AutoCAD® 2009
2D Training Manual

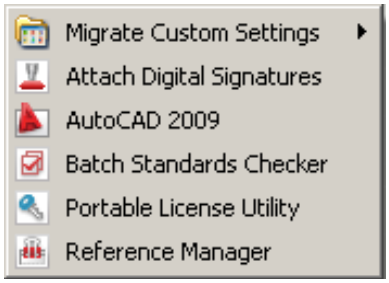
Written by Kristen S. Kurland

Chapter 1

Introduction

1.1 Launching AutoCAD

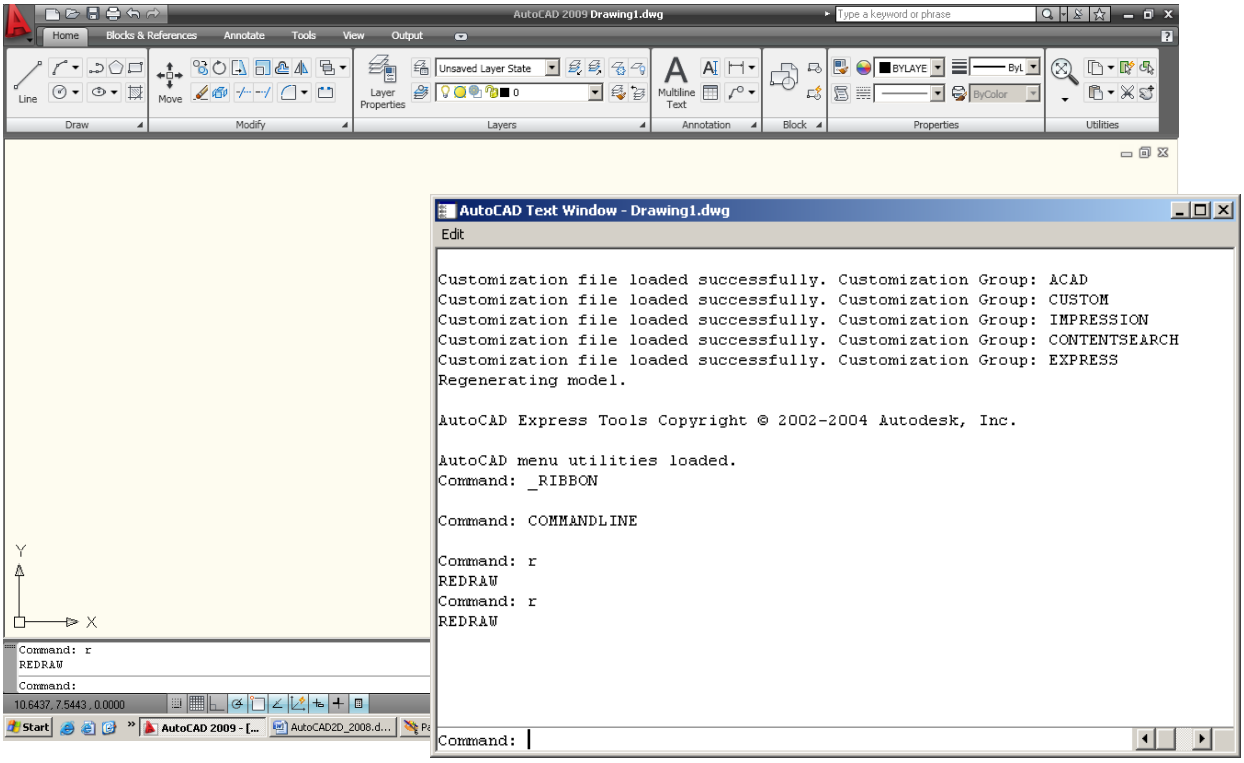
1. **Choose** Start from the Windows program manager.
2. **Choose** Programs, **Autodesk ,AutoCAD 2009**.
3. **Click** the AutoCAD 2009 for Windows icon.



1.2 Text and Graphics Screens

The graphics screen and the text screen are two different screens available in the drawing editor.

1. **Press** Function key **F2** on the keyboard.



TIPS:

The Cursor must be in the drawing window in order to select objects.

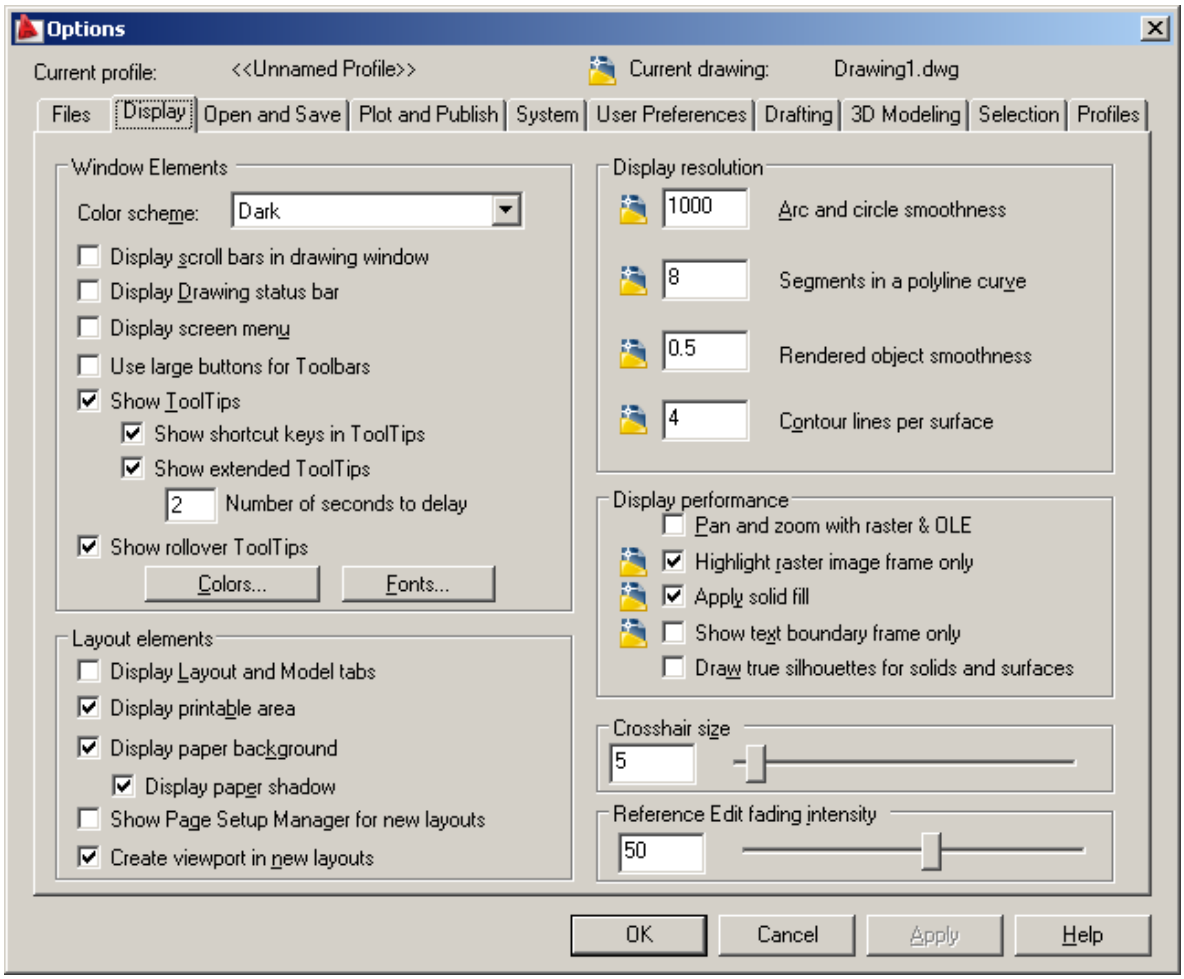
Maximize the AutoCAD windows to be full screen. This will make the drawings bigger and easier to read.

Use ALT + TAB to move between Windows applications.

1.3 Cursor

Controls the size of the crosshair. The allowable range is from 1 to 100 percent of the total screen. At 100% the ends of the crosshair are never visible. When the size is decreased to 99% or below, the crosshairs have a finite size, and the crosshairs' ends are visible when moved to the edge of the graphics area. The default size is 5%.

1. Choose **Tools, Options…**
2. Click the Display TAB.
3. Drag the slider bar in under crosshair size to set the cursor size.



1.4 Canceling a Command

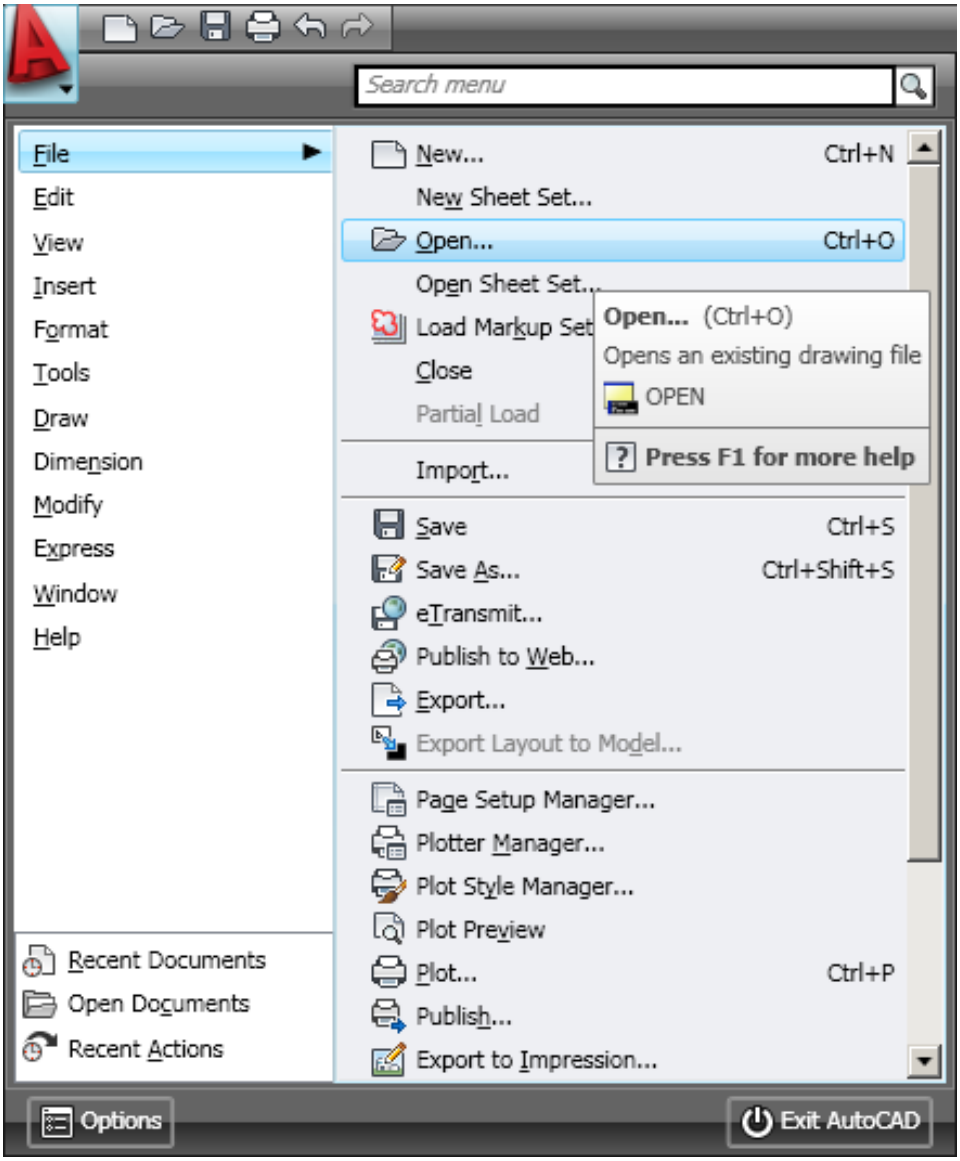
1. Press the **ESCAPE (ESC)** key on the keyboard.

TIP: Pressing ESC twice clears nested commands**.**

1.5 Menus and Colors

Menu Browser

1. Click on the A icon in the upper left corner of the drawing area.
2. Click the desired pulldown menu.
3. Click on the command to be executed from the pulldown.



Quick Access Toolbar

1. Click on one of the following icons for quick access to commands QNEW, OPEN, SAVE, PLOT, and UNDO/REDO.

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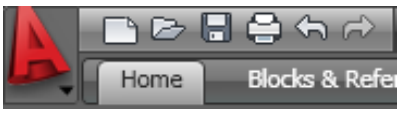


Right-click the Quick Access toolbar and click Customize Quick Access Toolbar. The Customize User Interface dialog opens and displays the list of commands available.

Drag commands you want to add from the command list pane in the Customize User Interface dialog box to the Quick Access toolbar.

Info Center

Quickly search for a variety of information sources, access product updates and announcements, and save topics with InfoCenter.

Ribbon

The ribbon provides a single, compact placement for operations that are relevant to the current workspace. It eliminates the need to display multiple toolbars, reducing clutter in the application window. The ribbon maximizes the area available for work using a single compact interface.

The ribbon can be displayed horizontally, vertically, or as a floating palette. The horizontal ribbon is displayed at the top of the drawing window by default when you create or open a drawing.

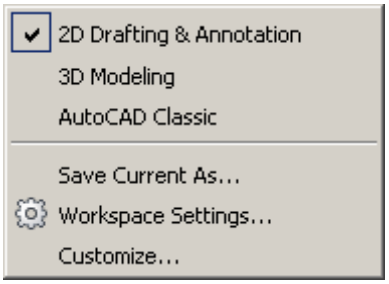
You can create your own panels to display on the ribbon; you can also modify the commands and controls on existing ribbon panels.

1.6 Workspaces

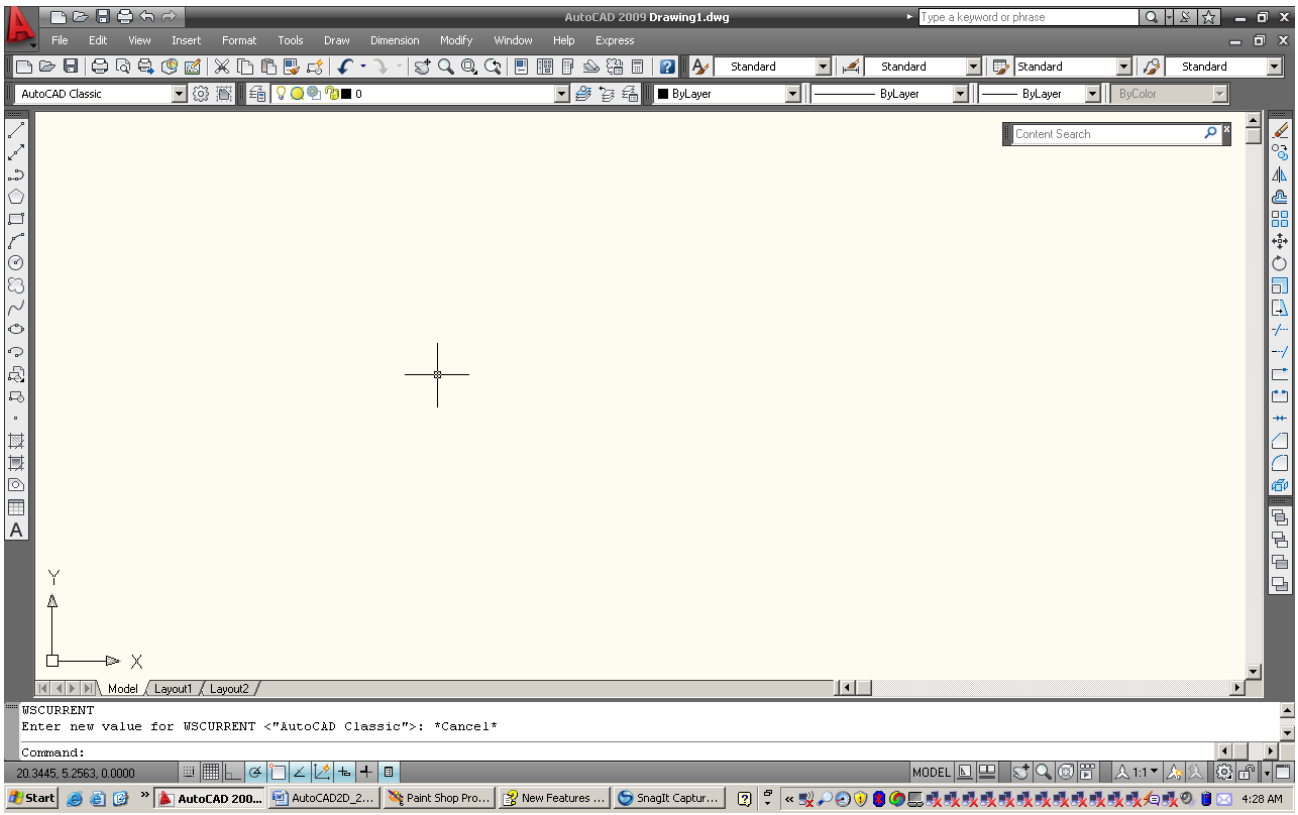
You can switch between the workspaces from the menu browser.

1. Click the Workspace switching icon in the lower left corner of the screen.

2. Click on one of the following workspace options



AutoCAD classic workspace



1.7 AutoCAD ClassicToolbars

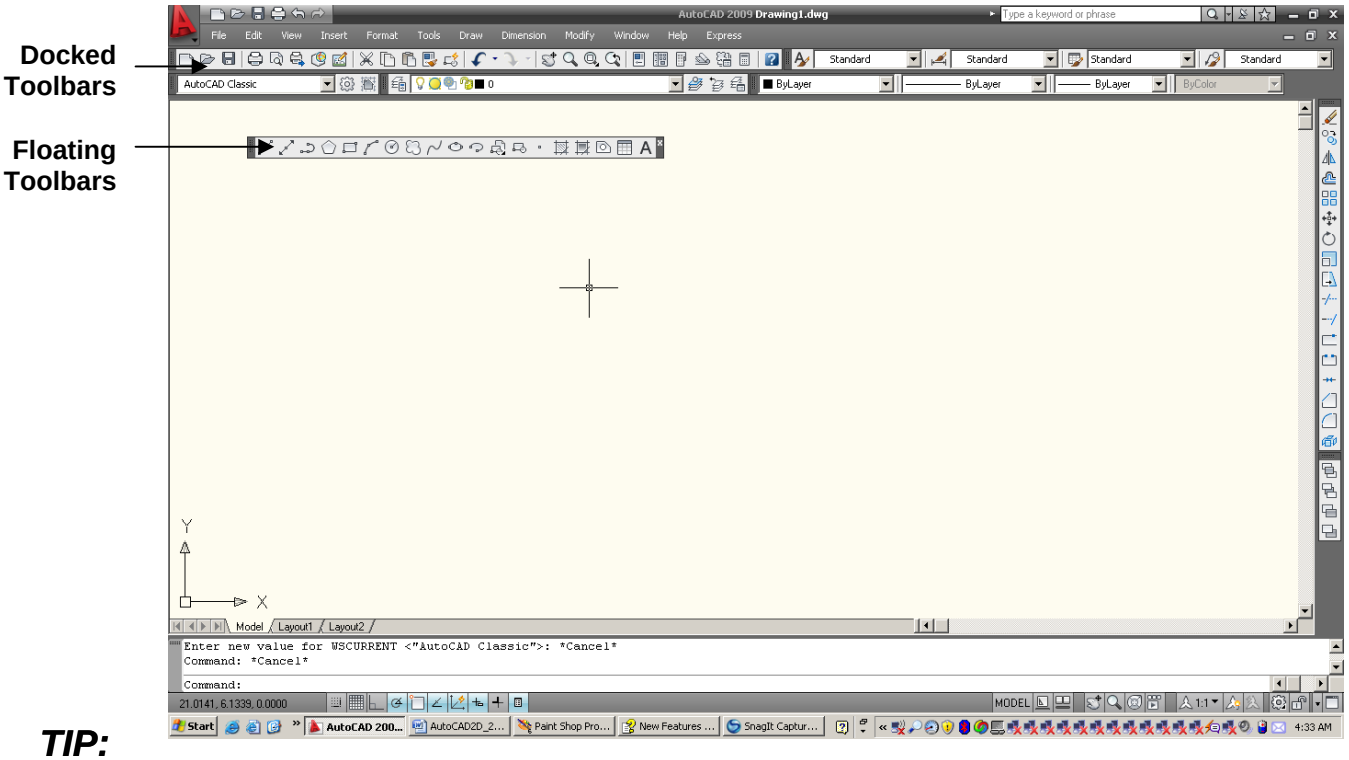
Toolbars can be docked on the screen or they can float about the screen.

To Float a Toolbar:

1. Choose the gray border surrounding each tool.
2. Drag the toolbar to any area on the screen.

To Dock a Toolbar:

1. Choose the title or gray border of the toolbar.
2. Drag the toolbar to the top, bottom, left, or right area of the graphics display.



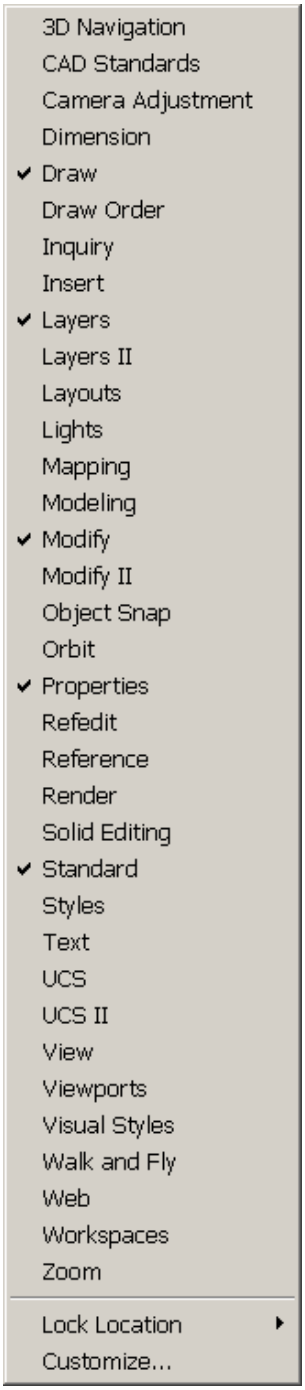
TIP:

-Holding the CTRL key while dragging will prevent docking.

Loading Toolbars

Right-clicking on an icon in any toolbar

This will show a list of all available toolbars.



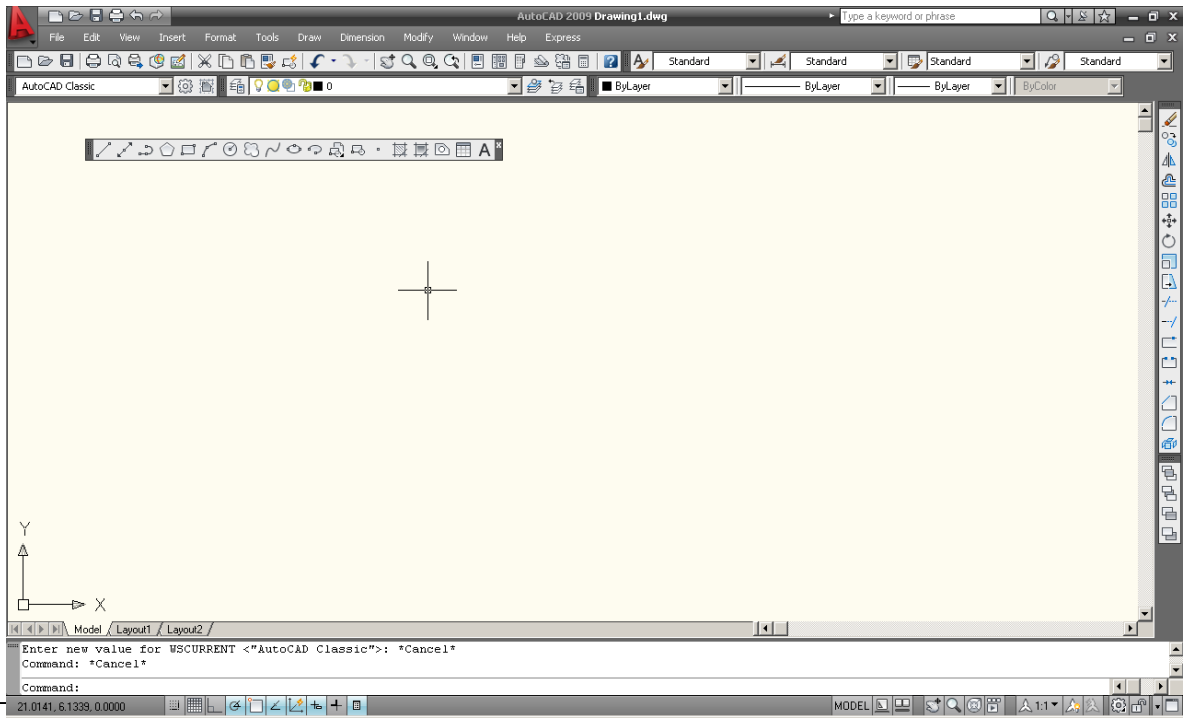
Help Tooltips

1. Move the mouse to the toolbar but do not pick the button.

1.8 Status Bar and Command Prompt

The Status Bar is the area below the command line that shows messages as well as coordinates, modes, and the current time.

To activate SNAP, GRID, ORTHO, OSNAP, MSPACE, PSPACE, and TILE, you must double-click on the mode to change.



Status Bar

TIP:

- Right click on the blank area of the status bar to see the tools to turn off/on.

1.9 Typing Commands

Typing a Command

All AutoCAD commands can be typed in at the command line. Many commands also have one or two letter aliases that can also be typed as shortcuts to the commands.

1. Type the desired command at the command prompt.

   Command : **LINE**

   or

2. Type the command's alias.
   Command: **L**

3. Press **ENTER**.

4. Type an option at the command prompt.

TIP:Many AutoCAD commands require you to press ENTER to complete the command. You know you are no longer in an AutoCAD command when you see a blank command line.

Reissuing the Last Command

The last used AutoCAD command can be re-entered by one of the following three methods of ENTER. The ENTER key on the keyboard will always act as ENTER, the SPACEBAR and RIGHT MOUSE will act as enter most of the time (exceptions include placing TEXT).

1. Press the **ENTER** key on the keyboard

   or

2. Press the **Space bar** on the keyboard.

   or

3. Click the right mouse button.

1.10 Pointing Device (Mouse)

AutoCAD uses either a mouse or digitizing tablet to select objects in a drawing.

Left Mouse Button

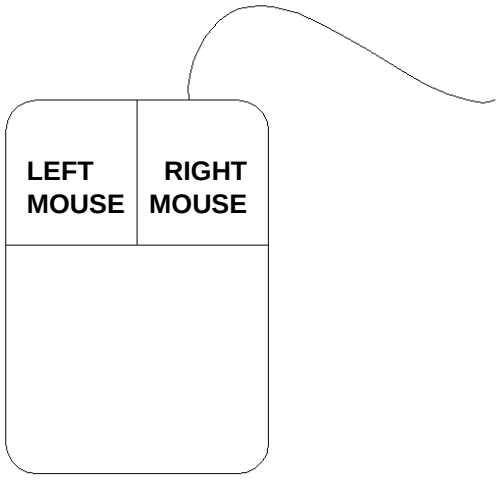
Used to pick or select objects

1. Click the left mouse button to select an object area in the drawing.
2. Press **ESC twice** to deselect an object (or to cancel a command).

Right Mouse Button

Used to enter a command, repeat last command, or access shortcut menus.

1. Click the right mouse button.



TIPS:

- SHIFT + the right mouse button brings up the object snap menus.
- Various screen locations for the mouse brings up different menus.

1.11 Undo and Redo

Reverses the last action.

1. Choose **Edit, Undo**.

   or

2. Click the Undo icon. 

   or

3. Press **CTRL + Z**.
4. Type U at the command prompt to undo the last command.

   Command: **U**

Redo

Reverses the effects of a single UNDO or U command.

1. Choose **Edit, Redo**.

   or

2. Click the Redo icon. 

   or

3. Type REDO at the command prompt to redo the last undo command.

   Command: **REDO**

TIPS:

-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.

-REDO must immediately follow the U or UNDO command.

