GTXImage CAD™ Series Version 21.0

GTX Corporation

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Introduction

This manual describes the installation and use of Version 21 of the GTXImage CAD Series. The Series consists of two software products - **GTXImage CAD** & **GTXImage CAD** *PLUS.*

GTXImage CAD loads raster images and allows image clean-up, restoration & comprehensive editing.

GTXImage CAD *PLUS* provides the same functionallity as GTXImage CAD, with the added capability to convert binary raster into AutoCAD vector entities & text.

	Product Functionality					
Product Level	Raster Cleanup	Raster Editing	CAD Drafting	R2V Conversion	ICR Text Conversion	Batch Conversion
GTXImage **CAD**	X	X	X			
GTXImage CAD *PLUS*	X	X	X	X	X	X

GTXImage CAD

GTXImage CAD provides capability to efficiently **clean, enhance** and **Edit** raster images. It includes the following features:

- Raster Snapping that emulates AutoCAD object snapping to raster data.

- Intelligent Object Picking™ (IOP) makes selection of raster objects as easy as handling CAD entities. Allows selection of complex raster data, which preserve raster intersections - instead of creating gaps.

- Intelligent Enhancement helps improve drawing quality and legibility while reducing raster file storage requirements by 20-30% over already compressed formats.

- Improve drawing quality by using image enhancement to clean, deskew, scale, rotate and remove speckles.

- Selective Vectorization helps you trace raster with AutoCAD vector entities.

- Output changes to a variety of printers and plotters as a hybrid (raster and vector) or full raster drawing.

- GTXImage CAD can load color images & has commands to work with colour images to create bitonal, reduce colors & separate colour images into separate binary images.

GTXImage CAD *PLUS*

GTXImage CAD *PLUS* includes all the functionality of GTXImage CAD. It also provides powerful geometry and text conversion tools for transforming raster images into intelligent AutoCAD drawings.

- Raster-to-Vector Conversion converts selected raster data into AutoCAD entities. It provides four modes of R2V conversion for CAD, Contour, Edge & Text.

- Text - The Intelligent Character Recognition (ICR) quickly converts raster text which typically comprises 20% of engineering drawings. ICR is different than OCR (optical character recognition) because of its unique training capability that allows you to train the software to recognize new fonts or custom lettering styles. Dictionary look-up and pattern matching make ICR perfect for recognizing the complicated text, such as part numbers.

- Batch - A powerful batch-conversion program, GTX OSR®, is included with GTXImage CAD PLUS.

Mastering the GTXImage CAD Series

User's Manual

You can use this manual to learn about the software in several ways:

Using the GTXImage CAD Series provides an overview of working with raster images and general principles about using GTX software (Page 16).

Getting Started demonstrates features and functions of the software in step-by-step lessons (Page 28).

The **Command Reference** describes each GTXImage CAD command and feature in an alphabetical list (Page 380).

Use the **Index** to look up any term for which you need more information.

Other Sources of Help

Use the Windows Help file: You can launch this from the GTX program group help icon, by typing **gHelp** from the command prompt, by selecting *HELP* from the GTX ICAD pull-down menu, or by pressing **F1** at a GTX command-line prompt. These execute our online help file, ICAD.chm, from the GTX program folder.

The GTX Web site at http://www.gtx.com contains product information, frequently asked questions (FAQ) and technical notes.

Technical Support

A maintenance agreement is available for purchase from GTX Corporation. Contact your GTX reseller for details.

E-mail questions to GTX at: support@gtx.com.

Installing the Software

The Installation section describes how to install and configure both levels of the GTXImage CAD Series.

The Appendix

Glossary of Terms - Defines special terms used in this manual which are printed in **boldface**.

Configuration File - Describes the configuration file and its parameters.

Raster Formats - Lists supported raster import and export formats.

Keyed Entry

The GTXImage CAD Series supports keyboard entry of commands:

The **[RETURN]** and **[Space]** key can be used interchangeably to end a command sequence or close a selection.

The **[Esc]** key cancels commands.

GTXImage CAD commands allow transparent commands to run while another command is running (i.e., a **Transparent Zoom** is done by typing an apostrophe before the command name: ZOOM or Z).

Software Configuration The GTXImage CAD Series receives many of its default values from a configuration file called *GTXRAST.CFG*. This file can be edited with any ASCII text editor (such as Notepad or WordPad). See Appendix A2 at the end of this manual for more information about the configuration file.

Raster Files

The raster files you will use with the GTXImage CAD Series are created with a scanner. Proper scanner calibration and following the scanner's instructions will generate better images. The better the image, the less cleanup is necessary.

File Format

For a list of accepted raster file formats, see Appendix A3, **Raster Formats**, on page 511.

Image Size

Some raster file formats allow a maximum size of 32,768 (32K) **pixels** in both the X and Y directions. If your drawing size is large you may need to reduce your image DPI to stay within the above limit. For example, at 400 **DPI**, 32K pixels equals approximately to an 80 inches (2 meter) square drawing. The maximum raster limits are also restricted by the available memory and your system's swap file size.

Installation

System Requirements

Minimum requirements to run GTXImage CAD are outlined below.

CPU	Pentium 4 or AMD Athlon equivalent processor 1.6 GHz or higher with SSE2 technology
Hard Drive	2 GB free hard disk space for installation
Memory	2 Gb RAM (8Gb Recommended)
Display	1024 X 768 or better
Environment	Windows 10 (32 & 64 Bit), Windows 8/8.1 (32 & 64 Bit) & Windows 7 (32 & 64 Bit)

Manipulating raster images is very RAM hungry and the more RAM and virtual memory you have the better the program will perform.

Installing the Software

To install your software:

Installation Steps
1.   Download the software from http://www.gtx.com/register/download.asp
2.   During installation ensure that you select the product that you have purchased – **GTXImage CAD** or **GTXImage CAD** *PLUS*.
3.   Read README.PDF and other files to see any recent changes to the installation!

Note: To install icons in common Windows folders, log in as the administrator to install. To install icons in a personal program group, log in as the person who will be using the software.

Licensing your GTX Software

Software license resources (32 bit or 64 bit)

A 'software based' license is provided by GTX in the form of a license file named GTXnet.lic for floating licenses and GTX.lic for single user licenses. Application for a 'software based' license begins by running a registration program on the Windows computer that will host the license. The main GTXImageCAD Series V21 file folder contains the 'software based' license tutorial Help file named GTX_Flex_Licensing.chm. Double click that file name to display instructions and information which cover the following topics:

1. Initiating the registration process for single user and floating licenses.
2. Descriptions of single user and multiple user floating license files
3. Floating license administration tool setup.
4. Resource links to floating license options and error code description.

Please also visit our website for step-by-step instructions:

http://www.gtx.com/supportsoftwarelicensing.asp

Critical Configuration File Parameters

Three configuration parameters are located in the configuration file, *GTXRAST.CFG*. These are:

ProductLevel

Product Level

ProductLevel specifies the software level you intend to run. The installation sets this variable. If this variable is not listed, the program will default to *PLUS*.

For GTXImage CAD   set   **ProductLevel = ICAD**

For GTXImage CAD *PLUS*   set   **ProductLevel = ICADPLUS**

Note: If the product level listed is different from the product you purchased, your program will only run in demo mode. If you keep running in demo mode although you bought the product, make sure your product level is set to the correct level in the configuration file!

Uninstalling the Software

To remove the GTXImage CAD Series software from your system, use **Programs and Features** from the Control Panel.

Alternatively, you may run the ImageCAD setup program from the CD ROM and select Uninstall.

Removing the GTX Application Files

Note: Uninstall may not remove files that were not placed on your system by SETUP, including modified configuration files, AutoCAD drawings or raster images. You may need to remove those files manually yourself.

Using the GTXImage CAD Series

The Purpose of This Section

This section describes the use of the GTXImage CAD Series software. The GTXImage CAD is based upon the AutoCAD 2018 OEM engine and GTXImage CAD commands follow the same convention as AutoCAD commands. As such, AutoCAD users will find GTXImage CAD very similar and easy to use.

For example, in AutoCAD, the command for moving a CAD entity is **MOVE**. In the GTXImage CAD Series, the command for moving raster data is **gMOVE**. Both commands use similar prompts to accomplish similar tasks.

Raster and Vector

The GTXImage CAD Series lets you draw and edit a combination of both CAD (**Vector**) and scanned (**Raster**) data. Raster images are made up of dots (**pixels**) which represent pictures by forming an electronic mosaic. If you zoom close to a raster image, you will see each pixel as a square.

CAD drawings are made up of vector **entities** in which coordinates and other measurements are expressed mathematically in the CAD drawing. For example, a line is defined mathematically as the X and Y coordinates of each of its endpoints. A circle is defined as the X and Y coordinates of its center point and the length of its radius. Load a sample drawing into GTXImage CAD, window some of the entities and type **LIST**. You will see a barrage of entity types and coordinates.

GTXImage CAD lets you **Attach** raster images within vector drawings, displaying the raster within **image entities**. Image entities do not store the image data; they only refer, or link, to raster files already residing on disk. You can set properties for an image entity in a drawing, but these properties are only stored in the CAD drawing, and only affect the appearance of the image and do not directly affect the raster file at all.

GTXImage CAD will let you copy, move, rotate, resize and clip a raster image object within a CAD drawing. The Image Adjust feature adjusts the display of an image's brightness, contrast, fade and transparency.

A GTXImage CAD drawing provides an excellent framework for managing and modifying one or more related raster images. It also lets you display images to scale with the CAD vector drawing coordinates.

Product Functionality

Loading Raster Images into GTXImage CAD

GTXImage CAD provide you with several ways of importing raster images. To load a raster image into GTXImage CAD, the software **attaches** it within a CAD drawing. This CAD drawing can be saved in a number of AutoCAD formats such as .DWG, .DWF & .DXF – normally .DWG. The

image is not stored in the .DWG file, but is viewed through it. When attaching an image, be sure of whether you are putting it in paper space or model space. Although the raster is only an image of a paper drawing, you can attach an image in model space and then scale the **image entity** so that it is to scale with the CAD coordinate system.

Learn to use GTXImage CAD's **IMAGE>Attach Image** command before using any other methods. GTXImage CAD also supports dragging and dropping raster files of TIFF or .CAL raster formats.

GTXImage CAD provides the **gATTACH** command to quickly attach images without prompting for insertion point, scale and orientation. Instead, it makes certain assumptions about how to place, scale and orient raster images.

Command	Benefits
IMAGE>Attach	▪ Command supported by GTXImageCAD ▪ Offers full control of insertion point, scale & orientation
Drag-and-Drop	▪ Easy to specify file - just drag it
gATTACH	▪ One-step image import

Scaling and Aligning Attached Raster Images

Once an image has been imported into GTXImage CAD, it should

be scaled and aligned to fit the drawing. Learn how to use the **SCALE** command to properly scale the drawing. If you are working with line diagrams without a scale factor, you will probably want to attach images at a scale of 1, duplicating the original paper drawing. Use the scale factor of scanned drawings to determine how to scale its image entity to match your CAD model space coordinates. For example:

Having attached *SIDE.CG4* from the GTX tutorial files at 0,0 orientation at scale of 1, the drawing would look like this:
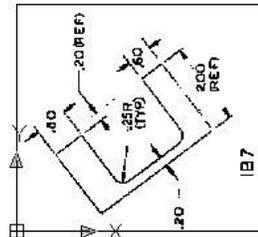
The dimensions in the raster image indicate the required size of the raster if it were a drawing represented within GTXImage CAD's model space.

The raster image will scale itself so that the geometry it portrays is to scale with GTXImage CAD's model space.

Note: Raster line widths will also scale along with raster images.

Orienting and Rotating Image Entities

Often paper drawings are scanned sideways. If you used the **Image>Attach** command to import a raster image, it may end up sideways. The best way to rotate the complete image is using the **gTURN** command:

gTURN Command

The **gTURN** command can be used to rotate an image, but it preserves the insertion point as the lower left point. Advantages of using this command are:

- The raster is turned inside the image entity, maintaining the lower left point insertion point.

- Because the raster turns rather than the image entity, the entity hasn't moved about any base point. If the image was in the right position before invoking gTURN then it requires no further moving or adjusting.

gROTATE Command

The **gROTATE** command can also rotate raster data within the raster image. It is useful for local rotation when raster editing, but is not recommended as an ideal tool for registering images within CAD drawings.

Creating New Raster Images

The **gCREATE** command allows you to create a new raster image. Just like scanned images that were captured from paper or mylar drawings, new images should be drawn at some drawing scale. New raster images are limited to realistic sizes. Create new raster images that are approximately paper-sized, and then scale the image entities using the **Scale** command.

For example, when creating a new image, you cannot make a raster image equivalent to a piece of paper the size of a city block. Instead, create a raster image the size of a large sheet of paper (E or A0 size). Then scale the image entity with the **Scale** command. This way you can avoid attempting to create oversized raster images.

Cropping Raster Data

Two commands are useful for erasing raster outside a rectangular area.

gCROP erases raster data outside a rectangular box. The raster image stays the same size.

gRESIZE removes the portion of the raster image outside a selected box. Raster data outside this box is erased because the raster image no longer occupies that space.
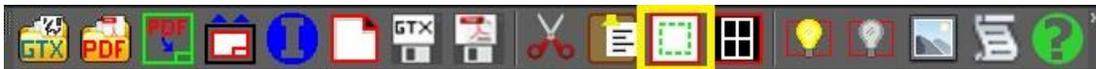
Modifying Images

gACTIVE loads the image into the GTX editor, making it the current image for editing. If there is no active image, raster editing commands will invoke **gACTIVE** before proceeding. When there are multiple images attached, **gACTIVE** command will require you to select the image to prepare. If there is only one image attached in GTXImage CAD, **gACTIVE** will automatically prepare it. **gINACTIVE** releases the image, recovering memory.

Cutting and Pasting Raster

The GTXImage CAD Series displays raster image entities which can be attached at any scale or orientation. Because of this, the **gCUT** and **gPASTE** commands may behave differently than you expect.

Scanning a paper drawing   Raster file at 270°

Image rotated through 90°   gCUT a section from a rotated image

gCUT produces the cut data as a new image, which is at a 270° orientation, just like the source raster file:

Image rotated at 30 °   gCUT from a rotated image

gCUT produces the cut data as a new image, which is at a 0° orientation, not at 30° like the source image entity. The border is larger than that specified during the gCUT command because it was cut at an angle.

Images oriented at angles other than 0 and 90 behave consistently. In this example, an image was inserted at a 30° orientation and gCUT is used to create a new image. The cut image is taken from the source image at 0° - not at 30°.

Converting Between Raster and Vector

The GTXImage CAD series provides many commands that will convert data between CAD and raster formats. These commands are found under the GTX Convert menu.

For more information, see the references for the following commands:

Vector-to-Raster Commands

gRASTER, **gBURN**

Raster-to-Vector Commands

gCONVRT, **gECONVRT**, **gTRACE**, **gVECTOR**

Intelligent Character Recognition

gTCONFIG, **gTRAIN, gTCONVRT**

Vector Cleanup

The commands under the GTXVClean menu aid in cleaning up drawings after conversion from raster. For more information on each command refer to the light blue backed icons in the 'Command Reference' section.

Snapping to Raster

GTXImage CAD provides many transparent raster snap methods that help you select points precisely within raster images. But these raster snaps can be used when drawing raster *or* vector CAD entities as well!

Raster AutoSnap

The GTXImage CAD also has an AutoSnap™ feature in which a ToolTip appears to indicate the type of raster snap being employed. The GTX command, **gAUTOSNAP**, controls whether AutoSnap™ will work within an image. Refer to the gAUTOSNAP command reference for more

information.

Saving Images

The GTXImage CAD Series provides several ways of saving modified images. In addition to saving .DWG files, GTXImage CAD's **SAVE** command will also save any changed images.

The **gSAVE** command lets you change the filename (and image name) of an image.

Command	Description
SAVE	Saves .DWG and any modified images.
SAVE AS	Allows saving in .DWG, .DXF or .DWF and any modified images.
GSAVE	Saves the current image.
QUIT or EXIT	When quitting GTXImage CAD, unsaved images can be saved. After which the .DWG is prompted to be saved.

GTXImage CAD Series Performance Factors

Because raster images can be so large, certain operating parameters affect system performance. In Minimum Requirements, we described the minimum recommended hardware configuration. The following GTXImage CAD parameters can also optimize display and editing speed:

Automatic Save Interval

GTXImage CAD saves modified raster images every time an **Automatic Save** is performed. You can increase the time interval between automatic saves and minimize this kind interruption by increasing the *SAVETIME* system variable.

Swap Space

Larger paging files can help performance when working with large raster images.

Aerial View

The aerial view maintains a second copy of the image in memory. To save memory you can keep this view closed. Additional processing time is taken when the Aerial View is set to automatically update. Turn off the automatic update option after loading the drawing.

Current View/Zoom

Redrawing images takes extra time. Performing a command generally takes longer if you are zoomed to extents than if you are zoomed in on a small area of the drawing.

Image Quality

Set the **Image Quality** to Draft to save additional time. High quality slows processes. Raster plotting occurs at high quality even when images are displayed in draft mode.

Multiple Images

GTXImage CAD easily handles multiple overlaid or tiled images. However, attaching multiple images can slow editing response.

Working with Color Images

The GTXImage CAD series has been designed to work predominantly with binary (Black & White) images and cannot directly edit color images like photos or high color pictures. However the new color commands allow extraction and conversion of color images to bitonal images that can then be edited and converted to AutoCAD entities.

Many color images use a limited range of colors. A good example of this is a road map where the data is displayed in a limited number of base colors.

In this example the roads are shown as black, the land contours are brown & the sea is blue. In these instances GTX provides commands to enable you to reduce the colors down to a more manageable range of colors. The GTX color related commands to do this are:

gBITONAL

gREDUCE

gSEPARATE

The best command to use depends mainly upon the number of colors that your image has.

Quick reduction to Binary.

If you wish to automatically reduce a color image down to a simple binary image you can use gBITONAL. The gBITONAL command has been optimized to operate on a broad range of color images & works well where an image has a clear-cut distinction between dark & light data.

However, the gBITONAL threshold may not give the exact results that you desire (by including or omitting important data from the binary image). In these instances it may be better to use the gREDUCE & gSEPARATE commands, where you have greater control of the color separation & threshold.

More than 256 colors.

If you have an image with greater than 256 colors you can firstly use gREDUCE to bring the

image down to 256 colors or less.

Less than 256 colors.

For images with colors less that 256 colors, either before or after using reduction, you can use the gSEPARATE command. This allows you to accurately select colors, or groups of colors, & break then out into separate binary color images.

These separate binary images can, if required, be further processed using the full range of GTX capabilities & commands detailed elsewhere in this help file.

For more detailed information about each command's operation refer to the respective section in the 'Command Reference'.

Supported Color file formats.

When working with color images in GTX you can only save in

.tif (uncompressed TIFF),

.**bmp** (Uncompressed Bitmap),

.png (Portable Network Graphics),

.tga (Taga) &

.pcx (PC Paintbrush) formats.

Also, if you originally loaded a **.jpg** (JFIF JPEG) file you will additionally be offered the option of saving in .jpg (JPEG) format.

JPEG Format files.

Although GTX can load & save .jpg (JPEG) files it is not an ideal operating format for GTX work. This is because JPEG is a format intended to work with pictures & photographs & when saving in JPEG format colors can automatically be added & removed from the color pallet. As such you can actively lose data by working & saving in JPEG format.

If possible we recommend that you save JPEG files in .TIF or .PNG format to avoid losing image definition.

Anti-Aliasing

Some color images are scanned with the 'anti-aliasing' option turned on. Anti-aliasng can enhance the quality of some images, particularly photographs. However it is not ideal for images with solid fill color as It tends to mix colors to achieve a more blurred image.

Whilst the gSEPARATE command will work with anti-aliased images the results may not be as good as required. As such we recommend that, if possible, images be scanned with anti-aliasing turned off.

Getting Started

Please visit the following link to access the Tutorials:

http://www.gtx.com/support/tutorials.asp

Using the Tutorials

To get the most from these lessons you should be familiar with GTXImage CAD Series image commands. Experiment with the GTXImage CAD **Image** command to attach images within AutoCAD, **ImageAdjust**, **ImageQuality**, **Transparency**, basic drawing and property-modifying commands (entity drawing commands, the **Properties** command, **Object Snapping** modes).

Following the Lessons

Try performing these exercises twice. The first time, follow the instructions literally to learn how each command and feature works. The second time, try to accomplish the same results as fast as possible, using the options and techniques with which you are most comfortable.

Standardizing Your Environment for the Tutorials

The lessons invoke GTXImage CAD Series features from menus.

The GTXImage CAD variables CMDDIA and FILEDIA control whether dialog boxes are used in certain commands. If CMDDIA and FILEDIA are set to 1, many GTXImage CAD Series commands use dialog boxes. If these variables are set to 0, many GTX commands will use command-line prompts. The lessons will describe the commands as they run with these variables set to "1".

Editing a Raster File

A typical editing session begins with enhancements that operate on most or all of the drawing. Next, make more detailed changes. Finally, focus on the overall picture to make sure the sections are properly matched and aligned.

1. The GTXImage CAD Series cannot modify images until you prepare them for editing. (**gACTIVE**)

2. Get rid of meaningless or unwanted raster to reduce memory requirements and to clean the image. (**gCROP**, **gSPECKL**, **gERASE**)

3. Because documents are usually scanned at a slight angle, deskew raster drawings before proceeding. (**gDESKEW**)

4. Next, perform editing functions that involve the whole drawing or large areas of the drawing. (**gMIRROR**, **gCUT**, **gPASTE**)

5. Completely edit the image in one area before moving on to another area. Insert missing objects

and correct distortions with GTXImage CAD commands. Leave new entities in vector format or rasterize them using **gRASTER**. Use **Raster Snapping** methods to select points on raster geometry just as you do using object snaps.

6. Create and edit text. If using GTXImage CAD *PLUS*, use **gTCONVERT** to convert raster text into AutoCAD text entities.

7. Look at the whole drawing and make any required changes (**gCOPY**, **gMOVE**, **gROTATE** and **gSMOOTH**).

8. Place new border and title block if needed.

     5. Save the file. (**SAVE**, **gSAVE**, **gCUT**).

Please visit the following link to access the Tutorials:

http://www.gtx.com/support/tutorials.asp

General Command Reference

General Information

The Command Reference, a comprehensive guide to the GTXImage CAD commands, lists all commands in alphabetical order.

In addition to the command listings, the *Command Reference* covers several topics in the appendixes: the GTXImage CAD *Series* configuration file, system variables, dimension variables, supported raster formats as well as other useful information.

For detailed information on Raster-specific functionality of GTXImage CAD, please refer to the Raster Command Reference in the next section.

Using GTXImage CAD Series Documentation

In addition to this Command Reference, several other resources are available to help you learn and use GTXImage CAD, both online and in print. You can access it from the Help menu.

Note: For the latest documentation corrections and additions, refer to the Readme.doc file.

Using This Reference

The Command Reference provides detailed information on all commands, command options, and system variables. The explanations reflect the default GTXImage CAD system variable settings and the default prototype drawing. If you change system variable settings, the prompts on your screen might differ from what you see here.

Executing Commands

The process of executing a command begins by starting the command, using one of several methods. For some commands, such as REGEN, no further action is required. For other commands, you must respond by providing additional information or actions in order to complete the command.

As you work with commands, note that right-clicking in the drawing area either acts as ENTER or displays a shortcut menu. You can control this behavior in the Options dialog box or with the SHORTCUTMENU system variable.

Starting Commands

You can start a command by doing one of the following:

- Select the command from a menu, toolbar, status bar, or shortcut menu.

- Enter the command name or command alias at the Command prompt and press ENTER or SPACEBAR.

In this Command Reference, near the beginning of each command description is a command access section, listing the specific ways you can start that command.

Standard toolbar:

View menu: Pan ▸ Realtime

Shortcut menu: With no objects selected, right-click in the drawing area, and choose Pan.

Command line: pan (or **'pan** for transparent use)

The availability of certain shortcut menus depends on the current setting of the SHORTCUTMENU system variable. The instructions in the command access sections assume that you have made the indicated menu available.

You can exit a command at any time by pressing ESC.

Using Transparent Commands

In many cases, you can start a command while using another command. The command you start is called a transparent command. Two right angle brackets (>>) precede prompts for transparent commands.

Command: **line**

Specify first point: **'grid**

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

Resuming LINE command

Specify first point:

In the Command Reference, the command access sections identify commands that you can use transparently.

Suppressing Dialog Boxes

Many commands provide both command line and dialog box interfaces. In most cases, when both a command line and dialog box interface are available, the command line version is preceded with a hyphen character. For example, the command line version of GROUP is   -Group.

Responding to Prompts

If a command does not immediately execute, GTXImage CAD either displays a dialog box or displays prompts on the command line requesting more information. Command line prompts are structured as follows:

Command: **commandname**

Current settings: Setting1 Setting2 Setting3

Instructional text [Option1/oPtion2/opTion3/...] <default option or value>:

The optional *current value line* displays the current settings for the command or for system variables related to the command. The succeeding prompts identify the type of input required to complete the command and, when applicable, list the available options in straight brackets and a default option or value in angle brackets. In some cases, GTXImage CAD determines default options and values based on the option or value that you last specified, or based on the settings of certain system variables.

Typically, the first word of a prompt indicates the type of action you can take. Most command line prompts begin with the word *enter*, *select*, or *specify*. These words indicate how you can respond to the prompt.

Terminology in command line prompts

If the prompt starts with...	*You can...*
Select	Select objects on the screen using your pointing device.
Enter	Enter text on the command line.
Specify	Use your pointing device or enter text on the command line to select a point location on the screen.

Selecting Options on the Command Line

To select one of the options contained within the straight brackets, you can enter the entire option name or only the capitalized letters, and then press ENTER or SPACEBAR. To select the default option enclosed in angle brackets (<>), if available, press ENTER or SPACEBAR.

You can also choose command options from a shortcut menu by right-clicking in the drawing area while the command is active. The options available on the command line appear in the shortcut menu.

Some prompts ask you to enter data, rather than (or as an alternative to) choosing an option. To do so, enter the text on the command line and press ENTER or SPACEBAR. However, be aware that when the command line prompt requests an object name, SPACEBAR inserts a space on the command line rather than acting as ENTER. This allows for the support of extended symbol names.

Just as default command options are often provided, prompts may include default values, enclosed in angle brackets (<>), when data is requested. For example, the POLYGON command displays the following prompt, suggesting **4** as the number of sides for your polygon:

Command: **polygon**

Enter number of sides <4>:

To accept the default value, press ENTER or SPACEBAR.

Using Wild-Card Characters on the Command Line

When the prompt directions ask for a name list, such as a block name, variable name, named view, and so on, you can use wild-card characters to specify names.

Repeating Commands

If no command is active, you can repeat the previous command by pressing ENTER or SPACEBAR or by right-clicking in the drawing area and choosing the Repeat *Command Name* option from the shortcut menu.

Accessing Recently Used Commands

You can access the six most recently used commands from a shortcut menu by right-clicking in the command window and choosing Recent Commands.

Standard File Selection Dialog Boxes

Several GTXImage CAD commands display standard file selection dialog boxes, in which you can navigate through your directory structure and select files to open. These dialog boxes may include the following options.

- **Look In** - Displays the current folder or drive. Click the arrow to view the hierarchy of the folder path and to navigate up the path tree.

- **Up One Level** - Navigates one level up in the current path tree.

- **Create New Folder** - Creates a new folder in the current path using a name that you specify.

- **List** - Displays the contents of the Files list in a multicolumn list.

- **Details** - Displays the contents of the Files list in a single-column list with file details.

- **Search the Web** - Displays the Browse the Web dialog box, from which you can access and store GTXImage CAD *Series* files on the Internet.

- **Look in Favorites** - Sets the Look In path to your system's *Favorites* folder.

- **Add to Favorites** - Creates a shortcut to the selected file or folder and adds that shortcut to your system's *Favorites* folder.

- **Files List** - Displays the files and folders present in the current path and of the selected file type.

- **File Name** - Displays the name of the file selected in the Files list. If you select multiple files, File Name displays each selected file within quotation marks. If you enter a file name in the File Name box, GTXImage CAD clears any selection. You must use quotation marks when entering multiple file names. You can use wild-card characters to filter files displayed in the Files list.

- **Files of Type** - Filters the list of applications by file type.

- **Preview** - Displays a preview image of the selected file. The Preview area is blank if you do not select a file.

- **Find File** - Displays the Browse/Search dialog box. You can browse through preview images of DWG files or search for files both locally and on a network.

- **Locate** - Uses the GTXImage CAD *Series* search path to locate the file specified in File Name. You set the search path on the Files tab in the Options dialog box.

References to Other Sections

At the end of most command descriptions, you'll find a section called "See Also." This refers to other GTXImage CAD *Series* commands and system variables related to the current command.

Typographical Conventions

To orient you to GTXImage CAD *Series* features as they appear on the screen, specific terms are set in typefaces that distinguish them from the body text. Throughout GTXImage CAD *Series* documentation, the following conventions are used:

Table 1: Typographical conventions

Text element	Example
GTXImage CAD Series commands	ADCENTER, DBCONNECT, SAVE
GTXImage CAD Series system variables	DIMBLK, DWGNAME, LTSCALE
GTXImage CAD Series named-objects, such as linetypes and styles	DASHDOT, STANDARD
Prompts	Select object to trim or [Project/Edge/Undo]:
Instructions after prompt sequences File names and file name extensions Folder or directory names	Select objects: Use an object selection method Readme file, .dwg file extension Sample folder, c:\program files\gtxImage CAD V21\sample
Text you enter	At the Command prompt, enter shape.
Keys you press on the keyboard	CTRL, F10, ESC, ENTER
Keys you press simultaneously on the keyboard	CTRL+C
Formal arguments specified in function definitions	The string and mode arguments

COMMAND OVERVIEW

ABOUT

Displays information about AutoCAD

Help menu: About

Command line: about (or **'about** for transparent use)

Copyright information and product information is displayed. Product information includes the version number and service pack, serial number, license type and expiration date, and the text of the license agreement. You can save the product information as a text file.

ADCCLOSE

Closes DesignCenter

Command line: adcclose

Closes the DesignCenter™ window.

ADCENTER

Standard toolbar:

Tools menu: DesignCenter

Command line: adcenter

Browses, finds, and previews content, and inserts content, which includes blocks, hatches, and external references (xrefs).
Use the buttons in the toolbar at the top of DesignCenter for display and access options.
When you click the Folders tab or the Open Drawings tab, the following two panes are displayed from which you can manage drawing content:

Content Area (DesignCenter)

Content Area (DesignCenter)

Displays the content of the "container" currently selected in the tree view. A container is a network, computer, disk, folder, file, or web address (URL) containing information accessible by DesignCenter. Depending on the container selected in tree view, the content area typically displays the following:

- Folders containing drawings or other files
- Drawings
- Named objects contained in drawings (named objects include blocks, xrefs, layouts, layers, dimension styles, and text styles)
- Images or icons representing blocks or hatch patterns
- Web-based content
- Custom content developed by third-party applications

From the content area, you can insert blocks or hatch patterns or attach external references in a drawing by dragging, by double-clicking, or by right-clicking and choosing Insert Block, Attach Xref, or Copy. You can drag or right-click to add other content to drawings, such as layers, dimension styles, and layouts. You can drag blocks and hatches from DesignCenter to tool palettes.

Note You can access relevant content area and tree view options on a shortcut menu by right-clicking in the tree view or the content area.

Load

Displays the Load dialog box (a standard file selection dialog box). Use Load to navigate to files on local and network drives or on the Web, and then to select content to load in the content area.

Back

Returns to the most recent location in the history list.

Forward

Returns to the next later location in the history list.

Up

Displays the contents of the container one level above the current container.

Stop (DC Online tab)

Stops the current transfer.

Reload (DC Online tab)

Reloads the current page.

Search

Displays the Search dialog box, where you can specify search criteria to locate drawings, blocks, and nongraphical objects within drawings.
Search also displays custom content saved on your desktop.

Favorites

Displays the contents of the *Favorites* folder in the content area. The *Favorites* folder contains shortcuts to items you access often. You can add items to *Favorites* either by right-clicking the

content area or right-clicking an item in the tree view, and then clicking Add to Favorites. To delete an item from *Favorites*, use the Organize Favorites option on the shortcut menu and then use the Refresh option on the shortcut menu.

Note The *DesignCenter* folder is automatically added to *Favorites.* This folder contains drawings with discipline-specific blocks that you can insert in drawings.

Home

Returns DesignCenter to your home folder. On installation, the home folder is set to *...\Sample\DesignCenter*. Change the home folder using the shortcut menu in the tree view.

Tree View Toggle

Displays and hides the tree view. Hide the tree view if you need more space in your drawing area. When the tree view is hidden, you can use the content area to navigate to containers and to load content.
The Tree View Toggle button is not available while you're using the History list in the tree view.

Preview

Displays and hides a preview of the selected item in a pane below the content area. If there is no preview image saved with the selected item, the Preview area is empty.

Description

Displays and hides a text description of the selected item in a pane below the content area. If a preview image is also displayed, the description is displayed below it. If there is no description saved with the selected item, the Description area is empty.

Views

Provides different display formats for the content that is loaded in the content area. You can select a view from the Views list or click the Views button repeatedly to cycle through the display formats. The default view varies for the type of content currently loaded in the content area.

Large Icon

   Displays the names of the loaded content in large icon format.

Small Icon

   Displays the names of the loaded content in small icon format.

List View

   Displays the names of the loaded content in a list.

Detail View

   Displays additional information about the loaded content. You can sort the items by name, size, type, and other properties, depending on the type of content that is loaded in the content area.

Refresh (Shortcut Menu Only)

Refreshes the display in the content area to reflect any changes you have made. Right-click the content area background and click Refresh on the shortcut menu.

Tree View (DesignCenter)

Tree View (DesignCenter)

Displays the hierarchy of files and folders on your computer and network drives, a list of open drawings, custom content, and a history of the last locations you accessed. Select an item in the tree view to display its contents in the content area.

Note In the *sample\designcenter* folder are drawings containing discipline-specific blocks that you can insert in drawings. These drawings are called symbol library drawings.

Use the buttons in the toolbar at the top of DesignCenter to access tree view options.

Folders

Displays the hierarchy of files and folders on your computer and network drives, including My Computer and Network Neighborhood.

You can use ADCNAVIGATE to navigate to a specific file name, directory location, or network path in the DesignCenter tree view. See ADCNAVIGATE.

Open Drawings

Displays all drawings open in the current work session, including drawings that are minimized.

History

Displays a list of the files that you opened most recently in DesignCenter. With the history displayed, right-click a file to display information about the file or to delete the file from the History list.

DC Online

Accesses the DesignCenter Online web page. When you establish a web connection, two panes are viewed on the Welcome page. The left side displays folders containing symbol libraries, manufacturer sites, and additional content libraries. When a symbol is selected, it is displayed on the right side and can be downloaded into your drawing.

ADCNAVIGATE

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

Command line: adcnavigate

Enter pathname *<current>*:

The path or drawing file name you specify is loaded in the tree view of the DesignCenter Folders tab.
At the prompt, you can enter a path in any of the following formats:

- Folder path: *c:\project files\electrical*
- Folder path and a file name: *c:\project files\electrical\circuit2.dwg*
- UNC (universal naming convention) network path: *\\server1\campus_project*

DesignCenter is displayed, with the Folder tab active, and the path or drawing file that you specified is loaded.
ADCNAVIGATE supports remote domains or workgroups if you have the target domain or workgroup mapped to a drive letter.

APERTURE

Controls the size of the object snap target box

⌨ **Command line: aperture** (or **'aperture** for transparent use)

   Object snap target height (1–50 pixels) <current>: *Enter a value* (**1**–**50**) *or press* ENTER

Object snap applies only to objects inside or crossing the object snap target box. The APBOX system variable controls whether the object snap target box is displayed. The number of pixels you enter using APERTURE controls the size of the object snap box. The higher the number, the larger the target box. You can also change this setting in the Options dialog box, Drafting tab.
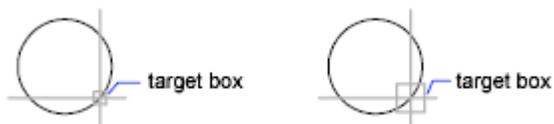
APERTURE controls the object snap target box, not the pickbox displayed at the Select Objects prompt. The object selection pickbox is controlled by the PICKBOX system variable.

ARC

Creates an arc

⬙ **Draw toolbar:**

⬙ **Draw menu:** Arc

⌨ **Command line: arc**

   Specify **start point** of arc or [**CEnter**]: *Specify a point, enter* **ce***, or press ENTER to* **start tangent to last line, arc, or polyline**

Start Point

Specifies the starting point of the arc.

Note If you press ENTER without specifying a point, the endpoint of the last drawn line or arc is used and you are immediately prompted to specify the endpoint of the new arc. This creates an arc tangent to the last drawn line, arc, or polyline.

Specify second point of arc or [CEnter/End]:

Second Point

Draws an arc using three specified points on the arc's circumference. The first point is the start point (1). The third point is the endpoint (3). The second point (2) is a point on the circumference of the arc.

Specify end point of arc:*Specify a point (3)*

You can specify a three-point arc either clockwise or counterclockwise.

Center

Specifies the center of the circle of which the arc is a part.

Specify center point of arc:

Specify end point of arc or [Angle/chord Length]:

End Point

Using the center point (2), draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point through the third point (3).
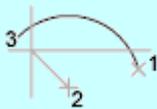
The arc does not necessarily pass through this third point, as shown in the illustration.

Angle

Draws an arc counterclockwise from the start point (1) using a center point (2) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

Specify included angle:*Specify an angle*
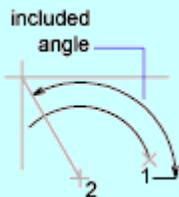
Chord Length

Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point. If the

chord length is negative, the major arc is drawn counterclockwise.

Specify length of chord:*Specify a length*

length of chord
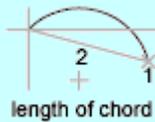
End

Specifies the endpoint of the arc.

Specify end point of arc:

Specify center point of arc or [Angle/Direction/Radius]:

Center Point

Draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point (3) through the second point specified (2).
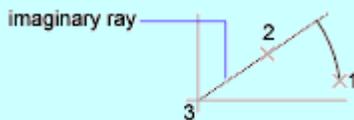
Angle

Draws an arc counterclockwise from the start point (1) to an endpoint (2), with a specified included angle. If the angle is negative, a clockwise arc is drawn.

Specify included angle: *Enter an angle in degrees or specify an angle by moving the pointing device counterclockwise*
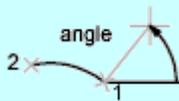
Direction

Begins the arc tangent to a specified direction. It creates any arc, major or minor, clockwise or counterclockwise, beginning with the start point (1), and ending at an endpoint (2). The direction is determined from the start point.

Specify tangent direction for the start point of arc:

direction
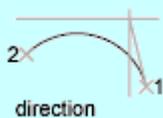
Radius

Draws the minor arc counterclockwise from the start point (1) to the endpoint (2). If the radius is negative, the major arc is drawn.

Specify radius of arc:
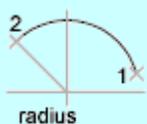
radius

Center

Specifies the center of the circle of which the arc is a part.

Specify center point of arc:

Specify start point of arc:

Specify end point of arc or [Angle/chord Length]:

End Point

Draws an arc counterclockwise from the start point (2) to an endpoint that falls on an imaginary ray drawn from the center point (1) through a specified point (3).
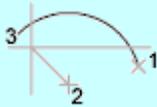
Angle

Draws an arc counterclockwise from the start point (2) using a center point (1) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

Specify included angle:
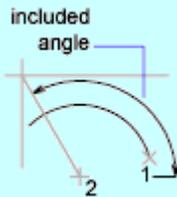
Chord Length

Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point. If the chord length is negative, the major arc is drawn counterclockwise.

Specify length of chord:
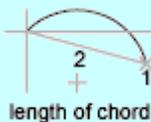
Tangent to Last Line, Arc, or Polyline

Tangent to Last Line, Arc, or Polyline

Draws an arc tangent to the last line, arc, or polyline drawn when you press ENTER at the first prompt.

Specify end point of arc: *Specify a point (1)*

AREA

Calculates the area and perimeter of objects or of defined areas

Inquiry toolbar:

Tools menu: Inquiry ➤ Area

Command line: area

Specify first corner point or [Object/Add/Subtract]: *Specify a point (1) or enter an option*

First Corner Point

First Corner Point

Calculates the area and perimeter you define by specifying points. All points must lie in a plane parallel to the *XY* plane of the current user coordinate system (UCS).

Specify next corner point or press ENTER for total: *Specify a point (2)*

defining an area and
perimeter

area defined
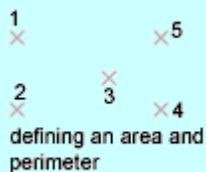
Continue to specify points to define a polygon and then press ENTER to complete the definition of the perimeter.

If you do not close the polygon, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated, that line length is added.

Object

Object

Calculates the area and perimeter of the selected object. You can calculate the area of circles, ellipses, splines, polylines, polygons, regions, and solids.

Note 2D solids (created with the SOLID command) do not have an area reported.

Select objects:

If you select an open polyline, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated that line length is ignored.

open polyline
selected
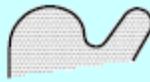
area defined

The centerline of a wide polyline is used to make area and perimeter (or length) calculations.
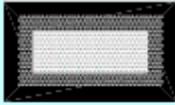
wide polyline

Add

Add

Turns on Add mode and keeps a running balance of the total area as you continue to define areas. The Add option calculates the individual areas and perimeters of defined areas and objects as well as the total area of all defined areas and objects. You can use the Subtract option to subtract specified areas from the total area.

   Specify first corner point or [Object/Subtract]: *Specify a point (1) or enter an option*

defined area to
be added

new area

First Corner Point

Calculates the area and perimeter you define by selecting points. All points must lie in a plane parallel to the *XY* plane of the current UCS.

   Specify next corner point or press ENTER for total (ADD mode): *Specify a point (2)*

Specify points to define a polygon (3). Press ENTER. AREA calculates the area and perimeter and returns the total area of all the areas defined by selecting points or objects since Add mode was turned on.

If you do not close the polygon, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated, that line length is added.

Object

Calculates the area and perimeter of the selected object.

   (ADD mode) Select objects:

AREA calculates the area and perimeter and returns the total area of all the areas defined by selecting points or objects since Add mode was turned on.

If you select an open polyline, the area is calculated as if a line were drawn from the last point entered to the first. When the perimeter is calculated that line length is ignored.

The centerline of a wide polyline is used to make area and perimeter calculations.

Subtract

Turns on Subtract mode and keeps a running balance of the total area as you subtract specified areas.

Subtract

Subtract

Similar to the Add option, but subtracts areas and perimeters.

selected area to be
subtracted

remaining area

ARRAY

Modify toolbar:
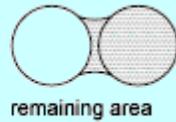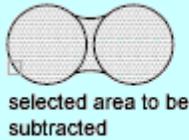
Modify menu: Array

Command line: array

Creates multiple copies of objects in a pattern. Use the **Rectangular Array** option to create an array of rows and columns of copies of the selected object. Use the **Polar Array** option to create an array by copying the selected objects around a center point.
Other options: **Select Objects**, **Preview Area**, **Preview**.

Rectangular Array

Rectangular Array

Creates an array of rows and columns of copies of the selected object.
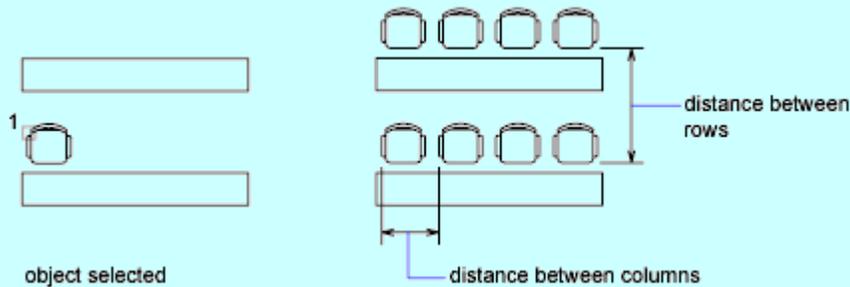
Rows

Specifies the number of rows in the array.
If you specify one row, you must specify more than one column. If you specify a large number of rows and columns for the array, it might take a while to create the copies. By default, the maximum number of array elements that you can generate in one command is 100,000. The limit is set by the

MAXARRAY setting in the registry. To reset the limit to 200,000, for example, enter **(setenv**

"MaxArray" "200000") at the Command prompt.

Columns

Specifies the number of columns in the array.
If you specify one column, you must specify more than one row. If you specify a large number of rows and columns for the array, it might take a while to create the copies. By default, the maximum number of array elements that you can generate in one command is 100,000. The limit is set by the

MAXARRAY setting in the registry. To reset the limit to 200,000, for example, enter **(setenv**

"MaxArray" "200000") at the Command prompt.

Offset Distance and Direction

Provides a space for you to specify the distance and direction of the array's offset.

Row Offset

Specifies the distance (in units) between rows. To add rows downward, specify a negative value. To specify row spacing with the pointing device, use the Pick Both Offsets button or the Pick Row Offset button.

Column Offset

Specifies the distance (in units) between columns. To add columns to the left, specify a negative value. To specify column spacing with the pointing device, use the Pick Both Offsets button or the Pick Column Offset button.

Angle of Array

Specifies the angle of rotation. This angle is normally 0, so the rows and columns are orthogonal with respect to the X and Y drawing axes of the current UCS. You can change the measurement conventions for angles using UNITS. The ANGBASE and ANGDIR system variables affect the angle of arrays.

Pick Both Offsets

Temporarily closes the Array dialog box so that you can use the pointing device to set the row and column spacing by specifying two diagonal corners of a rectangle.

Pick Row Offset

Temporarily closes the Array dialog box so that you can use the pointing device to specify the distance between rows. ARRAY prompts you to specify two points and uses the distance and direction between the points to specify the value in Row Offset.

Pick Column Offset

Temporarily closes the Array dialog box so that you can use the pointing device to specify the distance between columns. ARRAY prompts you to specify two points and uses the distance and direction between the points to specify the value in Column Offset.

Pick Angle of Array

Temporarily closes the Array dialog box so that you can specify the angle of rotation by entering a value or using the pointing device to specify two points. You can change the measurement conventions for angles using UNITS. The ANGBASE and ANGDIR system variables affect the angle of arrays.

Polar Array

Polar Array

Creates an array by copying the selected objects around a specified center point.

Center Point

Specifies the center point of the polar array. Enter coordinate values for *X* and *Y*, or choose Pick Center Point to use the pointing device to specify the location.

Pick Center Point

Temporarily closes the Array dialog box so that you can use the pointing device to specify the center point in the drawing area.

Method and Values

Specifies the method and values used to position objects in the polar array.

Method

Sets the method used to position objects. This setting controls which of the Method and Value fields are available for specifying values. For example, if the method is Total Number of Items & Angle to Fill, the related fields are available for specifying values; the Angle Between Items field is not available.

Total Number of Items

Sets the number of objects that appear in the resultant array. The default value is 4.

Angle to Fill

Sets the size of the array by defining the included angle between the base points of the first and last elements in the array. A positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation. The default value is 360. A value of 0 is not permitted.

Angle Between Items

Sets the included angle between the base points of the arrayed objects and the center of the array. Enter a positive value. The default direction value is 90.

Note You can choose the Pick buttons and use the pointing device to specify the values for Angle to Fill and Angle Between Items.

Pick Angle to Fill

Temporarily closes the Array dialog box so that you can define the included angle between the base points of the first and last elements in the array. ARRAY prompts you to select a point relative to another point in the drawing area.

Pick Angle Between Items

Temporarily closes the Array dialog box so that you can define the included angle between the base points of the arrayed objects and the center of the array. ARRAY prompts you to select a point relative to another point in the drawing area.

Rotate Items as Copied

Rotates the items in the array, as shown in the preview area.

More/Less

Turns the display of additional options in the Array dialog box on and off. When you choose More, additional options are displayed, and the name of this button changes to Less.

Object Base Point

Specifies a new reference (base) point relative to the selected objects that will remain at a constant distance from the center point of the array as the objects are arrayed. To construct a polar array, ARRAY determines the distance from the array's center point to a reference (base) point on the last object selected. The point used depends on the type of object, as shown in the following table.

Base point settings by object	
Object type	**Default base point**
Arc, circle, ellipse	Center point
Polygon, rectangle	First corner
Donut, line, polyline, 3D polyline, ray, spline	Starting point
Block, paragraph text, single-line text	Insertion point
Construction lines	Midpoint
Region	Grip point

Set to Object's Default

Uses the default base point of the object to position the arrayed object. To manually set the base point, clear this option.

Base Point

Sets a new *X* and *Y* base point coordinate. Choose Pick Base Point to temporarily close the dialog box and specify a point. After you specify a point, the Array dialog box is redisplayed.

Note To avoid unexpected results, set the base point manually if you are constructing a polar array and do not want to rotate the objects.

Select Objects

Select Objects

Specifies the objects used to construct the array. You can select objects before or after the Array dialog box is displayed. To select objects when the Array dialog box is displayed, choose Select Objects. The dialog box temporarily closes. When you finish selecting objects, press ENTER. The Array dialog box is redisplayed, and the number of objects selected is shown below the Select Objects button.

Note If you select multiple objects, the base point of the last selected object is used to construct the array.

Preview Area

Preview Area

Shows a preview image of the array based on the current settings in the dialog box. The preview image is dynamically updated when you move to another field after changing a setting.

Preview

Preview

Closes the Array dialog box and displays the array in the current drawing. Choose Modify to return to the Array dialog box to make changes.

ATTACHURL

Attaches hyperlinks to objects or areas in a drawing

⌨ **Command line:** attachurl

Enter hyperlink insert option [Area/Object] <Object>: *Enter a or press* ENTER

Area

Area

Creates the URLLAYER layer, draws a polyline on that layer, and attaches a URL to the polyline.

First corner: *Click in the drawing to indicate the lower-left corner of the area*

Other corner: *Click to indicate the upper-right corner of the area*

Enter hyperlink <current drawing>: *Enter a URL*

The polyline that represents the area is displayed in the color assigned to URLLAYER. The default color is red. When you move the cursor over the area in the drawing, the cursor changes to a hyperlink cursor to indicate that a URL is attached to the area.

Object

Object

Attaches a URL to the selected object.

Select objects: *Use an object selection method, and press* ENTER *to end selection*

Enter hyperlink <current drawing>: *Enter a URL*

When you move the cursor over the object in the drawing, the cursor changes to a hyperlink cursor to indicate that a URL is attached to the object.

ATTDEF

✎ **Draw menu:** Block ➤ Define Attributes

⌨ **Command line:** attdef

Defines the mode; attribute tag, prompt, and value; insertion point; and text options for an attribute.

Mode

Sets options for attribute values associated with a block when you insert the block in a drawing.
The default values are stored in the AFLAGS system variable. Changing the AFLAGS setting affects the default mode for new attribute definitions and does not affect existing attribute definitions.

Invisible

Specifies that attribute values are not displayed or printed when you insert the block. ATTDISP overrides Invisible mode.

Constant

Gives attributes a fixed value for block insertions.

Verify

Prompts you to verify that the attribute value is correct when you insert the block.

Preset

Sets the attribute to its default value when you insert a block containing a preset attribute.

Attribute

Sets attribute data. You can enter up to 256 characters. If you need leading blanks in the prompt or the default value, start the string with a backslash (\). To make the first character a backslash, start the string with two backslashes.

Tag

Identifies each occurrence of an attribute in the drawing. Enter the attribute tag using any combination of characters except spaces. Lowercase letters are automatically changed to uppercase.

Prompt

Specifies the prompt that is displayed when you insert a block containing this attribute definition. If you do not enter a prompt, the attribute tag is used as a prompt. If you select Constant in the Mode area, the Prompt option is not available.

Value

Specifies the default attribute value.

Insert Field Button

Displays the Field dialog box. You can insert a field as all or part of the value for an attribute.

Insertion Point

Specifies the location for the attribute. Enter coordinate values or select Specify On-screen and use the pointing device to specify the placement of the attribute in relation to the objects that it will be associated with.

Specify On-Screen

Displays a Start Point prompt when the dialog box closes. Use the pointing device to specify the placement of the attribute in relation to the objects that it will be associated with.

X

Specifies the *X* coordinate of the attribute's insertion point.

Y

Specifies the *Y* coordinate of the attribute's insertion point.

Z

Specifies the *Z* coordinate of the attribute's insertion point.

Text Options

Sets the justification, style, height, and rotation of the attribute text.

Justification

Specifies the justification of the attribute text. See TEXT for a description of the justification options.

Text Style

Specifies a predefined text style for the attribute text. Currently loaded text styles are displayed. To load or create a text style, see STYLE.

Height

Specifies the height of the attribute text. Enter a value, or choose Height to specify a height with your pointing device. The height is measured from the origin to the location you specify. If you select a text style that has fixed height (anything other than 0.0), or if you select Align in the Justification list, the Height option is not available.

Rotation

Specifies the rotation angle of the attribute text. Enter a value, or choose Rotation to specify a rotation angle with your pointing device. The rotation angle is measured from the origin to the location you specify. If you select Align or Fit in the Justification list, the Rotation option is not available.

Align Below Previous Attribute Definition

Places the attribute tag directly below the previously defined attribute. If you have not previously created an attribute definition, this option is not available.

Lock Position in Block

Locks the position of the attribute within the block reference.

Note In a dynamic block, an attribute's position must be locked for it to be included in an action's selection set.

ATTDISP

Globally controls visibility of block attributes in a drawing
View menu: Display ➤ Attribute Display

Command line: attdisp (or **'attdisp** for transparent use)

Enter attribute visibility setting [**Normal**/**ON**/**OFF**] <*current*>:

The drawing is regenerated after you change the visibility unless REGENAUTO, which controls automatic regeneration, is off. The current visibility of attributes is stored in the ATTMODE system variable.

Normal
Retains the current visibility of each attribute. Visible attributes are displayed. Invisible attributes are not displayed.

On
Makes all attributes visible.

Off
Makes all attributes invisible.

ATTDISP on

ATTDISP off
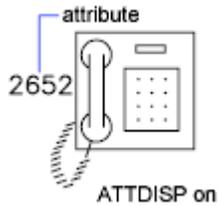
ATTEDIT

Modify menu: Object ➤ Attribute ➤ Single

Command line: attedit

Edits the attributes in a block.
To change attribute properties such as position, height, and style, use -ATTEDIT.

Block Name

Indicates the name of the selected block. The value of each attribute contained in the block is displayed in this dialog box.

List of Attributes

Displays the first eight attributes contained in the block. Edit the attribute values. If the block contains additional attributes, use Previous and Next to navigate through the list. You cannot edit attribute values on locked layers.

To use a field as the value, right-click and click Insert Field on the shortcut menu to display the Field dialog box.

Previous

Displays the previous eight attribute values. Previous is available only if the selected block contains more than eight attributes and you used Next to display the additional attributes.

Next

Displays the next eight attribute values. If the block contains no additional attributes, Next is unavailable.

ATTEXT

Extracts attribute data, informational text associated with a block, into a file

⌨ **Command line:** attext

The Attribute Extraction dialog box is displayed.

If you enter **-attext** at the Command prompt, ATTEXT displays prompts on the command line.

Specifies the file format for the attribute information, the objects from which you want to extract information, and the template and output file names for the information.

File Format

Sets the format for the file into which you are extracting the attribute data.

Comma Delimited File (CDF)

Generates a file containing one record for each block reference in the drawing that has at least one matching attribute tag in the template file. Commas separate the fields of each record. Single quotation marks enclose the character fields.

Space Delimited File (SDF)

Generates a file containing one record for each block reference in the drawing that has at least one matching attribute tag in the template file. The fields of each record have a fixed width; therefore, field separators or character string delimiters are not appropriate.

DXF Format Extract File (DXX)

Produces a subset of the AutoCAD Drawing Interchange File format containing only block reference, attribute, and end-of-sequence objects. DXF™ format extraction requires no template. The file name extension *.dxx* distinguishes the output file from normal DXF files.

Select Objects

Closes the dialog box so you can use the pointing device to select blocks with attributes. When the Attribute Extraction dialog box reopens, Number Found shows the number of objects you selected.

Number Found

Indicates the number of objects you selected using Select Objects.

Template File

Specifies a template extraction file for CDF and SDF formats. Enter the file name in the box, or choose Template File to search for existing template files using a standard file selection dialog box. The default file extension is *.txt*. If you select DXF under File Format, the Template File option is not available.

Output File

Specifies the file name and location for the extracted attribute data. Enter the path and file name for the extracted attribute data, or choose Output File to search for existing template files using a standard file selection dialog box. The *.txt* file name extension is appended for CDF or SDF files and the *.dxx* file name extension for DXF files.

ATTREDEF

Redefines a block and updates associated attributes

⌨ **Command line:** attredef

    Enter the name of the block you wish to redefine:

    Select objects for new block:

    Select objects:

    Insertion base point of new block: *Specify a point*

New attributes assigned to existing block references use their default values. Old attributes in the new block definition retain their old values. Any old attributes that are not included in the new block definition are deleted.

AUDIT

Evaluates the integrity of a drawing and corrects some errors

⬡ **File menu:** Drawing Utilities ➤ Audit

⌨ **Command line:** audit

    Fix any errors detected? [Yes/No] <N>: *Enter* **y** *or* **n***, or press*ENTER

For easy access, AUDIT places all objects for which it reports errors in the Previous selection set. However, editing commands affect only the objects that belong to the current paper space or model space.

If you set the AUDITCTL system variable to 1, AUDIT creates an ASCII file describing problems and the action taken and places this report in the same directory as the current drawing, with the file extension *.adt*.

If a drawing contains errors that AUDIT cannot fix, use RECOVER to retrieve the drawing and correct its errors.

BASE

Sets the insertion base point for the current drawing

⬡ **Draw menu:** Block ➤ Base

⌨ **Command line:** base (or **'base** for transparent use)

    Enter base point *<current>*: *Specify a point or press* ENTER

The base point is expressed as coordinates in the current UCS. When you insert or externally reference the current drawing into other drawings, this base point is used as the insertion base point.

BLIPMODE

Controls the display of marker blips

⌨ **Command line:** **blipmode** (or **'blipmode** for transparent use)

Enter mode [ON/OFF] <current>: *Enter* **on** *or* **off***, or press* ENTER

When Blip mode is on, a temporary mark in the shape of a plus sign (+) appears where you specify a point. BLIPMODE is off by default.
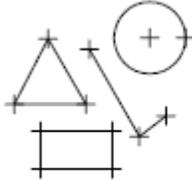
objects drawn with
BLIPMODE on
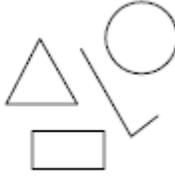
objects drawn with
BLIPMODE off

To remove marker blips, use REDRAW, REGEN, ZOOM, PAN, or other commands that redraw or regenerate the drawing.

BLOCK

🔷 **Draw toolbar:**

🔷 **Draw menu:** Block ➤ Make

⌨ **Command line:** **block**

Defines and names a block.

Name

Names the block. The name can have up to 255 characters and can include letters, numbers, blank spaces, and any special character not used by Microsoft® Windows® and the program for other purposes, if the system variable EXTNAMES is set to 1.
The block name and definition are saved in the current drawing.

Note You cannot use DIRECT, LIGHT, AVE_RENDER, RM_SDB, SH_SPOT, and OVERHEAD as valid block names.

Base Point

Specifies an insertion base point for the block. The default value is 0,0,0.

X

Specifies the *X* coordinate value.

Y

Specifies the *Y* coordinate value.

Z

Specifies the *Z* coordinate value.

Pick Insertion Base Point

Temporarily closes the dialog box so that you can specify an insertion base point in the current drawing.

Objects

Specifies the objects to include in the new block and whether to retain or delete the selected objects or convert them to a block instance after you create the block.

Select Objects

Closes the Block Definition dialog box temporarily while you select the objects for the block. When you finish selecting objects, press ENTER to redisplay the Block Definition dialog box.

Quick Select

Displays the Quick Select dialog box, which defines a selection set.

Retain

Retains the selected objects as distinct objects in the drawing after you create the block.

Convert to Block

Converts the selected objects to a block instance in the drawing after you create the block.

Delete

Deletes the selected objects from the drawing after you create the block.

Objects Selected

Displays the number of selected objects.

Settings

Specifies settings for the block.

Block Unit

Specifies the insertion units for the block reference.

Scale Uniformly

Specifies whether or not the block reference is prevented from being non-uniformly scaled.

Allow Exploding

Specifies whether or not the block reference can be exploded.

Description

Specifies the text description of the block.

Hyperlink

Opens the Insert Hyperlink dialog box, which you can use to associate a hyperlink with the block definition.

Open in Block Editor

Opens the current block definition in the Block Editor when you click OK.

BLOCKICON

Generates preview images for blocks displayed in DesignCenter

File menu: Drawing Utilities ➤ Update Block Icons

Command line: blockicon

Enter block names <*>: *Specify block names, or press* ENTER *to update all blocks*

Use this command to generate icons for blocks created with an earlier release. You can enter a series of comma-delimited block names or wild-card characters. For example, enter **b1,?2**, to specify that block B1 and all two-character blocks ending with 2 should be updated. After you enter the block names, a message is displayed describing the process as it proceeds. Press ESC at any time to stop.

BMPOUT

Saves selected objects to a file in device-independent bitmap format

Command line: bmpout

The Create Raster File dialog box (a standard file selection dialog box) is displayed. Enter the file name in the dialog box.

Select objects or <all objects and viewports>: *Press* ENTER *to select all objects and viewports or use an object selection method and press* ENTER

A bitmap file that contains the objects you select is created. The file reflects what is displayed on the screen.

Note When the FILEDIA system variable is set to 0 (Off), prompts are displayed on the command line.

BOUNDARY

Creates a region or a polyline from an enclosed area

Draw menu: Boundary

Command line: boundary

The Boundary Creation dialog box is displayed.

If you enter **-boundary** at the Command prompt, BOUNDARY displays prompts on the command line.

Defines the object type, boundary set, and island detection method for creating a region or polyline using a specified point within an area enclosed by objects.

Pick Points

Determines a boundary from existing objects that form an enclosed area around the specified point.

Island Detection

Controls whether BOUNDARY detects internal closed boundaries, called islands.

Object Type

Controls the type of the new boundary object. BOUNDARY creates the boundary as a region or a polyline object.

Boundary Set

Defines the set of objects BOUNDARY analyzes when defining a boundary from a specified point.

Current Viewport

Defines the boundary set from everything in the current viewport extents. Selecting this option discards any current boundary set.

New

Prompts you to select the objects that define the boundary set. BOUNDARY includes only the objects that can be used to create a region or closed polyline when it constructs the new boundary set.

For more information about the options in this dialog box, see HATCH.

BREAK

Breaks the selected object between two points

Modify toolbar:

Modify menu: Break

Command line: **break**

Select object: *Use an object selection method, or specify the first break point (1) on an object*

The prompts that are displayed next depend on how you select the object. If you select the object by using your pointing device, the program both selects the object and treats the selection point as the first break point. At the next prompt you can continue by specifying the second point or overriding the first point.

Specify second break point or [First point]: *Specify the second break point (2) or enter f*

Second Break Point

Specifies the second point to use to break the object.

First Point

Overrides the original first point with the new point that you specify.

Specify first break point:
Specify second break point:

before break

after break
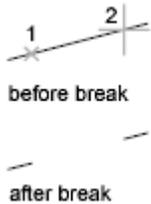
The portion of the object is erased between the two points that you specify. If the second point is not on the object, the nearest point on the object is selected; therefore, to break off one end of a line, arc, or polyline, specify the second point beyond the end to be removed.
To split an object in two without erasing a portion, enter the same point for both the first and

second points. You can do this by entering **@** to specify the second point.

Lines, arcs, circles, polylines, ellipses, splines, donuts, and several other object types can be split into two objects or have one end removed.
The program converts a circle to an arc by removing a piece of the circle starting counterclockwise from the first to the second point.

BROWSER

Launches the default web browser defined in your system's registry
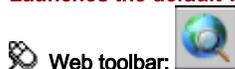
Web toolbar:

Command line: browser

> Enter Web location (URL) *<current>*: *Press* ENTER *or enter a new location; you don't need to enter http:// before the location*

Pressing ENTER displays your web browser, which automatically connects to the location you specify. Because BROWSER does not append "http://" to web locations, you can specify an FTP or file location to your default web browser.

Note BROWSER requires that you have Netscape Navigator 3.0 (or later) or Microsoft Internet Explorer installed.

CAL

Evaluates mathematical and geometric expressions

Command line: cal (or 'cal for transparent use)

CAL is an online geometry calculator that evaluates point (vector), real, or integer expressions. The expressions can access existing geometry using the object snap functions such as CEN, END, and INS.
You can insert AutoLISP® variables into the arithmetic expression and assign the value of the expression back to an AutoLISP variable.
You can use these arithmetic and vector expressions in any command that expects points, vectors, or numbers.
CAL topics:

CHAMFER

Bevels the edges of objects

Modify toolbar:

Modify menu: Chamfer

Command line: chamfer

> (TRIM mode) Current chamfer Dist1 = *current*, Dist2 = *current*

> Select **first line** or [**Undo**/**Polyline**/**Distance**/**Angle**/**Trim**/**mEthod**/**Multiple**]: *Use an object selection method or enter an option*

First Line

First Line

Specifies the first of two edges required to define a 2D chamfer, or the edge of a 3D solid to chamfer.

> Select second line or shift-select to apply corner: *Use an object selection method or hold down SHIFTand select an object to create a sharp corner*

If you select lines or polylines, their lengths adjust to accommodate the chamfer line. You can hold down SHIFT while selecting the objects to override the current chamfer distances with a value of 0. If the selected objects are line segments of a 2D polyline, they must be adjacent or separated by no more than one segment. If they're separated by another polyline segment, CHAMFER deletes the segment that separates them and replaces it with the chamfer.

If you select an edge on a 3D solid, you must indicate which one of the two surfaces adjacent to the edge is the base surface.

> Base surface selection...

> Enter surface selection option [Next/OK (current)] <OK>: *Enter* **n** *or* **o**, *or press* ENTER

Entering **o** or pressing ENTER sets the selected surface as the base surface. Entering **n** selects either of the two surfaces adjacent to the selected edge.

> Specify base surface chamfer distance <*current*>:

> Specify other surface chamfer distance <*current*>:

After you select the base surface and the chamfer distances, select the edges of the base surface to chamfer. You can select edges individually or all the edges at once.

> Select an edge or [Loop]: S*elect an edge, enter* **L***, or press* ENTER
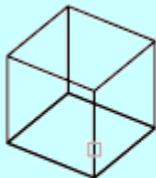
first edge selected   first base surface   second base surface
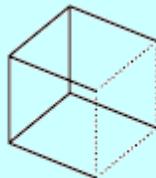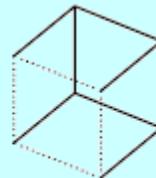
Edge

Selects an individual edge to chamfer.
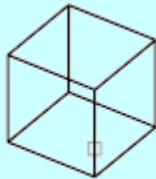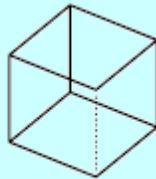
select edge   edge selected   chamfered edge
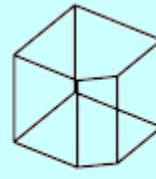
Loop

Switches to Edge Loop mode.

> Select an edge loop or [Edge]: *Select an edge, enter* **e***, or press* ENTER

Edge Loop

Selects all edges on the base surface.

selecting edge loop   edge loop selected   chamfered edge loop

Edge

   Switches to Edge mode.
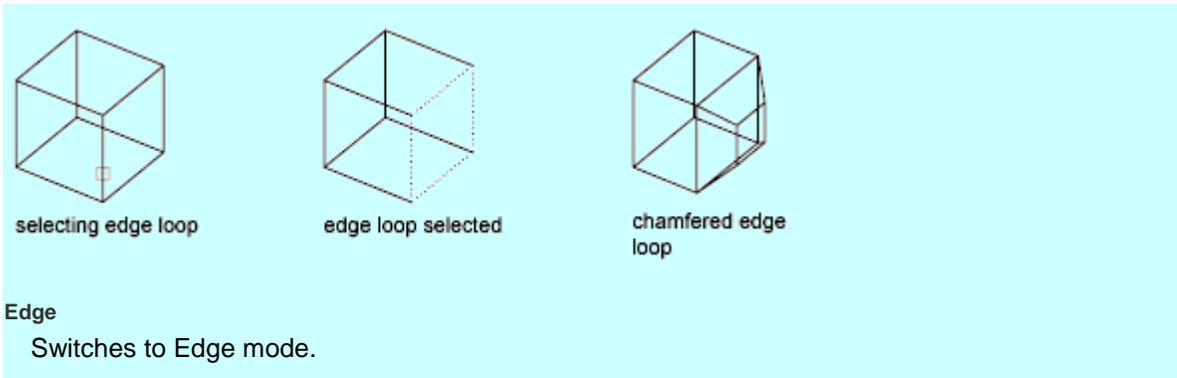
Undo

Undo

Reverses the previous action in the command.

Polyline

Polyline

Chamfers an entire 2D polyline.

   Select 2D polyline:

The intersecting polyline segments are chamfered at each vertex of the polyline. Chamfers become new segments of the polyline.
If the polyline includes segments that are too short to accommodate the chamfer distance, those segments are not chamfered.

selected polyline   result

Distance

Distance

Sets the distance of the chamfer from the endpoint of the selected edge.

   Specify first chamfer distance *<current>*:

   Specify second chamfer distance *<current>*:

equal distances   unequal distances
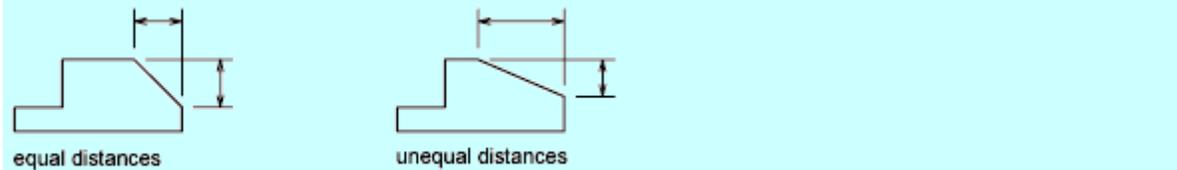
If you set both distances to zero, CHAMFER extends or trims the two lines so they end at the same point.

Angle

Angle

Sets the chamfer distances using a chamfer distance for the first line and an angle for the second line.

Specify chamfer length on the first line *<current>*:

Specify chamfer angle from the first line *<current>*:
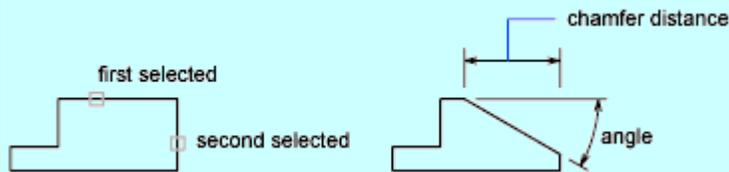
Trim

Trim

Controls whether CHAMFER trims the selected edges to the chamfer line endpoints.

Enter Trim mode option [Trim/No trim] *<current>*:

Note Trim sets the TRIMMODE system variable to 1; No Trim sets TRIMMODE to 0.

If the TRIMMODE system variable is set to 1, CHAMFER trims the intersecting lines to the endpoints of the chamfer line. If the selected lines do not intersect, CHAMFER extends or trims them so that they do. If TRIMMODE is set to 0, the chamfer is created without trimming the selected lines.

Method

Method

Controls whether CHAMFER uses two distances or a distance and an angle to create the chamfer.

Enter trim method [Distance/Angle] *<current>*:

Multiple

Multiple

Chamfers the edges of more than one set of objects. CHAMFER displays the main prompt and the Select Second Object prompt repeatedly until you press ENTER to end the command.

CHANGE

Changes the properties of existing objects

⌨ **Command line:** change

Select objects:

Except for zero-thickness lines, the objects selected must be parallel to the current user coordinate system (UCS).
If you select lines and other changeable objects in the same selection set, you get varying results depending on the object selection sequence. The easiest way to use CHANGE is to select only lines in a selection set or select only objects other than lines in a selection set.

Properties

Modifies properties of existing objects.
    Enter property to change [Color/Elev/LAyer/LType/ltScale/LWeight/Thickness/PLotstyle]:

Note The Plotstyle option is displayed only when you are using named plot styles.

If you select several objects with different values for the property you want to change, *varies* is displayed as the current value.
You can change several properties at a time. The Enter Property to Change prompt is redisplayed after each option is completed.

Color

Changes the color of the selected objects.
    Enter new color [Truecolor/COlorbook]<*current*>: *Enter a color name or a number from* **1** *through* **255***, enter* **t***, enter* **co***, enter* **bylayer** *or* **byblock***, or press* ENTER

For example, to change a color to red, enter **red** or **1**. If you enter **bylayer**, the object assumes the color of the layer on which it is located. If you enter **byblock**, the object inherits the color of the block of which it is a component.

True Color

  Specifies a true color to be used for the selected object.

  Red, Green, Blue: *Enter three integer values from 0 to 255 separated by commas to specify a true color*

Color Book

  Specifies a color from a loaded color book to be used for the selected object.

  Enter Color Book name: *Enter the name of a color book that has been installed, such as* **PANTONE**

  If you enter a color book name, the following prompt is displayed.

  Enter color name: *Enter the name of a color included in the selected color book, such as* **Pantone 573**

Elev

Changes the *Z*-axis elevation of 2D objects.
    Specify new elevation <*current*>:
You can change the elevation of an object only if all its points have the same *Z* value.
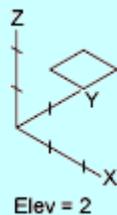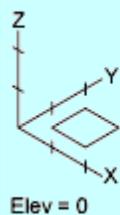
Elev = 0   Elev = 2

Layer

Changes the layer of the selected objects.
    Enter new layer name <*current*>:

Ltype

Changes the linetype of the selected objects.

Enter new linetype name *<current>*:

If the new linetype is not loaded, the program tries to load it from the standard linetype library file, *acad.lin*. If this procedure fails, use LINETYPE to load the linetype.

Ltscale

Changes the linetype scale factor of the selected objects.

Specify new linetype scale *<current>*:

Lweight

Changes the lineweight of the selected objects. Lineweight values are predefined values. If you enter a value that is not a predefined value, the closest predefined lineweight is assigned to the selected objects.

Enter new lineweight *<current>*:

Thickness

Changes the *Z*-direction thickness of 2D objects.

Specify new thickness *<current>*:

Changing the thickness of a 3D polyline, dimension, or layout viewport object has no effect.
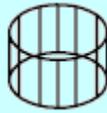
Plotstyle

Changes the plot style of the selected objects. A plot style is a collection of property settings that is saved in a plot style table.

Enter Plot Style or [?] *<current>*: *Enter a name or enter* **?**

Plot Style

Assigns the plot style to the selected objects. When you plot the drawing, objects are plotted with the properties defined in the designated plot style. Plot styles assigned to objects can be overridden by other plot style assignments.

?—List Plot Styles

Lists all the plot style names defined in the drawing.

CHPROP

Changes the color, layer, linetype, linetype scale factor, lineweight, thickness, and plot style of an object

⌨ **Command line:** chprop

Select objects:

Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness/PLotstyle]:

Note The Plotstyle option is displayed only when you are using named plot styles.

If you select several objects with different values for the property you want to change, *varies* is displayed as the current value.

Color

Color

Changes the color of the selected objects.

Enter new color [Truecolor/COlorbook]<*current*>: *Enter a color name or a number from **1** through **255**, enter **t**, enter **co**, enter **bylayer** or **byblock**, or press* ENTER

For example, to change a color to red, enter **red** or **1**. If you enter **bylayer**, the object assumes the color of the layer on which it is located. If you enter **byblock**, the object inherits the color of the block of which it is a component.

True Color

Specifies a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from 0 to 255 separated by commas to specify a true color*

Color Book

Specifies a color from a loaded color book to be used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* **PANTONE**

If you enter a color book name, the following prompt is displayed.

Enter color name: *Enter the name of a color included in the selected color book, such as* **PANTONE573**

Layer

Layer

Changes the layer of the selected objects.

Enter new layer name <*current*>:

Ltype

Ltype

Changes the linetype of the selected objects.

Enter new linetype name <*current*>:

If the new linetype is not loaded, the program tries to load it from the standard linetype library file, *acad.lin*. If this procedure fails, use LINETYPE to load the linetype.

Ltscale

Ltscale

Changes the linetype scale factor of the selected objects.
   Specify new linetype scale <*current*>:

Lweight

Lweight

Changes the lineweight of the selected objects. Lineweight values are predefined values. If you enter a value that is not a predefined value, the closest predefined lineweight is assigned to the selected objects.
   Enter new lineweight <*current*>:

Thickness

Thickness

Changes the *Z*-direction thickness of 2D objects.
   Specify new thickness <*current*>:

Changing the thickness of a 3D polyline, dimension, or layout viewport object has no effect.
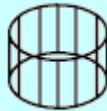
Plotstyle

Plotstyle

Changes the plot style of the selected objects. A plot style is a collection of property settings that is saved in a plot style table. This option is displayed only when you are using named plot styles.
   Enter Plot Style or [?] <*current*>: *Enter a name or enter* **?**

Plot Style

Assigns the plot style to the selected objects. When you plot the drawing, objects are plotted with the properties defined in the designated plot style. Plot styles assigned to objects can be overridden by other plot style assignments.

?—List Plot Styles

Lists all the plot styles defined in the drawing.

CIRCLE

Creates a circle

⬙ **Draw toolbar:**

⬙ **Draw menu:** Circle

⌨ **Command line:** circle

Specify <u>center point</u> for circle or [**3P (Three Points)**/**2P (Two Points)**/**Ttr (tan tan radius)**]: *Specify a point or enter an option*

Center Point

Center Point

Draws a circle based on a center point and a diameter or a radius.
Specify radius of circle or [Diameter]: *Specify a point, enter a value, enter d, or press* ENTER

Radius

Defines the radius of the circle. Enter a value, or specify a point (2). The distance between this point and the center point determines the radius of the circle.
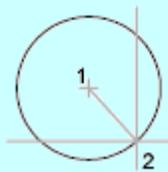
Diameter

Draws a circle using the center point and a specified distance for the diameter.

Specify diameter of circle <*current*>: *Specify a point (2), enter a value, or press* ENTER
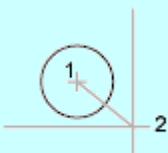
3P (Three Points)

3P (Three Points)

Draws a circle based on three points on the circumference.

Specify first point on circle: *Specify a point (1)*

Specify second point on circle: *Specify a point (2)*

Specify third point on circle: *Specify a point (3)*
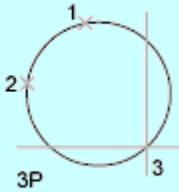
2P (Two Points)

2P (Two Points)

Draws a circle based on two endpoints of the diameter.

Specify first endpoint of circle's diameter: *Specify a point (1)*

Specify second endpoint of circle's diameter: *Specify a point (2)*
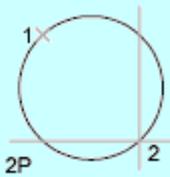
TTR (Tangent, Tangent, Radius)

TTR (Tangent, Tangent, Radius)

Draws a circle with a specified radius tangent to two objects.

Specify point on object for first tangent of circle: *Select a circle, arc, or line*

Specify point on object for second tangent of circle: *Select a circle, arc, or line*

Specify radius of circle *<current>*:

tan, tan, radius
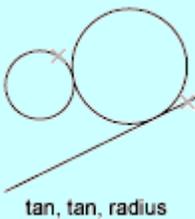
Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.

CLOSE

Closes the current drawing

📎 **File menu:** Close

⌨ **Command line:** close

The current drawing is closed. If you modified the drawing since it was last saved, you are prompted to save or discard the changes.
You can close a file that has been opened in read-only mode if you have made no changes or if you are willing to discard changes. To save changes to a read-only file, you must use the SAVEAS command.

COLOR

Sets the color for new objects

📎 **Format menu:** Color

⌨ **Command line:** color (or 'color for transparent use)

The Select Color dialog box is displayed.

If you enter -color at the Command prompt, COLOR displays prompts on the command line.

COMPILE

Compiles shape files and PostScript font files into SHX files

⌨ **Command line:** compile

The Select Shape or Font File dialog box (a standard file selection dialog box) is displayed. Enter the SHP or PFB file name in the dialog box. The compiled file is assigned this name with the file extension *.shx.*

CONVERTPSTYLES

Converts the current drawing to either named or color-dependent plot styles

⌨ **Command line:** convertpstyles

A drawing can use either named or color-dependent plot styles, but not both. CONVERTPSTYLES converts a currently open drawing from color-dependent plot styles to named plot styles, or from named plot styles to color-dependent plot styles, depending on which plot style method the drawing is currently using.

- **Converting drawings from color-dependent plot styles to named plot styles**
- **Converting drawings from named plot styles to color-dependent plot styles**

When you convert a drawing, CONVERTPSTYLES sets the PSTYLEMODE system variable to the appropriate setting (1 = named, 0 = color-dependent).

For example, a drawing using color-dependent plot styles assigns plot properties to objects and layers by color. In other words, all objects with the same color have the same plot properties. CONVERTPSTYLES converts the drawing to use named plot styles that can be applied to objects or layers independent of color assignment. In other words, all objects with the same color can have different plot properties.

Converting Drawings from Color-Dependent to Named Plot Styles

Converting Drawings from Color-Dependent to Named Plot Styles

A dialog box advises you to convert the drawing's plot style tables before converting the drawing. You can convert the color-dependent plot style tables assigned to the drawing to named plot style tables using CONVERTCTB.

The Select File dialog box (a standard file selection dialog box) is displayed, where you select the named plot style table file to attach to the converted drawing.

CONVERTPSTYLES requires you to select a named plot style table that was converted using CONVERTCTB or created from a PC2 or PCP file. Normally you should select the named plot style table that was converted from the color-dependent plot style table that was assigned to the same drawing.

CONVERTPSTYLES attaches the selected named plot style table to model space and to all layouts. Drawing layers are each assigned a named plot style (from the converted plot style table) that has the same plot properties that their color-dependent plot style had. Drawing objects that had the same color-dependent plot style as their layer are assigned the named plot style BYLAYER. Drawing objects that had a color-dependent plot style that was different from their layer are assigned a named plot style that has the same properties that their color-dependent plot style had.

After a drawing is converted to named plot styles, you can change the named plot style table assignment or assign other named plot styles tables to model space or layouts. You can also assign individual plot styles to drawing objects.

Converting Drawings from Named to Color-Dependent Plot Styles

Converting Drawings from Named to Color-Dependent Plot Styles

A dialog box advises you that the named plot styles attached to objects and named plot style tables attached to model space and layouts will be detached.

After a drawing is converted to color-dependent plot styles, you can assign a color-dependent plot style table. Plot styles will be applied by color.

COPY

Copies objects a specified distance in a specified direction

 Modify toolbar:

 Modify menu: Copy

Shortcut menu: Select the objects to copy, and right-click in the drawing area. Click Copy Selection.

 Command line: copy

Select objects: *Use an object selection method and press ENTER when you finish*

Specify base point or [Displacement]<Displacement>: *Specify a base point or enter* **d**

The two points you specify define a vector that indicates how far the copied objects are to be moved and in what direction.

If you press ENTER at the Specify Second Point prompt, the first point is interpreted as a relative X,Y,Z displacement. For example, if you specify **2,3** for the base point and press ENTER at the next prompt, the objects are copied 2 units in the X direction and 3 units in the Y direction from their current position.

The COPY command repeats for convenience. To exit the command, press ENTER.

Displacement

Specify displacement *<last value>*: *Enter coordinates to represent a vector*

The coordinate values that you enter specify a relative distance and direction.

COPYBASE

Copies objects with a specified base point

 Edit menu: Copy with Base Point

Shortcut menu: End any active commands, right-click in the drawing area, and choose Copy with Base Point.

 Command line: copybase

Specify base point:

Select objects:

The selected objects are copied to the Clipboard. Use PASTECLIP to move the copied objects from the Clipboard to a location in the same document or to another document. When you paste an object copied with COPYBASE, it is placed relative to the specified base point.

COPYCLIP

Copies objects or command line text to the Clipboard

 Standard toolbar:

 Edit menu: Copy

Shortcut menu: End any active commands, right-click in the drawing area, and choose Copy.

 Command line: copyclip

Select objects:

COPYCLIP copies all objects you select to the Clipboard. You can paste the contents of the Clipboard into a document or drawing as an OLE object.

Note You can also use CTRL+C to run COPYCLIP. If the cursor is in the drawing area, the selected objects are copied to the Clipboard. If the cursor is on the command line or in the text window, the selected text is copied to the Clipboard.

COPYHIST

Copies the text in the command line history to the Clipboard

Command line: copyhist

The text is copied to the Clipboard.

Tools menu: Customize Interface

Command line: cui

Manages customized user interface elements such as workspaces, toolbars, menus, shortcut menus, and keyboard shortcuts.
The Customize User Interface dialog box is displayed:

COPYLINK

Copies the current view to the Clipboard for linking to other OLE applications
Edit menu: Copy Link

Command line: copylink

COPYLINK copies the current view to the Clipboard. You can paste the contents of the Clipboard into a document as an OLE object.

CUSTOMIZE

Customizes tool palettes
Tools menu: Customize Tool Palettes

Shortcut menu: Right-click any tool palette and choose Customize.

Command line: customize

The Customize dialog box is displayed.

Provides an interface for customizing tool palettes and block authoring palettes.

Creates, modifies, and organizes palettes. Imports and exports palette files.

Palettes

Lists all available palettes. Click and drag a palette to move it up or down in the list. Right-click a palette in the list to rename, delete, or export the palette. (When you export a palette,

it's saved to a file with an *.xtp* extension.) Right-click in the Palettes area to import a palette or to create a new, blank one.

Palette Groups

Displays the organization of your palettes in a tree view. Click and drag a palette to move it into another group. Right-click a palette group, and then click Set Current on the shortcut menu to display that group of palettes.

The shortcut menu also provides options to

- Create a new group
- Delete an existing group
- Rename an existing group
- Remove a tool palette from a group
- Export a group (as an XPG file)
- Import a group

Current Palette Group

Displays the name of the palette group currently shown. Displays All Palettes when all available palettes are shown.

CUTCLIP

Copies objects to the Clipboard and removes the objects from the drawing

Standard toolbar:

Edit menu: Cut

Shortcut menu: End any active commands, right-click in the drawing area, and choose Cut.

Command line: cutclip

Select objects:

CUTCLIP moves the selected objects to the Clipboard, removing them from the drawing. You can paste the contents of the Clipboard into a document or drawing as an embedded OLE object (see PASTECLIP and PASTESPEC). CUTCLIP does not create OLE link information.

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames

Text toolbar:

Modify menu: Object ➤ Text ➤ Edit

Pointing device: Double-click a text object.

Shortcut menu: Select a text object, right-click in the drawing area, and click Edit.

Command line: ddedit

Object Selection

Displays the appropriate editing method for the type of text you select:

- Text created using TEXT or DTEXT displays the In-Place Text Editor without the Text Formatting toolbar and the ruler. Right-click to display options.
- Text created using MTEXT displays the In-Place Text Editor.
- Attribute definitions (not part of a block definition) display the Edit Attribute Definition dialog box.
- Feature control frames display the Geometric Tolerance dialog box.

DDEDIT repeats the prompt until you press ENTER to end the command.

Undo

Returns the text or attribute definition to its previous value. You can use this option immediately after editing.

DDPTYPE

Specifies the display style and size of point objects

Format menu: Point Style

Command line: ddptype (or **'ddptype** for transparent use)

The Point Style dialog box is displayed.

Shows the current point style and size. Change the point style by selecting an icon.

Point Display Images

Specifies the image used to display point objects. The point style is stored in the PDMODE system variable.

Point Size

Sets the point display size. The value you enter can be relative to the screen or in absolute units. The point display size is stored in the PDSIZE system variable. Subsequent point objects that you draw use the new value.

Set Size Relative to Screen

Sets the point display size as a percentage of the screen size. The point display does not change when you zoom in or out.

Set Size in Absolute Units

Sets the point display size as the actual units you specify under Point Size. Points are displayed larger or smaller when you zoom in or out.

DDVPOINT

Sets the three-dimensional viewing direction

◈ **View menu:** 3D Views ➤ Viewpoint Presets

⌨ **Command line: ddvpoint**

The Viewpoint Presets dialog box is displayed.