1.12 Function Keys and Accelerator Keys

F1	Displays Help
F2	Toggles Text Window
F3	Toggles OSNAP
F4	Toggles TABMODE
F5	Toggles ISOPLANE
F6	Toggles UCSDETECT
F7	Toggles GRIDMODE
F8	Toggles ORTHOMODE
F9	Toggles SNAPMODE
F10	Toggles Polar Tracking
F11	Toggles Object Snap Tracking
F12	Toggles Dynamic Input

ALT+F11	Displays the Visual Basic Editor
ALT+F8	Displays the Macros dialog box
CTRL+0	Toggles Clean Screen
CTRL+1	Toggles Properties palette
CTRL+2	Toggles DesignCenter
CTRL+3	Toggles the Tool Palettes Window
CTRL+4	Toggles Sheet Set Manager
CTRL+5	Toggles Info Palette
CTRL+6	Toggles dbConnect Manager
CTRL+7	Toggles Markup Set Manager
CTRL+8	Toggles the QuickCalc calculator palette
CTRL+9	Toggles the command window
CTRL+A	Selects objects in drawing
CTRL+SHITF+A	Toggles Groups
CTRL+B	Toggles Snap
CTRL+C	Copies objects to Clipboard
CTRL+SHIFT+C	Copies objects to Clipboard with Base Point
CTRL+D	Toggles Dynamic UCS
CTRL+E	Cycles through isometric planes
CTRL+F	Toggles running object snaps
CTRL+G	Toggles Grid
CTRL+H	Toggles PICKSTYLE
CTRL+I	Toggles COORDS

CTRL+J	Repeats last command
CTRL+L	Toggles Ortho mode
CTRL+M	Repeats last command
CTRL+N	Creates a new drawing
CTRL+O	Opens existing drawing
CTRL+P	Prints current drawing
CTRL+R	Cycles layout viewports
CTRL+S	Saves current drawing
CTRL+SHIFT+S	Brings up the Save As dialog box
CTRL+T	Toggles Tablet mode
CTRL+V	Pastes data from Clipboard
CTRL+SHIFT+V	Pastes data from Clipboard as a Block
CTRL+X	Cuts objects to Clipboard
CTRL+Y	Cancels the preceding Undo action
CTRL+Z	Reverses last action
CTRL+[	Cancels current command
CTRL+\	Cancels current command
CTRL+PAGE UP	Moves to the next layout tab to the left of the current tab
CTRL+PAGE DOWN	Moves to the next layout tab to the right of the current tab

1.13 On-Line Help

1. Choose Help, AutoCAD Help.

   or

2. Click the Help icon. 

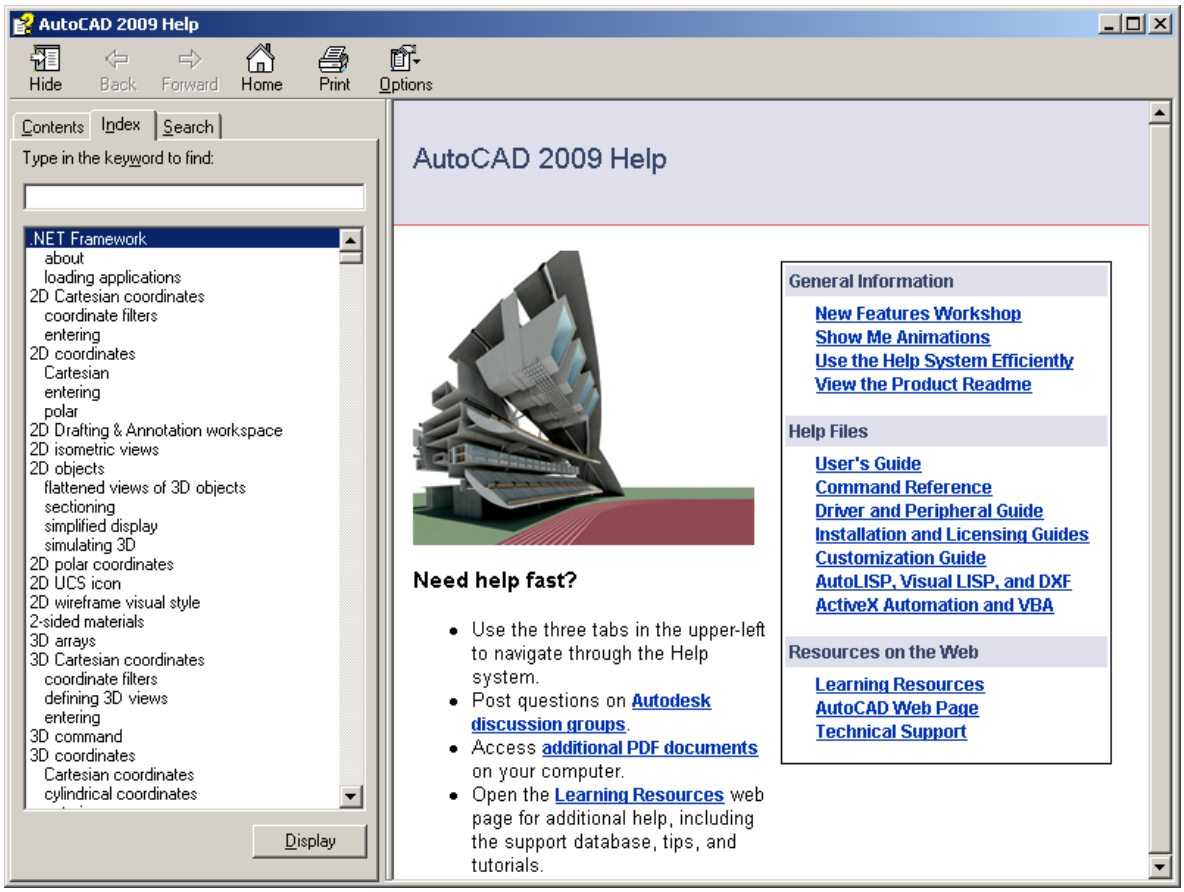
   or

3. Type HELP at the command prompt

   Command: **HELP**

   or

4. Press **Function Key F1**

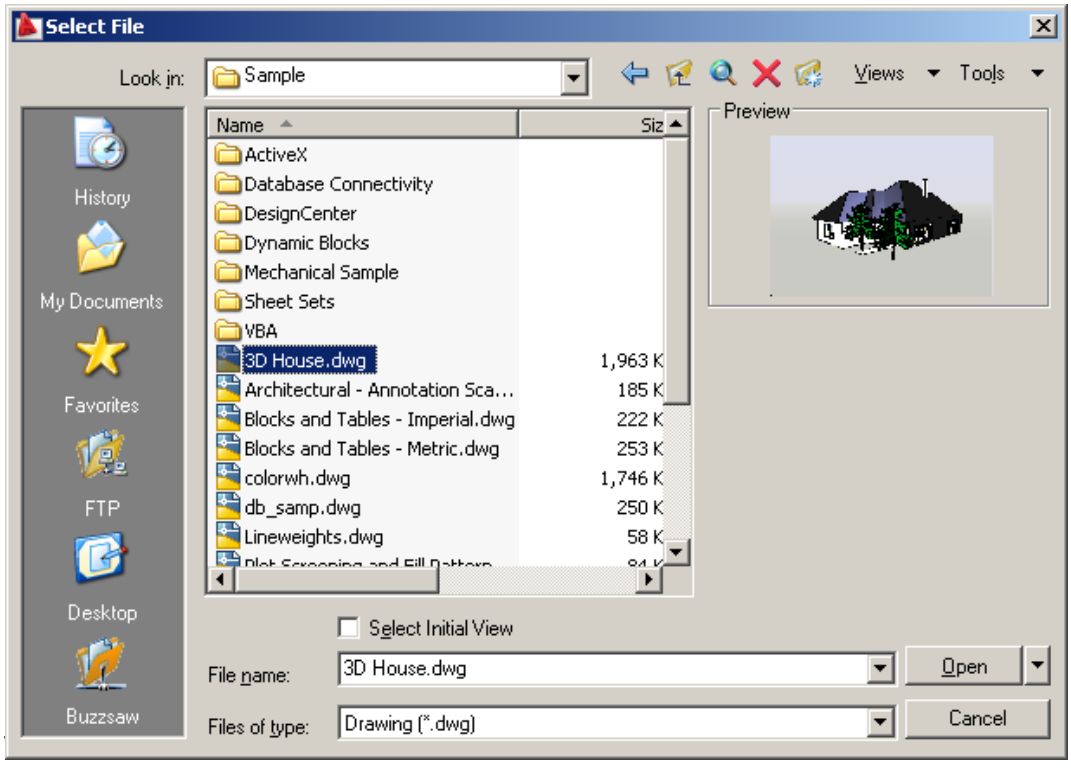


Chapter 2

Introduction to Commands

2.1 Open Existing Drawings

1.	**Choose**	File, OPEN.
		or
2.	**Press**	CTRL + O.
		or
3.	**Click**	the OPEN icon.
		or
4.	**Type**	OPEN at the command prompt. Command: **OPEN**
5.	**Press**	ENTER
6.	**Double Click**	the desired directory to find the drawing to open.
7.	**Click**	the drawing name to open.
8.	**Click**	The OK button.



-Preview shows a bitmap image of the drawing selected. This image is the view that was last saved in the drawing. It will not show a preview of drawings saved before R13 AutoCAD.

2.2 Creating a New Drawing

NEW Command

Creates a new drawing file.

1. **Choose** File, New.

or

2. **Press** CTRL + N

or

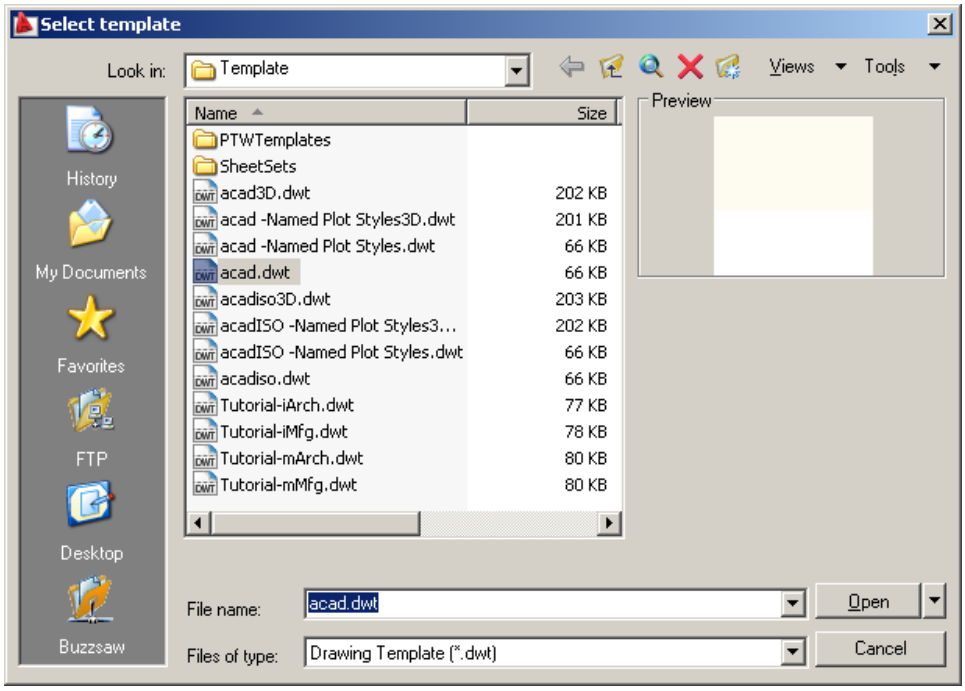
3. **Click** the New icon.

or

4. **Type** NEW at the Command prompt.

   Command: **NEW**

5. **Choose** One of the options for creating a new drawing.
6. **Click** The OK button.
7. **Save** the drawing as another name.



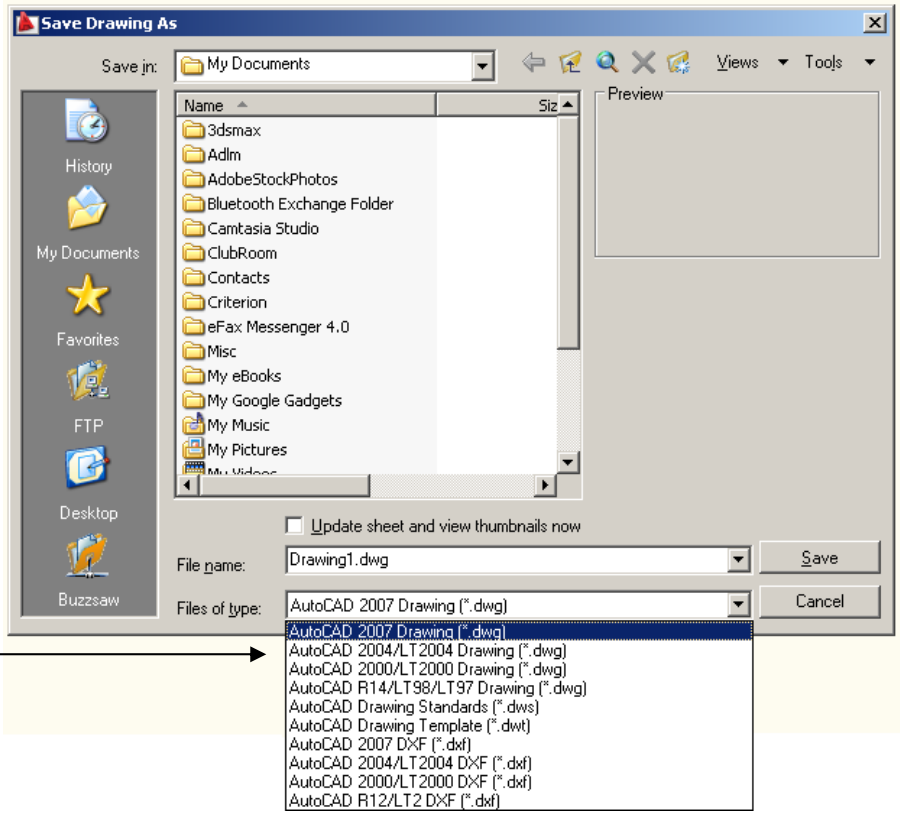
TIP:

New drawings can also be created from Template Files.

2.3 Saving Drawings

Saves the most recent changes to a drawing. The first time an unnamed drawing is saved the “Save As” dialog box appears. AutoCAD saves its drawings as files with extensions ending in .DWG.

1. **Choose** **File, Save or Saveas**.
   or
2. **Type** SAVE or SAVEAS at the command prompt.
   Command: **SAVE** or **SAVEAS**
3. **Press** **ENTER**
4. **Type** A new drawing name or keep the existing drawing name.
5. **Click** The OK button.



Various file types the drawings can be saved as

TIP:

Clicking the dropdown list for File type changes the format that the drawing can be saved in.

Quick Save

The QSAVE command is equivalent to clicking Save on the File menu.

If the drawing is named, AutoCAD 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, AutoCAD displays the Save Drawing As dialog box (see SAVEAS) and saves the drawing with the file name and format you specify.

1. **Press** CTRL + S.

or

2. **Click** the Save icon. 

or

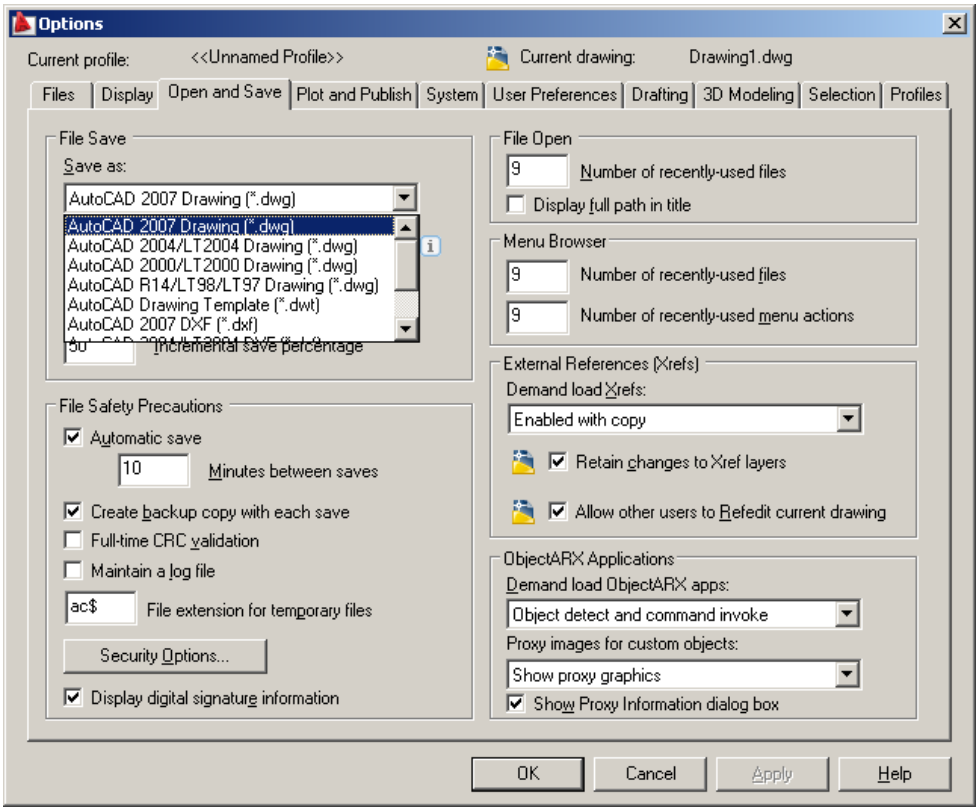
3. **Type** QSAVE at the command prompt,

   Command:**QSAVE**

TIPS:

Drawings can be saved as different versions of AutoCAD (e.g. R13, R14, R 2000, etc.)

AutoSave settings under Tools, Options…



2.4 File Safety Precautions

Autosave

AutoCAD automatically saves information in .SV$ files; however, users should save their drawings to .DWG files every 10 minutes. A value of zero (0) disables autosave.

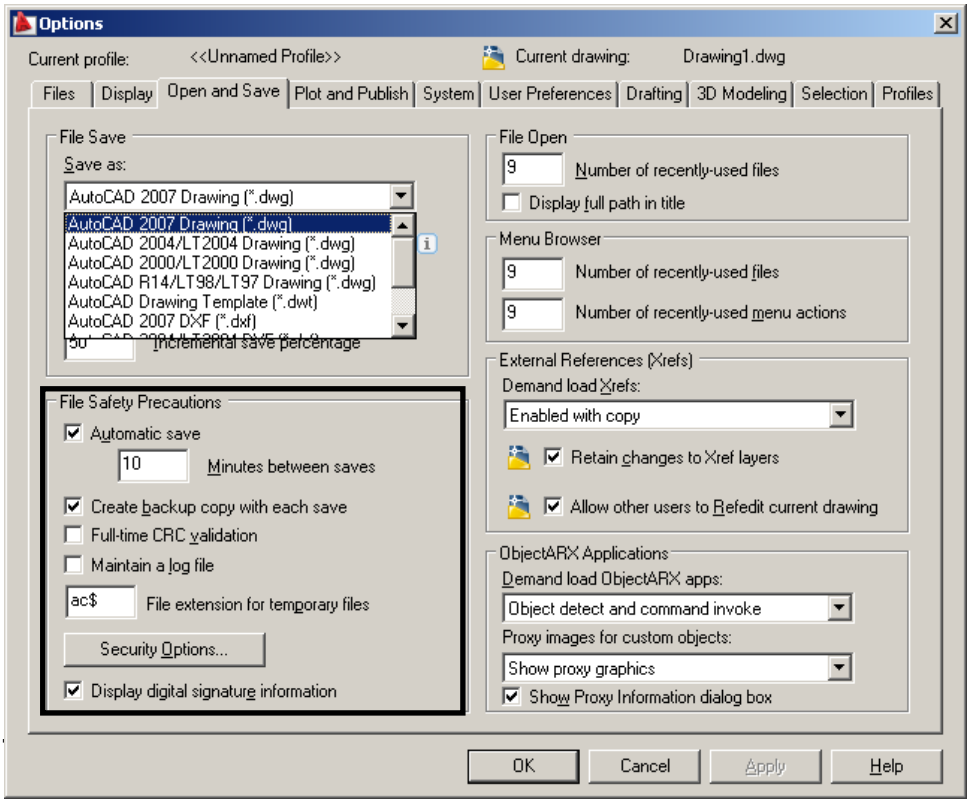
Temporary Files

These files have the extensions .ac$ (temporary drawing file).

After a system failure, if you are on a network, you should not delete temporary files until you have verified that they are not part of an active editing session.

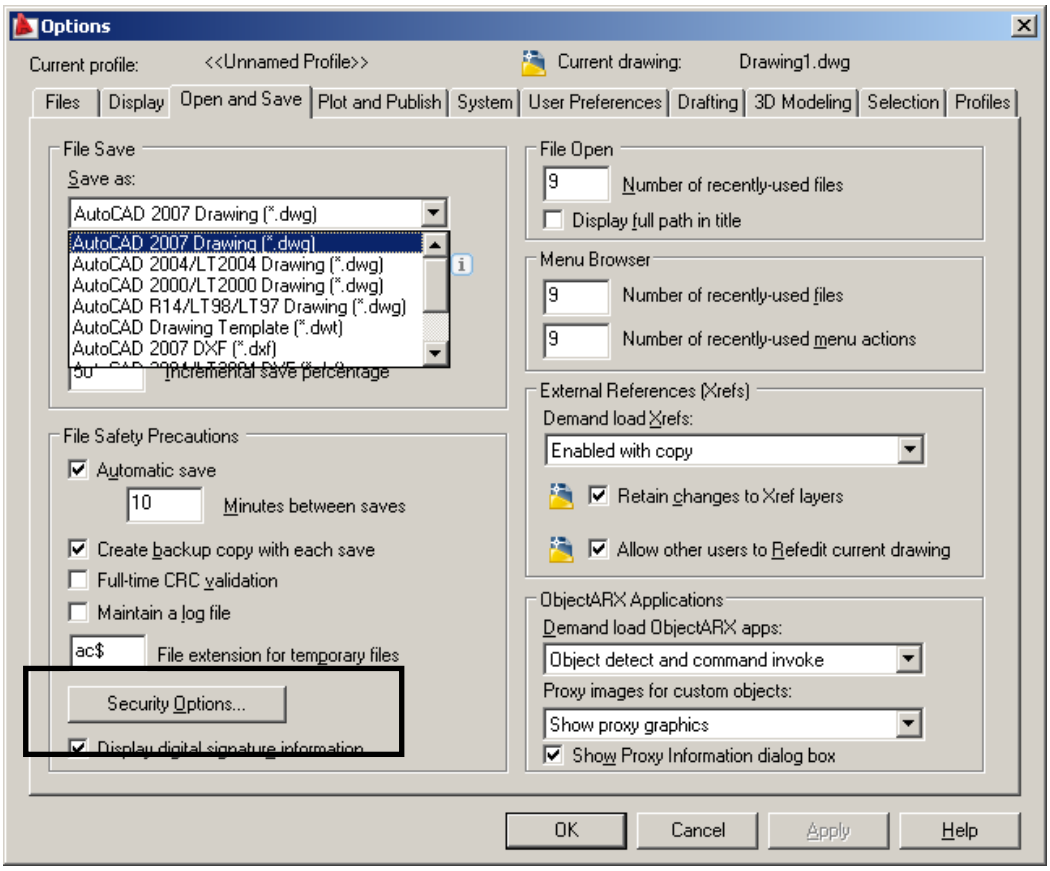
Other temporary files may be left in the drawing directory or the temporary file directory.

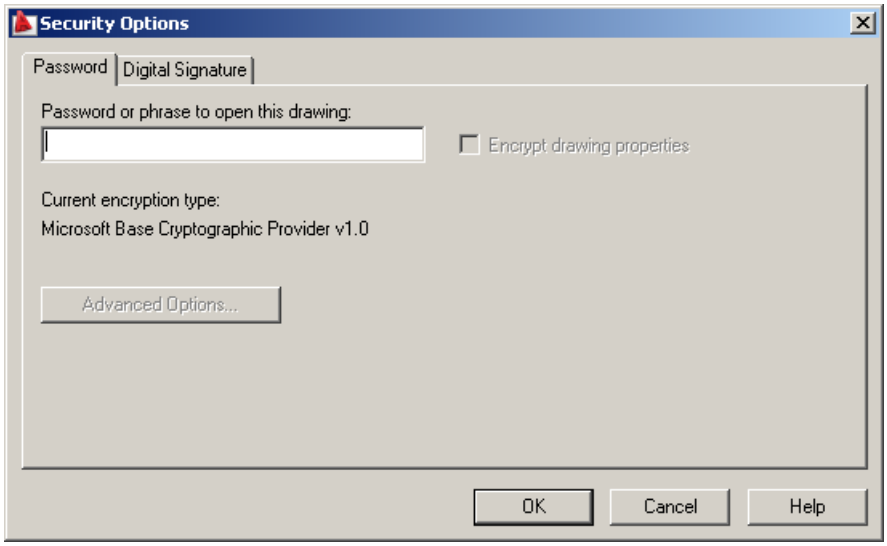
AutoSave and SV$ under Tools, Options...., Open and Save



Security Options

Specifies security settings to be used when your drawing is saved. The Password option adds a password to a drawing when it is saved.





2.5 Exiting AutoCAD

1. **Choose** File, Exit.

   or

2. **Type** QUIT at the command prompt.

   Command: **QUIT**

3. **Press** ENTER

4. **Click** Yes to save changes or No to discard changes.

Chapter 3

Draw Commands

3.1 Line Command

Creates single straight line segments

1. **Choose** Draw, Line.

or

2. **Click** the Line icon. 

or

3. **Type** LINE from the command prompt

   Command: **LINE** or **L**

4. **Press** ENTER
5. **Pick** From point: (**point**)
6. **Pick** Specify next point or [Close/Undo]:(**point**)
7. **Pick** Specify next point or [Close/Undo]:(**point**)
8. **Press** ENTER to end line sequence

or

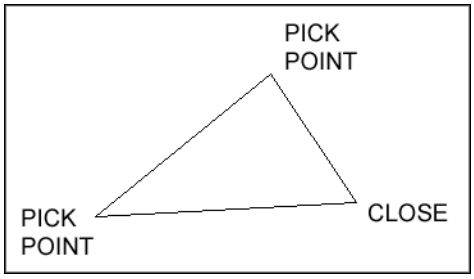
9. **Type** U to undo the last segment

   To point: **U** (undo)

or

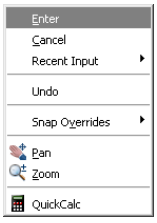
10. **Type** C to create a closed polygon

    To point : **C** (close)



TIPS:

- You can continue the previous line or arc by responding to the From point: prompt with a space or ENTER.
- Choose the right mouse button for the line pop-up menu to appear while in the line command

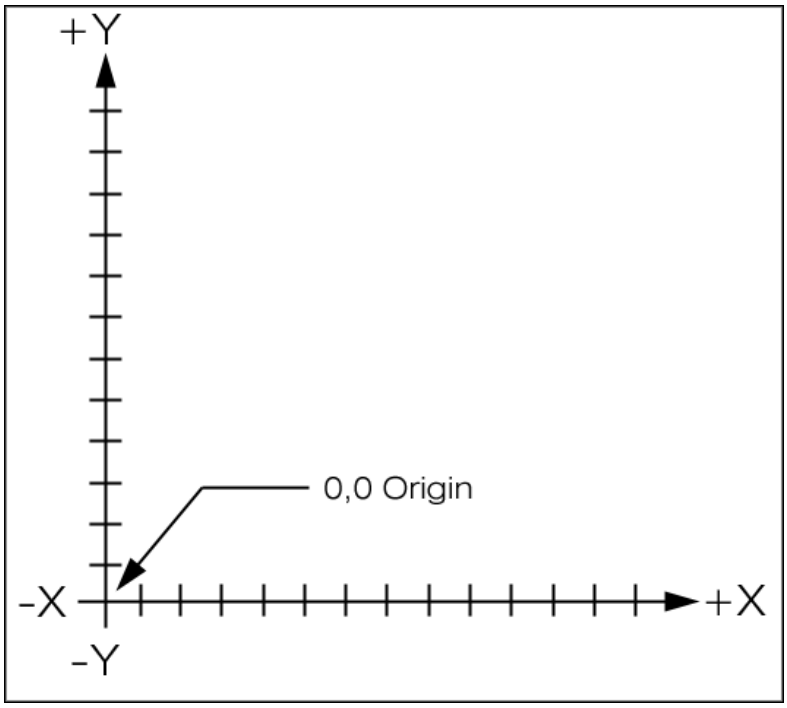


3.2 Cartesian Coordinate System

AutoCAD provides the user with an infinite two dimensional area to work with. Any entities place on the working two dimensional plane can be defined relative to the Cartesian coordinate system.

The Cartesian coordinate system divides a two dimensional plane with two perpendicular axis. The X axis runs horizontal across the bottom of the screen. The Y axis runs vertically along the left side of the screen. These two axis intersect at the bottom left corner of the screen.

Each of these axis is further divided into segments. Each segment is given a value. The X axis segments increase in value to the right. The positive X values are to the right of the intersection of the two axis. The negative X values are to the left. The positive Y values are above the intersection and increase up. The negative Y values are below.



Absolute Coordinates

1. **Type** x,y coordinate when AutoCAD asks for a point.

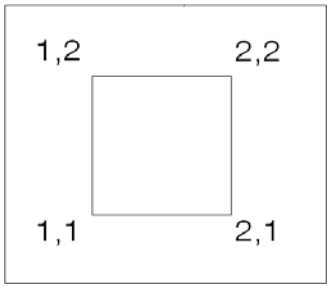
   From point: **1,1**

   To point: **2,1**

   To point: **2,2**

   To point: **1,2**

   To point: **1,1**



NOTE: If dynamic input (F12) is on, you must type the # sign before entering absolute coordinates (e.g.#1,1).

Relative Coordinates

1. **Type** @deltax,deltay when AutoCAD asks for a point.

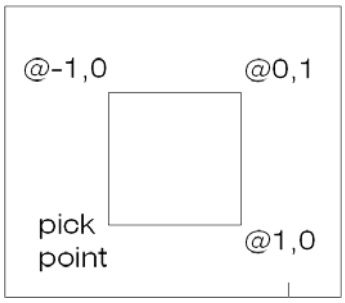
   From point pick point

   To point: **@1,0**

   To point: **@0,1**

   To point: **@-1,0**

   To point: **@0,-1**



Polar Coordinates

1. **Type** @distance<angle when AutoCAD asks for a point.

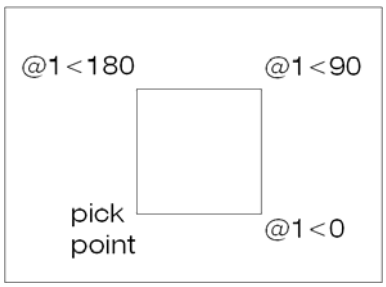
   From point: pick point

   To point:**@1<0**

   To point:**@1<90**

   To point:**@1<180**

   To point:**@1<270**



3.3 Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

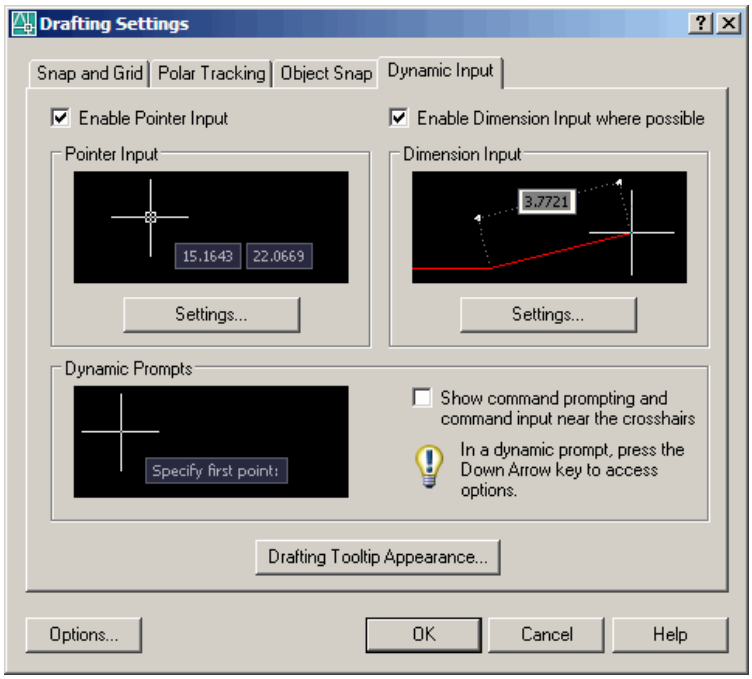
Turning Dynamic Input ON/OFF

1. Click **Dyn** on the status bar

   or

2. Press **F12**

Tip: Right-click Dyn and click Settings to control what is displayed by each component when Dynamic Input is on.



3.4 Orthogonal Lines

Controls lines from being drawn at various angles to straight lines. When the snap grid is rotated, ortho mode rotates accordingly.

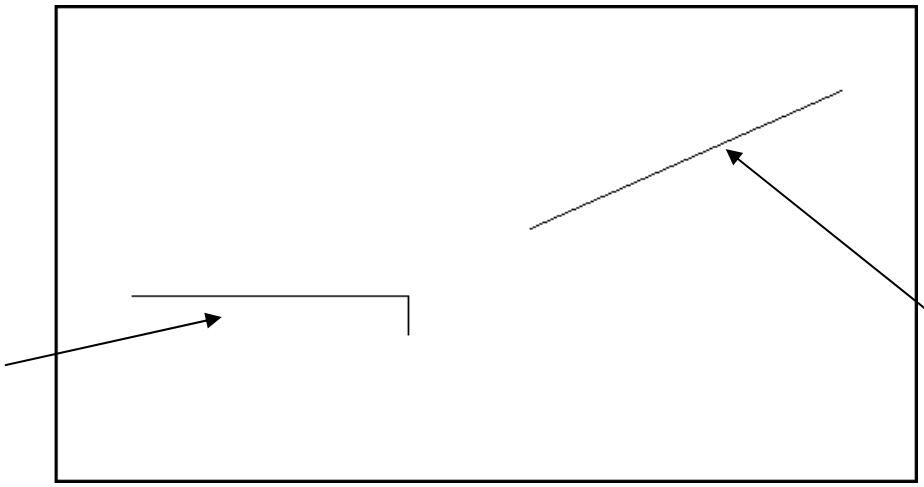
1. **Press** Function Key **F8**.

or

2. **Double Click** ORTHO from the Status Bar.

or

3. **Press** CTRL + L.



3.5 Polar Tracking

Polar Snaps work independently from snaps. With Polar Snaps on, AutoCAD shows the distances and angles being displayed as the cursor moves.

1. **Choose** Tools, Drafting Settings

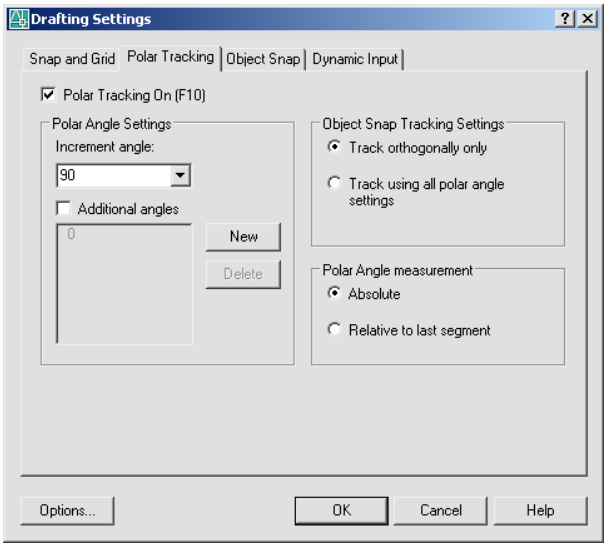
   or

2. **Type** DDSETTINGS at the command prompt.

   Command : **DDESTTINGS**

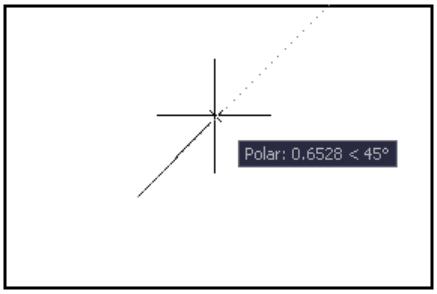
3. **Choose** the Polar trackingTAB from the dialog box.

4. **Select** the desired incremental angle from the dropdown list (or create a new angle).



5. **Pick** OK to exit the dialog box.

6. **Draw** a LINE using the Polar Snap references.



3.6 Circles

Circle Command

1. **Choose** Draw, Circle.

   or

2. **Click** the Circle icon. 

   or

3. **Type** CIRCLE at the command prompt.

   Command: **CIRCLE**

4. **Type** One of the following options:
   3P/2P/TTR/<<center point>>:

   or

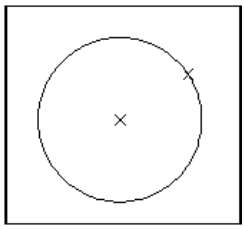
5. **Pick** A center point.
6. **Type** A radius or diameter.

   or

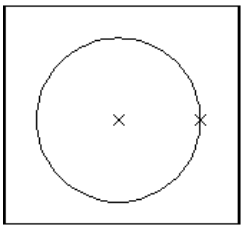
7. **Pick** A radius or diameter

   Diameter/<<radius>>:

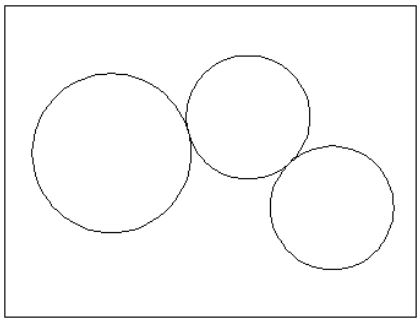
Circle, Center Radius



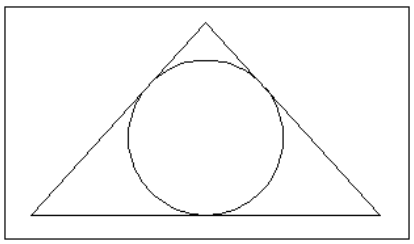
Circle, Center Diameter



Circle, Tangent, Tangent Radius

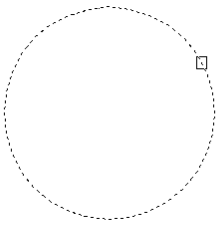


Circle, Tangent, Tangent, Tangent



TIPS:

- To create circles that are the same size, press ENTER when asked for the circle radius.
- When selecting a circle with a pickbox, be sure to select the circumference of the circle.



3.7 Arc Command

1. **Choose** Draw, Arc.

   or

2. **Click** the Arc icon. 

   or

3. **Type** ARC at the command prompt

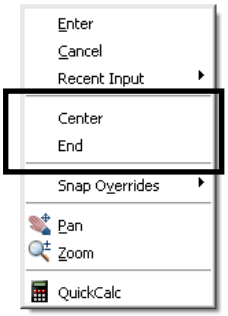
   Command: **ARC**

4. **Draw** One of the arcs.

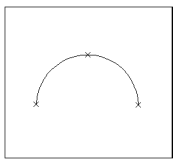
TIPS:

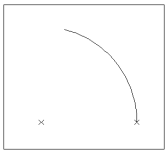
-Except for 3 point arcs, arcs are drawn in a COUNTERCLOCKWISE direction.

- While in the arc command, press the right mouse button to select the following options for arcs:

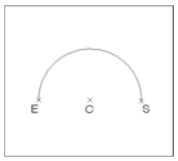


Arc Examples

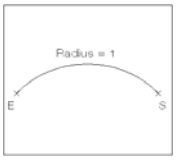
3 point arc 

Start ,center, chord length 

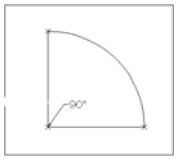
start, center, end



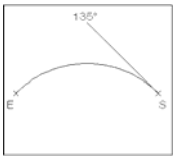
Start, end, radius



Start , center, included angle



Start, end, direction



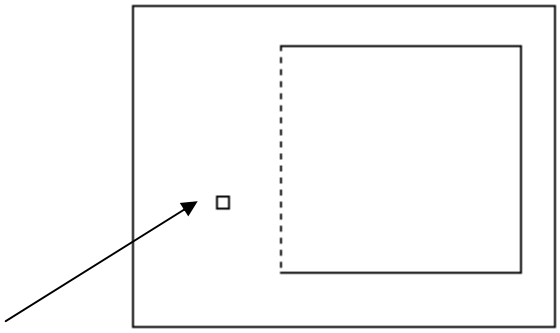
Chapter 4

Erase and Selection Sets

4.1 Erase and Selection Sets

Erasing Objects

1.	**Choose**	Modify, Erase.
		or
2.	**Click**	the Erase icon. 
		or
3.	**Type**	ERASE at the command prompt.
		Command : **ERASE** or **E**
4.	**Pick**	Object at the select object prompt.
		Select objects: (**pick object**)
5.	**Press**	ENTER when you are done choosing objects.
		Select objects: **ENTER**



Select objects with pickbox

TIP:

• If the cursor is not touching an object, AutoCAD will create a crossing or window selection as defined on the following pages.

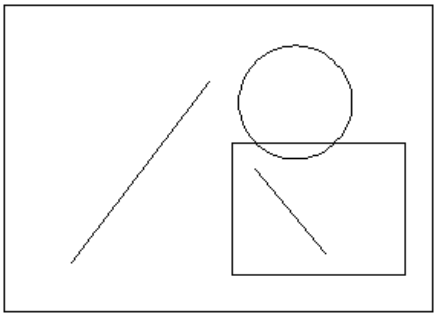
4.2 Selection Set Options

Type one of the following options at the Select objects: prompt: (point)One object.

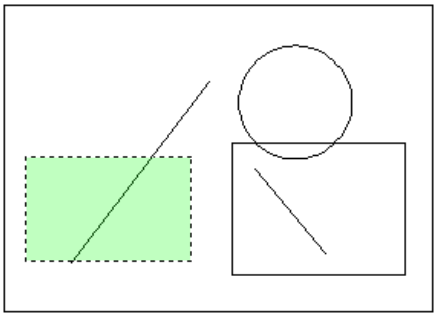
ALL	All objects within the drawing are selected unless they are on frozen or locked layers.
Multiple	Multiple objects selected without high lighting (faster edits).
Last	Last object.
Previous	All objects in the previous selection-set.
Group	Objects in a named group.
AUto	Automatic BOX (if pick in empty area).
SIngle	One selection (any type).
Add	Add mode: adds following objects to selection-set.
Remove	Remove mode: removes following objects from selection-set.

Window and Crossing

Window
Objects fully enclosed within Window.

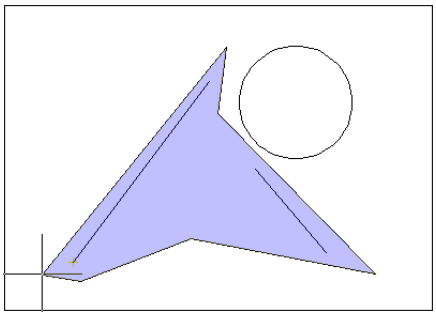


Crossing
Objects within or Crossing a window.



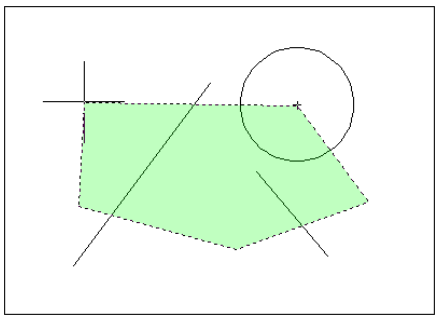
WPolygon

All entities within the boundaries of a polygon created by inputted points.



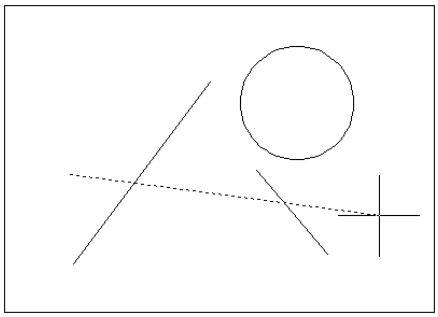
CPolygon

All entities within or touching the boundaries of a polygon created by input.



Fence

Objects that are crossed by a temporary line.



Remove from Selection Set

1. **Press** **SHIFT** and select entities to remove them from the selection set.

4.3 OOPS

Reinserts the last erased set of objects or block even if it was not the last command issued. Otherwise Oops acts like UNDO.

1. **Type** OOPS at the command prompt to reinsert erased objects

   Command: **OOPS**

4.4 Selection Preview

SELECTIONPREVIEW

Controls the display of selection previewing

Chapter 5

Basic Display Commands

5.1 ZOOM

Increases or decreases the apparent size of objects in the current viewport

1. **Choose** View, Zoom.

or

2. **Click** a Zoom icon.

or

3. **Type** ZOOM at the command prompt.

   Command: **Zoom** or **Z**

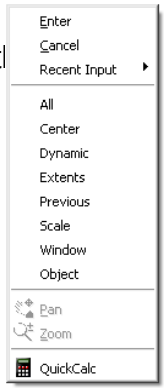
4. **Type** One of the following zoom options:

The following are basic zoom options:

All	Places entire drawing (all visible layers) on display at once. Forces a regeneration.
Extents	Displays current drawing content as large as possible.
Previous	Restores previous view.
Window	Designates rectangular area to be drawn as large as possible.
Number	Magnification relative to ZOOM All display
Number X	Magnification relative to current display (1X)
Center	Specifies center point and new display height.
Dynamic	Permits you to pan a box representing the viewing screen around the entire generated portion of the drawing and enlarge or shrink it.

TIPS:

-While in the ZOOM command, click with the right mouse button to see t
menu to the right.



5.2 PAN

Shifts the location of a view.

1. **Choose** View, Pan.

or

2. **Click** the Pan icon. 

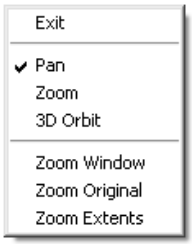
or

3. **Type** PAN from the command prompt.

   Command: **PAN** or **P**

TIPS:

- While in the PAN command, click with the right mouse button to see the following menu.



- Panning can also be done by using the window scroll bars

5.3 Redraw and Regen

Redraw refreshes the current view.

1. **Type** Redraw at the command prompt

   Command: **Redraw** or **R**

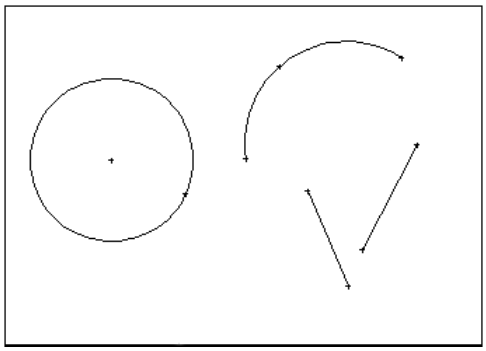
REGEN regenerates the entire drawing and recomputes the screen coordinates for all objects. It also re-indexes the drawing database for optimum display and object selection performance.

1. **Type** REGEN at the command prompt.

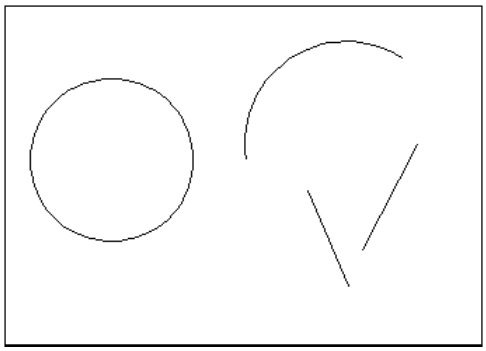
   Command: **REGEN or RE**

TIP: When BLIPMODE is on, marker blips left by editing commands are removed from the current viewport

Blips showing



Blips removed after redraw



5.4 Blipmode

Controls the display of marker blips. When Blip mode is on, a temporary mark in the shape of a plus sign (+) appears where points are specified. BLIPMODE is off by default.

1. **Type** BLIPMODE at the command prompt.

   Command: **BLIPMODE**

Chapter 6

Drawing Aids

6.1 SNAP Command

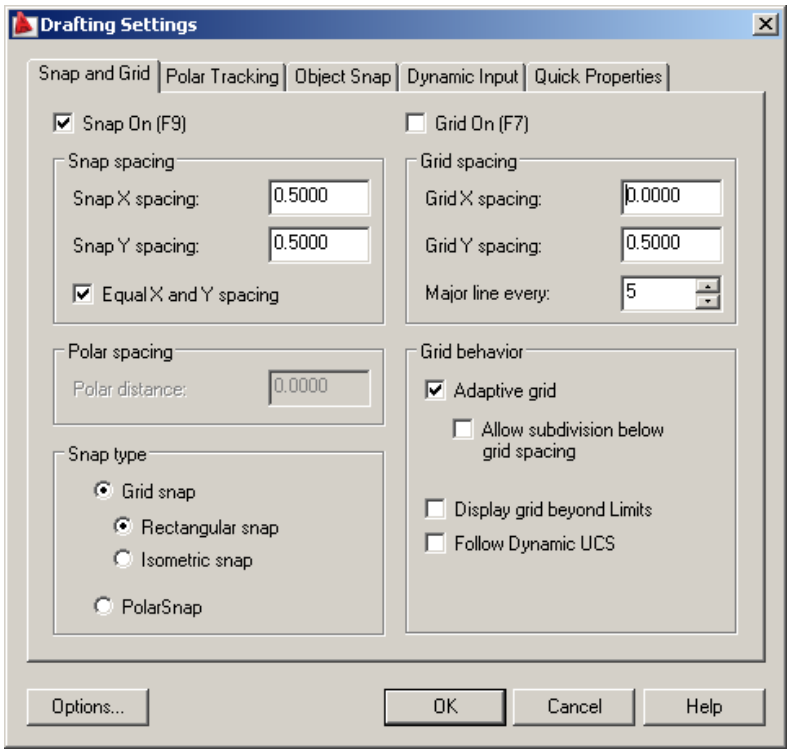
1. **Choose** Tools, Drafting Settings...

   or

2. **Type** SNAP at the command prompt.

   Command: **SNAP or SN**

3. **Type** One of the following options: Snap spacing or [ON/OFF/Aspect/Style/Type]:



Turn Snap On/OFF

1. **Press** Function Key **F9** to turn the snap ON/OFF.

   or

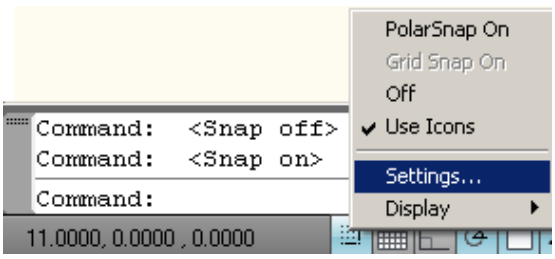
2. **Double Click** SNAP on the Status Bar.

   or

3. **Press** CTRL + B.

TIP:

Click with the right mouse button on the SNAP option from the status bar as a shortcut to changing the snap settings.



6.2 Grid Command

1. **Choose** Tools, Drafting Settings...

   or

2. **Type** DSETTINGS at the command prompt.

   Command : DSETTINGS (DS)

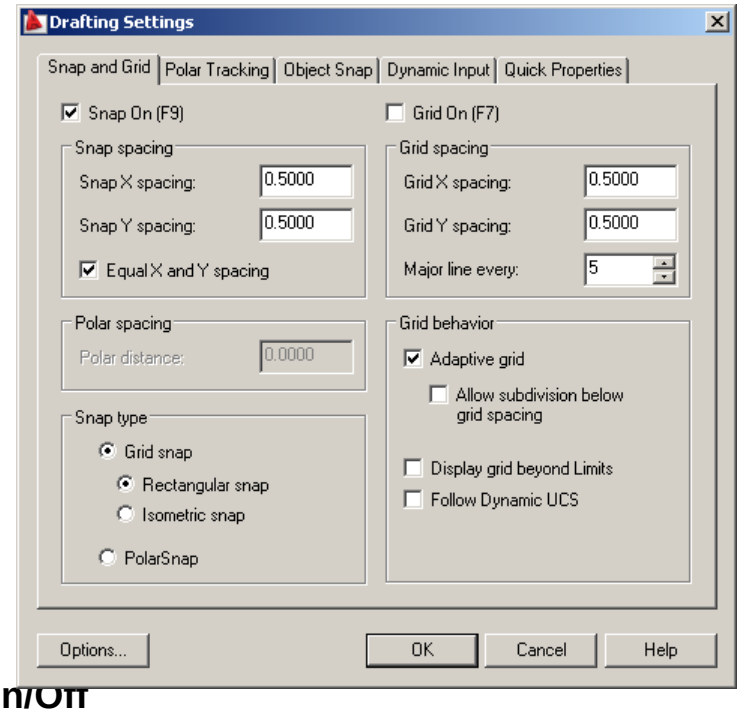
   or

3. **Type** GRID at the command prompt.

   Command: **GRID**

4. **Type** One of the following options:

   Grid spacing(X) or ON/OFF/Snap/Aspect <0000>:



Turn Grid On/Off

1. **Press** Function Key **F7** to turn the grid ON/OFF.

   or

2. **Double Click** GRID on the Status Bar.

   or

3. **Press** CTRL + G.

7.1 Running Object Snaps

An object snap mode specifies a snap point at an exact location on an object. OSNAP specifies running object snap modes, which remain active until you turn them off.

1. **Choose** Tools, Drafting Settings...

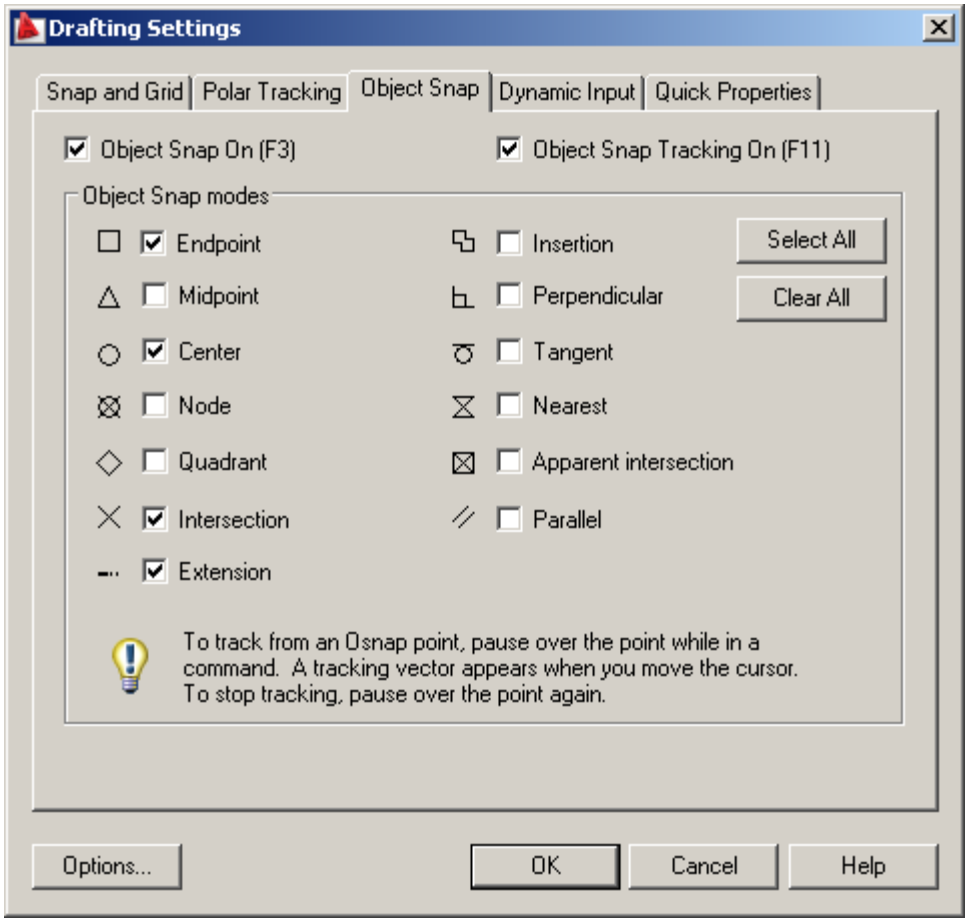
or

2. **Type** DDOSNAP at the command prompt

   Command: **DDOSNAP**

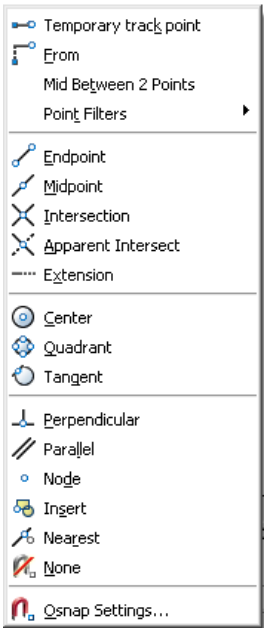
or

3. **Click** OSNAP on the Status Bar.
4. **Right Click** the Object Snap TAB.
5. **Choose** an object snap to turn ON/OFF from the dialog box.



7.2 Case by Case (Temporary Mode)

1. **Press** SHIFT + the RIGHT MOUSE BUTTON.



or

2. **Click** one of the object snaps located Object Snap toolbar icon.

or

3. **Type** The object snap at the prompt line.

   Command: Line

   From pt: **ENDP**

   To pt: **MID**

   To pt: **CEN**

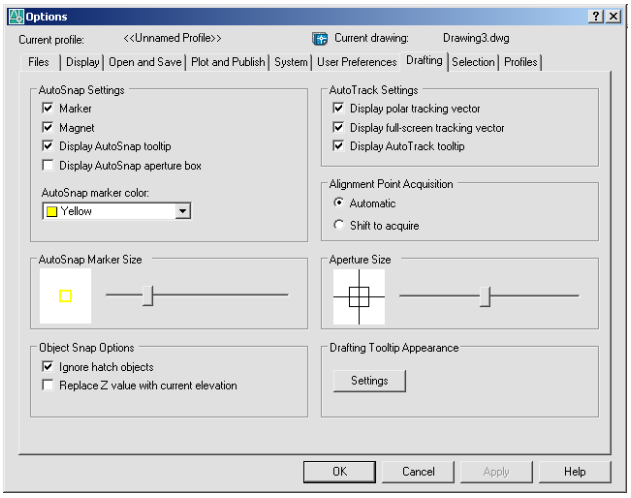
TIP:

Case by Case objects snaps will override running mode object snaps

7.3 Osnap Settings

When you use any of the object snap settings, AutoSnap displays a marker and a Snap tip when you move the cursor over a snap point.

1. **Choose** Tools, Options...
2. **Select** the Drafting tab in the Options dialog box.
3. **Change** settings and choose OK.



The following are object snap modes:

CENter	Center of Arc or Circle
ENDpoint	Closest endpoint of Line/Arc
INSertion	Insertion point of Text/Block/Shape/ Attribute
INTersection	Intersection of Lines/Arcs/Circles
MIDpoint	Midpoint of a line/Arc or midpoint
NEAerst	Nearest point on a Line/Arc/Circle/Point
APParent Int	Finds where two entities would intersect
NODe	Nearest point entity (or Dimension defini tion point)
NONe	None (off)
PERpendicular	Perpendicular to a Line/Arc/Circle
QUAdrant	Quadrant point on an Arc/Circle
QUIck	Quick mode (first find, not closest)
TANgent	Tangent to Arc or Circle

7.4 Aperture

Controls the size and appearance of the pickbox used for object snap selection.

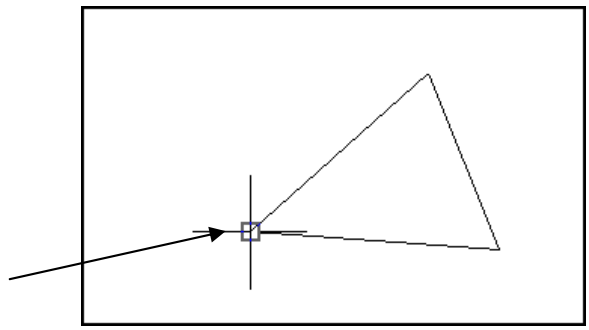
1. **Type** APERTURE at the command prompt

   Command: **APERTURE**

2. **Type** The size of the target box (3-8 is a good size)

   Size of target box in pixels (1-50): (**number**)

or



Aperture

Chapter 8
Setting Up a Drawing

List Command 8.1

1. **Choose** Tools, Inquiry, List.

   or

2. **Click** the List icon from the Inquiry Toolbar. 

   or

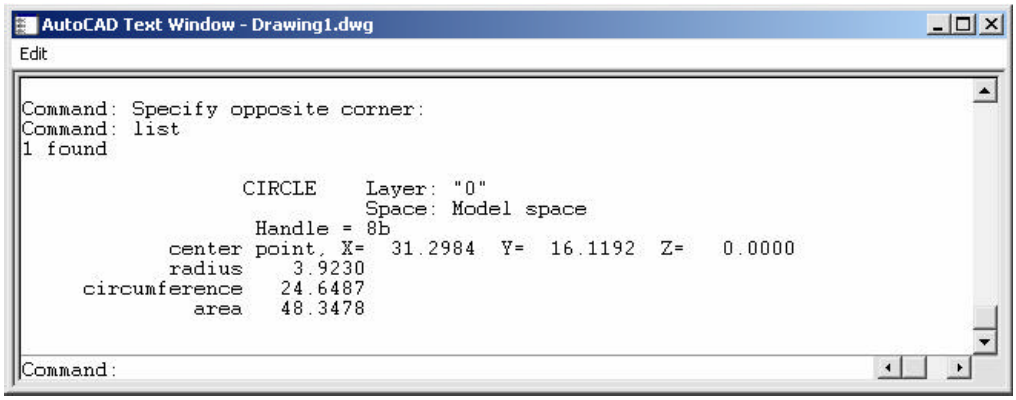
3. **Type** LIST at the command prompt.

   Command: **LIST or LI**

4. **Pick** The object or objects to list.

   Select objects: (**select**)

5. **Press** ENTER when you are finished choosing objects:



Measuring Distances 8.2

1. **Choose** Tools, Inquiry, Distance.

   or

2. **Click** the Distance icon from the Inquiry Toolbar. 

   or

3. **Type** DIST at the command prompt

   Command: **DIST**

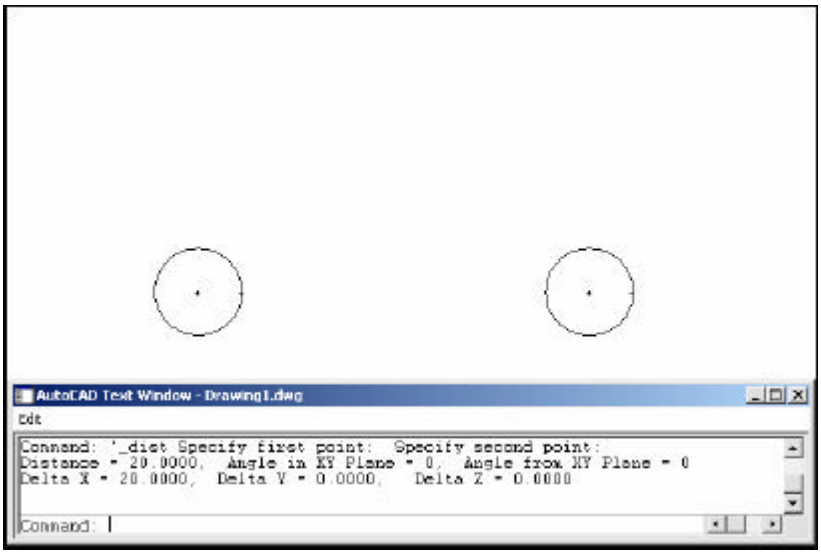
4. **Pick** The first point to measure from

   First point: **pick point**

5. **Pick** The second point to measure to

   Second point: **pick point**

Distance Between Circle Centers



TIP:

Be sure to use Object Snaps with the MEASURE command.

Calculating Areas 8.3

1. **Choose** Tools, Inquiry, Area.

   or

2. **Click** the Area icon. 

   or

3. **Type** AREA at the command prompt

   Command: **AREA**

4. **Pick** The first point for area calculation

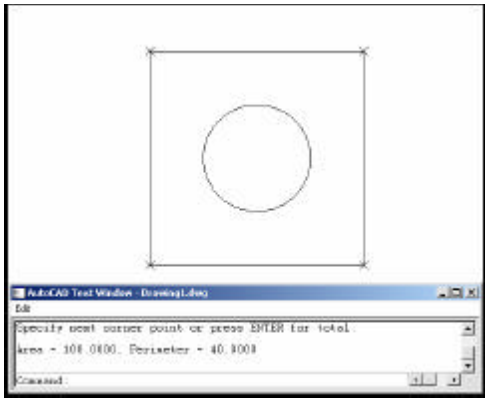
   <First point>/Object/Add/Subtract: **pick**

5. **Pick** Next point: **pick**

6. **Pick** Next point: **pick**

7. **Press** ENTER when you are finished choosing points.