Defines 3D view settings.

Set Viewing Angles

Sets the direction of the view relative to either the world coordinate system (WCS) or a user coordinate system (UCS).
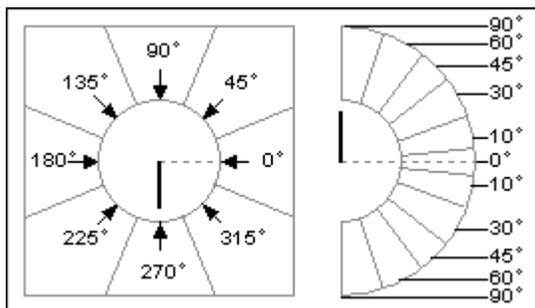
Absolute to WCS

Sets the view direction relative to the WCS.

Relative to UCS

Sets the view direction relative to the current UCS.

From

Specifies viewing angles.

X Axis

Specifies the angle from the *X* axis.

XY Plane

Specifies the angle from the *XY* plane.

You can also use the sample image to specify viewing angles. The black arm indicates the new angle. The gray arm indicates the current angle. Specify an angle by selecting the inner region of the circle or half-circle. Selecting the bounded outer regions rounds off the angle to the value displayed in that region. If you select the inner arc or an area inside it, the angle is not rounded off, and the result may be a fractional number.

Set to Plan View

Sets the viewing angles to display the plan view (*XY* plane) relative to the selected coordinate system.

DETACHURL

Removes hyperlinks in a drawing

⌨ **Command line:** detachurl

> Select objects: *Use an object selection method, and press* ENTER *to end selection*

Hyperlinks are removed from the selected objects. If an area is selected, the polyline is deleted. You can use PURGE to remove the URLLAYER layer.

DIM

Accesses Dimensioning mode

⌨ **Command line:** **dim** or **dim1**

> Dim: *Enter a Dimensioning mode command*

The Dim prompt indicates that you're in Dimensioning mode, in which you can use a special set of **dimensioning commands**. (DIM and DIM1 are provided only for compatibility with previous releases.)
Use DIM to remain in Dimensioning mode after you have used a dimensioning command. Use DIM1 to execute a dimensioning command and immediately return to the Command prompt.
To exit Dimensioning mode, enter **e** or **exit**, or press ESC.

Dimensioning Mode Commands

Dimensioning Mode Commands

The following commands are available at the DIM prompt.

Dimensioning mode commands	
Command	**Description**
EXIT	Exits Dimensioning mode and returns to the Command prompt. You can also press ESC to exit Dimensioning mode.
REDRAW	Redraws the current viewport.
STYLE	Changes the current text style.
UNDO or U	Erases the most recently created dimension objects and cancels any new dimension system variable setting. When you exit Dimensioning mode, UNDO reverses the effects of the entire dimensioning session.

The following table shows which AutoCAD commands are equivalent to the rest of the Dimensioning mode commands. For information about these Dimensioning mode commands, see the equivalent AutoCAD command.

Dimensioning mode command equivalents	
Dimensioning mode command	**Equivalent command**
ALIGNED	DIMALIGNED
ANGULAR	DIMANGULAR
BASELINE	DIMBASELINE
CENTER	DIMCENTER
CONTINUE	DIMCONTINUE
DIAMETER	DIMDIAMETER
HOMETEXT	DIMEDIT Home
HORIZONTAL	DIMLINEAR Horizontal
LEADER	LEADER
NEWTEXT	DIMEDIT New
OBLIQUE	DIMEDIT Oblique
ORDINATE	DIMORDINATE
OVERRIDE	DIMOVERRIDE
RADIUS	DIMRADIUS
RESTORE	-DIMSTYLE Restore
ROTATED	DIMLINEAR Rotated
SAVE	-DIMSTYLE Save
STATUS	-DIMSTYLE Status

TEDIT	DIMTEDIT
TROTATE	DIMEDIT Rotate
UPDATE	-DIMSTYLE Apply
VARIABLES	-DIMSTYLE Variables
VERTICAL	DIMLINEAR Vertical

DIMALIGNED

Creates an aligned linear dimension

 Dimension toolbar:

 Dimension menu: Aligned

 Command line: dimaligned

Specify first **extension line origin** or <**select object**>: *Specify a point for manual extension lines, or press*ENTER*for automatic extension lines*

After you specify either manual or automatic extension lines, the following prompt is displayed:
Specify **dimension line location** or [**Mtext**/**Text**/**Angle**]: *Specify a point or enter an option*

Extension Line Origin

Extension Line Origin

Specifies the first extension line origin (1). You are prompted to specify the second one.
Specify second extension line origin: *Specify a point (2)*

Object Selection

Object Selection

Automatically determines the origin points of the first and second extension lines after you select an object.

Select object to dimension:

For polylines and other explodable objects, only the individual line and arc segments are dimensioned. You cannot select objects in a nonuniformly scaled block reference.

If you select a line or an arc, its endpoints are used as the origins of the extension lines. The extension lines are offset from the endpoints by the distance specified in Offset from Origin on the Lines and Arrows tab of the New, Modify, and Override Dimension Style dialog boxes (see DIMSTYLE). This value is stored in the DIMEXO system variable.
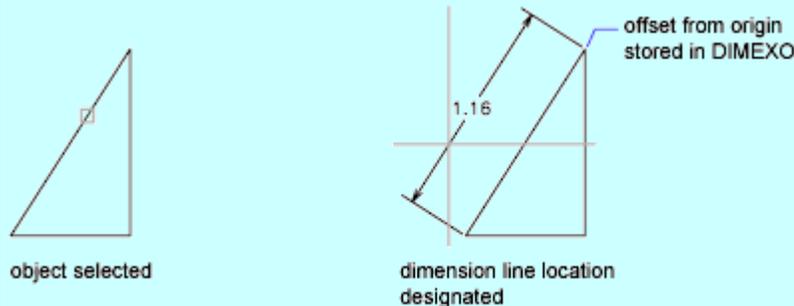
If you select a circle, the endpoints of its diameter are used as the origins of the extension line. The point used to select the circle defines the origin of the first extension line.

Dimension Line Location

Dimension Line Location

Specifies the placement of the dimension line and determines the direction to draw the extension lines. After you specify the location, the DIMALIGNED command ends.

Mtext

Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. The generated measurement is represented with angle brackets (< >). To add a prefix or a suffix to the generated measurement, enter the prefix or suffix before or after the angle brackets. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.

To edit or replace the generated measurement, delete the angle brackets, enter the new dimension text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]).

The current dimension style determines the appearance of the generated measurements.

Text

Text

Customizes the dimension text at the command line. The generated dimension measurement is displayed within angle brackets.

Enter dimension text <*current*>:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

Angle

Angle

Changes the angle of the dimension text.

Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

before angle

After you specify the angle, the Dimension Line Location prompt is redisplayed.
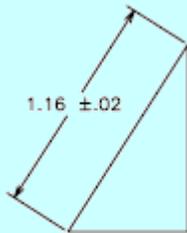
after angle

DIMANGULAR

Creates an angular dimension

 Dimension toolbar:

 Dimension menu: Angular

 Command line: dimangular

> Select **arc**, **circle**, **line**, or <**specify vertex**>: *Select an arc, circle, or line, or press*ENTER*to create the angular dimension by specifying three points*

After you define the angle to dimension, the following prompt is displayed:
> Specify **dimension arc line location** or [**Mtext**/**Text**/**Angle**]:

Arc Selection

Arc Selection

Uses points on the selected arc as the defining points for a three-point angular dimension. The center of the arc is the angle vertex. The arc endpoints become the origin points of the extension lines.

The dimension line is drawn as an arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line.

Circle Selection

Circle Selection

Uses the selection point (1) as the origin of the first extension line. The center of the circle is the angle vertex.
> Specify second angle endpoint: *Specify a point (2)*

The second angle endpoint is the origin of the second extension line and does not have to lie on the circle.
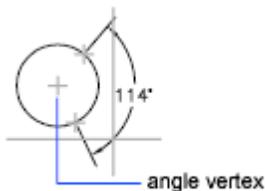
Line Selection

Defines the angle using two lines.

Select second line:

The program determines the angle by using each line as a vector for the angle and the intersection of the lines for the angle vertex. The dimension line spans the angle between the two lines. If the dimension line does not intersect the lines being dimensioned, extension lines are added as needed to extend one or both lines. The arc is always less than 180 degrees.

Three-Point Specification

Creates a dimension based on three points you specify.

Specify angle vertex: *Specify a point (1)*

Specify first angle endpoint: *Specify a point (2)*

Specify second angle endpoint: *Specify a point (3)*

The angle vertex can be the same as one of the angle endpoints. If you need extension lines, the endpoints are used as origin points.
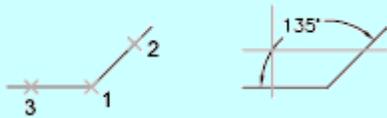
The dimension line is drawn as an arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line.

Dimension Arc Line Location

Specifies the placement of the dimension line and determines the direction to draw the extension lines. After you specify the location, DIMANGULAR ends.

Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]).

The current dimension style determines the appearance of the generated measurements.

Text

Text

Customizes the dimension text at the command line. The generated dimension measurement is displayed within angle brackets.

Enter dimension text *<current>*:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

Angle

Angle

Changes the angle of the dimension text.

Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

DIMARC

Creates an arc length dimension

Dimension toolbar:

Dimension menu: Arc Length

Command line: dimarc

Select arc or polyline arc segment: *Use an object selection method*

Specify arc length dimension location or [Mtext/Text/Angle/Partial/Leader]: Specify a point or enter an option

Arc Length Dimension Location

Arc Length Dimension Location

Specifies the placement of the dimension line and determines the direction of the extension lines.

Mtext

Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([]).

The current dimension style determines the appearance of the generated measurements.

Text

Text

Customizes the dimension text at the command line. The generated dimension measurement is displayed within angle brackets.

Enter dimension text *<current>*:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

Angle

Angle

Changes the angle of the dimension text.

Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

Partial

Partial

Reduces the length of the arc length dimension.

Specify first point for arc length dimension:

Specify a point on the arc where the arc length dimension should begin.

Specify second point for arc length dimension:

Specify a point on the arc where the arc length dimension should end.

Leader

Leader

Adds a leader object. This option is displayed only if the arc (or arc segment) is greater than 90 degrees. The leader is drawn radially, pointing towards the center of the arc being dimensioned.

Specify arc length dimension location or [Mtext/Text/Angle/Partial/No leader]:

Specify a point or enter an option. The leader is created automatically. The No Leader option cancels the Leader option before the leader is created. To remove a leader, delete the arc length dimension and recreate it without the leader option.

DIMBASELINE

Creates a linear, angular, or ordinate dimension from the baseline of the previous dimension or a selected dimension

Dimension toolbar:

Dimension menu: Baseline

Command line: dimbaseline

If no dimension was created in the current session, you are prompted to select a linear, ordinate, or angular dimension to use as the base for the baseline dimension.

> Select base dimension: *Select a linear, ordinate, or angular dimension*

Otherwise, the program skips this prompt and uses the dimension object that was last created in the current session. If the base dimension is linear or angular, the following prompt is displayed:

> Specify a **second extension line origin** or [**Undo**/**Select**] <Select>: *Specify a point, enter an option,*
> *or press* ENTER *to select a base dimension*

If the base dimension is ordinate, the following prompt is displayed:

> Specify **feature location** or [**Undo**/**Select**] <Select>:

To end the command, press ENTER twice, or press ESC. The current dimension style determines the appearance of the text.

Second Extension Line Origin

Second Extension Line Origin

By default, the first extension line of the base dimension is used as the extension line origin for the baseline dimension. To override this default behavior, explicitly select the base dimension; the extension line origin becomes the extension line of the base dimension closest to the pick point of the selection. When you select a second point, the baseline dimension is drawn and the Specify a Second Extension Line Origin prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the baseline dimension, press ENTER.

> Select base dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.

angular baseline dimensioning   linear baseline dimensioning
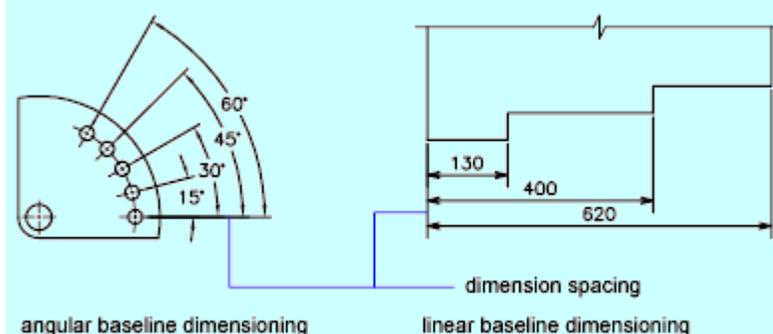
Feature Location

Feature Location

Uses the endpoint of the base dimension as the endpoint for the baseline dimension; you are prompted for the next feature location. When you select a feature location, the baseline dimension is drawn and the Specify Feature Location prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the baseline dimension, press ENTER.

Select base dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.

Undo

Undo

Undoes the last baseline dimension entered during this command session.

Select

Select

Prompts you to select a linear, ordinate, or angular dimension to use as the base for the baseline dimension. After you select a base dimension, the Specify Second Extension Line Origin prompt or the Specify Feature Location prompt is redisplayed.

DIMCENTER

Creates the center mark or the centerlines of circles and arcs

Dimension toolbar:

Dimension menu: Center Mark

Command line: dimcenter

Select arc or circle: *Use an object selection method*

You can choose between center marks and centerlines and specify their size when you set up the dimension style. See DIMSTYLE. You can also change center mark settings using the DIMCEN system variable.
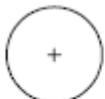
center mark   centerlines

DIMCONTINUE

Creates a linear, angular, or ordinate dimension from the second extension line of the previous dimension or a selected dimension

Dimension toolbar: [icon]

Dimension menu: Continue

Command line: dimcontinue

If no dimension was created in the current session, you are prompted to select a linear, ordinate, or angular dimension to use as the base for the continued dimension.

    Select continued dimension: *Select a linear, ordinate, or angular dimension*

Otherwise, the program skips this prompt and uses the dimension object that was last created in the current session. If the base dimension is linear or angular, the following prompt is displayed:

    Specify a **second extension line origin** or [**Undo**/**Select**] <Select>: *Specify a point, enter an option, or press* ENTER*to select a base dimension*

If the base dimension is ordinate, the following prompt is displayed:

    Specify **feature location** or [Undo/Select] <Select>:

To end the command, press ENTER twice, or press ESC. The current dimension style determines the appearance of the text.
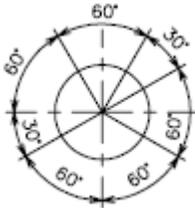
Second Extension Line Origin

Second Extension Line Origin

Uses the second extension line origin of the continued dimension for the first extension line origin of the next dimension. The current dimension style determines the appearance of the text.
After you select a continued dimension, the Specify a Second Extension Line Origin prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the continued dimension, press ENTER.

    Select continued dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.
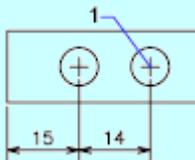
Feature Location

Feature Location

Uses the endpoint of the base dimension as the endpoint for the continued dimension; you are prompted for the next feature location. When you select a feature location, the continued dimension is drawn and the Specify Feature Location prompt is redisplayed. To end the command, press ESC. To select another linear, ordinate, or angular dimension to use as the basis for the continued dimension, press ENTER.

Select continued dimension: *Select a linear, ordinate, or angular dimension*

Select a base dimension, or press ESC to end the command.

Undo

Undo

Undoes the last continued dimension entered during the command session.

Select

Select

Prompts you to select a linear, ordinate, or angular dimension to use as the continued dimension. After you select a continued dimension, the Specify a Second Extension Line Origin prompt or the Specify Feature Location prompt is redisplayed. To end the command, press ESC.

DIMDIAMETER

Creates diameter dimensions for circles and arcs

Dimension toolbar:

Dimension menu: Diameter

Command line: dimdiameter

Select arc or circle:

DIMDIAMETER measures the diameter of the selected circle or arc and displays the dimension text with a diameter symbol in front of it.

Specify **dimension line location** or [**Mtext**/**Text**/**Angle**]: *Specify a point or enter an option*

Dimension Line Location

Dimension Line Location

Determines the angle of the dimension line and the location of the dimension text.

Mtext

Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([]).

The current dimension style determines the appearance of the dimension text.

Text

Text

Customizes the dimension text at the command line. The generated dimension measurement is displayed within angle brackets.

    Enter dimension text *<current>*:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

Angle

Angle

Changes the angle of the dimension text.

    Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

After you specify the angle, the Dimension Line Location prompt is redisplayed.

DIMDISASSOCIATE

Removes associativity from selected dimensions

⌨ **Command line:** dimdisassociate

    Select dimensions to disassociate: *Select one or more dimensions and then press* ENTER

DIMDISASSOCIATE filters the selection set to include only associative dimensions that are not on locked layers, and that are not in a different space than the current space (for example, if model space is active, associative dimensions in paper space are excluded). DIMDISASSOCIATE then disassociates these dimensions and reports the number of dimensions that are filtered out and the number that are disassociated.

DIMEDIT

Edits dimension text and extension lines on dimension objects

Dimension toolbar:

Command line: dimedit

Enter type of dimension editing [Home/New /Rotate/Oblique] <Home>: *Enter an option or press*ENTER

Home

Moves rotated dimension text back to its default position.

Select objects: *Use an object selection method to select dimension objects*
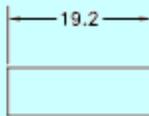
before Home   after Home

The selected dimension text is returned to its default position and rotation as specified in its dimension style.

New

Changes dimension text using the In-Place Text Editor.
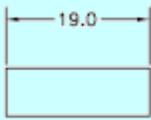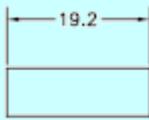
before New   after New

The generated measurement is represented with angle brackets (< >). To add a prefix or a suffix to the generated measurement, enter the prefix or suffix before or after the angle brackets. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.

To edit or replace the generated measurement, delete the angle brackets, enter the new dimension text, and then choose OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]).

Rotate

Rotates dimension text. This option is similar to the Angle option of DIMTEDIT.

Enter text angle:

Entering **0** places the text in its default orientation, which is determined by the vertical and horizontal text settings on the Text tab of the New, Modify, and Override Dimension Style dialog boxes. See DIMSTYLE. The DIMTIH and DIMTOH system variables control this orientation.

Select objects: *Use an object selection method to select dimension objects*
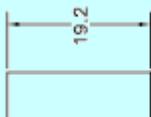
rotated text

Adjusts the oblique angle of the extension lines for linear dimensions. Linear dimensions are created with extension lines perpendicular to the direction of the dimension line. The Oblique option is useful when extension lines conflict with other features of the drawing.

Select objects: *Use an object selection method to select dimension objects*

Enter oblique angle (press ENTER for none): *Enter an angle or press* ENTER
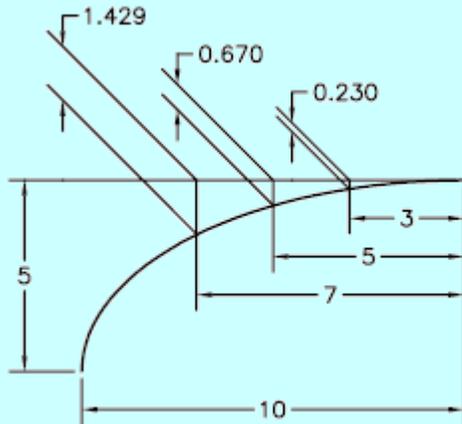
DIMLINEAR

Creates linear dimensions

 Dimension toolbar:

 Dimension menu: Linear

 Command line: dimlinear

Specify first **extension line origin** or <**select object**>: *Specify a point or press* ENTER *to select an object to dimension*

After you specify the extension line origin points or the object to dimension, the following prompt is displayed:

Specify **dimension line location** or [**Mtext**/**Text**/**Angle**/**Horizontal**/**Vertical**/**Rotated**]: *Specify a point or enter an option*

First Extension Line Origin

First Extension Line Origin

Prompts for the origin point of the second extension line after you specify the origin point of the first.

Specify second extension line origin: *Specify a point (2)*

point specification

Dimension Line Location

Dimension Line Location

Uses the point you specify to locate the dimension line and determines the direction to draw the extension lines. After you specify the location, the dimension is drawn.

dimension line location
designated

Mtext

Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.
To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]).
The current dimension style determines the appearance of the generated measurements.

Text

Text

Customizes the dimension text on the command line. The generated dimension measurement is displayed within angle brackets.

Enter dimension text *<current>*:

Enter the dimension text, or press ENTER to accept the generated measurement. To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).
Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

Angle

Changes the angle of the dimension text.
    Specify angle of dimension text:

Enter an angle. For example, to rotate the text 90 degrees, enter **90**.
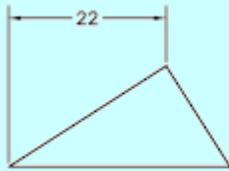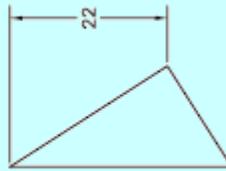
before Angle   after Angle

Horizontal

Creates horizontal linear dimensions.
    Specify dimension line location or [Mtext/Text/Angle]: *Specify a point or enter an option*
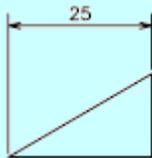
Dimension Line Location

Uses the point you specify to locate the dimension line. After you specify the location, the dimension is drawn.

Mtext, Text, Angle

These text editing and formatting options are identical in all dimension commands. See the option descriptions provided earlier in this command.

Vertical

Creates vertical linear dimensions.

Specify dimension line location or [Mtext/Text/Angle]: *Specify a point or enter an option*
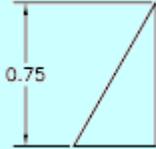
Dimension Line Location

Uses the point you specify to locate the dimension line. After you specify the location, the dimention is drawn.

Mtext, Text, Angle

These text editing and formatting options are identical in all dimension commands. See the option descriptions provided earlier in this command.

Rotated

Rotated

Creates rotated linear dimensions.

Specify angle of dimension line *<current>*: *Specify an angle or press* ENTER

Object Selection

Object Selection

Automatically determines the origin points of the first and second extension lines after you select an object.

Select object to dimension:

For polylines and other explodable objects, only the individual line and arc segments are dimensioned. You cannot select objects in a non-uniformly scaled block reference.

If you select a line or an arc, the line or arc endpoints are used as the origins of the extension lines. The extension lines are offset from the endpoints by the distance you specify in Offset from Origin in the Lines and Arrows tab of the New, Modify, and Override Dimension Style dialog boxes. See DIMSTYLE. This value is stored in the DIMEXO system variable.

object selected

dimension line
location designated

offset from origin
(DIMEXO system
variable)
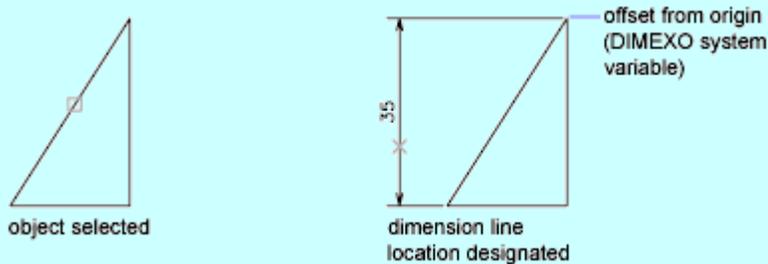
If you select a circle, the diameter endpoints are used as the origins of the extension line. When the point used to select the circle is close to the north or south quadrant point, a horizontal dimension is drawn. When the point used to select the circle is close to the east or west quadrant point, a vertical dimension is drawn.

DIMORDINATE

Creates ordinate point dimensions
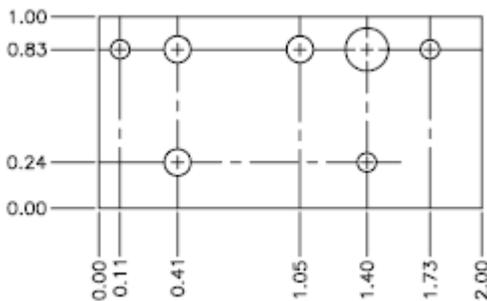
🖏 **Dimension toolbar:**

🖏 **Dimension menu:** Ordinate

⌨ **Command line: dimordinate**

Specify feature location: *Specify a point or snap to an object*

Specify **leader endpoint** or [**Xdatum**/**Ydatum**/**Mtext**/**Text**/**Angle**]: *Specify a point or enter an option*

Leader Endpoint Specification

Uses the difference between the feature location and the leader endpoint to determine whether it is an *X* or a *Y* ordinate dimension. If the difference in the *Y* ordinate is greater, the dimension

measures the *X* ordinate. Otherwise, it measures the *Y* ordinate.

Xdatum

Measures the *X* ordinate and determines the orientation of the leader line and dimension text. The Leader Endpoint prompt is displayed, where you can specify the endpoint.
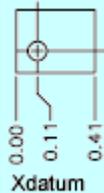
Xdatum

Ydatum

Measures the *Y* ordinate and determines the orientation of the leader line and dimension text. The Leader Endpoint prompts are displayed, where you can specify the endpoint.
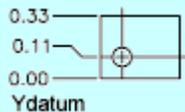
Ydatum

Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.

To edit or replace the generated measurement, delete the text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]).

The current dimension style determines the appearance of the generated measurements. After you choose OK, the Leader Endpoint prompt is redisplayed.

Text

Customizes the dimension text on the command line. The generated dimension measurement is displayed within angle brackets.

  Enter dimension text *<current>*: *Enter the dimension text, or press*ENTER to accept the generated measurement

To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes. After you press ENTER, the Leader Endpoint prompt is redisplayed.

Angle

Changes the angle of the dimension text.

  Specify angle of dimension text:

Enter an angle. For example, to rotate the text 45 degrees, enter **45**.

After you specify the angle, the Leader Endpoint prompt is redisplayed.

DIMOVERRIDE

Overrides dimensioning system variables

Dimension menu: Override

Command line: dimoverride

Enter **dimension variable name to override** or [**Clear overrides**]: *Enter the name of a dimension variable, or enter* c

Dimension Variable Name to Override

Overrides the value of the dimensioning system variable you specify.

Enter new value for dimension variable *<current>*: *Enter a value or press*ENTER

If you enter a new value, the Dimension Variable Name to Override prompt is redisplayed. If you press ENTER, you are prompted to select the dimensions.

Select objects: *Use an object selection method to select the dimensions*

The overrides to the selected dimensions are applied.

Clear Overrides

Clears any overrides on selected dimensions.

Select objects: *Use an object selection method to select the dimensions*

The overrides are cleared, and the dimension objects return to the settings defined by their dimension style.

DIMRADIUS

Creates radial dimensions for circles and arcs

Dimension toolbar:

Dimension menu: Radius

Command line: dimradius

Select arc or circle:

DIMRADIUS measures the radius of the selected arc or circle and displays the dimension text with a radius symbol in front of it.

Specify **dimension line location** or [**Mtext**/**Text**/**Angle**]: *Specify a point or enter an option*

Dimension Line Location

Determines the angle of the dimension line and the location of the dimension text.

Mtext

Displays the In-Place Text Editor, which you can use to edit the dimension text. To add a prefix or

a suffix, enter the prefix or suffix text before or after the generated measurement. Use control codes and Unicode character strings to enter special characters or symbols. See Control Codes and Special Characters.

To edit or replace the generated measurement, delete text, enter the new text, and then click OK. If alternate units are not turned on in the dimension style, you can display them by entering square brackets ([ ]).

The current dimension style determines the appearance of the generated measurements.

Text

Customizes the dimension text on the command line. The generated dimension measurement is displayed within angle brackets.

Enter dimension text *<current>*: *Enter the dimension text, or press* ENTER *to accept the generated measurement*

To include the generated measurement, use angle brackets (< >) to represent the generated measurement. If alternate units are not turned on in the dimension style, you can display alternate units by entering square brackets ([ ]).

Dimension text properties are set on the Text tab of the New, Modify, and Override Dimension Style dialog boxes.

Angle

Changes the angle of the dimension text.

Specify angle of dimension text: *Enter an angle*

For example, to rotate the text 45 degrees, enter **45**.

DIMREASSOCIATE

Associates selected dimensions to geometric objects

Dimension menu: Reassociate Dimensions

Command line: dimreassociate

Select dimensions to reassociate: *Select dimension objects*

Each selected dimension is highlighted in turn, and prompts for association points appropriate for the selected dimension are displayed. A marker is displayed for each association point prompt. If the definition point of the current dimension is not associated to a geometric object, the marker appears as an X; if the definition point is associated, the marker appears as an X inside a box.

Note The marker disappears if you pan or zoom with a wheel mouse.

Press ESC to terminate the command without losing the changes that were already specified. Use UNDO to restore the previous state of the changed dimensions.
The prompts for the different types of dimensions are:

Linear
Specify first extension line origin or [Select object] <next>: *Specify an object snap location, enter* **s** *and select a geometric object, or press* ENTER*to skip to the next prompt*

Specify second extension line origin <next>: *Specify an object snap location, or press* ENTER*to skip to the next dimension object, if any*

Aligned

Specify first extension line origin or [Select object] <next>: *Specify an object snap location, enter **s** and select a geometric object, or press* ENTER*to skip to the next prompt*

Specify second extension line origin <next>: *Specify an object snap location, or press* ENTER*to skip to the next dimension object, if any*

Angular (Three Point)

Specify angle vertex or [Select arc or circle] <next>: *Specify an object snap location, enter **s** and select an arc or a circle, or press* ENTER*to skip to the next prompt*

Specify first angle endpoint <next>: *Specify an object snap location or press* ENTER*to skip to the next prompt*

Specify second angle endpoint <next>: *Specify an object snap location or press* ENTER*to skip to the next dimension object, if any*

Angular (Two Line)

Select first line <next>: *Select a line, or press* ENTER*to skip to the next prompt*

Select second line <next>: *Select another line, or press* ENTER*to skip to the next dimension object, if any*

Diameter

Select arc or circle <next>: *Select an arc or a circle, or press* ENTER*to skip to the next dimension object, if any*

Leader

Specify leader association point <next>: *Specify an object snap location, or press* ENTER*to skip to the next dimension object, if any*

Ordinate

Specify feature location <next>: *Specify an object snap location, or press* ENTER*to skip to the next dimension object, if any*

Radius

Select arc or circle <next>: *Select an arc or a circle, or press* ENTER*to skip to the next dimension object, if any*

Note DIMREASSOCIATE does not change the setting of DIMLFAC in a dimension. Use DIMOVERRIDE to clear dimension linear factors in legacy drawings.

DIMREGEN

Updates the locations of all associative dimensions

⌨ **Command line:** dimregen

The locations of all associative dimensions in the current drawing are updated.
Associative dimensions need to be updated manually with DIMREGEN in three cases:

- After panning or zooming with a wheel mouse in a layout with model space active, update associative dimensions created in paper space.
- After opening a drawing that has been modified with a previous version of the program , update associative dimensions if the dimensioned objects have been modified.
- After opening a drawing containing external references that are dimensioned in the current drawing, update associative dimensions if the associated external reference geometry has been modified.

DIMSTYLE

 Styles toolbar:

 Format menu: Dimension Style

 Dimension menu: Style

 Command line: dimstyle

Creates new styles, sets the current style, modifies styles, sets overrides on the current style, and compares styles.

Current Dimstyle

Displays the name of the dimension style that is current. The default dimension style is STANDARD. The current style is applied to dimensions you create.

Styles

Lists dimension styles in the drawing. The current style is highlighted. Right-click in the list to display a shortcut menu with options to set the current style, rename styles, and delete styles. You cannot delete a style that is current or in use in the current drawing.
The item selected in List controls the dimension styles displayed. To make a style current, select it and click Set Current.
Unless you select Don't List Styles in Xrefs, dimension styles are displayed in externally referenced drawings using the syntax for externally referenced named objects. Although you cannot change, rename, or make current externally referenced dimension styles, you can create new styles based on them.

List

Controls the display of styles in the Styles list. Select All Styles if you want to see all dimension styles in a drawing. Select Styles in Use if you want to see only the dimension styles currently used by dimensions in the drawing.

Don't List Styles in Xrefs

When selected, suppresses display of dimension styles in externally referenced drawings in the Styles list.

Preview

Shows a graphic representation of the style selected in the Styles list.

Description

Describes the style selected in the Styles list relative to the current style. If the description is longer than the space provided, you can click in the pane and use arrow keys to scroll down.

Set Current

Sets the style selected under Styles to current. The current style is applied to dimensions you create.

New

Displays the Create New Dimension Style dialog box, in which you can define a new dimension style.

Modify

Displays the Modify Dimension Styles dialog box, in which you can modify dimension styles. Dialog box options are identical to those in the New Dimension Style dialog box.

Override

Displays the Override Current Style dialog box, in which you can set temporary overrides to dimension styles. Dialog box options are identical to those in the New Dimension Style dialog box. Overrides are displayed as unsaved changes under the dimension style in the Styles list.

Compare

Displays the Compare Dimension Styles dialog box, in which you can compare two dimension styles or list all the properties of one dimension style.

DIMTEDET

Moves and rotates dimension text

📎 **Dimension toolbar:**

📎 **Dimension menu:** Align Text

⌨ **Command line:** dimtedit

Select dimension: *Select a dimension object*

You are prompted for the new location of the dimension text.

Specify **new location for dimension text** or [**Left**/**Right**/**Center**/**Home**/**Angle**]: *Specify a point or enter an option*
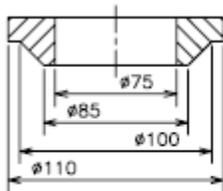
dimension text with left
and right justification
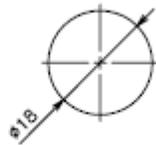
dimension text
positioned by cursor
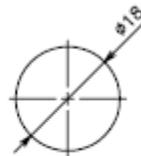
dimension text
positioned by angle

Location for Dimension Text

Updates the location of the dimension text dynamically as you drag it. To determine whether text appears above, below, or in the middle of the dimension line, use the Text tab in the New, Modify, and Override Dimension Style dialog box.

Left

Left-justifies the dimension text along the dimension line. This option works only with linear, radial, and diameter dimensions.
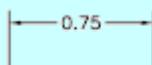
before Left
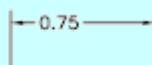
after Left

Right

Right-justifies the dimension text along the dimension line. This option works only with linear, radial, and diameter dimensions.
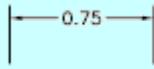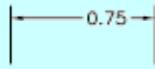
before Right   after Right

Center

Centers the dimension text on the dimension line.

Home

Moves the dimension text back to its default position.
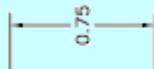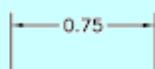
before Home   after Home

Angle

Changes the angle of the dimension text.

 Enter text angle:
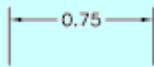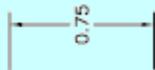
before Angle   after Angle 90 deg

The center point of the text does not change. If the text moves or the dimension is regenerated, the orientation set by the text angle is retained. Entering an angle of 0 degrees puts the text in its default orientation.

DIST

Measures the distance and angle between two points

Inquiry toolbar:

Tools menu: Inquiry ➤ Distance

Command line: dist (or **'dist** for transparent use)

Specify first point: *Specify a point*
Specify second point: *Specify a point*
Distance = *calculated distance*, Angle in XY plane = *angle*,
Angle from XY plane = *angle*
Delta X = *change in* X, Delta Y = *change in* Y, Delta Z = *change in* Z

The true 3D distance between points is reported. The angle in the *XY* plane is relative to the current *X* axis. The angle from the *XY* plane is relative to the current *XY* plane. DIST assumes the current elevation for the first or second point if you omit the *Z* coordinate value.
The distance is displayed using the current units format.
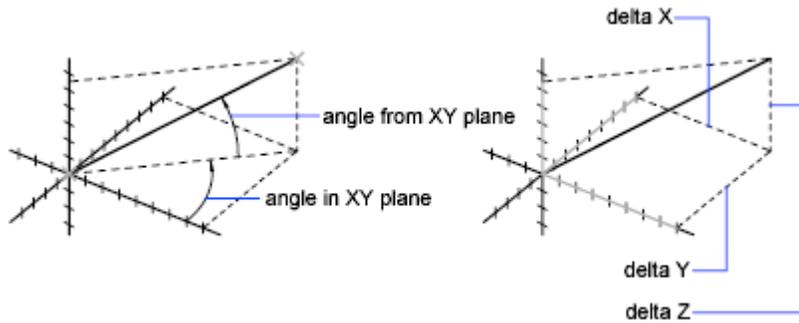
DIVIDE

Places evenly spaced point objects or blocks along the length or perimeter of an object

⊗ **Draw menu:** Point ➤ Divide

⌨ **Command line: divide**

> Select object to divide: *Use an object selection method*
> Enter <u>number of segments</u> or [Block]: *Enter a value from 2 through 32,767, or enter* **b**

Number of Segments

Number of Segments

Places point objects at equal intervals along the selected objects.
The illustration shows a polyline divided into five parts. Point Display mode (PDMODE) has been set such that the points can be seen.
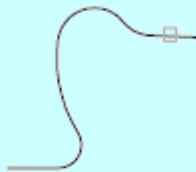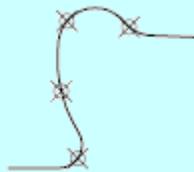
select polyline   divided into five parts

Block

Block

Places blocks at equal intervals along the selected object.

    Enter name of block to insert: *Enter the name of a block currently defined in the drawing*

    Align block with object? [Yes/No] <Y>: *Enter **y** or **n** or press* ENTER

Yes

Specifies that the *X* axes of the inserted blocks be tangent to, or collinear with, the divided object at the dividing points.

No

Aligns the blocks according to their normal orientation.

    Enter number of segments: *Enter a value from 2 through 32,767*

The illustration shows an arc divided into five equal parts using a block consisting of a vertically oriented ellipse.
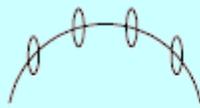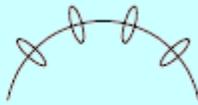
block not aligned   block aligned

DONUT

Draws filled circles and rings

Draw menu: Donut

Command line: donut

    Specify inside diameter of donut *<current>*: *Specify a distance or press* ENTER

If you specify an inside diameter of 0, the donut is a filled circle.

    Specify outside diameter of donut *<current>*: *Specify a distance or press* ENTER

    Specify center of donut or <exit>: *Specify a point (1) or press* ENTER *to end the command*

inside
diameter

outside
diameter
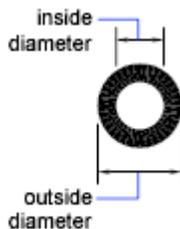
The location of the donut is set based on the center point. After you specify the diameters, you are prompted for the locations at which to draw donuts. A donut is drawn at each point specified (2). How the interior of a donut is filled depends on the current setting of the FILL command.
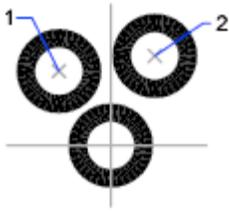
DRAGMODE

Controls the way dragged objects are displayed

⌨ **Command line: dragmode** (or **'dragmode** for transparent use)

Enter new value [<u>ON</u>/<u>OFF</u>/<u>Auto</u>] <*current*>: *Enter an option or press*ENTER

On
Permits dragging, but you must enter **drag** where appropriate in a drawing or editing command to initiate dragging.
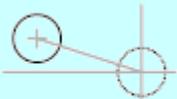
Off
Ignores all dragging requests, including those embedded in menu items.
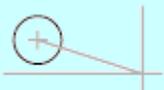
Auto
Turns on dragging for every command that supports it. Dragging is performed whenever it is possible. Entering **drag** each time is not necessary.

DRAWORDER

Changes the draw order of images and other objects

 Draw Order toolbar:

 Tools menu: Draw Order

Shortcut menu: Select an object, right-click, and then click Draw Order.

⌨ **Command line: draworder**

Select objects: *Use an object selection method*

Enter object ordering option [**Above objects**/**Under objects**/**Front**/**Back**] <Back>: *Enter an option or press* ENTER

Above Objects
Moves the selected object above the specified reference objects.
Select reference objects: *Use an object selection method*

Under Objects
Moves the selected objects below the specified reference objects.
Select reference objects: *Use an object selection method*

Front
Moves the selected objects to the top of the order of objects in the drawing.

Back
Moves the selected objects to the bottom of the order of objects in the drawing.

When you change the draw order (display and plotting order) of multiple objects, the relative draw order of the selected objects is maintained.

By default, when you create new objects from existing ones (for example, FILLET or PEDIT), the new objects are assigned the draw order of the original object you selected first. By default, while you edit an object (for example, MOVE or STRETCH), the object is displayed on top of all objects in the drawing. When you are finished editing, your drawing is partially regenerated so that the object is displayed according to its correct draw order. This can result in some edit operations taking slightly longer. You can use DRAWORDERCTL to change the default draw order settings. TEXTTOFRONT changes the draw order of all text and dimensions in the drawing.

DSETTINGS

Specifies settings for Snap mode, grid, polar and object snap tracking, and Dynamic Input

  Tools menu: Drafting Settings

    Shortcut menu: Right-click Snap, Grid, Polar, Osnap, Otrack, or Dyn on the status bar. Click Settings.

  Command line: dsettings (or **'dsettings** for transparent use)

The Drafting Settings dialog box is displayed.

Specifies drafting settings organized for drawing aids in four categories: **Snap and Grid**, **Polar Tracking**, **Object Snap**, and **Dynamic Input**.

Options

Displays the Drafting tab in the Options dialog box. You cannot access the Options dialog box from the Drafting Settings dialog box if you are running DSETTINGS transparently.

Snap and Grid Tab (Drafting Settings Dialog Box)

Snap and Grid Tab (Drafting Settings Dialog Box)

Specifies Snap and Grid settings.

Snap On

Turns Snap mode on or off. You can also turn Snap mode on or off by clicking Snap on the status bar, by pressing F9, or by using the SNAPMODE system variable.

Snap

Controls an invisible grid that restricts cursor movement to specified intervals.

Snap X Spacing

Specifies the snap spacing in the *X* direction. The value must be a positive real number. (SNAPUNIT system variable)

Snap Y Spacing

Specifies the snap spacing in the *Y* direction. The value must be a positive real number. (SNAPUNIT system variable)

Angle

Rotates the snap grid by the angle specified. (SNAPANG system variable)

X Base

Specifies an *X* base coordinate point for the grid. (SNAPBASE system variable)

Y Base

Specifies a *Y* base coordinate point for the grid. (SNAPBASE system variable)

Polar Spacing

Controls the PolarSnap™ increment distance.

Polar Distance

Sets the snap increment distance when PolarSnap is selected under Snap Type & Style. If this value is 0, the PolarSnap distance assumes the value for Snap X Spacing. The Polar Distance setting is used in conjunction with polar tracking and/or object snap tracking. If neither tracking feature is enabled, the Polar Distance setting has no effect. (POLARDIST system variable)

Grid On

Turns the grid dots on or off. You can also turn grid dots mode on or off by clicking Grid on the status bar, by pressing F7, or by using the GRIDMODE system variable.

Grid

Controls the display of a dot grid that helps you visualize distances.

Note The limits of the dot grid are controlled by the LIMITS command.

Grid X Spacing

Specifies the dot spacing in the *X* direction. If this value is 0, the grid assumes the value set for Snap X Spacing. (GRIDUNIT system variable)

Grid Y Spacing

Specifies the dot spacing in the *Y* direction. If this value is 0, the grid assumes the value set for Snap Y Spacing. (GRIDUNIT system variable)

Snap Type & Style

Controls Snap mode settings.

Grid Snap

Sets the snap type to Grid. When you specify points, the cursor snaps along vertical or horizontal grid points. (SNAPTYPE system variable)

Rectangular Snap: Sets the snap style to standard Rectangular snap mode. When the snap type is set to Grid snap and Snap mode is on, the cursor snaps to a rectangular snap grid. (SNAPSTYL system variable)

Isometric Snap: Sets the snap style to Isometric snap mode. When the snap type is set to Grid snap and Snap mode is on, the cursor snaps to an isometric snap grid. (SNAPSTYL system variable)

PolarSnap

Sets the snap type to Polar. When Snap mode is on and you specify points with polar tracking turned on, the cursor snaps along polar alignment angles set on the Polar Tracking tab relative to

the starting polar tracking point. (SNAPTYPE system variable)

Polar Tracking Tab (Drafting Settings Dialog Box)

Polar Tracking Tab (Drafting Settings Dialog Box)

Controls the AutoTrack settings.

Polar Tracking On

Turns polar tracking on and off. You can also turn polar tracking on or off by pressing F10 or by using the AUTOSNAP system variable.

Polar Angle Settings

Sets the alignment angles for polar tracking. (POLARANG system variable)

Increment Angle

Sets the polar increment angle used to display polar tracking alignment paths. You can enter any angle, or select a common angle of 90, 45, 30, 22.5, 18, 15, 10, or 5 degrees from the list. (POLARANG system variable)

Additional Angles

Makes any additional angles in the list available for polar tracking. The Additional Angles check box is also controlled by the POLARMODE system variable, and the list of additional angles is also controlled by the POLARADDANG system variable.

Note Additional angles are absolute, not incremental.

List of Angles

If Additional Angles is selected, lists the additional angles that are available. To add new angles, click New. To remove existing angles, click Delete. (POLARADDANG system variable)

New

Adds up to 10 additional polar tracking alignment angles.

Note Before adding fractional angles, you must set the AUPREC system variable to the appropriate decimal precision to avoid undesired rounding. For example, if the value of AUPREC is 0 (the default value), all fractional angles you enter are rounded to the nearest whole number.

Delete

Deletes selected additional angles.

Object Snap Tracking Settings

Sets options for object snap tracking.

Track Orthogonally Only

Displays only orthogonal (horizontal/vertical) object snap tracking paths for acquired object snap points when object snap tracking is on. (POLARMODE system variable)

Track Using All Polar Angle Settings

Applies polar tracking settings to object snap tracking. When you use object snap tracking, the cursor tracks along polar alignment angles from acquired object snap points. (POLARMODE system variable)

Note Clicking Polar and Otrack on the status bar also turns polar tracking and object snap tracking on and off.

Polar Angle Measurement

Sets the basis by which polar tracking alignment angles are measured.

Absolute

Bases polar tracking angles on the current user coordinate system (UCS).

Relative to Last Segment

Bases polar tracking angles on the last segment drawn.

Object Snap Tab (Drafting Settings Dialog Box)

Object Snap Tab (Drafting Settings Dialog Box)

Controls running object snap settings. With running object snap settings, also called Osnap, you can specify a snap point at an exact location on an object. When more than one option is selected, the selected snap modes are applied to return a point closest to the center of the aperture box.

Press TAB to cycle through the options.

Object Snap On

Turns running object snaps on and off. The object snaps selected under Object Snap Modes are active while object snap is on. (OSMODE system variable)

Object Snap Tracking On

Turns object snap tracking on and off. With object snap tracking, the cursor can track along alignment paths based on other object snap points when specifying points in a command. To use object snap tracking, you must turn on one or more object snaps. (AUTOSNAP system variable)

Object Snap Modes

Lists object snaps that you can turn on as running object snaps.

Endpoint

Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray, or to the closest corner of a trace, solid, or 3D face.
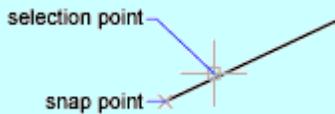
Midpoint

Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, region, solid, spline, or xline.
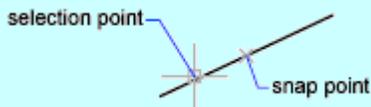
Center

Snaps to the center of an arc, circle, ellipse, or elliptical arc.
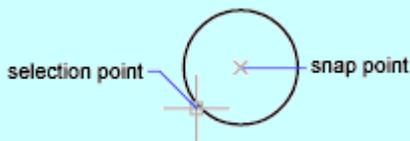
Node

Snaps to a point object, dimension definition point, or dimension text origin.

Quadrant

Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.
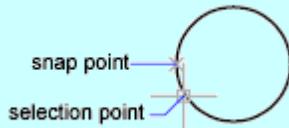
Intersection

Snaps to the intersection of an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, spline, or xline. Extended Intersection is not available as a running object snap.

Intersection and Extended Intersection do not work with edges or corners of 3D solids.

Note You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time.

Extension

Causes a temporary extension line or arc to be displayed when you pass the cursor over the endpoint of objects, so you can specify points on the extension.

Insertion

Snaps to the insertion point of an attribute, a block, a shape, or text.

Perpendicular

Snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, solid, spline, or xline. Deferred Perpendicular snap mode is automatically turned on when the object you are drawing requires that you complete more than one perpendicular snap. You can use a line, arc, circle, polyline, ray, xline, multiline, or 3D solid edge as an object from which to draw a perpendicular line. You can use Deferred Perpendicular to draw perpendicular lines between such objects. When the aperture box passes over a Deferred Perpendicular snap point, an AutoSnap tooltip and marker are displayed.

Tangent

Snaps to the tangent of an arc, circle, ellipse, elliptical arc, or spline. Deferred Tangent snap mode is automatically turned on when the object you are drawing requires that you complete more than one tangent snap. For example, you can use Deferred Tangent to draw a line that is tangent to two arcs, polyline arcs, or circles. When the aperture box passes over a Deferred Tangent snap point, a marker and an AutoSnap tooltip are displayed.

Note When you use the From option in conjunction with the Tangent snap mode to draw objects other than lines from arcs or circles, the first point drawn is tangent to the arc or circle in relation to the last point selected in the drawing area.

Nearest

Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, ray, spline, or xline.

Apparent Intersection

Snaps to the visual intersection of two objects that are not in the same plane but may appear to intersect in the current view. Extended Apparent Intersection is not available as a running object snap. Apparent and Extended Apparent Intersection do not work with edges or corners of 3D solids.

Note You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time.

Parallel

Draws a vector parallel to another object whenever you are prompted for the second point of a vector. After specifying the first point of a vector, if you move the cursor over a straight line segment of another object, the point is acquired. When the path of the object you create is parallel to the line segment, an alignment path is displayed, which you can use to create the parallel object.

Select All

Turns on all object snap modes.

Clear All

Turns off all object snap modes.

Dynamic Input Tab (Drafting Settings Dialog Box)

Dynamic Input Tab (Drafting Settings Dialog Box)

Controls pointer input, dimension input, dynamic prompting, and the appearance of drafting tooltips.

Enable Pointer Input

Turns on pointer iput. When pointer input and dimensional input are both turned on, dimensional input supersedes pointer input when it is available. (DYNMODE system variable)

Pointer Input

Displays the location of the crosshairs as coordinate values in a tooltip near the cursor. When a command prompts for a point, you can enter coordinate values in the tooltip instead of on the command line.

Preview Area

Shows an example of pointer input.

Settings

Displays the Pointer Input Settings dialog box.

Enable Dimension Input

Turns on dimensional input. Dimensional input is not available for some commands that prompt for a second point. (DYNMODE system variable)

Dimension Input

Displays a dimension with tooltips for distance value and angle value when a command prompts for a second point or a distance. The values in the dimension tooltips change as you move the cursor. You can enter values in the tooltip instead of on the command line.

Preview Area

Shows an example of dimensional input.

Settings

Displays the Dimension Input Settings dialog box.

Dynamic Prompts

Displays prompts in a tooltip near the cursor when necessary in order to complete the command. You can enter values in the tooltip instead of on the command line.

Preview Area

Shows an example of dynamic prompts.

Show Command Prompting and Command Input near the Crosshairs

Displays prompts in Dynamic Input tooltips. (DYNPROMPT system variable)

Drafting Tooltip Appearance

Displays the Tooltip Appearance dialog box.

DSVIEWER

The Aerial View window is displayed.

Displays the entire drawing; the current view is marked with a wide outline box.

View Menu (Aerial View Window)

Changes the magnification of the Aerial View by zooming in and out of the drawing or by displaying the entire drawing in the Aerial View window.
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. If both of these conditions exist at the same time, such as after using ZOOM Extents, both options are unavailable. All of the menu options are also available from a shortcut menu you can access by right-clicking in the Aerial View window.

Zoom In

Increases the magnification of the drawing in the Aerial View window by zooming in by a factor of 2, centered on the current view box.

Zoom Out

Decreases the magnification of the drawing in the Aerial View window by zooming out by a factor of 2, centered on the current view box.

Global

Displays the entire drawing and the current view in the Aerial View window.

Options Menu (Aerial View Window)

Provides toggles for automatic viewport display and dynamic updating of the drawing. All of the menu options are also available from a shortcut menu you can access by right-clicking in the Aerial View window.

Auto Viewport

Displays the model space view of the current viewport automatically when multiple viewports are displayed. When Auto Viewport is off, the Aerial View window is not updated to match the current viewport.

Dynamic Update

Updates the Aerial View window while you edit the drawing. When Dynamic Update is off, the Aerial View window is not updated until you click in the Aerial View window.

Realtime Zoom

Updates the drawing area in real time when you zoom using the Aerial View window.

DWGPROPS

Sets and displays the properties of the current drawing

⊗ **File menu:** Drawing Properties

⌨ **Command line:** dwgprops

The Drawing Properties dialog box is displayed

Displays read-only statistics or general information about your drawing, assigns summary properties, and assigns names and values to custom properties. These custom properties can help you identify your drawing.

Document properties are displayed in the list of field names in the Field dialog box.

DXBIN

Imports specially coded binary files

⊗ **Insert menu:** Drawing Exchange Binary

⌨ **Command line:** dxbin

The Select DXB File dialog box (a standard file selection dialog box) is displayed. Enter the name of the file to import.

ELEV

Sets elevation and extrusion thickness of new objects

⌨ **Command line:** **elev** (or **'elev** for transparent use)

Specify new default elevation *<current>*: *Specify a distance or press* ENTER

The current elevation is the default *Z* value for new objects when you specify only *X* and *Y* values for a 3D point.
The elevation setting is the same for all viewports regardless of their user coordinate systems (UCSs). New objects are created at the specified *Z* value relative to the current UCS in the viewport.

Specify new default thickness *<current>*: *Specify a distance or press* ENTER

The thickness sets the distance to which a 2D object is extruded above or below its elevation. A positive value is extruded along the positive *Z* axis; a negative value is extruded along the negative *Z* axis.

with elevation   with thickness
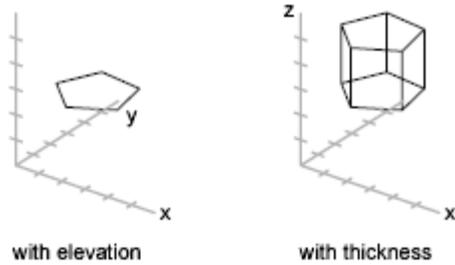
ELEV controls only new objects; it does not affect existing objects. The elevation is reset to 0.0 whenever you change the coordinate system to the world coordinate system (WCS).

ELLIPSE

Creates an ellipse or an elliptical arc

Draw toolbar:

Draw menu: Ellipse

Command line: ellipse

Specify **axis endpoint** of ellipse or [**Arc**/**Center**/**Isocircle**]: *Specify a point or enter an option*
The Isocircle option is available only when you set the Style option of SNAP to Isometric.

ERASE

Removes objects from a drawing

Modify toolbar:

Modify menu: Erase

Shortcut menu: Select the objects to erase, right-click in the drawing area, and click Erase.

Command line: erase

Select objects: *Use an object selection method and press*ENTER*when you finish selecting objects*
The objects are removed from the drawing.
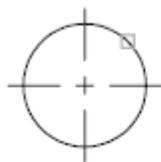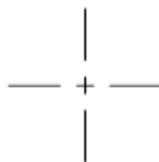
object selected   object erased

EXPLODE

Breaks a compound object into its component objects

⊗ **Modify toolbar:**

⊗ **Modify menu:** Explode

⌨ **Command line:** **explode**

   Select objects: *Use an object selection method and press* ENTER *when you finish*

The color, linetype, and lineweight of any exploded object might change. Other results differ depending on the type of compound object you're exploding. See the following list of objects that can be exploded and the results for each.

Note If you're using a script or an ObjectARX® function, you can explode only one object at a time.

2D and Lightweight Polyline

Discards any associated width or tangent information. For wide polylines, the resulting lines and arcs are placed along the center of the polyline.

3D Polyline

Explodes into line segments. Any linetype assigned to the 3D polyline is applied to each resulting line segment.

3D Solid

Explodes planar surfaces into regions. Nonplanar surfaces explode into bodies.

Arc

If within a nonuniformly scaled block, explodes into elliptical arcs.

Block

Removes one grouping level at a time. If a block contains a polyline or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

Blocks with equal X, Y, and Z scales explode into their component objects. Blocks with unequal X, Y, and Z scales (nonuniformly scaled blocks) might explode into unexpected objects.

When nonuniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with a "*E" prefix) and referenced with the nonuniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a nonuniformly scaled block cannot be exploded.

Exploding a block that contains attributes deletes the attribute values and redisplays the attribute definitions.

Blocks inserted with MINSERT and external references (xrefs) and their dependent blocks cannot be exploded.

Body

Explodes into a single-surface body (nonplanar surfaces), regions, or curves.

Circle

If within a nonuniformly scaled block, explodes into ellipses.

Leaders

Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), multiline text, or tolerance objects, depending on the leader.

Multiline Text

Explodes into text objects.

Multiline

Explodes into lines and arcs.

Polyface Mesh

Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.

Region

Explodes into lines, arcs, or splines.

EXPORT

Saves objects to other file formats

File menu: Export

Command line: export

The Export Data dialog box (a standard file selection dialog box) is displayed.
In Files of Type, select the format type to export objects to. In File Name, enter the name of the file to create. The objects are exported to the specified file format using the specified file name.
The following output types are available:

- *WMF:* Microsoft Windows® Metafile (see WMFOUT)
- *SAT:* ACIS solid object file (see ACISOUT)
- *STL:* Solid object stereolithography file (see STLOUT)
- *EPS:* Encapsulated PostScript file
- *DXX:* Attribute extract DXF™ file (see ATTEXT)
- *BMP:* Device-independent bitmap file (see BMPOUT)
- *3DS:* 3D Studio® file (see 3DSOUT)
- *DWG:* drawing file (see WBLOCK)

EXTEND

Extends an object to meet another object

Modify toolbar:

Modify menu: Extend

⌨ **Command line:** extend

Select boundary edges...

Select objects or <select all>: *Select one or more objects and press* ENTER*, or press* ENTER *to select all displayed objects*

Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]: *Select objects to extend, or hold down* SHIFT *and select an object to trim, or enter an option*

Boundary Object Selection

Boundary Object Selection

Uses the selected objects to define the boundary edges to which you want to extend an object.

Object to Extend

Object to Extend

Specifies the objects to extend. Press ENTER to end the command.

Shift-Select to Trim

Shift-Select to Trim

Trims the selected objects to the nearest boundary rather than extending them. This is an easy method to switch between trimming and extending.

Fence

Fence

Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

Specify first fence point: *Specify the starting point of the selection fence*

Specify next fence point or [Undo]: *Specify the next point of the selection fence or enter* **u**

Specify next fence point or [Undo]: *Specify the next point of the selection fence enter* **u**, *or press* ENTER

Crossing

Crossing

Selects objects within and crossing a rectangular area defined by two points.

Specify first corner: *Specify a point*

Specify opposite corner: *Specify a point at a diagonal from the first point*

Note Some crossing selections of objects to be extended are ambiguous. EXTEND resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

Project

Project

Specifies the projection method used when extending objects.

Enter a projection option [None/Ucs/View] *<current>*: *Enter an option or press* ENTER

selected boundary
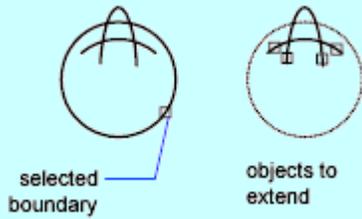
objects to extend

None

Specifies no projection. Only objects that intersect with the boundary edge in 3D space are extended.

arc not intersecting with boundary edge in 3D space

arc intersecting with boundary edge in 3D space

UCS
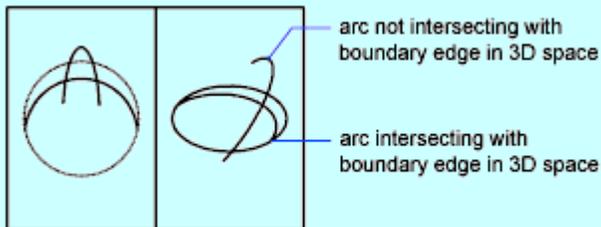
Specifies projection onto the *XY* plane of the current user coordinate system (UCS). Objects that do not intersect with the boundary objects in 3D space are extended.

projection of arcs that intersect with boundary edge onto current UCS

View

Specifies projection along the current view direction.

left viewport

right viewport
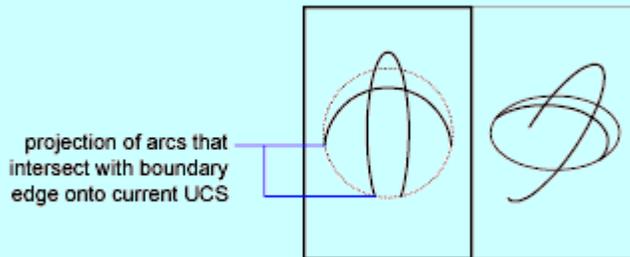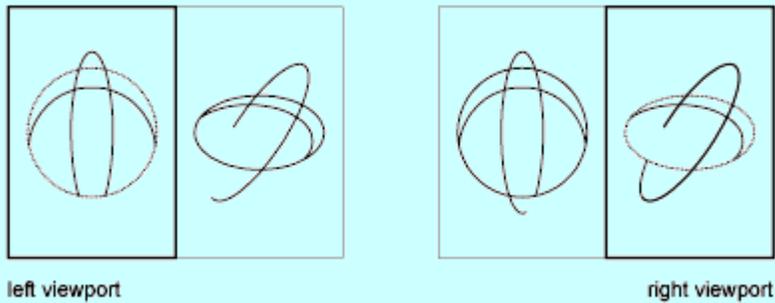
Edge

Edge

Extends the object to another object's implied edge, or only to an object that actually intersects it in 3D space.
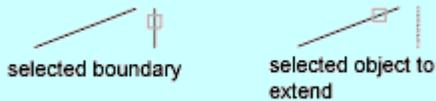
selected boundary   selected object to extend

Enter an implied edge extension mode [Extend/No extend] *<current>*: *Enter an option or press* ENTER

Extend

Extends the boundary object along its natural path to intersect another object or its implied edge in 3D space.

extended

No Extend

Specifies that the object is to extend only to a boundary object that actually intersects it in 3D space.

not extended
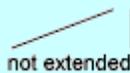
Undo

Undo

Reverses the most recent changes made by EXTEND.

FILL

Controls the filling of objects such as hatches, two-dimensional solids, and wide polylines

Command line: fill (or **'fill** for transparent use)

Enter mode [**ON**/**OFF**] *<current>*: *Enter* **on** *or* **off***, or press*ENTER

On

Turns on Fill mode. For the filling of a 3D object to be visible, its extrusion direction must be parallel to the current viewing direction, and hidden lines must not be suppressed.
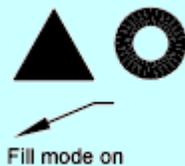
Fill mode on

Off

Turns off Fill mode. Only the outlines of objects are displayed and plotted. Changing Fill mode affects existing objects after the drawing is regenerated. The display of lineweights is not affected

by the Fill mode setting.

Fill mode off
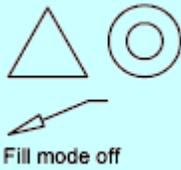
FILLET

Rounds and fillets the edges of objects

Modify toolbar:

Modify menu: Fillet

Command line: fillet

Current settings: Mode = *current*, Radius = *current*

Select **first object** or [**Undo**/**Polyline**/**Radius**/**Trim**/**Multiple**]: *Use an object selection method or enter an option*

First Object

First Object

Selects the first of two objects required to define a 2D fillet or selects the edge of a 3D solid to round or fillet the edge.

first selected object   second selected object   result
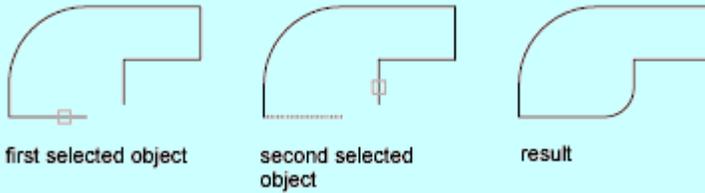
Select second object or shift-select to apply corner: Use an object selection method or hold down SHIFT and select an object to create a sharp corner

If you select lines, arcs, or polylines, their lengths adjust to accommodate the fillet arc. You can hold down SHIFT while selecting the objects to override the current fillet radius with a value of 0.
If the selected objects are straight line segments of a 2D polyline, they can be adjacent or separated by one other segment. If they are separated by another polyline segment, FILLET deletes the segment that separates them and replaces it with the fillet.
More than one fillet can exist between arcs and circles. Select the objects close to where you want the endpoints of the fillet.

selection points   result (with radius greater than 0)
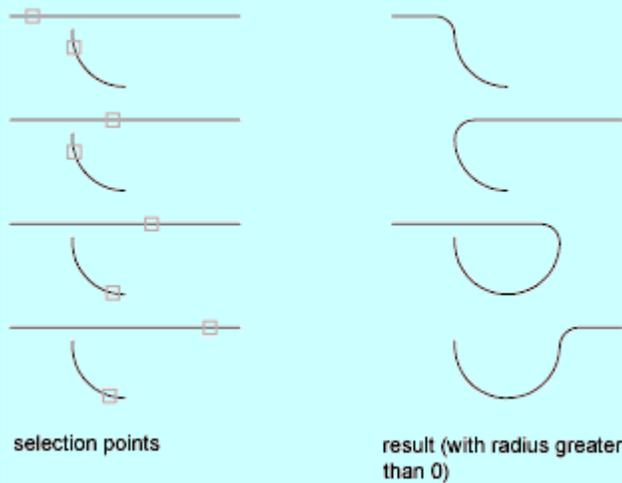
FILLET does not trim circles; the fillet arc meets the circle smoothly.

selection points   results (with radius greater than 0)
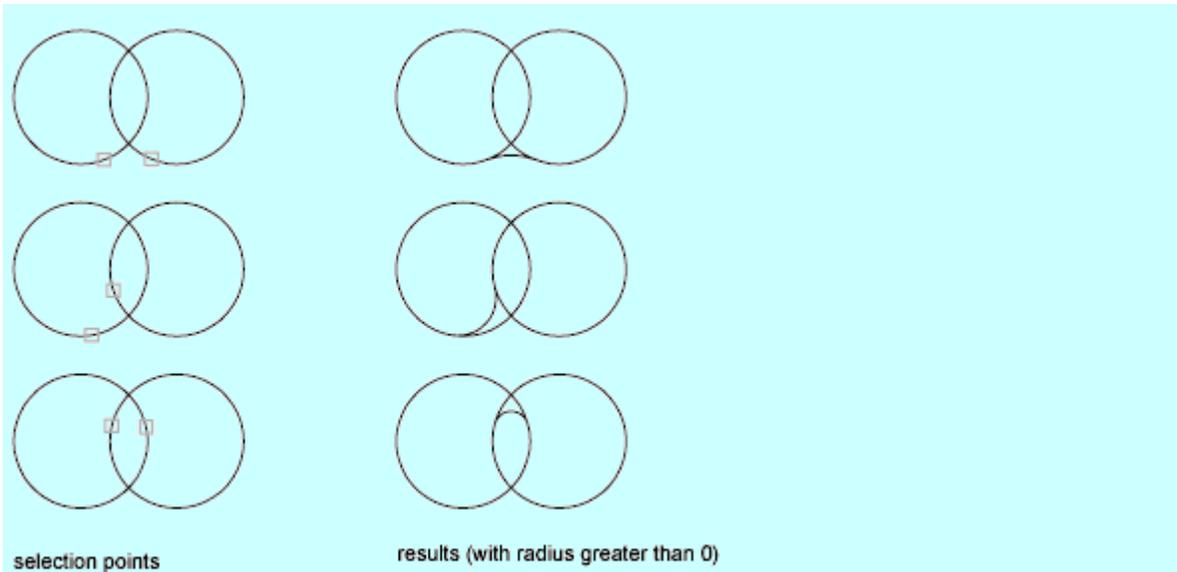
If you select a 3D solid, you can select multiple edges, but you must select the edges individually.

    Enter fillet radius *<current>*: *Specify a distance or press*ENTER

    Select an edge or [Chain/Radius]: *Select edge(s), enter* **c***, or enter* **r**

Edge

Selects a single edge. You can continue to select single edges until you press ENTER.

selecting edges   single edge fillets
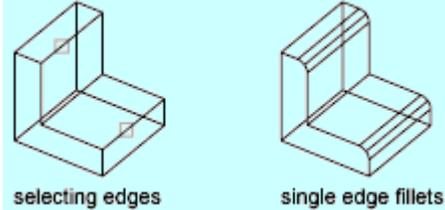
If you select three or more edges that converge at a vertex to form the corner of a box, FILLET computes a vertex blend that is part of a sphere if the three incident fillets have the same radii.

Chain

Changes from selection of single edges to selection of sequential tangent edges, called a *chain* selection.

    Select edge chain or <Edge/Radius>: *Select an edge chain, enter* **e***, or enter* **r**

Edge Chain

    Selects a tangential sequence of edges when you select a single edge. For example, if you select an edge on the top of a 3D solid box, FILLET also selects the other tangential edges on the top.

chain fillets

Edge

    Switches to a single-edge selection mode.

Radius

Defines the radius of the rounded edge.

Radius

Defines the radius of the rounded edge.

Enter fillet radius *<current>*: *Specify a distance or press*ENTER

The previous prompt is displayed:

Select an edge or [Chain/Radius]: *Select one or more edges, or enter* **c** *or* **r**

Undo

Undo

Reverses the previous action in the command.

Polyline

Polyline

Inserts fillet arcs at each vertex of a 2D polyline where two line segments meet.

Select 2D polyline:

If one arc segment separates two line segments that converge as they approach the arc segment, FILLET removes the arc segment and replaces it with a fillet arc.
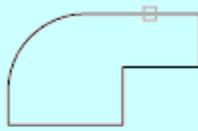
before   after

Radius

Radius

Defines the radius of the fillet arc.

Specify fillet radius *<current>*: *Specify a distance or press* ENTER

The value you enter becomes the current radius for subsequent FILLET commands. Changing this value does not affect existing fillet arcs.

Trim

Trim

Controls whether FILLET trims the selected edges to the fillet arc endpoints.

Enter Trim mode option [Trim/No trim] *<current>*: *Enter an option or press* ENTER

Trim

Trims the selected edges to the fillet arc endpoints.

No Trim

Does not trim the selected edges.

Multiple

Multiple

Rounds the edges of more than one set of objects. FILLET displays the main prompt and the Select Second Object prompt repeatedly until you press ENTER to end the command.

FILTER

Creates a list of requirements that an object must meet to be included in a selection set

⌨ **Command line: filter** (or **'filter** for transparent use)

The Object Selection Filters dialog box is displayed.

Selects, edits, and names filters for object selection.

Filter Property List

Displays a list of the filter properties that compose the current filter. The current filter is the filter that you select in Current in the Named Filters area.

Select Filter

Adds filter properties to the current filter.

Object Types and Logical Operators

Lists object types that you can filter and logical operators (AND, OR, XOR, and NOT) for grouping the filter expressions.

If you use logical operators, make sure that you pair and balance them correctly in the filter list. The number of operands you can enclose depends on the operation.

Logical operators		
Starting operator	Encloses	Ending operator
Begin AND	One or more operands	End AND
Begin OR	One or more operands	End OR
Begin XOR	Two operands	End XOR
Begin NOT	One operand	End NOT

For example, the following filter selects all circles except the ones with a radius greater than or equal to 1.0:

Object =Circle
**Begin NOT
Circle Radius >= 1.00
**End NOT

X, Y, Z Parameters

Define additional filter parameters depending on the object. For example, if you select Line Start, you can enter the *X*, *Y*, and *Z* coordinate values that you want to filter.

In the filter parameters, you can use relative operators such as < (less than) or > (greater than). For example, the following filter selects all circles with center points greater than or equal to 1,1,0 and radii greater than or equal to 1:

Object = Circle
Circle Center X >= 1.0000 Y >= 1.0000 Z >= 0.0000
Circle Radius >= 1.0000

Select

Displays a dialog box listing all items of the specified type in the drawing. Select the items to filter. For example, if you select the object type Color, Select displays a list of colors to choose for the filter.

Add to List

Adds the current Select Filter property to the filter list. Filter properties that you add to the unnamed filter remain available during the current work session unless you manually delete them.

Substitute

Replaces the filter property selected in the filter property list with the one displayed in Select Filter.

Add Selected Object

Adds one selected object in the drawing to the filter list.

Edit Item

Moves the selected filter property into the Select Filter area for editing. To edit a filter property, select it and choose Edit Item. Edit the filter property and choose Substitute. The edited filter replaces the selected filter property.

Delete

Deletes a selected filter property from the current filter.

Clear List

Deletes all the listed properties from the current filter.

Named Filters

Displays, saves, and deletes filters.

Current

Displays saved filters. Select a filter list to make it current. The named filter and its list of properties are loaded from the default file, *filter.nfl*.

Save As

Saves a filter and its list of properties. The filter is saved in the *filter.nfl* file. Names can contain up to 18 characters.

Delete Current Filter List

Deletes a filter and all its properties from the default filter file.

Apply

Exits the dialog box and displays the Select Objects prompt, where you create a selection set. The current filter is used on the objects you select.

FIND

The Find and Replace dialog box is displayed.

Specifies the text you want to find, replace, or select and controls the scope and results of the search.

Find Text String

Specifies the text string you want to find. Enter a text string including any wild-card characters, or choose one of the six most recently used strings from the list.

Replace With

Specifies the text string you want to use to replace the found text. Enter a string, or choose one of the most recently used strings from the list.

Search In

Specifies whether to search the entire drawing or only the current selection. If something is already selected, Current Selection is the default value. If nothing is selected, Entire Drawing is the default value. You can use the Select Objects button to temporarily close the dialog box and create or modify the selection set.

Select Objects Button

Closes the dialog box temporarily so that you can select objects in your drawing. Press ENTER to return to the dialog box. When you select objects, Search In displays Current Selection.

Options

Displays the Find and Replace Options dialog box, in which you can define the type of objects and words that you want to find.

Find/Find Next

Finds the text that you enter in Find Text String. If you have not entered text in Find Text String, this option is not available. Found text is displayed in the Context area. Once you find the first instance of the text, the Find option becomes Find Next, which you can use to find the next instance.

Replace

Replaces found text with the text that you enter in Replace With.

Replace All

Finds all instances of the text that you enter in Find Text String and replaces it with the text in Replace With. The Search In setting controls whether to find and replace text in the entire drawing or text in the current selection. The status area confirms the replacements and indicates the number of replacements that were made.

Select All

Finds and selects all loaded objects containing instances of the text that you enter in Find Text String. This option is available only when you set Search In to Current Selection. When you choose Select All, the dialog box closes and a message on the command line indicates

the number of objects that it found and selected. Note that Select All does not replace text; any text in Replace With is ignored.

Zoom To

Displays the area in the drawing that contains the results of a find or replace search. Although model space and all layouts defined for the drawing are searched, you can only zoom to text in the current Model or layout tab. When zooming to text found in a multiline text object, in some cases the found text string may not be displayed in a visible area of the drawing.

Object Type

Specifies the type of object the text was found in.

Context

Displays and highlights the currently found text string in its surrounding context. If you click Find Next, the Context area is refreshed and displays the next found text string in its surrounding context.

Status

Displays confirmation of find and replace searches.

GOTOURL

Opens the file or web page associated with the hyperlink attached to an object

Command line: gotourl

Select an object that has an attached hyperlink. The file or web page (URL) that is associated with the hyperlink opens.

GRAPHSCR

Switches from the text window to the drawing area

Command line: graphscr (or 'graphscr for transparent use)

GRAPHSCR closes the text window. You can also press F2 to open or close the text window. This command is ignored on dual-screen systems.

GRID

Displays a dot grid in the current viewport that is not plotted
Status bar: Grid

Command line: grid (or 'grid for transparent use)

Specify **grid spacing(X)** or [**ON**/**OFF**/**Snap**/**Aspect**] *<current>*: *Specify a value or enter an option*

Grid Spacing (X)

Sets the grid to the specified value. Entering **X** after the value sets the grid spacing to the specified value multiplied by the snap interval.

On

Turns on the grid using the current spacing.

GRID turned on

Off

Turns off the grid.

GRID turned off

Snap

Sets the grid spacing to the snap interval specified by the SNAP command.

Aspect

Changes the grid spacing in the *X* and *Y* directions.

Specify the horizontal spacing(X) *<current>*: *Enter a value or press* ENTER
Specify the vertical spacing(Y) *<current>*: *Enter a value or press* ENTER

Entering **X** following either value defines it as a multiple of the snap interval rather than the drawing units.

The Aspect option is not available when the current snap style is Isometric.
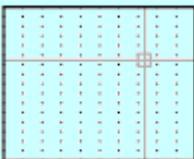
GRID set to Aspect

GROUP

Creates and manages saved sets of objects called groups

The Object Grouping dialog box is displayed.

If you enter **-group** at the Command prompt, GROUP displays prompts on the command line.

Displays, identifies, names, and changes object groups.

Group Name

Displays the names of existing groups.

Selectable

Specifies whether a group is selectable. When a group is selectable, selecting one object in the group selects the whole group. Objects on locked or frozen layers are not selected. When the PICKSTYLE system variable is set to 0, no groups are selectable.

Group Identification

Displays the name and description (if any) of the group selected in the Group Name list.

Group Name

Specifies the group name. Group names can be up to 31 characters long and can include letters, numbers, and the special characters dollar sign ($), hyphen (-), and underscore (_) but not spaces. The name is converted to uppercase characters.

Description

Displays the description of the selected group, if there is one.

Find Name

Lists the groups to which an object belongs.

Pick a member of a group: *Select one object*

The Group Member List dialog box is displayed, showing the groups to which the object belongs.

Highlight

Shows the members of the selected group in the drawing area.

Include Unnamed

Specifies whether unnamed groups are listed. When this option is cleared, only named groups are displayed.

Create Group

Specifies properties of new groups.

New

Creates a new group from the selected objects, using the name and description under Group Name and Description.

desk, computer, and
telephone selected

The group name is displayed in the Group Name list.

Selectable

Specifies that a new group is selectable.

Unnamed

Indicates that a new group is unnamed. A default name, *A*n*, is assigned to unnamed groups. The *n* represents a number that increases with each new group.

Change Group

Modifies existing groups.

Remove

Removes objects from the selected group. To use this option, clear the Selectable option.

Remove objects: *Use an object selection method*

The selected objects are removed from the group. The Object Grouping dialog box is displayed.

telephone selected

telephone removed from group

If you remove all the group's objects, the group remains defined. You can remove the group definition from the drawing by using the Explode option.

Note When you remove objects from a group and then later add them back during the same drawing session, they are returned to their previous position in the numerical order of the group.

Add

Adds objects to the selected group.

Select objects: *Use an object selection method*

The selected objects are added to the group. The Object Grouping dialog box is displayed.

chair selected

chair added to group

Note Group names are displayed in alphabetical order.

Rename

Renames the selected group to the name entered in Group Name under Group Identification.

Re-Order

Displays the Order Group dialog box, in which you can change the numerical order of objects within the selected group. Objects are numbered in the order in which you select them for inclusion in the group. Reordering is useful when creating tool paths. For example, you can change the cut order for the horizontal and vertical lines of a tool path pattern.

You can either change the numerical position of individual group members or ranges of group members or reverse the order of all members. The first object in a group is number 0, not number 1.

Description

Updates the selected group's description to the name that you enter in Description. You can use up to 64 characters for a description name.

Explode

Deletes the definition of the selected group. The group's objects remain in the drawing.

Selectable

Specifies whether the group is selectable.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill

Draw toolbar:

Draw menu: Hatch

Command line: hatch

The Hatch and Gradient dialog box is displayed.

If you enter **-hatch** at the Command prompt, HATCH displays prompts on the command line.

Note By default, the program will not create a hatch pattern that consists of over 10,000 hatch segments. The limit is set by the MaxHatch setting in the registry. To reset the limit to 50,000, for example, enter **(setenv "MaxHatch" "50000")** at the Command prompt. The limit can be reset to any value between 100 and 10,000,000.

Displays preview images for all predefined and custom patterns. The dialog box organizes patterns on four tabs with images arranged alphabetically on each tab. Click an image to select a pattern and click OK.

ANSI

Displays all ANSI patterns shipped with the product.

ISO

Displays all ISO patterns shipped with the product.

Other Predefined

Displays all patterns other than ANSI and ISO shipped with the product.

Custom

Displays a list of custom PAT files that you have added to the search path, which is set in the Options dialog box, Files tab.

Preview

Displays a preview image of the selected custom pattern.

Defines the appearance of the hatch pattern to be applied.

Type and Pattern

Specifies the type of hatch and the pattern.

Type

Sets the pattern type. User-defined patterns are based on the current linetype in your drawing. A custom pattern is a pattern that is defined in any custom PAT files that you have added to the search path. You can control the angle and scale of any pattern.

Predefined patterns are stored in the *acad.pat* or *acadiso.pat* files supplied with the product.

Pattern

Lists the available predefined patterns. The six most recently used predefined patterns appear at the top of the list. HATCH stores the selected pattern in the HPNAME system variable. The Pattern option is available only if you set Type to Predefined.

[...] Button

Displays the Hatch Pattern Palette dialog box, in which you can view preview images for all predefined patterns at once to help you make a selection.

Swatch

Displays a preview of the selected pattern. You can click the swatch to display the Hatch Pattern Palette dialog box. When the SOLID pattern is selected, you can click the right arrow to display a list of colors or the Select Color Dialog BoxSelect Color dialog box.

Custom Pattern

Lists the available custom patterns. The six most recently used custom patterns appear at the top of the list. The name of the selected pattern is stored in the HPNAME system variable. The Custom Pattern option is available only if you set Type to Custom.

[...] Button

Displays the Hatch Pattern Palette dialog box, in which you can view preview images for all custom patterns at once to help you make a selection.

Angle and Scale

Specifies an angle and a scale for the selected hatch pattern.

Angle

Specifies an angle for the hatch pattern relative to the *X* axis of the current UCS. HATCH stores the angle in the HPANG system variable.

Scale

Expands or contracts a predefined or custom pattern. HATCH stores the scale in the HPSCALE system variable. This option is available only if you set Type to Predefined or Custom.

Double

For user-defined patterns, draws a second set of lines positioned at 90 degrees to the original lines, creating a crosshatch. This option is available only if you set Type to User Defined on the Hatch tab. (HPDOUBLEsystem variable)

Relative to Paper Space

Scales the hatch pattern relative to paper space units. Using this option, you can easily display hatch patterns at a scale that is appropriate for your layout. This option is available only from a layout.

Spacing

Specifies the spacing of lines in a user-defined pattern. HATCH stores the spacing in the HPSPACE system variable. This option is available only if you set Type to User Defined.

ISO Pen Width

Scales an ISO predefined pattern based on the selected pen width. This option is available only if you set Type to Predefined and set Pattern to one of the available ISO patterns.

Hatch Origin

Controls the starting location of hatch pattern generation. Some hatches, such as brick patterns, need to be aligned with a point on the hatch boundary. By default, all hatch origins correspond to the current UCS origin.

Use Current Origin

Uses the setting stored in the HPORIGINMODE system variable. The origin is set to 0,0 by default.

Specified Origin

Specifies a new hatch origin. Click this option to make the following options available.

Click to Set New Origin

Specifies the new hatch origin point directly.

Default to Boundary Extents

Calculates a new origin based on the rectangular extents of the hatch. Choices include each of the four corners of the extents and its center. (HPORIGINMODE system variable)

Store as Default Origin

Stores the value of the new hatch origin in the HPORIGIN system variable.

Origin Preview

Shows the current location of the origin.

Controls the operation of islands and boundaries.

Islands

Specifies the method used to hatch or fill objects within the outermost boundary. If no internal boundaries exist, specifying an island detection style has no effect. Because you can define a precise set of boundaries, it is often best to use the Normal style.

Island Detection

Controls whether internal closed boundaries, called islands, are detected.

Normal

Hatches or fills inward from the outer boundary. If HATCH encounters an internal island, it turns off hatching or filling until it encounters another island within the island. You can also set the Normal style by adding **,N** to the pattern name in the HPNAME system variable.

Outer

Hatches or fills inward from the outer boundary. HATCH turns hatching or filling off if it encounters an internal island. This option hatches or fills only the outermost level of the structure and leaves the internal structure blank. You can also set the Outer style by adding **,O** to the pattern name in the HPNAME system variable.

Ignore

Ignores all internal objects and hatches or fills through them. You can also set the Ignore style by adding **,I** to the pattern name in the HPNAME system variable.

The Normal, Outer, and Ignore options are also available from a shortcut menu by right-clicking in the drawing area while you specify points or select objects to define your boundaries.

Boundary Retention

Specifies whether to retain boundaries as objects, and determines the object type applied to those objects.

Retain Boundaries

Creates boundary objects from the temporary hatch boundaries and adds them to the drawing.

Object Type

Controls the type of the new boundary object. The resulting boundary object can be a region or a polyline object. This option is available only if Retain Boundaries is checked.

Boundary Set

Defines the set of objects analyzed when defining a boundary from a specified point. The selected boundary set has no effect when you use Select Objects to define a boundary. By default, when you use the Add: Pick Point option to define a boundary, HATCH analyzes all objects in the current viewport extents. By redefining the boundary set, you can disregard certain objects when defining boundaries without having to hide or remove those objects. For large drawings, redefining the boundary set can also produce the boundary faster because HATCH examines fewer objects.

Current Viewport

Defines the boundary set from everything in the current viewport extents. Selecting this option discards any current boundary set.

Existing Set

Defines the boundary set from the objects that you selected with New. If you have not created a boundary set with New, the Existing Set option is not available.

New

Prompts you to select the objects that define the boundary set.

Gap Tolerance

Sets the maximum size of gaps that can be ignored when objects are used as a hatch boundary. The default value, 0, specifies that the objects must enclose the area with no gaps. Enter a value, in drawing units, from 0 to 5000 to set the maximum size of gaps that can be ignored when the objects serve as a hatch boundary. Any gaps equal to or smaller than the value you specify are ignored, and the boundary is treated as closed. (HPGAPTOL system variable)

Inherit Options

When you create a hatch with Inherit Properties, these settings control the location of the hatch origin.

Use Current Origin

Uses the current hatch origin setting

Use Source Hatch Origin

Uses the hatch origin of the source hatch

HATCHEDIT

Modifies an existing hatch or fill

 Modify II toolbar:

 Modify menu: Object ➤ Hatch

Shortcut menu: Select a hatch object to edit, and right-click in the drawing area. Click Hatch Edit.

Command line: hatchedit

Select hatch object: *Use an object selection method*

The Hatch Edit dialog box is displayed.
If you enter **-hatchedit** at the Command prompt, HATCHEDIT displays prompts on the command line.

Modifies the characteristics of an existing hatch or fill. The Hatch Edit dialog box displays the current properties of the selected hatch or fill object. You can modify only the properties in the Hatch Edit dialog box that are available.

Hatch Tab

Modifies the hatch pattern properties.

For information about these options, see the Hatch tab in the Hatch and Gradient dialog box.

Gradient Tab

Modifies the gradient fill properties. For information about these options, see the Gradient tab in the Hatch and Gradient dialog box.

More Options

Modifies the Inherit Options setting that is associated with Inherit Properties. All other options are not available.

For more information, see Inherit Properties in the Hatch and Gradient dialog box.

Boundaries

Modifies the boundaries of a hatch or fill.

For more information, see the Hatch and Gradient dialog box.

Options

Removes hatch associativity, separates a single hatch object that has several separate boundaries into individual hatch objects, and changes hatch draw order.

For more information, see the Hatch and Gradient dialog box.

Inherit Properties

Assigns the hatch or fill properties from another hatch object.

For more information, see the Hatch and Gradient dialog box.

Preview

Temporarily dismisses the dialog box and displays the selected object with the specified properties.

HELP

Displays Help

Standard toolbar:

Help menu: Help

⌨ **Command line:** **help** or **?** or press F1 (or **'help** for transparent use)

To display Help for a menu, open the menu, and then press F1.

Pressing F1 or entering **'help** while a command is active displays Help for that command. Choosing the Help button in a dialog box displays Help for that dialog box.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink
🔗 **Insert menu:** Hyperlink

Shortcut menu: To edit a hyperlink, select an object that contains a hyperlink, right-click in the drawing area, and choose Hyperlink➤ Edit Hyperlink.