   Area of Rectangle



Object Allows user to pick an object to calculate area (circle or polyline).

Add Adds separate areas for a total area calculation

Subtract Subtracts areas from each other.

TIPS:

Be sure to use Object Snaps with the MEASURE command

To subtract an area, you must first be in “add” mode to add the first area.

ID Command 8.4

1. **Choose** Edit, Inquiry, Locate Point.

or

2. **Click** the Locate Point Icon from the Inquiry Toolbar. 

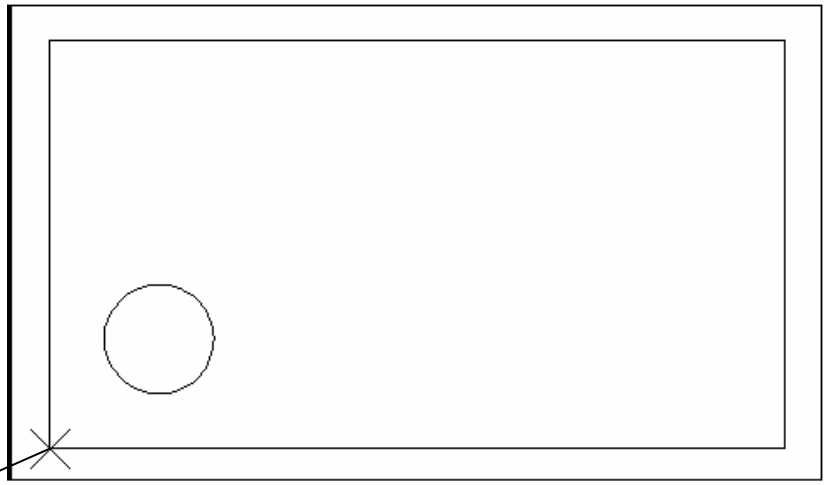
or

3. **Type** ID at the command prompt.

   Command: **ID**

4. **Pick** A point to identity

   Point : **pick point**



Using ID at the corner of the box rests the "0,0" origin for relative coordinates

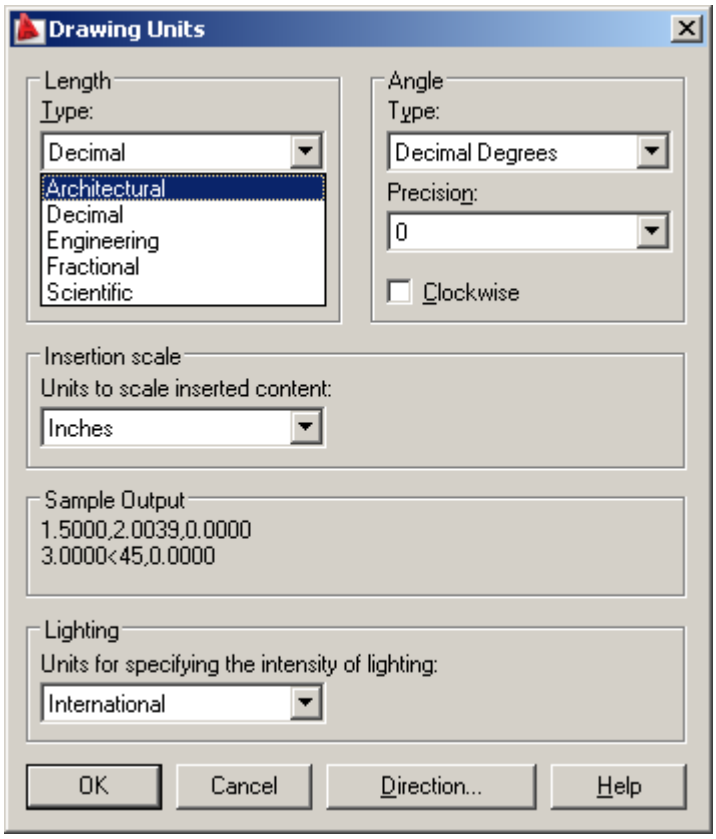
TIP:

AutoCAD returns the X,Y, and Z coordinates as well as making this the last point entered in the drawing (to move relative from)

Be sure to use Object Snaps with the ID command.

UNITS Command 8.5

1. **Choose** Format, Units...
   or
2. **Type** DDUNITS at the command prompt.
   Command: **DDUNITS or UN**
3. **Choose** a units and angle setting.
4. **Choose** a precision setting.



Drawing Limits 8.6

The drawing limits are two-dimensional points in the World Coordinate System that represent a lower-left limit and an upper-right limit.

The drawing limits also govern the portion of the drawing covered by the visible grid and determine the minimum area a ZOOM All displays.

1. **Choose** Format, Drawing Limits.

   or

2. **Type** LIMITS at the command prompt

   Command: **LIMITS**

3. **Type** One of the following options
   On/Off/Lower left corner <.000,0.000>: **0,0**

4. **Type** One of the following options for the upper right limit:

   Upper right corner <12.0000,9.0000>:**36,24**

Drawing with lower left limit of 0,0 and upper right limit of 36,24

TIPS:

You can also pick points to define the limits.

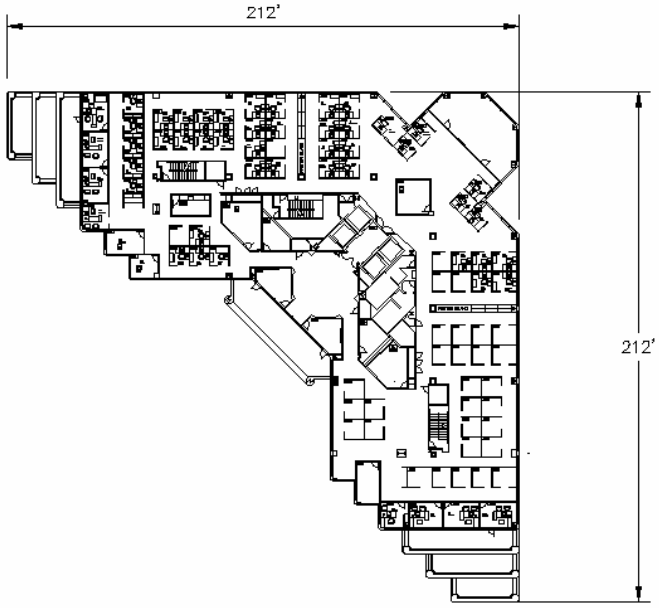
The limcheck variable controls whether or not you can draw outside the limits that are set. A setting of 0 (off) indicates that you can draw outside the limits and a setting of 1(on) indicates that you cannot.

Plot Scales and Paper Sizes 8.7

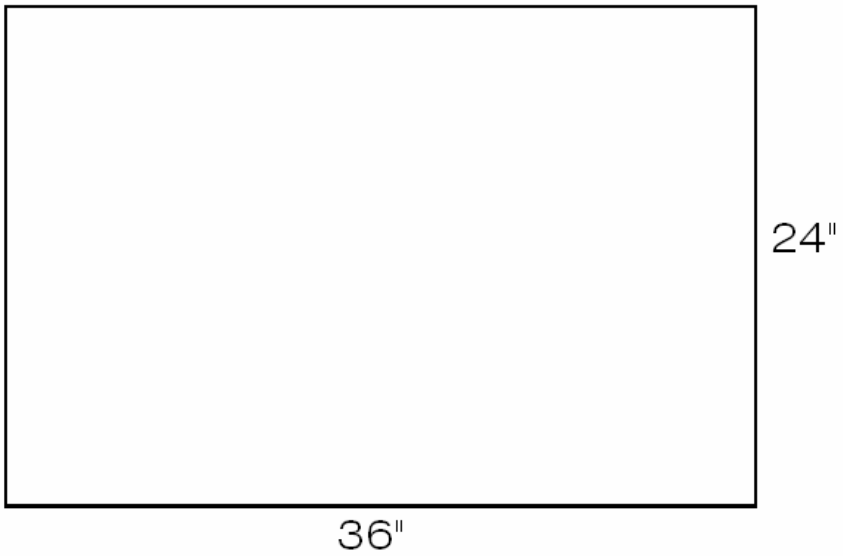
The following is an example of setting up an AutoCAD drawing for a D size sheet of paper (36 x24) with a scale of 1/16=1').

1.	**Size**	the object you're drawing.
2.	**Border Size**	36 x 24 plotted, 576' x 384' drawn.
		For some plotters, deduct a 1/2 margin on top, bottom, and left,and a 1 margin on the right.
3.	**Limits**	Lower left limit 0,0.
		Upper right limit 576', 384'.
4.	**Text Height**	for 1/8 notes, multiply by 192 which is the reciprocal of the plot scale.
		1/8 plotted, 24" drawn.
5.	**Hatch Scale**	for patterns other than architectural.
		Hatch Scale = 192
6.	**Dimension Scale**	Dimscale = 192
7.	**Ltscale**	Ltscale = 96

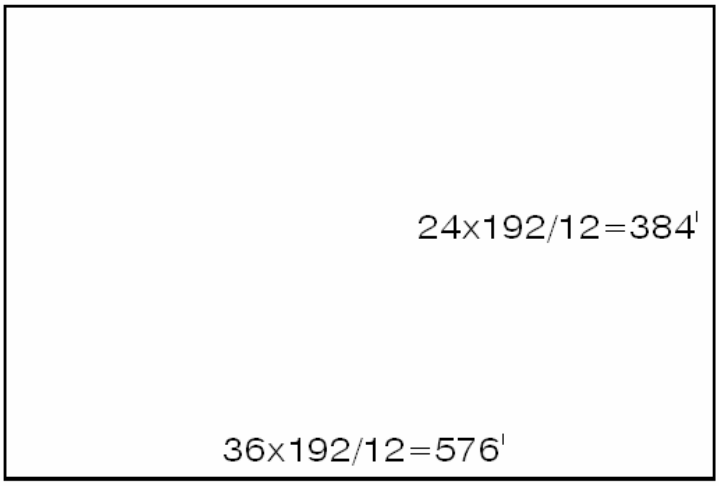
Determine your object size



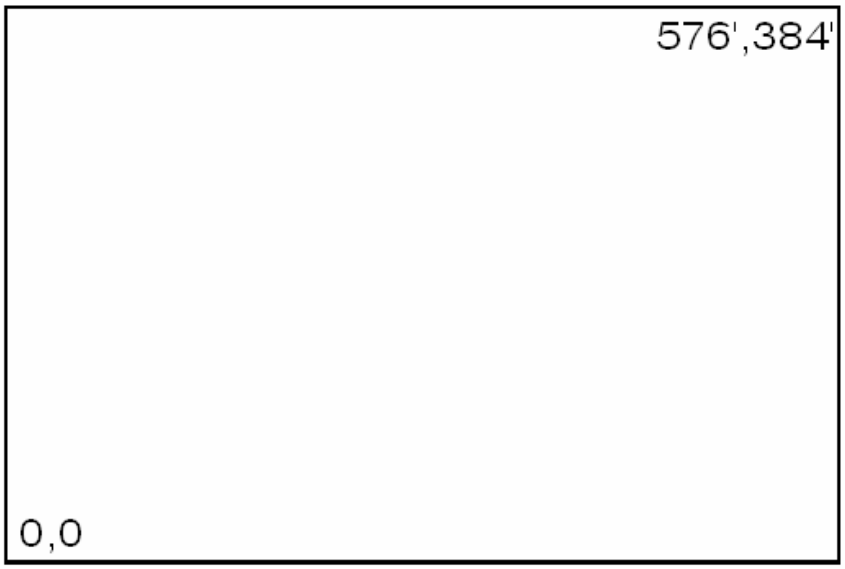
Decide Border (Paper) Size



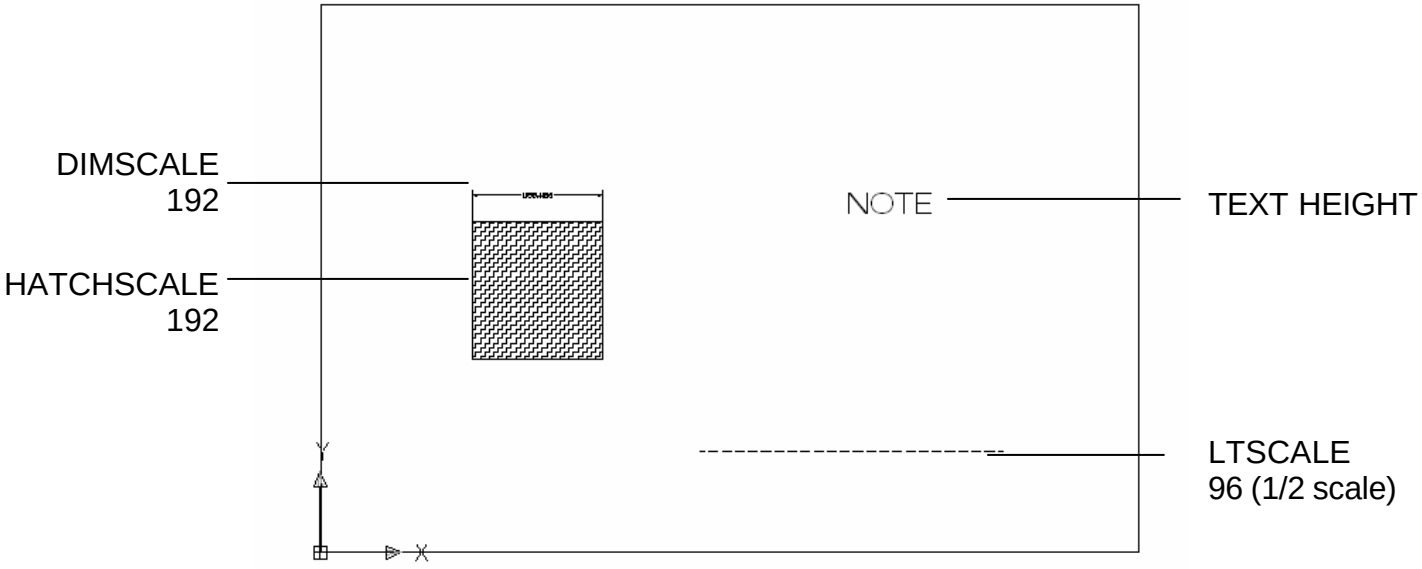
Decide the Scale Factor for Object which is at least 212', 212'. To do this, multiply the scale factor x paper size. (i.e.: 1/16"=1'-0' has scale factor 192)



Set Drawing Limits



Determine Dim Scale, Hatch Scale, Ltscale, and Text Height



Chapter 9
Plotting

Plot Command 9.1

1. **Choose** File, Plot.

or

2. **Click** the Plotter icon. 

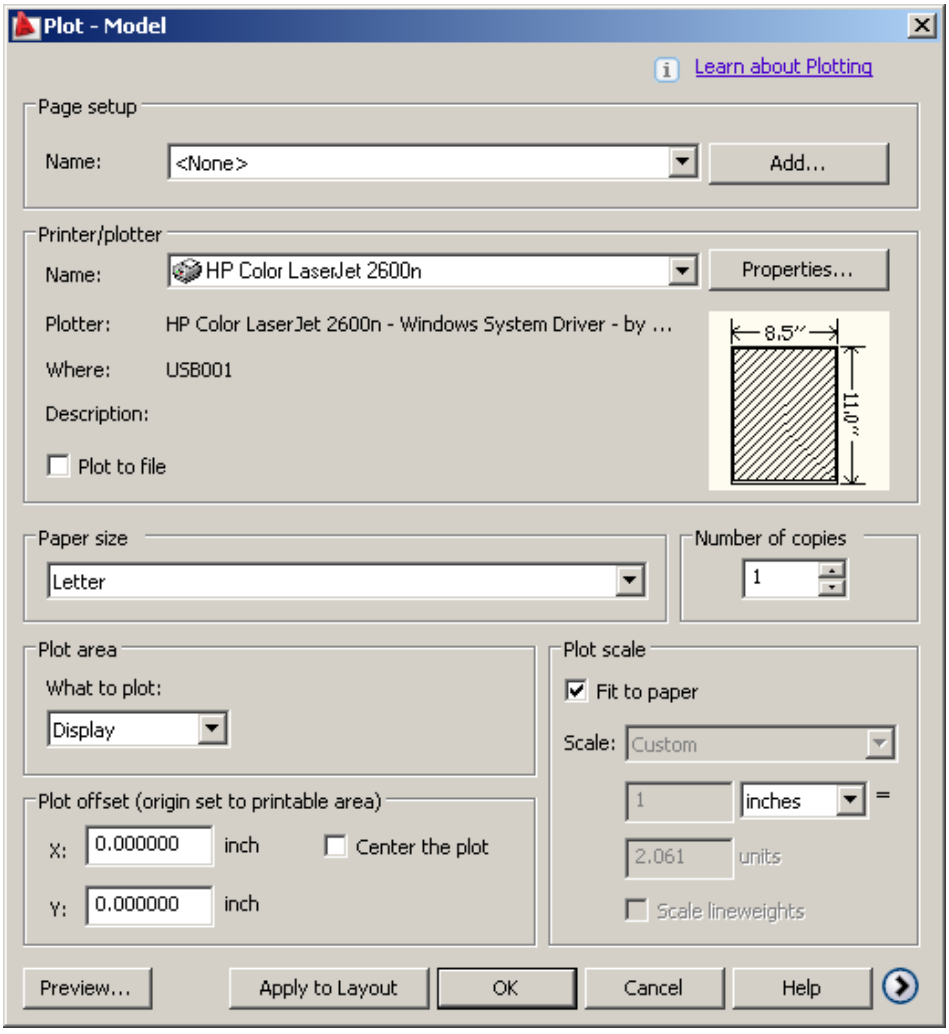
or

3. **Type** PLOT at the command prompt.

Command: **PLOT or PRINT**

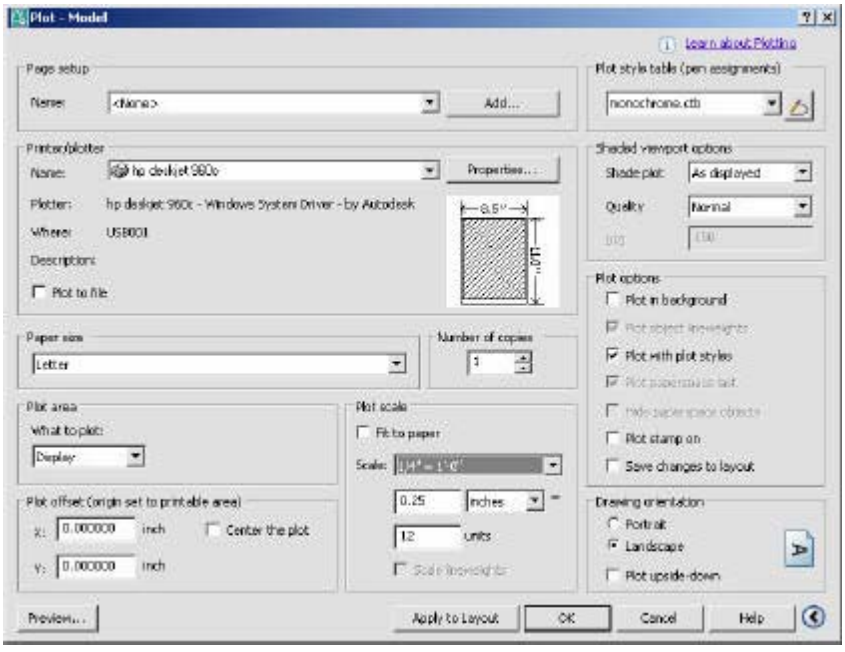
or

4. **Press** CTRL + P



Plot Settings

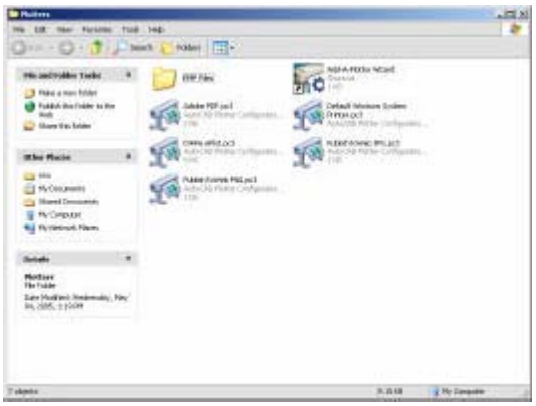
1. **Choose** the Plot Settings tab.
2. **Choose** the appropriate paper size based on the chosen plotter.
3. **Choose** the paper units (inches or mm).
4. **Choose** the drawing orientation (Portrait, Landscape, Upside down).
5. **Choose** the plotting area.
6. **Choose** the plot scale.
7. **Choose** plot to center or specify an x or y offset.
8. **Click** OK.



Adding a Plotter 9.2

Plotter Manager Wizard

1. **Choose** File, Plotter Manager
2. **Double-Click** the Add a Plotter Wizard icon.



AutoCAD adds a plotter configuration to a saved plot file called ?.PC3. You can then load from this file later.

3. **Click** Next >
4. **Choose** My Computer.

My Computer will configure a plotter using Autodesk Drivers. *System Printer* will configure AutoCAD using Window's printer drivers that are already installed.

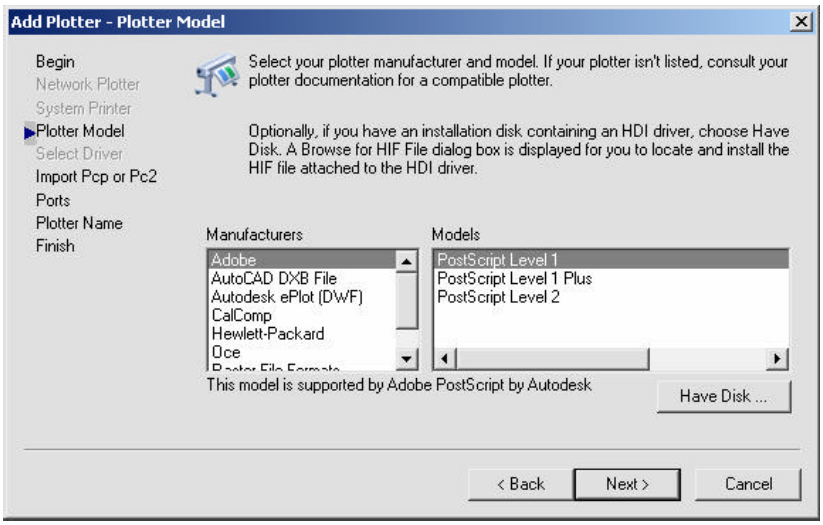
5. **Click** Next >

Choosing a Plotter Driver

6. **Choose** one of the Autodesk Plotter options.

   Your purchased plotter should be listed here. If it is not, you can choose "Have Disk..." and specify a location for a plotter driver.

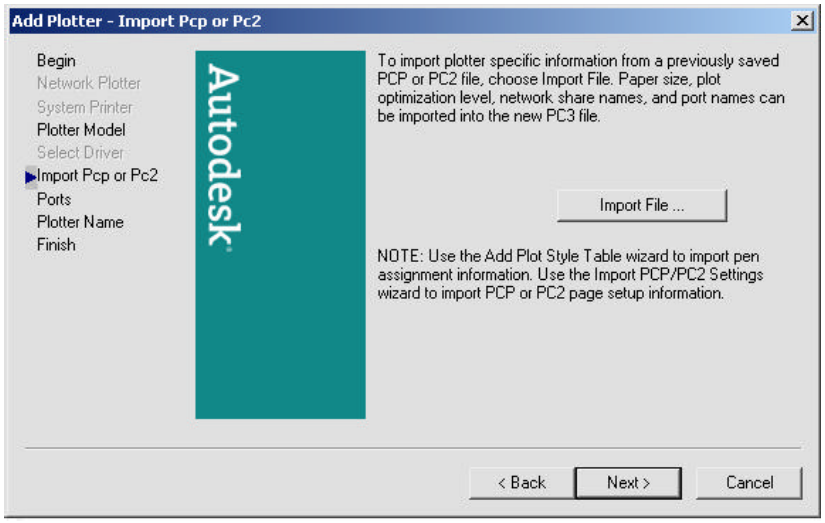
   You can also plot to a file by choosing the DXB, Autodesk ePlot, or Raster File options.



Importing a .PCP or .PC2 file

7. **Choose** Import File...if you wish to import a previously saved plot configuration file.

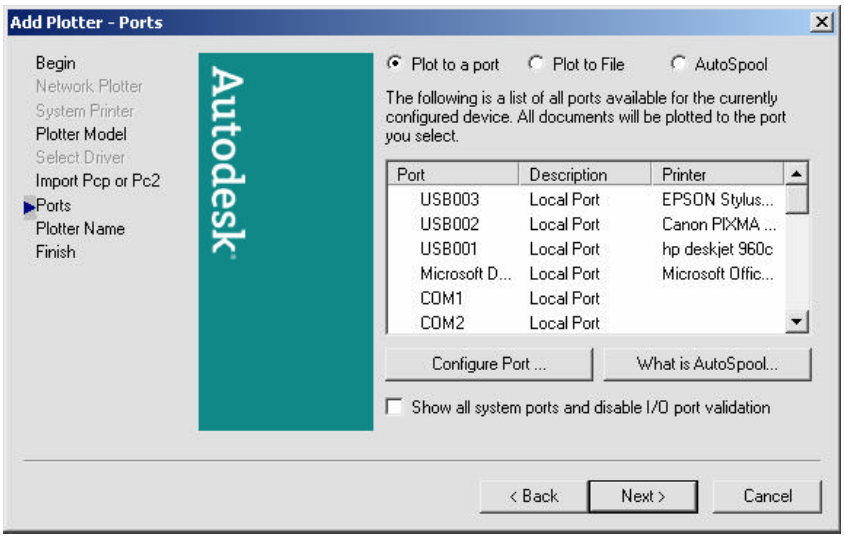
8. **Click** Next >



Define a Port

9. **Choose** an available port.
   NOTE: You can plot to a specific file name or "Autospool" to a file which can be automatically sent to the plotter at a later date.

10. **Choose** Next >



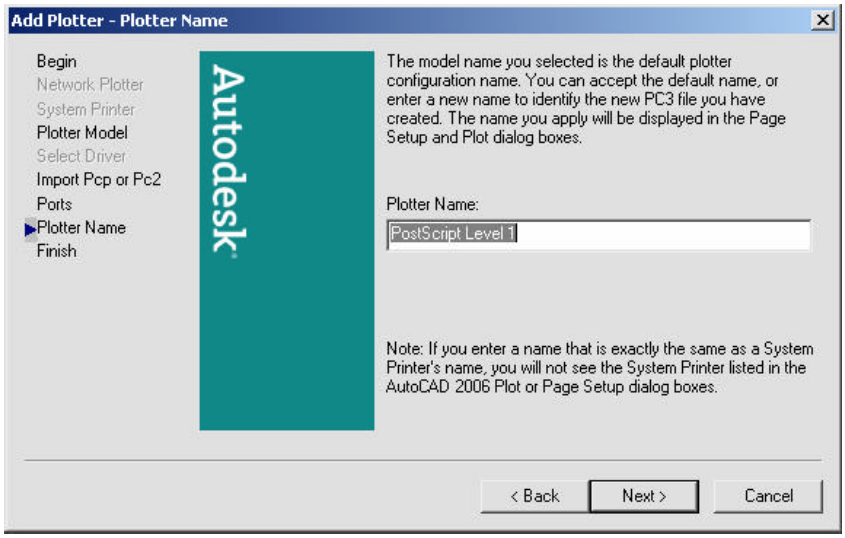
Saving a Plot Configuration Name

11. **Type** the file name you wish to save.

12. **Choose** Next >

13. **Click** Finish

AutoCAD will save a new Plot configuration icon.



Plot Styles 9.3

Add a Plot Style

A plot style controls how an object or layer is plotted by determining plotted properties such as lineweight, color, and fill style. Plot style tables collect groups of plot styles. The Plot Style Table Manager is a window that shows all the plot style tables available in AutoCAD.

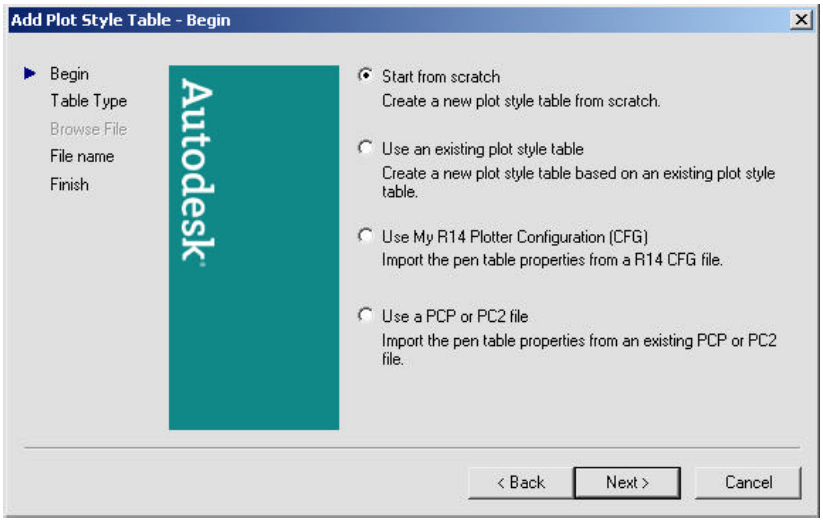
There are two plot style types: color-dependent and named. A drawing can only use one type of plot style table. You can convert a plot style table from one type to the other. You can also change the type of plot style table a drawing uses once it has been set.

1. **Choose** File, Plot Style Manager.
2. **Double-Click** Add a Plot Style Table Wizard icon.

3. **Click** Next >
4. **Choose** Start from Scratch to create a new Plot Style.

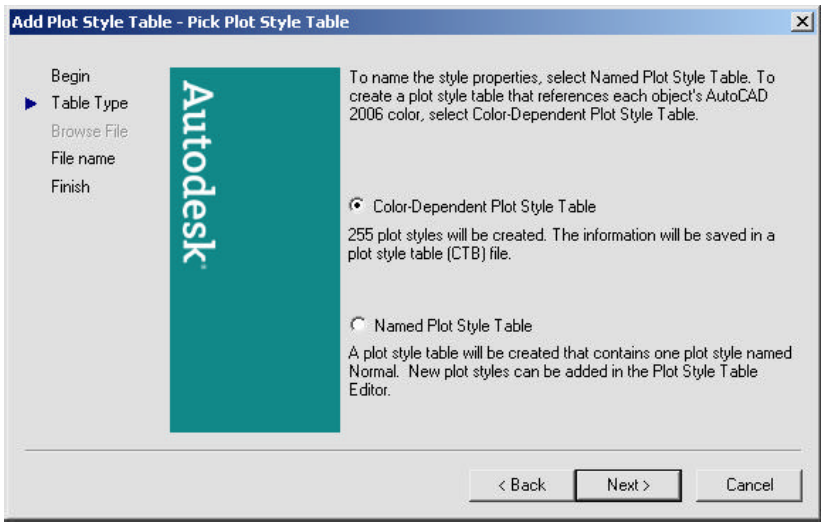
   You can also use a previously configured plot style, import a style from a previous release of AutoCAD, or import a pen table.

5. **Click** Next >



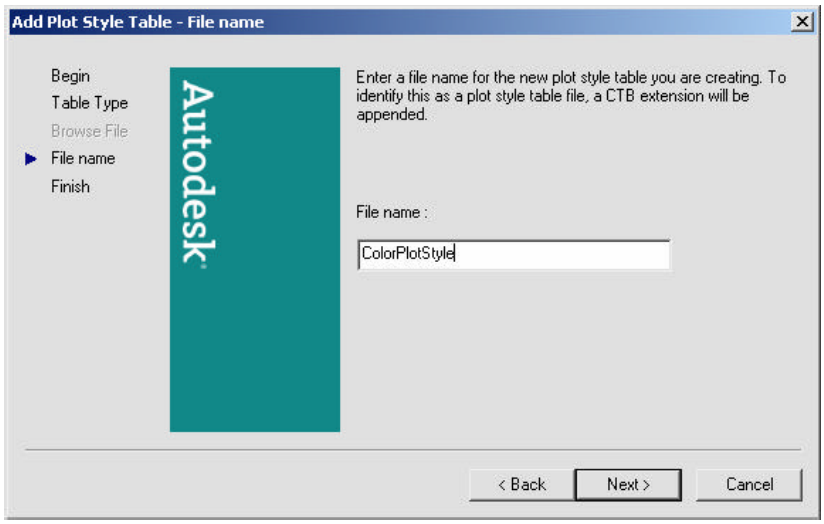
6. **Choose** Color-Dependent Plot Style Table

7. **Click** Next >

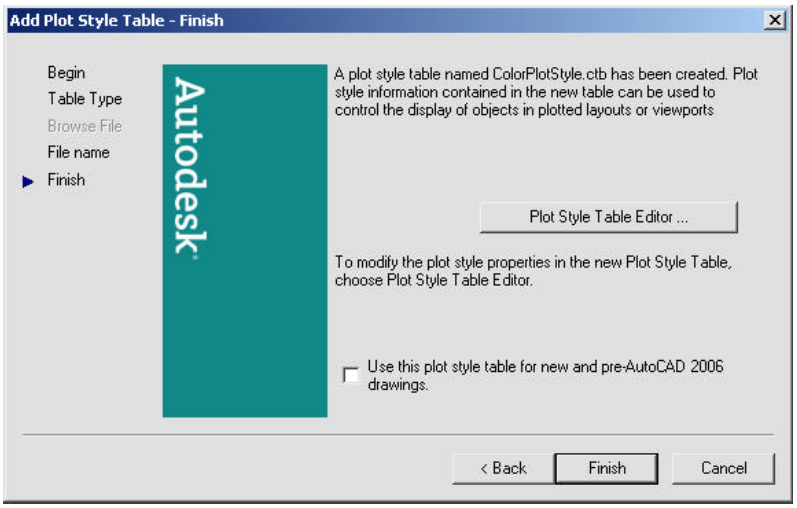


8. **Type** a name for the plot style table.

9. **Click** Next>



10. **Choose** "Plot Style Table Editor..."

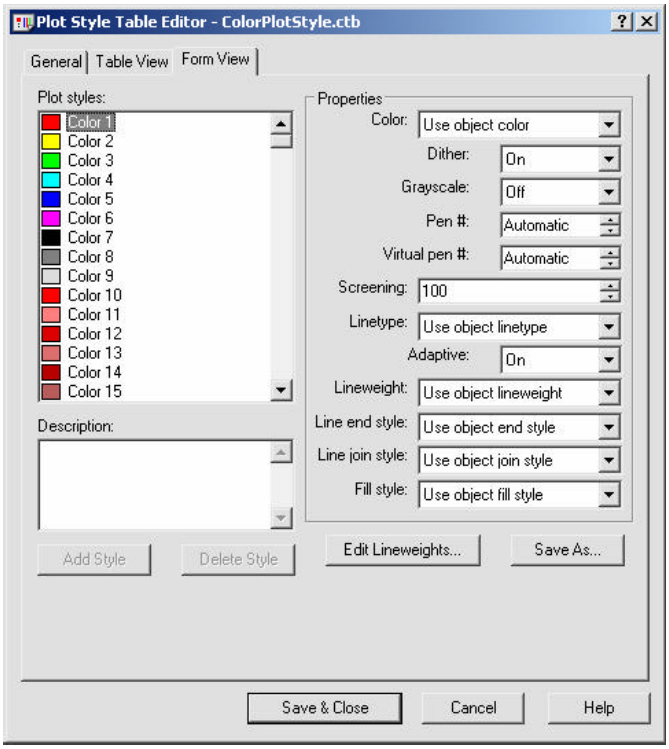


11. **Pick** an AutoCAD color and assign properties to it.

    For example, if you want all RED objects to be plotted with a pen width of .25 mm, choose that lineweight.

12. **Choose** Save and Close

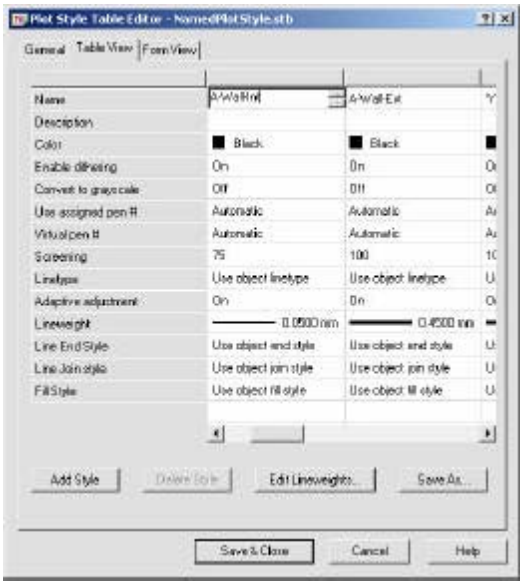
13. **Choose** Finish

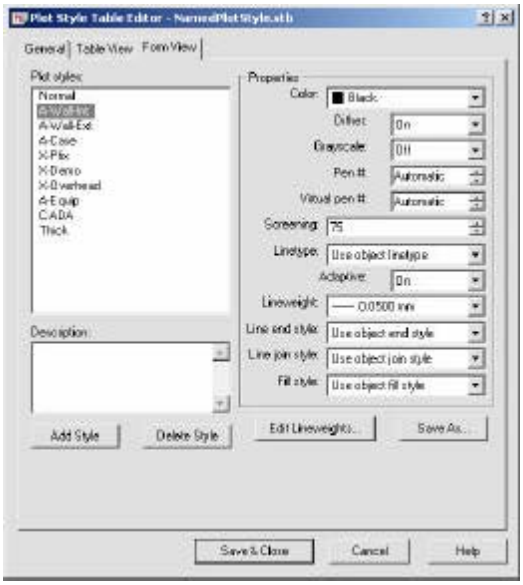


AutoCAD will save the file called COLORPLOTSTYLE.CBT

Named Plot Styles 9.4

1.	**Choose**	File, Plot Style Manager.
2.	**Double-Click**	Add a Plot Style Table Wizard icon.
3.	**Click**	Next >
4.	**Choose**	Start from Scratch to create a new Plot Style.
5.	**Click**	Next >
6.	**Choose**	Named Plot Style Table
7.	**Click**	Next >
8.	**Type**	a name for the plot style table.
9.	**Click**	Next>
10.	**Choose**	“Plot Style Table Editor...”
11.	**Create**	names for various styles.
12.	**Choose**	Save and Close.





Chapter 10
Edit Commands

Move Command 10.1

1. **Choose** Modify, Move.

or

2. **Click** the Move icon. 

or

3. **Type** MOVE at the command prompt

   Command: **MOVE or M**

4. **Pick** Objects to move

   Select objects: (**select**)

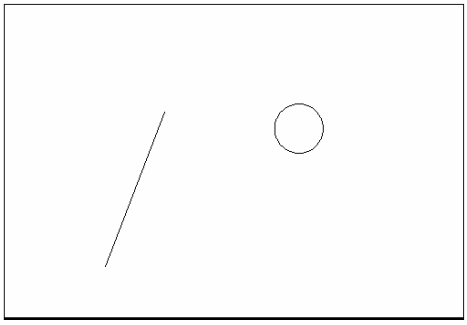
5. **Pick** A point to move from

   Base point or displacement: (**pick point**)

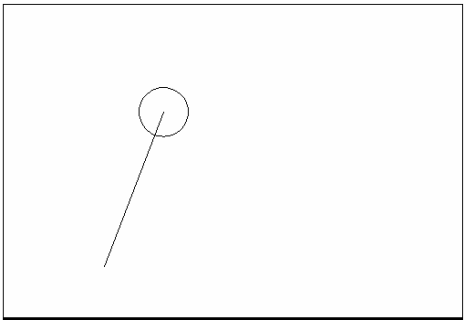
6. **Pick** A point to move to

   Second point of displacement: (**pick point**)

Circle before move



Circle after move



TIP:

To move an object a specified distance, type a distance at the second point of displacement prompt: **@1<0**

Copy Command 10.2

1. **Choose** Modify, Copy.

or

2. **Click** the Copy icon. 

or

3. **Type** COPY at the command prompt.

   Command: **COPY or CP**

4. **Pick** Objects to copy.

   Select objects: (**select**)

5. **Pick** A point to move from.

   Base point or displacement/Multiple: (**pick point**).

6. **Pick** A point to copy to.

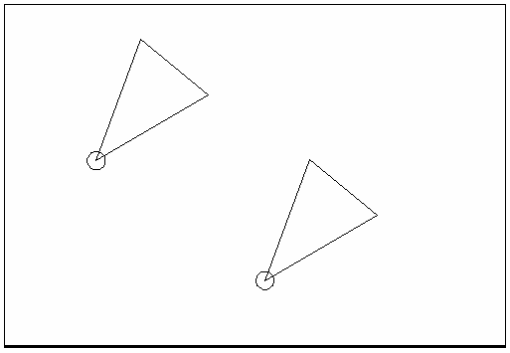
   Second point of displacement: (**pick point**)

or

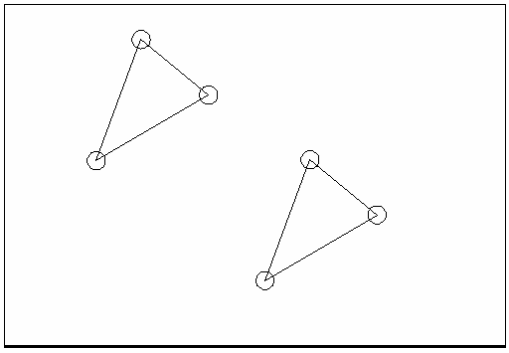
7. **Type** A point to copy to.

   Second point of displacement: @ **1<0**

Duplicate objects copied



Multiple objects copied



TIP:

- To copy many objects in the same copy command, type M for Multiple at the “Base point or displacement/Multiple” option.

Previous Selection 10.3

Places selected objects in the Previous selection set

1. **Choose** Modify, Move.

or

2. **Click** the Move icon. 

or

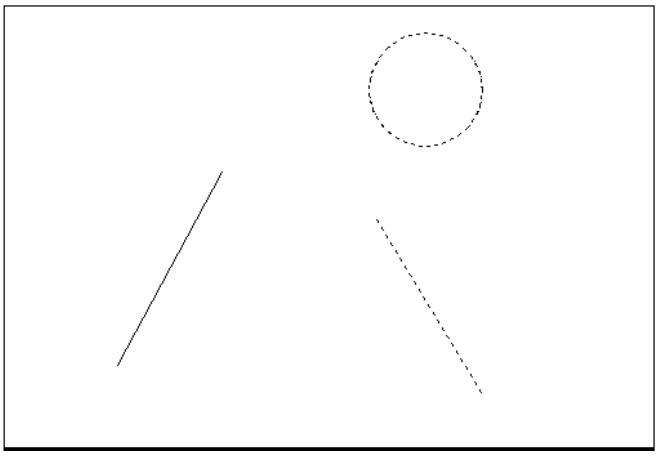
3. **Type** MOVE at the command prompt.

Command: **MOVE or M**

4. **Pick** Objects to move.

Select objects: (**P**)

Previous Selection Set Highlighted



TIP:

AutoCAD requires that objects be selected in order to be processed. The Select Objects prompt occurs after many commands, including the SELECT command itself.

Offset Command 10.4

Offset Distance

To offset a specified distance:

1. **Choose** Modify, Offset.

   or

2. **Choose** the Offset icon. 

   or

3. **Type** OFFSET at the command prompt.

   Command: **OFFSET or O**

4. **Type** The distance to offset.

   Offset distance or <Through point>: (**number**)

5. **Pick** The object to offset.

   Select object to offset: (**select object**)

6. **Pick** A side to offset object to.

   Side to offset: (**pick side**)

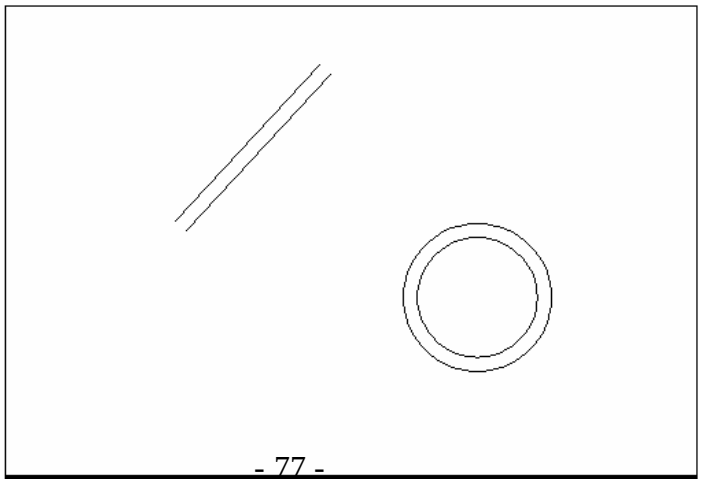
7. **Pick** Another object to offset

   Select object to offset: (**pick side**)

   or

8. **Press** Enter to end the command.

Offsetting objects by specifying a distance



Offset Through Point

To offset through point :

1. **Type** OFFSET at the command prompt

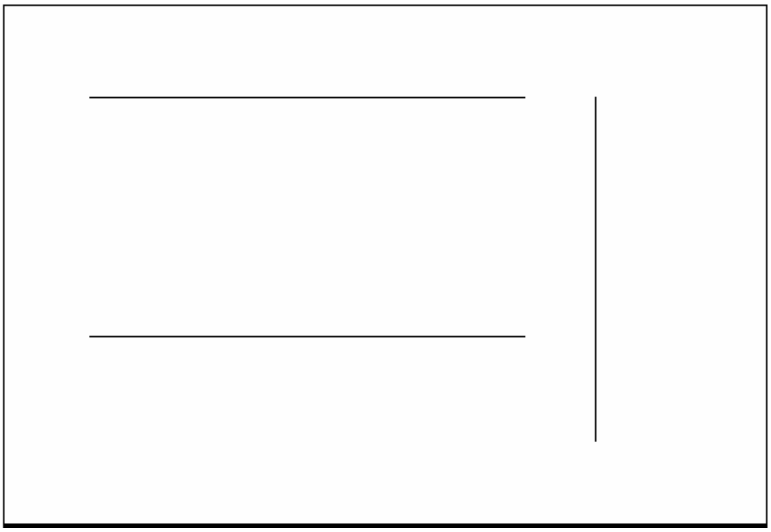
   Command: **OFFSET**

2. **Type** T to specify a through point

   Offset distance or <Through point>: (**T**)

3. **Pick** A point to offset through (HINT: use object snaps) Select object to offset: (**pick**) Through point: (**select object**)

Offset through a point



EXTEND 10.5

1. **Choose** Modify, Extend.

or

2. **Click** the Extend icon. 

or

3. **Type** EXTEND at the command prompt

   Command: **EXTEND**
   Select boundary edge(s)...

4. **Pick** The BOUNDARY edge to extend to

   Select objects: (**select**)

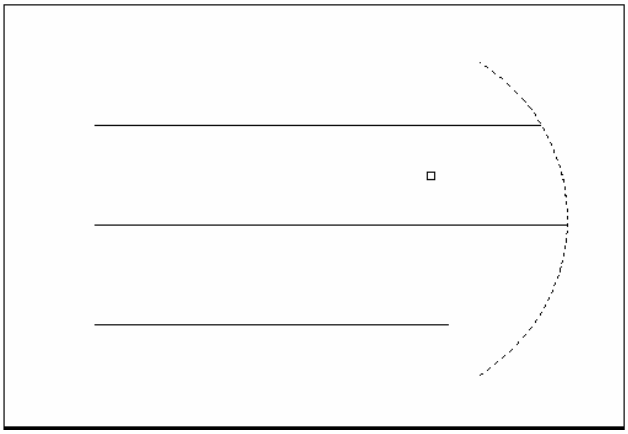
5. **Press** ENTER to accept the boundary edge

   Select objects: (**press enter**)

6. **Pick** The objects to extend

   <Select object to extend> / Project / Edge / Undo: Select an object, enter an option, or press enter : (**select**)

7. **Press** ENTER when you are done choosing objects



Lines Extended to an Arc (Arc is boundary edge)

TIP:

- Use the object selection option FENCE to choose multiple objects

TRIM 10.6

The TRIM command allows you to trim objects in a drawing so they end precisely at a cutting edge defined by one or more other objects in the drawing.

1. **Choose** Modify, Trim.

   or

2. **Click** the Trim icon. 
3. **Type** TRIM at the command prompt

   Command: **TRIM**

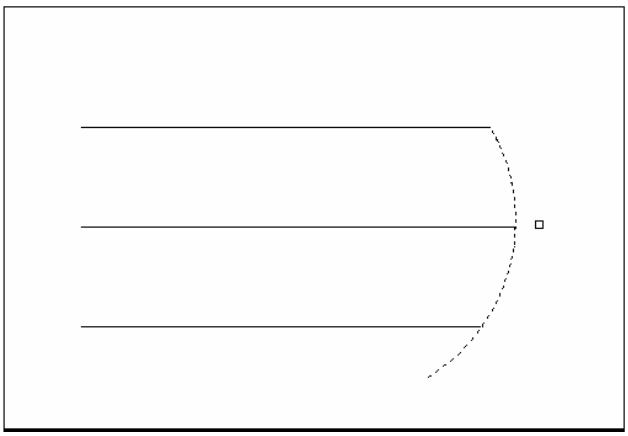
   Select cutting edge(s)...
4. **Pick** The CUTTING edge to extend to

   Select objects: (**select**)
5. **Press** ENTER to accept the cutting edge

   Select objects: (**press enter**)
6. **Pick** Objects to trim

   <Select object to trim> / Project / Edge / Undo:
   Select an object, enter an option, or press enter
7. **Press** ENTER when you are done choosing objects

   Select object to trim/Undo: (**press enter**)



Lines Trimmed to an Arc (Arc is cutting edge)

TIP: Hold the SHIFT key to interactively extend instead of trim.

Edgemode

Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

0 Uses the selected edge without an extension.

1 Extends the selected edge to its natural boundary.

MIRROR 10.7

1. **Choose** Modify, Mirror.

or

2. **Click** the Mirror icon. 

or

3. **Type** MIRROR at the command prompt.

   Command: **MIRROR**

4. **Pick** Objects to mirror.

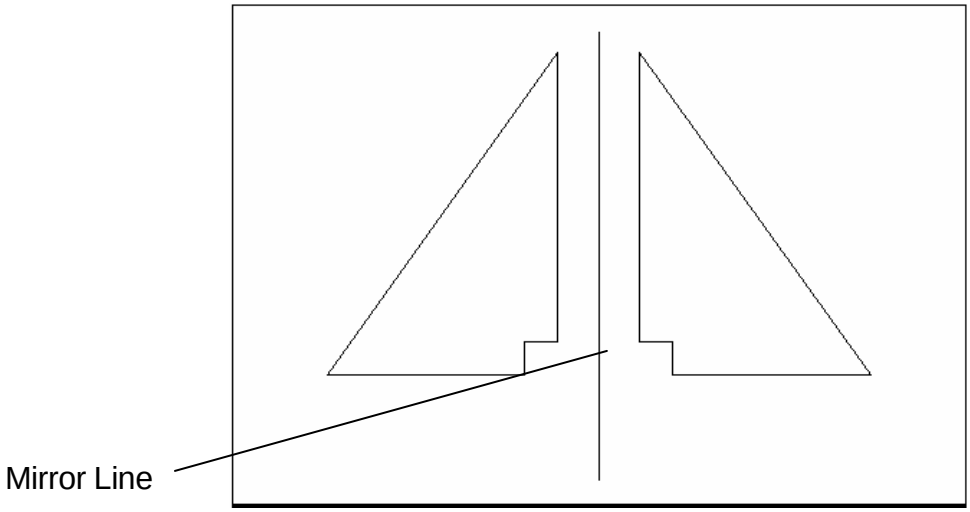
   Select objects:(**select**)

5. **Pick** First point of mirror line: (**point**)

6. **Pick** Second point: (**point**)

7. **Type** Yes to delete the original objects and No to keep them.

   Delete old objects? **Y** or **N**



Mirrtext 10.8

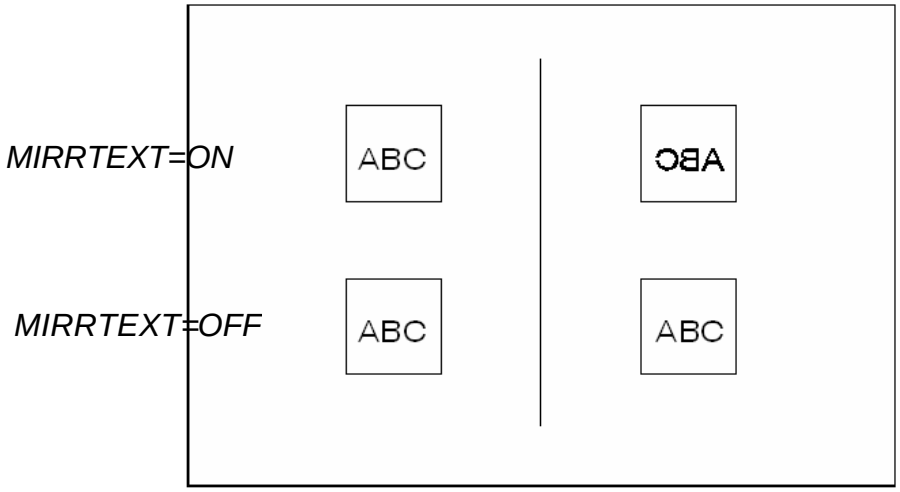
Mirror reflects (mirrors) text if 1, retains text direction if 0.

1. **Type** MIRRTEXT at the command prompt.

   Command: **MIRRTEXT**

2. **Type** 1 to reflect the text and 0 to retain the text.

   Current value <0> New value: **1** or **0**



ROTATE 10.9

1. **Choose** Modify, Rotate.

   or

2. **Click** the Modify icon. 

   or

3. **Type** ROTATE at the command prompt

   Command : **ROTATE**

4. **Pick** Objects to rotate:

   Select objects:(**select**)

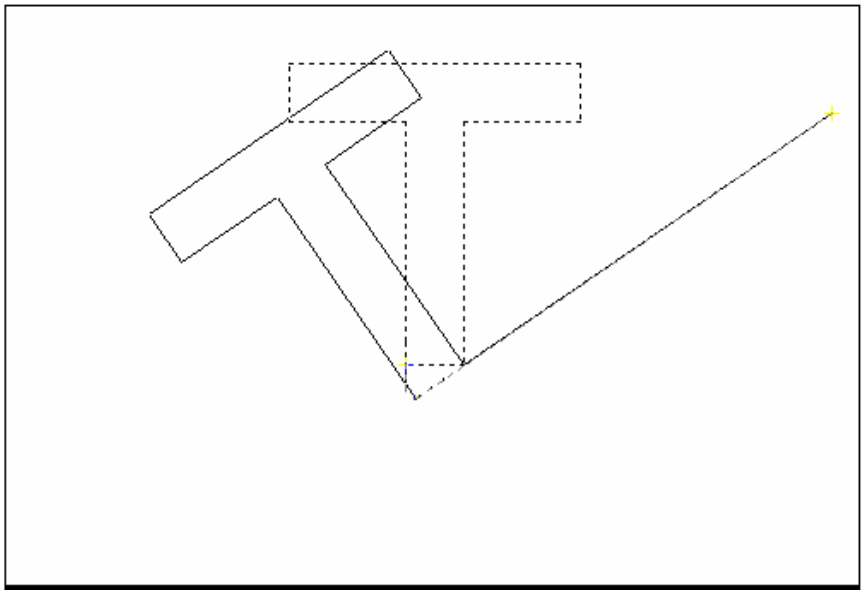
5. **Pick** A pivot point to rotate around

   Base point: (**point**)

6. **Type** A rotation angle<Rotation angle>/Reference: (**number**)

   or

7. **Pick** A rotation angle<Rotation angle>/Reference: (**point**)



Reference Angle Rotation

A positive angle causes counterclockwise rotation, and a negative angle produces clockwise rotation. If you respond to the last prompt with r, you can specify the current rotation and the new rotation you want. AutoCAD prompts:

1.	**Type**	R for a rotation angle<Rotation angle>/Reference: (**R**)
2.	**Choose**	An existing rotation angle Rotation angle:
		(**number or points**)
3.	**Choose**	A new rotation angle New angle:
		(**number or points**)

TIP:

You can show AutoCAD the reference angle (by pointing to the two endpoints of a line to be rotated), and then specify the new angle. You can specify the new angle by pointing or by dragging the object.

SCALE 10.10

1. **Choose** Modify, Scale.

or

2. **Click** the Scale icon. 

or

3. **Type** SCALE at the command prompt

   Command: **SCALE**

   Select objects: (**select objects**)

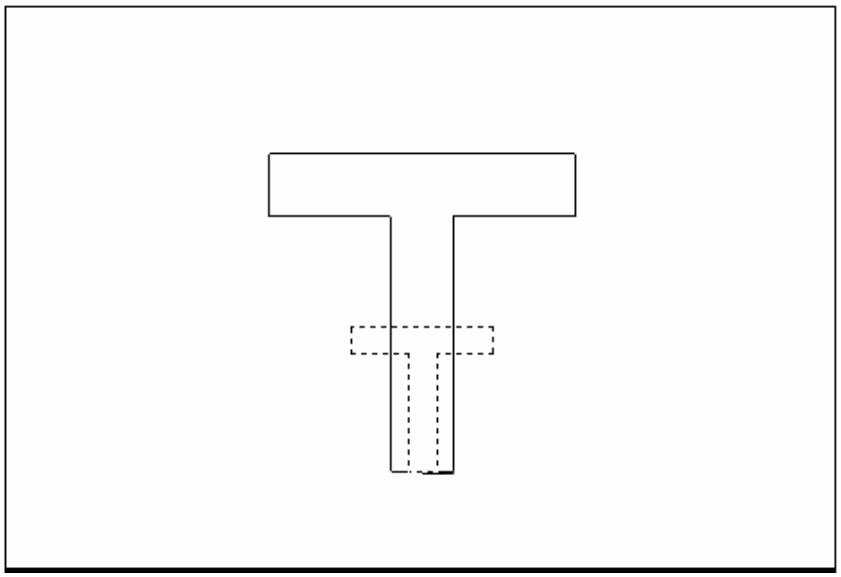
4. **Pick** A pivot point to scale about Base point: (**point**)

5. **Type** A rotation angle<Scale factor>/Reference:(**number**)

or

6. **Pick** A scale factor<Scale factor>/Reference: (**point**)

   Scale factor/Reference: (**points**)



Scale by Specifying Length

You can show AutoCAD the reference length (by pointing to the two endpoints of a line to be scaled), and then specify the new length. You can specify the new length by pointing, or by dragging the object.

1. **Type** R to define a reference length

   Scale factor/Reference: (R)

2. **Choose** A reference scale factor

   Reference length : (number or points)

3. **Choose** A new scale factor

   New length: (number or points)

Chapter 11
Text

Text Command 11.1

Text

Creates a single-line text object

1. **Type** TEXT at the command prompt

   Command: **TEXT**

   or

2. **Pick** the Single Line Text icon from the Text Toolbar. 
3. **Pick** A start point

   Justify/Style/<Start Point>: (**point**)

   or

4. **Type** J to change the justification or S to change the text style.
5. **Type** A text height

   Height <default>: (**type value or pick two points**)
6. **Type** A rotation angle

   Rotation angle <default>: (**angle or point**)
7. **Type** A text string

   Text: (**type text string**)
8. **Press** enter to exit the Text: prompt.

DTEXT (Dynamic Text)

Creates a single-line text object, showing the text dynamically on the screen as it is entered.

1. **Choose** Draw, Text, Single Line Text.

   or

2. **Type** DTEXT at the command prompt

   Command : **DTEXT**
3. **Follow** the steps 3-8 from above.

Text Justification 11.2

1. **Type** JUSTIFYTEXT at the command prompt

   Command: **JUSTIFYTEXT**

   or

2. **Pick** the Justify Text icon from the Text Toolbar. 

Text Justifications

A Aligns text between two designated endpoints (height and angle are not requested in this case).

C Centers the text around a specified point.

F Aligns the text between two designated endpoints with a specified height that varies only in its X scale factor.

M Centers the text both horizontally and vertically around a specified point.

R Right justifies the text at a designated endpoint.

S Selects a different text style.

TL Starts the top left portion of text at a given point.

TC Centers the top center of the text at a given point.

TR Ends the top of text at a given point.

ML Starts the middle left portion of the text at a given point.

MC Centers the middle of text at a given point.

MR Ends the text at the middle right portion at a given point.

BL Starts the bottom left portion of the text at a given point.

BC Centers the bottom center portion of the text at a given point.

BR Ends the bottom of text at a given point.

Text Styles 11.3

Style Command

1. **Choose** Format, Text Style...

   or

2. **Type** STYLE at the command prompt.

   Command: **STYLE**

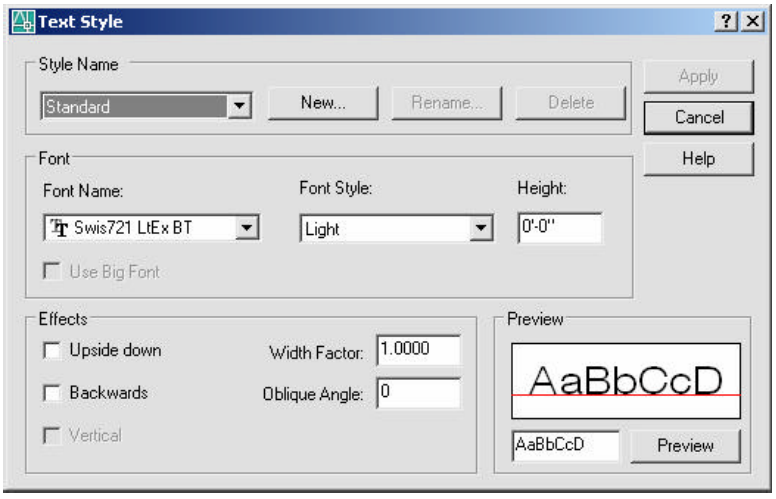
3. **Pick** the Text Style icon from the Text Toolbar.
4. **Choose** a style from the menu or create a NEW style.
5. **Choose** a font file.
6. **Type** a height for the text (set to zero to vary heights)
7. **Type** a width factor for each character.