⌨ **Command line:** **hyperlink**

Select objects: *Use an object selection method*

One of the following dialog boxes is displayed:

- Insert Hyperlink dialog box (for graphical objects that do not already contain hyperlinks)
- Edit Hyperlink dialog box (for graphical objects that already contain hyperlinks)

After you insert a hyperlink, the hyperlink icon is displayed when you move the cursor over the attached object. To open the hyperlink, right-click the selected object and choose Hyperlink. The PICKFIRST system variable must be set to 1 to open files associated with hyperlinks. If you enter **-hyperlink** at the Command prompt, HYPERLINK displays options on the command line, including an option to define an area to associate with a hyperlink.

Attaches hyperlinks to graphical objects.

Text to Display

Specifies a description for the hyperlink. This is useful when the file name or URL is not helpful in identifying the contents of the linked file.

The Insert Hyperlink dialog box includes the following tabs under Link To: Existing File or Web Page, View of This Drawing, E-mail Address

View of This Drawing Tab (Insert Hyperlink Dialog Box)

View of This Drawing Tab (Insert Hyperlink Dialog Box)

Specifies a named view in the current drawing to link to.

Select a View of This

Displays an expandable tree view of the named views within the current drawing, from which you can select one to link to.

E-mail Address Tab (Insert Hyperlink Dialog Box)

E-mail Address Tab (Insert Hyperlink Dialog Box)

Specifies an email address to link to. When the hyperlink is executed, a new email is created using the default system email program.

E-mail Address

Specifies an email address.

Subject

Specifies a subject for the email.

Recently Used E-mail Addresses

Lists recently used email addresses, from which you can select one to use for the hyperlink.

HYPERLINKOPTIONS

Controls the display of the hyperlink cursor, tooltips, and shortcut menu

⌨ **Command line:** hyperlinkoptions

Display hyperlink cursor, tooltip, and shortcut menu? [Yes/No] <Yes>:

Note Access to hyperlinks is not available if you choose No.

ID

Displays the coordinate of a location

✦ **Tools menu:** Inquiry ➤ ID Point

⌨ **Command line:** **id** (or **'id** for transparent use)

Point: *Use the pointing device to specify a point*

The UCS coordinate of the location is displayed on the command line.
ID lists the *X*, *Y*, and *Z* values of the specified point and stores the coordinate of the specified point as the last point. You can reference the last point by entering **@** at the next prompt that requests a point.
If you snap to an object in 3D space, the *Z* coordinate value is the same as that of the selected feature of the object.

IMAGE

Manages images

✦ **Reference toolbar:**

✦ **Insert menu:** Image Manager

Shortcut menu: Select an image, right-click in the drawing area, and choose Image ➤ Image Manager.

⌨ **Command line:** **image**

The Image Manager is displayed.
If you enter **-image** at the Command prompt, IMAGE displays prompts on the command line.

- Image Manager

Lists all the image files attached to the current drawing. You can view image details such as pixel width, color depth, and resolution of the selected image. You can attach new image files and detach, locate, reload, and unload attached images.

List of Images

Displays the names of all image files attached to the drawing. You can use F3 and F4 to toggle between list view and tree view.

List View

Lists the image definitions attached to the drawing. Each image name appears only once regardless of how many times the image is attached (inserted). You can sort the list of images by name, status (loaded, unloaded, or not found), type (TIFF, for example), date, size, or the saved path and file name. By default, the list is alphabetical by image name.

- To select multiple images, hold down SHIFT or CTRL while selecting items.
- To sort the list alphabetically or numerically by a specific column, click that column's heading.
- To change the width of the column, drag the line between the column headings to the right or left. The settings are saved and restored when you reopen the dialog box.
- To change an image name, select it and then click it again, or select it and then press F2. You cannot edit names of images that reside in external references (xrefs). Image names can include up to 255 characters and can contain letters, digits, spaces, and any special characters not used by Microsoft$^®$ Windows$^®$ or this program. The image name can be identical to the file name, but changing the image name does not change the file name.

Tree View

Displays all the image definitions and the levels of nesting of images within xrefs. The top level of the tree view shows images that you attached directly to the drawing, images nested in block references, and the names of externally referenced drawings containing images. The names of the images attached to the externally referenced drawings appear nested within the drawing at the next tree level.

To insert a copy of an already attached image, select it, and then choose Attach.

Tree view lists the image names only (not file names) and lists the image name just once, regardless of how many times the image is attached (insert).

You can edit an image name by selecting it and then clicking it again, or by selecting it and then pressing F2. However, you cannot select more than one image at a time.

Attach

Displays the Select Image File dialog box. See IMAGEATTACH. When you unload and then reload an image, that image is drawn on top. For more information, see DRAWORDER. Images remain loaded or unloaded from one drawing session to the next.

Detach

Removes the selected image definitions from the drawing database and erases all the associated image objects from the drawing and from the display.

Reload

Loads the most recent version of an image or reloads an image that was previously unloaded. Reloading does not control whether the image is displayed, but it ensures display of the most current image.

Unload

Unloads image data from working memory without erasing the image objects from the drawing. It is recommended that you unload images no longer needed for editing to improve performance. An unloaded image cannot be displayed or plotted.
You can selectively load and unload individual images from a working list of images associated with the drawing file.

Details

Opens the Image File Details dialog box, which displays the image name, saved path, active path, file creation date and time, file size and type, color system, color depth, width and height in pixels, resolution, default size in units, and a preview image.

Image Found At

Shows the path where the selected image was found. If you select multiple images, this field remains blank.
If the image file is not found in the path in the Saved Path column, several other paths are automatically searched for the image file. For the list of these paths and the order in which they are searched, see "Set Paths to Externally Referenced Drawings" in the *User's Guide*.

Browse

Opens the Select Image File dialog box (a standard file selection dialog box). The path you select appears under Image Found At.

Save Path

Stores the new path information. Press ESC while editing the path to restore the old path. If the referenced image cannot be found in the new path, the image's status changes to Not Found. If you do *not* click Save Path after editing the path, the original image path is used the next time you load the drawing.

- IMAGE Command Line

If you enter **-image** at the Command prompt, IMAGE displays prompts on the command line.
  Enter image option [?/Detach/Path/Reload/Unload/Attach] <Attach>: *Enter an option or press*ENTER

?—List Images

?—List Images

Lists the images by name in alphabetical order, the number of times each is attached to the drawing, and the path where the image is stored. Images are listed in alphabetical order, regardless of the setting of the MAXSORT system variable.

Images to list <*>: *Enter * to list all images, or enter an image name*

Detach

Detach

Detaches the named image from the drawing, marks it for deletion, and erases all occurrences of the image.

Enter list of images to detach: *Enter an image name, or enter * to detach all images*

Path

Path

Updates the path name (including file name) associated with a particular image. This option is useful if you change the location of an image file, rename the file, or replace an old image file with a new file; for instance, you can update *image01.pcx* and save it as *image02.pcx*.

Enter list of images for path modification: *Enter an image name, or enter * to list all images*

If you enter an asterisk (*), the following prompt is displayed:

Old path: *Lists the current path name for each image*

Enter New path: *Enter the new path name for the specified image*

Reload

Reload

Reloads the selected images, making that information available for display and plotting.

Enter list of images to reload: *Enter an image name, or enter * to reload all images*

Reloading...

Reload image *<image name>*: *<hard-coded path name>*

<image name> loaded and relinked.

Unload

Unload

Removes image data from working memory so that the images are not displayed, thus improving performance. All information associated with the image remains stored with the drawing. The image frame of each attached image remains visible.

Enter list of images to unload: *Enter a loaded image name, or enter * to unload all images*

Attach

Attach

Attaches a new image or a copy of an attached image to the current drawing. The Select Image File dialog box (a standard file selection dialog box) is displayed.

The file name without the extension is assigned as the image name. Image names can include up to 255 characters and contain letters, digits, spaces, and any special characters not used by Microsoft Windows or this program. If the file name is not a valid name for a nongraphical object, the Substitute Image Name dialog box is displayed. A valid image name is generated from the file name and an underscore and number are appended to the name. The number changes as necessary to avoid duplicate image names.

   Image *file name_1* created from file *file name.bmp*

If a definition with the same name and path exists in the drawing, the following prompts are displayed and the image is inserted as a copy:

   Image file name has already been loaded.

   Use IMAGE Reload to update its definition.

   Specify insertion point <0,0>: *Specify an insertion point*

   Base image size: Width: *current width*, Height: *current height*, *current unit*

   Specify scale factor: *Enter a value or press*ENTER

   Specify rotation angle <0>: *Enter a value or press*ENTER

If the FILEDIA system variable is set to 0, the following prompt is displayed instead of the dialog box:

   Enter image file name to attach <*last*>: *Enter an image name*

The last image name attached to the drawing during the current session is the default. To avoid errors when entering an image name, it is recommended that you specify both the image name and the file name as follows:

   imagename=path name\long file name.bmp

or

   imagename="path name\long file name.bmp"

If you enter a valid image name without a file extension, the program searches for the file in this order: first, an existing image definition in the drawing, and second, an image file in the folders in order of the search path. The program searches for all the image files with the specified name, regardless of extension, and uses the first name found. If no image name or image file is found, the message "Image Not Found" is displayed and the prompt is repeated.

To specify a long file name that does not conform to this program's naming rules, enter the name as follows:

   "*imagename=filename*"

You can use a dialog box to search for image files but still enter the *imagename=filename* convention on the command line. Enter a tilde (~) at the Enter Image File Name to Attach prompt. If you press ESC after the dialog box opens, the Enter Image Name prompt is displayed.

IMAGEATTACH

Attaches a new image to the current drawing

 Reference toolbar:

Insert menu: Raster Image

Command line: imageattach

The Select Image File dialog box (a standard file selection dialog box) is displayed. Once you select an image file, the Image dialog box is displayed.

Locates, inserts, names, and defines the parameters and details of attached images.

Name

Identifies the image you have selected to attach, either from the Select Image File dialog box (an unattached image) or from the list of previously attached images. To add another instance of an image file that is already attached, select the image name from the list and click OK.

Browse

Opens the Select Image File dialog box (a standard file selection dialog box). If Show Preview is selected, a preview of the selected file is displayed.

Found In

Displays the path where the image file was located.

Saved Path

Displays the path specified when the image file was attached to the current drawing.

Path Type

Specifies one of three types of folder path information to save with an attached image: an absolute path, a relative path, and no path.

Full Path

Specifies the absolute path to the image file.

Relative Path

Specifies a relative path to the image file.

No Path

Specifies only the image file name. The image file should be located in the folder with the current drawing file.

Insertion Point

Specifies the insertion point for the selected image. Specify On-Screen is the default. The default insertion point is 0,0.

Specify On-Screen

Directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter the insertion point in X, Y, and Z.

X

Sets the X coordinate value.

Y

Sets the Y coordinate value.

Z

Sets the Z coordinate value.

Scale

Specifies the scale factor of the selected image. Specify On-Screen directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter a value for the scale factor. The default scale factor is 1.
If INSUNITS is set to "unitless" or if the image does not contain resolution information, the scale factor becomes the image width in AutoCAD units. If INSUNITS has a value such as millimeters, centimeters, inches, or feet, and the image has resolution information, the scale factor is applied after the true width of the image in AutoCAD units is determined.

Rotation

Specifies the rotation angle of the selected image. If Specify On-Screen is selected, you may wait until you exit the dialog box to rotate the object with your pointing device or enter a rotation angle value on the command line. If Specify On-Screen is cleared, enter the rotation angle value in the dialog box. The default rotation angle is 0.

Details

Displays the Image Information section. You can view width and height in pixels, the resolution, and the size in units (such as millimeters, centimeters, meters, kilometers, inches, feet, yards, miles, unitless, and many others). The default value for unitless images is unitless. The image size is automatically converted to AutoCAD units and is displayed at the default width and height.

Resolution

Displays information about the vertical and horizontal resolution of the selected image.

Current AutoCAD Unit

Displays information about the default units of the selected image. The image is displayed at the default width and height in pixels.

Image Size in Pixels

Displays information about the width and height in pixels of the selected image.

Image Size in Units

Displays information about the default size in units of the selected image. The image size is automatically converted to AutoCAD units and is displayed at the default width and height.

IMAGEADJUST

Controls the image display of the brightness, contrast, and fade values of images

 Reference toolbar:

 Modify menu: Object ➤ Image ➤ Adjust

Shortcut menu: Select an image to adjust, right-click in the drawing area, and choose Image ➤ Adjust.

 Command line: imageadjust

Select image(s): *Select one or more images*

The Image Adjust dialog box is displayed.
If you enter **-imageadjust** at the Command prompt, IMAGEADJUST displays prompts on the command line.

IMAGECLIP

Uses clipping boundaries to define a subregion of an image object

⊗ **Reference toolbar:**

⊗ **Modify menu:** Clip ➤ Image

   Shortcut menu: Select an image to clip, right-click in the drawing area, and choose Image ➤ Clip.

⌨ **Command line: imageclip**

   Select image to clip: *Select an edge of an image*
   Enter image clipping option [**ON**/**OFF**/**Delete**/**New boundary**] <New>: *Enter an option or press*
      ENTER
The boundary you specify must be in a plane parallel to the image object.

On

On

Turns on clipping and displays the image clipped to the previously defined boundary.

Off

Off

Turns off clipping and displays the entire image and frame.
If you reclip the image while clipping is turned off, clipping is automatically turned back on. You are prompted to delete the old boundary even when clipping is turned off and the clipping boundary is not visible.

Delete

Delete

Removes a predefined clipping boundary and redisplays the full original image.

New Boundary

New Boundary

Specifies a new clipping boundary. The boundary can be rectangular or polygonal, and consists only of straight line segments. When defining a clipping boundary, specify vertices within the image boundary. Self-intersecting vertices are valid. Rectangular is the default option. If you use the pointing device to specify a point at the Enter Clipping Type prompt, the point is interpreted as the first corner of a rectangle.

 Enter clipping type [Polygonal/Rectangular] <Rectangular>: *Enter* **p** *or press*ENTER

Polygonal

Uses specified points to define a polygonal boundary.

 Specify first point: *Specify a point*

 Specify next point or [Undo]: *Specify a point or enter* **u**

 Specify next point or [Undo]: *Specify a point or enter* **u**

 Specify next point or [Close/Undo]: *Specify a point, or enter* **c** *or* **u**

You must specify at least three points to define a polygon.

If the image already has a clipping boundary defined, the following prompt is displayed:

 Delete old boundary? [No/Yes] <Yes>: *Enter* **n** *or press* ENTER

If you choose Yes, the entire image is redrawn and the command continues; if you choose No, the command ends.

Rectangular

Specifies a rectangular boundary by its opposite corners. The rectangle is always drawn parallel to the edges of the image.

 Specify first corner point: *Specify a point*

 Specify opposite corner point: *Specify a point*

IMAGEFRAME

Controls whether image frames are displayed and plotted

 Reference toolbar:

 Modify menu: Object ➤ Image ➤ Frame

 Command line: imageframe

 Enter image frame setting [0, 1, 2] <*current*>: *Enter an option or press* ENTER

0

Image frames are not displayed and not plotted.

1

Image frames are both displayed and plotted.

2

Image frames are displayed but not plotted.

IMAGEQUALITY

Controls the display quality of images

 Reference toolbar:

 Modify menu: Object ➤ Image ➤ Quality

 Command line: imagequality

> Enter image quality setting [High/Draft] *<current>*: *Enter an option or press* ENTER

The quality setting affects display performance; high-quality images take longer to display. Changing the setting updates the display immediately without causing a regeneration of the drawing.

Note Images are always plotted using a high-quality setting.

High

Produces a high-quality display of the image.

Draft

Produces a lower-quality display of the image.

IMPORT

Imports files in various formats

 Insert toolbar:

 Command line: import

The Import File dialog box (a standard file selection dialog box) is displayed.
In Files of Type, select the format type to import. In File Name, select the file to import. The file is imported into the drawing. The following input types are available:

- WMF—Windows metafile (see WMFIN)
- SAT—ACIS solid object file (see ACISIN)
- 3DS—3D Studio file (see 3DSIN)

For the WMF file type, choosing Options from the Tools menu in this dialog box displays the WMF In Options dialog box. You can display this dialog box directly by using WMFOPTS.
If FILEDIA = 0, the following prompt is displayed:

> Enter import file name: *Enter path and file name*

INSERT

Insert toolbar:

Insert menu: Block

Command line: insert

The Insert dialog box is displayed.

If you enter **-insert** at the Command prompt, INSERT displays prompts on the command line.

Specifies the name and position of the block or drawing to insert.

The last block you insert during the current editing session becomes the default block for subsequent uses of INSERT. The position of the inserted block depends on the orientation of the UCS.

Name

Specifies the name of a block to insert, or the name of a file to insert as a block.

Browse

Opens the Select Drawing File dialog box (a standard file selection dialog box) where you can select a block or drawing file to insert.

Path

Specifies the path to the block.

Preview

Displays a preview of the specified block to insert. A lightning bolt icon in the lower-right corner of the preview indicates that the block is dynamic.

Insertion Point

Specifies the insertion point for the block.

Specify On-Screen

Specifies the insertion point of the block using the pointing device.

X

Sets the X coordinate value.

Y

Sets the Y coordinate value.

Z

Sets the Z coordinate value.

Scale

Specifies the scale for the inserted block. Specifying negative values for the X, Y, and Z scale factors inserts a mirror image of a block.

Specify On-Screen

Specifies the scale of the block using the pointing device.

X

Sets the X scale factor.

Y

Sets the Y scale factor.

Z

Sets the Z scale factor.

Uniform Scale

Specifies a single scale value for X, Y, and Z coordinates. A value specified for X is also

reflected in the Y and Z values.

Rotation

Specifies the rotation angle for the inserted block in the current UCS.

Specify On-Screen

Specifies the rotation of the block using the pointing device.

Angle

Sets a rotation angle for the inserted block.

Block Unit

Displays information about the block units.

Unit

Specifies the INSUNITS value for the inserted block.

Factor

Displays the unit scale factor, which is calculated based on the INSUNITS value of the block
and the drawing units.

Explode

Explodes the block and inserts the individual parts of the block. When Explode is selected, you
can specify a uniform scale factor only.
Component objects of a block drawn on layer 0 remain on that layer. Objects having color
BYBLOCK are white. Objects with linetype BYBLOCK have the CONTINUOUS linetype.

INSERTOBJ

Inserts a linked or embedded object

 Insert toolbar:

⬙ **Insert menu:** OLE Object

⌨ **Command line: insertobj**

The Insert Object dialog box is displayed.

Inserts a linked or embedded object. Options vary, depending on whether you select Create New or Create from File.

Create New

Opens the application that's highlighted in the Object Type list so that you can create a new object to insert.

Object Type

Lists available applications that support linking and embedding. To create an object to embed, double-click an application to open it.

On the application's File menu, the Save option is replaced with a new Update option. Choosing Update inserts the object into the drawing or updates it.

Display as Icon

Displays the source application's icon in the drawing. Double-clicking the icon displays the embedded information.

Create from File

Specifies a file to link or embed.

File

Specifies the path and name of the file to be embedded or linked.

Browse

Displays the Browse dialog box (a standard file selection dialog box), in which you select a file to link or embed.

Link

Creates a link to the selected file rather than embedding it.

Display as Icon

Displays the source application's icon in the drawing. Double-clicking the icon displays the linked or embedded information.

ISOPLANE

Specifies the current isometric plane

⌨ **Command line: isoplane** (or **'isoplane** for transparent use)

Enter isometric plane setting [**Left**/**Top**/**Right**] <Top>: *Enter an option or press*ENTER

The isometric plane affects the cursor movement keys only when Snap mode is on and the snap style is Isometric. If the snap style is Isometric, Ortho mode uses the appropriate axis pair even if Snap mode is off. The current isometric plane also determines the orientation of isometric circles drawn by ELLIPSE. You can cycle through the isometric planes by pressing CTRL+E or F5.

Left
Selects the left-hand plane, defined by the 90-degree and 150-degree axis pair.

Top
Selects the top face of the cube, called the top plane, defined by the 30-degree and 150-degree axis pair.

Right
Selects the right-hand plane, defined by the 90-degree and 30-degree axis pair.

JPGOUT

Saves selected objects to a file in JPEG file format

⌨ **Command line:** jpgout

The Create Raster File dialog box (a standard file selection dialog box) is displayed. Enter the file name in the dialog box.

> Select objects or <all objects and viewports>: *Press* ENTER*to select all objects and viewports or use an object selection method and press* ENTER

A JPEG file is created that contains the objects you select. Shade Plot options are preserved in the file when you use this command.

Note When the FILEDIA system variable is set to 0 (Off), prompts are displayed on the command line.

JUSTIFYTEXT

Changes the justification point of selected text objects without changing their locations

✎ **Text toolbar:**

 Modify menu: Justify Text

 Modify menu: Object ➤ Text ➤ Justify

 Command line: justifytext

    Select objects: *Use an object selection method and press* ENTER *when you finish*

You can choose single line text objects, multiline text objects, leader text objects, and attribute objects.

    Enter a justification option
        [Left/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]<Existing>: *Specify a location to serve as the new justification point*

The justification point options shown above are described in the TEXT command. The justification point options for single line text are similar to those for multiline text except that the Align, Fit, and Left text options are equivalent to the bottom left (BL) multiline text attachment point.

LAYER

 Layers toolbar:

 Format menu: Layer

 Command line: layer (or **'layer** for transparent use)

Displays a list of the layers in the drawing and their properties. You can add, delete, and rename layers, change their properties, or add descriptions. Layer filters control which layers are displayed in the list and can also be used to make changes to more than one layer at a time.

New Property Filter

Displays the Layer Filter Properties dialog box, where you can create a layer filter based on one or more properties of the layers.

New Group Filter

Creates a layer filter that contains layers that you select and add to the filter.

Layer States Manager

Displays the Layer States Manager, in which you can save the current property settings for layers in a named layer state and then restore those settings later.

New Layer

Creates a new layer. The list displays a layer named LAYER1. The name is selected so that you can enter a new layer name immediately. The new layer inherits the properties of the currently selected layer in the layer list (color, on or off state, and so on).

Delete Layer

Marks selected layers for deletion. Layers are deleted when you click Apply or OK. 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.

Layers in a partially opened drawing are also considered referenced and cannot be deleted.

Note 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.

Set Current

Sets the selected layer as the current layer. Objects that you create are drawn on the current layer. (CLAYER system variable)

Current Layer

Displays the name of the current layer.

Search for Layer

Filters the layer list by name quickly as you enter characters. This filter is not saved when you close the Layer Properties Manager.

Status Line

Shows the name of the current filter, the number of layers displayed in the list view, and the number of layers in the drawing.

Invert Filter

Displays all layers that do not meet the criteria in the selected layer property filter.

Indicate Layers in Use

Displays icons in the list view to indicate whether layers are in use. In a drawing with many layers, clear this option to improve performance. (SHOWLAYERUSAGE system variable)

Apply to Layers Toolbar

Controls the display of layers in the list of layers on the Layers toolbar by applying the current layer filter.

Apply

Applies changes that have been made to layers and filters but does not close the dialog box.

The Layer Properties Manager has two panes:

- **Tree view**
- **List view**

Tree View

Displays a hierarchical list of layers and filters in the drawing. The top node, All, displays all layers in the drawing. Filters are displayed in alphabetical order. The All Used Layers filter is read-only. Expand a node to see nested filters. Double-click a property filter to open the Layer Filter Properties dialog box and view the definition of the filter.
If there are xrefs attached to the drawing, an XREF node displays the names of all the xrefs in the drawing and the layers in each xref. Layer filters defined in xref files are not displayed.

Tree View Shortcut Menu

Provides commands for items selected in the tree view.

Visibility

Changes the visibility state of all the layers in the selected filter (or All or All Used Layers, if selected).

- *On:* Objects on the layer are displayed, plotted, and regenerated and hide other objects when you use HIDE.
- *Off:* Objects on the layer are not displayed and not plotted but do hide other objects when you use HIDE. The drawing is not regenerated when you turn the layer on.
- *Thawed*: Objects on the layer are displayed and plotted and hide other objects when you use HIDE.
- *Frozen:* Objects on the layer are not displayed and not plotted but do hide other objects when you use HIDE. The drawing is regenerated when you thaw the layer.

Lock

Controls whether the objects on the layers in the selected filter can be modified.

- *Lock:* None of the objects on the layer can be modified. You can still apply object snaps to objects on a locked layer and perform other operations that do not modify those objects.
- *Unlock:* Objects on the layer can be modified.

Viewport

In the current layout viewport, controls the VP Freeze setting of the layers in the selected layer filter. This option is not available for model space viewports.

- *Freeze:* Sets Current VP Freeze for layers in the filter. In the current viewport, objects on the layer are not displayed and not plotted but do hide other objects when you use HIDE. The drawing is regenerated when you thaw the layer.
- *Thaw:* Clears Current VP Freeze for layers in the filter. In the current viewport, objects on the layer are displayed and plotted and hide other objects when you use HIDE. This option does not thaw layers that are set to Off or Frozen in the drawing.

Isolate Group

Turns off all layers not in the selected filter. The layers that are visible are the layers in the filter.

- *All Viewports:* In all viewports in a layout, sets Current VP Freeze for all layers that are not in the selected filter. In model space, turns off all layers that are not in the selected filter.
- *Active Viewport Only:* In the current layout viewport, sets Current VP Freeze for all layers that are not in the selected filter. In model space, turns off all layers that are not in the selected filter.

New Properties Filter

Displays the Layer Filter Properties dialog box, where you can create a new layer filter based on layer names and settings; for example, on or off, color, or linetype.

New Group Filter

Creates a new layer group filter named GROUP FILTER1 and adds it to the tree view. Enter a new name. Select the All filter or any other layer filter in the tree view to display layers in the list view, and then drag layers from the list view into the new layer group filter in the tree view.

You can use standard selection methods. Hold down CTRL to select more than one layer name. Hold down SHIFT and select the first and last layer names in a sequence to select all layers in the sequence.

Convert to Group Filter

Converts the selected layer property filter to a layer group filter. Changing the properties of the layers in a layer group filter has no effect on the filter.

Rename

Renames the selected filter. Enter a new name.

Delete

Deletes the selected layer filter. You cannot delete the All, All Used Layers, or Xref filters. This option deletes the layer filter but not the layers in the filter.

Properties

Displays the Layer Filter Properties dialog box, where you can modify the definition of the selected layer property filter. This option is available only when a layer property filter is selected.

Select Layers

Temporarily closes the Layer Filter Properties dialog box so that you can select objects in the drawing. This option is available only when a layer group filter is selected.

- *Add:* Adds the layers of the selected objects to the layer group filter that is selected in the tree view.
- *Replace:* Replaces the layers of the selected layer group filter with the layers of the objects that are selected in the drawing.

List View

Displays layers and layer filters and their properties and descriptions. If a layer filter is selected in the tree view, the list view displays only the layers in that layer filter. The All filter in the tree view displays all layers and layer filters in the drawing. When a layer property filter is selected and there are no layers that fit its definition, the list view is empty. You can use standard keyboard selection methods. To modify a property of a selected layer or of all layers in a selected filter, click the icon for that property. When a mixed icon or "Varies" is displayed for a layer filter, the property is not the same for all layers in the filter.

Status

Indicates the type of item: layer filter, layer in use, empty layer, or current layer.

Name

Displays the name of the layer or filter. Press F2 to enter a new name.

On

Turns the selected layers on and off. When a layer is on, it is visible and available for plotting. When a layer is off, it is invisible and not plotted, even if Plot is on.

Freeze

Freezes the selected layers in all viewports. You can freeze layers to speed up ZOOM, PAN, and many other operations; improve object selection performance; and reduce regeneration time for complex drawings.

Objects on frozen layers are not displayed, plotted, hidden, rendered, or regenerated.

Freeze the layers you want to be invisible for long periods. If you plan to switch visibility settings frequently, use the On/Off setting to avoid regenerating the drawing. You can freeze a layer in all viewports, in the current layout viewport, or in new layout viewports as they are created.

Lock

Locks and unlocks the selected layers. Objects on a locked layer cannot be modified.

Color

Changes the color associated with the selected layers. Clicking the color name displays the Select Color dialog box.

Linetype

Changes the linetype associated with the selected layers. Clicking the linetype name displays the Select Linetype dialog box.

Lineweight

Changes the lineweight associated with the selected layers. Clicking the lineweight name displays the Lineweight dialog box.

Plot Style

Changes the plot style associated with the selected layers. If you are working with color-dependent plot styles (the PSTYLEPOLICY system variable is set to 1), you cannot change the plot style associated with a layer. Clicking the plot style displays the Select Plot Style dialog box.

Plot

Controls whether the selected layers are plotted. If you turn off plotting for a layer, the objects on that layer are still displayed. Layers that are off or frozen are not plotted, regardless of the Plot setting.

Current VP Freeze (available only from a layout tab)

Freezes selected layers in the current layout viewport. You can freeze or thaw layers in the current viewport without affecting layer visibility in other viewports.

Current VP Freeze is an override to the Thaw setting in the drawing. That is, you can freeze a layer in the current viewport if it's thawed in the drawing, but you can't thaw a layer in the current viewport if it's frozen or off in the drawing. A layer is not visible when it is set to Off or Frozen in the drawing.

New VP Freeze (available only from a layout tab)

Freezes selected layers in new layout viewports. For example, freezing the DIMENSIONS layer in all new viewports restricts the display of dimensions on that layer in any newly created layout viewports but does not affect the DIMENSIONS layer in existing viewports. If you later create a viewport that requires dimensions, you can override the default setting by changing the current viewport setting.

Description

(Optional) Describes the layer or the layer filter.

List View Shortcut Menu

Provides options for modifying the list and modifying selected layers and layer filters.

Maximize Column

Changes the width of the column to maximize display of the column content. This option is available on the shortcut menu that is displayed when you right-click a column heading.

Maximize All Columns

Changes the width of all columns to maximize display of the column content. This option is available on the shortcut menu that is displayed when you right-click a column heading.

Show Filter Tree

Displays the tree view. Clear this option to hide the tree view.

Show Filters in Layer List

Displays filters at the top of the list view. The filters are listed in alphabetical order. Clear the check mark to display only layers in the list view.

Set Current

Sets the selected layer as the current layer. (CLAYER system variable)

New Layer

Creates a new layer. The list displays a layer named LAYER1. You can edit this layer name immediately. The new layer inherits the properties of the currently selected layer in the layer list (color, on or off state, and so on).

Delete Layer

Deletes selected layers from the drawing file definition. 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.

Note 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.

Change Description

Adds a description or changes the text in the description for the selected layer or filter. The description for a filter is added to the filter and to all layers in the filter.

Remove from Group Filter

Removes the selected layers from the group layer filter that is selected in the tree view.

Select All

Selects everything displayed in the list view.

Clear All

Removes selection from all items in the list view except the most recently selected layer or filter.

Select All but Current

Selects everything displayed in the list view except the current layer.

Invert Selection

Selects everything displayed in the list view except the items that are currently selected.

Invert Layer Filter

Displays all layers that do not meet the criteria in the selected layer property filter.

Layer Filters

Displays a list of layer filters, including All and All Used Layers. Click a filter to apply it to the list view.

Save Layer States

Displays the New Layer State to Save dialog box , in which you save the layer state and layer property settings of all layers in a drawing.

Restore Layer State

Displays the Layer States Manager , in which you can select a named layer state to restore settings of all layers in the drawing. Restores only those layer state and property settings that were selected when the named layer state was saved.

LAYOUT

Creates and modifies drawing layout tabs

Layouts toolbar:

 Insert Menu: Layout

 Command line: layout

Enter layout option [Copy/Delete/New/Template/Rename/SAveas/Set/?] <set>:

Note Many of these options are available by right-clicking a layout tab name.

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings

⬡ **Insert menu:** Layout➤ Layout Wizard

⬡ **Tools menu:** Wizards➤ Create Layout

⌨ **Command line: layoutwizard**

The Layout wizard is displayed.

The Layout wizard contains a series of pages that step you through the process of creating a new layout. You can choose to create a new layout from scratch or use an existing layout template on which to base your new layout.

Depending on which plotting device is currently configured, you can select a paper size from those available. You can select a predefined title block to apply to your new layout. The Preview area displays a preview image of the title block you select.

When you've finished using the wizard to specify your layout settings, you can modify any of the settings using the PAGESETUP command from within the new layout.

LEADER

Creates a line that connects annotation to a feature

⌨ **Command line: leader**

Specify leader start point:

Specify next point:

A leader line segment is drawn and prompts for points and options are displayed.

Specify **next point** or [**Annotation**/**Format**/**Undo**] <Annotation>: *Specify a point, enter an option, or press*ENTER

Point Specification

Point Specification

Draws a leader line segment to the point specified and continues to prompt you for points and options.

Specify next point or [Annotation/Format/Undo] <Annotation>: *Specify a point, enter an option, or press*ENTER

Annotation

Annotation

Inserts an annotation at the end of the leader line. The annotation can be single or multiple lines of text, a feature control frame containing geometric tolerances, or a block.

   Enter first line of annotation text or <options>: *Enter text or press*ENTER

If you enter text at the Annotation prompt, the text is inserted at the end of the leader line. You are prompted for additional lines of text until you end the command by pressing ENTER twice.
If you press ENTER at the Annotation prompt without entering text first, the following prompt is displayed:

   Enter an annotation option [Tolerance/Copy/Block/None/Mtext] <Mtext>: *Enter an option or press*ENTER

Tolerance

Creates a feature control frame containing geometric tolerances using the Geometric Tolerance dialog boxes (see TOLERANCE).

You can create datum indicators and basic dimension notation in these dialog boxes. After you specify the geometric tolerance, LEADER ends.
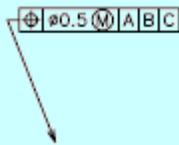
Copy

Copies text, a multiline text object, a feature control frame with geometric tolerances, or a block and connects the copy to the end of the leader line. The copy is associated with the leader line, meaning that if the copied object moves, the end of the leader line moves with it. The display of the hook line depends on the object copied.

   Select an object to copy:

The object is inserted and LEADER ends. The value of the current text gap (see DIMSTYLE or the DIMGAP system variable) determines where the text and multiline text objects are inserted. Blocks or feature control frames with geometric tolerances are attached to the end of the leader line.

Block

Inserts a block at the end of the leader line. The prompts are the same as for INSERT. The block reference is inserted at an offset from the end of the leader line and is associated to the leader line, meaning that if the block moves, the end of the leader line moves with it. No hook line is displayed.

   Enter block name or [?]: *Enter a block name or enter* ? *to list all blocks in the drawing*
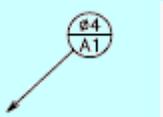
None

Ends the command without adding any annotation to the leader line.

Mtext

Creates text using the In-Place Text Editor when you specify an insertion point and a second point for the text boundary.

Enter the characters for the text. Enclose format strings for prefixes and suffixes in angle brackets (< >). Enclose format strings for alternate units in square brackets ([ ]).

The units settings and the current text style determine how the text is displayed. The multiline text is vertically centered and horizontally aligned according to the *X* axis direction of the last two vertices of the leader line. The text is offset from the hook line by the distance specified under Offset from Dim Line on the Text tab of the New, Modify, or Override Dimension Style dialog box. If the offset specified is negative, the multiline text is enclosed in a box as a basic dimension.

TRUE POSITION
PER DATUM
REFERENCE
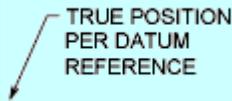
After you specify the text, LEADER ends.

Format

Format

Controls the way the leader is drawn and whether it has an arrowhead.

Enter leader format option [Spline/STraight/Arrow/None] <Exit>: *Enter an option or press*ENTER*to return to the previous prompt*

After each option, the Specify Next Point prompt is redisplayed.

spline leader line   straight leader line   sample arrowheads   no arrowhead
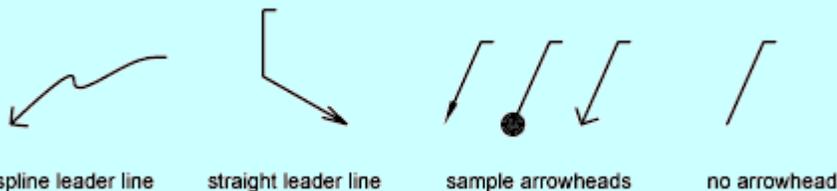
Spline

Draws the leader line as a spline. The vertices of the leader line are the control points, each of equal unit weight.

Straight

Draws the leader line as a set of straight line segments.

Arrow

Draws an arrowhead at the start point of the leader line.

None

Draws a leader line with no arrowhead at the start point.

Exit

Exits the Format options.

Undo

Undo

Undoes the last vertex point on the leader line. The previous prompt is displayed.

LENGTHEN

Changes the length of objects and the included angle of arcs

❖ **Modify menu:** Lengthen

⌨ **Command line:** lengthen

[Select an object](#) or [[DElta](#)/[Percent](#)/[Total](#)/[DYnamic](#)]: *Select one object or enter an option*

Object Selection

Object Selection

Displays the length and, where applicable, the included angle of the object.
LENGTHEN does not affect closed objects. The extrusion direction of the selected object need not be parallel to the *Z* axis of the current user coordinate system (UCS).

    Current length: *<current>*, included angle: *<current>*

    Select an object or [DElta/Percent/Total/DYnamic]: *Select one object, enter an option, or press* ENTER *to end the command*

Delta

Delta

Changes the length of an object by a specified increment, measured from the endpoint that is closest to the selection point. Delta also changes the angle of an arc by a specified increment, measured from the endpoint that is closest to the selection point. A positive value extends the object; a negative value trims it.

    Enter delta length or [Angle] *<current>*: *Specify a distance, enter* **a***, or press* ENTER

Delta Length

Changes the length of the object by the specified increment.

    Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

object selected

Angle

Changes the included angle of the selected arc by the specified angle.

    Enter delta angle *<current>*: *Specify an angle or press* ENTER

    Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

object selected

Percent

Sets the length of an object by a specified percentage of its total length. Percent also changes the angle of an arc by a specified percentage of the total included angle of the arc.

> Enter percentage length *<current>*: *Enter a positive nonzero value or press*ENTER

> Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

Total

Sets the length of a selected object by specifying the total absolute length from the fixed endpoint. Total also sets the included angle of a selected arc by a specified total angle.

> Specify total length or [Angle] *<current>*: *Specify a distance, enter a positive nonzero value,* enter **a**, *or press*ENTER
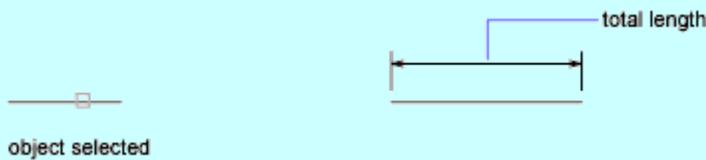
Total Length

Lengthens the object to the specified value from the endpoint that is closest to the selection point.

> Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

Angle

Sets the included angle of the selected arc.

> Specify total angle *<current>*: *Specify an angle or press*ENTER

> Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.
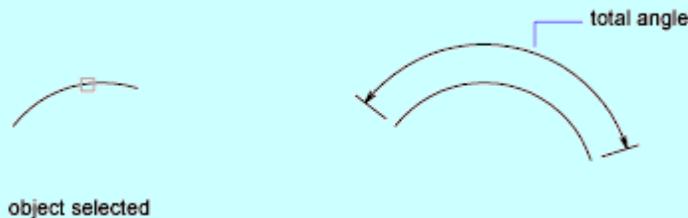
Dynamic

Turns on Dynamic Dragging mode. You change the length of a selected object by dragging one of its endpoints. The other end remains fixed.

> Select an object to change or [Undo]: *Select one object or enter* **u**

The prompt repeats until you press ENTER to end the command.

LIMITS

Sets and controls the limits of the grid display in the current Model or layout tab

⬙ **Format menu:** Drawing Limits

⌨ **Command line: limits** (or **'limits** for transparent use)

Specify **lower left corner** or [**ON**/**OFF**] *<current>*: *Specify a point, enter* **on** *or* **off***, or press*ENTER

LINE

Creates straight line segments

⬙ **Draw toolbar:**

⬙ **Draw menu:** Line

⌨ **Command line: line**

Specify first point: *Specify a point or press*ENTER *to* **continue** *from the last drawn line or arc*
Specify next point or [**Close**/**Undo**]:

Continue

Continues a line from the endpoint of the most recently drawn line.

before pressing ENTER   after pressing ENTER

If the most recently drawn line is an arc, its endpoint defines the starting point of the line, and the line is drawn tangent to the arc.
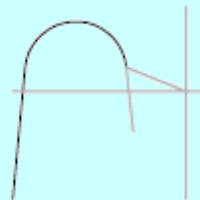
before pressing ENTER   after pressing ENTER

Close

Ends the last line segment at the beginning of the first line segment, which forms a closed loop of line segments. You can use Close after you have drawn a series of two or more segments.

before entering   after entering

Undo

Erases the most recent segment of a line sequence.

before entering   after entering

Entering **u** more than once backtracks through line segments in the order you created them.
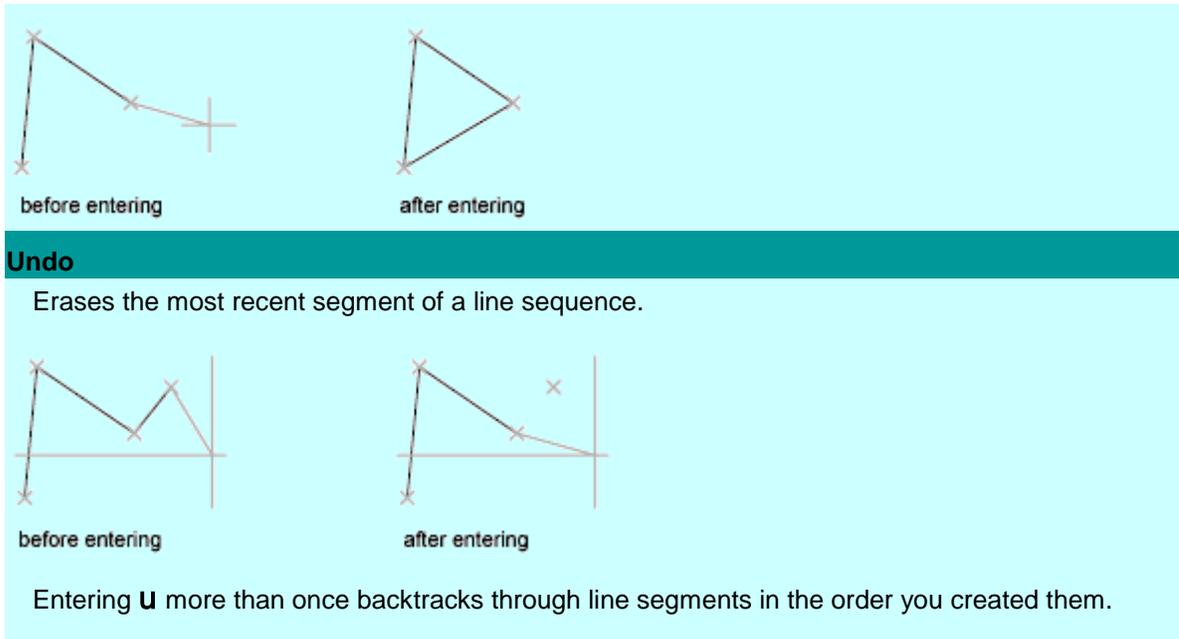
LINETYPE

Loads, sets, and modifies linetypes

🔷 **Format menu:** Linetype

⌨ **Command line: linetype** (or **'linetype** for transparent use)

The Linetype Manager is displayed.

If you enter **-linetype** at the Command prompt, LINETYPE displays prompts on the command line.

Loads linetypes and sets the current linetype.

Linetype Filters

Determines which linetypes to display in the linetype list. You can filter linetypes based on whether they are xref-dependent, or whether they are referenced by objects.

Invert Filter

Displays linetypes based on the opposites of the criteria you select. Linetypes that fit the inverse filter criteria are displayed in the linetype list.

Load

Displays the Load or Reload Linetypes dialog box, in which you can load into the drawing selected linetypes from the *acad.lin* file and add them to the linetype list.

Current

Sets the selected linetype to be the current linetype. Setting the current linetype to

BYLAYER means that an object assumes the linetype that is assigned to a particular layer.

Setting the linetype to BYBLOCK means that an object assumes the CONTINUOUS linetype until it is grouped into a block. Whenever the block is inserted, all objects inherit the block's linetype. The CELTYPE system variable stores the linetype name.

Delete

Deletes selected linetypes from the drawing. You can delete only unused linetypes. The BYLAYER, BYBLOCK, and CONTINUOUS linetypes cannot be deleted.

Note Be careful about deleting linetypes if you are working on a drawing in a shared project or one based on a set of layering standards. The deleted linetype definition remains stored in the *acad.lin* or *acadiso.lin* file and can be reloaded.

Show Details or Hide Details

Controls whether the Details section of the Linetype Manager is displayed.

Current Linetype

Displays the current linetype name.

List of Linetypes

Displays the loaded linetypes according to the option specified in Linetype Filters. To quickly select all or clear all linetypes, right-click in the linetype list to display the shortcut menu.

Linetype

Displays names of loaded linetypes. To rename a linetype, select it and then click it again and enter a new name. BYLAYER, BYBLOCK, CONTINUOUS, and xref-dependent linetypes cannot be renamed. The linetype name can include up to 255 characters. Linetype names can contain letters, digits, blank spaces, and the special characters dollar sign ($), hyphen (–), and underscore (_). Linetype names cannot include the special characters comma (,), colon (:), equal sign (=), question mark (?), asterisk (*), less-than and greater-than symbols (> <), forward and back slashes (/ \), vertical bar (|), quote ("), or back quote (`).

Appearance

Displays a sample of selected linetypes.

Description

Displays descriptions of the linetypes, which can be edited in the Details area.

Details

Provides alternative access to properties and additional settings.

Name

Displays the selected linetype name, which can be edited. The linetype name can include up to 255 characters. Linetype names can contain letters, digits, blank spaces, and the special characters dollar sign ($), hyphen (–), and underscore (_). Linetype names cannot include the special characters comma (,), colon (:), equal sign (=), question mark (?), asterisk (*), less-than and greater-than symbols (> <), forward and back slashes (/ \), vertical bar (|), quote ("), or back quote (`).

Description

Displays the description of the selected linetype, which can be edited.

Use Paper Space Units for Scaling

Scales linetypes in paper space and model space identically. Useful when working with multiple viewports. (PSLTSCALE system variable)

Global Scale Factor

Displays the global scale factor for all linetypes. (LTSCALE system variable)

Current Object Scale

Sets linetype scale for newly created objects. The resulting scale is the global scale factor multiplied by the object's scale factor. (CELTSCALE system variable)

ISO Pen Width

Sets the linetype scale to one of a list of standard ISO values. The resulting scale is the global scale factor multiplied by the object's scale factor.

LIST

Displays database information for selected objects

Inquiry toolbar:

Tools menu: Inquiry ➤ List

Command line: list

Select objects: *Use an object selection method*

The text window displays the object type, object layer, and *X,Y,Z* position relative to the current user coordinate system (UCS) and whether the object is in model space or paper space.
LIST reports color, linetype, and lineweight information if these items are not set to BYLAYER. The thickness of an object is displayed if it is nonzero. *Z* coordinate information defines the elevation. If the extrusion direction of the entry differs from the *Z* axis (0,0,1) of the current UCS, LIST also reports the extrusion direction in UCS coordinates.
LIST reports additional information related to the specific object selected.

LOAD

Makes shapes available for use by the SHAPE command

Command line: load

The Select Shape File dialog box (a standard file selection dialog box) is displayed. Enter the shape file name, or select a file name from the list.
You must load a shape (SHP) file the first time you need it; it is loaded automatically thereafter. The shape file must be available each time you edit the drawing.

LOGFILEOFF

Closes the log file opened by LOGFILEON

⌨ **Command line:** logfileoff

The program stops recording the text window contents and closes the log file. You can also control the log file with the OPTIONS command. Use the Maintain a Log File option on the Open and Save tab in the Options dialog box to turn the log file off and on. Use the Files tab to change the location of the log file.
Each drawing saves a log file (with the extension *.log*) that may need periodic deletion as the number of log files continues to grow.

LOGFILEON

Writes the text window contents to a file

⌨ **Command line:** logfileon

The contents of the text window are recorded in the log file until you exit the program or use the LOGFILEOFF command.
You can also control the log file with the OPTIONS command. Use the Maintain a Log File option on the Open and Save tab in the Options dialog box to turn the log file off and on. Use the Files tab to change the location of the log file.
Each drawing saves a log file (with the extension *.log*) that may need periodic deletion as the number of log files continues to grow.

LTSCALE

Sets the global linetype scale factor

⌨ **Command line:** ltscale (or 'ltscale for transparent use)

Enter new linetype scale factor *<current>*: *Enter a positive real value or press*ENTER

Use LTSCALE to change the scale factor of linetypes for all objects in a drawing. Changing the linetype scale factor causes the drawing to be regenerated.

LTSCALE = 1

LTSCALE = .5

LTSCALE = .25

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units

▧ **Format menu:** Lineweight

Shortcut menu: Right-click the Lwtbutton on the status bar and choose Settings.

⌨ **Command line:** lweight (or 'lweight for transparent use)

The Lineweight Settings dialog box is displayed.

If you enter **-lweight** at the Command prompt, LWEIGHT displays prompts on the command line.

Sets the current lineweight, sets the lineweight units, controls the display and display scale of lineweights in the Model tab, and sets the DEFAULT lineweight value for layers.

Lineweights

Displays the available lineweight values. Lineweight values consist of standard settings including BYLAYER, BYBLOCK, and DEFAULT. The DEFAULT value is set by the LWDEFAULT system variable, which has an initial value of 0.01 inches or 0.25 mm. All new layers use the default setting. The lineweight value of 0 plots at the thinnest lineweight available on the specified plotting device and is displayed at one pixel wide in model space.

Current Lineweight

Displays the current lineweight. To set the current lineweight, select a lineweight from the lineweight list and choose OK.

Units for Listing

Specifies whether lineweights are displayed in millimeters or inches. You can also set Units for Listing by using the LWUNITS system variable.

Millimeters (mm)

Specifies lineweight values in millimeters.

Inches (in.)

Specifies lineweight values in inches.

Display Lineweight

Controls whether lineweights are displayed in the current drawing. If this option is selected, lineweights are displayed in model space and paper space. You can also set Display Lineweight by using the LWDISPLAY system variable. Regeneration time increases with lineweights that are represented by more than one pixel. Clear Display Lineweight if performance slows down when working with lineweights turned on in a drawing. This option does not affect how objects are plotted.

Default

Controls the DEFAULT lineweight for layers. The initial DEFAULT lineweight is 0.01 inches or 0.25 mm. (LWDEFAULT system variable)

Adjust Display Scale

Controls the display scale of lineweights on the Model tab. On the Model tab, lineweights are displayed in pixels. Lineweights are displayed using a pixel width in proportion to the real-world unit value at which they plot. If you are using a high-resolution monitor, you can adjust the lineweight display scale to better display different lineweight widths. The Lineweight list reflects the current display scale.
Objects with lineweights that are displayed with a width of more than one pixel may increase regeneration time. If you want to optimize performance when working in the Model tab, set the lineweight display scale to the minimum value or turn off lineweight display altogether.

MATCHPROP

Applies the properties of a selected object to other objects

⊗ **Standard toolbar:**

⊗ **Modify menu:** Match Properties

⌨ **Command line: matchprop** or **painter** (or **'matchprop** for transparent use)

Select source object: *Select the object whose properties you want to copy*

Current active settings: *Currently selected matchprop settings*

Select destination object(s) or [Settings]: *Enter* s *or select one or more objects to copy properties to*

Destination Object(s)

Specifies the objects to which you want to copy the properties of the source object. You can continue selecting destination objects, or press ENTER to apply the properties and end the command.

Settings

Displays the Property Settings dialog box, in which you can control which object properties to copy to the destination objects. By default, in the Property Settings dialog box all object properties are selected for copying.

Specifies which basic properties and special properties to copy from the source object to the destination objects.

Color

Changes the color of the destination object to that of the source object. Available for all objects.

Layer

Changes the layer of the destination object to that of the source object. Available for all objects.

Linetype

Changes the linetype of the destination object to that of the source object. Available for all objects except attributes, hatches, multiline text, points, and viewports.

Linetype Scale

Changes the linetype scale factor of the destination object to that of the source object. Available for all objects except attributes, hatches, multiline text, points, and viewports.

Lineweight

Changes the lineweight of the destination object to that of the source object. Available for all objects.

Thickness

Changes the thickness of the destination object to that of the source object. Available only for arcs, attributes, circles, lines, points, 2D polylines, regions, text, and traces.

Plot Style

Changes the plot style of the destination object to that of the source object. If you are working in color-dependent plot style mode (PSTYLEPOLICY is set to 1), this option is unavailable. Available for all objects.

Dimension

In addition to basic object properties, changes the dimension style of the destination object to that of the source object. Available only for dimension, leader, and tolerance objects.

Polyline

In addition to basic object properties, changes the width and linetype generation properties of the destination polyline to those of the source polyline. The fit/smooth property and the elevation of the source polyline are not transferred to the destination polyline. If the source polyline has variable width, the width property is not transferred to the destination polyline.

Text

In addition to basic object properties, changes the text style of the destination object to that of the source object. Available only for single-line and multiline text objects.

Viewport

In addition to basic object properties, changes the following properties of the destination paper space viewport to match those of the source viewport: on/off, display locking, standard or custom scale, shade plot, snap, grid, and UCS icon visibility and location.

The settings for clipping and for UCS per viewport and the freeze/thaw state of the layers are not transferred to the destination object.

Hatch

In addition to basic object properties, changes the hatch properties of the destination object to that of the source object. To match the hatch origin, use Inherit Properties in HATCH or HATCHEDIT. Available only for hatch objects.

Table

In addition to basic object properties, changes the table style of the destination object to that of the source object. Available only for table objects.

MEASURE

Places point objects or blocks at measured intervals on an object
Draw menu: Point ➤ Measure

Command line: measure

Select object to measure:

Specify **length of segment** or [**Block**]: *Specify a distance or enter* **b**

The points or blocks drawn by MEASURE are placed in the Previous selection set, so you can choose them all by entering **p** at the next Select Objects prompt. You can use the Node object snap to draw an object by snapping to the point objects. You can then remove the points by entering **erase previous**.

The markers are placed in the user coordinate system (UCS) of the object being measured (except for 3D polylines in the current UCS). Markers are always placed on the object, regardless of the elevation settings.

If you use point objects for the markers, you can make the points easier to see by changing their appearance with the PDMODE system variable.

Length of Segment

Places point objects at the specified interval along the selected object, starting at the endpoint

---

closest to the point you used to select the object.

Measurement of closed polylines starts at their initial vertex (the first one drawn).

Measurement of circles starts at the angle from the center set as the current snap rotation angle. If the snap rotation angle is 0, then the measurement of the circle starts to the right of center, on its circumference.
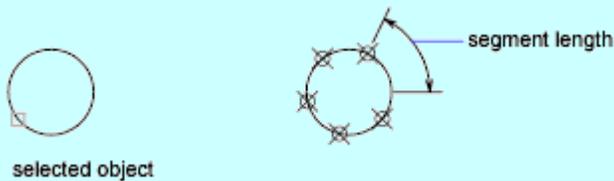
selected object

The illustration shows how MEASURE marks 0.5-unit distances along a polyline, with the PDMODE system variable set to 35.

select polylines   measure every 0.5 units
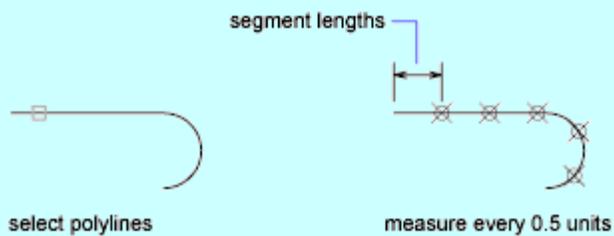
Block

Places blocks at a specified interval along the selected object.

Enter name of block to insert: *Enter the name of a block currently defined in the drawing*

Align block with object? [Yes/No] <Y>: *Enter* **y** *or* **n** *or press* ENTER

If you enter **y**, the block is rotated about its insertion point so that its horizontal lines are aligned with, and drawn tangent to, the object being measured. If you enter **n**, the block is always inserted with a 0 rotation angle.

Specify length of segment:

After you specify the segment length, the block is inserted at the specified interval. If the block has variable attributes, these attributes are not included.

MENU

Loads a customization file

⌨ **Command line: menu**

The Select Customization File dialog box (a standard file selection dialog box) is displayed. Enter or select a customization file name. When you press ENTER or choose Open, the named file is loaded.
A customization file is an XML-based file containing the menus, toolbars, tool palettes, and other interface elements that you can customize.
You can create a customized menu file and use MENU or CUI to load the file.

MINSERT

Inserts multiple instances of a block in a rectangular array

⌨ **Command line:** minsert

> Enter block name or [?]: *Enter a name, enter* **?** *to list the currently defined blocks in the drawing, or enter ~ to display the Select Drawing File dialog box*

Note You cannot precede the name of a block with an asterisk to explode the block's objects during insertion, as you can with <u>INSERT</u>.

> Specify **insertion point** or [**Scale**/**X**/**Y**/**Z**/**Rotate**/**PScale**/**PX**/**PY**/**PZ**/**PRotate**]: *Specify a point or enter an option*

Options at the insertion point preset the scale and rotation of a block before you specify its position. Presetting is useful for dragging a block using a scale factor and a rotation other than 1 or 0. If you enter one of the options, respond to the prompts by specifying a distance for the scale options or an angle for rotation.
Blocks inserted using MINSERT cannot be exploded.

Scale

Scale

Sets the scale factor for the *X*, *Y*, and *Z* axes. The scale for the *Z* axis is the absolute value of the specified scale factor.

> Specify scale factor for XYZ axes: *Enter a nonzero value*

> Specify insertion point:

> Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

> Enter number of rows (---) <1>: *Enter a positive value*

> Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

> Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one row, the following prompt is displayed:

> Specify distance between columns (| | |): *Enter a value or specify a distance*

X

X

Sets the *X* scale factor.

Specify X scale factor: *Enter a nonzero value*

Specify insertion point:

Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns (| | |): *Enter a value or specify a distance*

Y

Y

Sets the *Y* scale factor.

Specify Y scale factor: *Enter a nonzero value*

Specify insertion point:

Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns (| | |): *Enter a value or specify a distance*

Z

Z

Sets the *Z* scale factor.

Specify Z scale factor: *Enter a nonzero value*

Specify insertion point:

Specify rotation angle <0>:

The rotation angle sets the angle of the individual block inserts and also sets the angle of the entire array.

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value, or specify two points to define a box whose width and height represent the distance between rows and between columns*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns (| | |): *Enter a value or specify a distance*

Rotate

Rotate

Sets the angle of insertion for both the individual blocks and the entire array.

Specify rotation angle <0>:

Specify insertion point:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option or press* ENTER

X Scale Factor

Sets the *X* scale factor.

Enter Y scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value or specify a distance*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns (| | | |): *Enter a value or specify a distance*

Corner

Sets the scale factor by the specified point and the block insertion point.

Specify opposite corner: *Specify a point*

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value or specify a distance*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns (| | | |): *Enter a value or specify a distance*

XYZ

Specify X scale factor or [Corner] <1>: *Specify a nonzero value, enter* c*, or press* ENTER

You can determine the *X* and *Y* scale factors by entering a scale factor value or by specifying a corner point. If you specify a corner point, the specified point and the block insertion point determine the *X* and *Y* scale factors for the block.

Enter Z scale factor <use X scale factor>: *Enter a value or press* ENTER *to use the same scale factor*

Enter number of rows (---) <1>: *Enter a positive value*

Enter number of columns (| | |) <1>: *Enter a positive value*

If you specify more than one row, the following prompt is displayed:

Enter distance between rows or specify unit cell (---): *Enter a value or specify a distance*

If you specify more than one column and no unit cell, the following prompt is displayed:

Specify distance between columns (| | |): *Enter a value or specify a distance*

PScale

PScale

Sets the scale factor for the *X*, *Y*, and *Z* axes to control the display of the block as it is dragged into position.

Specify preview scale factor for XYZ axes: *Enter a nonzero value*

Specify insertion point:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

PX

PX

Sets the scale factor for the *X* axis to control the display of the block as it is dragged into position.

Specify preview X scale factor: *Enter a nonzero value*

Specify insertion point:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

PY

PY

Sets the scale factor for the *Y* axis to control the display of the block as it is dragged into position.

Specify preview Y scale factor: *Enter a nonzero value*

Specify insertion point:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

PZ

PZ

Sets the scale factor for the *Z* axis to control the display of the block as it is dragged into position.

Specify preview Z scale factor: *Enter a nonzero value*

Specify insertion point:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

PRotate

PRotate

Sets the rotation angle of the block as it is dragged into position.

Specify preview rotation angle:

Specify insertion point:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *Enter a value, enter an option, or press* ENTER

The descriptions of the X Scale Factor, Corner, and XYZ options match those of the corresponding options under Rotate.

MIRROR

Creates a mirror image copy of objects

 Modify toolbar:

 Modify menu: Mirror

 Command line: mirror

Select objects: *Use an object selection method and press* ENTER *to finish*

Specify first point of mirror line: *Specify a point*

Specify second point of mirror line: *Specify a point*
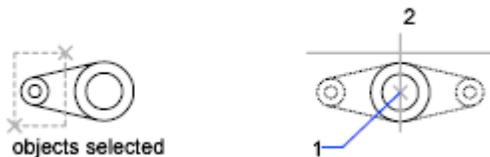
objects selected

The two specified points become the endpoints of a line about which the selected objects are mirrored. For mirroring in 3D, this line defines a mirroring plane perpendicular to the *XY* plane of the user coordinate system (UCS) containing the mirror line.

Erase source objects? [Yes/No] <N>: *Enter* **y** *or* **n**, *or press* ENTER

Yes

Places the mirrored image into the drawing and erases the original objects.

original objects
deleted

No

Places the mirrored image into the drawing and retains the original objects.

original objects
retained
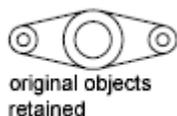
To manage the mirror properties of text objects, use the MIRRTEXT system variable. The default setting of MIRRTEXT is 1 (on), which causes a text object to be mirrored just like any other object. When MIRRTEXT is off (0), text is not reversed.

before mirroring   after mirroring   after mirroring
                          MIRRTEXT=1   MIRRTEXT=0
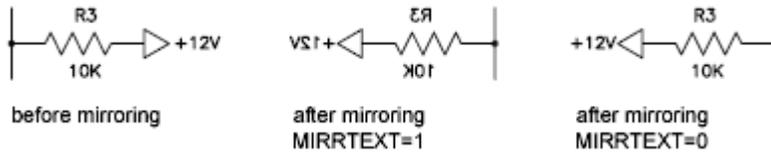
MLEDIT

Edits multiline intersections, breaks, and vertices
Modify menu: Object ➤ Multiline

Command line: mledit

The Multiline Edit Tools dialog box is displayed.
If you enter **-mledit** at the Command prompt, MLEDIT displays prompts on the command line.

Modifies multiline objects. The dialog box displays tools with sample images in four columns. The first column controls multilines that cross, the second controls multilines that form a *tee*, the third controls corner joints and vertices, and the fourth controls breaks in multilines.

Closed Cross

Creates a closed-cross intersection between two multilines.

Select first mline: *Select the foreground multiline*

Select second mline: *Select the intersecting multiline*

The closed-cross intersection is completed and the following prompt is displayed:
Select first mline or [Undo]: *Select another multiline or enter* **u**

First Mline

Edits another multiline. The Select Second Mline prompt is displayed.

first mline selected   second mline selected   result
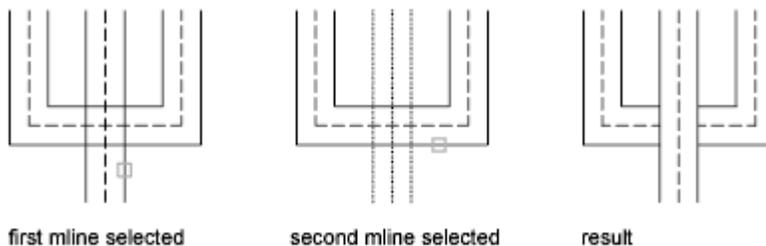
Undo

Undoes the closed-cross intersection. The Select First Mline prompt is displayed.

Open Cross

Creates an open-cross intersection between two multilines. Breaks are inserted in all elements of the first multiline and only the outside elements of the second multiline.

Select first mline: *Select a multiline*

Select second mline: *Select the intersecting multiline*

The open-cross intersection is completed, and the following prompt is displayed:

Select first mline or [Undo]: *Select another multiline or enter* **u**

First Mline

Edits another multiline. The Select Second Mline prompt is displayed.

first mline selected   second mline selected   result
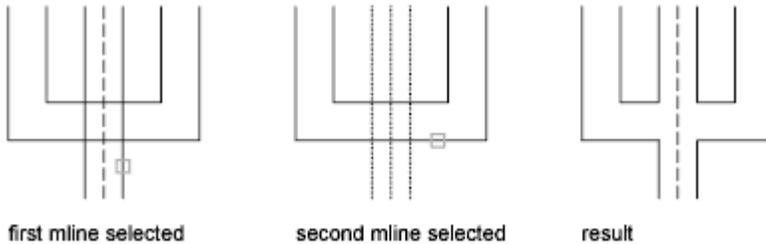
Undo

Undoes the open-cross intersection. The Select First Mline prompt is displayed.

Merged Cross

Creates a merged-cross intersection between two multilines. The order in which you select the multilines is not important.

Select first mline: *Select a multiline*

Select second mline: *Select the intersecting multiline*

The merged-cross intersection is completed, and the following prompt is displayed:

Select first mline or [Undo]: *Select another multiline or enter* **u**

First Mline

Edits another multiline. The Select Second Mline prompt is displayed.

first mline selected   second mline selected   result
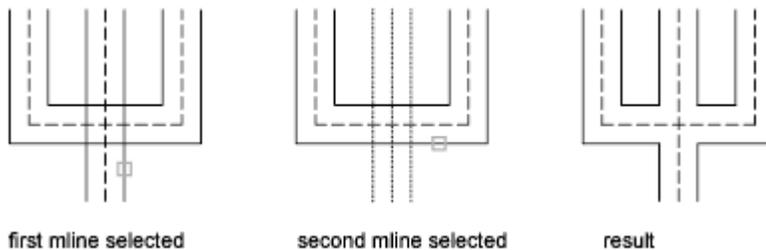
Undo

Undoes the merged-cross intersection. The Select First Mline prompt is displayed.

Closed Tee

Creates a closed-tee intersection between two multilines. The first multiline is trimmed or extended to its intersection with the second multiline.

Select first mline: *Select the multiline to trim*

Select second mline: *Select the intersecting multiline*

The closed-tee intersection is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**

First Mline

Edits another multiline. The Select Second Mline prompt is displayed.
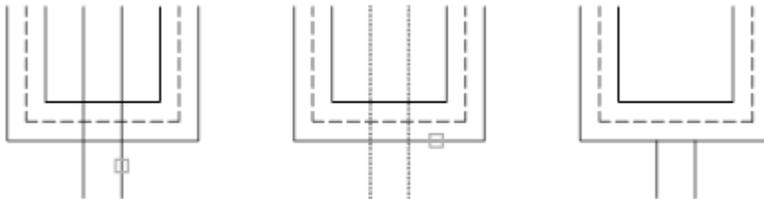
first mline selected   second mline selected   result

Undo

Undoes the closed-tee intersection. The Select First Mline prompt is displayed.

Open Tee

Creates an open-tee intersection between two multilines. The first multiline is trimmed or extended to its intersection with the second multiline.

Select first mline: *Select the multiline to trim or extend*

Select second mline: *Select the intersecting multiline*

The open-tee intersection is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**

First Mline

Edits another multiline. The Select Second Mline prompt is displayed.
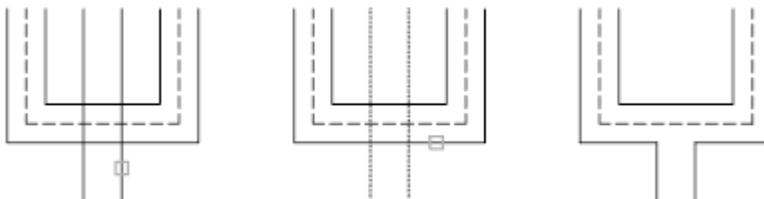
first mline selected   second mline selected   result

Undo

Undoes the open-tee intersection. The Select First Mline prompt is displayed.

Merged Tee

Creates a merged-tee intersection between two multilines.The multiline is trimmed or extended to its intersection with the other multiline.

Select first mline: *Select the multiline to trim or extend*

Select second mline: *Select the intersecting multiline*

The merged-tee intersection is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**

First Mline

Edits another multiline. The Select Second Mline prompt is displayed.

first mline selected   second mline selected   result
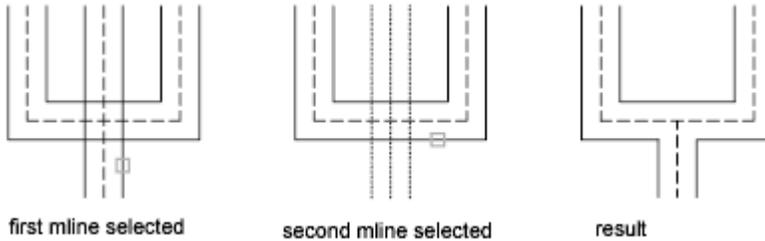
Undo

Undoes the merged-tee intersection. The Select First Mline prompt is displayed.

Corner Joint

Creates a corner joint between multilines. The multilines are trimmed or extended to their intersection.

Select first mline: *Select the multiline to trim or extend*

Select second mline: *Select the second half of the corner*

The corner joint is completed, and the following prompt is displayed:

Select first mline (or Undo): *Select another multiline or enter* **u**

First Mline

Edits another multiline. The Select Second Mline prompt is displayed.

first mline selected   second mline selected   result
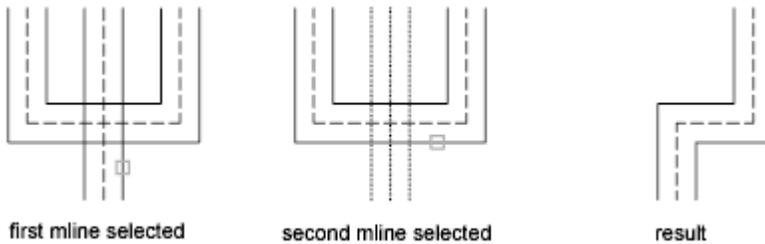
Undo

Undoes the corner joint. The Select First Mline prompt is displayed.

Add Vertex

Adds a vertex to a multiline.

Select mline: *Select a multiline*

A vertex is added at the selected point, and the following prompt is displayed:

Select mline or [Undo]: *Select another multiline or enter* **u**

Mline

Edits another multiline. The Select Mline prompt is displayed again.
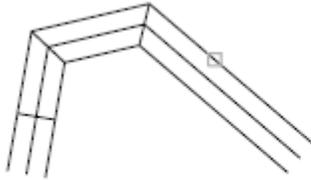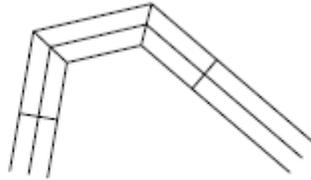
mline selected   result

Undo

Removes the added vertex. The Select Mline prompt is displayed.

Delete Vertex

Deletes a vertex from a multiline.

Select mline: *Select a multiline*

The vertex nearest to the selected point is deleted, and the following prompt is displayed:
Select mline or [Undo]: *Select another multiline or enter* **u**

Mline

Edits another multiline. The Select Mline prompt is displayed again.
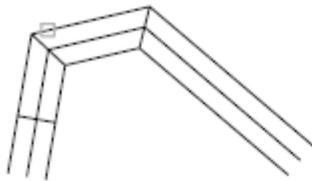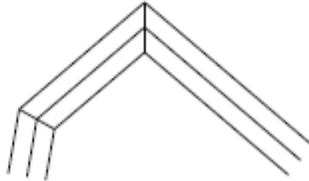
mline selected   result

Undo

Restores the deleted vertex. The Select Mline prompt is displayed.

Cut Single

Creates a visual break in a selected element of a multiline.

Select mline: *Select a multiline*

The selection point on the multiline is used as the first cut point, and the following prompt is displayed:
Select second point: *Specify the second cut point on the multiline*

The element is cut, and the following prompt is displayed:
Select mline or [Undo]: *Select another multiline or enter* **u**

Mline

Edits another multiline. The Select Mline prompt is displayed again.

first mline selected   second point selected   result
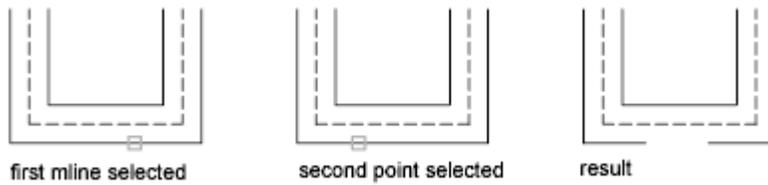
Undo

Undoes the cut. The Select Mline prompt is displayed.

Cut All

Creates a visual break through the entire multiline.

Select mline: *Select a multiline*

The selection point on the multiline is used as the first cut point, and the following prompt is displayed:

Select second point: *Specify the second cut point on the multiline*

All elements of the multiline are cut, and the following prompt is displayed:

Select mline or [Undo]: *Select another multiline or enter* **u**

Mline

Edits another multiline. The Select Mline prompt is displayed again.

first mline selected   second point selected   result
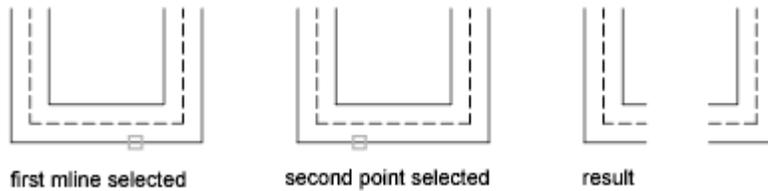
Undo

Undoes the cut. The Select Mline prompt is displayed.

Weld All

Rejoins multiline segments that have been cut.

Select mline: *Select a multiline*

The selection point on the multiline is used as the start of the weld, and the following prompt is displayed:

Select second point: *Specify the end of the weld on the multiline*

The multiline is welded, and the following prompt is displayed:

Select mline or [Undo]: *Select another multiline or enter* **u**

Mline

Edits another multiline. The Select Mline prompt is displayed again.
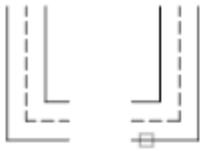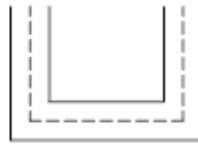
first mline selected   second point selected   result

Undo

Undoes the weld. The Select Mline prompt is displayed again.

MLINE

Creates multiple parallel lines

⬙ **Draw menu:** Multiline

⌨ **Command line:** **mline**

Current settings: Justification = *current*, Scale = *current*, Style = *current*

Specify **start point** or [**Justification**/**Scale**/**STyle**]: *Specify a point or enter an option*

Start Point

Start Point

Specifies the next vertex of the multiline.

Specify next point:

Specify next point or [Undo]: *Specify a point or enter* **u**

If you create a multiline with two or more segments, the prompt includes the Close option.

Specify next point or [Close/Undo]: *Specify a point or enter an option*
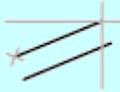
Next Point

Draws a multiline segment to the specified point using the current multiline style and continues to prompt for points.

Undo

Undoes the last vertex point on the multiline. The previous prompt is displayed.

Close

Closes the multiline by joining the last segments with the first segments.

Justification

Justification

Determines how the multiline is drawn between the points you specify.

Enter justification type [Top/Zero/Bottom] *<current>*: *Enter an option or press* ENTER

Top

Draws the multiline below the cursor, so that the line with the most positive offset is at the specified points.
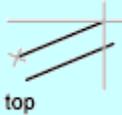
Zero

Draws the multiline with its origin centered at the cursor, so that the MLSTYLE Element Properties offset of 0.0 is at the specified points.
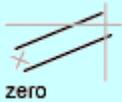
Bottom

Draws the multiline above the cursor, so that the line with the most negative offset is at the specified points.
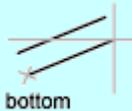
Scale

Scale

Controls the overall width of the multiline. This scale does not affect linetype scale.

Enter mline scale *<current>*: *Enter a scale or press* ENTER

The scale factor is based on the width established in the multiline style definition. A scale factor of 2 produces a multiline twice as wide as the style definition. A negative scale factor flips the order of the offset line—the smallest on top when the multiline is drawn from left to right. A negative scale value also alters the scale by the absolute value. A scale factor of 0 collapses the multiline into a single line. For information about the multiline style definition, see MLSTYLE.

Style

Style

Specifies a style to use for the multiline.

Enter mline style name or [?]: *Enter a name or enter* **?**

Style Name

Specifies the name of a style that has already been loaded or that's defined in a multiline library (MLN) file you've created.

?—List Styles

Lists the loaded multiline styles.

MLSTYLE

Creates, modifies, and manages multiline styles

Format menu: Multiline Style

Command line: mlstyle

Displays the Multiline Style dialog box.

Creates, modifies, saves, and loads multiline styles. The multiline style controls the number of elements and the properties of each element. MLSTYLE also controls the background color and the end caps of each multiline.

Current Multiline Style

Displays the name of the current multiline style to be used for subsequently created multilines.

Styles

Displays a list of multiline styles that are loaded in the drawing.

The list of multiline styles can include externally referenced multiline styles that is, multiline styles that exist in an externally referenced drawing (xref). Externally referenced multiline style names use the same syntax as other externally dependent nongraphical objects.

Description

Displays the description of the selected multiline style.

Preview Of

Displays the name and an image of the selected multiline style.

Set Current

Sets the current multiline style for subsequently created multilines. Select a name from the Styles list and choose Set Current.

Note You cannot make a multiline style from an xref the current style.

New

Displays the Create New Multiline Style dialog box, in which you can create a new multiline style.

Modify

Displays the Modify Multiline Style dialog box, in which you can modify a selected multiline style. You cannot modify the default STANDARD multiline style.

Note You cannot edit the element and multiline properties of the STANDARD multiline style or any multiline style that is being used in the drawing. To edit an existing multiline style, you must do so before you draw any multilines that use the style.

Rename

Renames the currently selected multiline style. You cannot rename the STANDARD multiline style.

Delete

Removes the currently selected multiline style from the Styles list. It does not delete the style from the MLN file.

You cannot delete the STANDARD multiline style, the current multiline style, or a multiline style that is in use.

Load

Displays the Load Multiline Styles dialog box, in which you can load multiline styles from a specified MLN file.

Save

Saves or copies a multiline style to a multiline library (MLN) file. If you specify an MLN file that already exists, the new style definition is added to the file and existing definitions are

not erased. The default file name is *acad.mln*.

MODEL

Switches from a layout tab to the Model tab

⌨ **Command line:** model

On the Model tab, you can create drawings in model space. The Model tab automatically sets the TILEMODE system variable to 1, and you can create model viewports to display various views of your drawing. Once you've completed your drawing, you can choose a layout tab to begin designing a layout environment from which to plot.
For possible performance gains when you switch between layout tabs or between the Model tab and a layout tab, use the LAYOUTREGENCTL system variable to control how the display list is updated.

MOVE

Moves objects a specified distance in a specified direction

⬡ **Modify toolbar:**

⬡ **Modify menu:** Move

Shortcut menu: Select the objects to move, and right-click in the drawing area. Click Move.

⌨ **Command line:** move

Select objects: *Use an object selection method and press* ENTER *when you finish*

Specify base point or [Displacement]<Displacement>: *Specify a base point or enter* **d**

Specify second point or <use first point as displacement>: *Specify a point or press* ENTER

The two points you specify define a vector that indicates how far the selected objects are to be moved and in what direction.
If you press ENTER at the Specify Second Point prompt, the first point is interpreted as a relative *X,Y,Z* displacement. For example, if you specify **2,3** for the base point and press ENTER at the next prompt, the objects move 2 units in the *X* direction and 3 units in the *Y* direction from their current position.

Displacement

Specify displacement *<last value>*: *Enter coordinates to represent a vector*

The coordinate values that you enter specify a relative distance and direction.

MREDO

Reverses the effects of several previous UNDO or U commands

Standard toolbar:

Command line: mredo

Enter **Number of Actions**number of actions or [**All**/**Last**]: *Specify an option, enter a positive number, or press*ENTER

Number of Actions
Reverses the specified number of actions.

All
Reverses all previous actions.

Last
Reverses only the last action.

MSPACE

Switches from paper space to a model space viewport

Command line: mspace

Commands operate in either model space or paper space. You use model space (the Model tab) to do drafting and design work and to create two-dimensional drawings or three-dimensional models. You use paper space (a layout tab) to create a finished layout of a drawing for plotting.

When you are in a layout, you can enter **mspace** on the command line to make the last viewport in the layout current, and then work in model space in that viewport within the layout. You can switch to model space by double-clicking a viewport, and you can switch to paper space by double-clicking an area of paper space.

MTEXT

Creates paragraphs of text as a single multiline text (mtext) object

Draw toolbar: A

Draw menu: Text ➤ Multiline Text

Pointing device: Double-click a multiline text object.

Command line: mtext

Current text style: *current* Text height: *current*

Specify first corner:

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]:

After you specify the point for the opposite corner, the in-place text editor is displayed.
If you specify one of the other options, or if you enter **-mtext** at the Command prompt, MTEXT
bypasses the in-place text editor and displays additional prompts on the command line.

Creates or modifies multiline text objects. Imports or pastes text from other files to use in
multiline text.
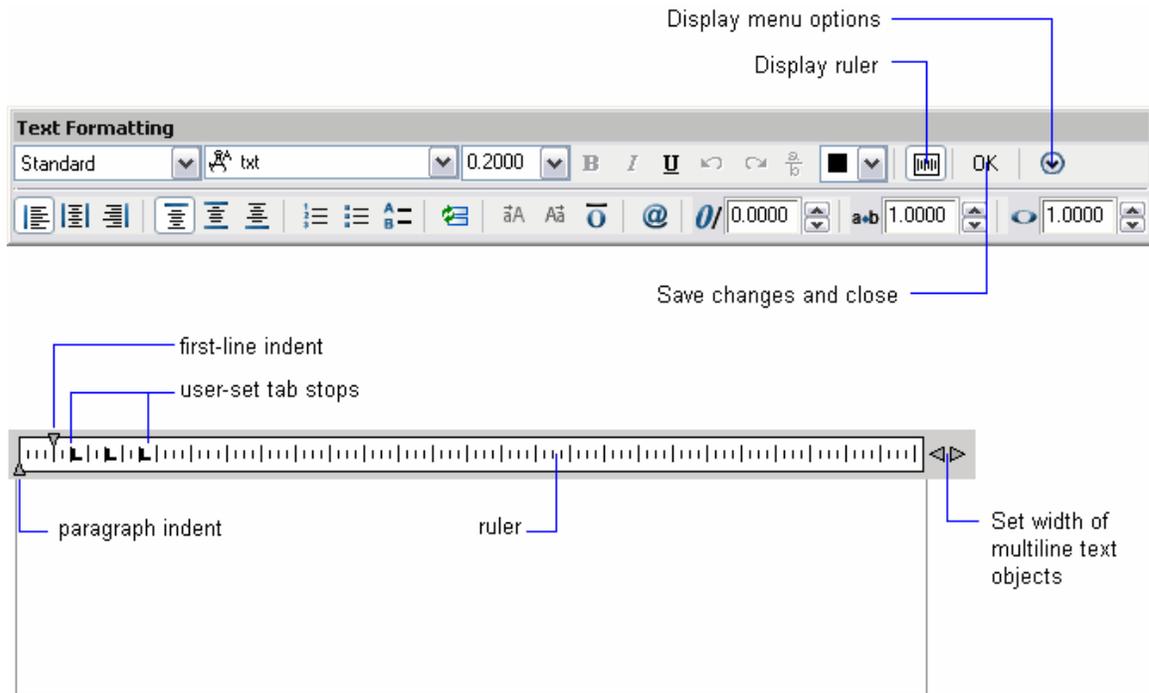
The In-Place Text Editor includes a **Text Formatting toolbar** and a **menu of options**. When a table
cell is selected for editing, the In-Place Text Editor displays column letters and row numbers.
If the text you enter overflows the bounding box you defined, a dashed line indicates the
defined width and height.

MULTIPLE

Repeats the next command until canceled

⌨ **Command line:** multiple

    Enter command name to repeat:

The command that you enter is repeated until you press ESC. Because MULTIPLE repeats only the command name, any parameters must be specified each time.
MULTIPLE does not repeat commands that display dialog boxes.

MVIEW

Creates and controls layout viewports

▨ **View menu:** Viewports ➤ 1 Viewport, 2 Viewports, 3 Viewports, 4 Viewports

⌨ **Command line:** mview

    Specify **corner of viewport** or [**ON**/**OFF**/**Fit**/**Shadeplot**/**Lock**/**Object**/**Polygonal**/**Restore**/**2**/**3**/**4**] <Fit>:
       Enter an option or specify a point

In a layout, you can create as many viewports as you want, but only up to 64 viewports can be active at one time (see MAXACTVP). Objects in model space are visible only in active viewports. Viewports that are not active are blank. Use the On and Off options to control whether viewports are active.

Corner of Viewport

Corner of Viewport

Specifies the first corner of a rectangular viewport.
    Specify opposite corner:

On

On

Makes a selected viewport active. An active viewport displays objects in model space. The MAXACTVP system variable controls the maximum number of viewports that can be active at one time. If your drawing contains more viewports than the number specified in MAXACTVP, you must turn one off to make another one active.
    Select objects: *Select one or more viewports*

Off

Off

Makes a selected viewport inactive. Objects in model space are not displayed in an inactive viewport.
    Select objects: *Select one or more viewports*

Fit

Fit

Creates one viewport that fills the layout to the edges of the printable area. When the paper background and printable area are turned off, the viewport fills the display.

fit

Shadeplot

Specifies how viewports in layouts are plotted.

Shade plot? [As displayed/Wireframe/Hidden/Rendered] *<As displayed>*: *Enter a shade plot option*

As Displayed

Specifies that a viewport is plotted the same way it is displayed.

Wireframe

Specifies that a viewport is plotted wireframe regardless of the current display.

Hidden

Specifies that a viewport is plotted with hidden lines removed regardless of the current display.

Rendered

Specifies that a viewport is plotted rendered regardless of the current display.

Select objects: *Select one or more viewports*

Lock

Prevents the zoom scale factor in the selected viewport from being changed when working in model space.

Viewport View Locking [ON/OFF]: *Enter* **on** *or* **off**

Select objects: *Select one or more viewports*

Object

Specifies a closed polyline, ellipse, spline, region, or circle to convert into a viewport. The polyline you specify must be closed and contain at least three vertices. It can be self-intersecting, and it can contain an arc as well as line segments.

Select object to clip viewport: *Select an object*

Polygonal

Creates an irregularly shaped viewport using specified points. The prompts are similar to those displayed when you specify a polygonal clip boundary for external references (xrefs), but you can specify arcs when you create a polygonal viewport boundary.

Specify start point: *Specify a point*

Specify next point or [Arc/Close/Length/Undo]: *Specify a point or enter an option*

Arc

Adds arc segments to the polygonal viewport.

[Angle/CEnter/CLose/Direction/Line/Radius/Second pt/Undo/Endpoint of arc] <Endpoint>: *Enter an option or press* ENTER

For a description of the options for creating arc segments, see the Arc option in PLINE.

Close

Closes the boundary. If you press ENTER after specifying at least three points, the boundary is closed automatically.

Length

Draws a line segment of a specified length at the same angle as the previous segment. If the previous segment is an arc, the new line segment is drawn tangent to that arc segment.

Undo

Removes the most recent line or arc segment added to the polygonal viewport.

Restore

Restore

Restores viewport configurations saved with the VPORTS command.

Enter viewport configuration name or [?] <*ACTIVE>: *Enter* **?***, enter a name, or press* ENTER

Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER

First Corner

Positions and sizes new viewports using the window selection method; the viewports are fit into the selected area.

Fit

Sizes the viewports to fill the drawing area.

2

2

Divides the specified area horizontally or vertically into two viewports of equal size.

Enter viewport arrangement [Horizontal/Vertical] <Vertical>: *Enter* **h** *or press* ENTER

Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER
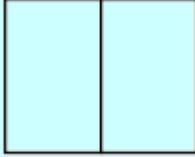
2/vertical

First Corner

Positions and sizes new viewports using the window selection method; the viewports are fit into the selected area.

Fit

Sizes the viewports to fill the drawing area.

3

3

Divides the specified area into three viewports.

Enter viewport arrangement

[Horizontal/Vertical/Above/Below/Left/<Right>: *Enter an option or press* ENTER

The Horizontal and Vertical options split the specified area into thirds. The other options split the area into three viewports: one large viewport and two smaller ones. The Above, Below, Left, and Right options specify where the larger viewport is placed.

Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER
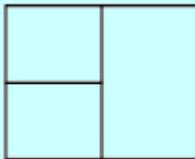
3/right

First Corner

Positions and sizes new viewports using the window selection method; the viewports are fit into the selected area.

Fit

Sizes the viewports to fill the drawing area.

4

4

Divides the specified area horizontally and vertically into four viewports of equal size.
   Specify first corner or [Fit] <Fit>: *Specify a point or press* ENTER
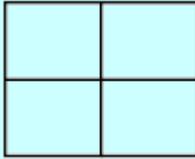
First Corner

Positions and sizes new viewports using the window selection method; he viewports are fit into the selected area.

Fit

Sizes the viewports to fill the drawing area.

MVSETUP

Sets up the specifications of a drawing

⌨ **Command line:** mvsetup

When you enter **mvsetup** at the command line, the prompts displayed depend on whether you are on the Model tab (model space) or on a layout tab (paper space).

On the Model tab, you set the units type, drawing scale factor, and paper size from the command line using MVSETUP. Using the settings you provide, a rectangular border is drawn at the grid limits.

On a layout tab, you can insert one of several predefined title blocks into the drawing and create a set of layout viewports within the title block. You can specify a global scale as the ratio between the scale of the title block in the layout and the drawing on the Model tab. The Model tab is most useful for plotting multiple views of a drawing within a single border.

To easily specify all layout page settings and prepare your drawing for plotting, you can also use the Page Setup dialog box, which is automatically displayed when you select a layout in a new drawing session.

NEW

Creates a new drawing
🗋 **File menu:** New

⌨ **Command line:** new

The behavior of the NEW command is determined by the Startup setting on the System tab in the Options dialog box.

- *Show the Startup Dialog Box:* NEW displays the Create New Drawing dialog box.

- *Do Not Show the Startup Dialog Box:* NEW displays the Select Template dialog box (a standard file selection dialog box).

If the FILEDIA system variable is set to 0 instead of 1, a prompt on the command line is displayed. If you set FILEDIA to 0, this prompt is displayed regardless of the Startup setting.

OFFSET

Creates concentric circles, parallel lines, and parallel curves

Modify toolbar:

Modify menu: Offset

Command line: offset

Current settings: Erase source = *current* Layer = *current* OFFSETGAPTYPE = *current*

Specify offset distance or [Through/Erase/Layer] <*current*>:*Specify a distance, enter an option, or press ENTER*

polyline

polyline with offset

The OFFSET command repeats for convenience. To exit the command, press ENTER.

Offset Distance

Offset Distance

Creates an object at a specified distance from an existing object.

> Select object to offset or [Exit/Undo]<exit>: *Select one object, enter an option, or press* ENTER *to end the command*

> Specify point on side to offset or [Exit/Multiple/Undo] <exit or next object>: *Specify a point (1) on the side of the object you want to offset or enter an option*
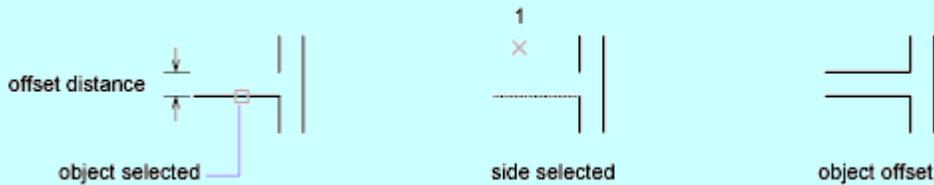
offset distance | object selected | side selected | object offset

Exit

Exits the OFFSET command.

Multiple

Enters the Multiple offset mode, which repeats the offset operation using the current offset distance.

Undo

Reverses the previous offset.

Through

Through

Creates an object passing through a specified point.

> Select object to offset or <exit>: *Select one object or press* ENTER *to end the command*

> Specify through point or [Exit/Multiple/Undo]<exit or next object>: *Specify a point (1) through which you want the offset object to pass or enter a distance*

object selected | through point | object offset

Exit

Exits the OFFSET command.

Multiple

Enters the Multiple offset mode, which repeats the offset operation and accepts additional through points.

Undo

Reverses the previous offset.

Erase

Layer

OOPS

Restores erased objects

⌨ **Command line:** oops

OOPS restores objects erased by the last ERASE command.

erased objects   before OOPS   after OOPS
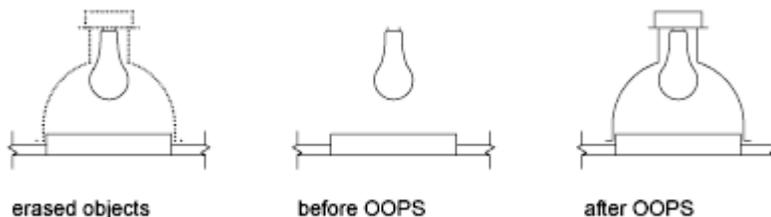
You can also use OOPS after BLOCK or WBLOCK because these commands can erase the selected objects after creating a block. However, you cannot use OOPS to restore parameters, actions, or grips in the Block Editor.
You cannot use OOPS to restore objects on a layer that has been removed with the PURGE command.

OPEN

⬙ **Standard toolbar:**

⬙ **File menu:** Open

⌨ **Command line:** open

Displays the drawing views and layers available for specifying what geometry to load into the selected drawing. When working with large drawing files, you can select the minimal amount of geometry you need to load when opening a drawing.
When a drawing is partially open, all named objects, as well as the specified geometry, are loaded into the file. Named objects include blocks, dimension styles, layers, layouts, linetypes, text styles, UCSs, views, and viewport configurations.

View Geometry to Load

Displays the selected view and available views in the drawing. Only model space views are available for loading. You can load paper space geometry by loading the layer that the paper space geometry is drawn on.

View Name

Displays the currently selected view. Geometry that is common to both the selected view and the layers is loaded into the drawing.

View List

Displays all the model space views available in the selected drawing file. When a view is selected, the program loads only the geometry in the selected view. The default view is Extents. Geometry that is common to both the selected view and the layers is loaded into the drawing. You can load geometry from only one view.

You can use the PARTIALOAD command to load additional geometry into a partially open drawing.

Note When a drawing is partially open, named objects are still loaded into the file along with the specified geometry from the selected view. All views are still available in the partially open drawing, but only geometry from the view specified to load is displayed in the drawing.

Layer Geometry to Load

Displays all the layers available in the selected drawing file. The geometry on selected layers is loaded into the drawing, including both model space and paper space geometry. The default loads no layer geometry into the drawing, but you can load geometry from one or more layers. If you specify to load no layer geometry into the drawing, no geometry from the selected view is loaded either because no layer geometry in that view or in the entire drawing is loaded. 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 selected drawing are not displayed in the Layer Geometry to Load list.
You can use the PARTIALOAD command to load additional geometry into a partially open drawing.

Note When a drawing is partially open, named objects are still loaded into the file along with the specified geometry from the selected layers. All layers are still available in the partially open drawing, but only geometry from the layers specified to load appears in the drawing.

Layer Name

Displays the layer names in the selected drawing.

Load Geometry

Loads geometry from the layer when the drawing is opened.

Load All

Loads geometry from all layers when the drawing is opened. You can right-click and use the shortcut menu to load geometry from all layers into the drawing.

Clear All

Loads no geometry from any layer when the drawing is opened. No geometry at all is loaded, including geometry specified to load from a view. You can right-click and use the shortcut menu to load no geometry into the drawing.

Index Status

Displays whether the selected drawing file contains a spatial or layer index. INDEXCTL controls whether layer and spatial indexes are saved with the drawing file.

Use Spatial Index

Controls whether a spatial index is used when partially opening a drawing. A spatial index can be used to locate what portion of the drawing is read; this minimizes the time required to open the drawing. If a drawing does not contain a spatial index, this option is unavailable.

Spatial Index

Displays whether the selected drawing file contains an index that organizes objects based on their location is space.

Layer Index

Displays whether the selected drawing file contains a list of the objects that are on each layer. A layer index can be used to locate what portion of the drawing is read; this minimizes the time required to open the drawing.

Unload All Xrefs on Open

Unloads all external references when opening the drawing. Selecting this option enhances the opening process. 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.

Open

Opens the drawing file, loading only combined geometry from the selected view and layers.

OPTIONS

Tools menu: Options

Shortcut menu: Right-click in the command window, or (with no commands active and no objects selected) right-click in the drawing area, and choose Options.

Command line: options

Customizes many settings.

Drawing Icon

Signifies that an option is saved with the drawing. An option saved with the drawing affects only the current drawing. An option saved in the registry (and not displayed with a drawing file icon) affects all drawings in a work session. Options that are saved in the registry are saved in the current profile.

Current Drawing

Displays the name of the current drawing above the tabs. (DWGNAME system variable)

The Options dialog box includes the following tabs:

- Files
- Display
- Open and Save
- Plot and Publish
- System
- User Preferences
- Drafting
- Selection
- Profiles

• **Files**

Lists the folders in which the program searches for support, driver, menu, and other files. Also lists optional, user-defined settings such as which dictionary to use for checking spelling.

Browse

Displays the Browse for Folder or Select a File dialog box (a standard file selection dialog box), in which you can specify a new location for the folder or file selected in the list.

Support File Search Path

Specifies the folders in which the program should look for text fonts, customization files, plug-ins, drawings to insert, linetypes, and hatch patterns that are not in the current folder.

Working Support File Search Path

Displays the active directories that the program searches for support files specific to your system. The list is read-only and displays valid paths from the Support Files Search Path that exist within the current directory structure and network mappings..

Device Driver File Search Path

Specifies the search path for device drivers for the video display, pointing devices, printers, and plotters.

Project Files Search Path

Specifies a project name for the drawing. The project name corresponds to a search path for external reference (xref) files associated with the project. You can create any number of project names with associated folders, but each drawing can have only one project name. (PROJECTNAME system variable)

Customization Files

Specifies the names and locations of various types of files.

Main Customization File

Specifies the default location of the main customization file (*acad.cui*).

Enterprise Customization File

Specifies the location of an enterprise customization file.

Help and Miscellaneous File Names

Specifies the names and locations of various types of files.

Help File

Specifies the location of the Help file.

Default Internet Location

Specifies the default Internet location used by both the BROWSER command and the Browse the Web button on the Web toolbar.

Configuration File

Specifies the location of the configuration file used to store hardware device driver information. This value is read-only and can be changed only by using the /c command line switch.

Text Editor, Dictionary, and Font File Names

Specifies a number of optional settings.

Text Editor Application

Specifies the text editor application to use for editing multiline text objects. (MTEXTED system variable)

Main Dictionary

Specifies the dictionary to use for checking spelling. (DCTMAIN system variable)

Custom Dictionary File

Specifies a custom dictionary to use (if you have one). (DCTCUST system variable)

Alternate Font File

Specifies the location of the font file to use if the original font cannot be located and an alternate font is not specified in the font mapping file. If you click Browse, the Alternate Font dialog box is displayed, from which you can choose an available font. (FONTALT system variable)

Font Mapping File

Specifies the location of the file that defines how to convert fonts that cannot be found. (FONTMAP system variable)

Print File, Spooler, and Prolog Section Names

Specifies settings related to plotting.

Plot File Name for Legacy Plotting Scripts

Specifies a default name for the temporary plot files used with plotting scripts created with AutoCAD Release 14 or earlier. The default name is the *drawing name* plus the *.plt* file extension. The default name used with AutoCAD 2000 and later drawings is the *drawing name-layout name* plus the *.plt* file name extension. Some plotting device drivers, however, use a different plot file extension. This option affects only the default plot file name used for plotting scripts created with earlier versions of AutoCAD.

Print Spool Executable

Specifies the application to use for print spooling. You can enter the executable file name as well as any command line arguments you want to use. For example, you can enter **myspool.bat %s** to spool plot files to *myspool.bat* and have a unique plot file name automatically generated.

Printer Support File Path

Specifies search path settings for printer support files.

Print Spooler File Location

Specifies the path for print spool files.

Printer Configuration Search Path

Specifies the path for printer configuration files (PC3 files).

Printer Description File Search Path

Specifies the path for files with a *.pmp* file extension, or printer description files.

Plot Style Table Search Path

Specifies the path for files with an *.stb* or *.ctb* extension, or plot style table files (both named plot style tables and color-dependent plot style tables).

Automatic Save File Location

Specifies the path for the file created when you select Automatic Save on the Open and Save tab. (SAVEFILEPATH system variable)

Color Book Locations

Specifies the path for color book files that can be used when specifying colors in the Select Color dialog box. You can define multiple folders for each path specified. This option is saved with the user profile.

Data Sources Location

Specifies the path for database source files. Changes to this setting do not take effect until you close and restart the program.

Template Settings

Specifies the drawing template settings.

Drawing Template File Location

Specifies the path to locate drawing template files used by the Start Up wizard and New dialog box.

Sheet Set Template File Location

Specifies the path to locate sheet set template files used by the Create Sheet Set wizard.

Default Template File Name for QNEW

Specifies the drawing template file used by the QNEW command.

Default Template for Sheet Creation and Page Setup Overrides

Specifies the default template file that is used for creating new sheets and to store page setup overrides that can be applied to Publish operations from the Sheet Set Manager.

Tool Palettes File Locations

Specifies the path for tool palette support files.

Authoring Palette File Locations

Specifies the path for the Block Authoring Palettes support files. Block Authoring Palettes are used in the Block Editor and provide tools for creating dynamic blocks.

Log File Location

Specifies the path for the log file created when you select Maintain a Log File on the Open and Save tab. (LOGFILEPATH system variable)

Plot and Publish Log File Location

Specifies the path for the log file that is created if you select the Automatically Save Plot and Publish Log option on the Plot and Publish tab.

Temporary Drawing File Location

Specifies the location to store temporary files. This program creates temporary files and then deletes them when you exit the program. If you plan to run the program from a write-protected folder (for example, if you are working on a network or opening files from a CD), specify an alternate location for your temporary files. The folder you specify must not be write-protected. The TEMPPREFIX system variable (read-only) also stores the current location of temporary drawing files.

Temporary External Reference File Location

Specifies the location of external reference (xref) files. This location is used for the copy of the xref when you select Enabled with Copy in the Demand Load Xrefs list on the Open and Save tab. (XLOADPATH system variable)

Texture Maps Search Path

Specifies the folders to search for rendering texture maps.

i-drop Associated File Location

Specifies the location of data files associated with i-drop content. When the location is not specified, the location of the current drawing file is used.

Browse

Displays the Browse for Folder or Select a File dialog box, depending on what you selected in the List of Folders and Files.

Add

Adds a search path for the selected folder.

Remove

Removes the selected search path or file.

Move Up

Moves the selected search path above the preceding search path.

Move Down

Moves the selected search path below the following search path.

Set Current

Makes the selected project or spelling dictionary current.

- ## Display

Customizes the display.

Window Elements

Controls display settings specific to the drawing environment.

Display Scroll Bars in Drawing Window

Displays scroll bars at the bottom and right sides of the drawing area.

Display Screen Menu

Displays the screen menu on the right side of the drawing area. The screen menu font is controlled by the Windows system font settings. If you use the screen menu, you should set the Windows system font setting to a font and font size that fits the screen menu size restrictions.

Use Large Buttons for Toolbars

Displays buttons in a larger format at 32 by 30 pixels. The default display size is 16 by 15 pixels.

Show Tooltips

Displays tooltips when you move the cursor over buttons on the toolbar.

Show Shortcut Keys in Tooltips

Displays shortcut keys when you move the cursor over buttons on the toolbar.

Colors

Displays the Color Options dialog box. Use this dialog box to specify colors of elements in the main application window.

Fonts

Displays the Command Line Window Font dialog box. Use this dialog box to specify the font for the command window text.

Layout Elements

Controls options for existing and new layouts. A layout is a paper space environment in which you can set up drawings for plotting.

Display Layout and Model Tabs

Displays the layout and Model tabs at the bottom of the drawing area.

Display Printable Area

Displays printable area in a layout. The printable area is represented by the area within the dashed line and is determined by the selected output device. Objects drawn outside of the printable area are clipped or omitted when the drawing is plotted.

Display Paper Background

Displays a representation of the specified paper size in a layout. The paper size and plot scale determine the size of the paper background.

Display Paper Shadow: Displays a shadow around the paper background in a layout. This option is unavailable if the Display Paper Background option is cleared.

Show Page Setup Manager for New Layouts

Displays the Page Setup Manager the first time you click a layout tab. Use this dialog box to set options related to paper and plot settings.

Create Viewport in New Layouts

Creates a single viewport automatically when you create a new layout.

Crosshair Size

Controls the size of the crosshairs. The valid range is from 1 to 100 percent of the total screen. At 100 percent, the ends of the crosshairs are never visible. When the size is decreased to 99 percent or below, the crosshairs have a finite size, and the ends of the crosshairs are visible when situated at the edge of the drawing area. The default size is 5 percent. (CURSORSIZE system variable)

Display Resolution

Controls the quality of the display of objects. If you set high values to improve display quality, the impact on performance is significant.

Arc and Circle Smoothness

Controls the smoothness of circles, arcs, and ellipses. A higher number produces smoother objects, but more time is required to regenerate, pan, and zoom the objects. You can improve performance by setting this option to a low value such as 100 for drawing, and increasing the value for rendering. The valid range is 1 to 20,000. The default setting is 1000. This setting is saved in the drawing. To change the default for new drawings, consider specifying this setting in the template files on which you base your new drawings. (VIEWRES command)

Segments in a Polyline Curve

Sets the number of line segments to be generated for each polyline curve. The higher the number, the greater the performance impact. Set this option to a low value such as 4 to optimize performance for drawing. Values range from -32767 to 32767. The default setting is 8. This setting is saved in the drawing. (SPLINESEGS system variable)

Rendered Object Smoothness

Controls the smoothness of shaded and rendered curved solids. The value you enter for Rendered Object Smoothness is multiplied by the value you enter for Arc and Circle Smoothness to determine how to display solid objects. To improve performance, set Rendered Object Smoothness to 1 or less when drawing. A higher number decreases display performance and increases rendering time. The valid range is 0.01 to 10. The default setting is 0.5. This setting is saved in the drawing. (FACETRES system variable)

Contour Lines per Surface

Sets the number of contour lines per surface on objects. A higher number decreases display performance and increases rendering time. The valid range is 0 to 2047. The default setting is 4. This setting is saved in the drawing. (ISOLINES system variable)

Display Performance

Controls display settings that affect performance.

Pan and Zoom with Raster and OLE

Controls the display of raster images and OLE objects when you use Realtime PAN and ZOOM. Clear this option to optimize performance. If dragging display is turned on and you select Pan and Zoom with Raster and OLE, a copy of the object moves with the cursor as you reposition the original. Dragging display controls whether an outline of the object is displayed while you drag it. The DRAGMODE system variable controls dragging display. (RTDISPLAY system variable)

Highlight Raster Image Frame Only

Controls the display of raster images during selection. If this option is selected, only the frame of the raster image is highlighted when selected. Select this option to optimize performance. (IMAGEHLT system variable)

Apply Solid Fill

Displays solid fills in objects. You must regenerate the drawing for this setting to take effect. This setting is saved in the drawing. Clear this option to optimize performance. (FILLMODE system variable)

Objects affected by FILL include hatches (including solid-fill), two-dimensional solids, wide polylines, multilines, and traces.

Show Text Boundary Frame Only

Displays the frames for text objects instead of displaying the text objects. After you select or clear this option, you must use REGEN to update the display. This setting is saved in the drawing. Select this option to optimize performance. (QTEXTMODE system variable)

Show Silhouettes in Wireframe

Controls whether silhouette curves of 3D solid objects are displayed as wireframes. This option also controls whether mesh is drawn or suppressed when a 3D solid object is hidden. This setting is saved in the drawing. Clear this option to optimize performance. (DISPSILH system variable)

Reference Edit Fading Intensity

Specifies the fading intensity value for objects during in-place reference editing. With in-place reference editing, you can edit a block reference or external reference from within the current drawing. While references are being edited in place, objects that are not being edited are displayed at a lesser intensity than objects that can be edited. The valid range is 0 through 90 percent. The default setting is 50 percent. (XFADECTL system variable)

• Open and Save

Controls options that relate to opening and saving files.

File Save

Controls settings related to saving a file.

Save As

Displays the valid file formats used when saving a file with SAVE, SAVEAS, and QSAVE. The file format selected for this option is the default format that all drawings are saved as when you use SAVE, SAVEAS, and QSAVE.

Note AutoCAD 2004 is the drawing file format used by the AutoCAD 2004, AutoCAD 2005, and AutoCAD 2006 releases.

Thumbnail Preview Settings

Displays the Thumbnail Preview Settings dialog box, which controls whether thumbnail previews are updated when the drawing is saved.

Incremental Save Percentage

Sets the percentage of potentially wasted space in a drawing file. Full saves eliminate wasted space. Incremental saves are faster but they increase the size of your drawing. If you set Incremental Save Percentage to 0, every save is a full save. For optimum performance, set the value to 50. If hard disk space becomes an issue, set the value to 25. If you set the value to 20 or less, performance of the SAVE and SAVEAS commands slows significantly. (ISAVEPERCENT system variable)

File Safety Precautions

Assists in avoiding data loss and in detecting errors.

Automatic Save

Saves your drawing automatically at the intervals you specify. You can specify the location of all Autosave files by using the SAVEFILEPATH system variable. SAVEFILE (read-only) stores the name of the Autosave file.

Note Automatic save is disabled when the Block Editor is open.

Minutes Between Saves: When Automatic Save is on, specifies how often the drawing is saved. (SAVETIME system variable)

Create Backup Copy with Each Save

pecifies whether a backup copy of a drawing is created when you save the drawing. The backup copy is created in the same location as the drawing. (ISAVEBAK system variable)

Full-Time CRC Validation

Specifies whether a cyclic redundancy check (CRC) should be performed each time an object is read into the drawing. CRC is an error-checking mechanism. If your drawings are being corrupted and you suspect a hardware problem or a software error, turn this option on.

Maintain a Log File

Writes the contents of the text window to a log file. To specify the location and name of the log file, use the Files tab in the Options dialog box. You can also set the log file location by using the LOGFILEMODE system variable. The LOGFILENAME system variable (read-only) stores the log file name of the current drawing.

File Extension for Temporary Files

Specifies a unique extension for temporary save files. The default extension is *.ac$.*

Security Options

Provides options for a digital signature and password that are invoked when you save a file.

Display Digital Signature Information

Presents digital signature information when a file with a valid digital signature is opened. (SIGWARN system variable)

File Open

Controls settings that relate to recently used files and open files.

Number of Recently Used Files to List

Controls the number of recently used files that are listed in the File menu for quick access. Valid values are 0 to 9.

Display Full Path In Title

Displays the full path of the active drawing in the drawing's title bar, or in the application window title bar if the drawing is maximized.

External References (Xrefs)

Controls the settings that relate to editing and loading external references.

Demand Load Xrefs

Controls demand loading of xrefs. Demand loading improves performance by loading only the parts of the referenced drawing needed to regenerate the current drawing. (XLOADCTL system variable)

- *Disabled:* Turns off demand loading.
- *Enabled:* Turns on demand loading and improves performance. Select the Enabled setting to enhance the loading process when working with clipped xrefs that contain a spatial or layer index. When this option is selected, other users cannot edit the file while it is being referenced.
- *Enabled with Copy:* Turns on demand loading but uses a copy of the referenced drawing. Other users can edit the original drawing.

Retain Changes to Xref Layers

Saves changes to layer properties and states for xref-dependent layers. When the drawing is reloaded, the properties currently assigned to xref-dependent layers are retained. This setting is saved in the drawing. (VISRETAIN system variable)

Allow Other Users to Refedit Current Drawing

Determines whether the current drawing file can be edited in place if it is being referenced by another drawing or multiple drawings. This setting is saved in the drawing. (XEDIT system variable)

ObjectARX Applications

Controls settings that relate to AutoCAD Runtime Extension applications and proxy graphics.

Demand Load ObjectARX Apps

Specifies if and when a third-party application is demand-loaded if a drawing contains custom objects created in that application. (DEMANDLOAD system variable)

- *Disable Load on Demand:* Turns off demand-loading.
- *Custom Object Detect:* Demand-loads the source application when you open a drawing that contains custom objects. This setting does not demand-load the application when you invoke one of the application's commands.
- *Command Invoke:* Demand-loads the source application when you invoke one of the application's commands. This setting does not demand-load the application when you open a drawing that contains custom objects.
- *Object Detect and Command Invoke:* Demand-loads the source application when you open a drawing that contains custom objects or when you invoke one of the application's commands.

Proxy Images for Custom Objects

Controls the display of custom objects in drawings. (PROXYSHOW system variable)

- *Do Not Show Proxy Graphics:* Specifies not to display custom objects in drawings.
- *Show Proxy Graphics:* Specifies to display custom objects in drawings.
- *Show Proxy Bounding Box:* Specifies to show a box in place of custom objects in drawings.

Show Proxy Information Dialog Box

Specifies whether a warning is displayed when you open a drawing that contains custom objects. (PROXYNOTICE system variable)

- ## Plot and Publish

Controls options related to plotting and publishing.

Default Plot Settings For New Drawings

Controls default plotting settings for new drawings or drawings created in AutoCAD Release 14 or earlier that have never been saved in AutoCAD 2000 or later format.

Use As Default Output Device

Sets the default output device for new drawings and for drawings created in AutoCAD Release 14 or earlier that have never been saved in AutoCAD 2000 or later format. The list displays any plotter configuration files (PC3) found in the plotter configuration search path and any system printers that are configured in the system.

Use Last Successful Plot Settings

Sets the plotting settings to match those of the last successful plot.

Add or Configure Plotters

Displays the Plotter Manager (a Windows system window). You can add or configure a plotter with the Plotter Manager.

Plot To File

Specifies the default location for plot to file operations. You can enter a location or click the [...] button to specify a new location.

Background Processing Options

Specifies options for background plotting and publishing. You can use background plotting to start a job you are plotting or publishing and immediately return to work on your drawing while your job is plotted or published as you work.
Backround plot and publish details are available through the Plot and Publish status bar icon shortcut menu.
You can plot or publish jobs in the background even if the program is not running by entering **acad/pl <DSD file name>** at the Windows Command Prompt.

Enable Background Plotting When

Specifies whether or not background plotting is turned on for plotting and publishing.

Plotting: Specifies that plot jobs are processed in the background. This setting is also affected by the BACKGROUNDPLOT system variable.

Publishing: Specifies that published jobs are processed in the background. This setting is also affected by the BACKGROUNDPLOT system variable.

Note When -PLOT, PLOT, -PUBLISH, and PUBLISH are used in a script (SCR file), the BACKGROUNDPLOT system variable value is ignored, and -PLOT, PLOT, -PUBLISH, and PUBLISH are processed in the foreground.

Plot and Publish Log File

Controls options for saving a plot and publish log file as a comma-separated value (CSV) file that can be viewed in a spreadsheet program.
This log file contains information about plot and publish jobs such as:

- Job ID
- Job name
- Sheet set name
- Category name
- Date and time started and completed
- Sheet name
- Full file path
- Selected layout name
- Page setup name
- Named page setup path
- Device name
- Paper size name
- Final status

Automatically Save Plot and Publish Log

Specifies that a log file that contains information about all plot and publish jobs is automatically saved.

Save One Continuous Plot Log

Specifies that a single log file that contains information about all plot and publish jobs is automatically saved.

Save One Log File Per Plot

Specifies that a separate log file is created for each plot and publish job.

General Plot Options

Controls options that relate to the general plotting environment, including paper size settings, system printer alert behavior, and OLE objects in a drawing.

Keep the Layout Paper Size If Possible

Uses the paper size specified in the Page Setup dialog box as long as the selected output device can plot to this paper size. If the selected output device cannot plot to this paper size, the program displays a warning message and uses the paper size specified either in the plotter configuration file (PC3) or in the default system settings if the output device is a system printer. You can also select this option by setting PAPERUPDATE to 0.

Use the Plot Device Paper Size

Uses the paper size specified either in the plotter configuration file (PC3) or in the default system settings if the output device is a system printer. You can also set this option by setting PAPERUPDATE to 1.

System Printer Spool Alert

Controls whether you are alerted when the plotted drawing is spooled through a system printer because of an input or output port conflict.

- *Always Alert (And Log Errors):* Alerts you and always logs an error when the plotted drawing spools through a system printer.
- *Alert First Time Only (And Log Errors):* Alerts you once and always logs an error when the plotted drawing spools through a system printer.
- *Never Alert (And Log First Error):* Never alerts you and logs only the first error when the plotted drawing spools through a system printer.
- *Never Alert (Do Not Log Errors):* Never alerts you or logs an error when the plotted drawing spools through a system printer.

OLE Plot Quality

Determines the quality of plotted OLE objects. The options are Monochrome, Low Graphics, High Graphics, and Automatically Select. The automatic option assigns a plot quality setting based on the type of OLE object. (OLEQUALITY system variable)

Use OLE Application When Plotting OLE Objects

Launches the application used to create the OLE object when plotting a drawing with OLE objects. You can use this option to optimize the quality of plotted OLE objects. This setting is saved in the drawing. (OLESTARTUP system variable)

Hide System Printer

Controls whether Windows system printers are displayed in the Plot and Page Setup dialog boxes. This option hides standard Windows system printers only. It does not hide Windows system printers that have been configured using the Add-a-Plotter wizard.

You can control the size of the list of devices in the Plot and Page Setup dialog boxes by moving a device's PC3 file out of the Plotters folder and its subfolders.

Specify Plot Offset Relative To

Specifies whether the offset of the plot area is from the lower-left corner of the printable area or from the edge of the paper (PLOTOFFSET system variable).

Printable Area

Specifies that the plot offset is relative to the printable area.

Edge of Paper

Specifies that the plot offset is relative to the edge of the paper.

Plot Stamp Settings

Opens the Plot Stamp dialog box.

Plot Style Table Settings

Opens the Plot Style Table Settings dialog box.

- # System

Controls system settings.

Current 3D Graphics Display

Controls settings that relate to system properties and configuration of the 3D graphics display system.

Current 3D Graphics Display System

Lists the available 3D graphics display systems. The default is the Heidi$^®$ 3D graphics display system (GSHEIDI10).

Properties

Displays a 3D Graphics System Configuration dialog box for the current 3D graphics display system. In the 3D Graphics System Configuration dialog box, you set options that affect the way objects are displayed and system resources are used in the 3D Orbit view. The options you set also affect the way objects are shaded with SHADEMODE.

Current Pointing Device

Controls options that relate to the pointing device.

Current Pointing Device

Displays a list of the available pointing device drivers.

- *Current System Pointing Device:* Sets the system pointing device as current.
- *Wintab Compatible Digitizer:* Sets the Wintab-compatible digitizer as current.

Accept Input From

Specifies whether the program accepts input from both a mouse and a digitizer or ignores mouse input when a digitizer is set.

Layout Regen Options

Specifies how the display list is updated in the Model and layout tabs. For each tab, the display list is updated either by regenerating the drawing when you switch to that tab or by saving the display list to memory and regenerating only the modified objects when you switch to that tab. Changing these settings can improve performance. (LAYOUTREGENCTL system variable)

Note The effectiveness of changing these settings depends on several factors. See LAYOUTREGENCTL for more information.

Regen When Switching Layouts

Regenerates the drawing each time you switch tabs.

Cache Model Tab and Last Layout

For the Model tab and the last layout made current, saves the display list to memory and suppresses regenerations when you switch between the two tabs. For all other layouts, regenerations still occur when you switch to those tabs.

Cache Model Tab and All Layouts

Regenerates the drawing the first time you switch to each tab. For the remainder of the drawing session, the display list is saved to memory and regenerations are suppressed when you switch to those tabs.

dbConnect Options

Controls options that relate to database connectivity.

Store Links Index in Drawing File

Stores the database index in the drawing file. Select this option to enhance performance during Link Select operations. Clear this option to decrease drawing file size and to enhance the opening process for drawings with database information.

Open Tables in Read-Only Mode

Specifies whether to open database tables in Read-only mode in the drawing file.

General Options

Controls general options that relate to system settings.

Single-Drawing Compatibility Mode

Limits the program to one open drawing at a time. If you clear this option, you can open multiple drawings at once. If multiple drawings are open, this option cannot be turned on until the additional drawings are closed. (SDI system variable)

Display OLE Text Size Dialog

Displays the OLE Text Size dialog box when OLE objects are inserted into drawings.

Show All Warning Messages

Displays all dialog boxes that include a Don't Display This Warning Again option. All dialog boxes with warning options are displayed, regardless of previous settings specific to each dialog box.

Beep on Error in User Input

Sounds an alarm beep when an invalid entry is detected.

Load acad.lsp with Every Drawing

Specifies whether the *acad.lsp* file is loaded into every drawing. If this option is cleared, only the *acaddoc.lsp* file is loaded into all drawing files. Clear this option if you do not want to run certain LISP routines in specific drawing files. (ACADLSPASDOC system variable)

Allow Long Symbol Names

Permits long names to be used for named objects in the drawing definition table. Object names can be up to 255 characters and include letters, numbers, blank spaces, and any special character not used by Windows and this program for other purposes. Long names can be used for layers, dimension styles, blocks, linetypes, text styles, layouts, UCS names,

views, and viewport configurations. This option is saved in the drawing. (EXTNAMES system variable)

Startup

Controls whether the Startup dialog box or no dialog box is displayed when starting this program or creating a new drawing.

Live Enabler Options

Specifies whether the program checks for object enablers. Using object enablers, you can display and use custom objects in drawings even when the ObjectARX application that created them is unavailable.

Check Web for Live Enablers

Checks the Autodesk website for object enablers.

Maximum Number of Unsuccessful Checks

Specifies the number of times to check for object enablers if unsuccessful.

• **User Preferences**

Controls options that optimize the way you work.

Windows Standard Behavior

Controls keystroke and right-click behavior.

Shortcut Menus in Drawing Area

Displays a shortcut menu in the drawing area when you right-click the pointing device. If this option is cleared, right-click is interpreted as ENTER.

Right-Click Customization

Displays the Right-Click Customization dialog box. This dialog box provides further definition for the Shortcut Menus in Drawing Area option. (SHORTCUTMENU system variable)

Insertion Scale

Controls the default scale for inserting blocks and drawings into a drawing.

Source Content Units

Sets the units used for an object being inserted into the current drawing when no insert units are specified with the INSUNITS system variable. If Unspecified-Unitless is selected, the object is not scaled when inserted. (INSUNITSDEFSOURCE system variable)

Target Drawing Units

Sets the units used in the current drawing when no insert units are specified with the INSUNITS system variable. (INSUNITSDEFTARGET system variable)

Fields

Sets preferences related to fields.

Display Background of Fields

Displays fields with a light gray background that is not plotted. When this option is cleared, fields are displayed with the same background as any text. (FIELDDISPLAY system variable)

Field Update Settings

Displays the Field Update Settings dialog box. (FIELDEVAL system variable)

Priority for Coordinate Data Entry

Controls how the program responds to input of coordinate data. (OSNAPCOORD system variable)

Running Object Snap

Specifies that running object snaps override coordinate entry at all times. You can also select this option by setting OSNAPCOORD to 0.

Keyboard Entry

Specifies that coordinate entry overrides running object snaps at all times. You can also select this option by setting OSNAPCOORD to 1.

Keyboard Entry Except Scripts

Specifies that coordinate entry overrides running object snaps, except in scripts. You can also select this option by setting OSNAPCOORD to 2.

Associative Dimensioning

Controls whether associative dimension objects or legacy-style, nonassociative dimension objects are created.

Make New Dimensions Associative

When selected, creates associative dimensions, which automatically adjust their locations, orientations, and measurement values when the geometric objects associated with them are modified. (DIMASSOC system variable)

Hyperlink

Controls settings that relate to the display properties of hyperlinks.

Display Hyperlink Cursor, Tooltip, and Shortcut Menu

Displays the hyperlink cursor and tooltip whenever the pointing device pauses over an object that contains a hyperlink. The shortcut menu provides additional hyperlink options when you select an object that contains a hyperlink and then right-click in the drawing area. If this option is cleared, hyperlinks in the drawing are ignored.

Undo/Redo

Controls Undo and Redo for Zoom and Pan.

Group Zoom and Pan Commands

When selected, groups multiple, consecutive zoom and pan commands as a single action for undo and redo operations.

Hidden Line Settings

Displays the Hidden Line Settings dialog box, where you can change the display properties of hidden lines.

Lineweight Settings

Displays the Lineweight Settings dialog box. Use this dialog box to set lineweight options, such as display properties and defaults, and also to set the current lineweight.

Edit Scale List

Displays the Edit Scale List dialog box. Use this dialog box to manage the list of scales displayed in several dialog boxes associated with layout viewports and plotting.

- ## Drafting

Specifies a number of general editing options.

AutoSnap Settings

Controls settings that relate to the visual aids, called AutoSnaps, that are displayed when you use object snaps. AutoSnap™ settings are saved in the registry. When the cursor, or aperture box, is on an object, you can press TAB to cycle through all the snap points available for that object.

Marker

Controls the display of the AutoSnap marker. The marker is a geometric symbol that is displayed when the crosshairs move over a snap point. (AUTOSNAP system variable)

Magnet

Turns the AutoSnap magnet on or off. The magnet is an automatic movement of the crosshairs that locks the crosshairs onto the nearest snap point. (AUTOSNAP system variable)

Display AutoSnap Tooltip

Controls the display of the AutoSnap tooltip. The tooltip is a label that describes which part of the object you are snapping to. (AUTOSNAP system variable)

Display AutoSnap Aperture Box

Controls the display of the AutoSnap aperture box. The aperture box is a box that appears inside the crosshairs when you snap to an object. (APBOX system variable)

AutoSnap Marker Color

Specifies the color of the AutoSnap marker.

AutoSnap Marker Size

Sets the display size for the AutoSnap marker.

Object Snap Options

Specifies options for object snaps. (OSNAP command)

Ignore Hatch Objects

Specifies that object snaps ignore hatch patterns when object snapping is turned on. (OSNAPHATCH system variable)

Replace Z Value with Current Elevation

Specifies that object snaps ignore the Z-value of the object snap location and use the Z-value of the elevation set for the current UCS.

AutoTrack Settings

Controls the settings that relate to AutoTrack™ behavior, which is available when polar tracking or object snap tracking is turned on (see DSETTINGS).

Display Polar Tracking Vector

Displays a vector along specified angles when polar tracking is on. With polar tracking, you can draw lines along angles. Polar angles are 90-degree divisors, such as 45, 30, and 15 degrees.

You can disable Display Polar Tracking Vector by setting TRACKPATH to 2.

Display Full-Screen Tracking Vector

Controls the display of tracking vectors. Tracking vectors are construction lines from which you can draw objects at specific angles or in specific relationships to other objects. If this option is selected, alignment vectors are displayed as infinite lines.

You can disable Display Full-Screen Tracking Vector by setting TRACKPATH to 1.

Display AutoTrack Tooltip

Controls the display of the AutoTrack tooltip. The tooltip is a label that displays the tracking coordinates. (AUTOSNAP system variable)

Alignment Point Acquisition

Controls the method of displaying alignment vectors in a drawing.

Automatic

Displays tracking vectors automatically when the aperture moves over an object snap.

Shift to Acquire

Displays tracking vectors when you press SHIFT and move the aperture over an object snap.

Aperture Size

Sets the display size for the AutoSnap aperture. When Display AutoSnap Aperture Box is selected (or when APBOX is set to 1), the aperture box is displayed in the center of the crosshairs when you snap to an object. The aperture size determines how close to a snap point you can be before the magnet locks the aperture box to the snap point. Values range from 1 to 50 pixels. (APERTURE system variable)

Drafting Tooltip Appearance

Controls the color, size, and transparency of drafting tooltips.

Settings

Displays the Tooltip Appearance dialog box.

• Selection

Sets options for selecting objects.

Pickbox Size

Controls the display size of the pickbox. The pickbox is the object selection tool that appears in editing commands. (PICKBOX system variable)

Selection Preview

Highlights objects as the pickbox cursor rolls over them. The PREVIEWEFFECT system variable controls the appearance of the highlighted objects.

When a Command Is Active

Displays selection previewing only when a command is active and the Select Objects prompt is displayed. (SELECTIONPREVIEW system variable)

When No Command Is Active

Displays selection previewing when no commands are active. (SELECTIONPREVIEW system variable)

Visual Effect Settings

Displays the Visual Effect Settings dialog box.

Selection Modes

Controls settings that relate to object selection methods.

Noun/Verb Selection

Allows you to select an object before starting a command. The command affects the previously selected object or objects. (PICKFIRST system variable)

You can use many editing and inquiry commands with noun/verb selection, including

ALIGN	DVIEW	PROPERTIES
ARRAY	ERASE	ROTATE
BLOCK	EXPLODE	SCALE
CHANGE	LIST	STRETCH
CHPROP	MIRROR	WBLOCK
COPY	MOVE	

Use SHIFT to Add to Selection

Adds an object to or removes it from the selection set when you press SHIFT and select an object. To clear a selection set quickly, draw a selection window in a blank area of the drawing. (PICKADD system variable)

Press and Drag

Draws a selection window when you select a point and dragging the pointing device to a second point. If this option is cleared, you can draw a selection window by selecting two separate points with the pointing device. (PICKDRAG system variable)

Implied Windowing

Initiates the drawing of a selection window when you select a point outside an object.

Drawing the selection window from left to right selects objects that are entirely inside the window's boundaries. Drawing from right to left selects objects within and crossing the window's boundaries. (PICKAUTO system variable)

Object Grouping

Selects all objects in a group when you select one object in that group. With GROUP you can create and name a set of objects for selection. You can also set this option by setting the PICKSTYLE system variable to 1.

Associative Hatch

Determines which objects are selected when you select an associative hatch. If this option is selected, boundary objects are also selected when you select an associative hatch. You can also set this option by setting the PICKSTYLE system variable to 2.

Grip Size

Controls the display size of grips. (GRIPSIZE system variable)

Grips

Controls the settings that relate to grips. Grips are small squares displayed on an object after it has been selected.

Unselected Grip Color

Determines the color of an unselected grip. If you choose Select Color from the color list, the Select Color dialog box is displayed. An unselected grip is displayed as a small, filled square. (GRIPCOLOR system variable)
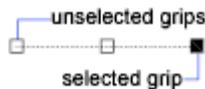
Selected Grip Color

Determines the color of a selected grip. If you choose Select Color from the color list, the Select Color dialog box is displayed. A selected grip is displayed as a small, filled square. (GRIPHOT system variable)

Hover Grip Color

Determines the color a grip displays when the cursor rolls over the grip. If you choose Select Color from the color list, the Select Color dialog box is displayed. (GRIPHOVER system variable)

Enable Grips

Displays grips on an object when you select it. You can edit an object with grips by selecting a grip and using the shortcut menu. Displaying grips in a drawing significantly affects performance. Clear this option to optimize performance. (GRIPS system variable)

Enable Grips within Blocks

Controls how grips are displayed on a block after you select it. If this option is selected, all grips are displayed for each object in the block. If this option is cleared, one grip is displayed at the insertion point of the block. You can edit an object with grips by selecting a grip and using the shortcut menu. (GRIPBLOCK system variable)

Enable Grips Within
Blocks off
(GRIPBLOCKS = 0)

Enable Grips Within
Blocks on
(GRIPBLOCKS = 1)

Enable Grip Tips

Displays grip-specific tips when the cursor hovers over a grip on a custom object that supports grip tips. This option has no effect on standard objects. (GRIPTIPS system variable)

Object Selection Limit for Display of Grips

Suppresses the display of grips when the initial selection set includes more than the specified number of objects. The valid range is 1 to 32,767. The default setting is 100. (GRIPOBJLIMIT system variable)

ORTHO

Constrains cursor movement to the horizontal or vertical direction

Status bar: Ortho

Command line: ortho (or **'ortho** for transparent use)

Enter mode [ON/OFF] <*current*>: *Enter* **on** *or* **off***, or press* ENTER

In the illustration, a line is drawn using Ortho mode. Point 1 is the first point specified, and point 2 is the position of the cursor when the second point is specified.

Ortho off   Ortho on
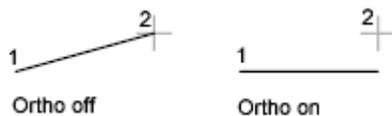
Ortho mode is used when you specify an angle or distance by means of two points using a pointing device. In Ortho mode, cursor movement is constrained to the horizontal or vertical direction (relative to the UCS) and the current grid rotation angle. Ortho mode is ignored in perspective views.
Horizontal is defined as being parallel to the X axis of the UCS and vertical as being parallel to the Y axis. ORTHO adjusts to the current snap rotation regardless of the UCS.

OSNAP

Sets running object snap modes

Object Snap toolbar:

Tools Menu: Drafting Settings

Shortcut menu: Press SHIFT while right-clicking in the drawing area and choose Osnap Settings.

Status bar: Osnap

Command line: osnap (or **'osnap** for transparent use)

The Object Snaps tab of the Drafting Settings dialog box is displayed.

If you enter **-osnap** at the Command prompt, OSNAP presents options on the command line.

Current osnap modes: *current*

Enter list of **object snap modes**: *Enter names of object snap modes separated with commas, or enter* **none** *or* **off**

PAGESETUP

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

File menu: Page Setup Manager

Shortcut menu: Right-click the Model tab or a layout tab and click Page Setup Manager.

Command line: pagesetup

The Page Setup Manager is displayed.

Specifies the page setup for the current layout or drawing sheet. You can also create named page setups, modify existing ones, or import page setups from other drawings.

Current Layout or Current Sheet Set

Lists the current layout to which the page setup will be applied. If the Page Setup Manager is opened from the Sheet Set Manager, displays the name of the current sheet set. If the Page Setup Manager is opened from a layout, displays the name of the current layout.

Layout icon

Displayed when the Page Setup Manager is opened from a layout.

Sheet set icon

Displayed when the Page Setup Manager is opened from the Sheet Set Manager.

Page Setups

Displays the current page setup, sets a different page setup as current, creates a new page setup, modifies an existing page setup, and imports page setups from other drawings.

Current Page Setup

Displays the page setup that is applied to the current layout. Displays "Not applicable" if the Page Setup Manager is opened from the Sheet Set Manager, because you cannot apply a page setup to an entire sheet set after it has been created.

Page Setup List

Lists the page setups that are available to apply to the current layout or when you publish a sheet set.

If the Page Setup Manager is opened from a layout, the current page setup is selected by default. The list includes the named page setups and layouts that are available in the drawing. Layouts that have a named page setup applied to them are enclosed in asterisks, with the named page setup in parentheses; for example, *Layout 1 (System Scale-to-fit)*. You can double-click a page setup in this list to set it as the current page setup for the current layout.

If the Page Setup Manager is opened from the Sheet Set Manager, only named page setups in the page setup overrides file (a drawing template [.*dwt*] file) that have Plot Area set to Layout or Extents are listed. The first page setup in the list is selected by default. Any of these page setups can be applied temporarily for a PUBLISH operation.

The shortcut menu also provides options for removing and renaming page setups.

Set Current

Sets the selected page setup as the current page setup for the current layout. You cannot set the current layout as the current page setup. Set Current is not available for sheet sets.

New

Displays the New Page Setup dialog box, in which you can enter a name for the new page setup and specify the page setup to use as a starting point.

Modify

Displays the Page Setup dialog box, in which you can edit the settings for the selected page setup.

Import

Displays the Select Page Setup From File dialog box (a standard file selection dialog box), in which you can select a drawing format (DWG), DWT, or drawing interchange format (DXF)™ file from which to import one or more page setups. If you select DWT as the file type, the Template folder opens automatically in the Select Page Setup From File dialog box. When you click OK, the Import Page Setups dialog box is displayed.

Selected Page Setup Details

Displays information about the selected page setup.

Device Name

Displays the name of the plot device specified in the currently selected page setup.

Plotter

Displays the type of plot device specified in the currently selected page setup.

Plot Size

Displays the plot size and orientation specified in the currently selected page setup.

Where

Displays the physical location of the output device specified in the currently selected page setup.

Description

Displays descriptive text about the output device specified in the currently selected page setup.

Display When Creating a New Layout

Specifies that the Page Setup dialog box is displayed when a new layout tab is selected or a new layout is created.
You can reset this function by turning on the Show Page Setup Dialog for New Layouts option on the Display tab of the Options dialog box.

PAN

Moves the view in the current viewport

 Standard toolbar:

 View menu: Pan ➤ Realtime

Shortcut menu: With no objects selected, right-click in the drawing area and choose Pan.

 Command line: pan (or **'pan** for transparent use)

The following prompt is displayed; you can pan the drawing display in real time.
Press ESC or ENTER to exit, or right-click to display a shortcut menu.

If you enter **-pan** at the Command prompt, PAN displays alternative prompts on the command line, and you can specify a displacement to pan the drawing display.

PARTIALOAD

Loads additional geometry into a partially opened drawing

 File Menu: Partial Load

 Command line: partiaload

The Partial Load dialog box is displayed. PARTIALOAD can be used only in a partially open drawing. To partially open a drawing, use OPEN and choose Partial Open in the Select File dialog box (a standard file selection dialog box). Any information that is loaded into the file using PARTIALOAD cannot be unloaded, not even with UNDO.

If you enter **-partiaload** at the Command prompt, PARTIALOAD displays prompts on the command line.

Displays the views and layers available for specifying additional geometry to load into a partially open drawing. You can select and load geometry from a view or a selected area and from layers. You cannot unload any information that is currently loaded in the drawing. If the drawing contains attached xrefs that were not loaded at the time that the drawing was partially opened, you can reload the xrefs by using the Xref Manager. See XREF.

View Geometry to Load

Displays the selected view and available views in the drawing. Views available for loading include only views defined in model space. You can load paper space geometry by loading the layer on which the paper space geometry is drawn.

View Name

Displays the currently selected view. Geometry that is common to both the selected view and the layers is loaded into the drawing.

View List

Displays all the model space views available in the selected drawing file. When a view is selected, only the geometry in the selected view is loaded. The default view is Extents. You can select to load geometry from only one view. Geometry that is common to both the selected view and the layers is loaded into the drawing. Any information currently loaded into the file cannot be unloaded.

Pick Window

Prompts you to use the pointing device to specify an area using window selection. The selected area becomes the view to load and is displayed in the View Geometry to Load list as New View.

Layer Geometry to Load

Displays all the layers available in the selected drawing file. The geometry on selected layers is loaded into the drawing, including both model space and paper space geometry. No layers are selected to load by default. You can load geometry from multiple layers. Geometry that is common to both the selected view and the layers is loaded into the drawing. Any information that is currently loaded into the file cannot be unloaded.

Layer Name

Displays the layer names in the selected drawing.

Load Geometry

Specifies whether geometry from a layer is loaded into the drawing or not. If Load Geometry is selected, the geometry from the layer is loaded into the drawing. Any information that is currently loaded into the file cannot be unloaded.

Load All

Selects Load Geometry for all layers. You can also right-click and choose Load All from the shortcut menu to load geometry from all layers.

Clear All

Specifies that no geometry be loaded into the drawing (clears the Load Geometry option for all layers). It is recommended that you load geometry from at least one layer into the drawing. If you do not select any layers to load, no geometry is loaded at all, including geometry from the selected view. A warning is issued if you do not select any layers to load into the drawing. You can also right-click and choose Clear All from the shortcut menu to specify that no geometry is loaded from any layer.

Index Status

Indicates whether the selected drawing file contains a spatial or layer index. INDEXCTL controls whether layer and spatial indexes are saved with the drawing file. A spatial index organizes objects based on their location in space. A layer index is a list showing which objects are on which layers.

- PARTIALOAD Command Line

Specify **first corner** or [**View**]: *Specify a point or enter* **v**

First Corner

Specifies the first corner of a user-defined view.
   Specify opposite corner: *Specify a point*

Geometry from the defined view is loaded into the partially open drawing.
   Enter layers to load or [?] <none>: *Enter a layer name, enter* **?**, *or press* ENTER

Layers to Load

Loads geometry from the selected layers into the current drawing, including both model space and paper space geometry. To load geometry from multiple layers into the drawing, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.
   Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER

Yes

   Does not load any external references into the drawing.

No

   Loads all external references into the drawing.

?—List Layers

Displays a list of layer names available in the current drawing. Enter one or more layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.
   Unload all Xrefs on open [Yes/No] <Yes>: *Enter* **y** *or* **n**, *or press* ENTER

Yes

   Does not load any external references into the drawing.

No

   Loads all external references into the drawing.

View

View

Loads the geometry from the specified view into the current drawing.

Enter view to load or [?] <*Extents*>: Enter a view name, enter ?, or press ENTERto load the Extents view

View to Load

Loads the geometry from the specified view into the partially open drawing.

Enter layers to load or [?] <none>: Enter a layer name, enter ?, or press ENTER

Layers to Load

Loads geometry from the layer or layers into the current drawing. To load geometry from multiple layers into the drawing, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: Enter y or n, or press ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

?—List Layers

Displays a list of layer names available in the current drawing. Enter one or more layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: Enter y or n, or press ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

?—List Views

Displays a list of model space views available in the selected drawing. Enter a view name.

Enter layers to load or [?] <none>: Enter a layer name, enter ?, or press ENTER

Layers to Load

Loads geometry from one or more layers into the current drawing. To load geometry from multiple layers into the drawing, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded nto the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: Enter y or n, or press ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

?—List Layers

A list of layer names available in the current drawing is displayed. Enter one or more layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

Unload all Xrefs on open [Yes/No] <Yes>: Enter y or n, or press ENTER

If you enter **yes**, no external references are loaded into the drawing. If you enter **no**, all external references are loaded into the drawing.

PARTIALOPEN

Loads geometry and named objects from a selected view or layer into a drawing

⌨ Command line: partialopen

Note It is recommended that you partially open a drawing by using OPEN and choosing Partial Open in the Select File dialog box to display the Partial Open dialog box.

When you are running a script and using PARTIALOPEN, FILEDIA can be set to 0 or 1. When FILEDIA is set to 0 and you enter **partialopen** or **-partialopen** at the Command prompt, PARTIALOPEN displays prompts on the command line.

   Enter name of drawing to open: *Enter a drawing name and press* ENTER

Enter **~** (tilde) at the prompt to ignore FILEDIA and display the Select File dialog box. In the Select File dialog box, select a file and choose Partial Open to display the Partial Open dialog box. If you enter a file name at the prompt, prompts are displayed on the command line.

   Enter **view to load** or [**?**] <*Extents*>: *Enter a view name, enter* **?***, or press* ENTER *to load the Extents view*

?—List Views

Displays a list of model space views available in the selected drawing. Enter a view name.

   Enter layers to load or [?] <none>: *Enter a layer name, enter* **?***, or press* ENTER

Layers to Load

Loads geometry from one or more layers into the current drawing. To load geometry from multiple layers, use a comma between layer names. If no layer geometry is specified, no layer geometry is loaded into the drawing, including geometry specified to load from a view.

   Unload all Xrefs on open [Yes/No] <N>: *Enter* **y** *or* **n***, or press* ENTER

Yes

   Does not load any external references into the drawing.

No

   Loads all external references into the drawing.

?—List Layers

Displays a list of layers available in the selected drawing. Enter one or more layer names. To enter multiple layers, separate the layer names with a comma.

   Unload all Xrefs on open [Yes/No] <N>: *Enter* **y** *or* **n***, or press* ENTER

Yes

   Does not load any external references into the drawing.

No

   Loads all external references into the drawing.

PASTEBLOCK

Pastes copied objects as a block

Edit menu: Paste as Block

Shortcut menu: End any active commands, right-click in the drawing area, and choose Paste as Block.

Command line: **pasteblock**

Specify insertion point: *Specify a point*

The objects copied to the Clipboard are pasted into the drawing as a block at the specified insertion point.

PASTECLIP

Inserts data from the Clipboard

Standard toolbar:

Edit menu: Paste

Shortcut menu: End any active commands, right-click in the drawing area, and choose Paste.

Command line: **pasteclip**

Specify insertion point:

If the Clipboard contains ASCII text, the text is inserted as a multiline text (mtext) object using the MTEXT defaults. A spreadsheet is inserted as a table object.
All other objects, except AutoCAD objects, are inserted as embedded or linked (OLE) objects. You can edit these OLE objects by double-clicking them in the drawing to open the application in which they were created.

Note You can also use CTRL+V to run PASTECLIP. If the cursor is in the drawing area, PASTECLIP behaves as described. If the cursor is on the command line, text from the Clipboard is pasted at the current prompt.

PASTEORIG

Pastes a copied object in a new drawing using the coordinates from the original drawing

Edit menu: Paste to Original Coordinates

Shortcut menu: End any active commands, right-click in the drawing area, and choose Paste to Original Coordinates.

Command line: **pasteorig**

The object copied to the Clipboard is pasted into the drawing at the same coordinates used in the original drawing.
PASTEORIG functions only when the Clipboard contains AutoCAD data from a drawing other than the current drawing.

PASTESPEC

Inserts data from the Clipboard and controls the format of the data

Edit menu: Paste Special

Command line: pastespec

The Paste Special dialog box is displayed.

Sets file formats and linking options for pasted files.

Source

Displays the name of the document that contains the information you copied. It may also show the specific section of the document you copied.

Paste

Pastes the contents of the Clipboard into the current drawing as an embedded object.

Paste Link

Pastes the contents of the Clipboard into the current drawing. If the source application supports an OLE link, a link is created to the original file.

As

Displays applicable formats in which you can paste the contents of the Clipboard into the current drawing.

If you select AutoCAD Entities, the metafile graphics in the Clipboard are converted to AutoCAD objects. If the metafile graphics are not converted, the metafile is displayed as an OLE object.

Display as Icon

Inserts a picture of the application icon instead of the data. To view or edit the data, double-click the icon.

PCINWIZARD

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

Tools menu: Wizards ➤ Import Plot Settings

Command line: pcinwizard

PCINWIZARD displays the Import PCP or PC2 Plot Settings wizard. Information that can be imported from PCP or PC2 files includes plot area, rotation, plot offset, plot optimization, plot to file, paper size, plot scale, and pen mapping.
The wizard prompts you for the name of the PCP or PC2 configuration file from which you want to import settings. You can view and modify the plot settings prior to importing them. The imported settings can be applied to the current Model tab or layout tab.

PEDIT

Edits polylines and three-dimensional polygon meshes

Modify II toolbar:

Modify menu: Object ➤ Polyline

Shortcut menu: Select a polyline to edit, right-click in the drawing area, and choose Polyline Edit.

Command line: pedit

Select polyline or [**Multiple**]: *Use an object selection method or enter* **m**

The remaining prompts depend on whether you have selected a **2D polyline**, a **3D polyline**, or **3D polygon mesh**.

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.

Multiple

Multiple

Enables selection for more than one object.

2D Polyline Selection

2D Polyline Selection

If you select a 2D polyline, the following prompt is displayed:

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: *Enter an option or press* ENTER*to end the command*

If the polyline you select is a closed polyline, Open replaces the Close option in the prompt. You can edit a 2D polyline if its normal is parallel to and in the same direction as the *Z* axis of the current UCS.

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.

before Close   after Close

Open

Removes the closing segment of the polyline. The polyline is considered closed unless you open it using the Open 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 objects to join the polyline, their endpoints must touch unless you use the Multiple option at the first PEDIT prompt. In this case, you can join polylines that do not touch if the fuzz distance is set to a value large enough to include the endpoints.

Select objects: *Use an object selection method*

If you previously selected multiple objects using the Multiple option, the following prompt is displayed:

Enter fuzz distance or [Jointype]<0.0000>: *Enter a distance or* **j**

Jointype

Sets the method of joining selected polylines.

Enter a vertex editing option

Enter join type [Extend/Add/Both}<Extend>: *Enter* **e**, **a**, *or* **b**

Extend

Joins the selected polylines by extending or trimming the segments to the nearest endpoints.

Add

Joins the selected polylines by adding a straight segment between the nearest endpoints.

Both

Joins the selected polylines by extending or trimming if possible. Otherwise joins the selected polylines by adding a straight segment between the nearest endpoints.

selected open polyline other objects selected polyline and objects joined
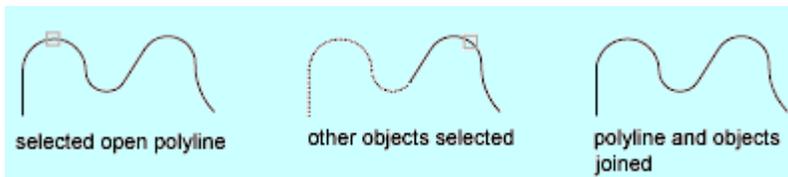
Width

Specifies a new uniform width for the entire polyline.
   Specify new width for all segments:

You can use the **Width** option of the Edit Vertex option to change the starting and ending widths of a segment.

varying width uniform width

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. The following prompt is displayed:
   [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <*current*>: *Enter an option or press* ENTER

Pressing ENTER accepts the current default, which is either Next or Previous.

beginning vertex

Next

Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline even if the polyline is closed.

Previous

Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline even if the polyline is closed.

Break

Saves the location of the marked vertex while you move the X marker to any other vertex.
   Enter an option [Next/Previous/Go/eXit] <*current*>: *Enter an option or press* ENTER

If one of the specified vertices is at an end of the polyline, the result is one truncated polyline. If both specified vertices are at endpoints of the polyline, or if just one vertex is specified and it is at an endpoint, you cannot use Break.

Next

   Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline, even if the polyline is closed.

Previous

   Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline, even if the polyline is closed.

saved marked vertex

next marked vertex

Go
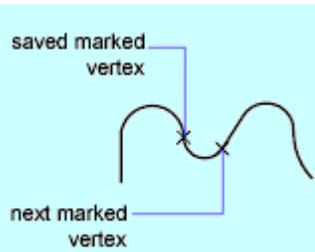
Deletes any segments and vertices between the two vertices you specify and returns to Edit Vertex mode.

segment deleted

Exit

Exits Break and returns to Edit Vertex mode.

Insert

Adds a new vertex to the polyline after the marked vertex.
Specify location for new vertex: *Specify a point (1)*

marked vertex

before Insert

after Insert

Move

Moves the marked vertex.
Specify new location for marked vertex: *Specify a point (1)*

marked vertex

before Move

after Move

Regen

Regenerates the polyline.

before Regen

after Regen

Straighten

Saves the location of the marked vertex while you move the X marker to any other vertex.
If you want to remove an arc segment that connects two straight segments of a polyline and then extend the straight segments until they intersect, use the FILLET command with a fillet radius of 0.

   Enter an option [Next/Previous/Go/eXit] <*current*>: *Enter an option or press* ENTER
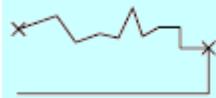
before Straighten   after Straighten

Next

  Moves the X marker to the next vertex.

Previous

  Moves the X marker to the previous vertex.

Go

  Deletes any segments and vertices between the two vertices you select, replaces them with single straight line segments, and returns to Edit Vertex mode. If you specify only one vertex by entering **go** without moving the X marker, the segment following that vertex is straightened if it is an arc.

Exit

  Exits Straighten and returns to Edit Vertex mode.

Tangent

Attaches a tangent direction to the marked vertex for use later in curve fitting. The following prompt is displayed:

   Specify direction of vertex tangent: *Specify a point or enter an angle*

Width

Changes the starting and ending widths for the segment that immediately follows the marked vertex.

   Specify starting width for next segment <*current*>: *Specify a point, enter a value, or press* ENTER

   Specify ending width for next segment <*starting width*>: *Specify a point, enter a value, or press* ENTER

You must regenerate the polyline to display the new width.

marked vertex   segment width changed

Exit

Exits Edit Vertex mode.

Fit

Creates an arc-fit polyline, 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.

original

Fit curve

Spline

Uses the vertices of the selected polyline as the control points, or frame, of a curve approximating a B-spline. This curve, called a spline-fit polyline, passes through the first and last control points unless the original polyline was closed. The curve is pulled toward the other points but does not necessarily pass through them. The more control points you specify in a particular part of the frame, the more pull they exert on the curve. Quadratic and cubic spline-fit polylines can be generated.

before Spline

after Spline

Spline-fit polylines are very different from the curves produced by the Fit option. Fit constructs pairs of arcs that pass through every control point. Both of these curves are different from true B-splines produced with the SPLINE command.
If the original polyline included arc segments, they are straightened when the spline's frame is formed. If the frame has width, the resulting spline tapers smoothly from the width of the first vertex to the width of the last vertex. All intermediate width information is ignored. Once spline-fit, the frame, if displayed, is shown with zero width and CONTINUOUS linetype. Tangent specifications on control point vertices have no effect on spline-fitting.
When a spline-fit curve is fit to a polyline, the spline-fit curve's frame is stored so that it can be recalled by a subsequent decurving. You can turn a spline-fit curve back into its frame polyline by using the PEDIT Decurve option. This option works on fit curves in the same manner as it does on splines.
Spline frames are not usually displayed on the screen. If you want to see them, set the SPLFRAME system variable to 1. Next time the drawing is regenerated, both the frame and the spline curve are drawn.

SPLFRAME set to 0

SPLFRAME set to 1

Most editing commands act the same when applied to spline-fit polylines or fit curves.

- MOVE, ERASE, COPY, MIRROR, ROTATE, and SCALE operate on both the spline curve and its frame, whether the frame is visible or not.
- EXTEND changes the frame by adding a new vertex where the initial or final line of the frame intersects the boundary geometry.
- BREAK and TRIM generate a polyline with only the fit spline, which is consistent with fit curves, where the curve fitting is permanent.
- EXPLODE deletes the frame and generates lines and arcs to approximate the spline-fit polyline.
- OFFSET generates a polyline with only the fit spline, which is consistent with its behavior with fit curves.
- DIVIDE, MEASURE, and the Object option of AREA and HATCH see only the fit spline, not the frame.

- STRETCH refits the spline to the stretched frame after a spline is stretched.

The Join option of PEDIT decurves the spline and discards the spline information of the original and any added polylines. Once the Join operation is complete, you can fit a new spline to the resulting polyline.
The Edit Vertex options of PEDIT have the following effect:

- The Next and Previous options move the X marker only to points on the frame of the spline, whether visible or not.
- The Break option discards the spline.
- The Insert, Move, Straighten, and Width options automatically refit the spline.
- The Tangent option has no effect on splines.

Object snap uses only the spline-fit curve itself, not the frame. If you want to snap to the frame control points, use PEDIT to recall the polyline frame first.
The SPLINETYPE system variable controls the type of spline curve approximated. Setting SPLINETYPE to 5 approximates a quadratic B-spline. Setting SPLINETYPE to 6 approximates a cubic B-spline.

quadratic B-spline   cubic B-spline

You can examine or change the fineness or coarseness of the spline approximation with the SPLINESEGS system variable, or you can use AutoLISP®. The default value is 8. If you set the value higher, a greater number of line segments are drawn and the approximation to the ideal spline becomes more precise. The generated spline occupies more space in the drawing file and takes longer to generate.
If you set SPLINESEGS to a negative value, the program generates segments using the absolute value of the setting and then applies a fit-type curve to those segments. Fit-type curves use arcs as the approximating segments. Using arcs yields a smoother generated curve when few segments are specified, but the curve can take longer to generate.

SPLINESEGS set to 1   SPLINESEGS set to 10

To change the number of segments used to fit an existing spline, change SPLINESEGS and respline the curve. You do not have to decurve it first.

Decurve

Removes extra vertices inserted by a fit or spline curve and straightens all segments of the polyline. Retains tangent information assigned to the polyline vertices for use in subsequent fit curve requests. If you edit a spline-fit polyline with a command such as BREAK or TRIM, you cannot use the Decurve option.

Ltype Gen

Generates the linetype in a continuous pattern through the vertices of the polyline. When turned off, this option generates the linetype starting and ending with a dash at each vertex. Ltype Gen does not apply to polylines with tapered segments.
Enter polyline linetype generation option [ON/OFF] <*current*>: *Enter* **on** *or* **off***, or press ENTER*

Ltype Gen set to Off   Ltype Gen set to On
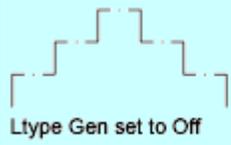
Undo

Reverses operations as far back as the beginning of the PEDIT session.

3D Polyline Selection

3D Polyline Selection

If you select a 3D polyline, the following prompt is displayed:

Enter an option [Close/Edit vertex/Spline curve/Decurve/Undo]: *Enter an option or press* ENTER

If the polyline you select is closed, Open replaces the Close option in the prompt.

Close

Creates the closing segment of the polyline, connecting the last segment with the first. The polyline is considered open unless you close it with Close.

Open

Removes the closing segment of the polyline. The polyline is considered closed unless you open it with Open.

Edit Vertex

Performs various editing tasks on one vertex of the polyline and segments that follow it.

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/eXit *<current>*: *Enter an option or press*ENTER

Pressing ENTER accepts the current default, which is either Next or Previous.

Next

Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline, even if the polyline is closed.

Previous

Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline, even if the polyline is closed.

Break

Saves the location of the marked vertex while you move the X marker to any other vertex.

Enter an option [Next/Previous/Go/eXit] *<current>*: *Enter an option or press* ENTER

Next

Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the polyline, even if the polyline is closed.

Previous

Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the polyline, even if the polyline is closed.

Go

Deletes any segments and vertices between the two vertices you specify and returns to Edit Vertex mode.

Exit

Exits Break and returns to Edit Vertex mode.

If one of the specified vertices is at an end of the polyline, the polyline is truncated. If both specified vertices are at endpoints of the polyline, or if just one vertex is specified and it is at an endpoint, you cannot use Break mode.

Insert

Adds a new vertex to the polyline after the marked vertex.
    Specify location for new vertex: *Specify a point*

Move

Moves the marked vertex.
    Specify new location for marked vertex: *Specify a point*

Regen

Regenerates the polyline.

Straighten

Saves the location of the marked vertex while you move the X marker to any other vertex.
    Enter an option [Next/Previous/Go/eXit] <*current*>: *Enter an option or press* ENTER

Next

Moves the X marker to the next vertex.

Previous

Moves the X marker to the previous vertex.

Go

Deletes any segments and vertices between the two vertices you select, replaces them with single straight line segments, and returns to Edit Vertex mode. If you specify only one vertex by entering **go** without moving the X marker, the segment following that vertex is made straight if it is an arc.

Exit

Exits Straighten and returns to Edit Vertex mode.

To remove an arc segment that connects two straight segments of a polyline and then to extend the straight segments until they intersect, use the FILLET command with a fillet radius of 0.

Exit

Exits Edit Vertex mode.

Spline Curve

Fits a 3D B-spline curve to its control points. The SPLFRAME system variable controls the accuracy and display of the control points for the 3D B-spline, whose curves can be approximated only by line segments. Negative values for spline segments are ignored.
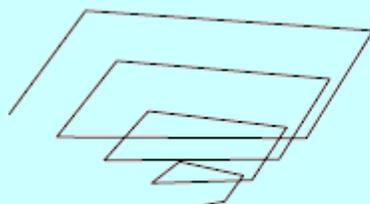
original 3D polyline   3D polyline after curve-fitting

Decurve

Removes extra vertices inserted by a fit or spline curve and straightens all segments of the polyline. Retains tangent information assigned to the polyline vertices for use in subsequent fit curve requests. If you edit a spline-fit polyline with a command such as BREAK or TRIM, you cannot use the Decurve option.Undo Reverses operations as far back as the beginning of the PEDIT session.

3D Polygon Mesh Selection

3D Polygon Mesh Selection

If you select a polygon mesh, the following prompt is displayed:

Enter an option [Edit vertex/Smooth surface/Desmooth/Mclose/Nclose/Undo]: *Enter an option or press* ENTER*to end the command*

Mclose and Nclose are replaced by Mopen and Nopen if the polygon mesh is currently closed in the *M* or *N* direction.

Edit Vertex

Edits individual vertices of a polygon mesh that can be seen as a rectangular *M* by *N* array, where *M* and *N* are the dimensions specified in 3DMESH. The SURFTAB1 and SURFTAB2 system variables store *M* and *N* values for RULESURF, TABSURF, REVSURF, and EDGESURF.

Enter an option [Next/Previous/Left/Right/Up/Down/Move/REgen/eXit *<current>*: *Enter an option or press* ENTER

Pressing ENTER accepts the current default, which is either Next or Previous.
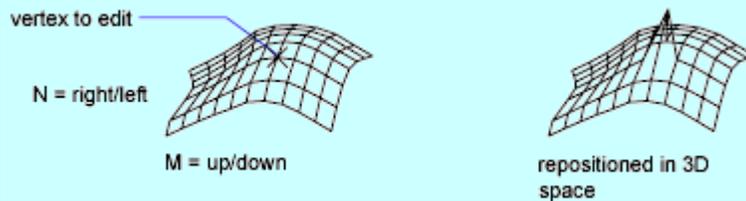
Next

Moves the X marker to the next vertex. The marker does not wrap around from the end to the start of the mesh, even if the mesh is closed.

Previous

Moves the X marker to the previous vertex. The marker does not wrap around from the start to the end of the mesh, even if the mesh is closed.

Left

Moves the X marker to the previous vertex in the *N* direction. The marker does not wrap around from the start to the end of the mesh, even if the mesh is closed.

Right

Moves the X marker to the next vertex in the *N* direction. The marker does not wrap around from the end to the start of the mesh, even if the mesh is closed.

Up

Moves the X marker to the next vertex in the *M* direction. The marker does not wrap around from the end to the start of the mesh, even if the mesh is closed.

Down

Moves the X marker to the previous vertex in the *M* direction. The marker does not wrap around from the start to the end of the mesh, even if the mesh is closed.

Move

Repositions the vertex and moves the editing mark.

Specify new location for marked vertex: *Specify a point*

Regen

Regenerates the polygon mesh.

Exit

Exits Edit Vertex mode.

Smooth Surface

Fits a smooth surface. The SURFTYPE system variable controls the type of surface this option fits. The types of surfaces include quadratic B-spline, cubic B-spline, and Bezier.
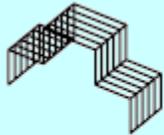
original polygon quadratic B-spline cubic B-spline Bezier
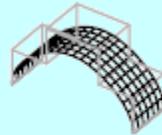
Desmooth

Restores the original control-point polygon mesh.

Mclose

Closes the *M*-direction polylines if the polygon mesh is open in the *M* direction.
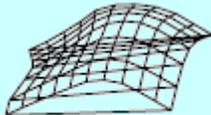
closed M direction

Mopen

Opens the *M*-direction polylines if the polygon mesh is closed in the *M* direction.

Nclose

Closes the *N*-direction polylines if the polygon mesh is open in the *N* direction.
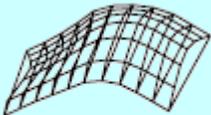
closed N direction

Nopen

Opens the *N*-direction polylines if the polygon mesh is closed in the *N* direction.

Undo

Reverses operations as far back as the beginning of the PEDIT session.

PLINE

Draw toolbar:

Draw menu: Polyline

Command line: pline

> Specify start point: *Specify a point (1)*
> Current line-width is *<current>*
> Specify **next point** or [**Arc**/**Close**/**Halfwidth**/**Length**/**Undo**/**Width**]: *Specify a point (2) or enter an option*

The PLINEGEN system variable controls the linetype pattern display around and the smoothness of the vertices of a 2D polyline. Setting PLINEGEN to 1 generates new polylines in a continuous pattern around the vertices of the completed polyline. Setting PLINEGEN to 0 starts and ends the polyline with a dash at each vertex. PLINEGEN does not apply to polylines with tapered segments.
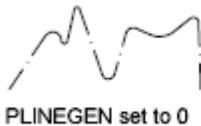
PLINEGEN set to 0   PLINEGEN set to 1

PLOT

Standard toolbar:

File menu: Plot
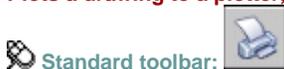
Shortcut menu: Right-click the Model tab or a layout tab and click Plot.

Command line: plot

The Plot dialog box is displayed. Click OK to begin plotting with the current settings.

If you enter **-plot** at the Command prompt, PLOT displays prompts on the command line.

Specifies device and media settings, and plots your drawing. The title of the Plot dialog box indicates the name of the current layout.

You can display more options in the Plot dialog box by clicking the More Options button.

Page Setup

Displays a list of any named and saved page setups in the drawing. You can base the current page setup on a named page setup saved in the drawing, or you can create a new named page setup based on the current settings in the Plot dialog box by clicking Add.

Name

Displays the name of the current page setup.

Add

Displays the Add Page Setup dialog box, in which you can save the current settings in the Plot dialog box to a named page setup. You can modify this page setup through the Page Setup Manager.

Printer/Plotter

Specifies a configured plotting device to use when plotting layouts.
If the selected plotter doesn't support the layout's selected paper size, a warning is displayed and you can select the plotter's default paper size or a custom paper size.

Name

Lists the available PC3 files or system printers from which you can select to plot the current layout. An icon in front of the device name identifies it as a PC3 file or a system printer.

- *PC3 file icon:* Indicates a PC3 file.
- *System printer icon:* Indicates a system printer.

Properties

Displays the Plotter Configuration Editor (PC3 editor), in which you can view or modify the current plotter configuration, ports, device, and media settings.

If you make changes to the PC3 file using the Plotter Configuration Editor, the Changes to a Printer Configuration File dialog box is displayed.

Plotter

Displays the plot device specified in the currently selected page setup.

Where

Displays the physical location of the output device specified in the currently selected page setup.

Description

Displays descriptive text about the output device specified in the currently selected page setup. You can edit this text in the Plotter Configuration Editor.

Plot to File

Plots output to a file rather than to a plotter or printer. The default location for plot files is specified in the Options dialog box, Plot and Publish tab, under Default Location for Plot-to-File Operations.

If the Plot to File option is turned on, when you click OK in the Plot dialog box, the Plot to File dialog box (a standard file navigation dialog box) is displayed.

Partial Preview

Shows an accurate representation of the effective plot area relative to the paper size and printable area. The tooltip displays the paper size and printable area.
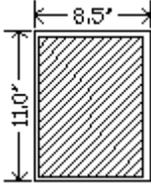
Paper Size

Displays standard paper sizes that are available for the selected plotting device. If no plotter is selected, the full standard paper size list is displayed and available for selection.

If the selected plotter doesn't support the layout's selected paper size, a warning is displayed, and you can select the plotter's default paper size or a custom paper size.

A default paper size is set for the plotting device when you create a PC3 file with the Add-a-Plotter wizard. The paper size that you select in the Page Setup dialog box is saved with the layout and overrides the PC3 file settings.

The actual printable area of the page, which is determined by the selected plotting device and paper size, is indicated in the layout by a dashed line.

If you are plotting a raster image, such as a BMP or TIFF file, the size of the plot is specified in pixels, not in inches or millimeters.

Number of Copies

Specifies the number of copies to plot. This option is not available when you plot to file.

Plot Area

Specifies the portion of the drawing to be plotted. Under What to Plot, you can select an area of the drawing to be plotted.

Layout/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 from the Model tab, plots the entire drawing area that is 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 may be regenerated to recalculate the extents before plotting.

Display

Plots the view in the current viewport in the selected Model tab or the current paper space view in the layout.

View

Plots a view that was previously saved with the VIEW command. You can select a named view from the list. If there are no saved views in the drawing, this option is unavailable.

When the View option is selected, a View list is displayed that lists the named views that are saved in the current drawing. You can select a view from this list to plot.

Window

Plots any portion of the drawing that you specify. When you select Window, the Window button becomes available. Click the Window button to use the pointing device to specify the two corners of the area to be plotted, or enter coordinate values.

Specify first corner: *Specify a point*
Specify other corner: *Specify a point*

Plot Offset

Specifies an offset of the plot area relative to the lower-left corner of the printable area or to 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.

The printable area of a drawing sheet is defined by the selected output device and is represented by a dashed line in a layout. When you change to another output device, the printable area may change.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y offset boxes. The plotter unit values are in inches or millimeters on the paper.

Center the Plot

Automatically calculates the X and Y offset values to center the plot on the paper. This option is not available when Plot Area is set to Layout.

X

Specifies the plot origin in the X direction relative to the setting of the Plot Offset Definition option.

Y

Specifies the plot origin in the Y direction relative to the setting of the Plot Offset Definition option.

Plot Scale

Controls the relative size of drawing units to plotted units. The default scale setting is 1:1 when plotting a layout. The default setting is Fit to Paper when plotting from the Model tab.

Fit to Paper

Scales the plot to fit within the selected paper size and displays the custom scale factor in the Scale, Inch =, and Units boxes.

Scale

Defines the exact scale for the plot. *Custom* defines a user-defined scale. You can create a custom scale by entering the number of inches (or millimeters) equal to the number of drawing units.

Note You can modify the list of scales with SCALELISTEDIT.

Inch(es) =/mm =/Pixel(s) =

Specifies the number of inches, millimeters, or pixels equal to the specified number of units.

Inch/mm/pixel

Specifies inches or mm for display of units in the Plot dialog box. The default is based on the paper size and changes each time a new paper size is selected. Pixel is available only when a raster output is selected.

Units

Specifies the number of units equal to the specified number of inches, millimeters, or pixels.

Scale Lineweights

Scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of plotted objects and are plotted with the linewidth size regardless of the plot scale.

Preview

Displays the drawing as it will appear when plotted on paper by executing the PREVIEW command. To exit the print preview and return to the Plot dialog box, press ESC, press ENTER, or right-click and then click Exit on the shortcut menu.

Apply to Layout

Saves the current Plot dialog box settings to the current layout.

More Options

Controls display of additional options in the Plot dialog box.

- Plot Style Table (Pen Assignments)
- Shaded Viewport Options
- Plot Options
- Drawing Orientation

Plot Style Table (Pen Assignments)

Sets the plot style table, edits the plot style table, or creates a new plot style table.

Name (Unlabeled)

Displays the plot style table that is assigned to the current Model tab or layout tab and provides a list of the currently available plot style tables.

If you select New, the Add Plot Style Table wizard is displayed, which you can use to create a new plot style table. The wizard that is displayed is determined by whether the current drawing is in color-dependent or named mode.

Edit

Displays the Plot Style Table Editor, in which you can view or modify plot styles for the currently assigned plot style table.

Shaded Viewport Options

Specifies how shaded and rendered viewports are plotted and determines their resolution level and dots per inch (dpi).

Shade Plot

Specifies how views are plotted. To specify this setting for a viewport on a layout tab, select the viewport and then, on the Tools menu, click Properties.

From the Model tab, you can select from the following options:

- *As Displayed:* Plots objects the way they are displayed on the screen.

- *Wireframe:* Plots objects in wireframe regardless of the way they are displayed on the screen.
- *Hidden:* Plots objects with hidden lines removed regardless of the way the objects are displayed on the screen.
- *Rendered:* Plots objects as rendered regardless of the way they are displayed on the screen.

Quality

Specifies the resolution at which shaded and rendered viewports are plotted.

You can select from the following options:

- *Draft*: Sets rendered and shaded model space views to be plotted as wireframe.
- *Preview*: Sets rendered and shaded model space views to be plotted at one quarter of the current device resolution, to a maximum of 150 dpi.
- *Normal*: Sets rendered and shaded model space views to be plotted at one half of the current device resolution, to a maximum of 300 dpi.
- *Presentation*: Sets rendered and shaded model space views to be plotted at the current device resolution, to a maximum of 600 dpi.
- *Maximum*: Sets rendered and shaded model space views to be plotted at the current device resolution with no maximum.
- *Custom*: Sets rendered and shaded model space views to be plotted at the resolution setting that you specify in the DPI box, up to the current device resolution.

DPI

Specifies the dots per inch for shaded and rendered views, up to the maximum resolution of the current plotting device. This option is available if you select Custom in the Quality box.

Plot Options

Specifies options for lineweights, plot styles, shaded plots, and the order in which objects are plotted.

Plot in Background

Specifies that the plot is processed in the background. (BACKGROUNDPLOT system variable)

Plot Object Lineweights

Specifies whether lineweights assigned to objects and layers are plotted.

Plot with Plot Styles

Specifies whether plot styles applied to objects and layers are plotted. When you select this option, Plot Object Lineweights is automatically selected also.

Plot Paperspace Last

Plots model space geometry first. Paper space geometry is usually plotted before model space geometry.

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. 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, in which you can specify the information that 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 the Plot Stamp On option, and then click the Plot Stamp Settings button that is displayed to the right of the option.

You can also open the Plot Stamp dialog box by clicking the Plot Stamp Settings button on the Plot and Publish tab of the Options dialog box.

Plot Stamp Settings button

Displays the Plot Stamp dialog box when the Plot Stamp On option is selected in the Plot dialog box.

Save Changes to Layout

Saves changes that you make in the Plot dialog box to the layout.

Drawing Orientation

Specifies the orientation of the drawing on the paper for plotters that support landscape or portrait orientation. The paper icon represents the media orientation of the selected paper. The letter icon represents the orientation of the drawing on the page.

Portrait

Orients and plots the drawing so that the short edge of the paper represents the top of the page.

Landscape

Orients and plots the drawing so that the long edge of the paper represents the top of the page.

Plot Upside-Down

Orients and plots the drawing upside down.

Icon

Indicates the media orientation of the selected paper and represents the orientation of the drawing on the page as a letter on the paper.

Note Drawing orientation is also affected by the PLOTROTMODE system variable.

Less Options

Hides the following options in the Plot dialog box:

- Plot Style Table (Pen Assignments)
- Shaded Viewport Options
- Plot Options
- Drawing Orientation

PLOTSTAMP

⌨ **Command line: plotstamp**

Specifies the information for the plot stamp.

Plot Stamp Fields

Specifies the drawing information you want applied to the plot stamp. The selected fields are separated by commas and a space.

Drawing Name

Includes the drawing name and path in the plot stamp information.

Layout Name

Includes the name of the layout in the plot stamp information.

Date and Time

Includes the date and time in the plot stamp information.

Note The date and time format is determined by the Regional Settings dialog box in the Windows Control Panel. Plot stamping uses the short date style for dates.

Login Name

Includes the Windows login name in the plot stamp information.

The Windows login name is contained in the LOGINNAME system variable.

Device Name

Includes the current plotting device name in the plot stamp information.

Paper Size

Includes the paper size for the currently configured plotting device in the plot stamp information.

Plot Scale

Includes the plot scale in the plot stamp information.

Preview

Provides a visual display of the plot stamp location based on the location and rotation values you have specified in the Advanced Options dialog box. You cannot preview the plot stamp any other way. This is not a preview of the plot stamp contents.

User Defined Fields

Provides text that can optionally be plotted, logged, or both plotted and logged at plot time. The selected value in each user-defined list will be plotted. For example, you might populate one list with media types or prices and the other with job names. If the user-defined value is set to <none>, then no user-defined information is plotted.

Add/Edit

Displays the User Defined Fields dialog box, where you can add, edit, or delete user-defined fields.

Plot Stamp Parameter File

Stores plot stamp information in a file with a *.pss* extension. Multiple users can access the same file and stamp their plots based on company standard settings.

Two PSS files are provided, *Mm.pss* and *Inches.pss*, which are located in the *Support* folder. The initial default plot stamp parameter file name is determined by the regional settings of the operating system when the program is installed.

Path

Specifies the location of the plot stamp parameter file.

Load

Displays the Plotstamp Parameter File Name dialog box (a standard file selection dialog box) in which you can specify the location of the parameter file you want to use.

Save As

Saves the current plot stamp settings in a new parameter file.

Advanced

Displays the Advanced Options dialog box, in which you can set the location, text properties, and units of the plot stamp. You can also create a log file, and set its location.

PLOTSTYLE

the current plot style for new objects or assigns a plot style to selected objects

Command line: plotstyle

To use the PLOTSTYLE command, the drawing must be configured to use named plot styles.

- If a drawing is in Color-Dependent mode, you can convert it to use named plot styles using the CONVERTPSTYLES command.
- To configure a new drawing to use named plot styles, set PSTYLEPOLICY to 0 before creating the new drawing.

If no selection set exists, the Current Plot Style dialog box is displayed. If a selection set exists, the Select Plot Style dialog box is displayed

If you enter **-plotstyle** at the Command prompt, PLOTSTYLE displays a prompt on the command line.

Specifies the plot style for new objects.

Current Plot Style

Displays the current plot style.