   Width factor <1>: (**enter**)

8. **Type** an obliquing (slant) angle.

   Obliquing angle <0>: (**angle or enter**)

9. **Type** Yes or No to place characters backwards.
   Backwards? (**Y** or **N**)
10. **Type** Yes or No to draw characters upside down.
    Upside down? (**Y** or **N**)
11. **Type** Yes or No to draw characters vertically



Font Files

AutoCAD supports the following font types:

.SHX	AutoCAD Fonts
.PFB	Adobe Type I Fonts
.PFA	
.TTF	Windows True Type Fonts

TIP:

To replace the font globally in a drawing, type style at the command prompt and keep the same style name but replace the font file with the new font. When AutoCAD regenerates, it will replace all text drawn with that style with the new font.

Multiline Text 11.4

Mtext Command

1. **Choose** Draw, Text, Multiline Text...

   or

2. **Pick** the Mtext icon. 

   or

3. **Type** MTEXT at the command prompt.

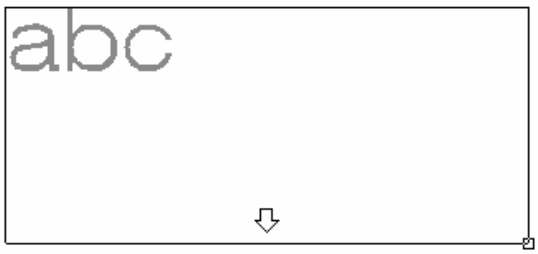
   Command: **MTEXT**

4. **Type** One of the following options

   Height/Justify/Rotation/Style/Width:

   or

5. **Pick** 2Points to define the text window.



6. **Type** text or change an MTEXT setting.

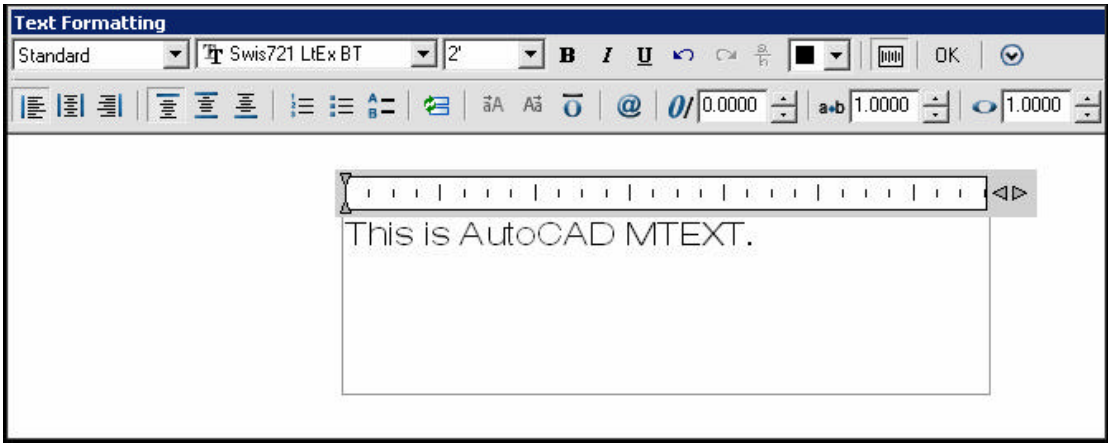
MTEXT options:

Rotation Controls the rotation angle of the text boundary.
Style Specifies the text style to use in paragraph text.
Height Specifies the height of uppercase text

Direction Specifies whether text is vertical or horizontal.

Width Specifies the width of the text boundary.

MTEXT Editor



Editing Text 11.5

DDEDIT

1. **Choose** Modify, Text...

   or

2. **Click** the Edit Text icon from the Text toolbar. 

   or

3. **Type** DDEDIT at the command prompt.

   Command: **DDEDIT** or **ED**

4. **Pick** The text to edit.

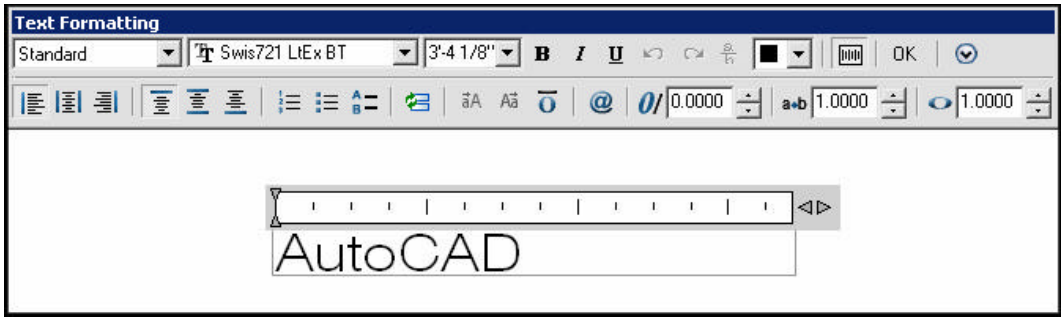
   Select objects: (pick text)

5. **Pick** Additional text or ENTER to end the command.

   Select objects: ENTER

Text Edit Dialog Box for TEXT and DTEXT Commands

Text Edit for MTEXT command



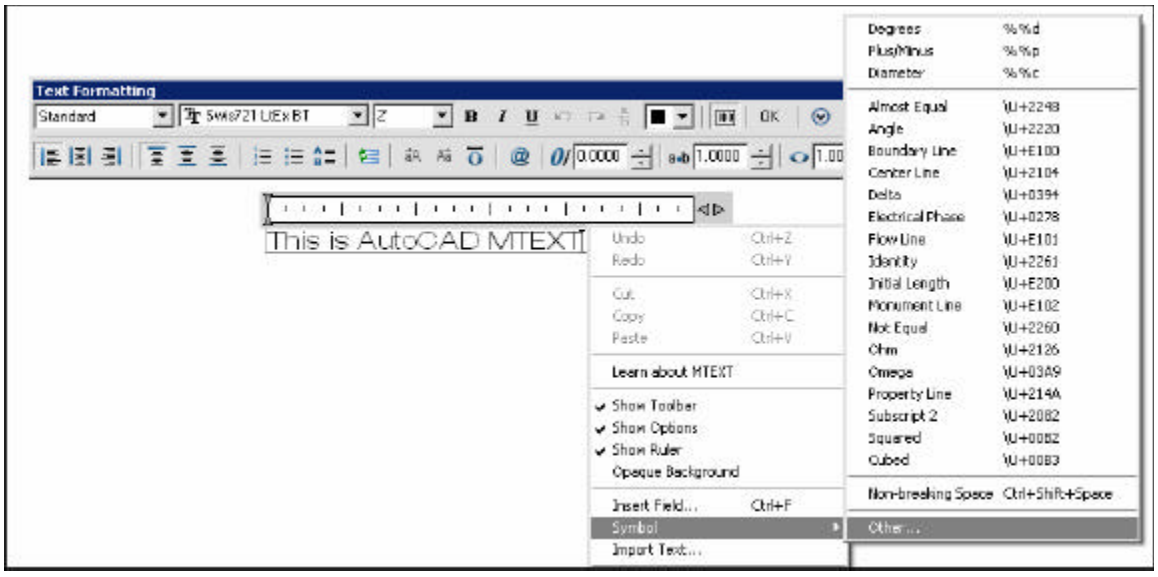
Special Control Codes 11.6

AutoCAD provides special control codes to return drafting symbols when using text.

1. **Type** The following characters to return equivalent symbol:

%%d	degree symbol (°)
%%c	diameter symbol (Ø)
%%p	plus minus symbol (±)
%%u	to start and stop underlining (NOTE)
%%o	to start and stop overscoring (NOTE)

The MTEXT command has additional symbols that can be accessed by right-clicking in the MText Editor for more Special Symbols.



Spell Check 11.7

1. **Choose** Tools, Spelling

   or

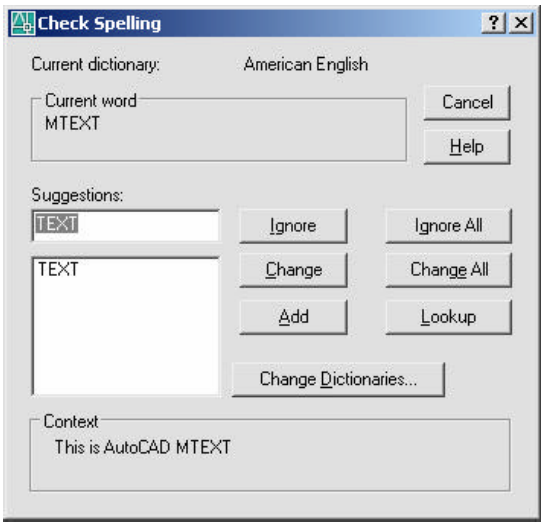
2. **Type** SPELL at the command prompt.

   Command: **SPELL**

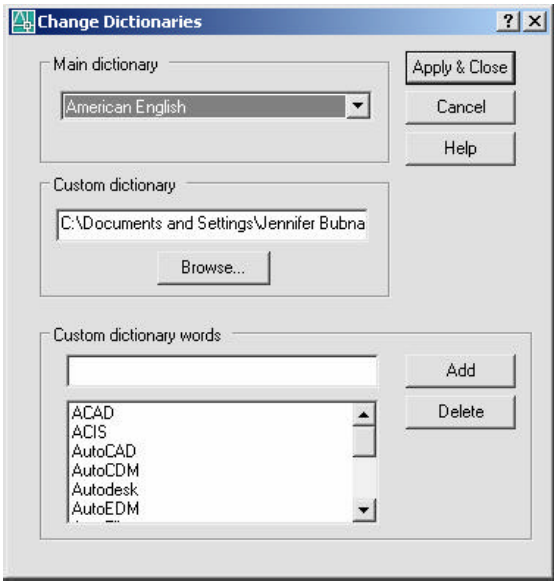
3. **Pick** The text to spell check.

   Select objects: (**pick text**)

4. **Choose** Change or Ignore to modify or accept the spelling of a word.



5. **Pick** Change Dictionaries to create your own dictionary.



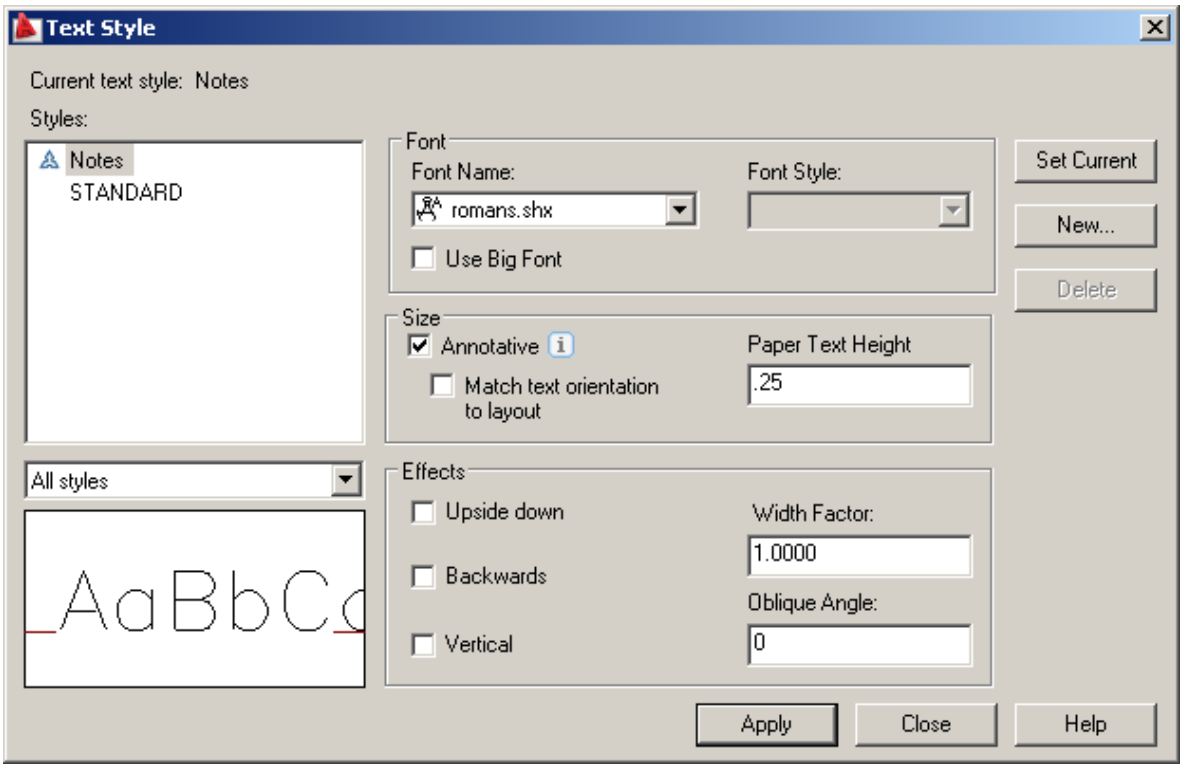
Annotative Text

Use annotative text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

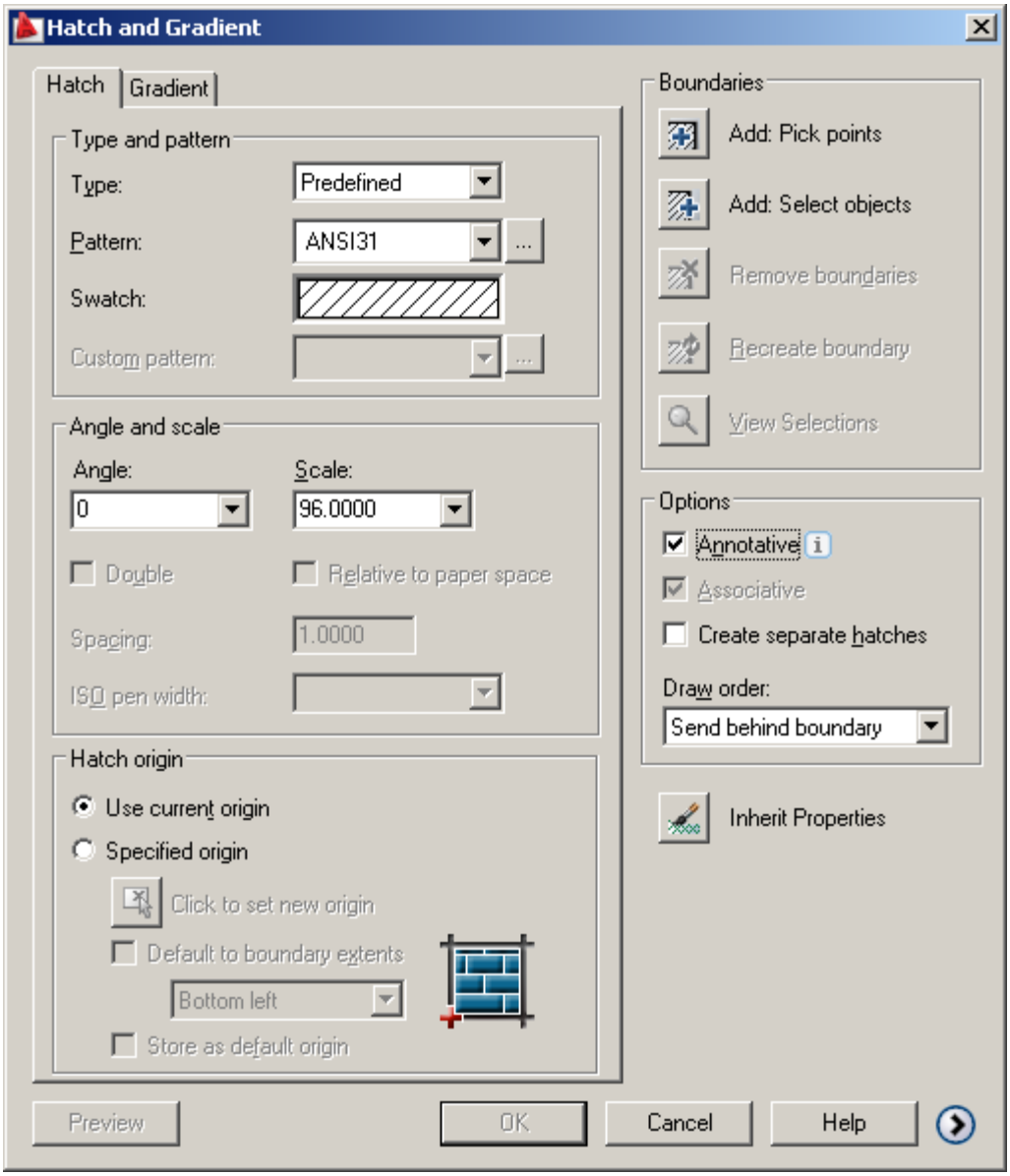
The current annotation scale automatically determines the display size of the text in model space or paper space viewports.

For example, you want text to display at a height of 3/16" on the paper, so you can define a text style to have a paper height of 3/16". When you add text to a viewport that has a scale of 1/2"=1'0", the current annotation scale, which is set to the same scale as the viewport's, automatically scales the text to display appropriately at 4.5".

1. Create an Annotative text style using the **STYLE** command.

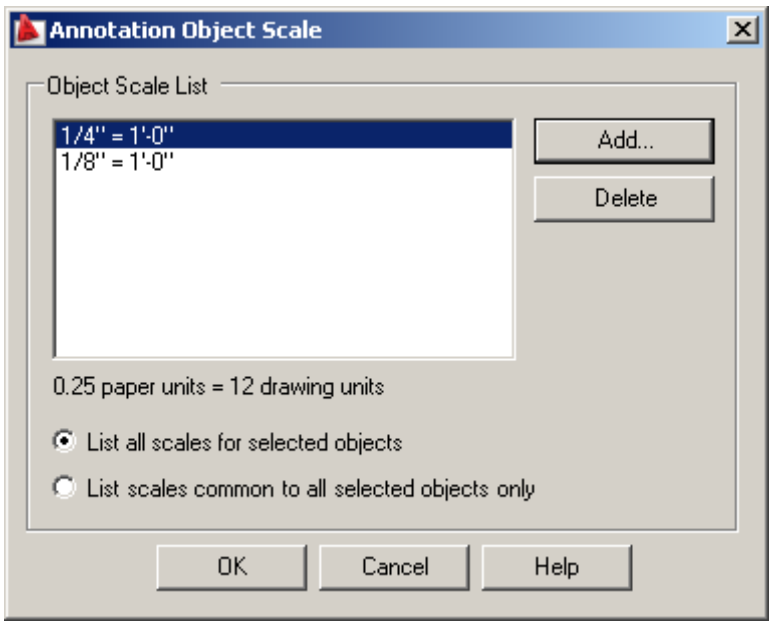


Annotative Hatch

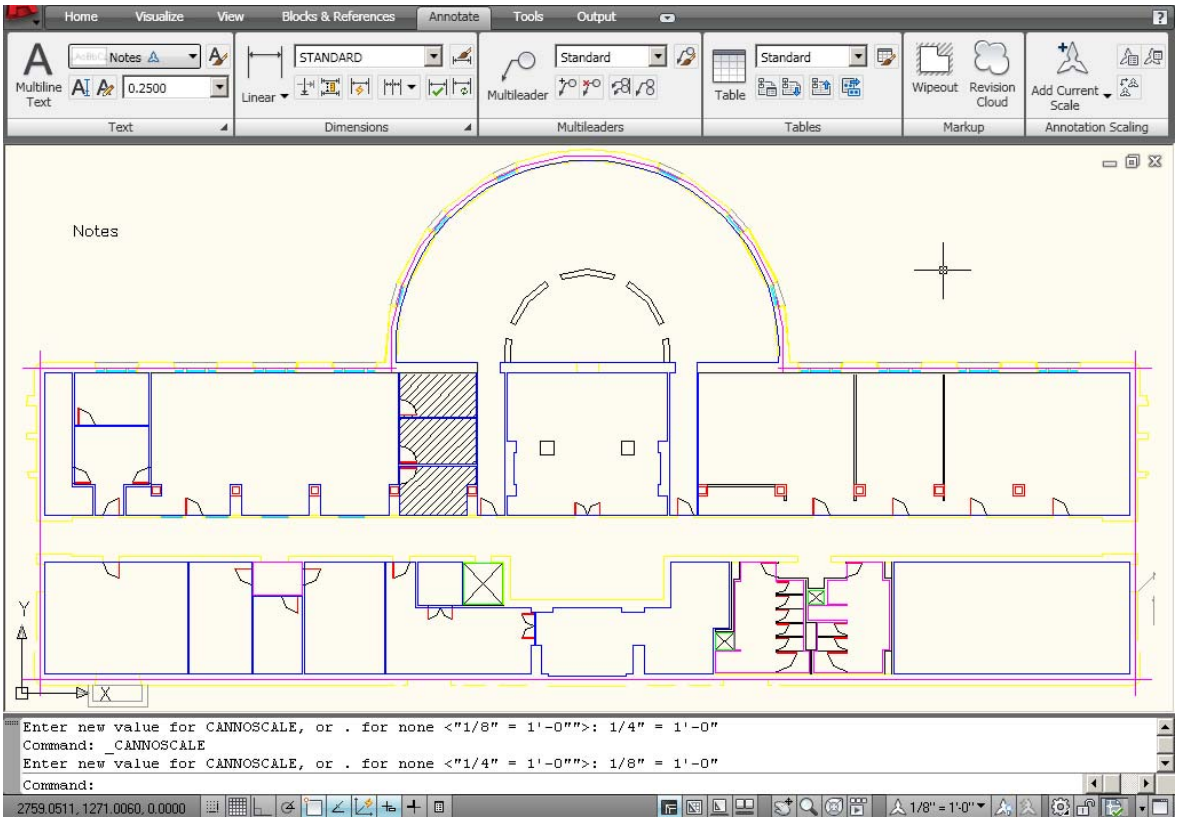


Scales

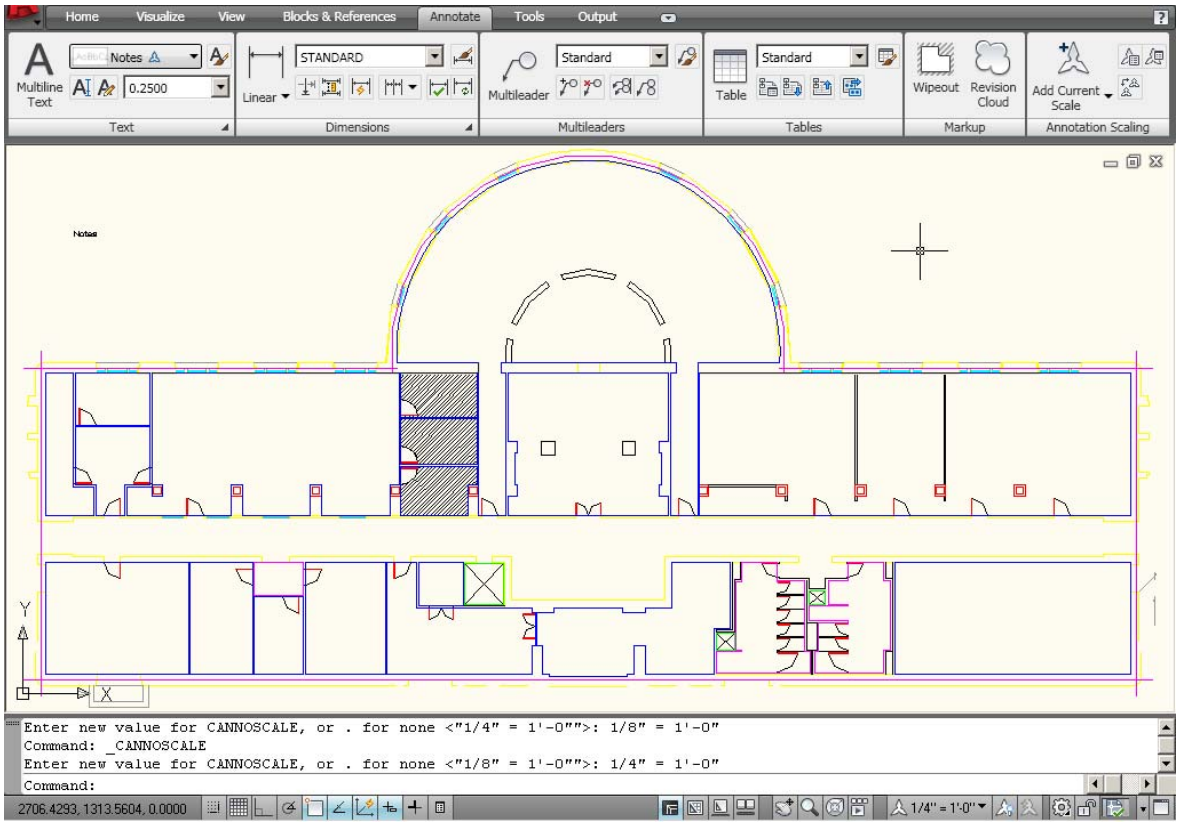
Add Scale



Change Scales



AutoCAD 2D Tutorial



Chapter 12
Layers, Linetypes, Colors

Introduction to Layers and Layer Dialog Box12.1

1. **Choose** Format, Layer.

or

2. **Type** LAYER at the command prompt.

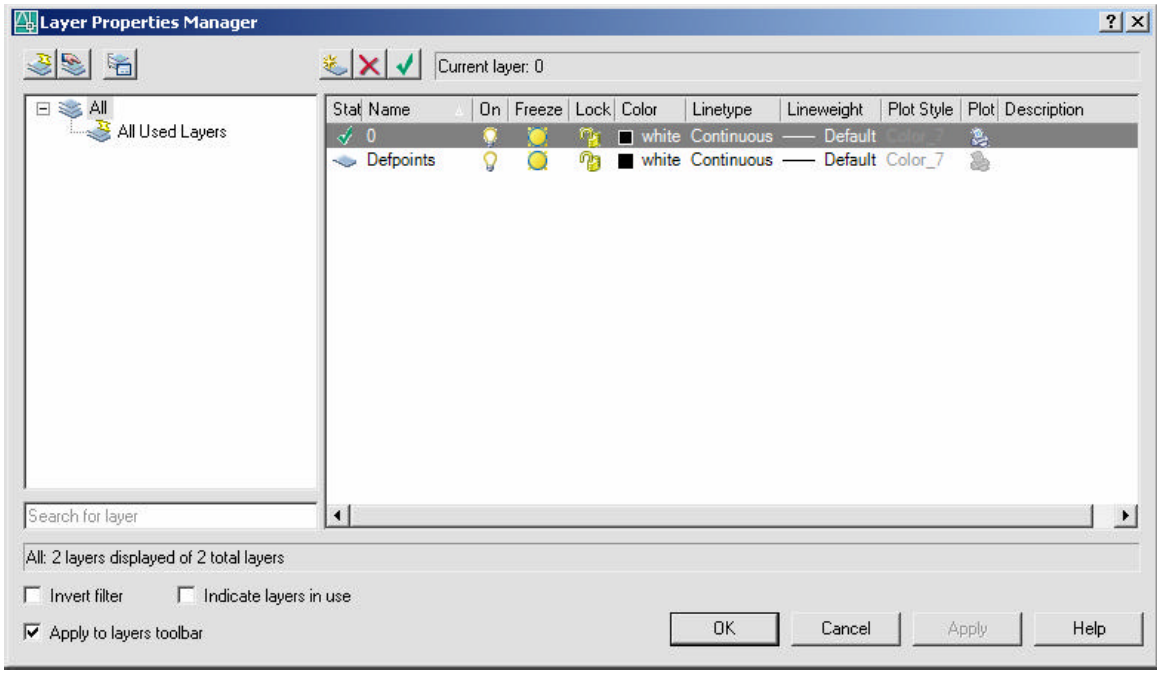
Command: **LAYER (or LA)**

or

3. **Pick** the layers icon from the Layer Control box on the object properties toolbar.

AutoCAD 2005

Layer Properties



Layer Options 12.2

?	Lists layers, with states, colors and linetypes.
Make	Creates a new layer and makes it current.
Set	Sets current layer.
New	Creates new layers .
ON	Turns on specified layers.
OFF	Turns off specified layers.
Color	Assigns color to specified layers.
Ltype	Assigns linetype to specified layers.
Freeze	Completely ignores layers during regeneration.
Thaw	Unfreezes specified layers Ltype.
Lock	Makes a layer read only preventing entities from being edited but available visual reference and osnap functions.
Unlock	Places a layer in read write mode and available for edits.
Plot	Turns a Layer On for Plotting
No Plot	Turns a Layer Off for Plotting
LWeight	Controls the line weight for each layer

TIP:

Layers can be set using the command line prompts for layers. To use this, type –LAYER or -LA at the command prompt

1. **Type** Command: -**LAYER** or **LA**
2. **Type** One of the following layer options

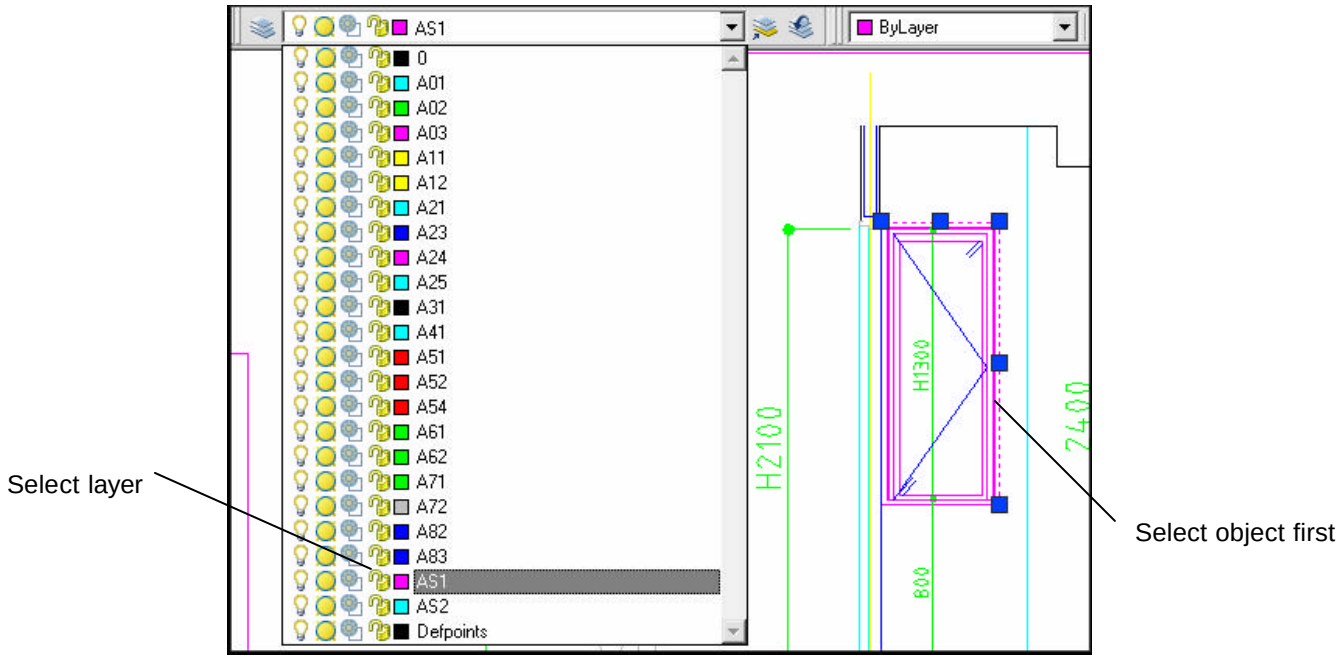
   ?/Make/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw:

Layer Shortcuts 12.3

Changing the Layer of an Object

1. **Click** Once on the object to change.
2. **Select** the desired layer from the Layer Control Box dropdown.