Plot Style List

Displays the available plot styles that can be assigned to an object, including the default plot style, NORMAL. A plot style is a collection of overrides for color, dithering, gray scale, pen assignments, screening, linetype, lineweight, end styles, join styles, and fill styles.

Active Plot Style Table

Sets the plot style table attached to the current layout. A plot style table is a collection of plot style definitions.

Editor

Displays the Plot Style Table Editor, in which you modify the plot styles in the plot style table.

Attached To

Displays the tab (Model tab or layout tab) that the plot style table is attached to.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration

📁 **File menu:** Plotter Manager

⌨ **Command line:** plottermanager

The Plotter Manager is displayed, where you can perform the following tasks:
- Double-click the Add-a-Plotter wizard to add and configure plotters and printers.
- Double-click a plotter configuration (PC3 file) to start the Plotter Configuration Editor.

Modifies a PC3 file's plotter port connections and output settings, including media, graphics, physical pen configuration, custom properties, initialization strings, calibration, and user-defined paper sizes. You can drag these configuration options from one PC3 file to another.

You can start the Plotter Configuration Editor with any of the following methods:
- Double-click a PC3 file in Microsoft® Windows® Explorer or right-click the PC3 file and click Open. (By default, PC3 files are stored in the *Plotters* folder. To find the location of your plotter files, on the Tools menu, click Options. In the Options dialog box, Files tab, click the plus sign to the left of Printer Support File Path. Click the plus sign to the left of the Printer Configuration Search Path file. Under Printer Configuration Search Path, click the path name to view the location of your plotter files.)
- Choose Edit Plotter Configuration from within the Add-a-Plotter wizard.
- Choose Properties in the Page Setup dialog box.
- Choose Properties in the Plot dialog box.

Depending on your configured plotting device, additional options may be available on the Device and Document Settings tab. For example, when you configure a nonsystem plotter, you can modify the pen characteristics.
The Plotter Configuration Editor includes the following tabs:
- General
- Ports
- Device and Document Settings

- ## General

Contains basic information about the plotter configuration (PC3) file. You can add or modify the information in the Description area. The remainder of the tab is read-only.

Plotter Configuration File Name

Displays the file name you assigned in the Add-a-Plotter wizard.

Description

Displays information you want to include about the plotter.

Driver Information

Displays the following information:

- Plotter driver type (system or nonsystem), name, model, and location
- HDI driver file version number (AutoCAD specialized driver files)
- UNC name of the network server (if the plotter is connected to a network server)
- I/O port (if the plotter is connected locally)
- Name of the system printer (if the configured plotter is the system printer)
- PMP (Plot Model Parameters) file name and location (if a PMP file is attached to the PC3 file)

• Ports

Changes communication settings between the configured plotter and your computer or network system. You can specify whether you want to plot through a port, plot to a file, or use AutoSpool.

If you plot through a parallel port, you can specify the timeout value. If you plot through a serial port, you can change the baud rate, protocol, flow control, and input and output timeout values.

Plot to the Following Port

Sends the drawing to the plotter through the selected port.

Plot to File

Sends the drawing to the file specified in the Plot dialog box.

AutoSpool

Uses the AutoSpool utility to plot the drawing. AutoSpool is specified on the Files tab in the Options dialog box.

Port List

Displays a list and description of available ports, both local and network. If the port is a network port, the name of the network printer is also displayed.

Show All Ports

Shows all available ports on the computer regardless of the ports available on the plotter.

Browse Network

Displays the network choices so you can connect to another instance of a nonsystem plotter.

Configure Port

Displays either the Configure LPT Port dialog box or the Settings for COM Port dialog box.

• Device and Document Settings

Controls many of the settings in the PC3 file. Click the icon for any of the nodes to view and change the specific settings. When you change a setting, your changes appear in angle

brackets (< >) next to the setting name. A check mark is also displayed over the icon of the node with a changed value.

Note Only the settings available for the configured device are displayed in the tree view. In addition, you might not be able to edit some settings if the device handles the setting through Custom Properties or doesn't support the capability.

Media Node

Specifies a paper source, size, type, and destination. Available settings depend on the supported features of your configured plotter. For Windows system printers, you must configure the media settings using the Custom Properties node.

Source and Size

Specifies the paper source and size.

- *Source:* Specifies the paper source; for example, sheet-fed or roll-fed. If you specify a tray, you can select the type of tray.
- *Width:* Specifies the width of the paper roll for a roll-fed source.
- *Automatic:* Enables the printer to specify the appropriate paper source.
- *Size:* Displays a list of available paper sources and both standard and custom paper sizes.
- *Printable Bounds:* Displays the print boundaries.

Media Type

Displays a list of the media types supported by the plotter configuration.

Duplex Printing

Determines double-sided printing and binding margin. Binding margin options are available only for plotters that support duplex printing.

- *None:* Indicates no double-sided printing.
- *Short Side:* Places the binding margin on the short side of the paper.
- *Long Side:* Places the binding margin on the long side of the paper.

Media Destination

Displays a list of available media destinations for the configured plotter, such as collating, cutting, and stapling. These options are available only for plotters that support this function.

Physical Pen Configuration Node (for Pen Plotters Only)

Controls the specific pens in the pen plotter. The lower pane of the Device and Document Settings tab displays a table used to describe the color, width, and speed of each pen in the plotter.

Note The physical pen information cannot be detected automatically; you must provide this information for your pen plotter under Physical Pen Characteristics.

Pen Configuration

Specifies settings for pen plotters.

- *Prompt for Pen Swapping:* Use more than one pen while plotting on a single-pen plotter. Under Physical Pen Characteristics, you can specify settings for as many pens as you want. You are prompted when you need to change pens.

- *Area Fill Correction:* Enables the program to compensate for pen widths when plotting filled areas and wide polylines. Each polygon is shrunk by half of the width of the pen used to draw it. This option prevents the plotter from oversizing the filled area when using a wide pen and ensures precision. Use this option if the plot must be accurate to half the pen width, such as in printed circuit artwork.

- *Pen Optimization Level:* Reduces plot time and increases the efficiency of the pens by optimizing the pen motion. For example, you can prevent pens from retracing duplicate lines. If your drawing uses many colors or widths, you can reduce the time needed to change pens by selecting Adds Pen Sorting. Every object that uses a particular pen will be plotted before switching to another pen. Each method in the list includes the optimization methods preceding it in the list (except for No Optimization).

Physical Pen Characteristics

To plot your drawing correctly on a pen plotter, you need to provide information about the pens in your plotter. For each pen in your plotter, specify a color and width. To optimize pen performance, you can specify a speed.

Note This information is required; the physical pen information cannot be detected automatically.

- *Color:* Specifies the colors of the pens in your plotter. The pen color that closely matches the object's color is selected. You can assign colors to your pens that correspond to each object's color. Use the Color list to select one of the following settings: No Pen, Red, Yellow, Green, Cyan, Blue, Magenta, Black, or Other. If you choose Other, the Select Color dialog box is displayed, providing access to the full color palette. Use No Pen to indicate that there is no pen in a particular position.

- *Speed:* Adjusts pen speed on a pen-by-pen basis. This feature is useful, for example, for slowing down pens that are skipping. Each pen manufacturer recommends a pen speed for each type of media. For best results, use those values. You can specify a pen speed in millimeters or inches per second.

- *Width:* Specifies the width of your pens so the program can determine if multiple pen strokes are needed to draw wide lines. You can specify the pen width in inches or millimeters. Be sure to select the pen width to match the actual width of the pen. The list provides a set of common pen widths.

Specifying pen characteristics here doesn't replace the pen tables you might have imported from PCP, PC2, or CFG files from AutoCAD Release 14, AutoCAD LT® 98, or earlier releases.

Graphics Node

Specifies settings for printing vector graphics, raster graphics, and TrueType text. Depending on the capabilities of the plotter, you can modify color depth, resolution, and dithering. You can select either color or monochrome output for vector drawings. When printing raster images on a plotter with limited memory, you can improve performance by making some changes to the quality of the printed output. If you use a nonsystem plotter that supports varying amounts of installed RAM, you can provide that information to improve performance.

Installed Memory

Provides the program with the amount of total memory (RAM) installed on a nonsystem plotter. This option is only available for nonWindows system printers that accept optional memory. If your plotter has extra memory, specify the total amount of memory.

- *Total Installed Memory:* Specifies the total installed memory in megabytes. The plotter driver uses the memory information to determine whether banding or image quality degradation is required to prevent the system from running out of memory.

Vector Graphics

Provides options for specifying the color depth, resolution, and dithering of vector output. Some of the Vector Graphics options are closely interrelated; changing an option can affect other available options.

- *Color Depth:* Displays a list for choosing color depth for the configured plotter. The color depth options change as you change the resolution and dithering values. More color depth uses more memory and takes longer to plot. You can specify either color or monochrome output.
- *Resolution:* Adjusts the DPI (dots per inch) resolution of the configured plotter. Changing the DPI resolution changes the options available in the Dithering list. A higher resolution setting uses more memory and takes longer to plot than a lower resolution setting.
- *Dithering:* Specifies a dithering choice for non-pen-based plotters. Some dithering choices cause slower plotting.

Raster Graphics (Non-Pen Plotters Only)

Specifies trade-offs between plotting speed and output quality when plotting raster objects. If you reduce the image quality, you increase output speed. If your system resources are limited, reducing image quality can reduce the chance of running out of memory while plotting. These options are available only for raster devices, not pen plotters.

- *Raster and Shaded/Rendered Viewports:* Specifies a position on the slider that balances output quality with memory and plotting speed when plotting raster images and shaded/rendered viewports. Position the slider at None to disable raster image printing. Degrading the image quality lets you plot in less time. Position the slider at Best for the best output at the expense of memory and plotting speed.
- *OLE:* Specifies a position on the slider that balances output quality with memory use and plotting speed when plotting OLE objects. Position the slider at None to disable OLE object printing. Degrading the image quality lets you plot in less time. Position the slider at Best for the best output at the expense of memory and plotting speed.
- *Trade-off:* Specifies where to compromise quality if you can't output at the highest quality. Move the slider to diminish resolution and color.

TrueType Text

On Windows system printers, specifies whether to plot TrueType text as a graphic image or as text. Plotting as a graphic guarantees that the text is printed as displayed, at the expense of slowing down the plotter and using more memory. Plotting as TrueType text prints faster and uses less memory; the plotter may use a different font for printing.

- *TrueType as Text:* Plots TrueType text as text.
- *TrueType as Graphics:* Plots TrueType text as graphics.

Merge Control

On raster plotters, controls the appearance of lines that cross. Merge control is not effective if your plotter is configured to plot everything as black or if you are using PostScript language.

- *Lines Overwrite:* Uses the last plotted line to obscure the lines under it. Only the topmost line is visible at the intersection.
- *Lines Merge:* Merges the colors of crossing lines.

Note Merge control may appear as an option for system printers that do not actually support the feature. Please check your printer's documentation to determine if merge control is supported.

Custom Properties Node

Modifies the device-specific properties for the plotter configuration. The settings for each plotter vary. If the plotter manufacturer has not included a Custom Properties dialog box for the device driver, the Custom Properties option is disabled. For some drivers, such as ePlot, this is the only tree view option that is displayed. For Windows system printers, most of the device-specific settings are made in this dialog box. For more information about the custom properties settings for your device, choose Help in the Custom Properties dialog box.

Initialization Strings Node (for Non-System Plotters Only)

Sets pre-initialization, post-initialization, and termination plotter strings. If you are plotting to an unsupported plotter in emulation mode, you can specify ASCII text initialization strings that prepare the plotter for printing, set device-specific options, and restore the plotter to its original state. Initialization strings should be used by advanced users only.

Pre-Initialization

Forces a plotter to emulate another plotter by sending a pre-initialization string to the plotter before it is initialized.

The text string is sent literally, except for a back slash (\). A back slash followed by a number, for example, \27, is interpreted and sent as a single character whose value is 27. The number 27 is the escape character. For example, \27%-12345X@PJL ENTER LANGUAGE = PostScript\10 sends an HP PJL command to a dual-language laser printer before it's initialized and switches the printer into PostScript mode. The \27 sends an escape character and the \10 sends a line feed character. The remainder of the text string is sent literally.

Post-Initialization

Sets a device-specific option that is not supported elsewhere in the program. Specify a post-initialization string that is sent to the plotter after it is initialized.

Termination

Restores the printer to its original state after plotting. Specify a termination string that is sent to the plotter after plotting is complete.

User-Defined Paper Sizes & Calibration Node

Attaches a PMP file to the PC3 file, calibrates the plotter, and adds, deletes, revises, or filters custom paper sizes. You can also modify standard paper sizes. This node accesses the Plotter Calibration and Custom Paper Size wizards. If the plotter you are using has been calibrated, a Plotter Model Parameter (PMP) file contains that calibration information. If the PMP file is not already attached to the PC3 file you are editing, you must create that association so you can use the PMP file. If the plotter was calibrated from within the Add-a-Plotter wizard while creating the current PC3 file, the PMP file is already attached. Use the PMP File Name option under User-defined Paper Sizes & Calibration to attach a PMP file to, or detach the PMP file from, the PC3 file you are editing.

Custom Paper Sizes (Nonsystem Printers Only)

Creates a customized paper size or changes the printable area of a standard or nonstandard paper size. With the Custom Paper Size wizard you can create a new paper size, or select from a list of available paper sizes (from a PMP file) if the plotter is not a

Windows system printer. If the plotter is a Windows system printer, use Custom Properties. You can add, delete, and edit your paper size.

Each plotter has a maximum printable area determined by where it grips the paper and how far the pen shuttle can reach. If you are creating a paper size that is larger than the paper sizes offered in the Plotter Configuration Editor, verify that the plotter is capable of plotting the new dimensions.

- *Add:* Starts the Custom Paper Size wizard. When you add a paper size, you can either create a new paper size from scratch or create a new one based on the listing of available paper sizes for the selected configured plotter. The new paper size is a user-defined size, not a standard size.
- *Delete:* Deletes the selected custom paper size from the list.
- *Edit:* Starts the Custom Paper Size wizard, where you can modify the selected paper size. You can change any of the custom paper size settings.

Note For Windows system printers, you can change the paper size settings and create custom paper sizes only in the Custom Properties dialog box.

Modify Standard Paper Sizes

Adjusts the printable area for standard paper sizes to match the printer's capabilities. (You can't create custom paper sizes for Windows system printers using the Plotter Configuration Editor.)

- *List of Standard Paper Sizes:* Displays the available set of standard paper sizes.
- *Modify:* Starts the Custom Paper Size wizard. You can modify Printable Area and File Name. The new paper size is a user-defined size, not a standard size.

Filter Paper Sizes

Filters the list of paper sizes displayed for the plotting device selected in the Plot and Page Setup dialog boxes. The list of paper sizes is displayed on the Plot Settings tab in the Plot dialog box and on the Layout Settings tab in the Page Setup dialog box. Select the paper sizes you want to display for this device.

- *Check All:* Hides all the paper sizes for the device.
- *Uncheck All:* Displays all the paper sizes for the device.

Plotter Calibration

Starts the Plotter Calibration wizard. If you need to correct scaling discrepancies, you can adjust the plotter calibration using the Plotter Calibration wizard.

Note You should perform a plotter calibration only if your drawings must be exactly to scale and your plotter or printer produces inaccurate plots. Plotter Calibration causes the program to rescale all plots sent to your plotter. If your plotter provides a calibration utility, it is recommended that you use it instead of the utility supplied with this program.

PMP File

Attaches a PMP file to or detaches a PMP file from the PC3 file you are editing. Use the Detach button to break the association between the PMP file and the PC3 file.

- *Attach:* Attaches a PMP file to the PC3 file. You can reuse calibration and custom paper size data stored in the PMP file.
- *Save PMP:* Saves a PMP file to a new file in the *AutoCAD 2006\drv* folder.
- *Detach:* Detaches the PMP file associated with the PC3 file you are editing.

Import

Imports file information from earlier versions of the program. If you have a PCP or PC2 file from an earlier version, you can import some of the information in those files into a PC3 file. PC3 files store plotter name, port information, pen optimization level, paper size, and resolution.

Save As

Saves a PC3 File to a new file name.

Defaults

Restores the settings on the Device and Document Settings tab back to the default settings

PNGOUT

Saves selected objects to a file in a Portable Network Graphics format

Command line: pngout

The Create Raster File dialog box (a standard file selection dialog box) is displayed. Enter the file name in the dialog box.

   Select objects or <all objects and viewports>: *Press* ENTER *to select all objects and viewports or use an object selection method and press* ENTER

A Portable Network Graphics file is created that contains the objects you select. The file reflects what is displayed on the screen.

Note When the FILEDIA system variable is set to 0 (Off), prompts are displayed on the command line.

POINT

Creates a point object

Draw toolbar:

Draw menu: Point

Command line: point

   Specify a point:

Points can act as nodes to which you can snap objects. You can specify a full three-dimensional location for a point. The current elevation is assumed if you omit the Z coordinate value.

The PDMODE and PDSIZE system variables control the appearance of point objects. PDMODE values 0, 2, 3, and 4 specify a figure to draw through the point. A value of 1 specifies that nothing is displayed.

0   1   2   3   4

Specifying the value 32, 64, or 96 selects a shape to draw around the point, in addition to the figure drawn through it:
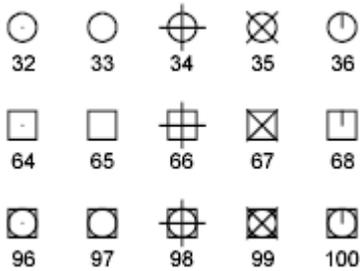
PDSIZE controls the size of the point figures, except for PDMODE values 0 and 1. A setting of 0 generates the point at 5 percent of the drawing area height. A positive PDSIZE value specifies an absolute size for the point figures. A negative value is interpreted as a percentage of the viewport size. The size of all points is recalculated when the drawing is regenerated. After you change PDMODE and PDSIZE, the appearance of existing points changes the next time the drawing is regenerated.

POLYGON

Creates an equilateral closed polyline

Draw toolbar:

Draw menu: Polygon

Command line: polygon

Enter number of sides *<current>*: *Enter a value between 3 and 1024 or press* ENTER

Specify center of polygon or [Edge]: *Specify a point (1) or enter* **e**
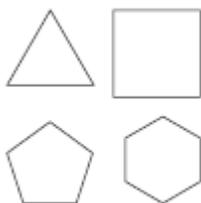
polygons

Center of Polygon

Defines the center of the polygon.

Enter an option [Inscribed in circle/Circumscribed about circle] *<current>*: *Enter* **i** *or* **c** *or press* ENTER

Inscribed in Circle

Specifies the radius of a circle on which all vertices of the polygon lie.

Specify radius of circle: *Specify a point (2) or enter a value*
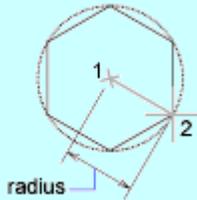
Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

Circumscribed about Circle

Specifies the distance from the center of the polygon to the midpoints of the edges of the polygon.

Specify radius of circle: *Specify a distance*
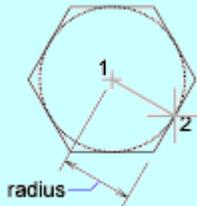
Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

Edge

Edge

Defines a polygon by specifying the endpoints of the first edge.

Specify first endpoint of edge: *Specify a point (1)*

Specify second endpoint of edge: *Specify a point (2)*
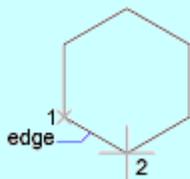
PREVIEW

Shows how the drawing will look when it is plotted

Standard toolbar:

File menu: Plot Preview

Command line: preview

PREVIEW displays a full-page preview of the current drawing. The preview is based on the current plot configuration, as defined by the settings in the Page Setup or Plot dialog box. Press ESC or ENTER to exit, or right-click in the preview area to access the **shortcut menu**, which offers additional preview options.

The cursor changes to a magnifying glass with plus (**+**) and minus (**–**) signs. Dragging the cursor toward the top of the screen while holding down the pick button enlarges the preview image. Dragging toward the bottom of the screen reduces the preview image.

The Preview window toolbar provides the following buttons:

Plot

Plots the drawing as it is displayed in the full-sheet preview, and then exits Plot Preview.

Pan

Displays the pan cursor, a hand cursor that you can use to pan the preview image. Hold down the pick button and drag the cursor in any direction. The pan cursor stays active until you click another button.

Zoom

Displays the zoom cursor, a magnifying-glass cursor that you can use to enlarge or reduce the preview image. To enlarge the image, hold down the pick button and drag the cursor toward the top of the screen. To reduce the image, hold down the pick button and drag the cursor toward the bottom of the screen.

Zoom Window

Zooms to display a specified window. Zoom Window works with both the zoom cursor and the pan cursor.

Zoom Original

Restores the initial full-sheet preview. Zoom Original works with both the zoom cursor and the pan cursor.

Close Preview Window

Closes the Preview window.

Preview Shortcut Menu

Preview Shortcut Menu

Controls Plot Preview options. To access this menu, right-click in the preview area.

Exit

Exits the preview.

Plot

Plots the drawing as it is displayed in the full-sheet preview, and then exits Plot Preview.

Pan

Displays the pan cursor, a hand cursor that you can use to pan the preview image. Hold down the pick button and drag the cursor in any direction. The pan cursor stays active until you click another option on the shortcut menu.

pan cursor

Zoom

Displays the zoom cursor, a magnifying-glass cursor that you can use to enlarge or reduce the preview image. To enlarge the image, hold down the pick button and drag the cursor toward the top of the screen. To reduce the image, hold down the pick button and drag the cursor toward the bottom of the screen.

zoom cursor

Zoom Window

Zooms to display a specified window. Zoom Window works with both the zoom cursor and the pan cursor.

Zoom Original

Restores the initial full-sheet preview. Zoom Original works with both the zoom cursor and the pan cursor.

PROPERTIES

Controls properties of existing objects

Standard toolbar:

Tools menu: Properties

Shortcut menu: Select the objects whose properties you want to view or modify, right-click in the drawing area, and click Properties.

Pointing Device: Double-click most objects

⌨ **Command line:** properties

The Properties palette is displayed. The Properties palette lists properties of the selected object or set of objects.

You can also view or modify properties of third-party application objects that are based on AutoCAD application programming interface (API) standards.

There are eight general properties common to all objects. All other object properties are specific to the type of object.

Displays the properties of the selected object or set of objects.

- 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 and layout, the name of the plot style table attached to the layer, the view properties, and information about the UCS.

You can specify a new value to modify any property that can be changed. Click the value and use one of the following methods:

- 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.

Object Type

Displays the type of object that is selected.

Toggle Value of PICKADD Sysvar

Turns the PICKADD system variable on (1) and off (0). When PICKADD is on, each object selected, either individually or by windowing, is added to the current selection set. When PICKADD is off, selected objects replace the current selection set.

Select Objects

Selects desired objects using any selection method. The properties common to the selected objects are displayed in the Properties palette. You can then modify the properties of the selected objects in the Properties palette, or you can make other changes to the selected objects by entering an editing command.

Quick Select

Displays the Quick Select dialog box . Use Quick Select to create selection sets based on filtering criteria.

The following shortcut menu options are available when you right-click the title bar.

Move

Displays a four-headed arrow cursor that you can use to move the palette. The palette does not dock.

Size

Displays a four-headed arrow cursor that you can use to drag an edge or a corner to make the palette smaller or larger.

Close

Closes the Properties palette.

Allow Docking

Controls whether the Properties palette docks when you drag it over a docking area at the side of the drawing.

Auto-hide

Controls display of the Properties palette. When this option is checked, only the palette title bar is displayed when the cursor moves outside the palette. When this option is not checked, the full palette is displayed when the cursor moves.

Description

Controls display of the description area at the bottom of the Properties palette. When this option is checked, descriptions are displayed. When this option is not checked, the description area is not displayed.

PROPERTIESCLOSE

Closes the Properties palette

Command line: propertiesclose

The Properties palette is closed.

PSETUPIN

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

Command line: psetupin

The Select Page Setup From File dialog box (a standard file selection dialog box) is displayed in which you can select the drawing *(.dwg)*, template (*.dwt*), or drawing interchange format (*.dxf*) file whose page setups you want to import.

If FILEDIA is set to 0 (zero) and you enter **-psetupin** at the Command prompt, PSETUPIN displays prompts on the command line.
When you select the drawing file that you want to use, the Import Page Setups dialog box is displayed.

PSPACE

Switches from a model space viewport to paper space

Command line: pspace

The program switches from model space to paper space when you are working on a layout tab.
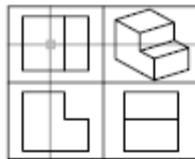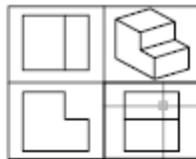
paper space   model space

On the layout tab, use paper space to create a finished layout of a drawing for printing. As part of designing your layout, you create layout viewports, which are windows containing different views of the model. By switching from paper space to model space (see MSPACE), you can edit the model and views within the current layout viewport.

You can make a viewport current by double-clicking inside it. You can switch to paper space by double-clicking an area of the paper space layout that is not within a viewport.

You can also switch between model space and paper space by choosing Model or Paper on the status bar.

PURGE

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

File menu: Drawing Utilities ➤ Purge

Command line: purge

The Purge dialog box is displayed.
If you enter **-purge** at the Command prompt, PURGE displays prompts on the command line.

Displays items that can be purged.

View Items You Can Purge

Switches the tree view to display a summary of named objects in the current drawing that you can purge.

Items Not Used in Drawing

Lists the named objects that are not used in the current drawing and that can be purged. You can list the items for any object type by clicking the plus sign or by double-clicking the object type. You purge items by selecting the item to purge.

Purge Nested Items removes items only when you select one of the following options:

- All Items or Blocks in the tree view
- The Purge All button

Confirm Each Item to Be Purged

Displays the Confirm Purge dialog box when you purge an item.

Purge Nested Items

Removes all unused named objects from the drawing even if they are contained within or referenced by other unused named objects. The Confirm Purge dialog box is displayed, and you can cancel or confirm the items to be purged.

View Items You Cannot Purge

Switches the tree view to display a summary of named objects in the current drawing that you can't purge.

Items Currently Used in Drawing

Lists named objects that cannot be removed from the drawing. Most of these objects are currently used in the drawing or are default items that cannot be removed. When you select individual named objects, information about why you can't purge the item is displayed below the tree view.

Tip

Displays information detailing why you can't purge the selected item.

Purge

Purges the selected items.

Purge All

Purges all unused items.

QDIM

Quickly creates a dimension

🔖 **Dimension toolbar:**

🔖 **Dimension menu:** Quick Dimension

⌨ **Command line: qdim**

Use QDIM to quickly create or edit a series of dimensions. The command is particularly useful for creating a series of baseline or continued dimensions, or for dimensioning a series of circles and arcs.

Select geometry to dimension: *Select the objects you want to dimension or the dimensions you want to edit and press* ENTER

Specify dimension line position, or
[**Continuous**/**Staggered**/**Baseline**/**Ordinate**/**Radius**/**Diameter**/**datumPoint**/**Edit**/**seTtings**] *<current>*: *Enter an option or press* ENTER

Continuous
Creates a series of continued dimensions.

Staggered
Creates a series of staggered dimensions.

Baseline
Creates a series of baseline dimensions.

Ordinate

Creates a series of ordinate dimensions.

Radius

Creates a series of radius dimensions.

Diameter

Creates a series of diameter dimensions.

Datum Point

Sets a new datum point for baseline and ordinate dimensions.

Select new datum point: *Specify a point*

The program returns to the previous prompt.

Edit

Edits a series of dimensions. You are prompted to add or remove points from existing dimensions.

Indicate dimension point to remove, or [Add/eXit] <eXit>: *Specify a point, enter* **a***, or press* ENTER *to return to the previous prompt*

Settings

Sets the default object snap for specifying extension line origins. The following prompt is displayed:

Associative dimension priority [Endpoint/Intersection]

The program returns to the previous prompt.

QLEADER

Creates a leader and leader annotation

Dimension toolbar:

Dimension menu: Leader

Command line: qleader

Use QLEADER to quickly create leaders and leader annotation. You can use the Leader Settings dialog box to customize the command so that it prompts you for the number of leader points and the annotation type suited to your drawing needs. You can use QLEADER to

- Specify leader annotation and annotation format
- Set the location where leaders attach to multiline text annotation
- Limit the number of leader points
- Constrain the angle of the first and second leader segments

If associative dimensioning is turned on with DIMASSOC, the leader start point can be associated with a location on an object. If the object is relocated, the arrowhead remains attached to the object and the leader line stretches, but the text or feature control frame remains in place.

Specify **first leader point**, or [**Settings**] <Settings>: *Specify the first leader point, or press* ENTER *to specify leader settings*

First Leader Point

Specify next point: *Specify the next leader point*

Specify next point: *Specify the next leader point, or press* ENTER *to specify the leader annotation*

The Number of Points setting on the Leader Line & Arrow tab of the Leader Settings dialog box determines the number of leader points you are prompted to specify.

The prompt that is displayed next depends on settings selected on the Annotation tab in the Leader Settings dialog box.

If Mtext and Prompt for Width are selected on the Annotation tab, the following prompts are displayed:

Specify the width <*current*>: *Specify the multiline text width by creating a text boundary or entering a value*

If you set the text width value to 0.00, the width of the multiline text is unlimited.

Enter first line of annotation text: *Enter the first line of text*

Press ENTER once to enter another line of text, or press ENTER again to complete the command.

If Copy an Object is selected on the Annotation tab, the following prompt is displayed:

Select an object to copy: *Select a text object, block reference, or tolerance object (feature control frame)*

The object is attached to the leader.

If Tolerance is selected on the Annotation tab, the Geometric Tolerance dialog box is displayed. Use the dialog box to create the tolerance feature control frame. When you choose OK, the feature control frame is attached o the leader.

If Block Reference is selected on the Annotation tab, the following prompts are displayed:

Enter block name or [?]: *Enter the name, or enter* ? *to display a list of blocks defined in the drawing*

Specify insertion point or [Scale/X/Y/X/Rotate/PScale/PX/PY/PZ/PRotate]: *Specify the block insertion point or enter an option*

For a description of the insertion options, see INSERT Command Line.

If None is selected on the Annotation tab, no annotation prompts are displayed.

Settings

Settings

Displays the Leader Settings dialog box.

QSAVE

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

Standard toolbar:

File menu: Save

Command line: qsave

The QSAVE command is equivalent to clicking Save on the File menu.

If the drawing is named, the program saves the drawing using the file format specified on the Open and Save tab of the Options dialog box and does not request a file name. If the drawing

is unnamed, the Save Drawing As dialog box (see SAVEAS) is displayed and the drawing is saved with the file name and format you specify.
If the drawing is read-only, use the SAVEAS command to save the changed file under a different name.

QSELECT

Creates a selection set based on filtering criteria

Tools menu: Quick Select

Shortcut menu: End any active commands, right-click in the drawing area, and choose Quick Select.

Command line: **qselect**

The Quick Select dialog box is displayed.

Specifies the filtering criteria and how you want to create the selection set from that criteria.

Apply To

Applies the filtering criteria to the entire drawing or to the current selection set (if one exists). To select a group of objects to which you want to apply the filtering criteria, use the Select Objects button. When you have finished selecting objects, press ENTER to redisplay the dialog box. Apply To is set to Current Selection.

If Append to Current Selection Set is selected, the filtering criteria is applied to the entire drawing.

Select Objects

Temporarily closes the Quick Select dialog box so that you can select the objects to which you want to apply the filter criteria. Press ENTER to return to the Quick Select dialog box. The Apply To box is changed to show Current Selection. The Select Objects button is available only when you select Include In New Selection Set and clear Append to Current Selection Set.

Object Type

Specifies the type of objects to include in the filtering criteria. If the filtering criteria are being applied to the entire drawing, the Object Type list includes all object types, including custom. Otherwise, the list includes only the object types of the selected objects.

If an application such as Autodesk Map was used to add a feature classification to an object, you can select a classification.

Properties

Specifies the object property for the filter. This list includes all searchable properties for the selected object type. The property you select determines the options available in Operator and Value.

If an application such as Autodesk® Map™ was used to add a feature classification to an object, you can select a classification property.

Operator

Controls the range of the filter. Depending on the selected property, options may include Equals, Not Equal To, Greater Than, Less Than, and *Wildcard Match. Greater Than and Less Than are not available for some properties. *Wildcard Match is available only for text fields that can be edited.

Value

Specifies the property value for the filter. If known values for the selected property are available, Value becomes a list in which you can choose a value. Otherwise, enter a value.

How to Apply

Specifies whether you want the new selection set to include or exclude objects that match the specified filtering criteria. Select Include in New Selection Set to create a new selection set composed only of objects that match the filtering criteria. Select Exclude from New Selection Set to create a new selection set composed only of objects that do not match the filtering criteria.

Append to Current Selection Set

Specifies whether the selection set created by QSELECT replaces or is appended to the current selection set.

Note QSELECT supports custom objects (objects created by another application) and their properties. If a custom object uses properties other than AutoCAD properties, the custom object's source application must be running in order for the properties to be available to QSELECT.

QTEXT

Controls the display and plotting of text and attribute objects

⌨ **Command line:** qtext (or 'qtext for transparent use)

Enter mode [ON/OFF] <*current*>: *Enter* **on** *or* **off***, or press* ENTER

When QTEXT (Quick Text) is on, each text and attribute object is displayed as a bounding box around the text object. Turning QTEXT mode on reduces the time it takes the program to redraw and regenerate drawings that contain many text objects.
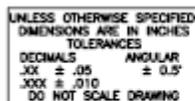
QTEXT off   QTEXT on

QUICKCALC

Opens the QuickCalc calculator

🟦 **Standard toolbar:**

🟦 **Tools menu:** QuickCalc

Shortcut menu: Right-click and click QuickCalc.

⌨ **Command line:** quickcalc

Displays the QuickCalc calculator.

QUIT

Exits the prgram

File menu: Exit

Command line: quit

Quits the program if there have been no changes since the drawing was last saved. If the drawing has been modified, you are prompted to save or discard the changes before quitting. You can quit a file that has been opened in read-only mode if you have made no modifications or if you are willing to discard them. To save modifications to a read-only drawing, use the SAVEAS command to save the drawing under another name.

RAY

Creates a semi-infinite line

Draw menu: Ray

Command line: ray

Specify start point: *Specify a point (1)*

Specify through point: *Specify a point for the ray to pass through (2)*

The ray is extended to the edge of the display in the direction defined by the starting point and the through point. The prompt for a through point is redisplayed so you can create multiple tays. Press ENTER to end the command.
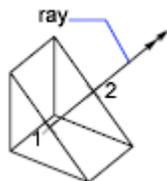
RECOVER

Repairs a damaged drawing

File menu: Drawing Utilities ➤ Recover

Command line: recover

In the Select File dialog box (a standard file selection dialog box), enter the drawing file name or select the damaged drawing file. Results are displayed in the text window.
When FILEDIA is set to 0 (zero), RECOVER displays the following prompt on the command line.

Enter name of drawing file to recover:

Enter ~ (tilde) at the prompt to ignore FILEDIA and display the Recover Drawing dialog box.

Note The RECOVER command performs recoveries or audit operations on DWG files only. Performing a recover on a DXF™ file will only open the file.

RECTANG

Draws a rectangular polyline

Draw toolbar:

Draw menu: Rectangle

Command line: rectang or **rectangle**

Current settings: Rotation = 0

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: *Specify a point or enter an option*

First Corner Point

First Corner Point

Specifies a corner point of the rectangle.

Specify other corner point or [Area/Dimensions/Rotation]: *Specify a point or enter an option*

Other Corner Point
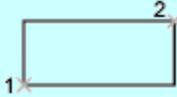
Creates a rectangle using the specified points as diagonally opposite corners.

Area

Creates a rectangle using the area and either a length or a width. If the Chamfer or Fillet option is active, the area includes the effect of the chamfers or fillets on the corners of the rectangle.

Enter area of rectangle in current units <100>: *Enter a positive value*

Calculate rectangle dimensions based on [Length/Width] <Length>: *Enter* **L** *or* **w**

Enter rectangle length <10>: *Enter a non-zero value*

or

Enter rectangle width <10>: *Enter a non-zero value*

Specify other corner point or [Area/Dimensions/Rotation]: *Move the cursor to display one of four possible locations for the rectangle and click the one that you want*

Dimensions

Creates a rectangle using length and width values.

Specify length for rectangles <0.0000> *Enter a non-zero value*

Specify width for rectangles <0.0000> *Enter a non-zero value*

Specify other corner point or [Area/Dimensions/Rotation]: *Move the cursor to display one of four possible locations for the rectangle and click the one that you want*

Rotation

Creates a rectangle at a specified rotation angle.

Specify rotation angle or [Points] <0> Specify an angle by entering a value, specifying a point, or entering **p** and specifying two points

Specify other corner point or [Area/Dimensions/Rotation]: *Move the cursor to display one of four possible locations for the rectangle and click the one that you want*

Chamfer

Chamfer

Sets the chamfer distances for the rectangle.

Specify first chamfer distance for rectangles *<current>*: *Specify a distance or press* ENTER

Specify second chamfer distance for rectangles *<current>*: *Specify a distance or press* ENTER

The values become the current chamfer distances for subsequent RECTANG commands.

Elevation

Elevation

Specifies the elevation of the rectangle.

> Specify the elevation for rectangles *<current>*: *Specify a distance or press* ENTER

The value becomes the current elevation for subsequent RECTANG commands.

Fillet

Fillet

Specifies the fillet radius of the rectangle.

> Specify fillet radius for rectangles *<current>*: *Specify a distance or press* ENTER

The value becomes the current fillet radius for subsequent RECTANG commands.

Thickness

Thickness

Specifies the thickness of the rectangle.

> Specify thickness for rectangles *<current>*: *Specify a distance or press* ENTER

The value becomes the current thickness for subsequent RECTANG commands.

Width

Width

Specifies the polyline width of the rectangle to be drawn.

> Specify line width for rectangles *<current>*: *Specify a distance or press* ENTER

The value becomes the current polyline width for subsequent RECTANG commands.

REDO

Reverses the effects of previous UNDO or U command

 Standard toolbar:

 Edit menu: Redo

Shortcut menu: With no commands active and no objects selected, right-click in the drawing area and choose Redo.

 Command line: redo

REDO reverses the effects of a single UNDO or U command. REDO must immediately follow the U or UNDO command.

REDRAW

Refreshes the display in the current viewport

 Command line: redraw (or **'redraw** for transparent use)

When BLIPMODE is on, marker blips left by editing commands are removed from the current viewport.
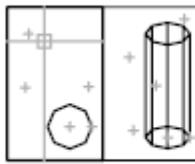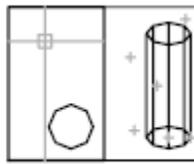
before REDRAW   after REDRAW

REDRAWALL

Refreshes the display in all viewports

View menu: Redraw

Command line: redrawall (or **'redrawall** for transparent use)

When BLIPMODE is on, marker blips left by editing commands are removed from all viewports.
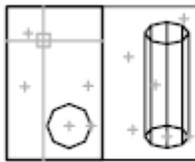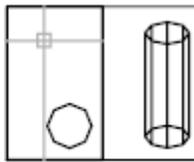
before REDRAWALL   after REDRAWALL

REFCLOSE

Saves back or discards changes made during in-place editing of a reference (an xref or a block)

Tools menu: Xref and Block In-Place Editing ➤ Save Reference Edits, or Xref and Block In-Place Editing ➤ Close Reference

Shortcut menu: With no objects selected during in-place reference editing, right-click in the drawing area and choose Close Refedit Session.

Command line: refclose

Enter option [Save/Discard reference changes] <Save>:

Save
Saves back to the xref source drawing or to the block definition in the current drawing all changes made to objects in the working set. If you remove an object from the working set and save changes, the object is deleted from the reference and added to the current drawing. The Save Back Changes to Reference button on the Refedit toolbar automatically saves reference editing changes.
Discard Reference Changes

Discards the working set; the source drawing or block definition is returned to its original state. 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. The Discard Changes to Reference button on the Refedit toolbar automatically discards reference editing changes without using REFCLOSE.

If you save or discard changes with REFCLOSE, you can still use the UNDO command to return to the reference editing session. If you have made unwanted changes to an xref and already saved back the changes, use UNDO to undo the unwanted changes; then use REFCLOSE to save back changes and restore the xref to its original state.

Note When you edit and save xrefs in place in a drawing, the preview image for the original reference drawing is no longer available unless you open and save the drawing again.

REFEDIT

Selects an external reference or block reference for editing

 Refedit toolbar:

 Tools menu: Xref and Block In-place Editing ➤ Edit Reference In-Place

 Command line: refedit

Select reference: *Select an xref or a block in the current drawing*

The Reference Edit dialog box is displayed.
If you enter **-refedit** at the Command prompt, REFEDIT displays prompts on the command line.

Specifies the reference to edit. To select a reference to edit, select an object in the reference. If you select an object that is part of one or more nested references, the nested references are displayed in the dialog box.

The Reference Edit dialog box includes the following tabs:

- **Identify Reference**
- **Settings**

Identify Reference Tab (Reference Edit Dialog Box)

Provides visual aids for identifying the reference to edit and controls how the reference is selected.

Reference Name

Displays the reference selected for in-place editing and any references nested within the selected reference.

Nested references are displayed only if the selected object is part of a nested reference. If multiple references are displayed, choose a specific xref or block to modify. Only one reference can be edited in place at a time. References that are inserted into a drawing using MINSERT cannot be selected for in-place reference editing.

Dynamic blocks are not displayed in the list and are not available for editing through this dialog box.

Preview

Displays a preview image of the currently selected reference.

The preview image displays the reference as it was last saved in the drawing. The reference preview image is not updated when changes are saved back to the reference.

Path

Displays the file location of the selected reference. If the selected reference is a block, no path is displayed.

Automatically Select All Nested Objects

Controls whether nested objects are included automatically in the reference editing session.

If this option is checked, all the objects in the selected reference will be automatically included in the reference editing session.

Prompt to Select Nested Objects

Controls whether nested objects must be selected individually in the reference editing session.

If this option is checked, after you close the Reference Edit dialog box and enter the reference edit state, you are prompted to select the specific objects in the reference that you want to edit.

Select nested objects: *Select objects within the reference that you want to edit*

Settings Tab (Reference Edit Dialog Box)

Settings Tab (Reference Edit Dialog Box)

Provides options for editing references.

Create Unique Layer, Style, and Block Names

Controls whether layers and other named objects extracted from the reference are uniquely altered.

If selected, named objects in xrefs are altered (names are prefixed with $#$), similar to the way they are altered when you bind xrefs. If cleared, the names of layers and other named objects remain the same as in the reference drawing. Named objects that are not altered to make them unique assume the properties of those in the current host drawing that share the same name.

Display Attribute Definitions for Editing

Controls whether all variable attribute definitions in block references are extracted and displayed during reference editing.

If Display Attribute Definitions for Editing is selected, the attributes (except constant 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 affect only subsequent insertions of the block; the attributes in existing block instances are not affected. Xrefs and block references without definitions are not affected by this option.

Lock Objects Not in Working Set

Locks all objects not in the working set. This prevents you from accidentally selecting and editing objects in the host drawing while in a reference editing state.

The behavior of locked objects is similar to objects on a locked layer. If you try to edit locked objects, they are filtered from the selection set.

REFSET

Adds or removes objects from a working set during in-place editing of a reference (an xref or a block)

Tools menu: Xref and Block In-place Editing ➤ Add to Working Set, or Xref and Block In-Place Editing ➤ Remove from Working Set

Command line: refset

Transfer objects between the Refedit working set and host drawing...

Enter an option [Add/Remove] <Add>: *Enter an option or press*ENTER

Objects that are part of the working set are visually distinct from other objects in the current drawing. All objects in the current drawing, except objects in the working set, appear faded.

Add

Adds objects to the working set.

 Select objects: *Select the objects you want to add*

An object that is part of the working set is added to the reference when changes are saved back, and the object is removed from the current drawing. The Add to Workset button on the Refedit toolbar prompts you to add objects to the working set without using REFCLOSE.

REGEN

Regenerates the entire drawing from the current viewport

⬡ **View menu:** Regen

⌨ **Command line: regen**

REGEN regenerates the entire drawing and recomputes the screen coordinates for all objects in the current viewport. It also reindexes the drawing database for optimum display and object selection performance.
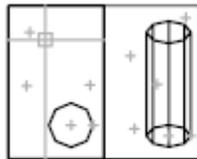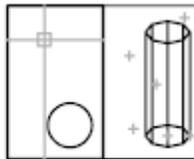
before REGEN   after REGEN

REGENALL

Regenerates the drawing and refreshes all viewports

⬡ **View menu:** Regen All

⌨ **Command line: regenall**

REGENALL regenerates the entire drawing and recomputes the screen coordinates for all objects in all viewports. It also reindexes the drawing database for optimum display and object selection performance.
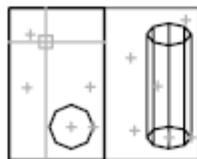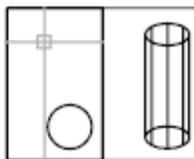
before REGENALL   after REGENALL

REGENAUTO

Controls automatic regeneration of a drawing

⌨ **Command line: regenauto** (or **'regenauto** for transparent use)

Enter mode [<u>ON</u>/<u>OFF</u>] *<current>*: *Enter* **on** *or* **off***, or press* ENTER

On
Regenerates the drawing immediately if any suppressed regenerations exist in the queue and continues to regenerate automatically whenever you perform an action that requires regeneration.
Off
Inhibits regeneration of the drawing until you use the REGEN or REGENALL command, or set

REGENAUTO to on.

If you perform an action that requires a regeneration and that action is irrevocable (such as thawing layers), the following message is displayed:

Regen queued

If you perform an action that requires a regeneration and that action is revocable, the following message is displayed:

About to regen—proceed?

If you click OK, the drawing is regenerated. If you click Cancel, the last action is cancelled and the drawing is not regenerated.

REGION

Converts an object that encloses an area into a region object

 Draw toolbar:

 Draw menu: Region

 Command line: region

Select objects: *Use an object selection method and press* ENTER *when you finish*

Regions are two-dimensional areas you create from closed shapes or loops. Closed polylines, lines, and curves are valid selections. Curves include circular arcs, circles, elliptical arcs, ellipses, and splines.
Closed 2D and exploded planar 3D polylines in the selection set are converted to separate regions and then converts polylines, lines, and curves to form closed planar loops (outer boundaries and holes of a region). If more than two curves share an endpoint, the resulting region might be arbitrary.
The boundary of the region consists of end-connected curves where each point shares only two edges. All intersections and self-intersecting curves are rejected.
If a selected polyline has been smoothed by either the Spline or Fit option of PEDIT, the resulting region contains the line or arc geometry of the smoothed polyline. The polyline is not converted to a spline object.
REGION deletes the original objects after converting them to regions unless the system variable DELOBJ is set to 0. If the original objects were hatched, hatch associativity is lost. To restore associativity, rehatch the region.

REINIT

Reinitializes the digitizer, digitizer input/output port, and program parameters file

 Command line: reinit

The Re-initialization dialog box is displayed

Defines the I/O port and the device file for reinitialization.

I/O Port Initialization

Reinitializes the I/O port for the digitizer.

Device File Initialization

Reinitializes the digitizer and the *acad.pgp* file.

RENAME

Changes the names of named objects

🔶 **Format menu:** Rename

⌨ **Command line: rename**

The Rename dialog box is displayed.

If you enter **-rename** at the Command prompt, RENAME displays prompts on the command line.

Renames named objects.

Named Objects

Lists named objects in a drawing by category: blocks, dimension styles, layers, linetypes, plot styles, table styles, text styles, UCSs, views, and model space viewport configurations. Select the objects you want to rename.

Items

Displays named objects of the type specified in Named Objects. Select the name you want to change.

Old Name

Specifies the named object to be renamed. Enter a name or select a name from the Items list. You can use the wild-card characters * and ? to rename groups of objects. You can't rename some standard objects, such as layer 0 and the CONTINUOUS linetype

Rename To

Specifies the new name you want to assign to the object. Enter a name and choose Rename To to apply the name change.

RESUME

Continues an interrupted script

⌨ **Command line: resume** (or **'resume** for transparent use)

You can interrupt a macro script that is running by pressing ESC or BACKSPACE. Any error encountered while processing input from a script file causes the script to be suspended. If a script is suspended while the program is active, you can use RESUME to continue the script.

ROTATE

Revolves objects around a base point

🔶 **Modify toolbar:**

🔶 **Modify menu:** Rotate

Shortcut menu: Select the objects to rotate, and right-click in the drawing area. Click Rotate.

⌨ **Command line: rotate**

Current positive angle in UCS: ANGDIR=*current* ANGBASE=*current*

Select objects: *Use an object selection method and press* ENTER *when you finish*

Specify base point: *Specify a point*

Specify **rotation angle** or [**Copy**/**Reference**]: *Enter an angle, specify a point, enter c, or enter* **r**

Rotation Angle

Determines how far an object rotates around the base point.

The axis of rotation passes through the specified base point and is parallel to the *Z* axis of the current UCS.

Copy

Creates a copy of the selected objects for rotation.

Reference

Rotates objects from a specified angle to a new, absolute angle.

Specify the reference angle *<last reference angle>*: *Specify an angle by entering a value or by specifying two points*

Specify the new angle or [Points] *<last new angle>*: *Specify the new absolute angle by entering a value or by specifying two points*

When you rotate a viewport object, the borders of the viewport remain parallel to the edges of the drawing area.

RSCRIPT

Repeats a script file

RSCRIPT is useful for demonstrations that repeat a script; for example, a script that must run over and over during a trade show or in a showroom.

If RSCRIPT is the last line in a script file, the file runs continuously until interrupted by ESC.

Note Consider turning off UNDO and any log files if you anticipate running the script over a long period; otherwise, these log files continue to grow and take up increasing amounts of disk space.

SAVE

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

⌨ **Command line: save**

The Save Drawing As dialog box (a standard file selection dialog box) is displayed. Save the drawing under the current file name, or enter a different file name to save a copy of the drawing under that name.

SAVE is available only from the command line. The Save option on the File menu or on the Standard toolbar is QSAVE. If the drawing is named, QSAVE saves the drawing without displaying the Save Drawing As dialog box. If the drawing is unnamed, the Save Drawing As dialog box is displayed. Enter a file name to both name and save the drawing.

SAVEAS

File menu: Save As

Command line: saveas

Sets options for DWG files and for drawing interchange file output.
The Saveas Options dialog box includes the following tabs:

- **DWG Options**
- **DXF Options**

DWG Options Tab (Saveas Options Dialog Box)

DWG Options Tab (Saveas Options Dialog Box)

Controls the drawing index and display of custom objects and specifies the default file format for saving drawings.

Save Proxy Images of Custom Objects

If you save to an earlier drawing file type, or the drawing contains custom objects from another application, you can select Save Proxy Images of Custom Objects to save images of the custom objects in the drawing file. If you do not choose this option, a frame is saved for each custom object in the drawing file. (PROXYGRAPHICS system variable)

Index Type

Determines whether layer or spatial indexes are created when you save a drawing. The indexes are used to improve performance during demand loading. Using indexes may slightly increase the time required to save a drawing. If a partially open drawing does not already contain spatial and layer indexes, this option is not available.

None

Creates neither layer nor spatial indexes when you save a drawing.

Layer

Loads only layers that are on and thawed.

Spatial

Loads only the portion of the drawing within a clipped boundary.

Layer & Spatial

Optimizes performance by specifying that the program loads only layers that are on, thawed, and within a clipped boundary area.

Save All Drawings As

Specifies the default file format that drawings are saved to. If you change the specified value, all subsequent uses of SAVE and QSAVE save the drawing to the new file format. You can also set this option on the Open and Save tab in the Options dialog box (see OPTIONS). Set the Save As option to AutoCAD 2004 Drawing to optimize performance while saving.

DXF Options Tab (Saveas Options Dialog Box)

DXF Options Tab (Saveas Options Dialog Box)

Sets drawing interchange file output options.

Format

Specifies whether the program creates an ASCII or a binary DXF file. ASCII-format DXF files can be read with a text editor and are compatible with a wider range of applications. Binary-format DXF files contain all of the information of an ASCII DXF file but in a more compact form. You can read and write to binary-format files faster than to ASCII-format files.

For more information about DXF files, see the *DXF Reference* in the Help system.

Select Objects

Controls whether the DXF file consists of selected objects or the entire drawing. When you select this option, the output file includes only selected objects and the block reference portions of any included blocks. The output file does not include the block definition tables.

Save Thumbnail Preview Image

Specifies whether an image of the drawing is displayed in the Preview area of the Select File dialog box. Save Thumbnail Preview Image is also controlled by the RASTERPREVIEW system variable.

Decimal Places of Accuracy

Saves the file using the specified number of bits of precision. The default precision is adequate in most cases; however, you might need to increase this value for certain drawings and for certain applications. If you encounter problems loading a DXF file you create, try increasing the precision. The only disadvantage of higher precision is increased file size. Lower precision is useful for specialized purposes but is not recommended in general.

SCALE

Enlarges or reduces selected objects proportionally in the *X*, *Y*, and *Z* directions

Modify toolbar:

Modify menu: Scale

Shortcut menu: Select the objects to scale, and right-click in the drawing area. Click Scale.

Command line: **scale**

Select objects: *Use an object selection method and press* ENTER *when you finish*

Specify base point: *Specify a point*

The base point you specify identifies the point that remains in the same location as the selected objects change size (and thus move away from the stationary base point).

Specify scale factor or [Copy/Reference]: *Specify a scale, enter* **c***, or enter* **r**

Scale Factor

Multiplies the dimensions of the selected objects by the specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects. You can also drag the cursor to make the object larger or smaller.

Copy

Creates a copy of the selected objects for scaling.

Reference

Scales the selected objects based on a reference length and a specified new length.

Specify reference length <1>: *Specify a beginning length from which to scale the selected objects*

Specify new length or [Points]: *Specify a final length to which to scale the selected objects, or enter* **p** *to define a length with two points.*

SCALETEXT

Enlarges or reduces selected text objects without changing their locations

Text toolbar: [icon]

Modify menu: Object ➤ Text ➤ Scale

Command line: scaletext

Select objects: *Use an object selection method, and press* ENTER *when you finish*

Enter a base point option for scaling
[Existing/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]<Existing>:
Specify a location to serve as a base point for scaling

With the base point prompt, you choose one of several locations to serve as base points for scaling, which is used individually for each selected text object. The base point for scaling is established on one of several insertion point locations for text options, but even though the options are the same as when you choose an insertion point, the justification of the text objects is not affected.

The base point options shown above are decribed in the TEXT command. The base point options for single line text are similar to those for multiline text except that the Align, Fit, and Left text options are equivalent to the bottom left (BL) multiline text attachment point.

Specify text height or [Match object/Scale factor]<0.5000>: *Specify a text height or enter an option*

Match Object

Match Object

Scales the text objects that you originally selected to match the size of a selected text object.

Select a text object with the desired height: *Select a text object to match*

Scale Factor

Scale Factor

Scales the selected text objects based on a reference length and a specified new length.

    Specify scale factor or [Reference]: *Specify a scale factor or enter* **r**

Scale Factor

  Scales the selected text objects using the numeric scale factor that you enter.

Reference

  Scales the selected text objects relative to a reference length and a new length.

   Specify reference length <1>: *Enter a length to serve as a reference distance*

   Specify new length: *Enter another length in comparison to the reference length*

  The selected text is scaled by a ratio of the values that you entered for the new length and the reference length. If the new length is less than the reference length, the selected text objects are reduced in size.

SCRIPT

Executes a sequence of commands from a script file

✖ **Tools menu:** Run Script

⌨ **Command line:** **script** (or **'script** for transparent use)

The Select Script File dialog box (a standard file selection dialog box) is displayed. Enter the file name of a script to run that script.

When FILEDIA is set to 0 (zero), SCRIPT displays the following prompt on the command line.

    Enter script file name *<current>*:

SELECT

Places selected objects in the Previous selection set

⌨ **Command line:** **select**

    Select objects: *Use an object selection method*

A small box, called the object selection target, replaces the crosshairs on the graphics cursor.

Objects must be selected in order to be processed. The Select Objects prompt occurs after many commands, including the SELECT command itself.

You can select objects individually with the pointing device, by drawing a selection window around them, by entering coordinates, or by using one of the selection methods listed below.

These methods can be used to select objects regardless of the command that initiated the Select Objects prompt. To view all options, enter **?** on the command line.

Expects a point or

Window/**Last**/**Crossing**/**BOX**/**ALL**/**Fence**/**WPolygon**/**CPolygon**/**Group**/**Add**/**Remove**/**Multiple**/**Previous**/**Undo** /**AUto**/**SIngle**

Select objects: *Specify a point or enter an option*

Window

Selects all objects completely inside a rectangle defined by two points. Specifying the corners from left to right creates a window selection. (Specifying the corners from right to left creates a crossing selection.)

Specify first corner: *Specify a point (1)*

Specify opposite corner: *Specify a point (2)*
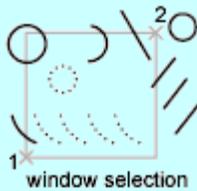
window selection

Last

Selects the most recently created visible object.

Crossing

Selects objects within and crossing an area defined by two points. A crossing selection is displayed as dashed or otherwise highlighted to differentiate it from window selection. Specifying the corners from right to left creates a crossing selection. (Specifying the corners from left to right creates a window selection.)

First corner: *Specify a point (1)*

Other corner: *Specify a point (2)*

crossing selection
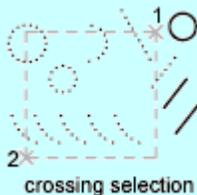
Box

Selects all objects inside or crossing a rectangle specified by two points. If the rectangle's points are specified from right to left, Box is equivalent to Crossing. Otherwise, Box is equivalent to Window.

Specify first corner: *Specify a point*

Specify opposite corner: *Specify a point*

All

Selects all objects on thawed layers.

select all
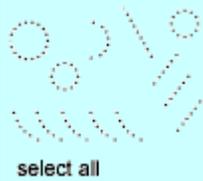
Fence

Selects all objects crossing a selection fence. The Fence method is similar to CPolygon except that that the fence is not closed, and a fence can cross itself. Fence is not affected by the PICKADD system variable.

First fence point: *Specify a point*

Specify endpoint of line or [Undo]: *Specify a point or enter **u** to undo the last point*

fence selection

WPolygon

Selects objects completely inside a polygon defined by points. The polygon can be any shape but cannot cross or touch itself. The last segment of the polygon is drawn so that it is closed at all times. WPolygon is not affected by the PICKADD system variable.

First polygon point: *Specify a point*

Specify endpoint of line or [Undo]: *Specify a point or enter **u** to undo the last point*

WPolygon selection

CPolygon

Selects objects within and crossing a polygon defined by specifying points. The polygon can be any shape but cannot cross or touch itself. The last segment of the polygon is drawn so that it is closed at all times. CPolygon is not affected by the PICKADD system variable.

First polygon point: *Specify a point*

Specify endpoint of line or [Undo]: *Specify a point or enter **u** to undo the last point*

CPolygon selection
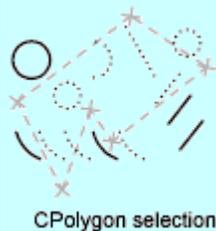
Group

Selects all objects within a specified group.

  Enter group name: *Enter a name list*

Add

Switches to the Add method: selected objects can be added to the selection set by using any of the object selection methods. Auto and Add are the default methods.

selected object to add
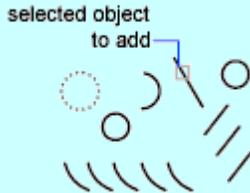
Remove

Switches to the Remove method: objects can be removed from the current selection set using any object selection method. An alternative to Remove mode is to hold down SHIFT while selecting single objects or use the Automatic option.

Multiple

Specifies multiple points without highlighting the objects, thus speeding up the selection process for complex objects. The Multiple method also selects two intersecting objects if the intersection point is specified twice.

Previous

Selects the most recent selection set. The Previous selection set is cleared by operations that delete objects from the drawing.

The program keeps track of whether each selection set was specified in model space or paper space. The Previous selection set is ignored if you switch spaces.

Undo

Cancels the selection of the object most recently added to the selection set.

Auto

Switches to automatic selection: pointing to an object selects the object. Pointing to a blank area inside or outside an object forms the first corner of a box defined by the Box method. Auto and Add are the default methods.

Single

Switches to the Single method: selects the first object or set of objects designated rather than continuing to prompt for further selections.

SETVAR

Lists or changes the values of system variables

Tools menu: Inquiry ➤ Set Variable

⌨ **Command line: setvar** (or **'setvar** for transparent use)

Enter **variable name** or [**?**] <*current*>: *Enter a variable name, enter* **?***, or press*ENTER

Variable Name
Specifies the name of the system variable you want to set. Enter new value for *variable_name* <*current*>: *Enter a new value or press*ENTER You can also change the value of system variables at the Command prompt by entering the name of the variable and its new value.

?—List Variables
Lists all system variables in the drawing and their current settings. Enter variable(s) to list <***>: *Enter a wild-card pattern or press* ENTER

SHAPE

Inserts a shape from a shape file that has been loaded using LOAD

⌨ **Command line:** shape

Enter **shape name** or [**?**]: *Enter a name, or enter* **?**

Shape Name
Loads the shape. Specify insertion point: Specify height <*current*>: *Specify a height or press*ENTER Specify rotation angle <0>: *Specify an angle or press*ENTER If a shape belongs to an external reference (xref) attached to the current drawing, the shape file is identified as externally dependent. Externally dependent shapes cannot be used in the current drawing unless they are reloaded.

?—List Shapes
Lists shapes and the files in which the shapes are defined. If you enter a question mark (**?**), the following prompt is displayed: Enter shape name(s) to list <***>: *Enter a name list or press*ENTER If you enter a name, the program lists the name of the file in which the shape definition exists and ends SHAPE. If you enter an asterisk (*****), the program lists shape names and ends SHAPE.

SKETCH

Creates a series of freehand line segments

⌨ **Command line:** sketch

The following information is based on the assumption that Tablet mode is on.

Record increment *<current>*: *Specify a distance or press* ENTER

The record increment value defines the length of the line segments. The pointing device must be moved a distance greater than the increment value to generate a line.

larger record increment   smaller record increment

Sketching is captured as a series of independent lines. Setting the SKPOLY system variable to a nonzero value produces a polyline for each contiguous sequence of sketched lines rather than multiple line objects.

Sketch. **PeneXitQuitRecordEraseConnect.** *Enter an option or press a pointer button*

Pen (Pick Button)

Raises and lowers the sketching pen. The pen must be raised before you can select menu items with the pointing device.

Exit—ENTER (button 3)

Records and reports the number of temporary lines sketched and ends the command.

Quit (button 4)

Discards all temporary lines sketched since the start of SKETCH or the last use of the Record option, and ends the command.

Record (button 2)

Records temporary lines as permanent and does not change the pen's position. Reports the number of lines using the following prompt:

nnn lines recorded.

Erase (button 5)

Erases any portion of a temporary line and raises the pen if it is down.

Select end of delete.

Connect (button 6)

Lowers the pen to continue a sketch sequence from the endpoint of the last sketched line or last Erase.

Connect: Move to endpoint of line.

. (Period) (button 1)

Lowers the pen, draws a straight line from the endpoint of the last sketched line to the pen's current location, and returns the pen to the up position.

SNAP

Restricts cursor movement to specified intervals

Status bar: Snap

Command line: snap (or **'snap** for transparent use)

Specify **snap spacing** or [**ON**/**OFF**/**Aspect**/**Rotate**/**Style**/**Type**] *<current>*: *Specify a distance, enter an option, or press*ENTER

Snap Spacing

Snap Spacing

Activates Snap mode with the value you specify.

On

On

Activates Snap mode using the current resolution, rotation, and style of the snap grid.
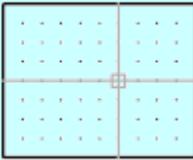
Off

Off

Turns off Snap mode but retains the current settings.

Aspect

Aspect

Specifies different spacing in the *X* and *Y* directions. This option is not available if the current snap style is Isometric.

Specify horizontal spacing *<current>*: *Specify a distance, or press*ENTER

Specify vertical spacing *<current>*: *Specify a distance, or press*ENTER
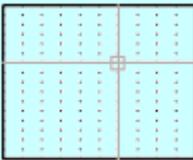
Rotate

Rotate

Sets the origin and rotation of the snap grid. The rotation angle is measured relative to the current UCS. You can specify a rotation angle between –90 and 90 degrees. A positive angle rotates the grid counterclockwise about its base point. A negative angle rotates the grid clockwise.

> Specify base point <current>: *Specify a point or press*ENTER
>
> Specify rotation angle <current>: *Specify an angle or press*ENTER

Style

Specifies the format of the snap grid, which is Standard or Isometric.

> Enter snap grid type [Standard/Isometric] <current>: *Enter* **s***, enter* **i***, or press*ENTER

Standard

Displays a rectangular grid that is parallel to the *XY* plane of the current UCS. *X* and *Y* spacing may differ.

> Specify snap spacing or [Aspect] <current>: *Specify a distance, enter* **a***, or press*ENTER

Spacing

Specifies the overall spacing of the snap grid.

Aspect

Specifies the horizontal and vertical spacing of the snap grid separately.

> Specify horizontal spacing <current>: *Specify a distance or press*ENTER
>
> Specify vertical spacing <current>: *Specify a distance or press*ENTER

Isometric

Sets an isometric grid, in which the grid points are initially at 30-degree and 150-degree angles. Isometric snap can be rotated but cannot have different Aspect values.

> Specify vertical spacing <current>: *Specify a distance or press*ENTER
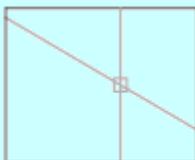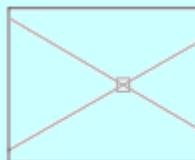
left isoplane   top isoplane   right isoplane

ISOPLANE determines whether the crosshairs lie in the top isometric plane (30- and 150-degree angles), the left isoplane (90- and 150-degree angles), or the right isoplane (30- and 90-degree angles).

Type

Specifies the snap type.

Polar

Sets the snap to polar tracking angles that are set in the POLARANG system variable.

Grid

Sets the snap to grid.

SPELL

Checks spelling in a drawing

Tools menu: Spelling

Command line: spell (or **'spell** for transparent use)

Select objects: *Use an object selection method and press* ENTER *when you finish*

If you enter **all** at the Select Objects prompt, the spelling in all text objects in the Model tab and all layout tabs is checked.
The Check Spelling dialog box is displayed if an unknown word is found in the selected text.

Corrects the spelling in text objects created with TEXT, MTEXT, LEADER, and ATTDEF. The Check Spelling dialog box is displayed only if an unknown word is found in the selected text.

Spell checking in attributes is limited to attribute values only. The spelling in text objects within block references and nested block references is checked, but spell checking in block definitions is performed only if the associated block reference has been selected.

Current Dictionary

Displays the name of the current dictionary.

Current Word

Displays the word being checked.

Suggestions

Displays a list of suggested replacement words from the current dictionary. Select a replacement in the list or enter a replacement word in the box.

Ignore

Skips the current word.

Ignore All

Skips all remaining words that match the current word.

Change

Replaces the current word with the word in the Suggestions box.

Change All

Replaces the current word in all selected text objects.

Add

Adds the current word to the current custom dictionary. The maximum word length is 63 characters.

Lookup

Lists words similar to the word selected in Suggestions.

Change Dictionaries

Displays the Change Dictionaries dialog box.

Context

Displays the phrase in which the current word was found.

SPLINE

Fits a smooth curve to a sequence of points within a specified tolerance

Draw toolbar:

Draw menu: Spline

Command line: spline

Specify **first point** or [**Object**]: *Specify a point or enter* **o**

First Point

First Point

Creates a spline using points you specify, using NURBS (nonuniform rational B-splines) mathematics.

　　Specify next point: *Specify a point*

Enter points until you have defined the spline curve. After you enter two points, the following prompt is displayed:

　　Specify next point or [Close/Fit Tolerance] <Start tangent>: *Specify a point, enter an option, or*
　　　*press*ENTER
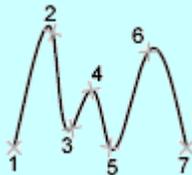
Next Point

Continuing to enter points adds additional spline curve segments until you press ENTER. Enter

undo to remove the last point specified. Once you press ENTER, you are prompted to specify

the start tangent for the spline curve.

Close

Closes the spline curve by defining the last point as co-incident with the first and making it tangent to the joint.

　Specify tangent: *Specify a point or press*ENTER

Specify a point to define the tangent vector or use the Tangent and Perpendicular object snap modes to make the spline tangent or perpendicular to existing objects.

closed spline
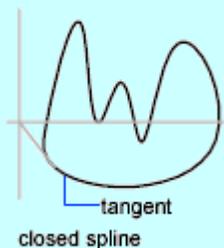
Fit Tolerance

Changes the tolerance for fitting of the current spline curve. The spline curve is redefined so that it fits through the existing points according to the new tolerance. You can repeatedly change the fit tolerance, but doing so changes the fit tolerance for all the control points regardless of the control point that is selected.

　Specify fit tolerance <*current*>: *Enter a value or press*ENTER

If you set the tolerance to 0, the spline curve passes through the fit points. Entering a tolerance greater than 0 allows the spline curve to pass through the fit points within the specified tolerance.

SPLINE returns to the previous prompt.
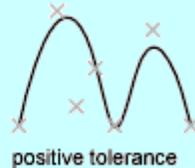
zero tolerance   positive tolerance

Start Tangent

Defines the tangency for the first and last points of the spline curve.

Specify start tangent: *Specify a point or press*ENTER

The Specify Start Tangent prompt specifies the tangency of the spline curve at the first point.

Specify end tangent:Specify a point or press ENTER
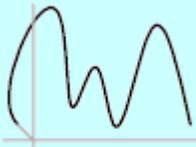
first tangent point

The Specify End Tangent prompt specifies the tangency of the spline curve at the last point.

last tangent point

If you specify tangency at both endpoints of the spline, you can enter a point or use the Tangent and Perpendicular object snap modes to make the spline tangent or perpendicular to existing objects. Press ENTER to calculate default tangents.

Object

Object

Converts 2D or 3D quadratic or cubic spline-fit polylines to equivalent splines and (depending on the setting of the DELOBJ system variable) deletes the polylines.

Select objects to convert to splines...

Select objects: *Select 2D or 3D spline-fit polylines and press* ENTER *when you finish*

SPLINEDIT

Edits a spline or spline-fit polyline

Modify II toolbar:

Modify menu: Object ➤ Spline

Shortcut menu: Select a spline to edit, right-click in the drawing area, and choose Spline Edit.

Command line: splinedit

Select spline:

Enter an option [**Fit data**/**Close**/**Move vertex**/**Refine**/**rEverse**/**Undo**]:

When you select a spline object or spline-fit polyline, grips appear at the control points. If the selected spline is closed, the Close option changes to Open. If the selected spline has no fit data, the Fit Data option is not available. Fit data consists of all fit points, the fit tolerance, and tangents associated with splines created with the SPLINE command. A spline can lose its fit data if you

- Use the Purge option while editing fit data
- Refine the spline by elevating the order, adding a control point, or changing the weight of a control point
- Change the fit tolerance
- Move a control point
- Trim, break, stretch, or lengthen the spline

Note SPLINEDIT automatically converts splined polylines to spline objects. A splined polyline is converted even if you select it and immediately exit SPLINEDIT.

Fit Data

Fit Data

Edits fit data using the following options:

Enter a fit data option

[Add/Close/Delete/Move/Purge/Tangents/toLerance/eXit] <eXit>: *Enter an option or*
*press*ENTER

Note If the selected spline is closed, the Close option is replaced by the Open option.

Add

Adds fit points to a spline.

Specify control point <exit>: *Specify a control point or press*ENTER

Specify new point <exit>: *Specify a point or press*ENTER

Specify new point <exit>: *Specify a point or press*ENTER

When you select a point, SPLINEDIT highlights it and the next point and interprets the new point to go between the highlighted points. Use the Undo option to remove the last point added. Selecting the last point on an open spline highlights only that point, and SPLINEDIT adds the new point after the last. If you select the first point on an open spline, you have the option of placing the new point before or after the first point.

Specify new point or [After/Before] <exit>: *Specify a point, enter an option, or press*ENTER

Specify new point <exit>: *Specify a point or press*ENTER

The point is added, and the spline is refitted through the new set of points.
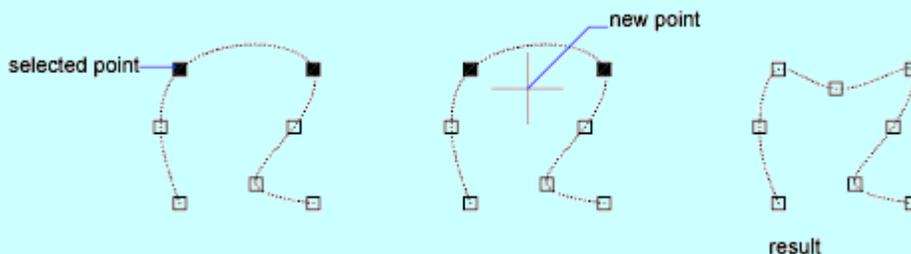
Close/Open

If the selected spline is closed, the Close option is replaced by the Open option.

Close

Closes an open spline and makes it tangent continuous (smooth) at its endpoints. If the spline's start point and endpoint are the same, this option makes the spline tangent continuous at both points.

Open

Opens a closed spline. If the spline's start point and endpoint were the same before you used the Close option to make it tangent continuous at both points, the Open option returns the spline to its original state. The start point and endpoint remain the same but lose their tangent continuity

(smoothness).

If the spline was open (its start point and endpoint were not the same) before you used the Close option to make it tangent continuous where the start point and endpoint meet, this option returns the spline to its original open state and removes tangent continuity.

Delete

Removes fit points from a spline and refits the spline through the remaining points.
    Specify control point <exit>: *Specify a control point or press*ENTER

Move

Moves fit points to a new location.
    Specify new location or [Next/Previous/Select Point/eXit] <N>: *Specify a point, enter an option, or press* ENTER

New Location

Moves the selected point to the new location you specify. The previous prompt is repeated.
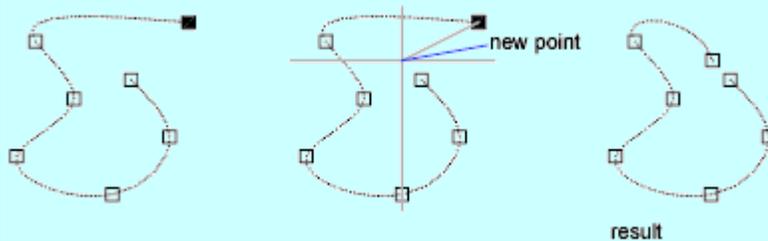
Next

Moves selection to the next point.

Previous

Moves selection to the previous point.

Select Point

Selects a point from the set of fit points.

    Specify fit point <exit>: *Specify a fit point or press* ENTER

Exit

Returns to the Enter a Fit Data Option prompt.

Purge

Removes a spline's fit data from the drawing database. After purging the spline's fit data, the main SPLINEDIT prompt is displayed without the Fit Data option.

Tangents

Edits the start and end tangents of a spline.

Specify start tangent or [System default]: *Specify a point, enter an option, or press* ENTER

Specify end tangent or [System default]: *Specify a point, enter an option, or press* ENTER

If the spline is closed, the prompt becomes Specify Tangent or [System Default].
The System Default option calculates the default tangents at the ends.
You can specify a point or use the Tangent or Perpendicular object snap mode to make the spline tangential or perpendicular to existing objects.

Tolerance

Refits the spline to the existing points with new tolerance values.

Enter fit tolerance *<current>*: *Enter a value or press*ENTER

zero tolerance

positive tolerance

Exit

Returns to the main SPLINEDIT prompt.

Close/Open

Close/Open

If the selected spline is closed, the Close option changes to Open.

Close

Closes an open spline and makes it tangent continuous (smooth) at its endpoints. If the spline's start point and endpoint are the same, this option makes the spline tangent continuous at both points.

Open

Opens a closed spline. If the spline's start point and endpoint were the same before you used the Close option to make it tangent continuous at both points, the Open option returns the spline to its original state. The start point and endpoint remain the same but lose their tangent continuity (smoothness).

If the spline was open (its start point and endpoint were not the same) before you used the Close option to make it tangent continuous where the start point and endpoint meet, this option returns the spline to its original open state and removes tangent continuity.

Move Vertex

Move Vertex

Relocates a spline's control vertices and purges the fit points.

Specify new location or [Next/Previous/Select Point/eXit] <N>: *Specify a point, enter an option, or press*ENTER

New Location

Moves the selected point to the new location you specify.

After you specify a new location for the selected point, SPLINEDIT recalculates and displays the spline with a new set of control points.

Next

Moves selection to the next point. The point marker does not wrap around from the end to the start of the spline, even if the spline is closed.

Previous

Moves selection to the previous point. The point marker does not wrap around from the start to the end of the spline, even if the spline is closed.

Select Point

Selects a point from the set of control points.

Specify fit point <exit>: *Specify a point or press*ENTER

Exit

Returns to the main SPLINEDIT prompt.

Refine

Refine

Fine-tunes a spline definition.

> Enter a refine option [Add control point/Elevate order/Weight/eXit] <eXit>: *Enter an option or press*ENTER

Add Control Point

Increases the number of control points that control a portion of a spline.

> Specify a point on the spline <exit>: *Specify a point or press*ENTER

SPLINEDIT adds a new control point close to the point you select, between the two control points affecting that portion of the spline.
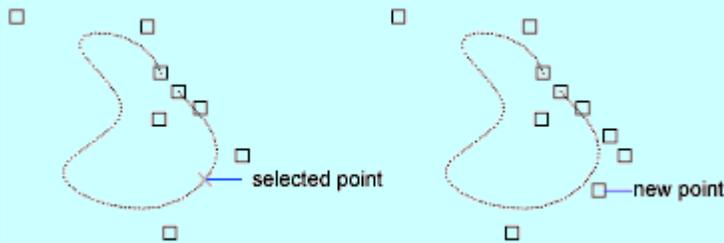
Elevate Order

Increases the number of control points on the spline.

> Enter new order <current>: *Enter an integer or press*ENTER

Entering a value greater than the current value increases the number of control points uniformly across the spline for more localized control. The maximum value for order is 26.

Weight

Changes the weight at various spline control points. A larger weight value pulls the spline closer to the control point.

> Enter new weight (current = *current*) or [Next/Previous/Select point/eXit] <N>: *Enter a value, enter an option, or press*ENTER

New Weight

Recalculates the spline based on the new weight value at the selected control point. The larger the integer value, the more the spline is pulled toward the control point.

Next

Moves selection to the next point. The point marker does not wrap around from the end to the start of the spline, even if the spline is closed.

Previous

Moves selection to the previous point. The point marker does not wrap around from the start to the end of the spline, even if the spline is closed.

Select Point

Selects a point from the set of control points.

> Specify fit point <exit>: *Specify a point or press*ENTER

Exit

Returns to the main Refine prompt.

Exit

Returns to the main SPLINEDIT prompt.

Reverse

Reverse

Reverses the spline's direction. This option is intended primarily for third-party applications.

Undo

Undo

Cancels the last editing action.

STATUS

Displays drawing statistics, modes, and extents

Tools menu: Inquiry ➤ Status

Command line: status (or **'status** for transparent use)

All coordinates and distances are displayed by STATUS in the format specified by UNITS. STATUS reports the number of objects in the current drawing. This includes graphical objects such as arcs and polylines, and nongraphical objects such as layers and linetypes, and block definitions. When used at the DIM prompt, STATUS reports the values and descriptions of all dimensioning system variables.

In addition, STATUS displays the following information.

Model or Paper Space Limits Are

Displays the grid limits defined by LIMITS. The first line shows the *XY* coordinate of the limit's lower-left corner, stored in the LIMMIN system variable. The second line shows the *XY* coordinate of the limit's upper-right corner, stored in the LIMMAX system variable. The notation Off to the right of the *Y* coordinate value indicates that limits checking is set to 0.

Model or Paper Space Uses

Displays the drawing extents, which includes all objects in the database and can exceed the grid limits. The first line shows the *XY* coordinate of the lower-left corner of the extents. The second line shows the *XY* coordinate of the upper-right corner. The notation Over to the right of the *Y* coordinate value indicates that the drawing extends outside the grid limits.

Display Shows

Lists the portion of the drawing extents visible in the current viewport. The first line shows the *XY* coordinate of the display's lower-left corner. The second line shows the *XY* coordinate of the upper-right corner.

Insertion Base Is

Displays the insertion point of the drawing, stored in the INSBASE system variable and expressed as an *X,Y,Z* coordinate.

Snap Resolution Is

Displays the snap spacing in the *X* and *Y* directions, stored in the SNAPUNIT system variable.

Grid Spacing Is

Displays the grid spacing in the *X* and *Y* directions, stored in the GRIDUNIT system variable.

Current Space

Shows whether model space or paper space is active.

Current Layout

Displays "Model" or the name of the current layout.

Current Layer

Lists the current layer of the drawing, stored by the CLAYER system variable.

Current Color

Lists the current color, stored by the CECOLOR system variable.

Current Linetype

Lists the current linetype, stored by the CELTYPE system variable.

Current Lineweight

Lists the current lineweight, stored by the CELWEIGHT system variable.

Current Plot Style

Lists the current plot style, stored by the CPLOTSTYLE system variable.

Current Elevation

Lists the current elevation of the drawing, stored in the ELEVATION system variable.

Thickness

Lists the current thickness, stored in the THICKNESS system variable.

Fill, Grid, Ortho, Qtext, Snap, Tablet

Shows whether these modes are on or off.

Object Snap Modes

Lists the running object snap modes specified by OSNAP.

Free Dwg Disk Space

Lists the amount of disk space available on the drive specified for this program's temporary files.

Free Temp Disk Space

Lists the amount of disk space available on the drive specified for temporary files.

Free Physical Memory

Lists the amount of installed memory free on your system.

Free Swap File Space

Lists the amount of free space in your swap file.

STRETCH

Moves or stretches objects

⊗ **Modify toolbar:**

⊗ **Modify menu:** Stretch

⌨ **Command line:** stretch

Select objects to stretch by crossing-window or crossing-polygon...

Select objects: *Use the cpolygon option or the crossing object selection method, and press ENTER. Individually selected objects and objects that are completely enclosed by the crossing selection are moved rather than stretched.*
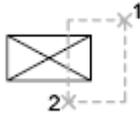
if
STRETCH moves only the vertices and endpoints that lie inside the crossing selection, leaving those outside unchanged. STRETCH does not modify 3D solids, polyline width, tangent, or curve-fitting information.

Base Point

Specify base point or [Displacement] *<last displacement>*: *Specify a base point or enter displacement coordinates*

Specify second point or *<use first point as displacement>*: *Specify a second point, or press*ENTER *to use the previous coordinates as a displacement*

Displacement

Specify displacement *<last value>*: *Enter displacement values for X,Y (and optionally Z)*
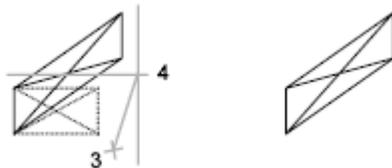
If you enter a second point, the objects are stretched the vector distance from the base point to the second point. If you press ENTER at the Specify Second Point of Displacement prompt, the first point is treated as an *X,Y,Z* displacement.

STYLE

⊗ **Format menu:** Text Style

⌨ **Command line:** style (or 'style for transparent use)

Creates, modifies, or sets named text styles.

Style Name

Displays text style names, adds new styles, and renames and deletes existing styles. The list contains defined style names and displays the current style by default. To change the current style, select another style from the list, or choose New to create a new style.
Style names can be up to 255 characters long. They can contain letters, numbers, and the special characters dollar sign ($), underscore (_), and hyphen (-).

New

Displays the New Text Style dialog box and automatically supplies the name "style*n*" (where *n* is the number of the supplied style) for the current settings. You can accept the default or enter a name and choose OK to apply the current style settings to the new style name.

Rename

Displays the Rename Text Style dialog box. The text style listed is renamed when you enter a new name and choose OK.

You can also use RENAME to change the name of an existing text style. If any existing text object uses the old style name, the text object takes the new name automatically.

Delete

Deletes a text style. Select a name from the list to make it current, and then choose Delete.

Font

Changes the style's font.

Note If you change the orientation or font file of an existing text style, all text objects with that style use the new values when the drawing is regenerated.

Font Name

Lists the font family name for all registered TrueType fonts and all compiled shape (SHX) fonts in the *Fonts* folder. When you select a name from the list, the program reads the file for the specified font. The file's character definitions are loaded automatically unless the file is already in use by another text style. You can define several styles that use the same font.

Font Style

Specifies font character formatting, such as italic, bold, or regular. When Use Big Font is selected, this option changes to Big Font Name and is used to select a Big Font file name.

Height

Sets the text height based on the value you enter. If you enter 0.0, you are prompted for the

text height each time you enter text using this style. Entering a height greater than 0.0 sets

the text height for this style. TrueType fonts can be displayed at a smaller height than SHX fonts with the same height setting.

Note The text height you specify may not be accurately represented by uppercase letters in TrueType fonts supplied with this program.

Use Big Font

Specifies an Asian-language Big Font file. Use Big Font is available only if you specify an SHX file under Font Name. Only SHX files are valid file types for creating Big Fonts.

Effects

Modifies characteristics of the font, such as its height, width factor, and obliquing angle and whether it is displayed upside down, backwards, or vertically aligned.

Upside Down

Displays the characters upside down.

Backwards

Displays the characters backwards.

Vertical

Displays the characters aligned vertically. Vertical is available only if the selected font supports dual orientation. Vertical orientation is not available for TrueType fonts.

Width Factor

Sets the character spacing. Entering a value less than 1.0 condenses the text. Entering a

value greater than 1.0 expands it.

Oblique Angle

Sets the obliquing angle of the text. Entering a value between -85 and 85 italicizes the text.

Note TrueType fonts using the effects described in this section might appear bold on the screen. Onscreen appearance has no effect on plotted output. Fonts are plotted as specified by applied character formatting.

Preview

Displays sample text that changes dynamically as you change fonts and modify the effects. To change the sample text, enter characters in the box below the character preview image.

Preview Text

Provides text to be displayed in the preview image.

Preview Button

Updates the sample text in the character preview image according to any changes you've made in the dialog box.

Note The preview image does not reflect the text height.

Apply

Applies style changes made in the dialog box to the text of the current style in the drawing.

Close

Applies changes to the current style. Cancel changes to Close whenever a change is made to any of the options under Style Name. Changing, renaming, or deleting the current style and creating a new style are actions that take place immediately and cannot be canceled.

Cancel

Cancel changes to Close whenever a change is made to any of the options under Style Name.

STYLESMANAGER

Command line: stylesmanager

The Plot Style Manager is displayed.
In the Plot Style Manager, you can double-click the Add-a-Plot-Style-Table wizard to add plot style tables. Double-click a plot style table (STB or CTB file) to start the Plot Style Table Editor.

Adds new plot style tables. Plot style tables contain and define plot styles, which can be assigned to objects. When complete, the wizard produces an STB or CTB file depending on the type of table you are creating. You can edit these files in the Plot Style Table Editor. To use your new plot style table, the selected options on the Plot and Publish tab of the Options dialog box must be appropriate for the type of plot style table (named or color-dependent) you created.

You can create a plot style table from scratch, use an existing plot style table as a starting point, use settings from your AutoCAD Release 14 CFG file, or use settings from a PCP or PC2 file.

You can create a plot style table from scratch, use an existing plot style table as a starting point, or use settings from a PCP, PC2, or CFG file.

You can create either a named plot style table or a color-dependent plot style table. With a named plot style table, you can add and define plot styles as you like; the file name has the extension *.stb*. A color-dependent plot style table creates 255 plot styles based on color; the file name has the extension *.ctb*. You can specify whether you want to use a plot style table for new drawings or for pre-AutoCAD 2000 drawings when they are saved in a later format.

Modifies the plot styles in a plot style table. If the plot style table is attached to a layout or the Model tab, and you change a plot style, any objects that use that plot style are affected. If the plot style table is color-dependent, the file extension is CTB. If the plot style table is named, the file extension is STB.

Open the Plot Style Table Editor with any of the following methods:
- Double-click a CTB or STB file in the Plot Style Manager.
- Right-click a CTB or STB file in the Plot Style Manager, and then choose Open from the shortcut menu.
- Choose Plot Style Table Editor from the Finish screen in the Add Plot Style Table wizard.
- In the Page Setup dialog box under Plot Style Table (Pen Assignments), select a plot style and click the Edit button.
- In the Current Plot Style and Select Plot Style dialog boxes, choose Editor.

The Plot Style Table Editor includes the following tabs:
- General
- Table View and Form View

General

Lists the plot style table file name, description, version number, location (path name), and table type. You can modify the description, and you can apply scaling to non-ISO linetypes and fill patterns.

Plot Style Table File Name

Displays the name of the plot style table file you are editing.

Description

Provides a description area for a plot style table.

File Information

Displays information about the plot style table you are editing: number of plot styles, path, and version number of the Plot Style Table Editor.

Apply Global Scale Factor to Non-ISO Linetypes

Scales all the non-ISO linetypes and fill patterns in the plot styles of objects controlled by this plot style table.

Scale Factor

Specifies the amount to scale non-ISO linetypes and fill patterns.

Delete R14 Color Mapping Table

Deletes color mapping tables that are used when you open pre-AutoCAD 2000 drawings. Named plot style tables that you create using *acadr14.cfg*, PCP, or PC2 files contain plot styles that are created from your AutoCAD Release 14 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. While the color mapping table exists, you cannot add, delete, or rename plot styles in that plot style table.

If you delete a color mapping table, plot styles cannot be automatically assigned to objects when pre-AutoCAD 2000 drawings are opened for the first time.

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.

Table View and Form View

Lists all of the plot styles in the plot style table and their settings. Plot styles are displayed in columns from left to right. You can use either the Table View tab or the Form View tab to adjust 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 is more convenient because the plot style names are listed at the left and the properties of the selected style are displayed to the right. You don't have to scroll horizontally to view the style and its properties. The first plot style in a named plot style table is NORMAL and represents an object's default properties (no plot style applied). You cannot modify or delete the NORMAL style.

Name

Displays the names of plot styles in named plot style tables. Plot styles in named plot style tables can be changed. Plot style names in color-dependent plot style tables are tied to object color and cannot be changed. The program accepts up to 255 characters for style names. You cannot have duplicate names within the same plot style table.

Description

Provides a description for each plot style.

Properties

Specifies the settings for the new plot style you are adding to the current plot style table.

Color

Specifies the plotted color for an object. The default setting for plot style color is Use Object Color. If you assign a plot style color, the color overrides the object's color at plot time.

You can choose Select Color to display the Select Color dialog box and select one of the 255 AutoCAD Color Index (ACI) colors, a true color, or a color from a color book. The color you specify is displayed in the plot style color list as Custom Color. If the plot device does not support the color you specify, it plots the nearest available color or, in the case of monochrome devices, black.

Enable Dithering

Enables dithering. A plotter uses dithering to approximate colors with dot patterns, giving the impression of plotting more colors than available in the AutoCAD Color Index (ACI). If the plotter does not support dithering, the dithering setting is ignored.

Dithering is usually turned off in order to avoid false line typing that results from dithering of thin vectors. Turning off dithering also makes dim colors more visible. When you turn off dithering, the program maps colors to the nearest color, resulting in a smaller range of colors when plotting. Dithering is available whether you use the object's color or assign a plot style color.

Convert to Grayscale

Converts the object's colors to grayscale if the plotter supports grayscale. If you clear Convert to Grayscale, the RGB values are used for object colors. Dithering is available whether you use the object's color or assign a plot style color.

Use Assigned Pen Number (Pen Plotters Only)

Specifies a pen to use when plotting objects that use this plot style. Available pens range from 1 to 32. If plot style color is set to Use Object Color, or you are editing a plot style in a color-dependent plot style table, you cannot change the assigned pen number; the value is set to Automatic.

If you specify 0, the field updates to read Automatic. The program determines the pen of the closest color to the object you are plotting using the information you provided under Physical Pen Characteristics in the Plotter Configuration Editor.

Virtual Pen Number

Specifies a virtual pen number between 1 and 255. Many non-pen plotters can simulate pen plotters using virtual pens. For many devices, you can program the pen's width, fill pattern, end style, join style, and color/screening from the front panel on the plotter.

Enter 0 or Automatic to specify that the program should make the virtual pen assignment from the AutoCAD Color Index.

The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens. If this is the case, all the other style settings are ignored and only the virtual pen is used. If a non-pen plotter is not configured for virtual pens, then the virtual and physical pen information in the plot style is ignored and all the other settings are used.

You can configure your non-pen plotter for virtual pens under Vector Graphics on the Device and Document Settings tab in the PC3 Editor. Under Color Depth, select 255 Virtual Pens.

Screening

Specifies 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. In order for screening to work, the Enable Dithering option must be selected.

Linetype

Displays a list with a sample and a description of each linetype. The default setting for plot style linetype is Use Object Linetype. If you assign a plot style linetype, the linetype overrides the object's linetype at plot time.

Adaptive Adjustment

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.

Lineweight

Displays a sample of the lineweight as well as its numeric value. You can specify the numeric value of each lineweight in millimeters. The default setting for plot style lineweight is Use Object Lineweight. If you assign a plot style lineweight, the lineweight overrides the object's lineweight at plot time.

Line End Style

Provides the following line end styles: Butt, Square, Round, and Diamond. The default setting for Line End Style is Use Object End Style. If you assign a line end style, the line end style overrides the object's line end style at plot time.

Line Join Style

Provides the following line join styles: Miter, Bevel, Round, and Diamond. The default setting for Line Join Style is Use Object Join Style. If you assign a line join style, the line join style overrides the object's line join style at plot time.

Fill Style

Provides the following fill styles: Solid, Checkerboard, Crosshatch, Diamonds, Horizontal Bars, Slant Left, Slant Right, Square Dots, and Vertical Bar. The default setting for Fill Style is Use Object Fill Style. If you assign a fill style, the fill style overrides the object's fill style at plot time.

Add Style

Adds a new plot style to a named plot style table.
The plot style is based on Normal, which uses an object's properties and doesn't apply any overrides by default. You must specify the overrides you want to apply after you create the new plot style. You cannot add a new plot style to a color-dependent plot style table; a color-dependent plot style table has 255 plot styles mapped to color. You also cannot add a plot style to a named plot style table that has a translation table.

Delete Style

Deletes the selected style from the plot style table.
Objects assigned this plot style retain the plot style assignment but plot as Normal because the plot style is no longer defined in the plot style table. You cannot delete a plot style from a named plot style table that has a translation table, or from a color-dependent plot style table.

Edit Lineweights

Displays the Edit Lineweights dialog box. There are 28 lineweights available to apply to plot styles in plot style tables. If the lineweight you need doesn't exist in the list of lineweights stored in the plot style table, you can edit an existing lineweight. You can't add or delete lineweights from the list in the plot style table.

Save As

Displays the Save As dialog box and saves the plot style table to a new name.

SYSWINDOWS

Arranges windows and icons when the application window is shared with external applications

Command line: syswindows

Enter an option [**Cascade**/**tile Horizontal**/**tile Vertical**/**Arrange icons**]: *Enter an option*

Cascade
Cascade
Overlaps windows, leaving title bars visible.
Tile Horizontal
Tile Horizontal
Arranges windows in horizontal, nonoverlapping tiles.
Tile Vertical
Tile Vertical
Arranges windows in vertical, nonoverlapping tiles.
Arrange Icons
Arrange Icons
Arranges the window icons.

TEXT

Creates a single-line text object

⬙ **Draw menu:** Text ➤ Single Line Text

⌨ **Command line: text**

Current text style: *current* Current text height: *current*

Specify **start point** of text or [**Justify**/**Style**]: *Specify a point or enter an option*

If TEXT was the last command entered, pressing ENTER at the Specify Start Point of Text prompt skips the prompts for height and rotation angle. The text that you enter in the In-Place Text Editor for single-line text is placed directly beneath the previous line of text. The point that you specified at the prompt is also stored as the insertion point of the text.

The In-Place Text Editor for single-line text consists of a bounding box that is the height of the text and expands as you type. Right-click to select options on the shortcut menu.

Each line of text is a separate object. To end a line and begin another, press ENTER. To exit TEXT, press ENTER twice.

You can enter special characters and format text by entering Unicode strings and control codes.

Use -TEXT to honor the TEXTEVAL system variable.

Start Point

Start Point

Specifies a start point for the text object.

Specify height <*current*>: *Specify a point (1), enter a value, or press* ENTER

The Specify Height prompt is displayed only if the current text style does not have a fixed height.

Specify rotation angle of text <*current*>: *Specify an angle or press* ENTER

Enter text in the In-Place Text Editor for single-line text.
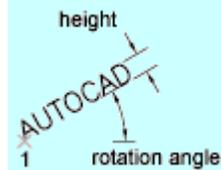
Justify

Justify

Controls justification of the text.

    Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:

You can also enter any of these options at the Specify Start Point of Text prompt.

Align

  Specifies both text height and text orientation by designating the endpoints of the baseline.

    Specify first endpoint of text baseline: *Specify a point (1)*

    Specify second endpoint of text baseline: *Specify a point (2)*

  Enter text in the In-Place Text Editor for single-line text.

  The size of the characters adjusts in proportion to their height. The longer the text string, the shorter the characters.
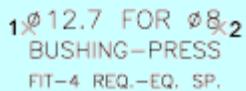
Fit

  Specifies that text fits within an area and at an orientation defined with two points and a height. Available for horizontally oriented text only.

    Specify first endpoint of text baseline: *Specify a point (1)*

    Specify second endpoint of text baseline: *Specify a point (2)*

    Specify height *<current>*:

  Enter text in the In-Place Text Editor for single-line text.
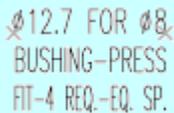
  The height is the distance in drawing units that the uppercase letters extend from the baseline. Designated text height is the distance between the start point and a point you specify. The longer the text string, the narrower the characters. The height of the characters remains constant.

Center

  Aligns text from the horizontal center of the baseline, which you specify with a point.

    Specify center point of text: *Specify a point (1)*

    Specify height *<current>*:

    Specify rotation angle of text *<current>*:

  Enter text in the In-Place Text Editor for single-line text.

  The rotation angle specifies the orientation of the text baseline with respect to the center point. You can designate the angle by specifying a point. The text baseline runs from the start point toward the specified point. If you specify a point to the left of the center point, the text is drawn upside down.

Middle

  Aligns text at the horizontal center of the baseline and the vertical center of the height you

specify. Middle-aligned text does not rest on the baseline.

Specify middle point of text: *Specify a point (1)*
Specify height *<current>*:
Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

The Middle option differs from the MC option in that it uses the midpoint of all text, including descenders. The MC option uses the midpoint of the height of uppercase letters.

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Right

Right-justifies the text at the baseline, which you specify with a point.

Specify right endpoint of text baseline: *Specify a point (1)*
Specify height *<current>*:
Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

AUTOCAD
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TL (Top Left)

Left-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-left point of text: *Specify a point (1)*
Specify height *<current>*:
Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

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TC (Top Center)

Centers text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-center point of text: *Specify a point (1)*
Specify height *<current>*:
Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

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TR (Top Right)

Right-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-right point of text: *Specify a point (1)*
Specify height *<current>*:
Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

AUTOCAD¹

ML (Middle Left)

Left-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.

Specify middle-left point of text: *Specify a point (1)*

Specify height *<current>*:

Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

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MC (Middle Center)

Centers the text both horizontally and vertically at the middle of the text. Available for horizontally oriented text only.

Specify middle-center point of text: *Specify a point (1)*

Specify height of text *<current>*:

Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

The MC option differs from the Middle option in that it uses the midpoint of the height of uppercase letters. The Middle option uses the midpoint of all text, including descenders.

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MR (Middle Right)

Right-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.

Specify middle-right point of text: *Specify a point (1)*

Specify height *<current>*:

Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

AUTOCAD⊲1

BL (Bottom Left)

Left-justifies text at a point specified for the baseline. Available for horizontally oriented text only.

Specify bottom-left point of text: *Specify a point (1)*

Specify height *<current>*:

Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

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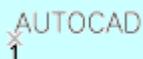
BC (Bottom Center)

Centers text at a point specified for the baseline. Available for horizontally oriented text only.

   Specify bottom-center point of text: *Specify a point (1)*
   Specify height *<current>*:
   Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

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BR (Bottom Right)

Right-justifies text at a point specified for the baseline. Available for horizontally oriented text only.

   Specify bottom-right point of text: *Specify a point (1)*
   Specify height *<current>*:
   Specify rotation angle of text *<current>*:

Enter text in the In-Place Text Editor for single-line text.

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Style

Style

Specifies the text style, which determines the appearance of the text characters. Text you create uses the current text style.
   Enter style name or [?] *<current>*: *Enter a text style name or enter* **?** *to list all text styles*

Entering **?** lists the current text styles, associated font files, height, and other parameters.

AUTOCAD
AUTOCAD
AUTOCAD
various styles

TEXTSCR

Opens the text window
View menu: Display ⊁ Text Window

Command line: textscr (or **'textscr** for transparent use)

TEXTSCR displays the command line in a separate window. You can press F2 to toggle between the drawing area and the text window. This command is ignored on dual-screen systems.
When the command line is hidden, you can turn it back on by entering **commandline** in the text window.

TIME

Tools menu: Inquiry ➤ Time

Command line: time (or **'time** for transparent use)

TIME displays the following information:

<u>Current time</u>: Wednesday, December 31, 2003 at 9:54:51:406 AM

Times for this drawing:

<u>Created</u>: Friday, December 12, 2003 at 1:21:36:203 AM

<u>Last Updated</u>: Wednesday, December 31, 2003 at 9:49:19:208 AM

<u>Total Editing Time</u>: 0 days 06:44:10.520

<u>Elapsed Timer</u> (on): 0 days 00:07:05.312

<u>Next Automatic Save In</u>: 0 days 01:59:15.570

Enter option [<u>Display</u>/<u>On</u>/<u>OFF</u>/<u>Reset</u>]: *Enter an option or press* ENTER

TOLERANCE

Dimension toolbar:

Dimension menu: Tolerance

Command line: tolerance

The Geometric Tolerance dialog box is displayed.

Specifies the symbols and values for a feature control frame.

After you select geometric characteristic symbols, the Geometric Tolerance dialog box closes and the following prompt is displayed:

Enter tolerance location: *Specify a location*

The feature control frame is placed at the specified location.

Sym

Displays the geometric characteristic symbol, which you select from the Symbol dialog box. The dialog box is displayed when you select one of the Sym boxes.

Tolerance 1

Creates the first tolerance value in the feature control frame. The tolerance value indicates the amount by which the geometric characteristic can deviate from a perfect form. You can insert a diameter symbol before the tolerance value and a material condition symbol after it.
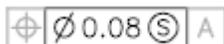
First Box

Inserts a diameter symbol in front of the tolerance value. Click the box to insert the diameter symbol.

Second Box

Creates the tolerance value. Enter a value in the box.

Third Box

Displays the Material Condition dialog box, in which you select a modifying symbol. These symbols act as modifiers to the geometric characteristic and the tolerance value of features that can vary in size.

Select the symbol you want to use. The dialog box closes. The symbol is inserted into the MC box for the first tolerance value in the Geometric Tolerance dialog box.

Tolerance 2

Creates the second tolerance value in the feature control frame. Specify the second tolerance value in the same way as the first.

Datum 1

Creates the primary datum reference in the feature control frame. The datum reference can consist of a value and a modifying symbol. A datum is a theoretically exact geometric reference used to establish the tolerance zone for a feature.

First Box

Creates the datum reference value. Enter a value in the box.

Second Box

Displays the Material Condition dialog box, in which you select a modifying symbol. These symbols act as modifiers to the datum reference.

Select the symbol you want to use. The dialog box closes. The symbol is inserted into the MC box for the primary datum reference in the Geometric Tolerance dialog box.

Datum 2

Creates the secondary datum reference in the feature control frame in the same way as the primary datum reference.

Datum 3

Creates the tertiary datum reference in the feature control frame in the same way as the primary datum reference.

Height

Creates a projected tolerance zone value in the feature control frame. A projected tolerance zone controls the variation in height of the extended portion of a fixed perpendicular part and refines the tolerance to that specified by positional tolerances. Enter a value in the box.
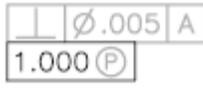
Projected Tolerance Zone

Inserts a projected tolerance zone symbol after the projected tolerance zone value.
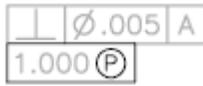
Datum Identifier

Creates a datum-identifying symbol consisting of a reference letter. A datum is a theoretically exact geometric reference from which you can establish the location and tolerance zones of other features. A point, line, plane, cylinder, or other geometry can serve as a datum. Enter the letter in the box.

TOOLBAR

Displays, hides, and customizes toolbars

View menu: Toolbars

Shortcut menu: Right-click any toolbar and choose Customize.

Command line: toolbar

The Customize User Interface dialog box (see CUI) is displayed.

If you enter **-toolbar** at the Command prompt, TOOLBAR displays prompts on the command line.

TOOLPALETTES

Opens the Tool Palettes window

Standard toolbar:

Tools menu: Tool Palettes Window

Command line: toolpalettes

The Tool Palettes window opens. You can click the Properties button or right-click the title bar, the current tool palette, the tab on a tool palette, or a tool to display a shortcut menu. The options displayed on each shortcut menu are appropriate to the cursor location. All the options on these shortcut menus are listed alphabetically and described below.

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

Allow Docking

Controls whether the tool palette docks when you drag it over a docking area at the side of the drawing. Clear this option to undock a docked tool palette.

Auto-hide

Controls display of the tool palette. 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 full tool palette is displayed when the cursor moves.

Close

Closes the tool palette window.

Copy

Copies the selected tool to the Clipboard.

Customize

Displays the Customize dialog box.

Cut

Removes the selected tool from the tool palette and places it on the Clipboard.

Delete Tool

Removes the selected tool from the tool palette.

Delete Tool Palette

Removes the current tool palette.

Move

Changes the cursor to a four-direction arrow.

Move Up

Moves the selected tool palette up one position.

Move Down

Moves the selected tool palette down one position.

New Tool Palette

Creates a new tool palette. Enter a name or press ENTER to use the default name.

Paste

Pastes a tool from the Clipboard to the current tab.

Properties

Displays the Tool Properties dialog box, where you can change the properties of the selected tool.

Rename

Renames the selected tool.

Rename Tool Palette

Renames the current tool palette.

Size

Changes the cursor to a four-direction arrow. Drag the right edge or the bottom edge of the tool palette to change its size.

Transparency

Displays the Transparency dialog box.

View Options

Displays the View Options dialog box, where you can control how tools are displayed.

TOOLPALETTESCLOSE

Closes the Tool Palettes window

⬙ **Tools menu:** Tool Palettes Window

⌨ **Command line:** **toolpalettesclose**

Closes the Tool Palettes window.

TRACE

Creates solid lines

⌨ **Command line:** **trace**

> Specify trace width *<current>*: *Specify a distance or press* ENTER
> Specify start point: *Specify a point (1)*
> Specify next point: *Specify a point (2)*
> Specify next point: *Specify a point (3) or press* ENTER *to end the command*

The endpoints of a trace are on the center line and are always cut square. TRACE automatically calculates the correct bevels for connection to adjacent segments. Each segment is drawn after you either specify the next segment or press ENTER. Because of the way bevels are handled, TRACE has no undo option.
Traces are solid filled when Fill mode is on. When Fill mode is off, only the outline of a trace is displayed.

TRANSPARENCY

Controls whether background pixels in an image are transparent or opaque

⬙ **Reference toolbar:**

⬙ **Modify menu:** Object ➤ Image ➤ Transparency

Shortcut menu: Select an image, right-click in the drawing area, and click Image ➤ Transparency.

⌨ **Command line:** **transparency**

Select image(s):

Enter transparency mode [ON/OFF] <*current*>: *Enter an option or press* ENTER

On

Turns transparency on so that objects beneath the image are visible.

Off

Turns transparency off so that objects beneath the image are not visible.

Note The PALETTEOPAQUE system variable controls whether dockable windows can be made transparent.

TREESTAT

Displays information about the drawing's current spatial index

Command line: treestat (or **'treestat** for transparent use)

The program indexes objects in a region by recording their positions in space. The result is called a *spatial index*. The spatial index is tree structured and has branching nodes to which objects are attached. The index has two major branches. The paper space branch is called a quad-tree and treats objects as two-dimensional. The model space branch is called an oct-tree and treats objects as either two- or three-dimensional. The model space branch can also be changed to a quad-tree when you are working on two-dimensional drawings.

TREESTAT displays information about each branch. The most important information is in the first two lines of the report—number of nodes, number of objects, maximum depth of the branch, and average number of objects per node.

If REDRAW and object selection are very slow, you can improve their performance. For example, if there are 50 megabytes of memory available and the current drawing has 50,000 objects with only 1,000 nodes in the index tree, increase the TREEDEPTH value to improve performance.

Each node consumes about 80 bytes of memory. The fewer objects per node of the oct-tree, the better the performance.

TRIM

Trims objects at a cutting edge defined by other objects

Modify toolbar:

Modify menu: Trim

Command line: trim

Current settings: Projection = *current* Edge = *current*

Select cutting edges...

Select objects or <select all>: *Select one or more objects and press* ENTER, *or press* ENTER *to select all displayed objects*

selected cutting edge
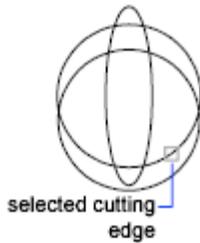
Select the objects that define the cutting edges to which you want to trim an object, or press ENTER to select all displayed objects as potential cutting edges. TRIM projects the cutting edges and the objects to be trimmed onto the *XY* plane of the current user coordinate system (UCS).

Note To select cutting edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

Select **object to trim** or **shift-select to extend** or [**Fence**/**Crossing**/**Project**/**Edge**/**eRase**/**Undo**]: *Select an object to trim, hold down* SHIFT *and select an object to extend it instead, or enter an option*

Specify an object selection method to select the objects to trim. If more than one trim result is possible, the location of the first selection point determines the result.

Object to Trim

Object to Trim

Specifies the object to trim. The prompt for selecting the object to trim repeats so you can trim multiple objects. Press ENTER to exit the command.

objects to trim

Shift-Select to Extend

Shift-Select to Extend

Extends the selected objects rather than trimming them. This option provides an easy method to switch between trimming and extending.

Fence

Fence

Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

Specify first fence point: *Specify the starting point of the selection fence*

Specify next fence point or [Undo]: *Specify the next point of the selection fence or enter* **u**

Specify next fence point or [Undo]: *Specify the next point of the selection fence, enter* **u**, *or press* ENTER

Crossing

Crossing

Selects objects within and crossing a rectangular area defined by two points.

Specify first corner: *Specify a point*

Specify opposite corner: *Specify a point at a diagonal from the first point*

Note Some crossing selections of objects to be trimmed are ambiguous. TRIM resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

Project

Project

Specifies the projection method used when trimming objects.

Enter a projection option [None/Ucs/View] *<current>*: *Enter an option or press* ENTER

None

Specifies no projection. The command trims only objects that intersect with the cutting edge in 3D space.

UCS
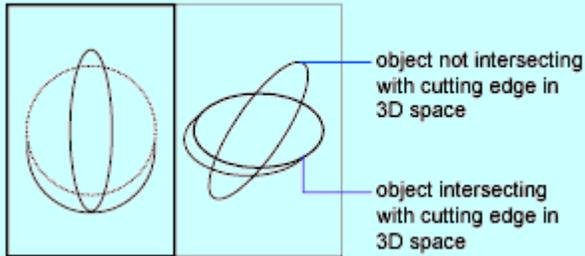
Specifies projection onto the *XY* plane of the current UCS. The command trims objects that do not intersect with the cutting edge in 3D space.
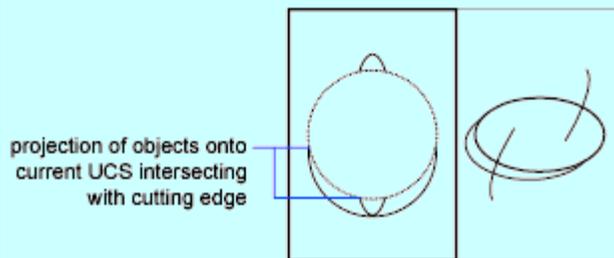
View

Specifies projection along the current view direction. The command trims objects that intersect the boundary in the current view.
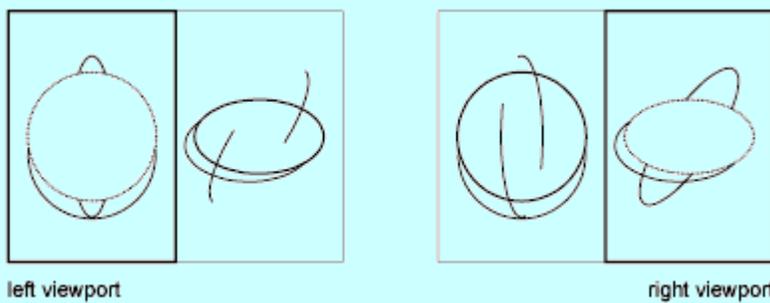
Edge

Edge

Determines whether an object is trimmed at another object's extrapolated edge or only to an object that intersects it in 3D space.

Enter an implied edge extension mode [Extend/No extend] *<current>*: *Enter an option or press ENTER*
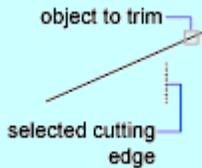
Extend

Extends the cutting edge along its natural path to intersect an object in 3D space.

No Extend

Specifies that the object is trimmed only at a cutting edge that intersects it in 3D space.

Note When trimming hatches, do not set Edge to Extend. If you do, gaps in the trim boundaries will not be bridged when trimming hatches, even when the gap tolerance is set to a correct value.
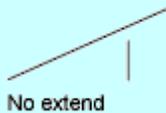
Erase

Erase

Deletes selected objects. This option provides a convenient method to erase unneeded objects without leaving the TRIM command.

Select objects to erase or <exit>: *Use an object selection method and press ENTER to return to the previous prompt*

Undo

Undo

Reverses the most recent change made by TRIM.

U

Reverses the most recent operation

Standard toolbar:

Shortcut menu: With no commands active and no objects selected, right-click in the drawing area and click Undo.

Command line: u

You can enter **u** as many times as you wish, backing up one step at a time, until the drawing is as it was when you began the current editing session.
When an operation cannot be undone, the command name is displayed but no action is performed. Operations external to the current drawing, such as plotting or writing to a file, cannot be undone.
If you changed modes or used transparent commands during a command, their effects are undone, along with the effects of the main command.
You can also press CTRL+Z to undo.

The U command is equivalent to entering **undo 1**.

UCS

Manages user coordinate systems

UCS toolbar:

Tools menu: New UCS

Command line: ucs

Enter an option [**New**/**Move**/**orthoGraphic**/**Prev**/**Restore**/**Save**/**Del**/**Apply**/**?**/**World**] <World>:

New

New

Defines a new coordinate system by one of six methods.
Specify origin of new UCS or [ZAxis/3point/OBject/Face/View/X/Y/Z] <0,0,0>:

Origin

Defines a new UCS by shifting the origin of the current UCS, leaving the direction of its X, Y, and Z axes unchanged.
Specify new origin point <0,0,0>: *Specify a point*

Specify a new origin point relative to the origin of the current UCS. If you do not specify a Z coordinate value for the origin, this option uses the current elevation.
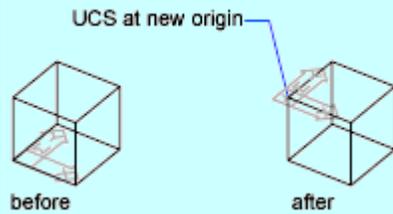
Z Axis

Defines a UCS with a particular positive Z axis.
Specify new origin point <0,0,0>: *Specify a point*

Specify point on positive portion of Z-axis *<current>*: *Specify a point*
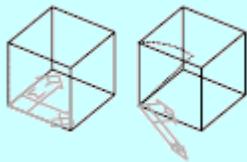
Specify a new origin and a point that lies on the new positive Z axis. The Z axis option tilts the XY plane.

3 Point

Specifies the new UCS origin and the direction of its positive X and Y axes. The Z axis follows by applying the right-hand rule. You can use this option to specify any possible coordinate system.
Specify new origin point <0,0,0>: *Specify a point (1)*

Specify point on positive portion of X-axis *<current>*: *Specify a point (2)*

Specify point on positive-Y portion of the UCS XY plane *<current>*: *Specify a point (3)*
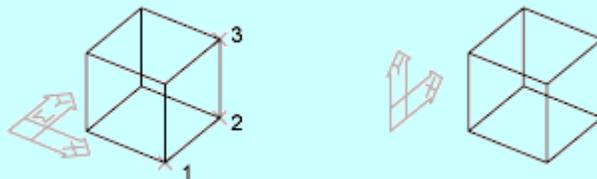
The first point specifies the origin of the new UCS. The second point defines the positive X axis. The third point defines the positive Y axis. The third point can be anywhere on the positive Y portion of the new UCS XY plane.

Object

Defines a new coordinate system based on a selected 3D object. The new UCS has the same extrusion direction (positive *Z* axis) as that of the selected object.

   Select object to align UCS: *Select an object*

select object   new UCS established

The following objects cannot be used with this option: 3D solids, 3D poly-lines, 3D meshes, viewports, mlines, regions, splines, ellipses, rays, xlines, leaders, and multiline text.

For objects other than 3D faces, the *XY* plane of the new UCS is parallel to the *XY* plane in effect when the object was drawn. However, the *X* and *Y* axes can be rotated differently.

The new UCS is defined as shown in the following table.

Define a UCS by selecting an object	
Object	**Method of determining UCS**
Arc	The center of the arc becomes the new UCS origin. The *X* axis passes through the arc endpoint that is closest to the selection point.
Circle	The center of the circle becomes the new UCS origin. The *X* axis passes through the selection point.
Dimension	The midpoint of the dimension text becomes the new UCS origin. The direction of the new *X* axis is parallel to the *X* axis of the UCS in effect when the dimension was drawn.
Line	The endpoint nearest the selection point becomes the new UCS origin. The new *X* axis is set so that the line lies in the *XZ* plane of the new UCS. The line's second endpoint has a *Y* coordinate of zero in the new UCS.
Point	The point becomes the new UCS origin.
2D polyline	The start point of the polyline is the new UCS origin. The *X* axis extends along the line segment from the start point to the next vertex.
Solid	The first point of the solid determines the new UCS origin. The new *X* axis lies along the line between the first two points.
Trace	The "from" point of the trace becomes the UCS origin, with the *X* axis lying along its centerline.
3D face	The new UCS origin is taken from the first point, the *X* axis from the first two points, and the *Y* positive side from the first and fourth points. The *Z* axis follows by application of the right-hand rule.
Shape, text, block reference, attribute definition	The insertion point of the object becomes the new UCS origin, and the new *X* axis is defined by the rotation of the object about its extrusion direction. The object you select to establish a new UCS has a rotation angle of zero in the new UCS.

Face

Aligns the UCS to the selected face of a solid object. To select a face, click within the boundary of the face or on the edge of the face. The face is highlighted and the UCS *X* axis is aligned with the closest edge of the first face found.

   Select face of solid object:

   Enter an option [Next/Xflip/Yflip] <accept>:

Next

Locates the UCS on either the adjacent face or the back face of the selected edge.

Xflip

Rotates the UCS 180 degrees around the *X* axis.

Yflip

Rotates the UCS 180 degrees around the *Y* axis.

Accept

If you press ENTER, accepts the location. The prompt repeats until you accept a location.

View

Establishes a new coordinate system with the *XY* plane perpendicular to your viewing direction (parallel to your screen). The UCS origin remains unchanged.

X, Y, Z

Rotates the current UCS about a specified axis.
> Specify rotation angle about *n* axis <0>: *Specify an angle*

In the prompt, *n* is *X, Y,* or *Z*. Enter a positive or negative angle to rotate the UCS. The right-hand rule determines the positive direction of rotation around the axis.

before   after

You can define any UCS by specifying an origin and one or more rotations around the *X, Y,* or *Z* axis.

world coordinate   rotation about X   rotation about Y   rotation about Z
system   axis = 90   axis = 90   axis = 90
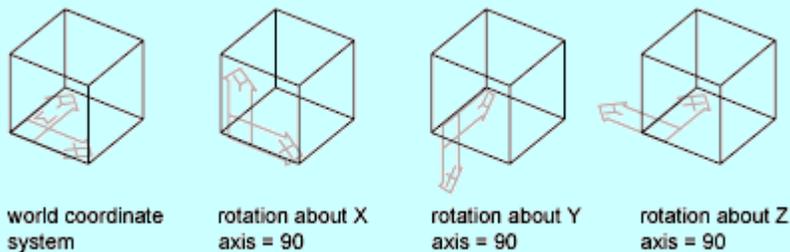
Move

Move

Redefines a UCS by shifting the origin or changing the *Z*-depth of the current UCS, leaving the orientation of its *XY* plane unchanged. Changing the *Z*-depth moves the UCS along its *Z* axis in a positive or negative direction relative to the current origin.

Specify new origin point or [Zdepth] <0,0,0>: *Specify a point or enter* **z**

New Origin

Changes the origin of the UCS.

Zdepth

Specifies the distance along the *Z* axis to move the UCS origin.

Specify Zdepth <0>: *Enter a distance or press* ENTER

If you have multiple viewports active, and you change viewports to specify a new origin or *Z*-depth, the change is applied to the UCS of the viewport that was current when the command was started and that viewport becomes current once the command has finished.

Note The UCS Move option does not add a UCS to the Previous list.

Orthographic

Orthographic

Specifies one of the six orthographic UCSs provided. These UCS settings are typically used when viewing and editing 3D models.

Enter an option [Top/Bottom/Front/BAck/Left/Right] <*current*>: *Enter an option or press* ENTER
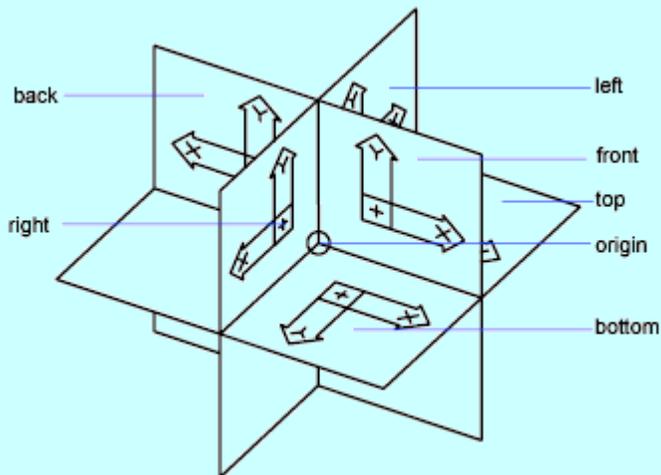
By default the orthographic UCS settings orient the current UCS relative to the world coordinate system (WCS) origin and orientation. The UCSBASE system variable controls the UCS upon which the orthographic settings are based. The origin or *Z*-depth of the orthographic UCS settings can be changed using the **Move** option of the UCS command.

Prev

Prev

Restores the previous UCS. The program retains the last 10 coordinate systems created in paper space and the last 10 coordinate systems created in model space. Repeating the Prev option steps back through one set or the other, depending on which space is current.
If you have saved different UCS settings in individual viewports and you switch between viewports, the different UCSs are not retained in the Previous list. However, if you change a UCS setting within a viewport, the last UCS setting is retained in the Previous list. For example, changing the UCS from World to UCS1 would retain World at the top of the Previous list. If you then switch viewports with Front as the current UCS, and then change the UCS to Right, the Front UCS is retained at the top of the Previous list. If you then choose the UCS Previous option twice in this viewport, the UCS setting changes to Front, and then back to World. See the UCSVP system variable.

Restore

Restore

Restores a saved UCS so that it becomes the current UCS. Restoring a saved UCS does not reestablish the viewing direction in effect when the UCS was saved.
   Enter name of UCS to restore or [?]: *Enter a name or enter* ?

Name

  Specifies a named UCS.

?—List UCSs

  Lists the names of currently defined UCSs.

  Enter UCS name(s) to list <*>: *Enter a name list or press* ENTER *to list all UCSs*

Save

Save

Saves the current UCS to a specified name. The name can have up to 255 characters and can include letters, numbers, blank spaces, and any special character not used by Microsoft® Windows® and this program for other purposes.
   Enter name to save current UCS or [?]: *Enter a name or enter* ?

Name

  Saves the current UCS with the specified name.

?—List UCSs

  Lists the names of currently defined UCSs.

  UCS name(s) to list <*>: *Enter a name list or press* ENTER *to list all UCSs*

Del

Del

Removes the specified UCS from the list of saved user coordinate systems.
   Enter UCS name(s) to delete <none>: *Enter a name list or press* ENTER
If you delete a named UCS that is current, the current UCS is renamed UNNAMED.

Apply

Apply

Applies the current UCS setting to a specified viewport or all active viewports when other viewports have a different UCS saved in the viewport. The UCSVP system variable determines whether a UCS is saved with a viewport.

 Pick viewport to apply current UCS or [All] <*current*>: *Specify a viewport by clicking inside it, enter* **a***, or press* ENTER

Viewport

 Applies the current UCS to the specified viewport and ends the UCS command.

All

 Applies the current UCS to all active viewports.

?—List UCSs

?—List UCSs

Lists names of user coordinate systems and provides the origin and *X, Y,* and *Z* axes for each saved UCS relative to the current UCS. If the current UCS is unnamed, it is listed as WORLD or UNNAMED, depending on whether it's the same as the WCS.

 UCS name(s) to list <*>: *Enter a name list*

World

World

Sets the current user coordinate system to the world coordinate system. The WCS is the basis for all user coordinate systems and cannot be redefined.

before   after

UCSICON

Controls the visibility and placement of the UCS icon

View menu: Display ➤ UCS Icon

Command line: ucsicon

The UCS icon represents the orientation of the user coordinate system (UCS) axes and the location of the current UCS origin. It also represents the current viewing direction relative to the *XY* plane.

A UCS can be stored with a viewport if the UCSVP system variable is set to 1 for that viewport.

Different coordinate system icons are displayed in paper space and model space. In both cases, a plus sign (**+**) appears at the base of the icon when it is positioned at the origin of the current UCS. The letter *W* appears in the *Y* portion of the icon if the current UCS is the same as the world coordinate system.

With the 3D UCS icon, a square is displayed in the *XY* plane at the origin when the UCS is the same as the world coordinate system.

A box is formed at the base of the icon if you are viewing the UCS from above (the positive *Z* direction). The box is missing if you are viewing the UCS from below.

With the 3D UCS icon, the *Z* axis is solid when viewed from above the XY plane and dashed when viewed from below 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—the 2D UCS icon is replaced by a broken pencil icon. The 3D UCS icon does not use a broken pencil icon.

> Enter an option [**ON**/**OFF**/**All**/**Noorigin**/**ORigin**/**Properties**] *<current>*: *Enter an option or press* ENTER

All   No origin   Origin

UCSMAN

UCS toolbar:

 Tools menu: Named UCS
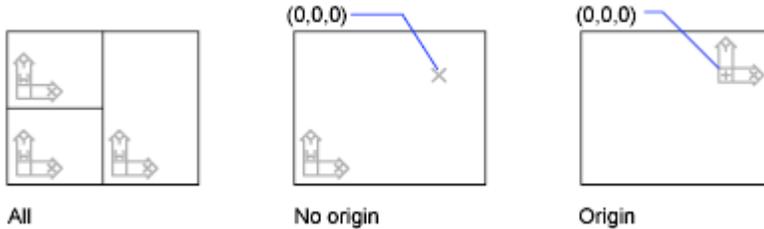
Command line: ucsman

Displays and modifies defined and unnamed user coordinate systems, restores named and orthographic UCSs, and specifies UCS icon and UCS settings for viewports.

The UCS dialog box includes the following tabs:

- **Named UCSs**
- **Orthographic UCSs**
- **Settings**

Named UCSs Tab (UCS Dialog Box)

Named UCSs Tab (UCS Dialog Box)

Lists user coordinate systems and sets the current UCS.

Current UCS

Displays the name of the current UCS. If the UCS has not been saved and named, it is listed as UNNAMED.

UCS Names List

Lists the coordinate systems defined in the current drawing. If there are multiple viewports and multiple unnamed UCS settings, the list includes only the unnamed UCS of the current viewport. Unnamed UCS definitions that are locked to other viewports (when the UCSVP system variable is set to 1 in that viewport) are not listed in the current viewport. A pointer indicates the current UCS. UNNAMED is always the first entry if the current UCS is unnamed. World is always listed and cannot be renamed or deleted. If you define other coordinate systems for the active viewport during the current editing session, a Previous entry is next. You can step back through these coordinate systems by selecting Previous and Set Current repeatedly.
To add a UCS name to this list, use the Save option of the UCS command. To rename or delete a customized UCS, right-click the UCS name in the list and use the shortcut menu.

Set Current

Restores the selected coordinate system. You can also restore a selected coordinate system by double-clicking its name in the list, or by right-clicking the name and choosing Set Current. The Current UCS text is updated.

Details

Displays the UCS Details dialog box, which displays UCS coordinate data. You can also view the details about a selected coordinate system by right-clicking the name and choosing Details.

Rename (Shortcut Menu Only)

Renames a customized UCS. You cannot rename the World UCS. You can also rename a UCS by double-clicking its name in the list.

Delete (Shortcut Menu Only)

Deletes a customized UCS. You cannot delete the World UCS.

Orthographic UCSs Tab (UCS Dialog Box)

Orthographic UCSs Tab (UCS Dialog Box)

Changes the UCS to one of the orthographic UCS settings.

Current UCS

Displays the name of the current UCS. If the UCS has not been saved and named, it is listed as UNNAMED.

Orthographic UCS Names

Lists the six orthographic coordinate systems defined in the current drawing. The orthographic coordinate systems are defined relative to the UCS specified in the Relative To list. The Depth column lists the distance between the orthographic coordinate system and the parallel plane passing through the origin of the UCS base setting (stored in the UCSBASE system variable).

Name

Specifies the name of the orthographic coordinate system.

Depth

Specifies the distance between the XY plane of the orthographic UCS and a parallel plane that passes through the origin of the coordinate system specified by the UCSBASE system variable. The parallel plane of the UCSBASE coordinate system can be an XY, YZ, or XZ plane.

Note You can specify the depth or a new origin for the selected orthographic UCS. See Depth option.

Set Current

Restores the selected coordinate system. You can also restore a selected coordinate system by double-clicking its name in the list, or by right-clicking the name and choosing Set Current.

Details

Displays the UCS Details dialog box, which displays UCS coordinate data. You can also view the details about a selected coordinate system by right-clicking the name and choosing Details.

Relative To

Specifies the base coordinate system for defining the orthographic UCSs. By default, WCS is the base coordinate system. The list displays all the named UCSs in the current drawing.
Whenever you change the Relative To setting, the origin of the selected orthographic UCS is restored to its default position.
If you save an orthographic coordinate system in a drawing as part of a viewport configuration, or if you select a setting in Relative To other than World, the orthographic coordinate system name changes to UNNAMED to distinguish it from the predefined orthographic coordinate system.

Reset (Shortcut Menu Only)

Restores the origin of the selected orthographic coordinate system. The origin can be changed from its default 0,0,0 position using the Move option of the UCS command. To reset the origin of the selected orthographic coordinate system, right-click the name in the list and choose Reset Origin from the shortcut menu. The origin is restored to its default location (0,0,0) relative to the specified base coordinate system.

Depth (Shortcut Menu or Double-Click)

Specifies the distance between the XY plane of the orthographic UCS and a parallel plane that

passes through the origin of the coordinate system. In the Orthographic UCS Depth dialog box, enter a value or choose the Select New Origin button to use the pointing device to specify a new depth or a new origin.

Settings Tab (UCS Dialog Box)

Displays and modifies UCS icon settings and UCS settings saved with a viewport.

UCS Icon Settings

Specifies the UCS icon display settings for the current viewport.

On

Displays the UCS icon in the current viewport.

Display at UCS Origin Point

Displays the UCS icon at the origin of the current coordinate system in the current viewport. If this option is cleared, or if the origin of the coordinate system is not visible in the viewport, the UCS icon is displayed at the lower-left corner of the viewport.

Apply to All Active Viewports

Applies the UCS icon settings to all active viewports in the current drawing.

UCS Settings

Specifies UCS behavior when the UCS setting is updated.

Save UCS with Viewport

Saves the coordinate system setting with the viewport. This option sets the UCSVP system variable. If this option is cleared, the viewport reflects the UCS of the viewport that is current.

Update View to Plan When UCS Is Changed

Restores Plan view when the coordinate system in the viewport is changed. (UCSFOLLOW system variable)

UNDO

Reverses the effect of commands

 Standard toolbar:

 Command line: undo

Enter the **number** of operations to undo or [**Auto**/**Control**/**BEgin/End**/**Mark/Back**]: *Enter a positive number, enter an option, or press* ENTER *to undo a single operation*

UNDO displays the command or system variable name on the command line to indicate that you have stepped past the point where the command was used.

Note UNDO has no effect on some commands and system variables, including those that open, close, or save a window or a drawing, display information, change the graphics display, regenerate the drawing, or export the drawing in a different format.

Number

Undoes the specified number of preceding operations. The effect is the same as entering **u** multiple times.

Auto

Groups the actions of a single command, making them reversible by a single U̲ command. When the Auto option is on, starting a command groups all actions until you exit that command. You can undo the group of actions as if it were one action.
UNDO Auto is not available if the Control option has turned off or limited the UNDO feature.
    Enter UNDO Auto mode [ON/OFF] *<current>*: *Enter* **on** *or* **off***, or press* ENTER

Control

Limits or turns off UNDO.
    Enter an UNDO control option [All∕None/One/Combine] <All>: *Enter an option or press* ENTER

All

    Turns on the full UNDO command.

None

    Turns off the U and UNDO commands and discards any UNDO command information saved earlier in the editing session. The Undo button on the Standard toolbar is unavailable.

    The Auto, Begin, and Mark options are not available when None or One is in effect. If you attempt to use UNDO while it is turned off, the following prompt is displayed:

    Enter an UNDO control option [All/None/One] <All>:

One

    Limits UNDO to a single operation.

    The Auto, Begin, and Mark options are not available when None or One is in effect. The main prompt for the UNDO command changes to show that only a Control option or a single step of the UNDO command is available when the One option is in effect.

    Control/<1>:

    If you enter **C**, the previous prompt is displayed:

    Enter an UNDO control option [All/None/One] <All>:

Combine

    Controls whether multiple, consecutive zoom and pan commands are combined as a single operation for undo and redo operations.

    Combine zoom and pan operations? [Yes/No] <Yes>: *Enter* **y** *or* **n** *or press* ENTER

Begin, End

Groups a sequence of actions into a set. After you enter the Begin option, all subsequent actions become part of this set until you use the End option. Entering **undo begin** while a group is already active ends the current set and begins a new one. UNDO and U treat grouped actions as a single action.

If you enter **undo begin** without **undo end**, using the Number option undoes the specified number of commands but does not back up past the begin point. If you want to go back to before the begin point, you must use the End option, even if the set is empty. The same applies to the U command. A mark placed by the Mark option disappears inside an UNDO group.

Mark, Back

Mark, Back

Mark places a mark in the undo information. Back undoes all the work done back to this mark. If you undo one operation at a time, you are informed when you reach the mark.
You can place as many marks as necessary. Back moves back one mark at a time, removing the mark. If no mark is found, Back displays the following prompt:
   This will undo everything. OK? <Y>: *Enter* **y** *or* **n** *or press* ENTER

Enter **y** to undo all commands entered in the current session. Enter **n** to ignore the Back option.

When you use the Number option to undo multiple actions, UNDO stops if it encounters a mark.

UNITS

Controls coordinate and angle display formats and precision
 Format menu: Units

 Command line: units (or **'units** for transparent use)

The Drawing Units dialog box is displayed.

If you enter **-units** at the Command prompt, UNITS displays UNITS Command Line prompts on the command line.

Defines the unit and angle formats.

Length

Specifies the current unit of measurement and the precision for the current units.

Type

Sets the current format for units of measure. The values include Architectural, Decimal, Engineering, Fractional, and Scientific. The Engineering and Architectural formats produce feet-and-inches displays and assume that each drawing unit represents one inch. The other formats can represent any real-world unit.

Precision

Sets the number of decimal places or fractional size displayed for linear measurements.

Angle

Specifies the current angle format and the precision for the current angle display.

Type

Sets the current angle format.

Precision

Sets the precision for the current angle display.

The following conventions are used for the various angle measures: decimal degrees appear as decimal numbers, grads appear with a lowercase *g* suffix, and radians appear with a lowercase *r* suffix. The degrees/minutes/seconds format uses *d* for degrees, ' for minutes, and " for seconds; for example:

123d45'56.7"

Surveyor's units show angles as bearings, using N or S for north or south, degrees/minutes/seconds for how far east or west the angle is from direct north or south, and E or W for east or west; for example:

N 45d0'0" E

The angle is always less than 90 degrees and is displayed in the degrees/minutes/seconds format. If the angle is precisely north, south, east, or west, only the single letter representing the compass point is displayed.

Clockwise

Calculates positive angles in the clockwise direction. The default direction for positive angles is counterclockwise.

When prompted for an angle, you can point in the desired direction or enter an angle regardless of the setting specified for Clockwise.

Insertion Scale

Controls the unit of measurement for blocks and drawings that are inserted into the current drawing. A block or a drawing that is created with units that are different from the units specified with this option is scaled when inserted. The insertion scale is the ratio of the units used in the source block or drawing and the units used in the target drawing. Select Unitless to insert the block without scaling it to match the specified units.

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

Sample Output

Displays an example of the current settings for units and angles.

Direction

Displays the Direction Control Dialog BoxDirection Control dialog box.

VIEW

Saves and restores named views

🔖 **View menu:** Named Views

⌨ **Command line:** view

The View dialog box is displayed.

If you enter **-view** at the Command prompt, VIEW displays prompts on the command line.

Note The VIEW command cannot be used transparently.

Saves and restores named views and orthographic views.

The View dialog box includes the following tabs:

- **Named Views**
- **Orthographic and Isometric Views**

Named Views Tab (View Dialog Box)

Named Views Tab (View Dialog Box)

Creates, sets, renames, modifies, and deletes named views.

Current View

Displays the name of the current view. When the View dialog box is first displayed, it lists the current view as CURRENT.

View Names

Lists the named views in the current drawing and the information that is stored with each named view. A pointer indicates the current named view.

The following information is stored with a named view:

- *Category*: The category in which the named view is listed on the View List tab in the Sheet Set Manager.
- *Location*: The Model tab, or the name of the layout tab on which the named view was saved.
- *VP*: Whether the named view is associated with a viewport on a sheet in a sheet set.
- *Layers*: Whether layer visibility settings are saved with the named view.
- *UCS*: The name of the user coordinate system (UCS) that is saved with the named view.
- *Perspective*: Whether the named view is a perspective view.

Set Current

Restores the selected named view. You can also restore a named view by double-clicking its name in the list or by right-clicking its name and clicking Set Current on the shortcut menu.

New

Displays the New View dialog box.

Update Layers

Updates the layer information saved with the selected named view to match the layer visibility in the current model space or layout viewport.

Edit Boundaries

Displays the selected named view centered and zoomed out and with the rest of the drawing area in a lighter color to show the boundaries of the named view. You can specify opposite corners of a new boundary repeatedly until you press ENTER to accept the results.

Details

Displays the View Details dialog box.

Delete

Deletes the selected named view.

Rename (Shortcut Menu Only)

Renames the selected named view. You can also press F2.

Delete Layer Information (Shortcut Menu Only)

Deletes the layer visibility information that was saved with the selected named view.

Detach from Viewport (Shortcut Menu Only)

Removes the association between the selected named view and its viewport in the Sheet Set Manager.

Orthographic and Isometric Views Tab (View Dialog Box)

Orthographic and Isometric Views Tab (View Dialog Box)

Restores orthographic or isometric views.

Current View

Displays the name of the current view. The default value is CURRENT.

View Names

Lists the orthographic and isometric views. A pointer indicates the current orthographic or isometric view. The views are relative to the UCS listed in the Relative To list.

Set Current

Restores the selected orthographic view. You can also restore a selected view by double-clicking its name in the list or by right-clicking the name and choosing Set Current. When an orthographic view is restored, the program zooms to the extents of the view.

Relative To

Specifies the base coordinate system for defining the orthographic view. By default, the world coordinate system (WCS) is the base coordinate system. (UCSBASE system variable)

The UCSBASE system variable controls the orientation of orthographic UCSs relative to the WCS or to named UCSs. Changing this setting also changes the orientation of the predefined orthographic UCSs. A warning message is displayed when you change the value of the UCSBASE system variable.

Restore Orthographic UCS with View

Restores the associated UCS when you make an orthographic view current. This option sets the UCSORTHO system variable.

VIEWRES

Sets the resolution for objects in the current viewport

⌨ **Command line:** viewres

Do you want fast zooms [Yes/No] <Y>: *Press* ENTER *(Fast Zoom is no longer a functioning option of this command and remains for script compatibility only.)*

Enter circle zoom percent (1–20000) <current>: *Enter an integer from 1 to 20,000 or press* ENTER

The model is regenerated.
VIEWRES controls the appearance of circles, arcs, ellipses, and splines using short vectors. The greater the number of vectors, the smoother the appearance of the circle or arc. For example, if you create a very small circle and then zoom in, it might appear to be a polygon.

Using VIEWRES to increase the zoom percentage and regenerate the drawing updates and smooths the circle's appearance. Decreasing the zoom percentage has the opposite effect.
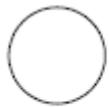
VIEWRES
at 500
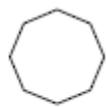
VIEWRES
at 15

Note Increasing the zoom percentage in VIEWRES may increase the time it takes to regenerate the drawing.

When a paper space layout is made current for the first time and a default viewport is created in the layout, the viewing resolution for this initial viewport is the same as the viewing resolution for the Model tab viewport.
The VIEWRES setting is saved in the drawing. To change the default for new drawings, consider specifying the VIEWRES setting in the template files on which you base your new drawings.

VPCLIP

Clips viewport objects and reshapes the viewport border

> **Shortcut menu:** Select the viewport to clip, right-click in the drawing area, and choose Viewport Clip.

Command line: vpclip

Select viewport to clip:
Select clipping **object** or [**Polygonal**/**Delete**] <Polygonal>:

Clipping Object

Specifies an object to act as a clipping boundary. Objects that are valid as clipping boundaries include closed poly-lines, circles, ellipses, closed splines, and regions.

Polygonal

Draws a clipping boundary. You can draw line segments or arc segments by specifying points. The following prompt is displayed:

Specify start point:
Specify next point or [Arc/Close/Length/Undo]:

The descriptions of the Next Point, Arc, Close, Length, and Undo options match the descriptions of the corresponding options in the PLINE command.

Delete

Deletes the clipping boundary of a selected viewport. This option is available only if the selected viewport has already been clipped. If you clip a viewport that has been previously clipped, the original clipping boundary is deleted, and the new clipping boundary is applied.

VPLAYER

Sets layer visibility within viewports

⌨ **Command line:** vplayer

Enter an option [?/Freeze/Thaw/Reset/Newfrz/Vpvisdflt]:

?—List Frozen Layers

?—List Frozen Layers

Displays the names of frozen layers in a selected viewport.
    Select a viewport:

The command line window displays the names of any layers that are frozen in the selected viewport.

Freeze

Freeze

Freezes a layer or set of layers in one or more viewports. Objects on frozen layers are not displayed, regenerated, or plotted.
    Enter layer name(s) to freeze: *Enter one or more layer names*
    Enter an option [All/Select/Current] <Current>: *Enter an option or press* ENTER

All

Applies the changes in all viewports.

Select

Temporarily switches to paper space, allowing you to select the viewports where you can apply the layer settings.

    Select objects: *Select one or more viewports and press* ENTER

Current

Applies the changes in the current viewport only.

Thaw

Thaw

Thaws layers in specific viewports.
    Enter layer name(s) to thaw: *Enter one or more layer names*
    Enter an option [All/Select/Current] <Current>: *Enter an option or press* ENTER

All

Applies the changes in all viewports.

Select

Applies the changes in selected viewports.

    Select objects: *Select one or more viewports and press* ENTER

Current

Applies the changes in the current viewport only.

Reset

Reset

Sets the visibility of layers in specified viewports to their current default setting.
>   Enter layer name(s) to reset: *Enter one or more layer names*
>
>   Enter an option [All/Select/Current] <Current>: *Enter an option or press* ENTER

All

   Applies the changes in all viewports.

Select

   Applies the changes in selected viewports.

>   Select objects: *Select one or more viewports and press* ENTER

Current

   Applies the changes in the current viewport only.

Newfrz (New Freeze)

Newfrz (New Freeze)

Creates new layers that are frozen in all viewports.
>   Enter name(s) of new layers frozen in all viewports: *Enter one or more layer names*

Vpvisdflt (Viewport Visibility Default)

Vpvisdflt (Viewport Visibility Default)

Thaws or freezes the specified layers in subsequently created viewports.
>   Enter layer name(s) to change viewport visibility: *Enter one or more layer names*
>
>   Enter a viewport visibility option [Frozen/Thawed] <*current*>: *Enter* **f** *or* **t***, or press* ENTER

VPOINT

Sets the viewing direction for a three-dimensional visualization of the drawing

View menu: 3D Views ➤ Viewpoint

Command line: vpoint

>   Current view direction: VIEWDIR=*current*
>
>   Specify a **view point** or [**Rotate**] <**display compass and tripod**>: *Specify a point, enter* **r***, or press*
>   ENTER*to display a compass and axis tripod*

View Point

   Using the *X,Y,Z* coordinate you enter, creates a vector that defines a direction from which the drawing can be viewed. The view defined is as if the viewer is looking from the point back at the origin (0,0,0).

as seen by viewer
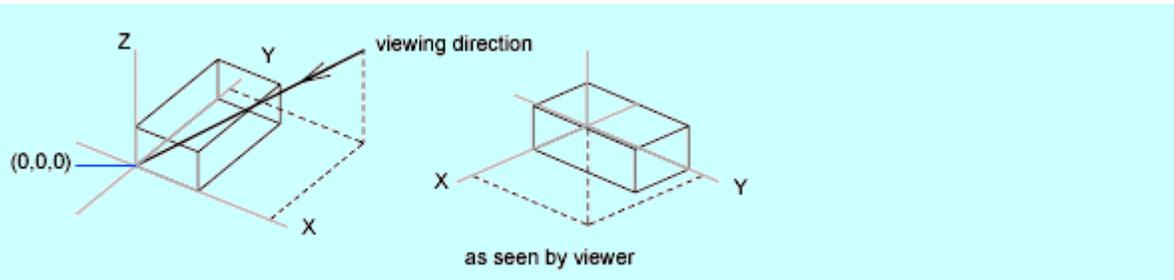
Rotate

Specifies a new viewing direction using two angles.

Enter angle in XY plane from X axis *<current>*: *Specify an angle*

The first angle is specified with respect to the *X* axis, in the *XY* plane.

Enter angle from XY plane *<current>*: *Specify an angle*

The second angle is specified up or down from the *XY* plane.
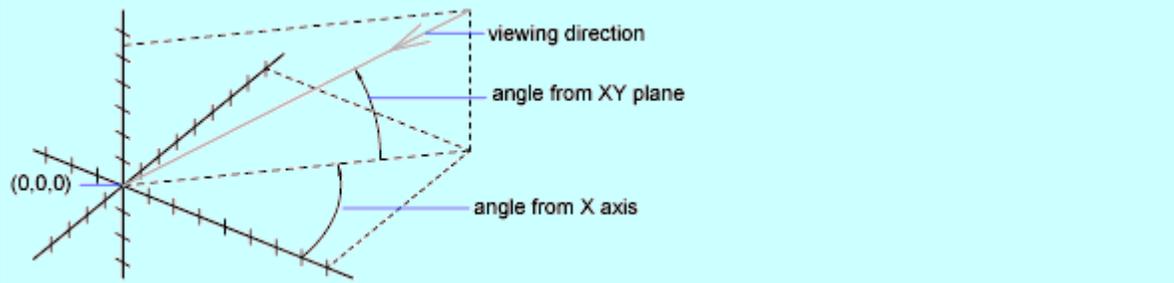
Compass and Axis Tripod

Displays a compass and axis tripod, which you use to define a viewing direction in the viewport.
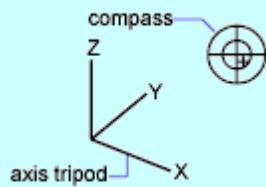
The compass is a two-dimensional representation of a globe. The center point is the north pole $(0,0,n)$, the inner ring is the equator $(n,n,0)$, and the entire outer ring is the south pole $(0,0,-n)$.

You can move the small crosshairs on the compass to any portion of the globe with your pointing device. As you move the crosshairs, the axis tripod rotates to conform to the viewing direction indicated on the compass. To select a viewing direction, move your pointing device to a location on the globe and click.

VPORTS

Creates multiple viewports in model space or paper space

Layouts toolbar:

View menu: Viewports

Command line: vports

The Viewports dialog box is displayed.

If you enter **-vports** at the Command prompt, VPORTS displays prompts on the command line.

Creates new viewport configurations, or names and saves a model space viewport configuration. The options available in this dialog box depend on whether you are configuring model space viewports (on the Model tab) or layout viewports (on a layout tab).

Tabs for model space viewports: **New Viewports—Model Space**, **Named Viewports—Model Space**

Tabs for layout viewports: **New Viewports—Layouts**, **Named Viewports—Layouts**

New Viewports Tab—Model Space (Viewports Dialog Box)

New Viewports Tab—Model Space (Viewports Dialog Box)

Displays a list of standard viewport configurations and configures model space viewports.

New Name

Specifies a name for the new model space viewport configuration you are creating. If you do not enter a name, the viewport configuration you create is applied but not saved. If a viewport configuration is not saved, it cannot be used in a layout.

Standard Viewports

Lists and sets the standard viewport configurations, including CURRENT, which is the current configuration.

Preview

Displays a preview of the viewport configuration you select and the default views assigned to each individual viewport in the configuration.

Apply To

Applies the model space viewport configuration to the entire display or to the current viewport.

- *Display:* Applies the viewport configuration to the entire Model tab display. Display is the default setting.
- *Current Viewport:* Applies the viewport configuration to the current viewport only.

Setup

Specifies either a 2D or a 3D setup. When you select 2D, the new viewport configuration is initially created with the current view in all of the viewports. When you select 3D, a set of standard orthogonal 3D views is applied to the viewports in the configuration.

Change View To

Replaces the view in the selected viewport with the view you select from the list. You can choose a named view, or if you have selected 3D setup, you can select from the list of standard views. Use the Preview area to see the choices.

Named Viewports Tab—Model Space (Viewports Dialog Box)

Named Viewports Tab—Model Space (Viewports Dialog Box)

Displays any saved viewport configurations in the drawing. When you select a viewport configuration, the layout of the saved configuration is displayed in Preview.

Current Name

Displays the name of the current viewport configuration.

New Viewports Tab—Layouts (Viewports Dialog Box)

New Viewports Tab—Layouts (Viewports Dialog Box)

Displays a list of standard viewport configurations and configures layout viewports.

Standard Viewports

Displays a list of standard viewport configurations and configures layout viewports.

Preview

Displays a preview of the viewport configuration you select and the default views assigned to each individual viewport in the configuration.

Viewport Spacing

Specifies the spacing you want to apply between the layout viewports you are configuring.

Setup

Specifies either a 2D or a 3D setup. When you select 2D, the new viewport configuration is initially created with the current view in all of the viewports. When you select 3D, a set of standard orthogonal 3D views is applied to the viewports in the configuration.

Change View To

Replaces the view in the selected viewport with the view you select from the list. You can choose a named view, or if you have selected 3D setup, you can select from the list of standard views. Use the Preview area to see the choices.

Named Viewports Tab—Layouts (Viewports Dialog Box)

Named Viewports Tab—Layouts (Viewports Dialog Box)

Displays any saved and named model space viewport configurations for you to use in the current layout. You cannot save and name a layout viewport configuration.

WBLOCK

Writes objects or a block to a new drawing file

⌨ **Command line:** wblock

The Write Block dialog box is displayed.

Entering **– wblock** at the Command prompt displays a standard file selection dialog box in which to specify a name for the new drawing file, followed by prompts on the command line. If FILEDIA is set to 0, the standard file selection dialog box is suppressed.

Saves objects or converts a block to a file.

The Write Block dialog box displays different default settings depending on whether nothing is selected, a single block is selected, or objects other than blocks are selected.

Source

Specifies blocks and objects, saves them as a file, and specifies insertion points.

Block

Specifies an existing block to save as a file. Select a name from the list.

Entire Drawing

Selects current drawing as a block.

Objects

Specifies a base point for the block. The default value is 0,0,0.

Base Point

Specifies a base point for the block. The default value is 0,0,0.

Pick Point

Temporarily closes the dialog box so that you can specify an insertion base point in the current drawing.

X

Specifies the *X* coordinate value for the base point.

Y

Specifies the *Y* coordinate value for the base point.

Z

Specifies the *Z* coordinate value for the base point.

Objects

Sets the effect of block creation on objects used to create a block.

Retain

Retains the selected objects in the current drawing after saving them as a file.

Convert to Block

Converts the selected object or objects to a block in the current drawing after saving them as a file. The block is assigned the name in File Name.

Delete from Drawing

Deletes the selected objects from the current drawing after saving them as a file.

Select Objects Button

Temporarily closes the dialog box so that you can select one or more objects to save to the file.

Quick Select Button

Opens the Quick Select dialog box, which you can use to filter your selection set.

Objects Selected

Indicates the number of objects selected.

Destination

Specifies the new name and location of the file and the units of measurement to be used when the block is inserted.

File Name and Path

Specifies a file name and path where the block or objects will be saved.

[...]

Displays a standard file selection dialog box.

Insert Units

Specifies the unit value to be used for automatic scaling when the new file is dragged from DesignCenter™ or inserted as a block in a drawing that uses different units. Select Unitless if you do not want to automatically scale the drawing when you insert it. See INSUNITS.

WMFIN

Imports a Windows metafile

Insert menu: Windows Metafile

Command line: wmfin

The Import WMF dialog box (a standard file selection dialog box) is displayed. Enter a file name. The *.wmf* file extension is added automatically.

In the Import WMF dialog box, if you click Tools ➤ Options, the WMF In Options dialog box is displayed. You can also open this dialog box directly by using WMFOPTS. After opening the selected WMF file, the following prompt is displayed:

Specify **insertion point** or [**Scale**/**X**/**Y**/**Z**/**Rotate**/**PScale**/**PX**/**PY**/**PZ**/**PRotate**]: *Specify a point or enter an option*

WMFOPTS

Sets options for WMFIN

Command line: wmfopts

The WMF In Options dialog box is displayed.

Controls whether metafiles maintain relative line widths and whether they are imported as wireframes or solid objects.

Wire Frame (No Fills)

Imports objects as wireframes. If you clear this option, objects are imported as filled objects.

Wide Lines

Maintains the relative line width of lines and borders. If you clear this option, lines are imported with zero width.

WMFOUT

Saves objects to a Windows metafile

Command line: wmfout

The Create WMF standard file selection dialog box is displayed. Enter a file name. A *.wmf* file extension is added.

Select objects: *Use an object selection method and press* ENTER *when you finish selecting objects*

The selected objects are saved to a file in Windows metafile format.

Note If one or more raster objects are attached to the drawing *and* if the image application *(ism.arx)* is loaded, the maximum resolution of the resulting WMF file is 2K by 2K. The image

application is loaded the first time you run IMAGE, IMAGEADJUST, IMAGEATTACH, IMAGECLIP, IMAGEFRAME, or IMAGEQUALITY.

XATTACH

Attaches an external reference to the current drawing

Reference toolbar:

Insert menu: External Reference

Command line: xattach

The Select Reference File dialog box (a standard file selection dialog box) is displayed. The External Reference dialog box is displayed after a file is selected.

Attaches a drawing as an external reference (xref). If you attach a drawing that itself contains an attached xref, the attached xref appears in the current drawing. Like blocks, attached xrefs can be nested. If another person is currently editing the xref, the drawing attached is based on the most recently saved version.

Name

Displays the xref name in the list after an xref is attached. When an attached xref is selected in the list, its path is displayed in Path.

Browse

Choose Browse to display the Select a Reference dialog box (a standard file selection dialog box), in which you can select new xrefs for the current drawing.

Found In

Displays the path where the xref was found.
If no path was saved for the xref or if the xref is no longer located at the specified path, the program searches for the xref in the following order:

- 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

Saved Path

Displays the saved path, if any, that is used to locate the xref. This path can be an absolute (fully specified) path, a relative (partially specified) path, or no path.

Reference Type

Specifies whether the xref is an attachment or an overlay.

Path Type

Specifies whether the saved path to the xref is the full path, a relative path, or no path. You must save the current drawing before you can set the path type to Relative Path. For a nested xref, a relative path always references the location of its immediate host and not necessarily the currently open drawing.
The Relative Path option is not available if the referenced drawing is located on a different local disk drive or on a network server.

Insertion Point

Specifies the insertion point of the selected xref.

Specify On-Screen

Displays prompts on the command line and makes the X, Y, and Z options unavailable. The descriptions of the options displayed on the command line match the descriptions of the corresponding options under Insertion Point in INSERT.

X

Specifies the *X* coordinate value for the point at which an xref instance is inserted into the current drawing. The insertion point in the current drawing is aligned with the insertion point defined in the BASE command in the referenced file.

Y

Specifies the *Y* coordinate value for the point at which an xref instance is inserted into the current drawing. The insertion point in the current drawing is aligned with the insertion point defined in the BASE system variable in the referenced file.

Z

Specifies the *Z* coordinate value for the point at which an xref instance is inserted into the current drawing. The insertion point in the current drawing is aligned with the insertion point defined in the BASE system variable in the referenced file.

Scale

Specifies the scale factors for the selected xref.

Specify On-Screen

Displays prompts on the command line and makes the X, Y, and Z Scale Factor options unavailable. The descriptions of the options displayed on the command line match the descriptions of the corresponding options under Insertion Point in INSERT.

X Scale Factor

Specifies the *X* scale factor for the xref instance.

Y Scale Factor

Specifies the *Y* scale factor for the xref instance.

Z Scale Factor

Specifies the *Z* scale factor for the xref instance.

Uniform Scale

Ensures that the *Y* and *Z* scale factors are equal to the *X* scale factor.

Rotation

Specifies the rotation angle for the xref instance.

Specify On-Screen

Displays prompts on the command line and makes the Angle option unavailable. You are prompted for a rotation angle, as described in INSERT.

Angle

Specifies the rotation angle at which an xref instance is inserted into the current drawing.

Block Unit

Displays information about the block units.

Unit

Displays the specified INSUNITS value for the inserted block.

Factor

Displays the unit scale factor, which is calculated based on the INSUNITS value of the block and the drawing units.

XBIND

Binds one or more definitions of named objects in an xref to the current drawing

 Reference toolbar:
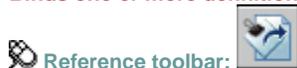
 Modify menu: Object ⮞ External Reference ⮞ Bind

 Command line: xbind

The Xbind dialog box is displayed.
If you enter **-xbind** at the Command prompt, XBIND displays prompts on the command line.

Note The Bind option of XREF binds the xref file. Use XBIND for individual dependent definitions.

XCLIP

Defines an xref or block clipping boundary and sets the front and back clipping planes

 Reference toolbar:

Shortcut menu: Select an xref, right-click in the drawing area, and choose Xref Clip.

 Command line: xclip

Select objects: *Use an object selection method and press ENTER* when you finish selecting objects

Enter clipping option [**ON**/**OFF**/**Clipdepth**/**Delete**/**generate Polyline**/**New boundary**] <New>: *Select an option or press* ENTER

On

On

Displays the clipped portion of the xref or block in the host drawing.

Off

Off

Displays all of the geometry of the xref or block in the host drawing, ignoring the clipping boundary.

Clipdepth

Clipdepth

Sets the front and back clipping planes on an xref or block. Objects outside the volume defined by the boundary and the specified depth are not displayed.

Specify front clip point or [Distance/Remove]:

Front Clip Point

Creates a clipping plane passing through and perpendicular to the clipping boundary.

Specify back clip point or [Distance/Remove]:

Distance

Creates a clipping plane the specified distance from and parallel to the clipping boundary.

Specify distance from boundary:

The previous prompt is displayed.

Remove

Removes both the front and back clipping planes.

Delete

Delete

Removes a clipping boundary for the selected xref or block. To temporarily turn off a clipping boundary, use the Off option. Delete erases the clipping boundary and the clipdepth. The ERASE command cannot be used to delete clipping boundaries.

Generate Polyline

Generate Polyline

Automatically draws a polyline coincident with the clipping boundary. The polyline assumes the current layer, linetype, lineweight, and color settings. Use this option when you want to modify the current clipping boundary using PEDIT and then redefine the clipping boundary with the new polyline. To see the entire xref while redefining the boundary, use the Off option.

New Boundary

New Boundary

Defines a rectangular or polygonal clipping boundary or generates a polygonal clipping boundary from a polyline. If a boundary already exists, the following prompt is displayed:

Delete old boundary? [Yes/No] <Yes>:

The command continues only if all previous boundaries are deleted.

Specify clipping boundary(s):

[Select polyline/Polygonal/Rectangular] <Rectangular>: *Specify a clipping boundary option or press* ENTER

Select Polyline

Defines the boundary using the selected polyline. The polyline can be open but must consist of straight line segments and cannot intersect itself. The boundary created using this method is parallel to the user coordinate system (UCS) plane on which the polyline lies.

Polygonal

Defines a polygonal boundary by using the points you specify for the vertices of a polygon. The clipping boundary is applied in the current UCS regardless of the current view.

Rectangular

Defines a rectangular boundary by using the points you specify for opposite corners. The clipping boundary is applied in the current UCS and is independent of the current view.

XLINE

Creates an infinite line

 Draw toolbar:

 Draw menu: Construction Line

 Command line: xline

Specify a **point** or [**Hor**/**Ver**/**Ang**/**Bisect**/**Offset**]: *Specify a point or enter an option*

Point

Point

Specifies the location of the infinite line using two points through which it passes.

Specify through point: *Specify the point (2) through which you want the xline to pass, or press* ENTER *to end the command*

The xline is created through the specified point.
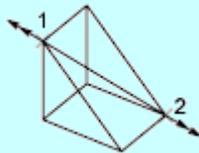
Hor

Hor

Creates a horizontal xline passing through a specified point.

    Specify through point: Specify the point (1) through which you want the xline to pass, or press
        ENTER to end the command

The xline is created parallel to the *X* axis.
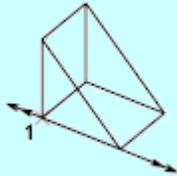
Ver

Creates a vertical xline passing through a specified point.

    Specify through point: Specify the point (1) through which you want the xline to pass, or press
        ENTER to end the command

The xline is created parallel to the *Y* axis.
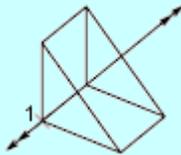
Ang

Creates an xline at a specified angle.

Enter angle of xline (0) or [Reference]: *Specify an angle or enter* **r**
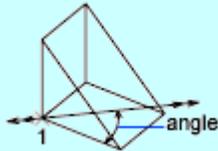
Angle of Xline

Specifies the angle at which to place the line.

Specify through point: *Specify the point through which you want the xline to pass*

An xline is created through the specified point, using the specified angle.

Reference

Specifies the angle from a selected reference line. The angle is measured counterclockwise from the reference line.

Select a line object: *Select a line, polyline, ray, or xline*

Enter angle of xline <0>:

Specify through point: *Specify the point through which you want the xline to pass, or press* ENTER *to end the command*

An xline is created through the specified point, using the specified angle.

Bisect

Bisect

Creates an xline that passes through the selected angle vertex and bisects the angle between the first and second line.

Specify angle vertex point: *Specify a point (1)*

Specify angle start point: *Specify a point (2)*

Specify angle end point: *Specify a point (3) or press* ENTER *to end the command*

The xline lies in the plane determined by the three points.
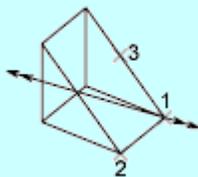
Offset

Offset

Creates an xline parallel to another object.

Specify offset distance or [Through] <*current*>: *Specify an offset distance, enter* **t**, *or press* ENTER

Offset Distance

Specifies the distance the xline is offset from the selected object.

Select a line object: *Select a line, polyline, ray, or xline, or press* ENTER *to end the command*

Specify side to offset? *Specify a point and press* ENTER *to exit the command*

Through

Creates an xline offset from a line and passing through a specified point.

Select a line object: *Select a line, polyline, ray, or xline, or press* ENTER *to end the command*

Specify through point: *Specify the point through which you want the xline to pass and press* ENTER *to exit the command*

XPLODE

Breaks a compound object into its component objects

⌨ **Command line:** xplode

Select objects to XPlode.

Select objects: *Use an object selection method and press* ENTER *when you finish selecting objects*

The program reports how many objects were selected and, of those, how many objects cannot be exploded. If you select more than one valid object, the following prompt is displayed.

Enter an option [Individually/Globally] <Globally>: *Enter* **i**, *enter* **g**, *or press* ENTER
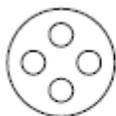
block before
XPLODE

component
objects after
XPLODE

Individually

Individually

Applies changes to the selected objects one at a time. The following prompt is displayed for each object.

Enter an option [All/Color/LAyer/LType/Inherit from parent block/Explode] <Explode>: *Enter an option or press* ENTER

All

Sets the color, linetype, lineweight, and layer of the component objects after you explode them. The prompts associated with the Color, Linetype, Lineweight, and Layer options are displayed.

Color

Sets the color of the objects after you explode them.

New color [Truecolor/COlorbook]<BYLAYER>: *Enter a color name or number from* **1** *through* **255***, enter* **t***, enter* **co***, enter* **bylayer** *or* **byblock***, or press* ENTER

Entering **bylayer** causes the component objects to inherit the color of the exploded object's

layer. Entering **byblock** causes the component objects to inherit the color of the exploded object.

Enter **t** for a true color to be used for the selected object.

Red, Green, Blue: *Enter three integer values from* **0** *to* **255** *separated by commas to specify a true color*

Enter **CO** for a color from a loaded color book to be used for the selected object.

Enter Color Book name: *Enter the name of a color book that has been installed, such as* **PANTONE**

If you enter a color book name, you are prompted to enter the color name in the color book.

Enter color name: *Enter the name of a color included in the selected color book, such as* **PANTONE573**

Layer

Sets the layer of the component objects after you explode them. The default option is to inherit the current layer rather than the layer of the exploded object.

Enter new layer name for exploded objects <*current*>: *Enter an existing layer name or press* ENTER

LType

Sets the linetype of the component objects after you explode them.

Enter new linetype name for exploded objects <BYLAYER>: *Enter a linetype name, or press* ENTER

You can enter the name of any linetype that is loaded in the drawing. Entering **bylayer** causes

the component objects to inherit the linetype of the exploded object's layer. Entering **byblock** causes the component objects to inherit the linetype of the exploded object.

Inherit from Parent Block

Sets the color, linetype, lineweight, and layer of the component objects to that of the exploded object if the component objects' color, linetype, and lineweight are BYBLOCK and the objects are

---

drawn on layer 0.

Explode

Breaks a compound object into its component objects exactly as the <u>EXPLODE</u> command does.

Globally

Globally

Applies changes to all the selected objects.

> Enter an option [All/Color/LAyer/LType/Inherit from parent block/Explode] <Explode>: *Enter an option or press* ENTER

The descriptions of the All, Color, Layer, LType, Inherit from Parent Block, and Explode options match the descriptions of the corresponding options under Individually.

XREF

Controls external references to drawing files

🕭 **Reference toolbar:**

🕭 **Insert menu:** Xref Manager

> **Shortcut menu:** Select an xref, right-click in the drawing area, and choose Xref Manager.

⌨ **Command line:** **xref**

The Xref Manager is displayed.

If you enter **-xref** at the Command prompt, XREF displays prompts on the command line.

Attaches, overlays, lists, binds, detaches, reloads, unloads, renames, and modifies paths to external references (xrefs) in the current (or host) drawing.

List of External References

Displays the xrefs in the drawing in a tree view or a list view. You can use the F3 and F4 keys to switch between list view and tree view. Selecting any field selects the xref.

List View

Displays a flat listing of the attached xrefs and their associated data. You can sort the list of references by name, status, type, file date, file size, or the saved path and file name.

Reference Name

Lists the names of the xrefs as stored in the definition table for the drawing.

Status

Shows whether the xref is loaded, unloaded, not found, unresolved, orphaned, or marked for unloading or reloading.

- *Loaded:* Currently attached to the drawing.
- *Unloaded:* Marked to be unloaded from the drawing once the Xref Manager is closed.

- *NotFound:* No longer exists in the valid search paths.
- *Unresolved*: Cannot be read by this program.
- *Orphaned:* Attached to another xref that is unresolved, or not found.

Size

Shows the file size of the corresponding reference drawing. The size is not displayed if the xref is unloaded, not found, or unresolved.

Type

Indicates whether the xref is an attachment or an overlay.

Date

Displays the last date the associated drawing was modified. This date is not displayed if the xref is unloaded, not found, or unresolved.

Saved Path

Shows the saved path of the associated xref (this is not necessarily where the xref is found).

Tree View

Displays a hierarchical representation of the xrefs, displaying the relationships between xref definitions. Tree view shows the level of nesting relationship of the attached xrefs, whether they are attached or overlaid, and whether they are loaded, unloaded, marked for reload or unload, or not found, or unresolved.

Attach

Displays the External Reference dialog box if an external reference is selected or displays the Select Reference File dialog box if no external reference is selected. See XATTACH.

Detach

Detaches one or more xrefs from your drawing, erasing all instances of a specified xref and marking the xref definition for deletion from the definition table. Only the xrefs attached or overlaid directly to the current drawing can be detached; nested xrefs cannot be detached. An xref referenced by another xref or block cannot be detached.

Reload

Marks one or more xrefs for reloading. This option rereads and displays the most recently saved version of the drawing.

Unload

Unloads one or more xrefs. Unloaded xrefs can be easily reloaded. Unlike detaching, unloading does not remove the xref permanently. It merely suppresses the display and regeneration of the xref definition to improve performance.

Bind

Displays the Bind Xrefs dialog box. The Bind option makes the selected xref and its dependent named objects (such as blocks, text styles, dimension styles, layers, and linetypes) a part of the current drawing.

Open

Opens the selected xref for editing in a new window. The new window is displayed after the Xref Manager is closed.

Xref Found At

Displays the full path of the currently selected xref. This is where the xref is actually found and is not necessarily the same as the saved path.

Browse

Displays the Select New Path dialog box (a standard file selection dialog box), in which you can select a different path or file name.

Save Path

Saves the path, as it appears in Xref Found At, to the currently selected xref.

ZOOM

Increases or decreases the apparent size of objects in the current viewport

⊗ **Standard toolbar:**

⊗ **View menu:** Zoom

Shortcut menu: With no objects selected, right-click in the drawing area and choose Zoom to zoom in real time.

⌨ **Command line: zoom** (or **'zoom** for transparent use)

Note You cannot use ZOOM transparently during VPOINT or DVIEW or while ZOOM, PAN, or VIEW is in progress.

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale /Window/Object] <real time>:

All

Zooms to display the entire drawing in the current viewport. In a plan view, All zooms to the grid limits or current extents, whichever is greater. In a 3D view, ZOOM All is equivalent to ZOOM Extents. The display shows all objects even if the drawing extends outside the grid limits.
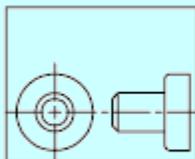
before ZOOM All   after ZOOM All

In the illustration, LIMITS is greater than the extents of the drawing.

Because it always regenerates the drawing, you cannot use ZOOM All transparently.

Center

Zooms to display a window defined by a center point and a magnification value or height. A smaller value for the height increases the magnification. A larger value decreases the

magnification.

Specify center point: *Specify a point (1)*
Enter magnification or height *<current>*: *Enter a value or press* ENTER
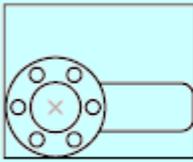
before ZOOM center
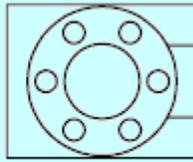
after ZOOM center,
magnification increased

Dynamic

Zooms to display the generated portion of the drawing with a view box. The view box represents your viewport, which you can shrink or enlarge and move around the drawing. Positioning and sizing the view box pans or zooms to fill the viewport with the image inside the view box.
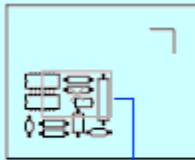
view box
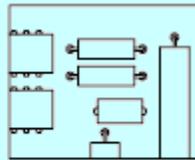
new view

The panning view box is displayed first. Drag it to the location you want and click. The zooming view box is then displayed. Resize it and press ENTER to zoom, or click to return to the panning view box.

Press ENTER to fill the current viewport with the area currently enclosed by the view box.

Extents

Zooms to display the drawing extents and results in the largest possible display of all the objects.

before ZOOM Extents
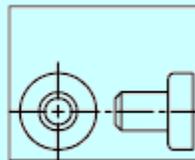
after ZOOM Extents

Previous

Zooms to display the previous view. You can restore up to 10 previous views.
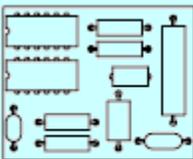
original view
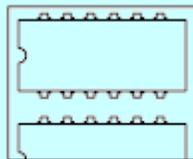
current view
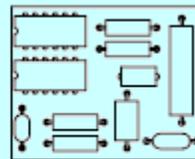
after ZOOM Previous

Note If you change the shading with the SHADEMODE command, the view is changed. If you enter ZOOM Previous after changing the shading, it restores the previous view, which is shaded differently, but not zoomed differently.

Scale

Zooms the display at a specified scale factor.

Enter a scale factor (nX or nXP): *Specify a value*

Enter a value followed by **X** to specify the scale relative to the current view. For example, entering

.5x causes each object to be displayed at half its current size on the screen.

ZOOM .5x

Enter a value followed by **xp** to specify the scale relative to paper space units. For example,

entering **.5xp** displays model space at half the scale of paper space units. You can create a layout with each viewport displaying objects at a different scale.

Enter a value to specify the scale relative to the limits of the drawing. (This option is rarely used.) For example, entering **2** displays objects at twice the size they would appear if you were zoomed to the limits of the drawing.

ZOOM 2

Window

Zooms to display an area specified by two opposite corners of a rectangular window.

Specify first corner: *Specify a point (1)*
Specify opposite corner: *Specify a point (2)*

before ZOOM Window   after ZOOM Window
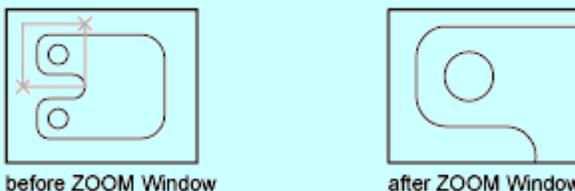
Object

Zooms to display one or more selected objects as large as possible and in the center of the drawing area. You can select objects before or after you start the ZOOM command.

Real Time

Using the pointing device, zooms interactively to a logical extent.

Press ESC or ENTER to exit, or right-click to display the shortcut menu

The cursor changes to a magnifying glass with plus (+) and minus (–) signs. See Zoom Shortcut Menu for a description of the options that are available while zooming in real time.

ZOOM Realtime cursor

The current drawing area is used to determine the zooming factor. ZOOM uses half of the window height to move to a zoom factor of 100%. Holding down the pick button at the midpoint of the window and moving vertically to the top of the window zooms in to 100%. Conversely, holding the pick button down at the midpoint of the window and moving vertically to the bottom of the window zooms out by 100%.

Note If you place the cursor at the bottom of the window, hold down the pick button, and move vertically to the top of the window, the zoom-in factor is 200%.

When you have reached the zoom-in limit, the plus sign in the cursor disappears, indicating that you can no longer zoom in. When you have reached the zoom-out limit, the minus sign in the cursor disappears, indicating that you can no longer zoom out.

When you release the pick button, zooming stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again and continue to zoom the display from that location.

To exit zooming at the new position, press ENTER or ESC.

Raster Command Reference

This section defines the GTXImage CAD series commands and features. The first few pages include new and changed commands, a set of overview tables listing each command and feature with icon and brief description. Read detailed command descriptions through the rest of the section, indexing along the top margin of the page to find each command name.