   AutoCAD will move the object to the new layer.



Making a Layer Current

1. **Click** once on the Make Object's Layer Current icon. 
2. **Select** object whose layer will become current:

Match Properties

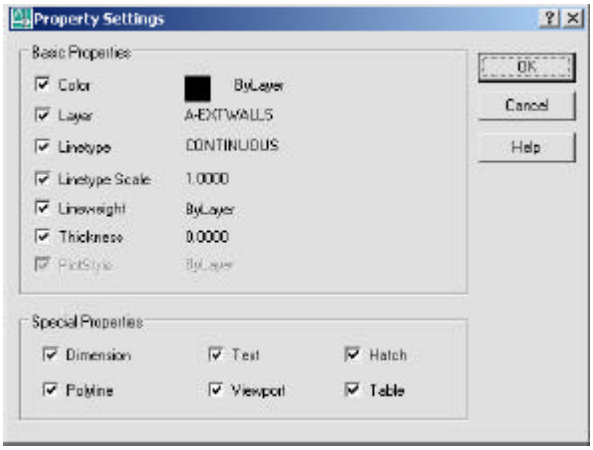
1. **Choose** Modify, Match Properties.

or

2. **Click** the Match Properties Icon from the Standard toolbar.

or

3. **Type** Command : **MATCHPROP or MA**
4. **Select** the object whose properties you want to copy (1).
5. **Select** the objects to which you want to apply the properties (2).



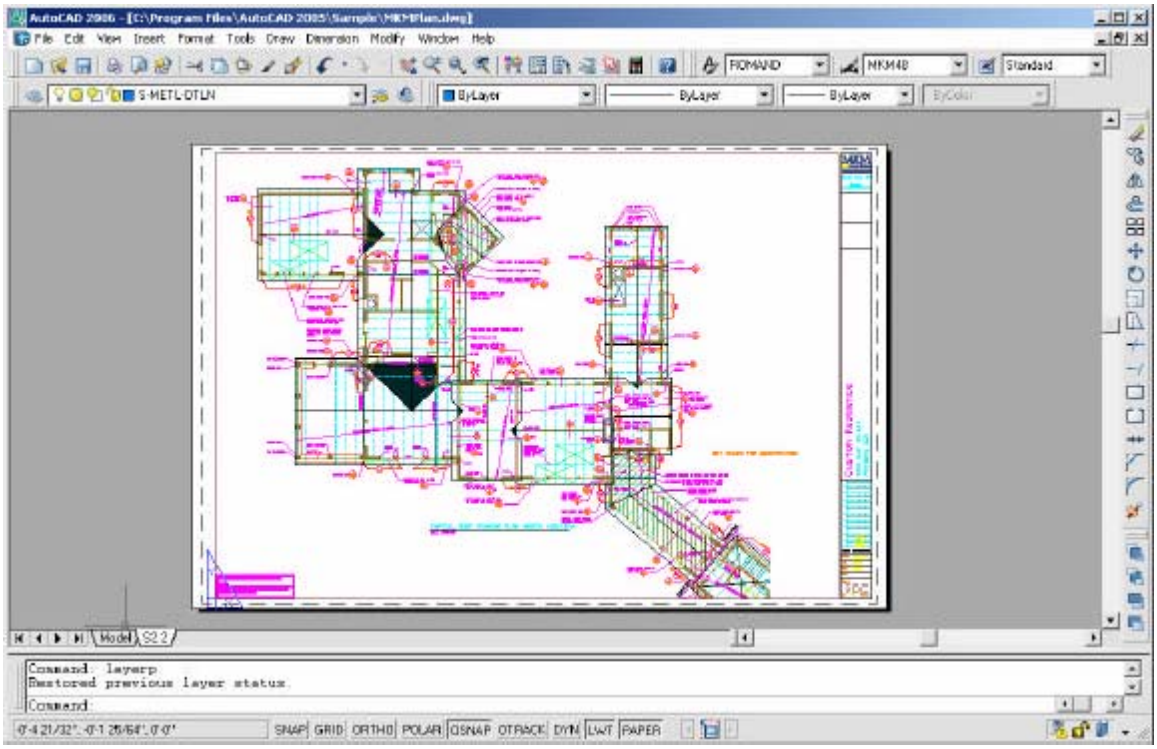
Layer Previous 12.4

1. **Open** an AutoCAD drawing with layers.
2. **Turn** layers on/off.
3. **Zoom** or perform any AutoCAD Command.
4. **Type** LAYERP at the command prompt.

   Command: LAYERP

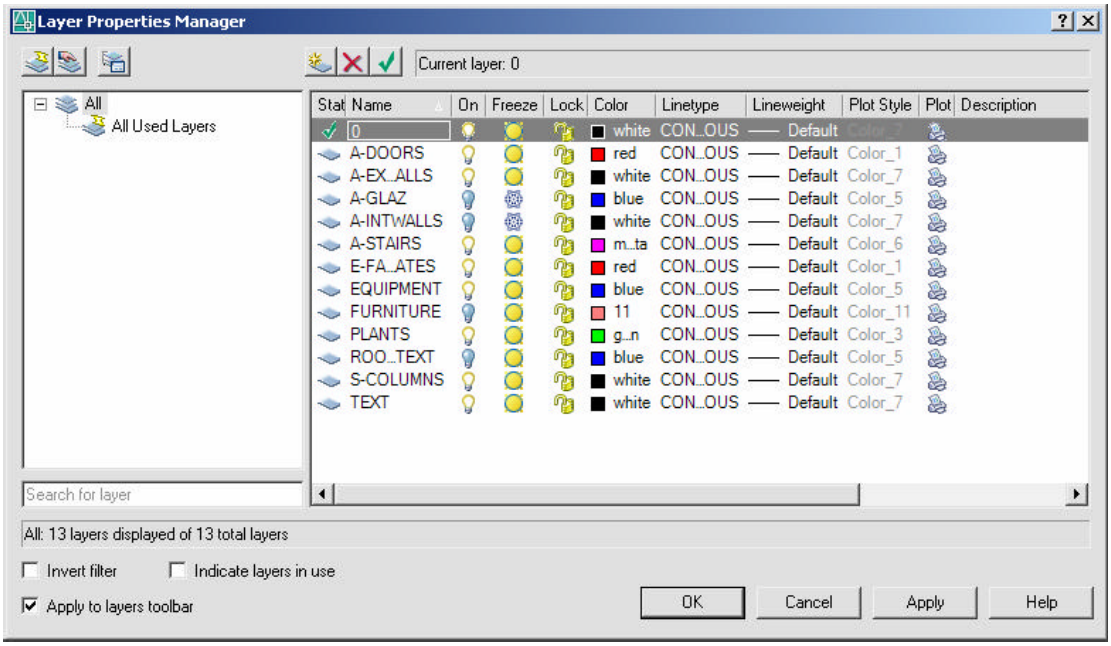
   or

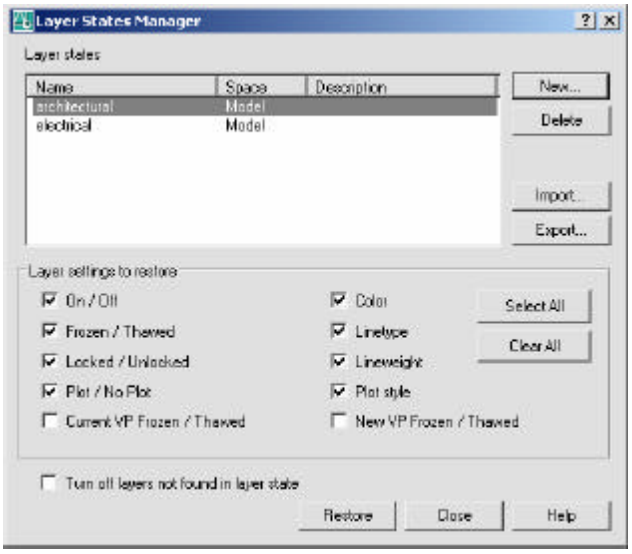
5. **Click** the Layer Previous icon. 



Layer States 12.5

1. **Choose** the layer icon.
2. **Select** various layers to be ON,OFF,FROZEN,LOCKED, etc.
3. **Choose** the Save State button.
4. **Choose** Restore State to restore the layer settings.





Color Command 12.6

1. **Choose** Format, Color.

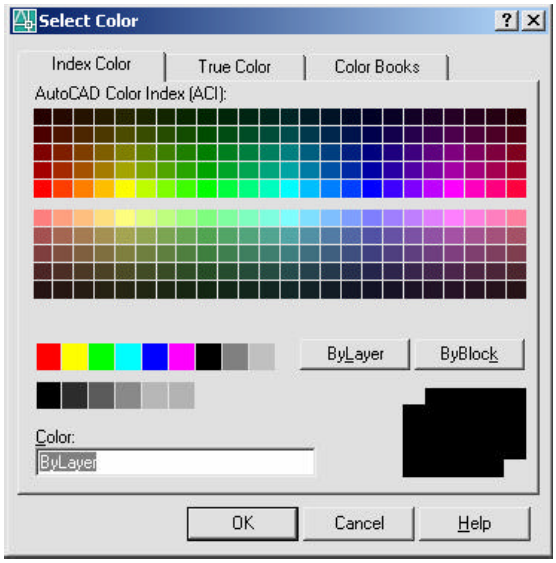
or

2. **Type** DDCOLOR at the command prompt.

   Command: **DDCOLOR or COL**

or

3. **Choose** Color on the Object Properties toolbar and then select a color from the list or select Other to display the Select Color dialog box.



TIP:

These settings ignore the current layer settings for color.

By Layer

If you enter bylayer, new objects assume the color of the layer upon which they are drawn.

By Block

If you enter byblock, AutoCAD draws new objects in the default color (white or black, depending on your configuration) until they are grouped into a block. When the block is inserted in the drawing, the objects in the block inherit the current setting of the COLOR command.

Linetypes 12.7

Loading and Changing Linetypes

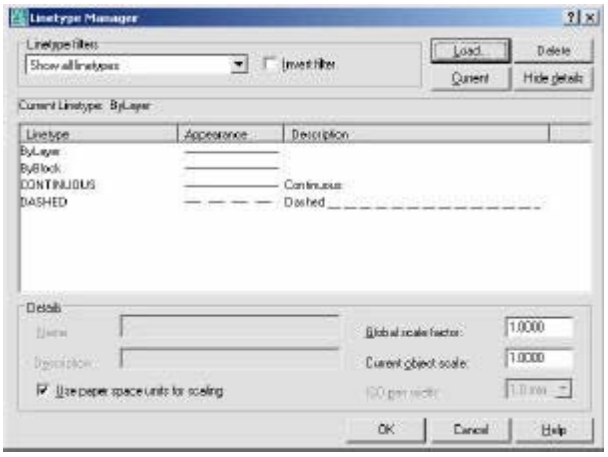
1. **Choose** Format, Linetype...

   or

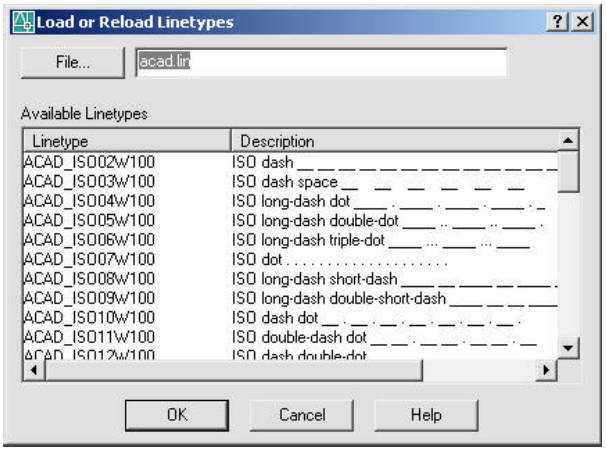
2. **Type** DDLTYPE at the command prompt.

   Command:**DDLTYPE or LT**

3. **Choose** Load... to see a list of available linetypes.



4. **Choose** the desired linetype to assign.



5. **Click** OK.

Lineweights 12.8

Loading and Changing Lineweights

1. **Choose** Format, Lineweight...

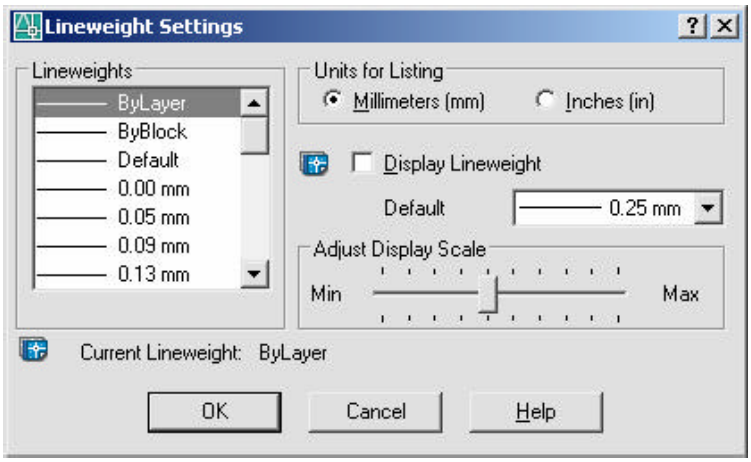
or

2. **Type** LINEWEIGHT at the command prompt.

Command: **LINEWEIGHT or LWEIGHT**

or

4. **Pick** a lineweight to make current from the Object Properties menu.



TIPS:

- Lineweights can also be assigned to layers.
- The Display Lineweights feature can be turned on/off on the status bar to show or not show lineweights in the drawing, thus making regenerations faster.

- 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.

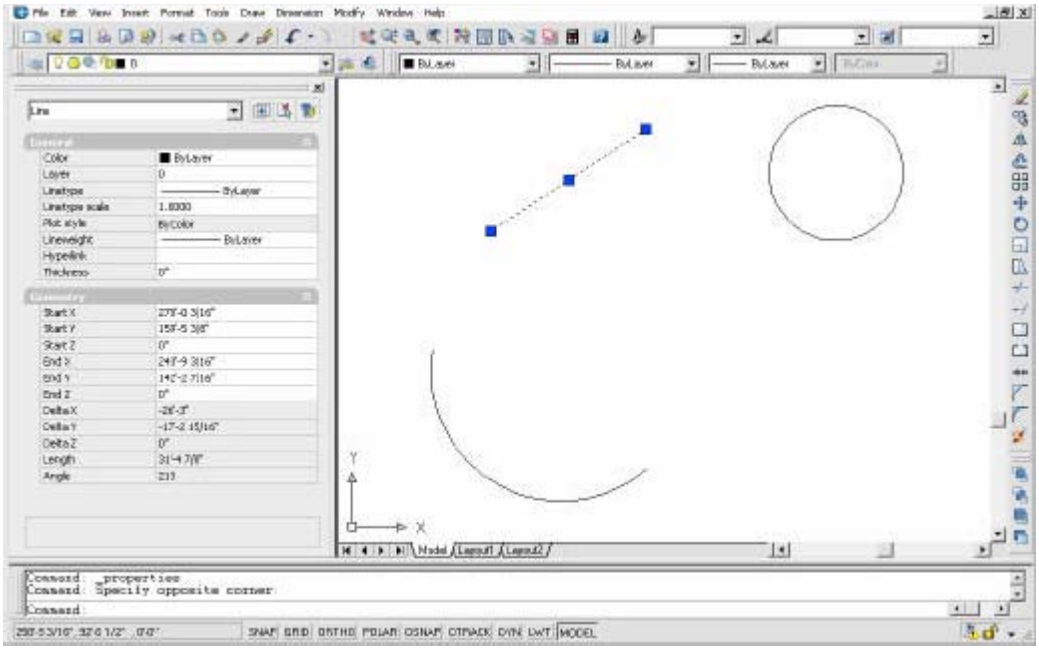
Object Properties 12.9

1. **Choose** Modify, Properties.
   or
2. **Click** the Properties icon. 
   or
3. **Type** DDCHPROP or DDMODIFY at the command prompt.

   Command: **DDCHPROP** (CH) or
   DDMODIFY (MO)
4. **Pick** Objects whose properties you want to change
   Pick a window for DDCHPROP, single object for DDMODIFY.

   Select objects:(select)
5. **Press** ENTER to accept objects.

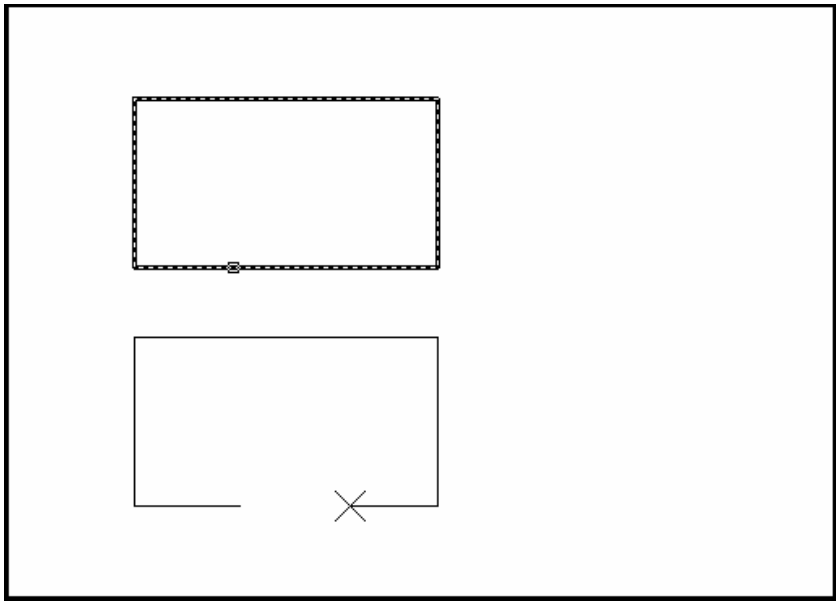
   Select objects: (press enter)
6. **Choose** One of the following properties to change.



Chapter 13
More Edit Commands

Break 13.1

1. **Choose** Modify, Break.
   or
2. **Click** the Break icon. 
   or
3. **Type** BREAK at the command prompt. Command: **BREAK**
4. **Pick** Object to break.
   Select object: (**select one object**)
5. **Pick** A second break point.
   Enter second point : (**point**)



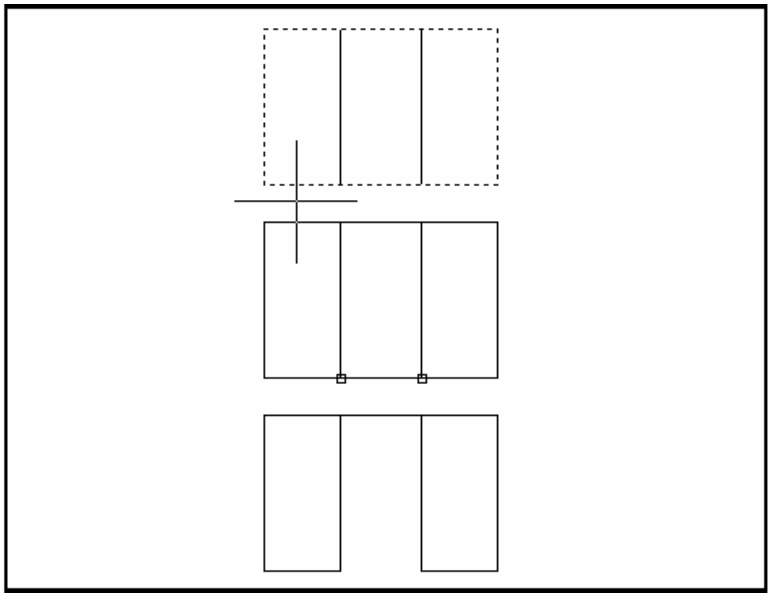
or

6. **Type** **F** to choose a different break point
   Enter second point (or F for first point):(**F**)

7. **Pick** The first break point on the object

   Enter first point: (**point**)

8. **Pick** A second break point



TIP:
You can also type coordinates instead of picking a break point. Enter second point (or F for first point): **@3'<0**

If you break a circle, it changes to an arc by deleting the portion from the first point to the second, going counterclockwise.

Breaking a Polyline with nonzero width will cause the ends to be cut square.

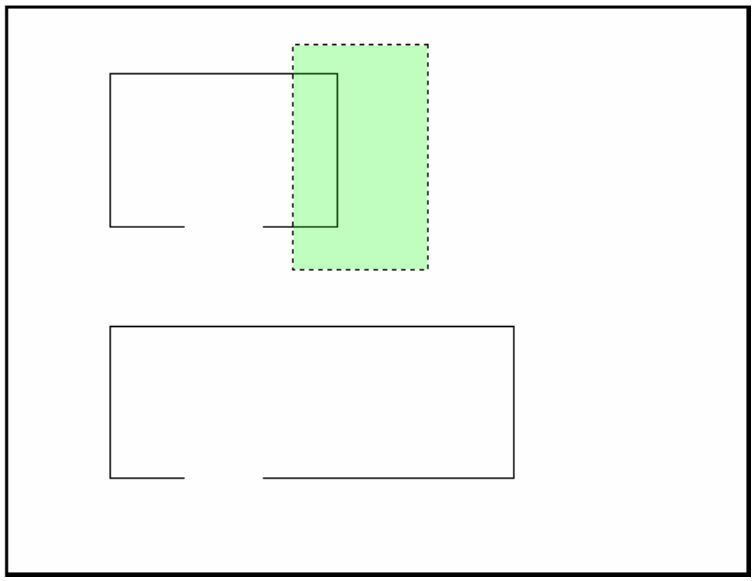
Stretch 13.2

1. **Choose** Modify, Stretch.
   or
2. **Click** the Stretch icon. 
3. **Type** STRETCH at the command prompt.

   Command : **STRETCH** Select objects to stretch by window...
4. **Type** C to choose CROSSING window

   Select objects: **C**
5. **Pick** A first corner to stretch. First corner: (**point**)
6. **Pick** The opposite corner to window the objects to stretch.

   Other corner: (**point**)



7. **Press** ENTER to accept objects to stretch.
8. **Pick** A base point to stretch from Base point:

   (**point**)

9. **Pick** A point to stretch to Newpoint: (**point**)

or

10. **Type** A distance to stretch. New point: @**1<0**

TIP:

The Stretch command must use a CROSSING window or a CROSSING POLYGON window.

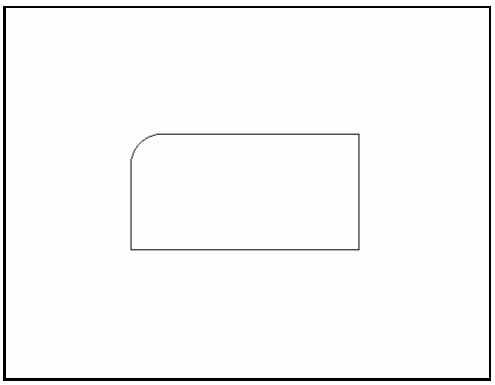
Fillet 13.3

1. **Choose** Modify, Fillet.
   or
2. **Click** the Fillet icon. 
   or
3. **Type** FILLET at the command prompt. Command: **FILLET**
4. **Pick** First object to fillet. Polyline/Radius/Trim<Select two objects>: select first object.
5. **Pick** Second object to fillet.
   Select second object: select second object.
   or
6. **Type** One of the following options:

P Fillets an entire Polyline.

R Sets the fillet radius.

T Sets the trim mode (trim cuts the fillet corner and no trim keeps the fillet corner).



TIP:

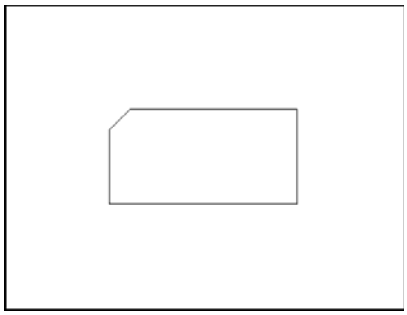
- You can also fillet PARALLEL lines as well as PLINES with LINES
- Type a radius of zero (0) to create a clean 90 degree corner.

Chamfer 13.4

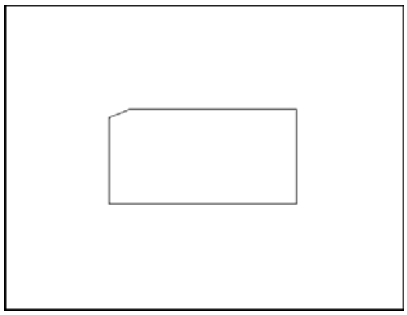
1. **Choose** Modify, Chamfer.
   or
2. **Click** the Chamfer icon. 
   or
3. **Type** CHAMFER at the command prompt.
   Command: **CHAMFER**
4. **Pick** First object to chamfer.
   Polyline/Distance/Angle/Trim/Method<Select first line>: **select first object**
5. **Pick** Second object to chamfer.
   Select second object: select second object.
   or
6. **Type** One of the following options:

P	Chamfers entire Polyline.
D	Sets chamfer distances.
A	Uses a distance and angle method instead of two distances.
T	Sets the trimmode
M	Sets the method to distance or angle.

Chamfer with equal distances



Chamfer with different distances



Array 13.5

Rectangular Array

To draw rectangular array:

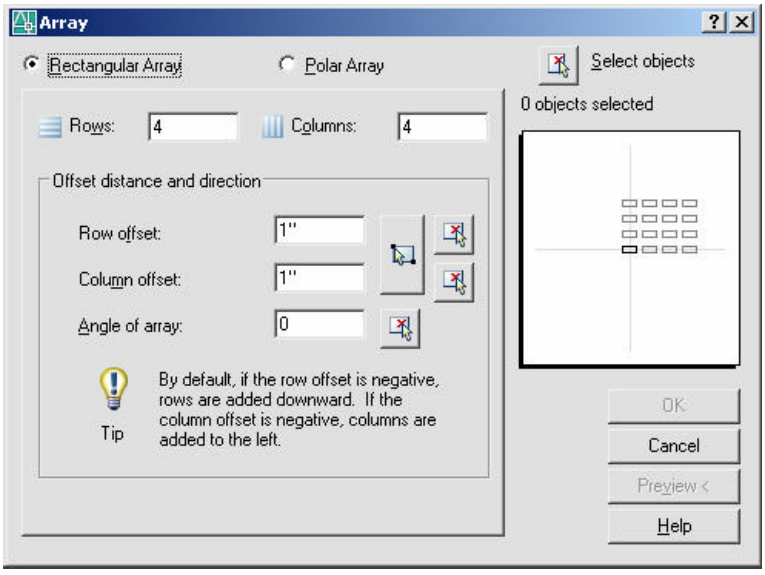
1. **Choose** Modify, Array.

or

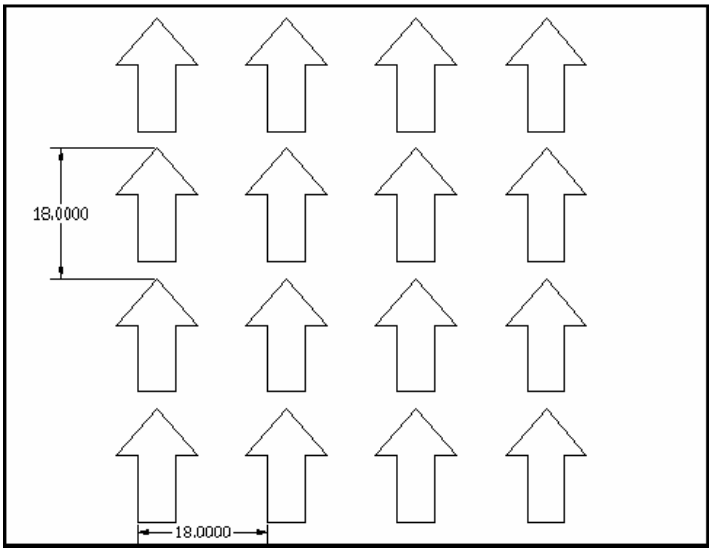
2. **Click** the Array icon.

or

3. **Type** *ARRAY at the command prompt.* Command : **ARRAY**
4. **Pick** Objects to array. Select objects : (**select**)
5. **Type** *The number of rows top to bottom.* Number of rows(---) <1>: (**number**)
6. **Type** The number of columns left to right. Number of columns (|||)<1>: (**number**)
7. **Type** The unit cell distance between items in each row. *Distance between rows:* (**+ number=up, -number =down**)
8. **Type** The unit cell distance between items in each column. Distance between columns:(**+number=right, - number =left**)



Unit Cell Distance Between Rows



Unit Cell Distance Between Columns

Polar Array

To draw a polar array:

1. **Choose** Modify, ARRAY.

   or

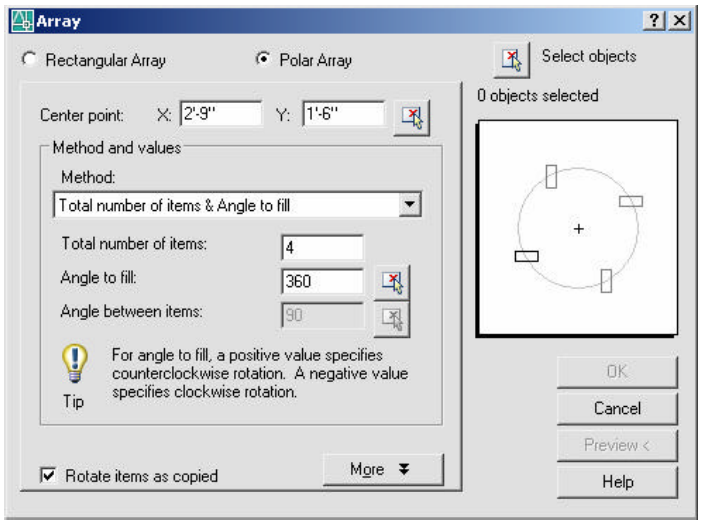
2. **Click** the Array icon. 