Command Overview

Icon	Command	Short	Description
	gACLEAN	GACL	Automatically deskews and deletes noise in the active image.
	gACTIVE	GAC	Prepares a selected image as the active image, ready for raster editing or conversion
	gADESKEW	GAD	Automatically rotates the current raster image to align raster to an orthogonal orientation.
	gARC		Draws a raster arc in the current raster image
	gARRAY	GAR	Creates a circular or rectangular array from selected raster data
	gATTACH	GA	Quickly loads raster images
	gAUTOSNAP	GAS	Enables AutoSnap™ options within active image entities
	gBITONAL	GBIT	Automatically reduces a color image to a binary (1-bit) image
	gBURN	GBU	Converts all vectors to raster, "burning" them into the current raster image
	gCALIKE	GCA	Changes circles and arcs to have the same radius as a selected circle or arc

	gCARC		The gCARC command stretches arcs
	gCCIRCLE		The gCCIRCLE command stretches raster circles.
	gCCIRCLE2P		Change circle – two point pick.
	gCELEV	GCE	Assigns incremental elevations to multiple polylines
	gCHANGE	GCH	Changes the size or position of a line, arc or circle
	gCHGLAY	GCHL	Changes the layer of objects to that of a specified object
	gCIRCLE	GC	Draws a raster circle in the current raster image
	gCLEANIOP		Automatically deskews the active image and places speckles and other noise in a selection set
	gCLINE		The gCLINE command stretches raster lines.
	gCONCEN	GCONC	Moves circles and arcs to be concentric about a location or about the centerpoint of a specified circle or arc
	gCONVRT	GCO	Converts selected raster data to vector
	gCOPY	GCP	Copies selected raster data to a new location on the image, retaining the original
	gCREATE	GCR	Creates a new raster database or flush the current raster database
	gCROP	GCRO	Deletes all information outside of a specified window
	gCTEXT		Change raster text.
	gCUT	GCU	Copies selected raster data to a file or buffer

	gDESKEW	GD	Aligns raster data in current raster image to 0 and 90 degrees
	gDETACH		Removes the selected image
	gDRO_BK		Sets all image entities' draw order behind other entities
	gECONVRT	GEC	Creates a vector edge around selected raster
	gEDGE	GED	Creates a raster edge around selected raster
	gERASE	GE	Removes selected raster data
	gFRZLAY	GFR	Freezes the layer of a specified object
	gHELP	GH	Launches the online help
	gHTEXT		Edits hybrid text
	gHYBRIDMODE		Creates vector over GTX drawn raster objects
	gINACTIVE	GINA	Deactivates the currently active image, freeing memory
	gINFO	GI	Displays information about the raster database
	gINVERT	GIN	Reverses the raster foreground and background data Useful for raster scanned from negative images
	gISOLAY	GISL	Freezes all layers except that of a selected object
	gJOIN	GJO	Joins lines
	gLINE	GLN	Draws lines on current raster image
	gLINE2		Draws multiple separated lines
	gMIRROR	GMI	Mirrors selected raster data
	gMCOPY		Makes multiple copies of selecter raster

	gMOVE	GM	Moves selected raster data
	gOFFSET	GOFF	Creates concentric raster circles, parallel raster lines and parallel raster arcs
	gPASTE	GP	Pastes raster data from a raster file or a buffer created with gCUT
	gRAHEAD	GRAH	Erases "arrowheads" (solids with three points)
	gRASTER	GRA	Converts selected vector entities to raster
	gRECTANGLE		Draws a raster rectangle
	gREDUCE	GRED	Reduces a multi-color image to a more
	gREFLCT	GREF	Mirrors the entire image in either the X or Y axis, replacing the original image
	gRELIMIT	GREL	Combines trim and extend for lines
	gRESIZE	GRE	Resizes the image entity (Adds or removes pixels without scaling the image entity)
	gROTATE	GR	Rotates selected raster data
	gROTRECTANGLE		Draws a raster rectangle at any rotated orientation
	gRUB	GRU	Erases raster data under selected vector data
	gSAVE	GSA	Saves the raster image
	gSCALE	GS	Scales selected raster data
	gSEPARATE	GSEP	Separates colors to discrete bitonal images
	gSLICE		Creates a 2 pixel wide gap between raster objects
	gSMOOTH	GSM	Smoothes selected raster

	gSPECKL	GSP	Perform raster object speckle removal to selectively erase background noise or fill in holes in the raster image
	gTCONFIG	GTCFG	Configures Text Window IOP, Text Line IOP and the gTCONVRT/gTRAIN commands
	gTCONVRT	GTC	Converts raster text to CAD text entities
	gTEXT		Types raster text aided by the text formatting control
	gTRACE	GT	Traces vector lines, arcs, and circles over selected raster geometry (raster is preserved)
	gTRAIN	GTR	Previews and edits text recognition files used by gTCONVRT
	gTRANS_ON		Makes all images transparent
	gTURN	GTU	Reorients the work area (and image) by 90, 180 or 270 (-90) degrees
	gVECTOR	GV	Traces raster to vector lines, arcs and circles
	gVSKEW	GVS	Deskews vector data
	gWARP	GW	Corrects distortions in an image
	ImageFrame		Configures the Image Frame
	ImageTransparency		Controls image transparency

Raster Picking Options

Most commands affect a subset of the raster image. To gMOVE a raster circle, you need to first pick that circle. The GTXRaster CAD series uses Intelligent Object Picking (IOP) to pick raster entities from the drawing as if they were intelligent CAD entities. These options can be typed, found in the GTX Edit>Raster Pick, the toolbar or the screen menu.

+ ADD - Add data to the selection set (after using "remove")

— Remove - Remove selected data from the selection set

ALL - Select all raster on the current raster image

Window - Select all raster within a rectangular window selected by two pick points

PWindow - Select all raster within a polygon window selected by three or more vertices

CRossing - Select connected raster objects that are completely contained by or crossing the edge of a window selected by two pick points

PCrossing - Select raster that is completely contained by or crossing the edge of a polygon window selected by three or more vertices

Inside - Select raster objects completely contained by a window selected by two pick points

Outside - Select all raster outside a rectangular window selected by two pick points

PInside - Select connected raster objects that are completely contained by a polygon selected by three or more vertices

RCrossing - Select raster that is completely contained by or crossing the edge of a

rotatable rectangular window selected by two baseline points and a height point

RInside - Select raster objects completely contained by a rotatable rectangular window selected by two baseline points and a height point

RWindow - Select all raster within a rotatable rectangular window selected by two baseline points and a height point

Fence - Select connected raster objects that are intersected by a polyline of two or more pick points

Object - Select contiguous raster data within the current view by a single pick point

ARc - Select raster arcs under a reference arc defined by three pick points

2 Pick Circle - Select raster circle using a 2-point reference circle

CIrcle - Select raster arcs or a circle under a reference circle defined by three pick points

Line - Select raster lines under a reference line defined by two pick points

PRevious - Pick the objects in the previous selection set

Segment - Selects a linear raster segment in both directions from a single pick point until it stops or finds an acute angle or a branch

AllText - Selects all text and text-like raster in the image below the height defined in the GTX Text Configuration maximum character size. Note This can also be used to remove text from a selection set.

TEXTLine - Select text-sized raster elements that are intersected by a reference line. Separates text from touching raster elements

TEXTWindow - Select raster text elements that are within a rectangular window. Separates text from touching raster elements

unDer - Pick raster under selected vector objects

Undo - Remove data from the selection set added by the last pick made

View - Select all raster data within the present view

Working with GTX's Intelligent Object Picking (IOP's)

Although GTX's IOP's are not commands in their own right, they are operators that allow you to precisely select the raster data that you wish to work with. IOP's are fundamental to the effective use of GTX and users should spend time to understand how the various IOP's work.

To better understand the IOP operation we have highlighted some of the ways in which you can use the IOP's for practical raster selection. The GTX IOP's work with most of the raster commands, but for this example we have used the raster erase command – gERASE.

Start by loading the GTX sample file called GTXVBELT.CG4 then initiate the gERASE command & try the following IOP selections. Note that in these examples RED is the normal (passive) Raster & GREEN is the selected areas.

Window IOP's

Firstly there are six Window selection IOP's – **Window** (Implied Cutting Window), **Crossing** Window & **Inside** Window. Using rectangular pick or a rotated baseline and height pick on the raster:

_____
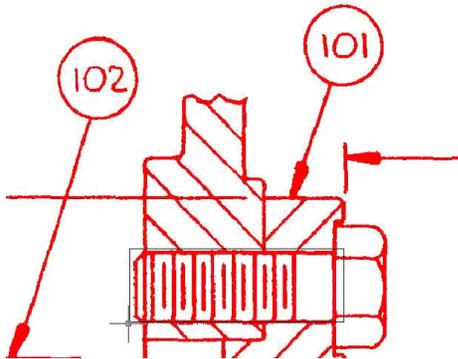
 Using the **IOP Window** (Cutting) option the raster selection would pick:
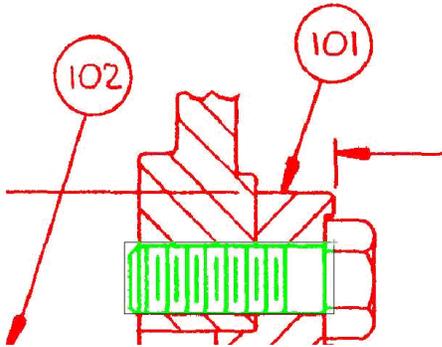
It only selects the cut rectangular window area.

Using the **IOP Crossing** (Window) the raster selection would highlight:
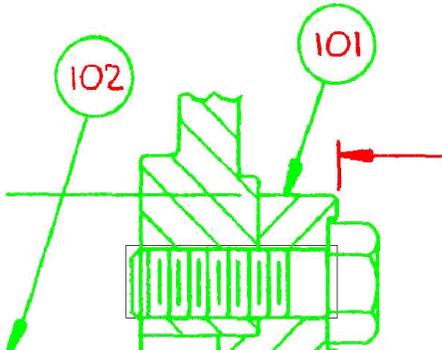
The Crossing IOP extends to include all crossing raster elements.

This is ideal for quickly picking up large areas of the drawing especially views & sections. Isolating text is often an advantage & note how the text is not selected.

_____

 Using the **IOP Inside** (Window) the raster selection would select:
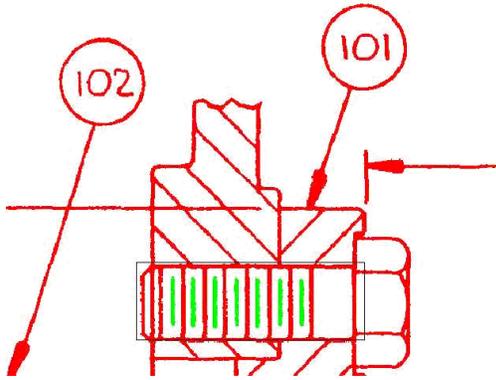
Inside is great for only picking elements totally within the selected Window.

It's Ideal for selecting text, broken lines & parts list items.

_____

 IOP Rwindow to select all raster contained within a rotatable rectangular window selected by two baseline points and a height point
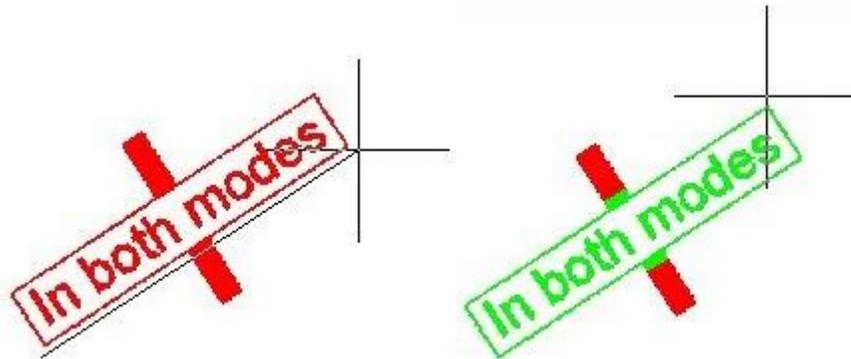
Select two points of baseline   Select Rwindow height

IOP Rcrossing Select raster that is completely contained by or crossing the edge of a   rotatable rectangular window selected by two baseline points and a height point
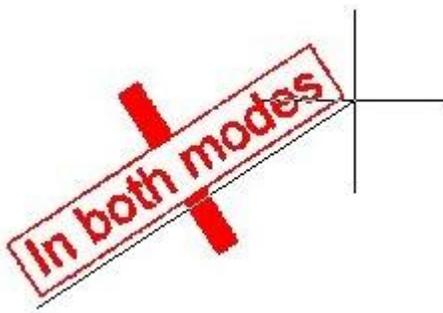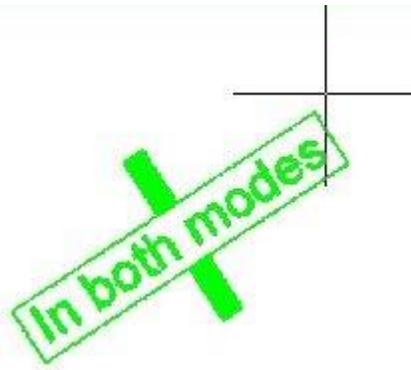
Select two points of baseline   Select Rcrossing height

 IOP RInside Select raster objects completely contained by a rotatable rectangular window selected by two baseline points and a height point
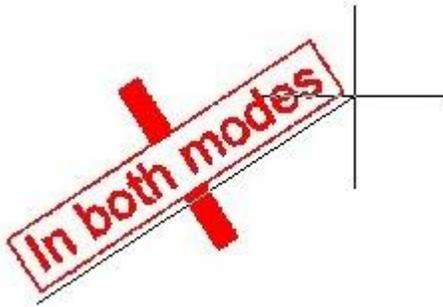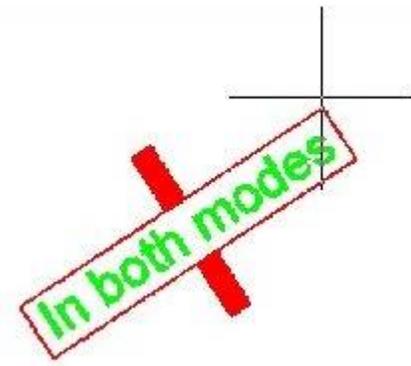
Select two points of baseline   Select Rinside height

_____

Polygon Window IOP's

Next there are three Polygon Area selection IOP's – **Pwindow** (Implied

Cutting area), **Pcrossing** & **Pinside.**

They are similar in operation to the Rectangular Window selections only you select a freehand polygon. Clicking Right Mouse button closes the polygon. Example of Pwindow:

_____
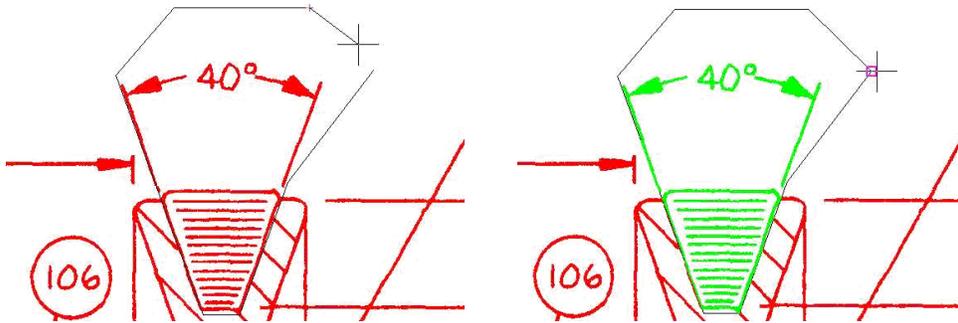
Element Selection IOP's – **Line**, **Circle** & **Arc**

 Using **IOP Line** it is possible to pick or move a raster line which crosses other raster data. The key benefit is that it leaves any crossing intersections intact & you do not need to repair any intersecting raster.

Just select two points over the raster line you wish to select:
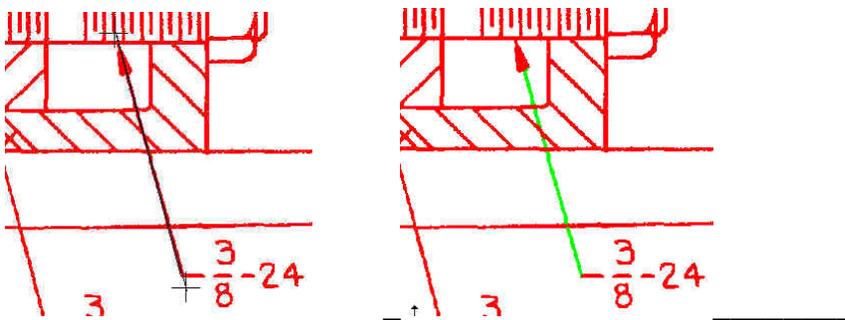
Similarly using **IOP Circle** it is possible to pick or move a raster circle which crosses other raster data. Again it leaves the intersections intact & can greatly reduce editing time. There are two IOP circle options:

 Three Point IOP circle selection - or **Two Point IOP** circle selection (with bounding circle). Below is a two point IOP Circle example:

_____
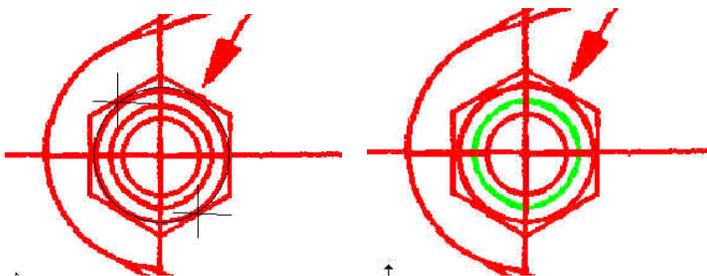
In the same way using **IOP arc** it is possible to isolate or move a raster arc which crosses other raster information. You need to select three points to identify the arc:

_____
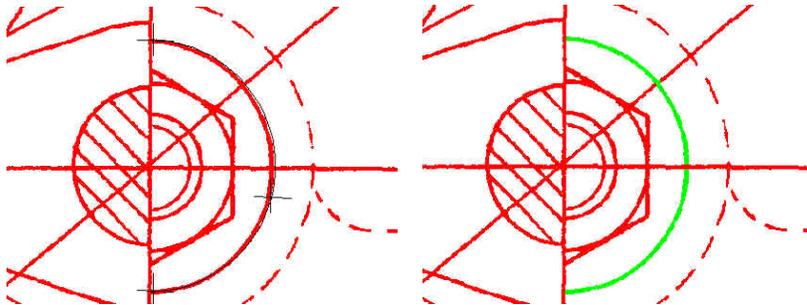
IOP Text Selection

There are three IOP selections that will isolate text. All commands use the text height settings in the GTX config file (GTXRAST.CFG) or if you have the GTXRaster CAD PLUS module you can change the sizes in the TEXT Recognition Config dialog (gTCONFIG).

IOP Textwin selects text within a given window.

_____

IOP Textline selects text intersecting a drawn two point line.

_____

390
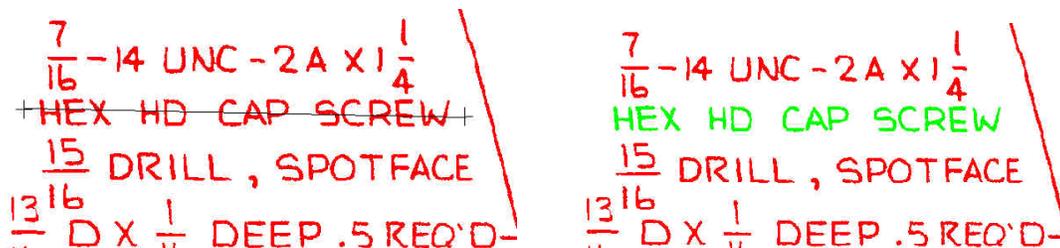
123 **IOP Alltext** is a very powerful selection that identifies text over the whole drawing with just one pick!

_____
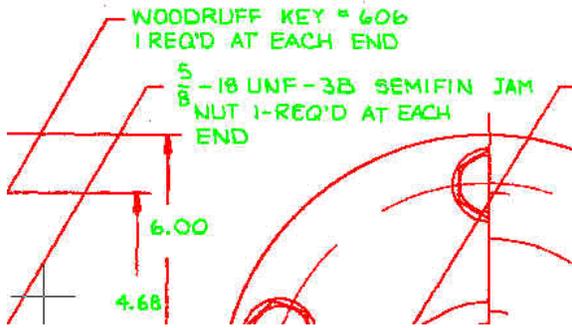
Other useful IOP selections:

IOP Fence is very useful for selecting drawing data when converting & smoothing. With careful use of IOP Fence you can quickly pick complete views but not select text. Ideal for selecting parts lists & drawing borders.

_____
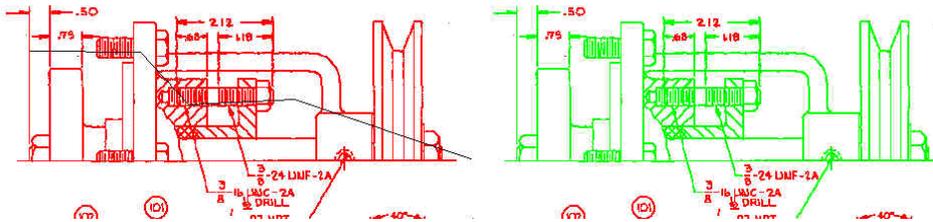
IOP Object allows you to pick a single connected piece of raster data with one pick.

_____

IOP Segment is ideal for picking free running raster lines. The selection stops at logical intersections. This can be very useful when working on maps to isolate specific contour lines.
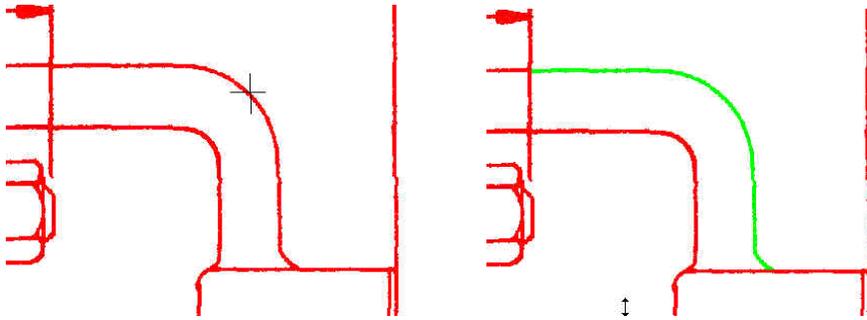
Raster Object Snapping Options

The GTXImage CAD series provides transparent "raster snaps". While using a CAD or Raster command, select Control-Right-mouse button to choose raster snaps from a pop-up menu.

In addition to transparent raster snaps, the GTXImage CAD series includes the **gAUTOSNAP** command, enabling **Endpoint**, **Intersection**, **Nearest** and **Node** AutoSnap™ methods within active raster entities. The following table lists each raster snap option.

Icon	Name	Keyboard	Description
	None	'rnone	Disables previous snap mode settings
	Endpoint	'rend	Snaps to the nearest end of a raster entity
			Endpoint activates as running snap under AutoCAD's End AutoSnap™.
	Intersection	'rint	Snaps to the intersection of two or more raster entities
			Intersection activates as running snap under AutoCAD's Intersection AutoSnap™.
	Nearest	'rnea	Snaps to the centerline of the nearest raster object
			Nearest activates as running snap under AutoCAD's Nearest AutoSnap™.
	Edge	'redge	Snaps to the edge of the nearest raster object.
			Edge activates as running snap under AutoCAD's Node AutoSnap™.
	Center	'rcen	Snaps to the center of a circle or arc defined by three raster points
	Tangent	'rtan	Snaps to the point that, with the previous point, makes a line tangent to a circle or arc defined by three raster points. The first point defines the side of the circle or arc for the tangent
	Midpoint	'rmid	Snaps to the midpoint of a line defined by two raster points
	MidArc	'rmida	Snaps to the midpoint of a raster arc defined by the endpoints and a circumference point
	Perpendicular	'rper	Snaps to a point where the present entity being drawn will intersect perpendicular to a line defined by two points
	Perp-Arc	'rpera	Snaps to the point perpendicular to an arc defined by three raster points

Command Definitions

This section defines each command. The GTXImage CAD variables control the use of dialog boxes or prompts for many commands. FILEDIA controls dialogs for file commands and CMDDIA controls dialogs for other commands. Setting these variables to "1" enables the dialogs. Setting the variables to "0" disables dialogs and causes these commands to use GTXImage CAD command line prompts.

gACTIVE (Active Image)

Type	Menu	Availability
GAC	GTX Imaging>Active Image	All Products

Icon:

Every raster editing/enhancing command requires an active image to be specified. The gACTIVE command is invoked automatically when there is none specified, so you will not explicitly invoke this command unless you are working on more than one image at a time and wish to switch from one to another.

gACTIVE sets active images to transparent.

Options:

Number of Images

Note: Invoke gINACTIVE to make an active image inactive and save memory. CLIPPING only masks the DISPLAY of images. When editing a clipped image, the entire image is being edited (selections include clipped portions of an image, but after the edit, the clipping is still active).

The behavior of **gACTIVE** varies depending on the number of images currently attached to the drawing:

One Image: If there is only one image currently attached, **gACTIVE** will automatically set it as the active raster image.

Two or More Images: If there are two or more images, a prompt will request that you select the one to edit.

Image Type

gACTIVE behaves differently depending on the number of images attached.

Bi-Tonal: Bi-tonal images are activated quickly and easily.

Color or Grayscale: Color or grayscale images are reduced to bi-tonal (two colors) before becoming active. The color or grayscale image is not modified; rather, the command generates a bitonal version of the original.

gADESKEW

Type	Menu	Availability
GAD or GADESKEW	GTX Enhance>Auto Deskew	GTXImage CAD GTXImage CAD PLUS

Icon:

Drawings are often scanned in slightly rotated. gDESKEW straightens selected raster by rotating it to align with the X or Y axis. gDESKEW rotates selected raster so a selected line on the drawing becomes "square". The *AutoDeskew* configuration parameter causes the gATTACH command to automatically deskew upon loading a new raster image.

Instructions:

1. To deskew entire image, select gADESKEW.

2. The deskew will take place, based on long raster lines found in the margin of the raster image. If no such lines are found, they will not deskew.

gARC

TYPE	MENU	AVAILABILITY
gArc	GTX Raster Edit >Raster Arc	GTXImage CAD GTXImage CAD PLUS

Icon:

Draws a raster arc in the current raster image

Instructions:
From point:
Select or type the first endpoint.
To point:
Select or type a second endpoint.
To point:
Select or type a third endpoint.
The arc appears after the third point is specified.

Note: If there is no active image, gACTIVE will be invoked before continuing with gARC. If there is no image attached gCREATE will be invoked to set up an image into which raster arc can be drawn.

gARRAY

Type	Menu	Availability
GAR or GARRAY	GTX Raster Edit >Array	GTXImage CAD
		GTXImage CAD PLUS

Icon:

gARRAY copies raster data in rectangular or polar arrays. When creating a rectangular array, gARRAY arranges multiple copies of the selected raster in rows and columns. When creating a polar array, the command arranges copies of the selected raster placed about a circle.

Instructions:

1. Invoke gARRAY.

```
Select raster <Window>:
```

2. Select the raster data to be copied. Use any valid raster selection method.

3. Press **[RETURN]**.

```
Rectangular or Polar array (R/P) <R>:
```

4. Enter **R** for rectangular or **P** for polar.

Rectangular

If using a rectangular array you must specify the number of rows and columns to be copied.

```
Number of rows (---)<1>:
```

a. Enter the number of rows you desire.

```
Number of columns (||||)<1>:
```

b. Enter the number of vertical columns you desire.

```
Unit cell or distance between rows (---):
```

c. You may enter a vertical distance numerically or draw a "unit cell" representing the distances between rows and columns graphically. If you enter a distance numerically then press **[RETURN]**, you will receive the following prompt:

```
Distance between columns (||||):
```

d. Enter the distance that will separate the columns. The selected raster is copied.

Note: Positive x or y distances places copies to the right or above the original. Negative distances places them to the left or below the original.

If you drew a unit cell instead of typing a distance, a rectangular box will rubber-band from the anchor point. Drag the box to represent the distance between both the rows and columns. The selected raster is copied.

Polar

If using a polar array you must specify the center point of the array, the number of copies, the angle to fill and whether or not to rotate the raster data as it is copied around the polar array.

```
Center point of array:
```

a. Select a point that represents the axis around which all selected data will be copied.

```
Number of items:
```

b. Enter the number of items that will fill the angle you will specify.

```
Angle to fill (+=ccw,-=cw)<360>:
```

c.Enter an **integer** value representing the angle over which the copies will be evenly distributed.

```
Rotate objects as they are copied? <Y>:
```

d. Respond **Y** to rotate the objects as they are copied. This will keep the raster objects in the same orientation to the center point as they are copied. Responding **N** will keep the raster objects in the same orientation as the original raster object selected.

The raster data is copied.

gATTACH

Type	Menu	Availability
GA, GATTACH	GTX Imaging>Attach Raster File	All Products

Icon:

gATTACH loads an image into GTXImage CAD with fewer steps than used in the IMAGE command. Instructions:

Select **gATTACH**.

The **gATTACH** file dialog will appear.

Select the appropriate file type in the "List Files of Type:" combo-box, select the directory in which the file resides. All files of the indicated type will appear in the "File Name" box. Select the file you wish to load and press OK.

Loading a Raster File

The image is attached at 0,0 and at the orientation specified by the AUTOROTATE configuration parameter.

Raster Resolution

If the GTXRAST.CFG setting is "FILE" the file's resolution will be used, but if a specific resolution is set in GTXRAST.CFG, it will override that of the file.

Color Images

There is a difference in the way the gATTACH handles & opens color or black & white (binary) images. If you use the gATTACH command to open a 'color image' you will additionally be asked how you wish to pre-process the color image.
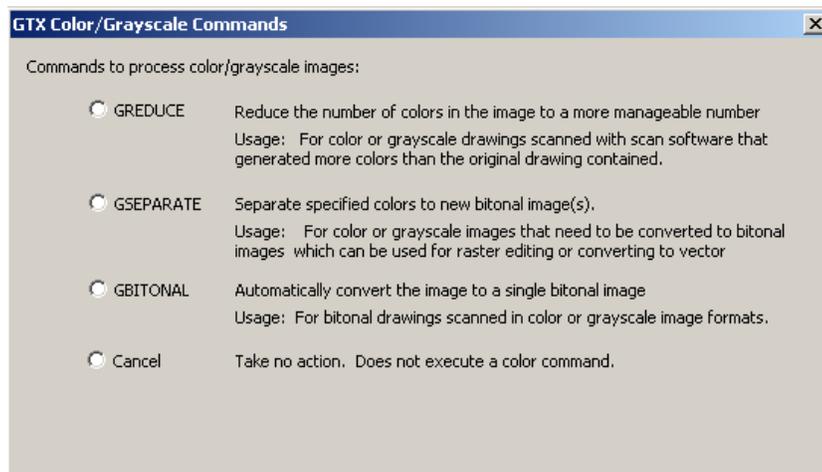
The best command to use depends mainly upon the number of colors that your image has.

gBITONAL

If you wish to automatically reduce a color image down to a simple binary image you can use

---

gBITONAL The gBITONAL command has been optimized to operate on a broad range of color images & works well where an image has a clear-cut distinction between dark & light data.

However, the gBITONAL threshold may not give the exact results that you desire (by including or omitting important data from the binary image). In these instances it may be better to use the gREDUCE & gSEPARATE commands, where you have greater control of the color separation & threshold.

gREDUCE

If you have an image with greater than 256 colors you can firstly use gREDUCE to bring the image down to 256 colors or less.

gSEPARATE

For images with colors less that 256 colors, either before or after using reduction, you can use the gSEPARATE command. This allows you to accurately select colors, or groups of colors, & break then out into separate binary color images.

These separate binary images can, if required, be further processed using the full range of GTX capabilities & commands detailed elsewhere in this help file.

For more detailed information about each command's operation refer to the respective gBITONAL, gREDUCE & gSEPARATE section in the 'Command Reference'.

Cancel

Cancel does no pre-processing & just loads the image into AutoCAD. In this state you cannot work on the color image using GTX commands.

gAUTOSNAP

Type	Menu	Availability
GAS, GASP, GASNAP or GAUTOSNAP	GTX Raster Edit>Raster Snaps>Raster Auto Snap	GTXImage CAD GTXImage CAD PLUS

Icon:

gAUTOSNAP enables GTXImage CAD's AutoSnap™ options within an active image.

Details:

Instructions:

GAUTOSNAP: <0>,1:

1. Type **1** to raster-enable AutoSnap™.

2. Invoke **OSNAP** command. The dialog includes a Select Settings section.

3. Select the snap modes you wish to enable. The Endpoint, Intersection, Intersection and Node settings activate GTX raster snaps. Because GTXImage CAD does not provide "Edge" snapping, its Node snap is used to activate running edge raster snap.

CAD Object Snap	Raster Snap
Endpoint	Endpoint
Intersection	Intersection
Nearest	Nearest
Node	Edge

Select **OK**.

The aperture box will appear around the crosshairs and the marker will snap to raster features.

Note: AutoSnap™ uses glyphs (symbols) to represent the snap type. GTX running snaps use ToolTip labels to identify the snap being used. Snaps work instantaneously even though the ToolTips appear after a short delay.

gAUTOCLEAN

Type	Menu	Availability
GACL	GTX Enhance>Auto Clean	GTXImage CAD
GACLEAN	Raster	GTXImage CAD PLUS

Icon:

gAUTOCLEAN automatically deskews the image and removes noise and speckles without forming a selection set; it is completely automatic. Another command, GCLEANIOP is interactive, forming a selection set that can be modified before erasing the noise.

Instructions:

Issue the Clean IOP command. The program will automatically deskew the image and delete noise.

Note: If you are concerned about losing information from your drawing, you may use gCLEANIOP instead

gBITONAL

Type	Menu	Availability
gBITONAL, gBIT	GTX Convert>Convert Image to Bitonal	All Products

Icon:

gBITONAL automatically reduces a color image to a binary image.

The gBITONAL command only works when you have one or more color (or greyscale) images loaded.

Instructions:

Issue the gBITONAL command. The program will then automatically reduce a multi-color image to a binary image.

If you have no color/greyscale images loaded & you activate the command you will get an error telling you that you don't have a suitable image for gBITONAL to work on.

If you have more than one color/greyscale image loaded you will be offered a dialog box to select the image you wish make bitonal.

Notes:

gBITONAL only works if you have a color (or greyscale) image loaded.

On large color images the gBITONAL command can take some time to process.

The gBITONAL algorithm has been optimized to operate on a broad range of color images. As such the command works well where an image has a clear-cut distinction between dark & light data.

However, the gBITONAL threshold may not give the exact results that you desire (by including or omitting important data from the binary image). In these instances it may be better to use the gSEPARATE command, where you have greater control of the color separation & threshold.

gBURN

Type	Menu	Availability
GBU or GBURN	GTX Convert>Burn All to Raster	GTXImage CAD GTXImage CAD PLUS

Icon:

gBURN converts all vector entities into raster data with a designated line width.

Note: The raster database must be established before gBURN can be used. The database can be established either by loading a raster file or by using gCREATE. If the database has not been established, you will see a message and gCREATE will be invoked automatically.

Only entities within the raster limits can be rasterized! Those outside the limits will be ignored or erased. When entities are partially outside, the part inside is rasterized; the part outside is erased.

GTXImage CAD will not rasterize AutoCAD 3d Objects.

GTXImage CAD will not rasterize hatches with gradient/fill pattern colors.

Options:

Width Options

Table   Use the width settings according to color as set in the Width Table

Fixed   Type in a value or select the width graphically using **Box** or **Pick**.

Instructions:

Dialog

1. Invoke **gBURN**. The following dialog will appear.

2. To specify line width you can a) enter a **Fixed** line width, b) specify it graphically by drawing a **Box** or by using **Pick** to select raster in the drawing of the desired width, or c) use a **Table** to correlate the colors of vectors to raster line widths.

Fixed Line Width:

a. Select the **Fixed** radio button. Type the raster width to be used (in CAD units, not pixels), or use the Pick or Box option to graphically specify the width.

Using Box:

a. Select the **Box** button.

```
Select first corner of box:
```

b. Select a point.

```
Select opposite corner of box:
```

c. As you move the cursor, a rubber-banding rectangle appears with one corner at the point selected. Move the cursor until the box is the desired width and the length exceeds the width. Select the point.

The best way to use this feature is to pick a point on the edge of an existing horizontal or vertical raster line and stretch the box along the line. You can then easily match the width. The length of the rectangle can be any size that is longer than the width as the smallest side is used as the width measurement. A reference rectangle is used instead of a square to make it easier to see. As soon as the second point has been selected, the rasterization takes place.

Using Pick

a. Select Pick.
```
Select raster <WIDTH>:
```

b. Select a piece of raster data of the desired raster width. The width of the selected raster will be entered in the "Fixed" text box.

Using the LineWidth Configuration Table:

a. Select the **Table** radio button and the **Table Setup** button. The LineWidth Configuration Table appears, correlating CAD color numbers and line widths expressed in current units.

b. To change the width associated with a particular CAD color, select that ACAD Color row. Enter the value in the **Width** text field. Select **OK** to close the dialog or select another row to change its width.

Line Width Configuration Dialog
```
Enter width <.XX>:
```

c. Enter the value for the width in the units being used and press **[RETURN]**. The Width Table will reflect the changes.

d. Select another color number to change its width or press [0] and **[RETURN]** to continue the command.

The entities will be changed to raster with the widths defined by their colors.

The highlighted entities will be converted to raster. Some residual vector markings overlay the raster until you redraw the image.

Prompts

1. Invoke **gBURN**.
```
Raster width Box/Pick/<Table>/.01:
```

2. There are three ways to respond to this prompt. You can 1) type in a value as the width to be used for all vector lines to be rasterized for plotting; 2) use the LineWidth Configuration Table to correlate CAD color numbers with line widths; or 3) use the Box or Pick options to specify a standard linewidth graphically (as described above).

Fixed Line Width:

a. You can type in a value (in mm or inch) as the width to be used for all the vector lines to be rasterized. The value in parentheses is the default value or one previously used. If you enter a value it will remain the default line width during this editing session.

Note: If you defined the width previously in this session with a typed value or the "box" option, that value will appear as the default. You can press [RETURN] to accept this value or enter a number to represent the new width. The selected entities will all be converted to raster with the designated width.

Using the LineWidth Configuration Table:

a. If you type in "T" or "Table", the LineWidth Configuration Table appears. The listing correlates CAD color numbers and line widths expressed in current units.

b. If you defined the width previously in this session with a typed value or the "box" option, that value will appear as the default. You can press **[RETURN]** to accept this value or enter a number to represent the new width. The selected entities will all be converted to raster of the designated width.

Note: The original default option is to use the Width Table set up by colors. This allows you to set different widths for entities being rasterized according to their color. If you press [RETURN] or [T], the Width Table will be displayed. The widths will be in inches or millimeters according to the unit set in GTXRAST.CFG.

```
LINEWIDTH CONFIGURATION TABLE
To change a line-width for different CAD colors, enter the number for the entry. When
you are satisfied with the settings, enter 0 to continue.
GTXImage CAD Color #s     Width in Inches
0.  To Continue
1.  Color 1 (Red)   .01
2.  Color 2 (Yellow).01
3.  Color 3 (Green) .02
4.  Color 4 (Cyan)  .02
5.  Color 5 (Blue)  .03
6.  Color 6 (Magenta).03
7.  Color 7 (White) .04
Enter the color number (1-7) of the width to change<0>:
```

c. Enter the number and press **[RETURN]**

```
Enter width <.XX>:
```

d. Enter the value for the width in the units being used and press **[RETURN]**. The Width Table will reflect the changes.

e. Select another color number to change its width or press [0] and **[RETURN]** to continue the command.

Note: The width settings you change will remain in effect until you change them or you exit GTXImage CAD.

The vectors will be burned to raster with the table's widths.

Note: Polylines and other non-filled entities that have a width attribute will be drawn with the GTXImage CAD width attribute.

Rasterization of certain entities may take a while to process. Rasterizing text and ellipses takes longer than lines, for instance. Select less data at a time to improve rasterization speeds.

You may need to zoom in to see line styles.

This command does not work for 3-D entities.

gCALIKE

Type	Menu	Availability
GCA or GCALIKE	GTX Vector Clean>Calike	GTXImage CAD PLUS

Icon:

gCALIKE modifies the radius of circles or arcs to a specified radius or to match that of a selected circle or arc.

Options:

The circles can be changed in terms of a Reference circle or arc, a specified Radius or Diameter.

Instructions:

Select objects:

1. Select one or more circles and/or arcs that you will want to set to identical radii.

   Select reference circle or arc:

2. Choose a circle or arc with the desired radius.

   Enter Diameter <Radius> <#.##>:

3. The reference object's radius will appear in the angle brackets. A rubber banding line stretches from the center of the reference object to the cursor. Press [RETURN] to accept the value, graphically select with the left mouse button the desired radius, type a new desired radius value, or type D and then enter a new diameter.

gCARC

Type	Menu	Availability
GCARC	GTX Edit>Change Arc	GTXImage CAD PLUS

Icon:

The gCARC command stretches arcs. Change the radius of an arc by moving its circumference and endpoint. This command also merges or smoothes fragmented and irregular arcs. To smooth, choose the raster entity and redraw it without changing its geometry.

Arc selection requires **5 point selection**

Command: _gcarc

Select start point of reference arc: (point #1)

Select point on arc: (point #2)

Select end of reference arc: (point #3 - the selection is highlighted)

Select new circumference point: (point #4 - keep or move the point)

Select new endpoint: (point #5)

Select start point of reference arc: (The command is ready for a second arc selection. Press the return key or escape key to terminate the command.)

gCCIRCLE

Type	Menu	Availability
GCCIRCLE	GTX Edit>Change Circle	GTXImage CAD PLUS

Icon:

The gCCIRCLE command stretches raster circles. Change the radius of a circle by moving its circumference. This command also merges or smoothes fragmented and irregular circles. To smooth, choose the raster entity and redraw it without changing its geometry.

Circle selection requires **4 point selection**

Command: _gccircle

Select point on circumference of circle: (point #1)

Select second point for circle: (point #2)

Select third point for circle: (point #3 - the selection is highlighted)

Current radius is ###

Enter circle radius: (point #4 or a numeric value may be typed)

Select point on circumference of circle: (The command is ready for a second circle selection. Press the return key or escape key to terminate the command.)

gCCIRCLE2P

Type	Menu	Availability
GCCIRCLE2P	GTX Edit>Change 2P Circle	GTXImage CAD PLUS

Icon:

The gCCIRCLE command stretches raster circles. Change the radius of a circle by moving its circumference. This command also merges or smoothes fragmented and irregular circles. To smooth, choose the raster entity and redraw it without changing its geometry.

 2 point circle selection requires **3 point selection**

 Command: _gccircle2p

Select point on circumference of circle: (point #1)

Select second point for circle: (A vector circle is stretched over the raster circle to point #2. The selection is highlighted)

Current radius is ###

Enter circle radius: (point #3 or a numeric value may be typed)

Select point on circumference of circle: (The command is ready for a second circle selection. Press the return key or escape key to terminate the command.)

gCELEV

Type	Menu	Availability
GCE or GCELEV	GTX Vector Clean>Contour Elevation	GTXImage CAD PLUS

Icon:

gCELEV sets the elevation of a series of PLINE objects in increments.

Instructions:

Set the base or starting elevation.

Set the Contour Interval – the increments by which elevation is changed.

Press **OK**.

Stop/<Pick Contour>:

Select the polyline to assign the starting elevation.

Stop/<Pick Contour>:

Select the polyline to assign consequent elevations.

When done, type **S**.

gCHANGE

Type	Menu	Availability
GCH or GCHANGE	GTX Raster Edit>Change	GTXImage CAD GTXImage CAD PLUS

Icon:

gCHANGE stretches raster lines, arcs and circles. Change the radius of an arc by moving its circumference and endpoint. Change the radius of a circle by moving its circumference. Change the length and orientation of a line by moving one endpoint.

This command also merges or smoothes fragmented and irregular lines or arcs. To smooth, choose the raster entity and redraw it without changing its geometry. To smooth or merge entire sections of a drawing, use the gSMOOTH command. In the following illustrations, dashed lines portray rubber-banding lines.

Options:

Object Type	Options
Line	Select a line to be changed.
Arc	Select an arc to be changed.
Circle	Select a circle to be changed.

Instructions:

1. Select gCHANGE.

   ```
   Select raster <LINE>:
   ```

2. The original default is "line". If you want to select another type of object, select it from the menu or type the first letter of the name.

Using Line

a. Select a point on the raster line to serve as the endpoint that will NOT change. Then select a point to indicate the rest of the line that will be altered. The raster line will be highlighted and a rubber-banding reference line will appear, attached at the first point used to pick the line (If the first point was outside the raster line, the endpoint closest to that point is selected).

   ```
   Select new endpoint:
   ```

Note: If you choose points to change a raster line drawn in a broken or dashed line style, all the segments will be selected, but the new line will be redrawn as a smoothed, solid line.

Turn on the Ortho Mode ON if you want to change lines so they are perfectly horizontal or vertical or so they snap to a specified angle.

b. Move the cursor. The reference line will rubber band as it follows the cursor. Press **[SELECT]** to select the new point where the line should end. The reference line will disappear and the raster line will be redrawn at the new orientation and length.

Using Arc

a. Select the endpoint that will NOT change. Then select a point on the circumference of the arc and the other endpoint of the arc. These three points define the raster arc and the selected arc will be highlighted.

   ```
   Select new circumference point:
   ```

b. You will see a rubber-banding line extending from the first endpoint selected. Select a point to be on the circumference of the new arc.

```
Select new endpoint:
```

c. The rubber-banding line will be attached at the second point. Select the endpoint for the new arc in any of the usual ways. The raster arc will be redrawn through the three points. If the original arc was in several pieces, they will be selected but the new arc will be solid.

Using Circle

a. Select the "circle" option from the menu.

```
Select point on circumference of circle:
```

b. Specify the raster circle by selecting three points on the circumference. After the raster circle has been selected, it will be highlighted. A rubber-banding line from the center of the circle will appear.

```
Enter circle radius:
```

c. Enter a value in current units or select a point to indicate the radius of the new circle. The circle will be redrawn with the original center point and the new radius.

Note: If the original circle was broken into arcs or drawn in a dashed line style, they will be selected but the new circle will be unbroken.

gCHGLAY

Type	Menu	Availability
GCHL GCHGLAY	GTX Vector Clean>Change Layer	GTXImage CAD PLUS

Icon:

gCHGLAY changes the layer of objects to that of a specified object.

Instructions:

```
Select Objects to Change Layer:
```

1. Select the objects you want to move to a destination layer.

```
Dialog/<Select object on NEW layer>:
```

2. Either type **D** to select a layer or to define a new layer, or select a sample object that is already residing on the desired destination layer.

If you selected a sample object, the selected objects are moved to the same layer as the sample. If you selected dialog, continue reading the following instructions:

The Manual Layer Selection dialog appears.

3. Select the destination layer name from the list or select **New Layer** to define a new destination layer for the selected objects.

4. When done, select **OK**.

The selected objects are moved to the specified layer.

gCIRCLE

Type	Menu	Availability
GC or GCIRCLE	GTX Raster Edit>Raster Circle	GTXImage CAD GTXImage CAD PLUS

Icon:

Draws a raster circle in the current raster image

Instructions:

Center Point:

Select or type the centerpoint of the circle.

Radius [Diameter]: Select or type the radius, or type "D" and then specify the Diameter.

The circle appears after the radius or diameter is specified.

Note: If there is no active image, gACTIVE will be invoked before continuing with gCIRCLE. If there is no image attached gCREATE will be invoked to set up an image into which raster line can be drawn.

gCLEANIOP

Type	Menu	Availability
GCLEANIOP	GTX Enhance>Clean IOP Raster	GTXImage CAD GTXImage CAD PLUS

Icon:

gCLEANIOP deskews the image and removes noise and speckles. gCLEANIOP creates a selection set of all noise. You can use ADD and REMOVE in combination with other IOP methods to refine the selection set. When finished, the command deletes the highlighted raster.

Instructions:

The image automatically deskews and noise is highlighted.

```
Select Raster <WINDOW>:
```

Add or remove raster using standard IOP methods. When done adding and removing raster, press [RETURN].

gCLINE

Type	Menu	Availability
GCLINE	GTX Edit>Change Line	GTXImage CAD
		GTXImage CAD PLUS

Icon:

The gCLINE command stretches raster lines. Change the length and orientation of a line by moving one endpoint. This command also merges or smoothes fragmented and irregular lines. To smooth, choose the raster entity and redraw it without changing its geometry.

Line selection requires **3 point selection**

Command: _gcline

Select one end of reference line: (point #1)

Select other end of reference line: (point #2 - the selection is highlighted)

Select new endpoint: (point #3)

Select one end of reference line: (The command is ready for a second line selection. Press the return key or escape key to terminate the command.)

gCONCEN

Type	Menu	Availability
gCONCEN or GCONC	GTX Vector Clean>Concen	GTXImage CAD PLUS

Icon:

gCONCEN moves circles and arcs to be concentric about a location or about the centerpoint of a specified circle or arc.

Instructions:

```
Select Objects:
```

1. Select one or more circles or arcs

```
Pick point/<Select circle/arc>:
```

2. Select another arc or circle about which the previously selected objects should become concentric. Alternatively, type **P** and select the location itself.

The selected objects are moved so their centerpoints align about the centerpoint of the target object or about the indicated location.

gCONVRT

Type	Menu	Availability
GCO or GCONVRT	GTX Convert>Vectorize	GTXImage CAD PLUS

Icon: R▶V

gCONVRT converts raster data to AutoCAD vector geometry.

Instructions:

Conversion Type

CAD conversion, raster data is converted into lines, arcs, circles and "arrowheads" (triangular SOLID entities). This option is best for mechanical drawings and line diagrams.

Contour causes raster data to be converted into polyline or spline entities. This option is best for maps and other documents containing contours.

Rectify

Ortho - Improve the drawing by making cleaner intersections; snapping lines to horizontal or vertical. Where applicable, fillet arcs are snapped to increments of 90 degrees, producing a high quality filleted corner.

Reference snaps lines that are close to a specified reference angle to that angle. Parallel lines at this angle separated by a distance close to the offset distance are snapped to that distance.

1. Select **Reference**.

   `First point for snap angle:`

2. Select the first point for the angular line.

   `Second point for snap angle:`

3. Select the second point to define the angle.

   `Reference angle <XX.XXXXX>:`

4. You can type in a new value for the angle or press **[RETURN]** if it is satisfactory. You cannot redefine it by selecting points.

   `Parallel alignment offset <0.0>:`

5. This value determines the distance between the imaginary grid lines parallel to the given reference angle. Type in a value in current drawing units. Using the **(0.0)** value which is always the default means no grid snapping for this angle.

   `First point for snap angle:`

6. You can define another reference angle and grid or press **[RETURN]** if you do not want to use another one. When you are finished defining the angles, press **[RETURN]**. The command then

---

continues as above.

Note: If you use the Reference Angle option, orthogonal rectification will not occur automatically. However, you can enter "0" and "90" reference angles to achieve orthogonal rectification.

None converts raster exactly as it appears without any rectification.

Vector Color

Vector entities do not have any width. You can preserve information stored in the line width of the raster image; you can assign a color to vectors resulting from different line widths.

By Width causes the values in the line-width configuration table will be used to determine the color of the vector entities. **Table Setup** displays the table and values can be changed if desired. If several colors are given the same width value, only the lowest number color will be used for that line width.

1. Select the **No Widths** check box to render resulting vectors in the current layer's color or select the **Table Setup** button to color-code resulting vectors for up to 16 different line widths.

Line Width Configuration Table Dialog

2. Click on the color to modify a color's width value. Enter the new value in the Width input window at the lower right corner.

Note: If you set your GTXImage CAD display options so that the background of the screen is white, the Line Width Configuration Table's "white" color (color #7) will refer to *black vectors.*

---

3. Select another color to change or click **OK** to exit the Line Width Configuration dialog.

Leaving **By Width** inactive causes resulting vectors to inherit the color and linestyle of their layers.

Hatches

When this is active, vectors generated from hatched areas will be isolated on their own layer. The name for this layer is established by the Hatch_Layer_Name configuration parameter in *GTXRAST.CFG*. You can create this layer in advance in your drawing template and give it a color if you always want hatched data to appear the same.

```
Hatch angle? <0.000>/Pick:
```

1. Type the hatch angle or type **P** to select two points designating the angle.

```
Parallel alignment offset <0,0>
```

2. Type the distance between hatches or select two points designating the distance.

```
Hatch angle? <45.000>/Pick:
```

When done, press **[RETURN]**.

```
Parallel alignment offset <0,0>
```

Press **[RETURN]** again.

Linestyles

When this is active, lines that appear to have regular breaks in the raster image will be isolated on their own layer. The name for this layer is established by the Linestyle_Layer_Name configuration

parameter in *GTXRAST.CFG*. As with Hatch_Layer_Name, you can create this layer in advance in your drawing template and give it a color and linestyle if you always want hatched data to appear the same.

Arrowheads

Controls whether arrowheads (3-sided solids) are generated during CAD type conversion.

Linear alignment

Linear Alignment causes broken lines to be merged into unbroken lines as they are vectorized. This is used to convert dashed geometry as a single entity (then the entities can be moved to a dashed layer). If this box is empty, collinear geometry with gaps or breaks will be vectorized without closing the gaps. The gap is established by the Gap Jump setting.

Gap Jump

Sets the distance in pixels for Linear Alignment to jump when combining multiple collinear lines into single lines (as in recognizing dashed linestyles). This can be set by manually entering the distance or by selecting Pick Gap and then graphically showing a gap distance.

Other Options

Save Raster preserves raster data after converting it to vector. If this box is empty, the raster will be erased after conversion.

Elevation sets the elevation at which resulting vectors will be created. The default value is 0. To create contours at different elevation, set this value before converting raster to vector.

Elevation

Set the Elevation to the level at which resulting vectors should be created (normally you want this to be 0).

When all parameters are set as desired, click OK to launch the conversion process.

Prompt Mode

gCONVRT also works exclusively with GTXImage CAD prompts when the CMDDIA variable is set to 0.

gCOPY

Type	Menu	Availability
GCP or GCOPY	GTX Raster Edit>Cut/Copy	GTXImage CAD GTXImage CAD PLUS

Icon:

gCOPY copies raster data from one location to another location in the image entity.

Options:

Movement Options

Base Point   Indicate the offset distance for the copy by selecting two points.

Note: Turn the ORTHO Mode ON to place copies in a perfectly horizontal, vertical, or specified direction from the last position.

Displacement   Enter X and Y offset values to indicate copy placements.

Instructions:

1. Select **gCOPY**.

    `Select raster <WINDOW>:`

2. Select the raster to be copied. The selected raster will be highlighted.

3. When all of the raster to be copied is highlighted, press **[RETURN]**.

    `Base point or displacement:`

    There are several ways to respond to this prompt. To use the default option, either select a reference point or type in how far over the selected data should be copied. Since the system does not know whether X,Y value typed in this is the base point or the displacement distance, the response to the second point clarifies this.

Using Base Point

a. Select a base point either by using the cursor or by typing the coordinates. This will be a reference point for the copy process.

    `Second point of displacement:`

b. A rubber-banding line will appear attached at the base point. Select a second point to define the relative distance and angle for placement of the copy. The copy will appear in the designated location in the display raster color. The original raster will be un-highlighted. The command will terminate.

Using Displacement

a. At the "base point or displacement" prompt, type in the X and Y value for the displacement in the same format used to enter coordinates. You will see the same prompt as above.

    `Second point of displacement:`

b. Press **[RETURN]** to indicate that the original entry was the displacement. The raster data you selected will be copied to its new location in the main raster color. The original raster will be un-highlighted. The command will terminate.

---

gCREATE

Type	Menu	Availability
GCR or GCREATE	GTX Imaging>Create Raster Area	GTXImage CAD GTXImage CAD PLUS

Icon:

gCREATE establishes an unnamed raster image entity on the current layer. The raster data can be modified within the limits, which can be changed later using the gRESIZE command.

The MaxNumPixels configuration parameter limits the number of pixels allowed in a new raster image. The default value is:

```
MaxNumPixels= 973440000
```

which at 400 dpi could generate an image measuring 78"x78".

Instructions:

1. Select **gCREATE**.

```
Enter raster resolution DPI <200>:
```

2. Enter a positive **integer**.

```
select lower left corner of raster limits <0.000,0.000>:
```

3. Normally, you enter "0,0" for this value. However, you can select any point either by typing coordinates or selecting it graphically.

```
Select upper right corner of raster limits:
```

4. A rubber-banding box will appear attached at the first point. Select the point for the upper right corner either with coordinates or graphically.

Warning: When creating a new raster image in a hybrid environment, consider the drawing's scale. Raster images will normally be about the same size as paper media. Create images at "paper" size and then scale them against your vector data. You do not want to create a raster image the size of a house or a map - it will take a long time if it works at all!

---

5. The program will check whether there is sufficient memory for the work area you designate. If not, you will get an error message.

gCROP

Type	Menu	Availability
GCRO or GCROP	GTX Enhance>Crop	GTXImage CAD GTXImage CAD PLUS

Icon:

gCROP Deletes all raster data outside of a specified rectangular window. For instance, you may use it to get rid of "dirty" edges or an unwanted frame.

Instructions:

1. Select the gCROP command.

    ```
    Select first corner of window:
    ```

2. Select a point for one corner of the window.

    ```
    Select opposite corner of window:
    ```

3. A rectangular window will rubber-band from the first point selected. Position it to include all the raster data you want to save. Press **[SELECT]** to select the point.

Note: If your drawing is skewed or rotated, use gDESKEW first.

To reduce the raster limits by eliminating the outside area, use gRESIZE or use the gCUT command to create a new file.

The raster data outside the window will disappear. The raster limits stay the same.

gCTEXT

Type	Menu	Availability
GCTEXT	GTX Edit>Change text	GTXImage CAD GTXImage CAD PLUS

Icon:

The change text feature begins by framing an area of the current raster image that contains raster text data that is to be replaced by new rastertext. The framed area is defined by a baseline at any orientation angleand the replacement raster text will be drawn at that angle. The replacement text is entered with the aid of the Text Formatting control.

 Instructions:

1.Select insert point of baseline

The point chosen will be the lower left point of the first text character of the replacement raster text.

 2.Select end point of baseline

This point defines the width of the existing raster data that will be replaced by either new raster text or blank space.

3.Select rectangle height

This point completes the frame of raster data that will be replaced by new raster text or blank space..

Height

All raster data contained within the framed area will be highlighted. All highlighted raster data will be removed and replaced by the following steps.

4.Specify new text height

Enter a numeric value that will set the height of the new raster text. The height of the existing raster text may be determined ahead of time by using the Dist command.

5. The Text Formatting control with the In-Place Text Editor will appear. Type over the raster text that was highlighted in step #3.

6. To end the editing click the OK button on the Text Formatting toolbar or click outside the In-Place Text Editor.

gCtext Advanced Instructions:

The change text command provides the choice of multiple IOP (intelligent object picking) selections on any part of the raster data in the current image object, however only the area of the data highlighted by the baseline and height framing is subject to editing of raster text. Any other highlighted raster will be erased when the gCtext command has completed.

In this example the editable text in the following images is the word BRACKET. That word was framed using the baseline and height selection. The two text lines at the bottom of this example were subsequently highlighted using an IOP selection, in this case the Inside selection.

This is the sequence gCtext command responses that complete this example;

Command: gctext

Select insert point of baseline:

Select end point of baseline:

Select rectangle height:

PC NO.	NAME	MAT.	QUAN.	NOTES
101	BRACKET	C.I.	I	
102	GEAR	C.I.	I	
103	SHAFT	STEEL	I	
104	GLAND	C.I.	I	
105	PULLEY	A.I.	I	DIE CAST
106	BUSHING	BRO.	I	

The word BRACKET was framed by a baseline and height point.

Select raster <INSIDE>:

Select opposite corner of window:

The last two lines of text were selected and highlighted by the Inside IOP

Select raster <INSIDE>:

Multiple IOP choices on other sections of raster data are allowed. Press the keyboard Return to escape further picking choices

Specify new text height

The In-Place Text Editor opens and the word BRACKET is edited.

To end editing click the OK button on the Formatting toolbar or click outside the In-Place Text Editor

PC NO.	NAME	MAT.	QUAN.	NOTES
101	BRACKETS	C.I.	1	
102	GEAR	C.I.	1	
103	SHAFT	STEEL	1	
104	GLAND	C.I.	1	

The word BRACKET has been changed and all IOP selected raster data is removed.

gCUT

Type	Menu	Availability
GCU or GCUT	GTX RasterEdit>Cut	GTXImage CAD
		GTXImage CAD PLUS

Icon:

gCUT copies raster data from the active image entity to a raster file or to memory. The default filename is set in the configuration file.

Note:

1. Raster data cut from images is oriented relative to the raster file's coordinate system. When cutting raster data from a rotated image entity, the raster is not necessarily saved as displayed within GTXImage CAD. Also, when pasting raster data into a rotated image, the raster data appears at the image entity's native rotation.

2. If the "raster limits" box does not enclose all of the highlighted raster data, the data outside the rectangle will not be saved in the file.

3. If the rectangle does not fall entirely within the raster limits, the limits will be used instead of the point(s) selected.

If you want to save the file with a format other than the one specified in the GTXRAST.CFG file, type in the extension that defines that format.

Options:

Mode Options

Copy   Copy the data and retain it on the screen.

Erase   Copy the data and erase it from the drawing.

Destination Options

File   Copy the data to a raster file.

Buffer   Copy the data to a temporary buffer (to be gPASTE-d later).

Instructions:

1. Select gCUT.

   ```
   Command: gCUT
   Mode <Copy>/Erase:
   ```

2. The default value is in brackets. Press **[RETURN]** to select the default or select the desired mode from the menu or by entering its abbreviation.

   ```
   Select raster <WINDOW>:
   ```

3. Select the raster data.

4. Press **[RETURN]** when all the raster data to be saved in the new file has been highlighted. You will next be prompted to define the image limits.

   ```
   Select lower left corner of raster limits:
   ```

5. Type in the coordinates for the point or position the cursor and press **[SELECT]**. A rubber-banding box will appear attached at the point.

```
Select upper right corner of raster limits:
```

6. Select the point for the upper right corner of the limits. The two points you have selected are stored as the upper and lower limits of the image.

```
File/<Buffer>:
```

7. Enter **B** to copy the data to a temporary raster buffer. The selected data within the rectangle will be copied to a raster buffer for later insertion using **gPASTE**. Enter **F** to save the selected data to a raster file (Continue with step 8).

Dialogs:

a. After specifying F or File, gCUT will save the selected data to a raster file. gCUT will display the SAVE Raster File dialog.

    Enter the desired file name and select the appropriate path for the new file.

b. To change the default file type, select the scroll box labelled "List Files of Type:" and select the desired file format.

c. The file name entered does not need to include the extension. The file type extension displayed in the List Files of Type: box will be used.

d. Click OK to return to the drawing editor.

    Prompts:

a. After specifying F or File, gCUT will save the selected data to a raster file. gCUT will display the following prompts:

```
Enter raster file name <cut_data.ext>:
```

b. The default file name is "cut_data" with the extension indicated in the *GTXRAST.CFG* file. If you have previously used **gCUT**, the name of that file will be the default shown in brackets. Accept the default file name by pressing **[RETURN]** or type a new name for the raster file and press **[RETURN]**.

    If you do not include the extension, the format and extension set in the *GTXRAST.CFG* by "RasterFormat" and "RasterSaveExt" entries will be used. If no path is indicated, the file is saved using the path defined in the *GTXRAST.CFG* "RasterPath" entry. If no path was defined, it is saved in the current directory.

c. If a file already exists with the same name you typed as a new name, you will be asked whether to replace it. If you respond "yes", the new file will overwrite the old one. If you accept the default, "No", you will again be prompted for a raster file name. The request is repeated until an acceptable name is entered or you cancel the command without saving a file.

d. When a file has been saved, the highlighting will disappear. If the "erase" mode was selected, the highlighted data will also disappear. Otherwise the raster data remains as it was before the command was invoked.

```
Raster file FILENAME.EXT saved.
```

gDESKEW

Type	Menu	Availability
GD or GDESKEW	GTX Enhance>Deskew	GTXImage CAD
		GTXImage CAD PLUS

Icon:

Drawings are often scanned in slightly rotated. gDESKEW straightens selected raster by rotating it to align with the X or Y axis.

The gADESKEW (Auto Deskew) command may be faster and easier with drawings that have titleblocks.

The deskew uses reference line that is rotated to horizontal or vertical (whichever is closer). Pick points as far apart as possible. Use transparent zoom or raster snapping features to draw the reference line precisely.

The lower left corner is used as the rotation point.

Processing time depends on image size, and resolution, and available memory.

Instructions:

1. Select gDESKEW.

    Command: gDESKEW

2. press **[RETURN]**.

    Select raster <WINDOW>:

3. Select the raster data to deskew. If deskewing the entire drawing, type: **ALL**. Press **[RETURN]**.

    Select one end of reference line:

4. Select a point on a raster line that should be perfectly horizontal or vertical. A rubber-banding reference line will extend from it to the cursor.

    Select other end of reference line:

5. Move the cursor to the second point on the line and select it. The deskew will take place, based on the reference line you indicated. You will see a message: De-skewing XX.XXXXXX degrees.

    Do you wish to displace deskewed raster? Y/<N>:

6. If the raster is located correctly, press **[RETURN]**. The command will be completed. If you wish to move the deskewed raster, type: **Y**.

    Base point or displacement:

There are two ways to respond. You can either select the first of two points that will show the movement graphically or type in the offset values in the form X,Y. Since the system does not know whether the X,Y value typed in is the base point or the displacement distance, the response to the second prompt clarifies this.

Using Base Point

1. Select the reference point for the moving process.

    Second point of displacement:

Note: 1. If you skew the same file several times the raster data could become distorted. If this happens, re-load the raster image and deskew it again using gDESKEW once.

2. If the image needs to be rotated more than 5 degrees you should re-scan the drawing. Deskewing the entire image more than a few degrees can distort the raster data.

3. Large files may take several minutes to complete the operation.

Using transparent zoom or the "end" or "int" raster snapping methods can help you more accurately pick your reference line.

2. A rubber-banding line will appear attached at the base point. Select a second point to define the relative distance and angle for the new placement of the deskewed raster. It will appear in the designated location. The command will be completed.

Using Displacement

1. Type in the X and Y values for the displacement in the same format used to enter coordinates. You will see the same prompt as above.

   ```
   Second point of displacement:
   ```

2. Press **[RETURN]** in response to the prompt to indicate that the original entry was the displacement. The deskewed raster will appear in the new location.

gDETACH

Type	Menu	Availability
GDETACH	GTX Imaging>Detach Raster Image	GTXImage CAD GTXImage CAD PLUS

Icon:

Removes the selected image definition from the drawing database and erases an image object from the drawing and from the display. If there is only a single image object defined in the drawing it will be removed in a single step. The GTX Image Selection control is presented if there are more than one image objects defined in the drawing. A single image must be selected for detachment.

DrawOrder Back

Type	Menu	Availability
GDRO_BK	GTX Imaging>Modify Images>Send Images to Back	GTXImage CAD GTXImage CAD PLUS

Icon:

DrawOrder Back Moves all raster image entities' draworder to back.

gECONVRT

Type	Menu	Availability
GEC or GECONVRT	GTX Convert>Vectorize Edge	GTXImage CAD PLUS

Icon:

gECONVRT converts raster data to CAD vector geometry.

Options:

Contour: When checked, raster contours are converted into polyline entities, which are good for designating contours. When this checkbox is empty, raster data is converted into lines, arcs, and circles.

Save Raster: Retains raster after conversion to vector. If this box is empty, the raster is erased during conversion.

Elevation: Sets the elevation at which resulting vectors will be created. The default value is 0. To create contours at different elevation, set this value before converting raster to vector.

Instructions:

1. Invoke gECONVRT.

    ```
    Specify raster <WINDOW>:
    ```

2. Select the desired raster and press **[RETURN]**. The **Edge Convert Setup** dialog will appear:

    Width Recognition

    The edge of a solid raster area has no width, so this option is disabled. Proceed to the conversion type.

    CAD or Contour Conversion:

3. Checking the **Contour** box will cause lines and polylines to be generated. Leaving this box un-checked will cause only lines, arcs and circles to be generated.

    a.   If the **Contour** option is checked, the **Polyline Smooth Index** input window will contain a value between **1** and **9**. A value of 1 would follow the contours least smoothly, and 9 results in the closest match to the contour.

    With lower resolution images you may want to choose values lower than 8 or 9 so that the polylines do not follow the square contours of individual pixels!

4. Set the **Elevation** to the desired level at which resulting vectors will be created.

5. Set **Save Raster** to your preference

6. Select **OK** to start the conversion.

gEDGE

Type	Menu	Availability
GED or GEDGE	GTX Enhance>Edge	GTXImage CAD GTXImage CAD PLUS

Icon:

gEDGE removes the inside of solid raster regions, leaving only the edge. Removes all raster except a single pixel width contour around the edge of selected raster data.

Instructions:

Command Dialog:

1. Invoke gEDGE.

```
Select raster <WINDOW>:
```

2. Select the raster you wish to "hollow out". Press **[RETURN]**.

3. The insides of the raster are erased, leaving a 1-pixel edge around the outside.

gERASE

Type	Menu	Availability
GE or GERASE	GTX Raster Edit>Erase	GTXImage CAD GTXImage CAD PLUS

Icon:

gERASE removes selected raster data from the drawing.

Instructions:
```
Select raster <WINDOW>:
```

1. Invoke **gERASE**.

2. Select the raster you want to remove.

3. When all the raster data to be removed is highlighted, press **[RETURN]**. The selected raster is then erased.

gFRZLAY

Type	Menu	Availability
GFR	GTX Vector Clean>Freeze Layer	GTXImage CAD PLUS

Icon:

gFRZLAY freezes the layer of a selected object.

Instructions:

Dialog/<Select object>:

Choose an object residing on the layer you wish to freeze or type **D** to use the dialog.

Object:

The layer of the selected object becomes frozen.

Dialog:

A dialog appears. Choose the layer or layers to freeze and select OK.

HELP

Type	Menu	Availability
HELP	Help>Help	GTXImage CAD GTXImage CAD PLUS

Icon:

HELP starts the GTXImage CAD Help file. The Help window functions as any other Windows application Help file, including "search for keyword" capabilities.

For more information on available sources of help, refer to the Introduction.

gHTEXT

Type	Menu	Availability
GHTEXT	GTXConvert>Edit Hybrid Text	GTXImage CAD
		GTXImage CAD PLUS

Icon: **A'**

Hybrid text is raster text with a vector text overlay created by using the gText command while the gHybridmode function is enabled. Edited hybrid text will be oriented at the same angle as the text that is chosen to be edited. The gHtext command initiates a vector text editor aided by the Text Formatting control.

 Instructions:

Command: ghtext

1.Select text object:

     Mouse click on the vector text to be edited.

 2.Selection of a vector text string opens the Text Formatting control with the   In-Place Text Editor.

3.Place the cursor at the point where text is to be added or deleted.

    Type new text or swipe over existing text to delete.

 4.When editing is complete, click the OK button on the Text Formatting control or click somewhere outside of the In-Place Text Editor. The changed hybrid text is completed.

gHYBRIDMODE

Type	Menu	Availability
GHYBRIDMODE	GTXConvert> Hybrid Mode	GTXImage CAD GTXImage CAD PLUS

Icon:

Enabling the hybrid mode will result in a vector overlay of raster that is drawn using any of the GTX raster drawing commands

Hybrid mode value 1 is enabled and value 0 is disabled.

Command: ghybridmode

HYBRIDMODE: <0>,1 :

Image Frame

Type	Menu	Availability
ImageFrame	GTX Imaging>Modify Images>Image Frame	GTXImage CAD GTXImage CAD PLUS

Icon:

ImageFrame sets frames for all image entities to on or off. Sets the GTXImage CAD ImageFrame variable.

Image Transparency

Type	Menu	Availability
Transparency	GTX Imaging>Modify Images>Image Transparency	GTXImage CAD GTXImage CAD PLUS

Icon:

Transparency makes all images transparent when selected from the Imaging toolbar. Otherwise lets you set the transparency mode for selected images.

gINACTIVE

Type	Menu	Availability
GINA or GINACTIVE	GTX Imaging>Deactivate Image	GTXImage CAD GTXImage CAD PLUS

Icon:

gINACTIVE deactivates the active image entity, freeing memory. In order to edit the image you must reinvoke gACTIVE and choose the image.

gINFO

Type	Menu	Availability
GI or GINFO	GTX Imaging> Raster Info	GTXImage CAD GTXImage CAD PLUS

Icon:

gINFO displays details about the current raster image and the software license. It includes the default values for raster file name and format, as well as the resolution, the raster limits and data extents (in current drawing units). The file name and format will be that of the last raster file loaded or saved.

Instructions:

1. Select **gINFO**. You will see a listing on the text screen display similar to the following. The values shown are sample values only.

> **Resolution:**   Shows the resolution of the current raster database in terms of dots per inch (DPI). This value is obtained from the resolution from the file unless overridden by the *GTXRAST.CFG* **Resolution** setting or by a resolution established using **gCREATE**.

> **Raster data limits:** Defines the rectangular area indicating the size of the present raster image, including bounding white space, set with **gCREATE** or **gRESIZE** or by the limits of the raster drawing that was loaded. The values are given in current drawing units.

> **Raster data extents:**   Defines smallest rectangular area that can enclose the raster data in terms of CAD units. It is defined by the X and Y coordinates of the lower left corner, followed by the X and Y coordinates of the upper right corner of the rectangle. The extents determine where raster data is currently drawn.

> **Drawing limits:**   Defines the CAD drawing limits.

> **Source FileName:** The name of the raster file to which the image refers.

> **Active FileName:** The filename to which **gSAVE** will save.

> **Raster File type:**   This reflects the format of the last raster file saved with **gSAVE** or the setting in *GTXRAST.CFG* if no file has been saved yet.

> **Undo store size:**   Number of raster undo's being recorded.

> **Image modified?**   Flag identifying whether image has been changed since the last save (TRUE or FALSE).

> **Version:**   Shows GTX program, version number, and ARX version of the GTX program that is running.

gINVERT

Type	Menu	Availability
GIN or GINVERT	GTX Enhance>Invert	All Products

Icon:

gINVERT Reverses the background and main raster colors of the current raster image. Use it on files scanned in reverse (light on dark). This does not just invert the *display* of an image - it truly inverts the raster image itself.

Instructions:

The color of the background and the color of the data will be switched. You can then edit the raster data with any of the GTXImage CAD commands.

Undo gINVERT by invoking the command again.

gISOLAY

Type	Menu	Availability
GISL or ISOLAY	GTX Vector Clean>Isolate Layer	GTXImage CAD PLUS

Icon:

gISOLAY freezes all layers except that of the selected object, effectively isolating its layer.

Instructions:

```
Select objects:
```

Select objects on the layer or layers you want to isolate.

Press **[RETURN]**.

The layers of the objects chosen in step 1 are isolated by freezing all other layers.

gJOIN

Type	Menu	Availability
GJO or GJOIN	GTX Vector Clean>Join	GTXImage CAD PLUS

Icon:

gJOIN joins vector lines together.

Instructions:

Select line startpoint.

Select the endpoint of the line you wish to join with another line.

```
Select object
```

Select the first line (whose endpoint you selected).

```
Select object
```

Continue selecting lines until all lines you wish to join have been selected.

Press [RETURN]

All selected lines are merged into one line.

gLINE

Type	Menu	Availability
GLN or GLINE	GTX RASTER Edit >Raster Line	GTXImage CAD
		GTXImage CAD PLUS

Icon:

gLINE draws a raster line in the current raster image.

Instructions:

From point:

     Select or type the first endpoint.

To point:

     Select or type the other endpoint.

The command continues, allowing you to continue entering point coordinates.

To point:

Note: If there is no active image, **gACTIVE** will be invoked before continuing with **gLINE**. If there is no image attached, gCREATE will be invoked to set up an image into which the raster line can be drawn.

     Enter the endpoint for the next line or press **[RETURN]** to end the command.

gLINE2

Type	Menu	Availability
GLINE2	GTX Edit>Draw Line2	All Products

Icon:

GLINE2 draws multiple raster lines in the current raster image.

Instructions:

Command: gline2

Line width <0.010>/Box/Pick:

From point:

> Select or type the first endpoint.

To point:

> Select or type the other endpoint.

From point:

> Select or type the second endpoint.

To point:

> Select or type the other endpoint.

The command continues, allowing you to continue entering point coordinates.

> Press **[RETURN]** to end the command.

gMIRROR

Type	Menu	Availability
GMI or GMIRROR	GTX Raster Edit >Mirror	GTXImage CAD GTXImage CAD PLUS

Icon:

gMIRROR creates a mirror image of selected raster data. This is differentfrom reflect which mirrors the entire current raster image. You will define a line for the data to be mirrored across. You may either retain or discard the original data.

Instructions

1. Select gMIRROR.

```
Select raster <WINDOW>:
```

2. Select the raster data to be mirrored. The selected raster will be highlighted.

3. When you have selected all the items to be mirrored, press **[RETURN]**.

```
First point of mirror line:
```

4. Select a point for the start of the reference line.

```
Second point:
```

5. Select another point to define the reference line.

```
Delete old objects? Yes/<No>
```

6. Press **[RETURN]** to preserve the original raster geometry. If you only want to keep the new mirrored data, enter **Y**.

gMOVE

Type	Menu	Availability
GM or GMOVE	GTX Raster Edit>Move	GTXImage CAD GTXImage CAD PLUS

Icon:

Function:

gMOVE moves selected raster data from one area to another on the drawing. The displacement points define the relative distance and angle that the items will be moved from the original position.

Note: Turn ORTHO mode ON to move objects in perfectly horizontal or vertical directions.

Options:

Base point   Indicate the offset distance for the copy by selecting two points.

Displacement   Enter X and Y offset values to indicate copy placement.

Instructions:

1. Invoke gMOVE.

```
Select raster <WINDOW>:
```

2.   Select the raster data to be moved. The selected raster will be highlighted.

3.   When you have selected all the items to be moved, press **[RETURN]**.

```
Base point or displacement:
```

There are two ways to respond. You can either select the first of two points that will show the movement graphically or type in the offset values in the form X,Y. Since the system does not know whether the X,Y value typed in is the base point or the displacement distance, the response to the second prompt clarifies this.

Using Base Point

a. Select a point either by using the cursor or by typing the coordinates. This will be a reference point for the moving process.

```
Base point or displacement:
Second point of displacement:
```

b. A rubber-banding line will appear attached at the base point. Select a second point to define the relative distance and angle for the new placement of the data. The data will appear in the designated location in the main raster color. The raster will disappear from the original location. The command will be terminated.

Using Displacement

a. Type in the X and Y values for the displacement in the same format used to enter coordinates. You will see the same prompt as above.

```
Second point of displacement:
```

Press **[RETURN]** in response to the prompt to indicate that the original entry was the displacement. The raster data you selected will disappear from the original location and be displayed in the new location in the raster color.

gOFFSET

Type	Menu	Availability
gOFFSET	GTX Raster Edit>Offset	GTXImage CAD
gOFF		GTXImage CAD PLUS

Icon:

gOFFSET creates a copy of a selected line, arc, or circle parallel or symmetrical to the original. This command is similar to the CAD OFFSET command.

Instructions:

Having prepared an image for editing,

1. Invoke gOFFSET.

   ```
   Offset distance or Through <Through>:
   ```

2. Either enter a distance or use the mouse to indicate the distance through which copies will be offset.

   ```
   Select raster <LINE>:
   ```

3. At this point you enter "C" for circle, "A" for arc, or click two points on a raster line.

   Line

   Click two points along the raster line.

   Arc/Circle

   Click three points along the circumference of the arc or circle.
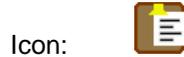
   Offset

   ```
   Side to offset?
   ```

4. Click a point to one side of the line, arc or circle. (You can click inside or outside a circle).

   A copy of the original entity will be placed at the specified offset distance on the indicated side.

gPASTE

Type	Menu	Availability
GP or GPASTE	GTX Raster Edit>Paste	GTXImage CAD GTXImage CAD PLUS

Icon:

gPASTE inserts a saved raster image at a specified location on the graphics screen. You can use this command to insert saved raster onto an existing image or onto a blank graphics screen to create a new image. The image may be scaled or rotated as it is placed. The default values for file format and name will be taken from the *GTXRAST.CFG* file or the current database, but can be overridden. Raster files of any acceptable format can be used.

1: Raster data cut from images is oriented relative to the raster file's coordinate system. When cutting raster data from a rotated image entity, the raster is not necessarily saved as displayed within GTXImage CAD. Also, when pasting raster data into a rotated image, the raster data appears at the image entity's native rotation.

2: If there is no active image, *gACTIVE will be invoked before continuing with gPASTE. If there is no image attached, gCREATE will be invoked to set up an image into which raster data can be pasted.*

Options:

Source Options:

> File   Insert raster data from a raster file.
>
> Buffer   Insert raster data from a temporary buffer (previously created with gCUT).

Paste Options:

> Move   Move the pasted data to a different location.
>
> Rotate   Specify the desired orientation (angle) of the image.
>
> Scale   Specify the factors by which to change the X and Y. dimensions.
>
> Fit   A "Scale" option that allows you to show the scale graphically.

Instructions:

1. Invoke the gPASTE command.
   ```
   File/<Buffer>:
   ```

2. Type **F** to paste data from a raster file. Type **B**, **Buffer** or **[RETURN]** if it is the default as shown above to paste data from a temporary buffer (created with **gCUT**).

Pasting from a File:

gPASTE will determine the raster file you wish to paste. If *CMDDIA* is set to **1**, the following dialog will appear:

Enter the desired file name and select the appropriate path for the new file.

If *CMDDIA* is set to **0**, **gPASTE** will issue the following prompt:

```
Command: GPASTE
Enter raster file name <cut_data.ext>:
```

Type the name of the raster file that you want to paste. "Cut_data" or the last file pasted is the default, shown in brackets. If no path is entered, the program looks for the file in the current directory, then for the ACAD path, then for the DOS path.

Pasting from a Buffer

gPASTE will paste raster data from a buffer only if you previously performed a **gCUT** to that buffer.

After determining the source of the raster data to be pasted, the following prompt will appear:

```
Insertion point <0.0000,0.0000>:
```

3. Select the point where the lower left corner of the file should be placed. The raster data of the file will appear at that location in the main raster color. The **gPASTE** menu will replace the GTXRAST menu.

```
Loading raster -- filename.ext ...
Move/Rotate/Scale
```

4. If you are satisfied with the default settings and the present location of the raster, press **[RETURN]** to exit from the command. To change the location, rotation or scale, select the desired option. Options can be selected in any order and any number of times until you press **[RETURN]** to exit from the command.

The same data can be selected for further raster editing with the "previous" option of the editing command. For instance, if you want to have more than one copy of the pasted data at the same size and orientation, you can select **gCOPY** and "previous".

Using Move

1. Select "Move" by entering "m" or selecting "Move" from the menu. The pasted data will be highlighted. You will be prompted to indicate a base point for the move.

```
Base point or displacement:
```

2. Select a reference point. This may be a significant point such as the center of a cross.

```
Second point of displacement:
```

A rubber-band line will extend from the base point.

3. Select a second point to indicate the distance and angle that the base point should move. The raster data will move to the new location and be un-highlighted. The reference line disappears.

```
Move/Rotate/Scale:
```

4. You can continue to select options or press **[RETURN]** to terminate the command.

Using Rotate

1. Enter "r" or select "Rotate" from the menu to change the orientation of the reference box. The pasted data will be highlighted.

```
Base point:
```

2. Select a point for the center of rotation.

```
Rotation Angle <0.0>/Reference:
```

3. A rubber-band line will extend from the base point. The rotation point is the lower left corner of

the box. The rotation is counter-clockwise from the X-axis if a positive angle is entered.

There are several ways to respond. You can accept the default angle by pressing **[RETURN]** or use either of the two options. The default angle, initially zero degrees, reflects the value used for a previous **gPASTE** operation.

Rotation Angle

Using rotation angle, you can specify the angle with reference to the X-axis by typing in a value (in degrees) or by rubber-banding the reference line.

When you designate the angle by either method, the raster will be redrawn in the new orientation.

Using Reference Angle

Using reference angle, you must enter "r" or select "Reference" from the menu. You then indicate a different base line (not the X-axis) for the graphic angle.

```
Reference angle <0.0>:
```

- You can either enter a degree value to define the new base line in relation to the X-axis or you can define it graphically by selecting the first point for the base line.

```
Reference angle <0.0>: Second point:
```

- Select the second point for the base line.

```
New angle:
```

• Indicate the final position for the new base line by entering a value. You can also indicate the rotation angle graphically if you chose the original base point (the center of rotation) as the first point of the new base line. The selected data will rotate the same amount. The angle of rotation will actually be the angle between the reference line positions.

For example, if the desired rotation would bring a certain line to a vertical position, you could draw the base line over that line and then type "90" for the new angle. The default angle then will be 90 minus the angle of the base line.

- As soon as the rotation angle has been specified, the highlighted data will rotate about the base point. The highlighting will be removed.

```
Move/Rotate/Scale
```

You can select another option to change or press **[RETURN]** to terminate the command.

Using Scale

1. If you enter "s" or select "Scale" from the menu, the pasted data will be highlighted and you can change the size of the image. The default value is initially set to X=1, Y=1.

```
Base point:
```

2. Select the point about which the raster will be scaled. This point will not change its location. It also serves as the first point for the scale rectangle if you use the graphic method of scaling.

```
Scale factors <1.0,1.0>/Fit:
```

3. The data is scaled from the base point. You can type the numerical values by which to scale the X- and Y- dimensions or select "Fit" and change the height and width graphically.

If you select "Fit", you will be prompted to select the original size.

```
Select original size:
```

Select two corners of a box representing the size of the original image.

`Select new size:`

Draw the window to show the relative size of the desired image to the reference box. The data that filled the original rectangle will be enlarged or compressed to fit into the new rectangle. The X and Y dimensions can be different.

After you indicate the scale by either method, the pasted data is redrawn at the new size and un-highlighted.

`Move/Rotate/Scale:`

4. You can select another option to change or press **[RETURN]** to terminate the command. You can further adjust the image after you have placed it with the "previous" option in the desired command (**gMOVE**, **gSCALE**, etc.).

gRAHEAD

Type	Menu	Availability
GRAH or GRAHEAD	GTX Vector Clean>Remove Arrow Heads	GTXImage CAD PLUS

Icon:

Function:

gRAHEAD erases "arrowheads" (three-sided solids).

Instructions:
```
Filter Arrow heads - All/<Select objects>:
```

Type **A** for all or press **[RETURN]** to select a group of objects

If you typed **A**, then all 3-sided solids are erased. If you chose to select objects, the following prompt is shown:
```
Select objects:
```

Window or pick objects. Press **[RETURN]** when done. arrowheads are highlighted.

Press **[RETURN]** to erase them.

gRASTER

Type	Menu	Availability
GRA or GRASTER	GTX Convert>Rasterize	GTXImage CAD GTXImage CAD PLUS

Icon:

gRASTER converts selected vector entities into raster data with a designated line width. The line style will be preserved.

Options:

Width Options

Table   Use the width settings according to color as set in the Width Table

Box   Define a value graphically to be used for the width of the vector selected to be converted to raster.

Value   Type in a value in inches or millimeters to use for the width of the vector selected to be converted to raster.

Notes:

If there is no active image, gACTIVE will be invoked before continuing with gRASTER. If there is no image attached, gCREATE will be invoked to set up an image into which raster data can be created.

Only entities within the raster limits can be rasterized. Those selected outside the limits will be ignored or erased. When entities are partially outside, the part inside is rasterized; the part outside is erased.

GTXImage CAD will not rasterize AutoCAD 3D objects.

GTXImage CAD will not rasterize hatches with gradient/fill pattern colors .

Instructions:

Dialog

1. Invoke gRASTER. The following prompt will appear:
   Select objects:

2. Select the vector entities to be converted. When all entities to be converted are highlighted, press **[RETURN]**. The **Rasterize Setup** dialog will appear.

3. To specify line width, you can enter a **Fixed** line width, specify it graphically by drawing a **Box** or use a **Table** of vector color-to-raster line width correlations.

Fixed Line Width:

> Select the **Fixed** radio button. Type a value (in mm or inch) as the width to be used for all the vector lines to be rasterized, or use the value in the **Fixed Width** text field. If you enter a value it will remain the default line width during this editing session.

Using the LineWidth Configuration Table:

a. Select the Table radio button and the Table Setup button. The LineWidth Configuration Table appears, correlating CAD color numbers and line widths expressed in current units.
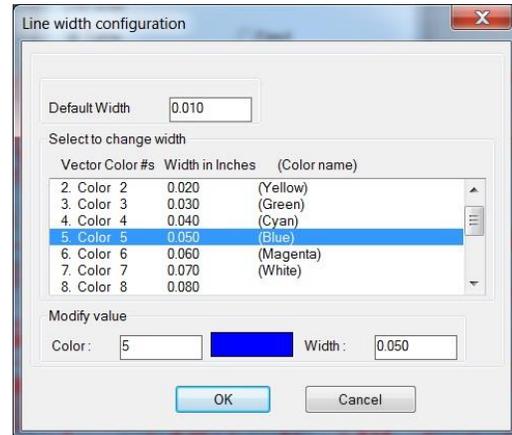
b. To change the width associated with a particular color, select that ACAD Color row. Enter the value in the Width text field. Select OK to close the dialog or select another row to change its width.

```
Enter width <.XX>:
```

c. Enter the value for the width in the units being used and press **[RETURN]**. The Width Table will reflect the changes.

d. Select another color number to change its width or press [0] and **[RETURN]** to continue the command.

The entities will be changed to raster with the widths defined by their colors.

Using Box

a. Select **Box**.

```
Select first corner of box:
```

b. Select a point.

```
Select opposite corner of box:
```

c. As you move the cursor, a rubber-banding rectangle appears with one corner at the point selected. Move the cursor until the box is the desired width and the length exceeds the width. Select the point.

The best way to use this feature is to pick a point on the edge of an existing horizontal or vertical raster line and stretch the box along the line. You can then easily match the width. The length of the rectangle can be any size that is longer than the width as the smallest side is used as the width measurement. A reference rectangle is used instead of a square to make it easier to see. As soon as the second point has been selected, the rasterization takes place.

The highlighted entities will be converted to raster. Some residual vector markings overlay the raster until you redraw the image.

Using Pick

a. Select a vector entity

b. Select the **Pick** option

```
Select raster <width>:
```

c. Next select a raster entity or enter a width

A dialog box will appear with a fixed width value of the raster entity selected.

d. Select **OK**

The vector entity selected will be converted to line width of the previous raster that was selected using the **Pick** command.

GTXImage CAD Prompts

1. Select **gRASTER**.

```
Select objects:
```

2. Select the vector entities to be converted. When all entities to be converted are highlighted, press **[RETURN]**.

```
Raster width Box/<Table>/.01:
```

3. There are three ways to respond to this prompt. You can 1) type in a value as the width to be used for all vector lines to be rasterized for plotting; 2) use the LineWidth Configuration Table to correlate CAD color numbers with line widths; or 3) use the Box option to specify a standard linewidth graphically.

Fixed Line Width:

a. You can type in a value (in mm or inch) as the width to be used for all the vector lines to be rasterized. The value in parentheses is the default value or one previously used. If you enter a value it will remain the default line width during this editing session.

Note: If you defined the width earlier in the session, that value will appear as the default. You can press [RETURN] to accept this value or enter a number to represent the new width. The selected entities will all be converted to raster with the designated width.

Using the LineWidth Configuration Table:

a. If you type in "T" or "Table", the LineWidth Configuration Table appears. The listing correlates color numbers and line widths expressed in current units.

b. If you defined the width previously in this session with a typed value or the "box" option, that value will appear as the default. You can press **[RETURN]** to accept this value or enter a number to represent the new width. The selected entities will all be converted to raster of the designated width.

Note: The original default option is to use the Width Table set up by colors. This allows you to set different widths for entities being rasterized according to their color. If you press [RETURN] or [T], the Width Table will be displayed. The widths will be in inches or millimeters according to the unit set in GTXRAST.CFG.
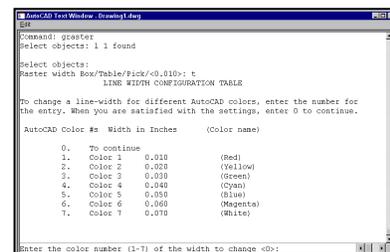
c. Enter the number and press **[RETURN]**

```
Enter width <.XX>:
```

d. Enter the value for the width in the units being used and press **[RETURN]**. The Width Table will reflect the changes.

e. Select another color number to change its width or press [0] and **[RETURN]** to continue the command.

Note: The width settings you change will remain in effect until you change them or you exit GTXImage CAD.

The entities will be changed to raster with the widths defined by their colors.

Using Box

a. At the prompt, type "B" or select "Box" from the menu.

`Select first corner of box:`

b. Select a point.

`Select opposite corner of box:`

c. As you move the cursor, a rubber-banding rectangle appears with one corner at the point selected. Move the cursor until the box is the desired width and the length exceeds the width. Select the point.

The best way to use this feature is to pick a point on the edge of an existing horizontal or vertical raster line and stretch the box along the line. You can then easily match the width. The length of the rectangle can be any size that is longer than the width as the smallest side is used as the width measurement. A reference rectangle is used instead of a square to make it easier to see. As soon as the second point has been selected, the rasterization takes place.

The highlighted entities will be converted to raster. Some residual vector markings overlay the raster until you redraw the image.

Notes:

1. The next time you use gRASTER, the value of the width you used will be the default fixed line width.

2. If the width value is less than 1 pixel, the raster width will automatically be reset to 1 pixel, regardless of the way the width was set.

3. Polylines and other non-filled entities that have a width attribute will be drawn with the CAD width attribute.

4. Rasterization of certain entities may take a while to process. Rasterizing text and ellipses takes longer than lines, for instance. Select less data at a time to improve rasterization speeds.

5. gRASTER does not convert 3-D entities

CALS Plotter Driver

An alternative method for rasterizing vector data is to use the Raster File Format HDI plotter driver. This can be configured using the Autodesk Plotter Manager from the Windows Control Panel.

gRECTANGLE

Type	Menu	Availability
GRECT, GRECTAN or GRECTANGLE	GTX Edit>Draw Rectangle	All Products

Icon:

gRectangle draws a raster rectangle in the current raster image.
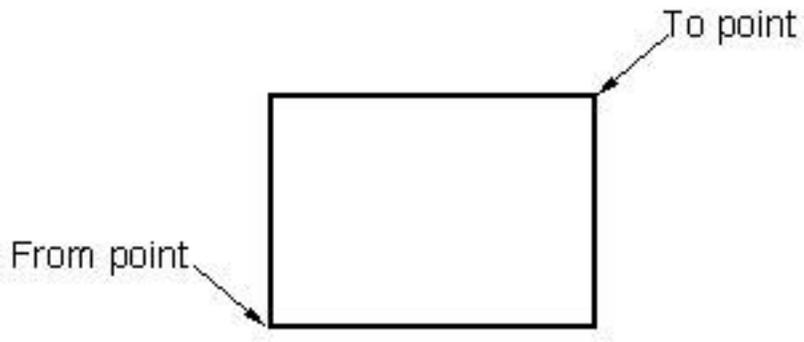
Instructions:

Command: grectangle

Line width <0.010>/Box/Pick:

1.Select first corner of window:

    Select or type the first corner point.

2.Select opposite corner of window:

    Select or type the second corner   point.

gREDUCE

Type	Menu	Availability
gREDUCE gRED	GTX CONVERT>Reduce Color Image	All Products

Icon:

gREDUCE reduces the number of colors in an image to a more manageable number of colors (between 4 to 256 colors).
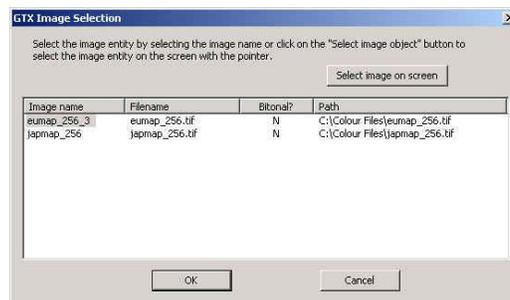
The gREDUCE command only works when you have one or more color (or grayscale) images loaded.

Instructions:

Invoke the gREDUCE command.

If you have no color/grayscale images loaded & you will get an error telling you that you don't have a suitable image for gREDUCE to work on.
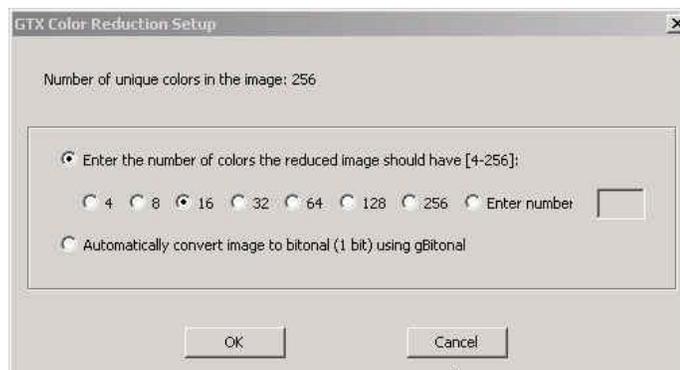
If you have more than one color/greyscale image loaded you will be offered the 'Image selection' dialog box. You can then either select the image by name or you can pick it interactively using the 'Select Image on Screen' button.

Once the Image has been selected the system will display the color reduction dialog. You can select the numbers of colors you wish to reduce to from a preset list in the range of between 4 to 256 colors.

Or you can specify the exact number of colors.

Or you can reduce to a bitonal image.

Notes:

gREDUCE only works if you have a color (or greyscale) image loaded.

You cannot use the command to increase the number of colors.

gREFLCT

Type	Menu	Availability
GREF or gREFLCT	GTX Enhance>Reflect	All Products

Icon:

gREFLCT reflects the raster within an image about the X or Y axis. This command can be useful if an image was scanned through the back of transparent media.

Instructions:

1. Select gREFLCT.

   ```
   Select axis to mirror data about <Xaxis>/Yaxis:
   ```

2. Select "X" or "Y" or press **[RETURN]** to select the default value. The entire image is flipped about the chosen axis and the raster limits are rotated to the new orientation.

gRELIMIT

Type	Menu	Availability
GREL or GRELIMIT	GTX Vector Clean>Relimit	GTXImage CAD PLUS

Icon:

gRELIMIT combines the effects of both the TRIM and EXTEND commands for vector lines

Instructions:

```
Select line end point:
```

1. The **End** AutoSnap becomes enabled to help select endpoints. Select the end of the line you want to trim or extend.

```
Select Object:
```

2. Select the boundary line.

The line will lengthen or shorten to meet the boundary.

If the line cannot shorten or lengthen to meet the boundary, the following message is displayed:

```
Object does not intersect an edge.
```

gRESIZE

Type	Menu	Availability
GRE or GRESIZE	GTX Enhance>Resize	GTXImage CAD
		GTXImage CAD PLUS

Icon:

gRESIZE changes the size of the raster work area. You can increase the dimensions in both positive and negative directions. This is useful to accommodate pasting in additional raster files, scaling up the drawing, or making edits that would move data beyond the existing limits. You can also reduce the work area to reduce memory requirements; for instance, when the actual data you need occupies only a small portion of the work area.

Instructions:

1. Select gRESIZE.

```
This command could fail in limited memory situations
Do you want to save the raster image first? Yes/<No>:
```

2. If you have not recently saved your data, respond "Yes". You will be prompted for a file name and then returned to the command.

```
Select lower left corner of raster limits <X.XXX,X.XXX>:
```

3. Select a point with the mouse or type in the coordinates for the point to be the lower left corner of the work area. Negative numbers are acceptable.

```
Select upper right corner of raster limits:
```

4. Select a point with the mouse or type in the coordinates for the point to be the upper right corner of the work area. Make sure that all the existing raster data is enclosed in the resulting window. Any raster data not included will be eliminated from the database.

Lines marking the raster work area move to the new boundaries. Existing data retains the same coordinates as before.

Warning: When resizing an image, consider the drawing's scale. Raster images will normally be about the same size as paper media. Ensure your image remains within a normal "paper" size and then scale it against your vector data. You do not want to create a raster image the size of a house or a map - it will take a long time if it works at all!

---

gROTATE

Type	Menu	Availability
GR or GROTATE	GTX Raster Edit>Rotate	GTXImage CAD GTXImage CAD PLUS

Icon:

gROTATE rotates selected raster data. You will define the center point and a rotation angle to specify the new orientation for the data. The rotation angle may be selected by entering a value or by picking two points to indicate the angle graphically. gROTATE can also be used to "deskew" a specified section of a drawing.

Options:

Rotation Angle   Specify the rotation angle with respect to the X axis.

Reference Angle   Draw a new base line and indicate the desired angle for this line.

Instructions:

1. Select gROTATE.

   ```
   Select raster <WINDOW>:
   ```

2. Select the raster data to be rotated. When everything you want to be rotated is highlighted, press **[RETURN]** to continue the command.

   ```
   Base point:
   ```

3. Select a point on the entity to be used as the center point for the rotation process. The entity will turn about this "anchor" point as it rotates. It will also be used as the base point for a graphic representation of the rotation angle.

   ```
   Rotation angle <0.0>/Reference:
   ```

4. There are several ways to respond to this prompt. You can enter the default value, a new rotation angle, or use reference angles to graphically designate the rotation.

   Using the Default Angle

a. The default angle, initially zero degrees, reflects the value used for a previous **gROTATE** operation. You can accept the default angle by pressing **[RETURN]**.

   ```
   Delete old objects Yes/<No>:
   ```

   Using Rotation Angle

a. You can specify the angle with reference to the X axis by typing in a value (in degrees) or by rubber banding the reference line.

b. When you designate the angle by either method, the raster will be redrawn in the new orientation.

   Using Reference Angle

a. Enter "R" or select "Reference" from the menu to use a different base line (not the X axis) for the graphic angle.

   ```
   Reference angle <0.0>:
   ```

b. You can either enter a degree value to define the new base line or define it graphically by selecting a point. If you select a point, you will be prompted for the second point.

```
Reference angle <0.0>: Second point:
```

Select the second point to define the line that should be rotated to a particular angle.

```
New angle:
```

c. Type in the value for the angle the new base line should take. The selected data will rotate the same amount. The angle of rotation will actually be the angle between the reference line positions. For example, if the rotation should bring a certain line to a vertical position, you could draw the base line over that line and then type "90" for the new angle. The default angle will then be 90 minus the angle of the base line.

5. Respond "yes" if you do not want to retain the original data in addition to the rotated copy.

Notes:

1. If raster data is placed so that it is outside of the set raster drawing limits, the portion outside the drawing limits will be lost. If this happens, use UNDO, use gRESIZE to extend the raster limits as necessary, then use gROTATE to rotate the data. Nothing will be lost because all raster is within the new raster limits.

2. Rotating raster data may distort it somewhat. The "line", "arc", and "circle" intelligent raster object picking methods may not work on raster data that has been rotated several times.

3. *gROTATE* can rotate raster right off the image. To rotate an entire raster image 90 degrees, rotate the image entity using the **ROTATE** command, or use gTURN so that the image is automatically resized.

4. Positive values rotate raster in a counter-clockwise direction.

As soon as the rotation angle has been defined by whatever method is used, the highlighted data will rotate as specified about the base point. The command terminates.

gROTRECTANGLE

Type	Menu	Availability
GROTRECTANGLE	GTX Edit>Draw Rotated Rectangle	GTXImage CAD GTXImage CAD PLUS

Icon: ◇

Draw a raster rectangle at any rotated orientation in the current raster image

by defining the rectangle's baseline and height.

 Instructions:

Command: grotrectangle

Line width <0.010>/Box/Pick:

 1.Enter a line width for the raster lines of the rectangle

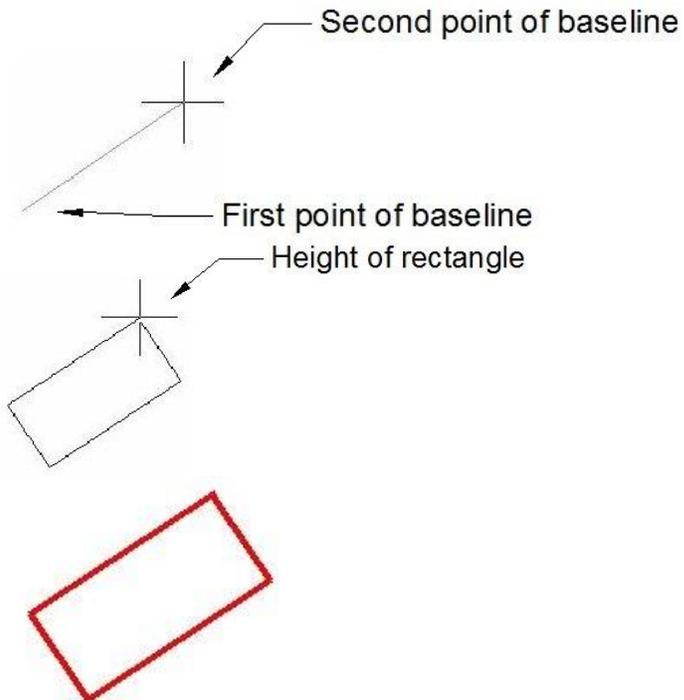**2.**Select insert point of baseline:

   Select or type the first corner point of the baseline.

3.Select end point of baseline

   Select or type the second corner point of the baseline..

4.Select rectangle height

   The raster rectangle is drawn when the height point is selected.

gRUB

Type	Menu	Availability
GRU or GRUB	GTX Raster Edit>Rub	GTXImage CAD GTXImage CAD PLUS

Icon:

gRUB erases raster data after it has been converted or traced to vector entities. You select the vector entities overlying the raster that is to be erased. This lets you use the raster to evaluate the conversion and then erase it easily.

Instructions:

1. Select gRUB.

    ```
    Select objects:
    ```

2. Select the vector entities over the raster to be erased. When all the desired entities are highlighted, press **[RETURN]**. You will see a message indicating the percent of rub completed.

Note: The width at which gRUB erases is determined by the linewidth configuration table. Automatically generated polylines have no actual width, so they are not very effective for erasing underlying raster. *gERASE* *can be more effective in this situation.*

---

gSAVE

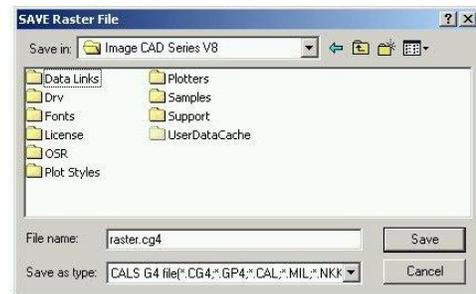| Type	Menu	Availability
GSA or GSAVE	GTX Imaging>Save Raster File	GTXImage CAD GTXImage CAD PLUS

Icon:

gSAVE saves changes made to a raster image.

A filename can be up to 256 characters long. If the extension is not included, the default raster format and its extension (such as ".TIFF", or ".CG4") will be used.

TIFF subformats (uncompressed, G4, etc.) default to the TIFFFORMAT setting in *GTXRAST.CFG*.

Instructions:

1. Invoke gSAVE. The SAVE Raster File dialog appears:

2. Select the output format from the List Files of Type box. Enter the File Name and press OK.

3. If the file name already exists, you will be prompted to overwrite the file or cancel the save.

The drawing is saved.

Renaming an Image

If you saved the image to a new name, note that the image entity still points to the originally attached raster file. If you want the drawing to point to the new filename you must do the following:

1. Select **Insert>Raster Image...** to invoke the **IMAGE** command.

2. Select the GTXARCH image. Above under the **Saved Path** column you can see the original raster filename. At the bottom of the dialog you can see an "Image Found At" text box, which points at the recently saved filename.

3. Select **Save Path**. This tells GTXImage CAD that you want the changed filename as the one to which the image points.

Notes:

The lower left corner of the drawing raster limits is set as the insertion point to be used when the file is loaded or pasted.

Always save when leaving for any length of time and before loading new files. If performing many tasks on a single drawing, it is a good idea to save the files periodically.

---

4. Select **OK**.

Saving Color Images

There is a difference in the way the gSAVE operates depending upon if you are saving a color or black & white (binary) image. Color images have a more restricted number of file saving formats.

Supported Color file formats.

When working with color images in GTX you can only save in

.tif (uncompressed TIFF),

.**bmp** (Uncompressed Bitmap),

.png (Portable Network Graphics),

.pcx (PC Paintbrush) formats.

Also, if you originally loaded a **.jpg** (JFIF JPEG) file you will additionally be offered the option of saving in .jpg (JPEG) format.

gSCALE

Type	Menu	Availability
GS or GSCALE	GTX Raster Edit>Scale	GTXImage CAD GTXImage CAD PLUS

Icon:

gSCALE changes the size of raster geometry within an image entity. You may scale the data about a reference point by entering X axis and Y axis scaling factors or you may scale the raster data graphically.

Instructions:

1. Select gSCALE.

```
Select raster <WINDOW>:
```

2. Select the raster data to be scaled. When all of the data to be scaled is highlighted, press **[RETURN]**.

```
Base point:
```

3. Select a point near the data selected. This point is the reference point to scale the entities about (it will not change its location). It also serves as the first point for the scale rectangle if you use the graphic method of scaling.

```
Scale factors<1.0,1.0>/Fit:
```

The value in the brackets is the default value, either the initial default of 1.0,1.0 (scale by 1 in both X and Y directions) or any values previously used with the **gSCALE** command.

4. There are several ways to respond to the prompt. You can use the default value or enter the numerical scaling factors for X and Y. You can also indicate the size graphically be selecting "Fit".

Using the Default Scaling Factors

```
Scale factors<1.0,1.0>/Fit:
```

a. The default scaling values reflect the scale factors used for a previous **gSCALE** operation or 1.0,1.0 if the command has not been used. You can accept the default scale factors by pressing **[RETURN]**.

Entering Numerical Scaling Factors

```
Scale factors<1.0,1.0>/Fit:
```

a. Enter two values separated by a comma to indicate the X and Y scale factors respectively. To enlarge the image, enter scale factors greater than 1. To shrink the image, enter scale factors between 0 and 1 (Entering "2,2" doubles each dimension and increases the area by a factor of 4. Entering ".5,.5" cuts each dimension in half and decreases the image to 1/4 its original size.)

Using "Fit"

```
Scale factors<1.0,1.0>/Fit:
```

a. After selecting the base point, select "Fit" from the menu or enter "F". A rubber-banding box anchored at the base point will appear.

```
Select original size:
```

b. Rubber-band a reference rectangle from the base point representing the original size of the raster data being scaled.

```
Select new size:
```

c. Rubber-band a new reference rectangle from the base point representing the new size of the raster data. The "Fit" option generates scale factors based on the size ratio between the reference rectangles.

Notes:

1.   *gSCALE* can scale raster data right outside the image entity, and the data is lost! To avoid this, choose the base point to avoid rotating the data outside the limits of the image.

2.   After scaling, you can easily re-select the rotated data and reposition it with *gMOVE* "previous" if necessary - this even works if the rotated data overlaps with preexisting raster.

3.   To expand the image, save the file and then use *gRESIZE* to enlarge the raster work area.

4.   Scaling a raster image with different X and Y factors will cause raster arcs and circles to become elliptical. The resulting shapes cannot be selected using the intelligent raster object "arc" and "circle" picking methods.

---

5.   The data is scaled to the dimensions indicated by the scaling factors. The command is terminated.

gSEPARATE

Type	Menu	Availability
gSEPARATE gSEP	GTX Convert>Separate Color image	All Products

Icon:

gSEPARATE separates out individual colors, or selected groups of colors from a color image, & reduces them to a single binary (1 bit) image. The command can separate out to multiple binary images with each run of the command
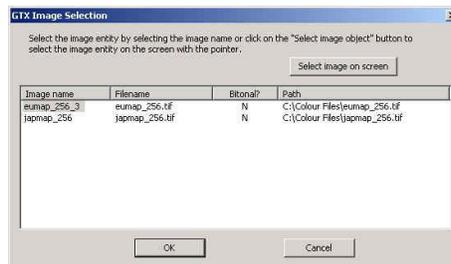
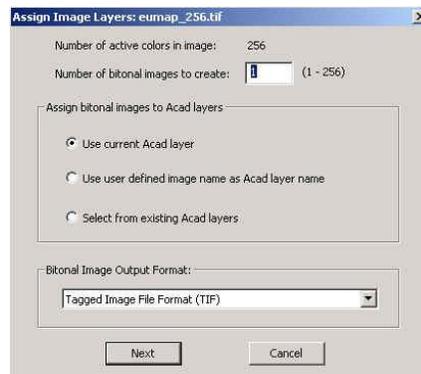The gSEPARATE command only works when you have one or more color (or grayscale) images loaded.

Instructions:

Invoke the gSEPARATE command.

If you have no color/grayscale images loaded & you will get an error telling you that you don't have a suitable image for gSEPARATE to work on.

If you have more than one color/greyscale image loaded you will be offered the 'Image selection' dialog box. You can then either select the image by name or you can pick it interactively using the 'Select Image on Screen' button.

Once the Image has been selected the system will display the 'Assign Image Layers' dialog.

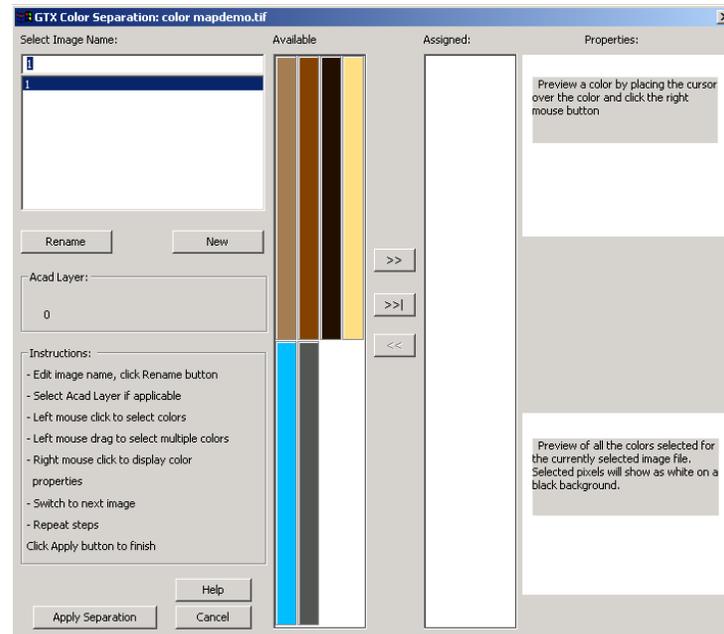You can enter the number of binary images you wish to create (between 1 to 256 images).

The binary images created can be assigned to different layers. The layering naming can be controlled by selecting one of three options:

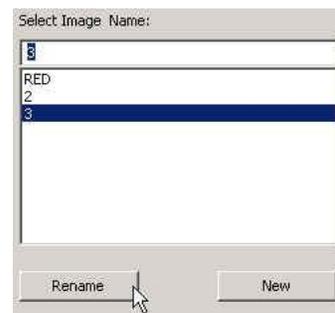1) **Use current ACAD Layer**. In this instance the binary images will be put on the current AutoCAD layer.

2) **Use User defined image name as Acad layer name.** This option will use the layer name to create a new layer with that name & the image will be placed upon that layer.

3) **Select from existing Acad Layers.** This prompts the user to select the layer from the existing list of defined layers, for every new binary image created.

Next you can select the file format for the new binary images. Click on the down arrow to see & select the image format.
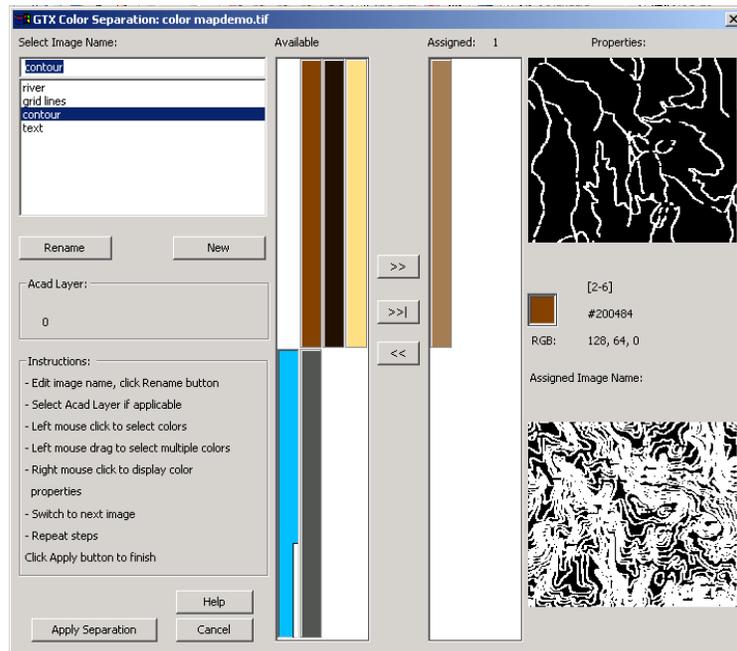
Select **NEXT** to move to the Color Separation dialog.

You can rename your binary image & layer in the top left hand corner of the dialog. By default the image names are numbers 1,2,3 etc. Highlight the layer number & you can enter a new name in the box above. Then select **RENAME** to change the image name. You can also add new images at this stage by selecting the **NEW** button.
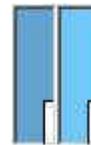

You can now move onto select the colors for a particular binary image.

Firstly highlight the image name you wish to work on. Then using the cursor 'drag & drop' the required colors from the left (Available) payne into the right (Assigned) payne. You can pick the colors individually or in blocks. To select a block of colors **hold down** the left mouse button over the group of colors you wish to select.
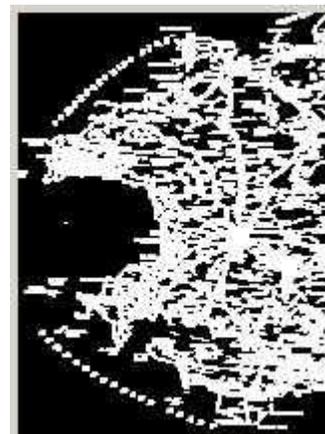
Dialog operation instructions are:

- Edit image name, click **Rename** button

- Select Acad Layer if applicable

- **Left mouse click** to select colors

- **Left mouse drag** to select multiple colors

- **Right mouse click** to display color properties

 - Switch to next image

- Repeat above steps

- Click **Apply** button to finish

Colors with a notch in the bottom Right Hand corner indicate that they are colors that have already been selected on another image.

To help you select the correct colors & information the dialog automatically displays a view of the selected data on that layer.

When you have correctly assigned the colors to your respective images/layers select the **Apply Separation** button to create the new binary images.

Notes:

gREDUCE only works if you have a color (or greyscale) image loaded.

Anti-Aliasing

Some color images are scanned with the 'anti-aliasing' option turned on. Anti-aliasng can enhance the quality of some images, particularly photographs. However it is not ideal for images with solid fill color as It tends to mix colors to achieve a more blurred image.

Whilst the gSEPARATE command will work with anti-aliased images the results may not be as good as required. As such we recommend that, if possible, images be scanned with anti-aliasing turned off.

gSLICE

Type	Menu	Availability
GSLICE	GTX Edit>Slice	All Products

Icon:

GSLICE separates raster objects that should not be joined.
A two pixel wide gap is created along a pick line selected by
a starting point and an ending point.

Instructions:

1.From Point:

   Select or type the first endpoint

2.To Point:

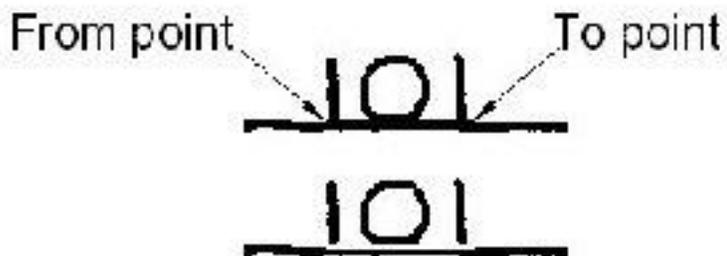   Select or type the other endpoint

3.From Point:

   Select or type the first endpoint

4.To Point:

   Select or type the other endpoint

The gSlice command continues prompting for the starting point of the
next slice until you select **[RETURN]**.

gSMOOTH

Type	Menu	Availability
GSM or gSMOOTH	GTX Enhance>Smooth	GTXImage CAD GTXImage CAD PLUS

Icon:

gSMOOTH smoothes raster objects in a selected portion of the raster data. gSMOOTH can also make raster lines orthogonal or parallel to a specified angle, combine fragmented raster lines. gSMOOTH can smooth raster linewidth to a fixed value or snap raster linewidth to a series of user-defined linewidths. By smoothing raster, gSMOOTH can reduce the file size by up to 30 percent.

Instructions:

Dialog:

Conversion Type indicates the type of raster-to-vector conversion being used to smooth the raster. CAD interprets raster as lines, arcs and circles. Contour interprets raster as polyline contours.

Ortho improves the drawing by snapping lines to horizontal or vertical.
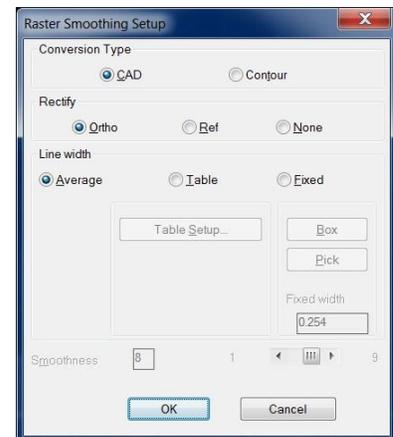
None smoothes the data exactly as it appears without any rectification.

Reference snaps lines close to any reference angle to that angle and parallel lines at this angle separated by a distance close to the offset distance will snap to the offset distance.

Line Width Options

Box defines a value graphically to be used for the width of the data selected to be smoothed.

Value applies a literal value to use for the width of the selected data to be smoothed.

Note: Be sure to enter values in current units (mm or inches). The graphically defined width will also be translated to mm or inches when it appears as the default.

Average smoothes raster geometry to an average of that segment's width.

Table snaps the lines to the nearest value listed in the Width Table.

GTXImage CAD Prompts

1. Select **gSMOOTH**.

   ```
   Select raster <WINDOW>:
   ```

2. Select the raster to be smoothed. When all the data to be converted is highlighted, press **[RETURN]**.

   ```
   Width Box/Table/<Average>/0.01:
   ```

3. There are several ways to respond to the prompt. You can enter a value to be the fixed line

width for all selected data or you can select this value graphically with the "Box" option. You can average each raster element's width or you can set up a Table of LineWidths to which raster elements are snapped.

Using a Fixed Line Width

a. The value shown in the prompt is the value used last or the one set for **RasterWidth** in the configuration file. To accept the default value, press **[RETURN]**. Type a number that represents the width in current size units.

Using Box

a. At the prompt, type "B" or select "Box" from the menu.

```
Select first corner of box:
```

b. Select a point on the edge of a horizontal or vertical raster line.

```
Select opposite corner of box:
```

c. As you move the cursor, a rubber-banding rectangle appears with one corner at the point selected. Stretch the box along the line and match the width. The length of the rectangle can be any size that is longer than the width as the smallest side is used as the width measurement. Select the second point.

Using The Average Width

a. At the prompt for width, type "A" or select **Average** from the menu. Each element will be smoothed to its average width. If parts of a line are significantly different in width, they will be averaged separately.

Using The LineWidth Table

At the prompt for width, type "T" or select Table from the menu. The LineWidth Table will be displayed. The widths will be displayed in the units set in *GTXRAST.CFG*. This allows you to set up to seven different widths to which the lines snap.

Note: The colors do not affect raster smoothing. In this command they only determine the title under which a line width is defined.
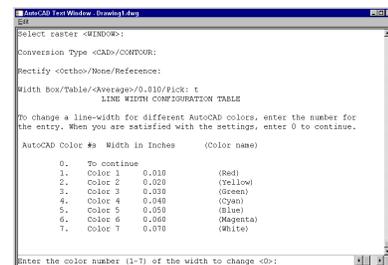
---

c. Enter the value for the width in the size units being used (inches or millimeters) and press **[RETURN]**. The Width Table will reflect the new value.

d. Select another color number to change its width or type "0" and **[RETURN]** to continue the command. You can set up to seven different widths.

```
Rectify <Ortho>/None/Reference:
```

4. There are several ways to respond to the prompt. The default is orthogonal rectification. You can create reference angles to snap lines to a particular non-orthogonal angle (such as hatching) or you can use no rectification.

Using Orthogonal Rectification

a. If "Ortho" is the default mode, just press **[RETURN]** to accept the default of orthogonal rectification. Otherwise, type "O" and **[RETURN]**. All lines within a few degrees of 0 and 90 degrees will be rectified to 0 and 90 degrees.

Using Reference Angle(s)

a. Type "R" or select **Reference** from the menu. The option will prompt you to draw a line showing the reference angle to which lines will be snapped.

```
First point for snap angle:
```

b. Select the first point to define the angular line.

```
Second point for snap angle:
```

c. Select the second point to define the angle.

```
Parallel alignment offset <0.0>:
```

d. This value determines the distance between the imaginary grid lines parallel to the given reference angle. Type in a value in current drawing units. Using the (0.0) value which is always the default means no grid snapping for this angle.

```
First point for snap angle:
```

e. You can define another reference angle and grid or press **[RETURN]** if you do not want to use another one. When you are finished defining the angles, press **[RETURN]**.

5. After you respond to the last prompt, the conversion will take place and you will see the message "Processing Image..."

gSPECKL

Type	Menu	Availability
GSP or GSPECKL	GTX Enhance>Speckle	All Products

Icon:

gSPECKL filters and removes speckles from the raster image. gSPECKL can also fill in small holes in solid raster.

Options:

Pick Options

gSPECKL allows only Window, PWindow or View raster picking to and to add or remove raster data to/from the selection set.

Mode Options

       Delete   Remove speckles from the raster.

       Fill   Fill in small holes in the raster.

Instructions:

1. Select gSPECKL.

```
Speckle size <0.00>/Box/Pick:
```

2. There are two ways to respond to this prompt. You can enter a value or you can use the "Box" option to graphically specify the speckle size.

   Entering a Value

a. Specify an absolute value and hit **[RETURN]**. This value will represent one side of a square area. The speckles or holes to be affected must be within this tolerance.

   Using a Box

a. To use the graphics method, type "B" or select **Box** from the menu.

   Using Pick

a. Single-click with the left mouse button on a sample speckle in the image. Raster data of that size and smaller will be considered to be speckles.

```
Select first corner of box:
```

b. You may want to window an actual speckle or hole to gauge the proper size. Select a point for one corner of the window or box. A rubber-banding box will appear with one corner attached at the point.

```
Select opposite corner of box:
```

c. Drag the box to the desired size and select the second point. The largest side of the box will be used as the reference dimension to define maximum speckle/hole size.

3. After specifying the speckle size, the following prompt will appear:

```
<Delete Speckle>/Fill hole:
```

4. This sets the mode of the **gSPECKL** command. Select the desired mode from the menu or type its abbreviation. Type "D" and **[RETURN]** for "Delete" or type "F" and **[RETURN]** for "Fill".

    ```
    Select raster <WINDOW>:
    ```

5. Select the area of the drawing to be processed with the "Window", "PWindow" or "View" option. All of the speckles or holes of the designated size or smaller in the window will be highlighted.

CAUTION:

1. Set the speckle size smaller than punctuation marks such as the period (.) and colon (:). If your speckle size is larger than these, they will be erased and valuable information could be lost!

2. With large drawings, memory demands can be great for this command. If you have problems, despeckle portions of the drawing, saving in between (*gSAVE).*

3. In the "Select" mode, the "window" option only adds speckles that are entirely inside the window and "PWindow" selects speckles that are entirely or partially inside the area. In the "Remove" mode, either window option removes data up to the window boundary.

---

6. Continue to create a selection set using "Add" and "Remove" to define the area to process. When you are satisfied with the area selections, press **[RETURN]**. The highlighted speckles or holes in the display will be processed.

gTCONFIG

Type	Menu	Availability
GTCFG or GTCONFIG	GTX Convert>Text Config	GTXImage CAD PLUS

Icon:

gTCONFIG sets up parameters for text IOP and the text conversion commands, gTCONVRT and gTRAIN.

Instructions:

Text Configuration dialog has four panes: Raster Input, Character Recognition, Word Recognition, and Text Output. This is the order in which GTX Intelligent Character Recognition (ICR) functions:

1. ICR analyzes the selected raster text, filters out raster that is too large or small to be text, and gathers string formation information to assist in the detection of space characters.

2. If a piece of raster appears to be two or more touching characters, that raster is broken down into its component characters. Raster text is compared to samples recorded in character recognition (.ICR) files. A guess is made as to each character's identity. Each guessed character is assigned a **confidence level**.

> If only character recognition is being performed, characters with a confidence above the Auto-Convert Threshold are automatically converted into text. Characters with a confidence below that threshold are presented in the Character Verification dialog.

3. When Word Recognition is performed, the Auto-Convert Threshold is not used. Instead, characters are formed into strings which are then compared to words and patterns in the dictionary files (.DCT). These files contain words and patterns that are likely to appear in converted drawings.

> If a recognized word is close to one found in the dictionary, but a character is missing or perhaps incorrectly recognized, that character can be changed so the word matches the one found in the dictionary. One threshold, Force Fit, controls how close a word has to be to a dictionary term for it to be changed. Another threshold, Verify, determines how likely the user will be interrupted
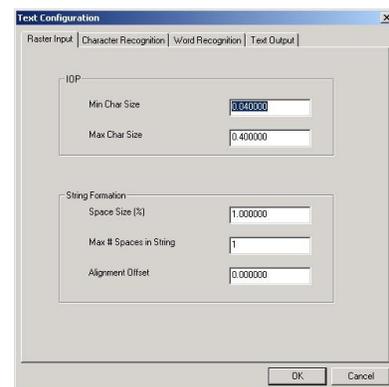
4. When using character recognition (not word recognition), ambiguous characters are shown to the user so that 0 and O, 1 and I, 5 and S, and other pairs of ambiguous characters appear correctly in the CAD text. Regardless of the conversion method, ICR generates AutoCAD text at user-specified size, using the font of the current text style.

Raster Input

The Raster Input pane contains parameters that control the interpretation of incoming raster text characters and space recognition.

IOP

Sets the minimum and maximum size raster text "blobs" that textwin or textline IOP methods are allowed to select. Text IOP separates raster text from touching raster geometry (i.e., text with a touching or intersecting raster underline). Text IOP needs to

know the typical size of text characters so it can tell them apart from their surroundings. It also selects the minimum and maximum size raster that gTCONVRT will attempt to recognize as text. Entities larger or smaller than these values will not be selected by text IOP or will be ignored by text conversion if another selection method was used.

Min Char Size

Sets the smallest size of raster that text IOP will select or that gTCONVRT will attempt to recognize. The default value is set by the CharMinSize parameter in *GTXRAST.CFG*.

Max Char Size

Sets the largest size of raster that text IOP will select or that gTCONVRT will attempt to recognize. The default value is set by the CharMaxSize parameter in *GTXRAST.CFG*.

Note: Set the Min Char Size between the sizes of speckles and of typical period/full stop raster characters ("."). That way, speckles are not converted but small text characters are converted. Also, if there are many "touching" Characters, you may want to double or triple the Max Char Size so that gTCONVRT does not ignore touching raster text.

String Formation

Controls how gTCONVRT merges separated raster strings by inserting spaces in recognized text, how it breaks colinear text into separate CAD text entities.

Space Size

Sets the distance between two characters (in terms of the percentage of a character's width) required before a space character is inserted between the two text strings.

Max # Spaces

Sets the maximum number of the above-mentioned space characters allowed before gTCONVRT separates the text into separate ASCII text entities. If two text strings are far enough apart, they will exceed this "breaking point" and be portrayed by two separate CAD text entities.

Alignment Offset

Sets the minimum vertical distance (relative to both of the text strings' orientations) between separate lines of text in terms of a percentage of a typical character's height. By default, this parameter is set to 0%, meaning that the top of one character could be level with the bottom of another and they would still be merged, into the same CAD text string.
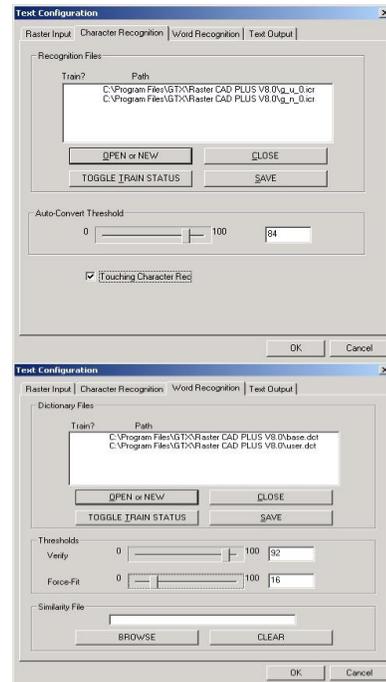
Character Recognition

Recognition Files: Lists the *.ICR* files being used by gTCONVRT to recognize raster text.

Open/New: Opens a recognition file or creates a new recognition file and sets it for training.

Close: Closes the selected recognition file.

Train: Sets the selected recognition file for training.

Save: Saves a trained recognition file.

Word Recognition

Dictionary Files

OPEN or NEW – Opens an existing ASCII dictionary file (*.dct) or creates a new one from scratch.

TOGGLE TRAIN STATUS – Controls whether the selected dictionary file is being trained. When you make corrections during gTCONVRT, new words will be added to the dictionary file that is being trained. These may later be edited with a standard text editor, such as Windows Notepad.

CLOSE – Closes the selected dictionary file.

SAVE – Saves changes made to the selected dictionary file (only if it was being trained).

Thresholds

Verify – Controls the likelihood that gTCONVRT will display the text verification dialog to verify its guess at what a given word was. If the average confidence of all characters in the text string falls below this value, the text verification dialog will appear.
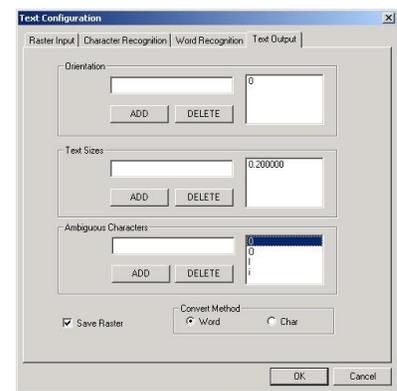
ForceFit – Controls how hard gTCONVRT will try to change characters to make a given string fit an existing dictionary file word or pattern.

Text Output

2. Text recognition works on one orientation of text at a time. The default orientation is zero degrees. To recognize at 90 degrees, Delete **0** from the orientation *list box*, type **90** in the orientation *text box* and press [RETURN].

3. To use a pre-trained recognition file (suffix is .ICR) select **Open**. The **Open Character Recognition/Training File** dialog appears. Choose the .ICR recognition file and select **OK**.

4. If you want to train this or any other opened recognition file on the new drawing, select **TRAIN**. The letter "T" will appear to the left of the training file.

Note: When you train a character recognition file, you will need to save it afterwards! To save, go back into the text configuration dialog, select the Character Recognition pane, and Save. THE FILE IS SAVED ONLY AFTER YOU CLOSE OUT OF THE TEXT CONFIGURATION DIALOG! After selecting SAVE, select OK to make the dialog save the file.

---

5. If training a new recognition file, set the **Auto-Convert Threshold** to **100** to force the **Character Validation** dialog to appear for every character. If working with an existing recognition file, a value of 70 to 90 will be preferable so the dialog only appears for uncertain characters.

6. Set the values for IOP minimum and maximum character size to limit the raster selected with Text IOP.

7. The **String Formation** parameters should work well at the default settings.

8. Leave the **Save Raster** button un-selected to keep raster text as it is being converted to AutoCAD text.

9. Select **OK**.

Orientation   Sets the text orientations allowed to be recognized from raster text characters. The default list is established by the CharOrient

---

parameter in *GTXRAST.CFG*.

Ambiguous Characters

Lists characters that could be easily misrecognized for another (i.e., the letter "O" could easily be drawn the same way as the numeral "0". Therefore, it is common for each of these characters to be included in this list).

Auto-Convert Threshold

This is a simple but important parameter. It determines how frequently the gTCONVRT command asks you to validate its guess at characters. If the confidence level of a recognized character is below this Auto-Convert Threshold, then the Character Verification dialog will appear, allowing you to see the character for yourself and override the program's attempt at recognizing the character.

Notes:

1. The Auto-Convert Threshold is the Confidence Level above which a character must be recognized for it to be automatically converted to text without using the verification dialog. This is set by the *CharThreshold* parameter in *GTXRAST.CFG*.

2. With a trained recognition file, gTCONVRT should accurately recognize legible characters. Recognition of text in messy drawings may be unreliable.

3. gTCONVRT assigns a *Confidence Level* from 0 to 100 to each character. Confidence measures the degree to which a raster character matches those on which the recognition file has trained.

Text Sizes   Lists the sizes allowed for text resulting from raster text recognition during **gTCONVRT**. Resulting text will be snapped to the nearest perceived text size. The default list is established by the **TextHeight** parameter in *GTXRAST.CFG*.

Save Raster   If deactivated, gTCONVRT will delete raster text as it is converted to CAD text. We recommend you normally leave this parameter on so the original raster text remains for final text checking purposes/QA.

gTCONVRT

Type	Menu	Availability
GTC or GTCONVRT	GTX Convert/Text Convert	GTXImage CAD PLUS

Icon: R▶T

gTCONVRT converts raster text to CAD text entities. Invokes the text editor window, allowing you to systematically check the results of the recognition process. This command can perform two tasks:

• It converts selected raster text to CAD text entities.

Warning: The Character Verification dialog is actually a separate windows application. Because of this, if you happen to click somewhere within GTXImage CAD's window, the text verification dialog will disappear. Nothing seems to respond & it looks like GTXImage CAD has locked-up. It has not but to free it up use [ALT-TAB] to return to the Character Verification application if this happens.

• If a recognition file is set for training, it can train it on new text characters.

Character Recognition Instructions:

1. Invoke **gTCONVRT**.

2. If **gTCONFIG** has not yet been invoked during the GTXImage CAD session, **gTCONVRT** invokes it before beginning.

   ```
   Select raster <WINDOW>:
   ```

3. Use the best IOP method for selecting the desired raster text. TextWin and TextLine were especially designed for this command, though Fence, Inside and Window also work.

4. **gTCONVRT** recognizes and assigns a confidence level to each separate raster "blob" selected in step 3.

5. Two kinds of characters will appear in the **Character Verification** dialog; (a) those that were recognized at a confidence below the **Auto-Convert Threshold** or (b) those that were listed as "Ambiguous" in the **gTCONFIG** dialog.

   Character Validation: character recognition thought this "D" was an "O".

6. Depending on the results of character recognition, accept or correct the character.

Using the Character Validation Dialog

Program's Guess	Action
Correct	Press Return
Wrong	Type the correct character(s) and press **[RETURN]**
"Spurious" (The raster is not really a character)	Select **Ignore**. Character will not be trained and will be recognized as a character.

7. Characters whose confidence is above the Auto-Convert Threshold are automatically converted to CAD text, bypassing the Character Validation dialog.

8. **gTCONVRT** ends. To edit or change resulting text, use the **Edit/Properties** or **Change** commands.

Options:

Ignore   Ignores the current character. Use this if a single character is represented by two separate pieces of raster - ignore one half and correct the other so the correct text is written.

Cancel   Ends recognizing all raster in the selection set. Returns to command prompt. No text entities result.

Train CR   Setting a recognition file for training in gTCONFIG activates a "Train CR" section in the lower left corner of the Character Verification dialog.

If training a recognition file, the Character Verification dialog will train on each character that appears. If the raster that appears is a poor representation of that character (for example, a character that was scanned poorly and is either missing a part or is attached to some geometry), select No before accepting or correcting it. This way character recognition is not trained on misleading raster text.

Word Recognition Instructions:

To recognize words rather than individual characters, set the **mode** switch to Word on the **Text Output** pane of the text configuration dialog (See **gTCONFIG**, page 488).

Word recognition otherwise works very similarly to character recognition, except that a word verification dialog appears instead of the character verification dialog. You are given the opportunity to change the word before it is converted into CAD text.

You can set a dictionary file to train during conversion. This adds new words to the training dictionary, which is really just an ASCII text list of words that are likely to be in your drawings. Open *BASE.DCT* or *USER.DCT* in Notepad for an example.

gTEXT

Type	Menu	Availability
GTEXT	GTX Edit>Raster Text	GTXImage CAD
		GTXImage CAD PLUS

Icon:

Type raster text in the current image aided by the Text Formatting control.

Instructions:
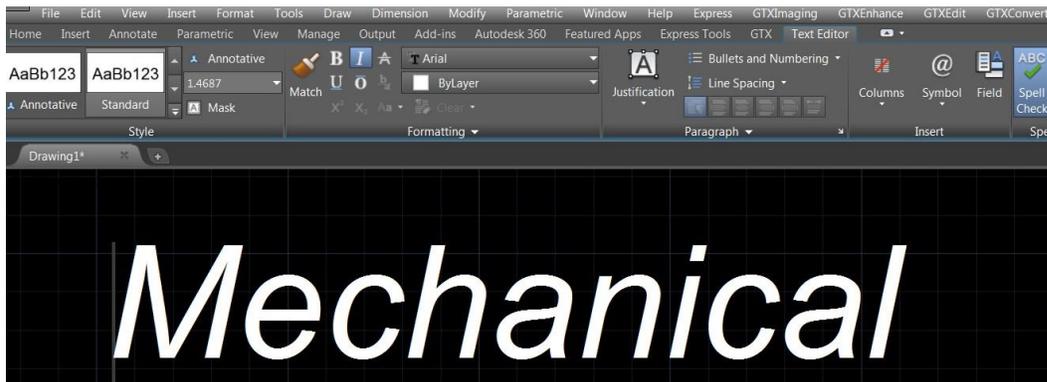
Command: gtext

1.Specify start point of text:

The start point is the bottom left point of the first character in the text string.

2.Specify height:

Enter a numeric value for the height of the text characters.

3.Specify rotation angle of text:

Text can be entered at any orientation angle that is specified. The default angle is zero degrees. After selection of the rotation angle the Text Formatting control appears with the In-Place Text Editor. Begin adding text entries. The keyboard Return key will add a new line for text entry.

4.To end editing click the OK button on the Text Formatting toolbar or click outside the In-Place Text Editor. After closing the text editor the raster text will appear.

-Raster text is created upon closure of the text editor

-Raster text with a vector text overlay is created if the GTX gHybridmode command was enabled before initiating the gText command.

gTRACE

Type	Menu	Availability
GT or GTRACE	GTX Convert/Trace Vectors	GTXImage CAD GTXImage CAD PLUS

Icon:

Function: gTRACE traces over raster objects with vector entities. For a straight line, two points are picked to determine the extents of the raster line. Circles and arcs can also be copied to vector by picking three points.

Note: If vectorizing raster geometry depicted in a special linetype, you might want to establish a current layer with that linetype first. The resulting vector geometry is placed on the current CAD layer and is rendered in its linetype.

Vector entities will be created using the current image and linetype settings.

Options:

Line   Default option. Trace a raster line or series of lines with one vector line. Intersections are preserved. If points are picked beyond the endpoints of a line it will not be extended.

Circle   Trace a raster circle or series of arcs with a vector circle. A raster circle or arc selected with the "circle" option will be traced as a vector circle.

Arc   Trace a raster arc or series of arcs with a vector arc. If points are picked on a portion of a line or raster arc, only the portion between the points will be traced. You can use the "arc" option to trace part of a circle, part of an arc or a whole arc.

Instructions:

1. Select **gTRACE**.

   ```
   Select raster <LINE>:
   ```

2. The default mode begins as "line". Select another from the menu or type the first letter of the name and press **[RETURN]**. You will be prompted to select points.

Using Line

a. Select the first point at one end of the line or line segments to be traced.

   ```
   Select other end of reference line:
   ```

b. Select the second point on the opposite end of the line or line segments that you are tracing.

c. Continue to trace another line or select another option. Press **[RETURN]** to exit the command.

Using Circle

a. At the prompt, select "circle" or type it followed by **[RETURN]**. You will be prompted to select three points to define the circle to be copied.

   ```
   Select point on circumference of circle:
   ```

b. Select a point on the circumference of the circle.

   ```
   Select second point for circle:
   ```

c. Select another point on the circumference of the circle.

```
Select third point for circle:
```

d. Select a third point on the circumference of the circle. The circle will be overlaid with a vector circle on the current layer. Continue to trace another circle or select another option. Press **[RETURN]** to exit from the command.

Using Arc

a. At the prompt, select "arc" or type it followed by **[RETURN]**. You will be prompted to select three points as if you were drawing a three-point arc.

```
Select start point of reference arc:
```

b. Select a point at one end of the arc.

```
Select point on arc:
```

c. Select a point near the midpoint of the arc.

```
Select end of reference arc:
```

d. Rubber-band the arc so that it overlays the arc to be traced and select the point.

```
Select raster <ARC>:
```

e. The arc will be overlaid with a vector arc. Continue to trace another arc or select another option. Press **[RETURN]** to exit from the command.

gTRAIN

Type	Menu	Availability
GTR or GTRAIN	GTX Convert/Text Train	GTXImage CAD PLUS

Icon:

While training files in gTCONVRT, mistakes can be made. gTRAIN searches through a recognition file and lets you correct or delete mistakes made during training.

Warning: The Character Verification dialog is actually a separate windows application. Because of this, if you happen to click somewhere outside this dialog in the AutoCAD window it will appear to disappear and lock up AutoCAD. If this happens, use [CTRL-ESC] or [ALT-TAB] to return to the Character Verification application.

gTRAIN searches through a training recognition file for specified characters or orientations of characters and displays these in the Character Verification dialog. It displays each raster text sample, allowing you to keep, change or discard each character sample. Changes are stored to the recognition file when it is saved and closed within **gTCONFIG**.

Note: If you select both of these options, gTRAIN will search for the specified characters at the specified orientations.

Search Options:

Training Characters   Searches for specified characters.

Orientations   Searches for characters at specified orientations.

Editing Options:

Correct   Replace the identity displayed by the Character Verification dialog with a new character identity (an "F" was trained as "P"; typing F will correct the typo).

Keep   Accepts the character evaluation of the sample raster text.

Discard   Discards the current sample raster text.

Cancel   Cancels the gTRAIN command, making no further changes. Changes before selecting Cancel are made to the *.ICR* file.

Instructions:

1. Select **gTCONFIG**.

2. The Text Configuration dialog appears.

3. Open a recognition file or select one that is already loaded.

4. Set the file to **Train**. "ON" appears below the **Train** button.

5. Select **OK**.

6. Invoke **gTRAIN**. The Text Training dialog appears:

Text Training dialog

7. If you leave the "Training Characters" and "Orientations" check boxes empty, gTRAIN will display every character in the training recognition file. The characters to be displayed can be limited to certain characters or orientations.

8. To limit the search to only certain characters select **Training Characters** and select **Edit List...**. The Text Training Characters dialog appears:

Text Training Characters dialog

9. Build a list of individual characters to observe with gTRAIN. Enter characters separately.

Note: If you enter a multi-character string the command will search for raster characters that have been recognized as that string (i.e., you could type the unicode \U+2205 to search for the unicode text diameter symbol if you had previously trained that using gTCONVRT).

---

10. To limit the search to characters at specific orientations check the **Orientations** box and select **Edit list...**. The Text Orientations dialog appears:

Text Orientations dialog

11. Add each required orientation to the list, then select OK.

12. Back in the Text Training dialog, verify that the lists are correct, then select **OK**. The Character Verification dialog appears:

Character Verification dialog

The dialog will present each raster text sample, its trained identity and its orientation (i.e., In 104, the raster is displayed in the top half of the dialog and the character is recognized as the number "7" at 45 degrees in the lower half).

If the evaluation is correct, as with gTCONVRT, press [RETURN] or select KEEP to accept the identity of the raster text sample and proceed to the next sample.

If the evaluation is incorrect, but is still a good example of that character at its orientation, make the necessary corrections to the text id and its orientation, then press [RETURN] or select KEEP to make the change and proceed to the next sample.

If the evaluation is incorrect and is also a bad example (unidentifiable or at least very messy) select Discard to remove the text sample from the recognition file.

To cancel the gTRAIN command, select [CANCEL]. All changes made prior to cancelling the command will have been saved in memory.

To Keep Your Changes

gTRAIN does not automatically save changes to the recognition file. To write the changes to the file invoke gTCONFIG and save the training recognition file. If you choose to keep the recognition file as it was when it was first loaded, you can close it without saving changes.

gTURN

Type	Menu	Availability
GTU or GTURN	GTX Enhance/Turn	GTXImage CAD GTXImage CAD PLUS

Icon:

gTURN quickly rotates raster data in the active image by a multiple of 90 degrees. This also swaps the image's X and Y raster limits.

This command is useful when you are working with images that have been scanned sideways and appear sideways when attached within GTXImage CAD.

Options:

90   Rotate the raster data counterclockwise 90 degrees.

180   Rotate the raster data counterclockwise 180 degrees.

270   Rotate the raster data counterclockwise 270 degrees.

Instructions:

1.   Select **gTURN**.

   `Specify rotation angle <90>/180/270:`

2.   Enter the angle or press **[RETURN]** for the default. The image raster data is rotated.

gVECTOR

Type	Menu	Availability
GV or GVECTOR	GTX Convert/Draw Vectors	GTXImage CAD GTXImage CAD PLUS

Icon:

Function: gVECTOR traces raster with lines, arcs or circles like **gTRACE**. Unlike **gTRACE**, **gVECTOR** erases originating raster data.

gWARP

Type	Menu	Availability
GW or GWARP	GTX Enhance/Warp	GTXImage CAD GTXImage CAD PLUS

Icon:

gWARP corrects distortion in a raster image by stretching the image data between source and destination points.

Options:

The Image Warp dialog establishes warp source and destination point pairs. These point pairs define the way the image is to be stretched.

Add Point Pairs creates one pair at a time.

Generate Grid creates a lower left corner, an upper right corner, and intermediate grid points.

Instructions:

Warping requires the definition of source and destination point pairs. When CMDDIA is set to 1, the Image Warp dialog allows the creation of individual point pairs or an entire array of them. When CMDDIA is set to 0, a series of prompts gathers the same information.

Add Point Pairs

The Add Point Pairs button lets you add individual pairs of source and destination points.

```
Source point on the drawing (1):
```

1. Type or select a point on the image representing a known coordinate.

```
Warp point (#.###,#.###):
```

2. Type or select the real location to which the source point is to be warped or displaced.

```
Source point on the drawing (2):
```

3. Enter another source point or press [RETURN] to go back to the **Image Warp** dialog.

The specified points appear in the **Coordinate List Table**.

Generate Grid

The Generate Grid button lets you create a rectangular grid of destination points and then for each point, enter the source points. This is backwards from the prompts for Add Point Pairs, but it provides a convenient way of generating a list of known real coordinates and then provides the location of each of those features in the raster image.

```
Select base point of grid:
```

1. This point may be the lower left corner of the image, or one corner of a grid of known coordinates.

```
Select opposite corner of grid:
```

2. Select the corner opposite the base point. This defines the total area the grid will occupy.

```
Select first corner for cell size:
```

3. Select one corner to define the cell size. This might be the same point as the base point.

```
Select opposite corner for cell size:
```

4. Select the other corner of the cell to define its overall size. The grid will fill up with cells whose size is defined by the first and opposite corners.

    For each grid point generated by steps 1-4, the following prompt will appear:
    ```
    New source point on the drawing (#.###,#.###):
    ```

5. A line will be rubber-banding from one of the grid points. Select the raster point nearby that is supposed to be representing this real point in space.

    When all the grid (destination) points have prompted for their respective source points, the **Image Warp** dialog will re-appear showing the entire list of point pairs in the Coordinate List Table.

    ### Save List

    The **Save List** button saves the list of point pairs in an ASCII text file. The default filename is *warp.txt*.

    ### Delete Point Pair

    Deletes the selected point pair in the Coordinate List Table.

    ### Modify Point Pair

    Re-enters the selected point pair in the Coordinate List Table.

    ### Clear All Points

    Erases ALL point pairs in the Coordinate List Table.

    ### Prompt Instructions

    When CMDDIA = 0, the following prompts are issued instead of command dialogs.
    ```
    Enter number of warp points:
    ```

Enter the number of warp points for **gWARP** to create.

```
Source point on the drawing (1):
```

Select or enter the coordinates of the first source point. This will be the location in the raster image that will be stretched to a new (destination) position.

```
Warp Point (#.###,#.###):
```

A rubber-banding line will stretch from the source point to the cursor. Select or enter the coordinates of the destination point. This will be the location to which the source point is to be stretched.

```
Source point on the drawing (2):
```

Continue entering source and destination point pairs until all the warp points are done.

```
Now warping image
```

Appendix 1: Glossary of Terms

Term	Definition
Active Image	The image currently prepared by the GTXImage CAD series for raster editing or conversion.
ASCII	American Standard Code for Information Exchange - Pronounced "Ask-ee". The code used by most computers to represent text.
Attach	Lets you "attach" an image within a drawing. The raster data is not stored in the image - instead, an image entity "points" to the raster file on disk.
Auto-Convert Threshold	The confidence level above which gTCONVRT will automatically convert a raster character into ACAD text without using the Character Verification Dialog.
AutoSnap™	AutoCAD's method of dynamically snapping the crosshair to vector objects
Character Verification	The QA process that lets the user verify that ICR correctly creates ACAD text entities, even if it guesses the character wrong.
Confidence Level	A number between 1 and 100 representing the degree to which a character matches those it has already seen.
Contours	Continuous curves that can be represented by the polyline (or pline) vector entities. Typically found on topographical maps.
Deskew	To rotate a raster image so that it is no longer skewed.
Despeckle	To remove speckles or small marks that contain no meaningful information from a drawing.
Display Color	The color in which unselected raster is displayed.
DPI	Dots Per Inch -The number of dots per linear inch, calibrating a raster image to real space.
Highlight Color	The color in which selected raster is displayed.
ICR	Intelligent Character Recognition - Conversion of raster text into ACAD text entities.
Integer	A number evenly divisible by one (1,2,3,4,5 are integers. 1.25, 3/4, and .05 are not integers).

Term	Definition
IOP	Intelligent Object Picking™ - The proprietary method by which GTXImage CAD series selects raster shapes from a scanned drawing as if they were distinct vector entities.
Object Snapping	ACAD mode of snapping the cursor to a point relative to vector geometry.
Pixel	Pixels, or "PIcture ELements", are the tiny dots with which a scanner portrays a drawing.
Polyline Smooth Index	A number representing the smoothness of polylines generated when vectorizing raster contours.
Raster File	Pixel data from a scanned drawing stored as a file. There is a wide range of standards and formats for raster files.
Raster Limits	The area in a raster image in which features can exist; the extents of a raster database.
Raster Snapping	The ability to pick a point relative to raster geometry, such as snapping to the end of a raster "line".
Rasterize	To convert a CAD vector object into raster format.
REF file	Accompanies a raster file and has the same filename. Stores information about its scale, orientation (loadrotate setting or results of invoking gTURN), units (inch/mm).
Scanner	A device which converts a drawing into raster data by measuring and recording the intensity of light that is reflected from it.
Skewed	Slightly rotated. When a drawing is fed slightly crooked into a scanner, the resulting raster image is skewed.
Transparent Zoom	The ability to change your view of a drawing while in the middle of a command. Type: 'Z, [RETURN]. ACAD starts Zoom command. Old command resumes upon completion.
Vectorize	To convert raster data into vector format, as used in a CAD environment.

Appendix 2: Configuration

The GTXImage CAD Series receives many of its default values from a configuration file, *GTXRAST.CFG*. This ASCII file can be modified with a text editor such as Notepad. The list below shows the default contents of *GTXRAST.CFG* if you had installed it for using inches. The entries can be listed in any order.

Here is a standard configuration file:

```
REM * GTX Configuration *REM ** Use "REM" or "rem" for comment lines, blank lines
are ok. **
REM **Defines inch or millimeter measurement standard**
DrawingUnits 1Inch=1

REM **Defines the default path for saving raster files.**
RasterPath =CURRENT DIRECTORY

REM **Used with gCreate and color images when gActivated.**
RasterFormat =TIF

REM **Defines the available raster file formats that will be loaded with an AutoCAD
REM **open when a raster file with the same name is in the same directory.
RasterLoadExt =TIF CG4 RLC BMP PCX PNG *

REM **The maximum size an image can be created or actived.
REM **Number of pixels = DPI * DPI * ImageHeight * ImageWidth.
REM **This limit (973440000) could create a  400DPI 78" X 78" image.
REM **If set to 0 it's unlimited - warning could exceed memory capacity.
MaxNumPixels=973440000

REM **Defines the marking color for selected raster in editing.  Can use any color
from 0-255.**
HighlightColor =GREEN

REM **Defines the width of rasterized vector objects.**
RasterWidth =0.01

REM **Yes option creates a backup of the original raster file.**
BackupRas =NO

REM **Yes option saves the vector entity when doing Gburn or Graster.**
SaveVector=NO

REM **Defines the Backup Raster Extension for the BackupRas variable. You must put a
'.' before the extension.**
BackupRasExt=.GTX

REM **Defines the Dot Per Inch resolution within the raster limits.**
REM **Used for gcreate.**
Resolution =200

REM **Set the number of undos that are stored. Not available at this time**
REM MAXUndo=3

REM **Sets the subformat to be used when saving TIFF files.**
REM Subformat          Value
REM TIFF G3                 1
REM TIFF G4                 2
REM TIFF Uncompressed       3
TiffFormat = 2
```

```
REM **Disables a specific command.  Multiple variables can be set but must be on
separate lines.**
REM disable=gattach
REM disable=gload

REM * Text Configuration *
REM **Sets a recognition file for GTXImage CAD PLUS to automatically load when it
runs.**
REM CharFile =c:\program files\gtximage cad series V21\g_u_0.icr

REM **Sets a character to be listed in the ambiguous characters list box in the
gTConfig dialog.**
REM Ambigchar =I

REM **Sets the initial value for the Orientation list in the gTConfig dialog.**
CharOrient =0

REM **Sets the initial value of the AutoConvert Threshold setting in the gTConfig
dialog.**
CharThreshold =100

REM **Sets the initial value for the Min Char Size in the gTConfig dialog.**
CharMinSize =0.04

REM **Sets the initial value for the Max Char Size in the gTConfig dialog.**
CharMaxSize =0.4

REM **Sets the initial value for the Space Size setting in the gTConfig dialog.**
CharSpaceSize =1.0

REM **Sets the initial value for Alignment Offset setting in the gTConfig dialog.**
CharAlign =0

REM **Sets the initial value for Text Height setting in the gTConfig dialog.**
TextHeight =0.2

REM **Sets the initial value for Max # Spaces setting in the gTConfig dialog.**
MaxSpaceSize =1

REM **Sets the initial value for the word-mode verification level.
WordVerifyLevel =100

REM **Sets the initial value for the word-mode force-fit level.
WordForceLevel =0

REM ** 0 means touching character recognition is OFF, 1 means ON
TouchingCR =0

REM ** 0 means word mode CR, 1 means character mode CR
CRMode =1

REM ** Character similarity file for word recognition
REM SimFile = c:\program files\gtximage cad series V21\gtxtext.sim

REM ** dictionary file for word recognition
REM WordFile =c:\program files\gtximage cad series V21\base.dct
REM WordFile =c:\program files\gtximage cad series V21\User.dct

REM ** sets the default GAP JUMP size
REM GAP_SIZE_TO_JUMP = 50

REM ** sets the layer name to be created during hatch recognition
HATCH_LAYER_NAME = HatchLayer

REM ** sets the layer name to be created during linestyle recognition
LINESTYLE_LAYER_NAME = LinestyleLayer
```

```
REM * Line Width Configuration *
REM **Defines the rasterized vector width for particular colors.  Can use any color
from 0-255.**
Pcolor 1=0.010
Pcolor 2=0.020
Pcolor 3=0.030
Pcolor 4=0.040
Pcolor 5=0.050
Pcolor 6=0.060
Pcolor 7=0.070
Pcolor 8=0.080
Pcolor 21=0.090
Pcolor 23=0.100
Pcolor 30=0.110
Pcolor 41=0.120
Pcolor 63=0.130
Pcolor 153=0.140
Pcolor 191=0.150
Pcolor 161=0.160


REM **Set PclrUnit=0 if you want to multiply your pcolor setting by the CAD scale of
the Image.
REM **Set PclrUnit=1 if you do NOT want to multiply your pcolor setting by the CAD
scale of the Image.
PclrUnit=1

REM **Product level. Settings include ICAD, ICADPLUS.**
ProductLevel=ICADPLUS
Demomode=NO

REM **If equal to 0, it sets zero-width polylines to be rasterized just as all other
entities.
REM **If set to 1 all plines will be rasterized as 1 pixel wide.**
Zero_Pline=0

REM **Determines the number of minutes of idle time the GTXImage CAD series license
server
REM **will allow before de-activating an inactive license and making it available
again.**

REM **Set RefDelLoad=1 if ref file needs to be deleted on doing a gATTACH, set to 0
otherwise**
RefDelLoad = 0

REM **Images are placed on this layer when attaching.
REM **The current layer is used if no layer name is listed.
DefaultRasterLayer=Raster

REM ** Defines the layer color for the layer named in the
REM ** DefaultRasterLayer config parameter.
REM ** Can use any color from 0-255.
RasterColor = RED

REM ** Set to 'YES' when another 3rd party application will also be    *
REM ** editing raster images.  This will cause more overhead to keep   *
REM ** keep ICAD in sync with the 3rd party application raster edits. *
REM ** Default is NO.                                                  *
3P_APP_EDIT=NO

REM ** When set to 'YES', ICAD will not perform transparent saving ofREM ** edited
raster images.
REM ** Useful when another 3rd party application will also be performingREM **
transparent saving.
REM ** Default is NO.
3P_APP_SAVING=NO


REM **
```

```
REM ** A raster image attached with the 'gattach' command willREM ** automatically
be the active image (gactive) when the parameter
REM ** is set to YES.
REM ** Default is YES.
GACTIVE_ON_GATTACH=YES

REM ** Keep the following setting at 'YES' to prevent
REM ** holes from forming in the warped image
WARP_CORNER_PTS=YES
```

Configuration Entry Descriptions

The configuration file entries are explained below and valid entries are given. Any items not included in the file or not given a value will use the default values.

Basic Configuration Parameters

DrawingUnits   This entry has two functions. It defines whether configuration values should be inches or millimeters and also how the World Coordinate Units should be applied to the raster image. The raster image can be either "1inch" or "1mm"; the other linear measurements in the configuration file will be in the same unit (i.e., if you specify "mm" the RasterWidth parameter will express millimeters and the value should be adjusted accordingly). The DrawingUnits value defines what inch or 1 mm on the raster image maps to in CAD units. The value entered must be a real number.

RasterPath   Defines the default path for saving raster files. You can override the directory entered here by entering the full path when prompted for a file name. The default is to save the file in the current directory.

RasterFormat   Defines the default storage format of new raster files. This is used by **gCREATE** and when using **gACTIVE** on color images.

RasterLoadExt   Defines the available raster formats that will be loaded with GTXImage CAD open when a raster file with the same name is in the same directory.

MaxNumPixels   Sets the maximum number of pixels allowed in a new raster image. Default setting is 973,440,000, which at 400 dpi is equivalent to 78 x 78 inches or 1990 x 1990 millimeters.

HighlightColor   Defines the default color used to highlight selected raster data. Choose a color that can be seen against the raster image and against the background color. Valid entries are the same for RasterColor. The color can be changed with the **gCOLOR** or **gLAYER** commands. The default is **GREEN**.

RasterWidth   Sets the default raster line width to be used when vector entities are converted to raster. The entry should be an appropriate real number, or **T** if you want to specify the Line Width Table values. The default is **.01**. This value will be in inches or mm according to the **DrawingUnits** entry. All the values can be reset when you use the **gRASTER** command.

BackupRas   "Yes" causes raster save to create a backup raster file. "No" deactivates the backup option.

BackupRasExt   Sets the file extension for backup raster files.

Resolution   Defines the default **resolution** for new raster images.

MaxUndo	Sets the number of raster undos that are stored.
Tiffformat	Sets the subformat to be used when saving TIFF files:
TIFF G3	1
GIFF G4	2
TIFF UNCOMPRESSED	3

Disable Disables a specified command. Enter a single command name (i.e., **gMOVE**) after the word "Disable". The software will not allow the user to invoke the specified command. To disable multiple commands, add more *Disable [Command Name]* lines to the configuration file.

Text Configuration Parameters

CharFile sets a recognition file for **GTXImage CAD** *PLUS* to automatically load when it is run. If you want the program to load multiple recognition files, add more *CharFile [Recognition File]* lines to the configuration file.

AmbigChar sets a character to be listed in the ambiguous characters list box in the gTCONFIG dialog upon starting **GTXImage CAD** *PLUS.*

CharOrient sets the initial values for the **Orientation** list in the **gTCONFIG** dialog, determining the allowed orientations for building CAD text entities. The default setting is **0**. Valid entries are any angle between -360 and 360 degrees.

CharThreshold sets the initial value of the **AutoConvert Threshold** setting in the **gTCONFIG** dialog. This is the character **confidence level** above which the character verification dialog appears. The default value is **100**, which makes the character verification dialog appear every time. Valid entries are integers from 0 to 100.

CharMinSize sets the initial value for the **Min Char Size** setting in the **gTCONFIG** dialog, determining the minimum value for raster text characters when using Text IOP or **gTCONVRT**. The default value is **.04**.

CharMaxSize sets the initial value for the **Max Char Size** setting in the **gTCONFIG** dialog, determining the maximum value for the raster characters when using Text IOP or **gTCONVRT**. The default value is **.4**.

CharSpaceSize sets the initial value for the **Space Size** setting in the **gTCONFIG** dialog, defining the distance between raster characters in terms of a typical character's width necessary for **gTCONVRT** to insert a space character. The default value is **1.0**. Valid entries are any positive real number.

CharAlign sets the initial value for the **Alignment Offset** setting in the **gTCONFIG** dialog, determining the minimum vertical distance, relative to the text strings' orientations, between separate lines of text in terms of a percentage of a typical character's height. The default setting is **0**. Valid entries are any real number.

TextHeight sets the initial value for the TextHeight value in the text configuration dialog.

MaxSpaceSize sets the initial value for the **Max # Spaces** setting in the **gTCONFIG** dialog, determining the maximum number of spaces that can be inserted into a text string by **gTCONVRT** before it breaks the text string into two separate CAD text entities. The default setting is **1**. Valid entries are integers 1 through 100.

MaxSpaceSize sets the initial value for the **Max Space Size** dialog.

WordVerifyLevel Sets the initial value for the word-mode verification level.

WordForceLevel sets the initial value for the word-mode force-fit level.

TouchingCR controls whether ICR splits touching characters. 0 means OFF, 1 means ON.

CRMode controls whether ICR is performing *Character* recognition or *Word* recognition. 0 means word mode CR, 1 means character mode CR.

SimFile establishes the character similarity file to use for word recognition. *Gtxtext.sim* is hardcoded into GTX, but if another file is loaded it will override the default similarity settings.

WordFile establishes the dictionary file(s) to load on startup.

Raster-to-Vector Conversion Parameters

GAP_SIZE_TO_JUMP sets the default GAP JUMP size.
HATCH_LAYER_NAME sets the layer name to be created during hatch recognition.
LINESTYLE_LAYER_NAME sets the layer name to be created during linestyle recognition

PColor   Establishes a value in the line width configuration table.

Pcolor [Color Value]=0.050 [Line Width]

PClrUnit   Determines behavior of linewidth color table when rasterizing or vectorizing with a scaled image. Set PclrUnit=0 if you want to multiply your pcolor setting by the scale of the Image. Set PclrUnit=1 if you do NOT want to multiply your pcolor setting by the scale of the image.

Other Parameters

ProductLevel sets the product level for the GTXImage CAD series to run. Settings include ICAD, ICADPLUS,

Zero_Pline   Determines how polylines of zero width are rasterized.   0 causes zero-width plines to rasterized like other entities. 1 causes them to rasterize at 1 pixel wide.

RefDelLoad   Specifies the behavior of gATTACH as it loads a raster file with an associated .REF file. If set to 1, the .REF is deleted. If set to 0, the .REF file is kept.

DefaultRasterLayer   Establishes the name of a layer on which images are placed when attached using **gATTACH**.

RasterColor defines the layer color for the layer named in the DefaultRasterLayer config parameter. Any color from 0-255 can be used.

BkCompMode   When set to 1, this overrides GTXImage CAD's normal behavior to make it behave more like earlier versions of the GTXImage CAD Series. When a drawing containing attached images is opened, any image that does not have the same name as the drawing is detatched, deleting its image entities. Any raster file with the same name as the drawing will then be attached using any .REF files to register the raster in the vector drawing.

3P_APP_EDIT   This should be set to **Yes** when another 3rd party application will also be editing raster images. This will cause more overhead to keep RCAD in sync with the 3rd party application raster edits. When set to 'YES', RCAD will not perform transparent saving of

edited raster images.

3P_APP_SAVING　　　　　This is used when another 3rd party application will also be performing transparent saving.

GACTIVE_ON_GATTACH　　Controls whether a raster image attached with the **gATTACH** command should automatically become active (**gACTIVE**).

Line Width Configuration

The line width configuration table is important to several functions of the GTXImage CAD Series. Because CAD vector entities have no line width, the GTXImage CAD lets you assign raster line widths to CAD vector colors.

When converting raster to vector, **gCONVRT** can assign different colors to vectors resulting from different raster line widths. The **gRASTER** command converts vector data to raster. When rasterizing, CAD vector entities of different colors can be rasterized to different line widths. When smoothing raster the **gSMOOTH** command can snap raster data to the closest line width listed in this table.

The format for listing a color/linewidth listing is:

```
Pcolor Color# = LineWidth
```

In which **Color#** is the AutoCAD color number to be associated with the raster linewidth. **LineWidth** is the width of the raster line (this can be measured using the **Dist** command).

Appendix 3: Raster File Formats

The following table shows the read and write raster formats which GTXImage CAD Series can load and save.

Raster Format	Ext.
BMP Windows and OS/2 bitmap format	.bmp, .dib , .rle
CALS-I Mil-R-Raster	.gp4, .mil , .rst, cg4, .cal
GIF CompuServe Graphics Exchange Format	.gif
JFIF	.jpg
PCX	.pcx
PICT	.pct
PNG	.png

Raster Format	Ext.
TARGA	.tga
TIFF /LZW	.tif

Note: The PCX file writer provided by AutoCAD does not facilitate the saving of the resolution of the file. Files of this type are loaded without a resolution and the gINFO command cannot report a resolution. The GTXImage CAD Series software can save a PCX file into any of the other GTX supported file formats with a resolution of 200 dpi. When saved to another format, the file will always have a default resolution of 200 dpi.

Rastrans Raster Format Translator V5.0

The Raster Format Translator (*RASTRANS*) is a stand-alone utility included with the GTXImage CAD Series the same input/output formats as GTXImage CAD. Rastrans translates raster files from one raster format to another. It can scale translated files separately in the X and Y axes. GTXImage CAD series includes both Windows and DOS versions of GTX Rastrans. The DOS version lets you create DOS batch scripts.

Windows GUI Version

The Windows version of GTX Rastrans is the executable, *Rastrans.exe*, located in the GTX directory.

Rastrans main screen

Upon running Rastrans, the main screen will appear.

The left side of the Rastrans main screen contains the **Input Files List,** showing all the raster files to be translated.

Input Files List

This list box displays the files to be batch-translated to the selected output file format.

Add Files

Select to add raster files to the Input Files List.

Select any or all files to be translated into the Output File Format. Multiple files can be selected by holding **[Shift]** or **[Control]** while using the left mouse button to pick more than one file at a time.

Remove Files

Removes files from the Input Files List.

About RASTRANS

Displays copyright information.

Output File Directory

Specifies output directory for translated files. Selecting **Use Input File Directory** sends output to the same directory as the source files.

Output File Format

Displays the file format to which input files are to be translated. Selecting **Print to [Your Printer]** sends the output directly to the current Windows system printer. Scale may need to be adjusted to fit a drawing.

DPI

Use Original DPI sets resulting raster output at the same **DPI** as the input file. **New DPI** specifies an override dpi for output files.

Resolution

Use Original Resolution sets resulting raster output at the same **Resolution** as the input file. **New DPI** specifies an override resolution for output files.

Translate

Begins translating input files into files of the output file format in the specified output directory.

Quit

Exits Rastrans.

Restart

After translating, the **Translate** button is renamed **Restart**. Clears the Input Files List, allowing you to do another batch conversion if desired.

Rastrans.wri

For additional documentation, refer to *Rastrans.wri* located in the GTX directory.

SUPPORTED FORMATS

Raster Format	Ext.	Read	Write	Preserve Headers
Adobe PostScript	PS		X	
Boeing Bedmics	B4	X	X	X
GTX Uncompressed Runlength	RNL	X	X	
CalComp Raster Format	CRF		X	
CCITT G3 TIFF header	TIF	X	X	X
CCITT G4 TIFF header	TIF	X	X	X
CImage Corp	DSI	X		
JEDMICS Tiled G4	C4	X	X	X
FileNET G4 Tiled	IMG	X	X	
FileNET G3 Tiled	IMG	X		
FileNET G4 Banded	IMG	X		
FileNET G3 Banded	IMG	X		
GraphTEC RLC	RLC		X	

HP Printer Control Language	PCL		X	
HP Raster Transfer Language	RTL		X	
HP Graphics Language Std. 2	HPG		X	
PC Paint format	PCX	X	X	
Image Systems Runlength	RLC	X	X	
Image Systems Runlength 2	RLC	X		
Japan Digital Labs 4000E	JD4		X	
Japan Digital Labs	JDL		X	
CCITT G3-GTX	G3	X	X	
CCITT G4-GTX	G4	X	X	
Cals G4 - (MIL-STD-28002A)	CG4	X	X	
Oce Graphics/NKK (Japan)	OCE		X	
Optigraphics TIFF	OPT		X	
Raster Graphics	RLE		X	
TIFF uncompressed	TIF	X	X	X
TIFF Version 6 Tiled	TIF	X	X	X
TIFF Packbits	TIF	X		
Versatec Block Format	VBF		X	
Versatec VDS	VDS	X		

DOS Version of GTX Rastrans

GTXRastrans is also available as a DOS application.

Usage:

```
Rastrans infilename {outfilename|lpt1|lpt2|lpt3} [-rof
{calsg4|gtxg3|gtxg4|tiffg4|tiffg3|tiffu|tiled|rnl|rlc|jdl|crf|rtl|pcl|vb
f|ps|opt|pcx|rle|nkk|grphtec|hpg}] [-dpi input resolution]
```

Argument	Description
Infilename	raster input filename
	Input file types are autodetected independent of file extension (except b4, c4, rlc)
	Input file types recognized are gtxrnl, gtxg3, gtxg4, calsg4, tiffg3, tiffg4, tiff packbit, tiff uncompressed, tiled tiff, rlc, rlc2, dsig4, pcx, filenet img, b4, c4, cit
Outfilename	the raster output filename
-rof	optional – specifies the output file format type
-dpi	optional – input resolution (50 to 10000)
-xdpi	optional - output x resolution (50 to 10000)

Argument	Description
-ydpi	optional - output y resolution (50 to 10000)
-odpi	optional - output x and y resolution (50 to 10000)
-rot	optional - rotation of 90, 180, or 270

Raster File Formats

GTX OSR uses most of the same raster formats as GTXImage CAD. Formats exclusive to AutoCAD are not available.

Table 2: Raster File Formats

Raster Format	Ext.	Read	Write
Boeing Bedmics	B4	X	X
GTX Uncompressed Runlength	rnl	X	X
CCITT G3-GTX Header	g3	X	X
CCITT G4-GTX Header	g4	X	X
CImage Corp.	dsi	X	
JEDMICS Tiled G4	c4	X	X
FileNET G4 Tiled	img	X	X
FileNET G3 Tiled	img	X	
FileNET G4 Banded	img	X	
FileNET G3 Banded	img	X	
GraphTEC RLC	rlc		X
HP Deskscan	tif	X	X
PC Paint format	pcx	X	X
Image Systems Runlength	rlc	X	
Image Systems Runlength 2	rlc		X
CCITT G3-TIFF	tif	X	X
CCITT G4-TIFF	tif	X	X
Cals G4 - (MIL-STD-28002A) (28002 Type I supported. 28002 Type II not supported)	cg4	X	X
TIFF Uncompressed	tif	X	X
TIFF Version 6 Tiled	tif	X	X
TIFF Packbits	tif	X	
Fuji/Xerox MMR	mmr	X	X
Intergraph	cit	X	X

INDEX