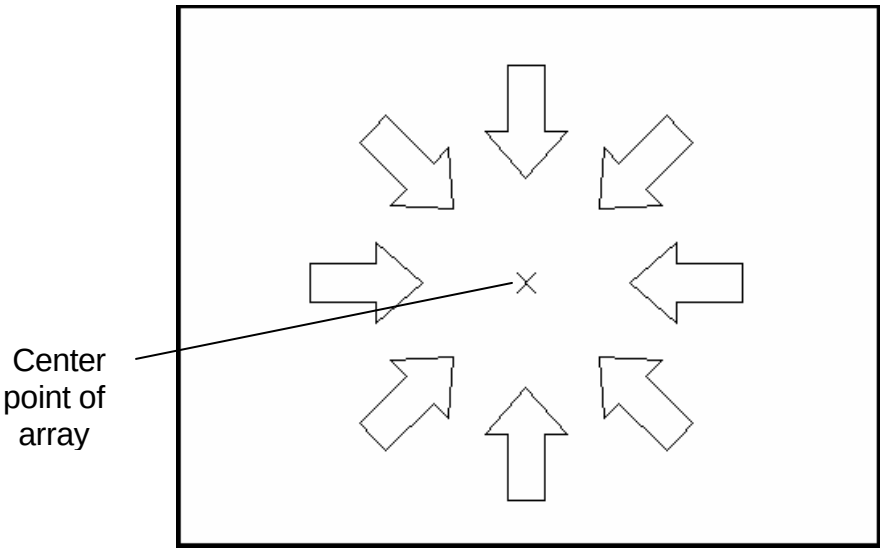
   or

3. **Type** *ARRAY at the command prompt.* Command: **ARRAY**
4. **Pick** Objects to array. Select objects:(**select**)
5. **Type** P to draw a polar array. Rectangular or Polar array (R/P): **P**
6. **Pick** A center point for the array. Center point of array: **pick point**
7. **Type** The TOTAL number of items in the array. *Number of items*: **number**
8. **Type** The number of degrees to rotate the objects. Degrees to fill (+=CCW, -+CW)<360>: **number**

9. **Type** Yes or No to rotate objects.

Rotate objects as they are copied?<y> **Y** or **N**





Lengthen 13.6

1. **Choose** Modify, LENGTHEN.

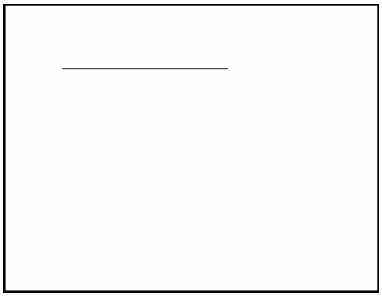
o
r

2. **Type** LENGTHEN at the command prompt. Command: **_lengthen**

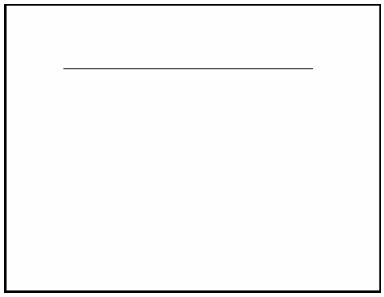
Select an object or [**DElta**/Percent/Total/
Enter delta length or [Angle] <0.0000>:**2**

Select an object to change or [Undo]: **pick object**

Object before lengthen



Object after lengthen



Chapter 14
Advanced Display Commands

Transparent Commands 14.1

Transparent commands are those started while another is in progress. Precede transparent commands with an apostrophe.

1. **Type** LINE at the command prompt.
   Command: **LINE**

   Specify first point: **(pick point)**

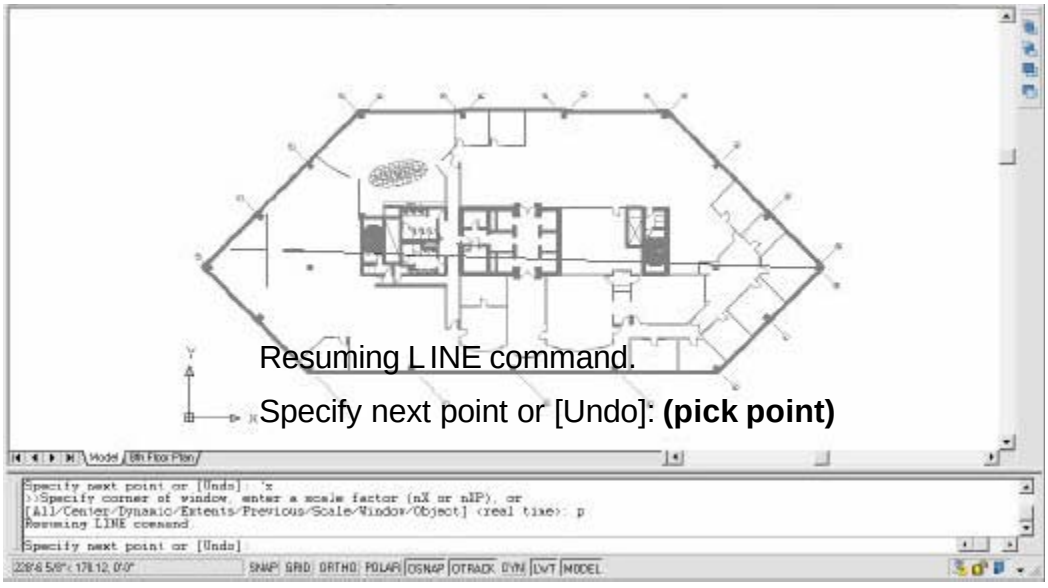
   Specify next point or [Undo]: **'zoom**

   >>Specify corner of window, enter a scale factor

   (nX or nXP), or

   [All/Center/Dynamic/Extents/Previous/Scale/ Window]
   <real time>: **(pick corner)**

   >>>>Specify opposite corner: (**pick other corner)**



TIP:

Commands that do not select objects, create new objects, or end the drawing session usually can be used transparently.

Multiple Command 14.2

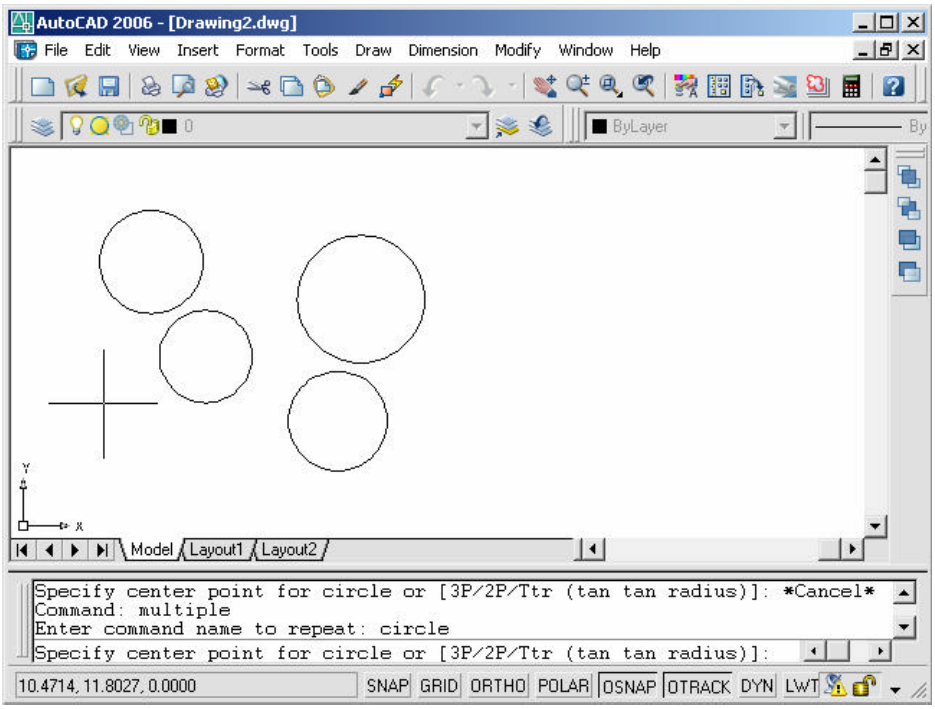
Multiple repeats the specified command until canceled

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

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

1. **Type** MULTIPLE before each command

   Command: **multiple circle**



Calculator (CAL Command) 14.3

Evaluates mathematicaland geometric expressions

1. **Type** CAL at the command prompt. Command: **cal**

   (or 'cal) Initializing...>>

   Expression: **1+1**

   2

Numeric operators

( ) Groups expressions

^ Indicates exponentiation

*** , /** Multiplies, divides

+, - Adds, subtracts

Vector operators

( ) Groups expressions

& Determines the vector product of vectors (as a vector)

[a,b,c]&[x,y,z] = [(b*z) - (c*y) , (c*x) - (a*z) , (a*y) - (b*x)]

* Determines the scalar product of vectors (as a real number)

[a,b,c]*[x,y,z] = ax + by + cz

, / Multiplies, divides a vector by a real number a

[x,y,z] = [a*x,a*y,a*z]

+ , - Adds, subtracts vectors (points)

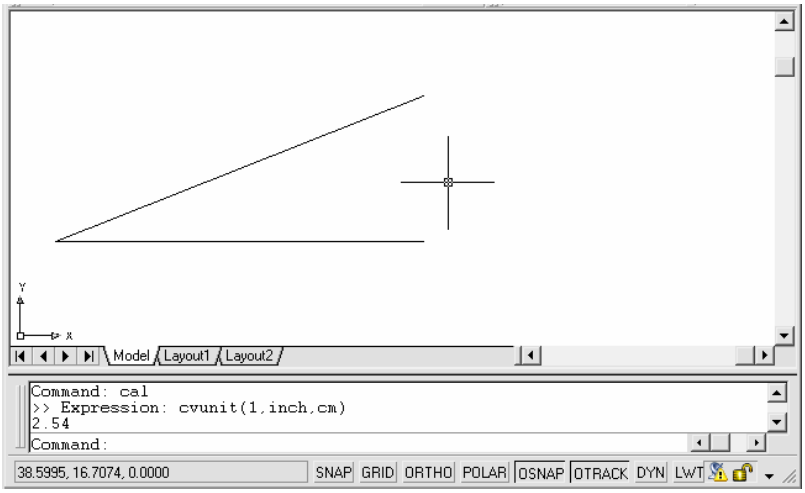
[a,b,c] + [x,y,z] = [a+x,b+y,c+z]

Converts units of measure

1. **Type** CAL at the command prompt.
Command: **cal (or 'cal)**

Initializing...>> Expression: **cvunit(1,inch,cm)**

2.54



Determines Angles

1. **Type** CAL at the command prompt.
Command: **cal (or 'cal)**

Initializing...>> Expression: **ang(end,end,end)**

45

Chapter 15
Polylines

Pline Command 15.1

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.

1. **Choose** Draw, Polyline.

or

2. **Pick** the Pline icon. 
3. **Type** PLINE at the command prompt

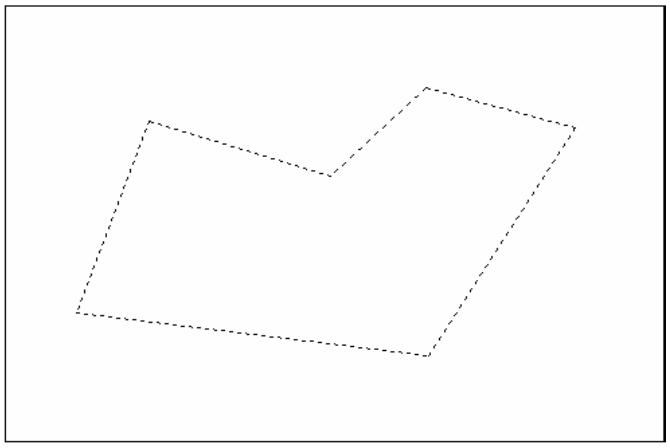
   Command : **PLINE** or **PL**
4. **Pick** A point on the drawing to start the polyline

   From point:(**select**)
5. **Type** One of the following options
   Arc/Close/Halfwidth/Length/Undo/Width/<endpoint of line>:

or

6. **Pick** A point to continue drawing
   Arc/Close/Halfwidth/Length/Undo/Width/<endpoint of line>: (**pick point**)

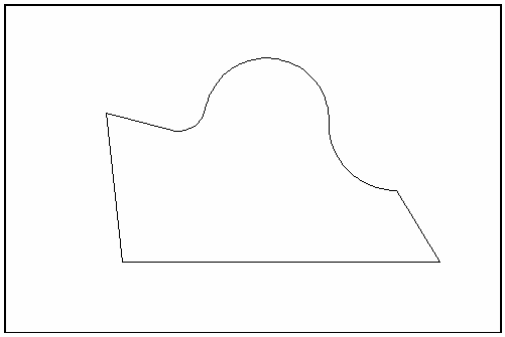
Polyline as one segment



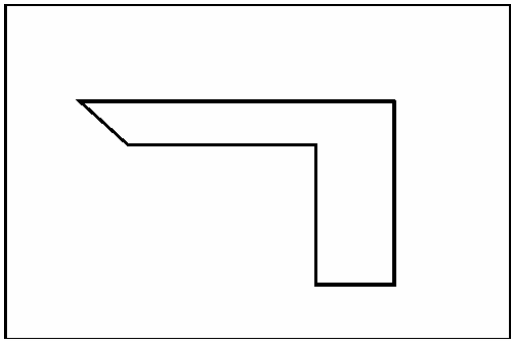
PLINE options:

Arc	Toggles to arc mode and you receive the following:Angle/CEnter/CLose/Direction/Halfwidth/ Line/ Radius /Second Pt/Undo/Width/<endpt of arc>:
Close	Closes a polyline as it does in the line command.
Halfwidth	Specifies the halfwidth of the next polyline segments. Can be tapered.
Length	Specifies the length to be added to the polyline in the current direction.
Undo	Undoes the previous pline segment as with the line command.
Width	Specifies the width of the next polyline segments. Can be tapered.

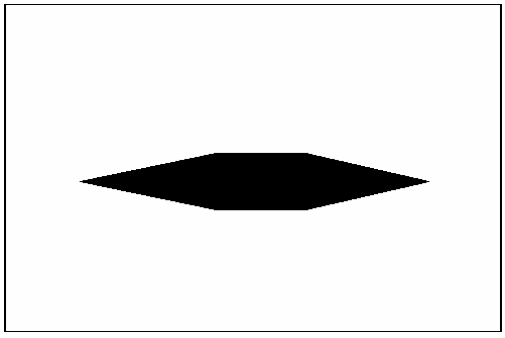
Polyline with arcs



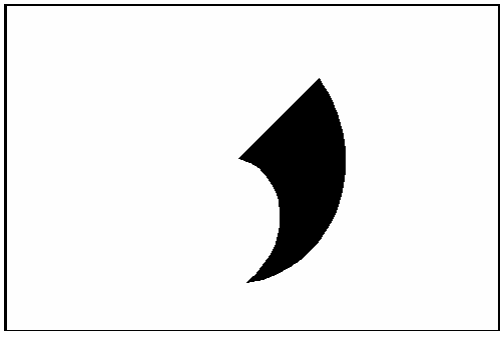
Polyline with width .125



Tapered width polyline



Tapered width arc polyline



Editing Polylines 15.2

1. **Choose** *Modify, Polyline.*

   or

2. **Pick** the Pedit icon from the Modify II toolbar. 
3. **Type** PEDIT at the command prompt

   Command: **PEDIT**
4. **Pick** Pick a polyline to edit

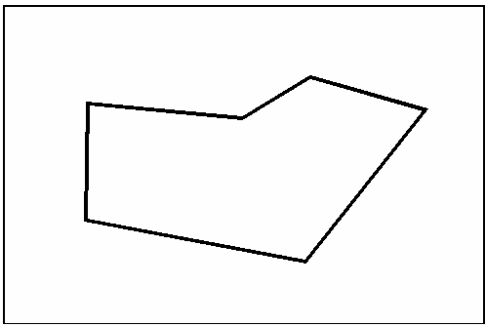
   Select Polyline:(**pick**)
5. **Type** One of the following options:Close/Join/ Width/Edit vertex/FitCurve/Spline/Curve/

   Decurve/Undo/eXit

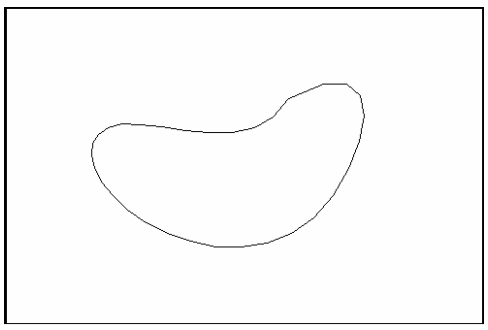
PEDIT options:

Close	Closes open polyline segments
Join	Connects polylines, lines, and arcs to existing polylines.
Width	Changes the width for all polyline segments.
Fit curve	Creates curved arc segments around pline vertices at the direction you specify.
Spline Curve	Creates a curve through control points on a polyline.
Decurve	Straightens curved segments.
Edit Vertex	Displays the following Edit Vertex Options:

Polyline width change



Splined Polyline



PLINEGEN

Sets how linetype patterns are generated around the vertices of a two-dimensional polyline. Does not apply to polylines with tapered segments.

0	Polylines are generated to start and end with a dash at each vertex
1	Generates the linetype in a continuous pattern around the vertices of the polyline.

Edit Vertex Options

1. **Type** One of the following vertex options: Next/Previous/Break/Insert/Move/Regen/Straighten/ Tangent/Width/eXit<N>:

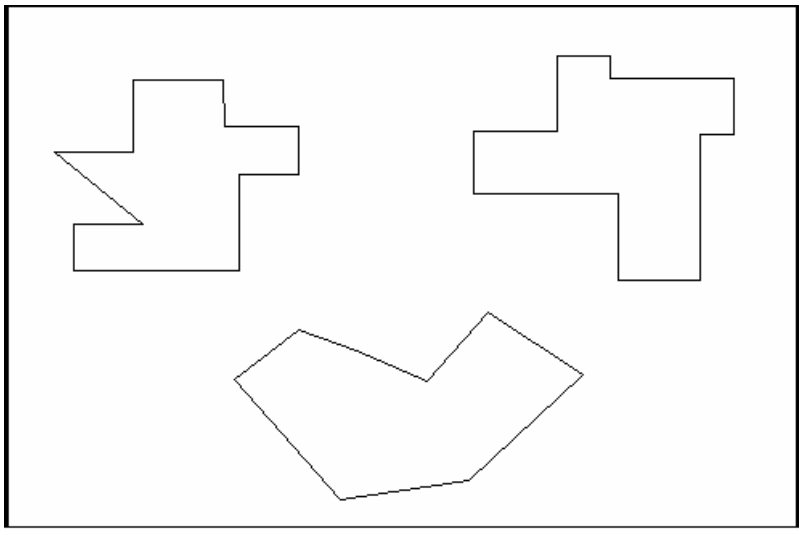
Next	Moves the X to the next vertex
Previous	Moves the X to the previous vertex
Break	Remembers the currently marked vertex and allows you to move to another vertex. You can then remove the segments between these vertices. Closed plines will open.
Insert	Adds a new vertex after the currently marked vertex.
Move	Moves the location of the currently marked vertex.
Regen	Regenerates the pline. Used with the width option.
Straighten	Remembers the currently marked vertex and allows you to move to another vertex. You can then replace the segments between these vertices with a straight one.
Tangent	Attaches a tangent direction to the current vertex for later use in curve fitting.
Width	Changes starting and ending widths for the segment following the marked vertex.
eXit	Exits from editing vertices.

Editing Multiple Polylines 15.3

1. **Type** the PEDIT at the command prompt. Command: **PEDIT**

   Select polyline or [Multiple]:**M**

   Pick multiple polylines to edit.



Explode Command 15.4

1. **Choose** *Modify,Explode.*

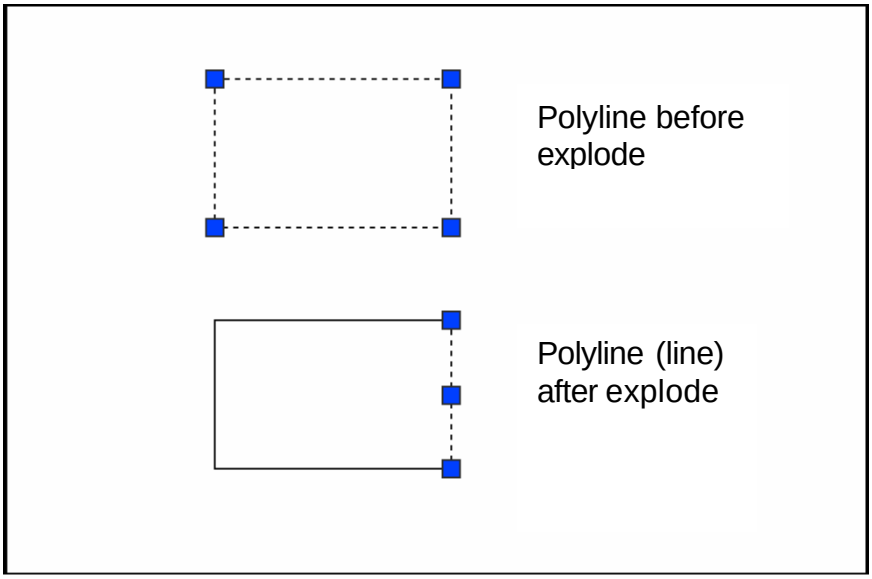
or

2. **Pick** the Explode icon. 
3. **Type** EXPLODE at the command prompt.

   Command: **EXPLODE**

or

4. **Pick** The object to explode. Select objects: (**pick**)



Turning Lines into Polylines 15.5

Use the PEDIT command to pick lines. AutoCAD will ask if you want to turn these lines into polylines. You can then use the JOIN option under PEDIT to join additional lines to the polyline.

1. Command: **pedit**

   Select polyline or [Multiple]: **pick line**

   Object selected is not a polyline

   Do you want to turn it into one? <Y>

   Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: j

TIP:

• Lines and Arcs must have a common endpoint to join them together.

Chapter 16
More Draw Commands

Polygon 16.1

1. **Choose** Draw, Polygon.

   or

2. **Click** the Polygonicon. 

   or

3. **Type** Polygon at the command prompt.

   Command: **POLYGON**

4. **Type** The number of sides for the polygon (3-1024)

   Number of sides <default>: **number**

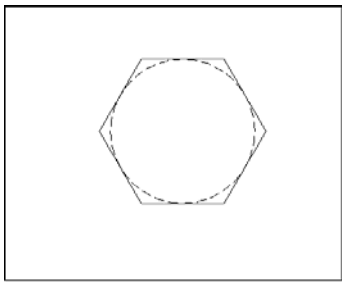
5. **Pick** The center of the polygon. Edge/<Center of polygon>: **pick**

   or

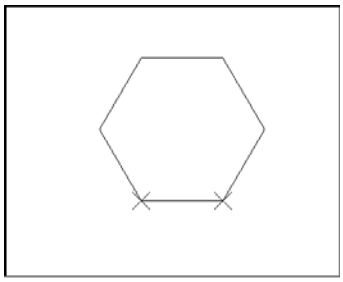
6. **Type** **E** to define the polygon by two edges.

7. **Type** **I** or **C** to place the polygon inside or outside of an imaginary circle. Inscribed in circle/Circumscribed about circle (I/C):

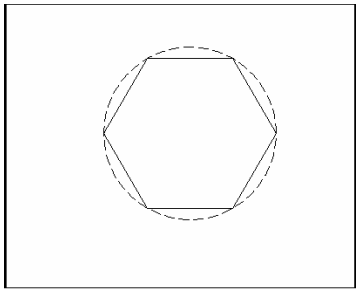
Polygon Inscribed in an imaginary circle



Polygon drawn with Edge



Polygon circumscribed around an imaginary circle



Rectangle 16.2

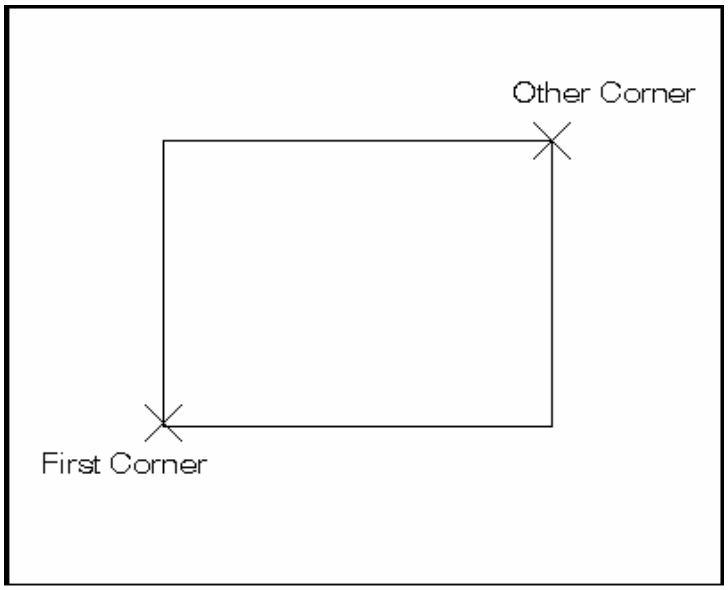
1. **Choose** Draw, Rectangle.

   or

2. **Click** the Rectangle icon. 

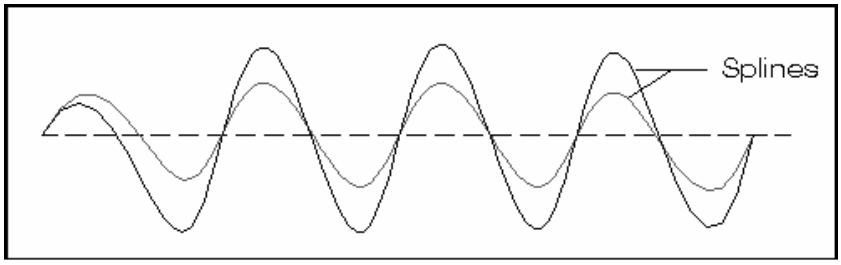
   or

3. **Type** Rectang at the command prompt Command:
   RECTANG Chamfer/Elevation/Fillet/Thickness/Width/
   <First corner>:
4. **Pick** first corner.
5. **Pick** other corner or type coordinates **(i.e. @4,2)**.

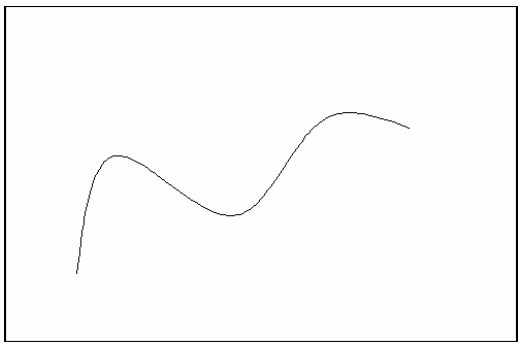


Spline 16.3

The SPLINE command creates a particular type of spline known as a nonuniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points



1.	**Choose**	Draw, Spline.
		or
2.	**Click**	the Spline icon. 
		or
3.	**Type**	SPLINE at the command prompt
		Command: **SPLINE**
4.	**Pick**	A start point for the spline
		Object / <Enter first point>: (**pick point**)
5.	**Pick**	Points until you are done drawing splines
		Enter point: (**pick points**)
6.	**Press**	Enter or close to complete the spline
7.	**Pick**	Starting tangent point for the spline
		Enter start tangent (**pick point**)
8.	**Pick**	Ending tangent point for the spline
		Enter end tangent: (**pick point**)



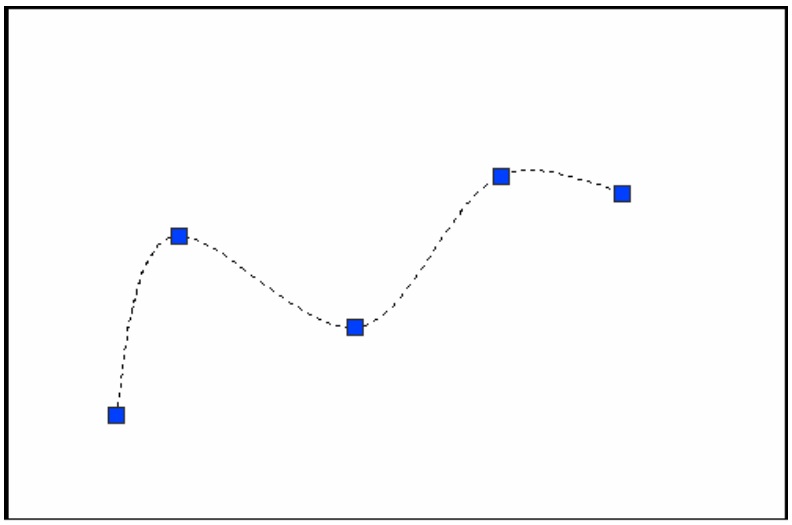
Spline options:

Object	Converts 2D or 3D spline-fit polylines to equivalent Splines
Points	Points that define the spline
Close	Closes a spline.
Fit Tolerance	Allows you to set a tolerance value that creates a smooth spline.

TIP: Refer to AutoCAD online help topic for more information on spline options.

Editing Splines 16.4

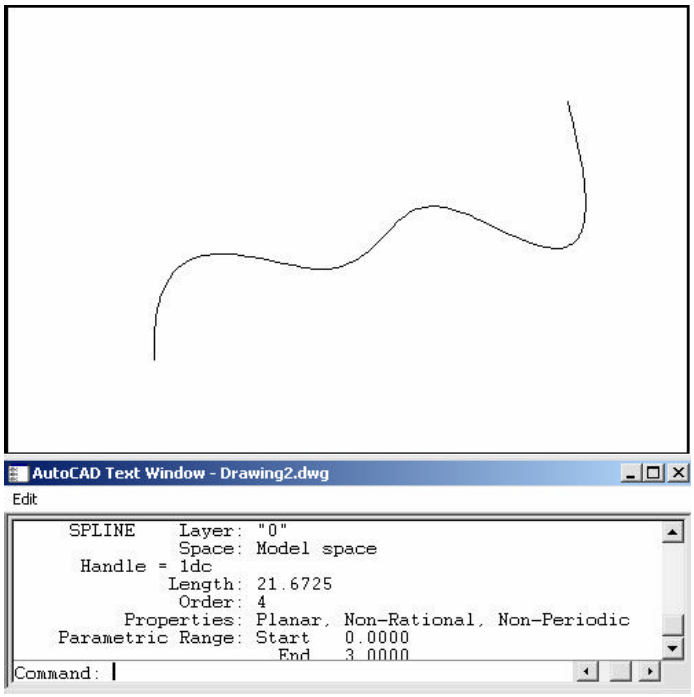
1. **Choose** Modify, Object, Spline.



TIP: Drawings containing splines use less memory and disk space than those containing spline-fit polylines of similar shape.

Covert PLINE to Spline 16.5

1. **Draw** a PLINE.
2. **Type** PEDIT to edit the polyline as a spline.
3. **Choose** Draw, Spline.
4. **Type** Object at the command prompt.
5. **Click** once on the polyline to turn it into a spline.



TIP: Use the LIST command to determine if an object is a PLINE or SPLINE.

Donut 16.6

Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

1. **Choose** Draw, Donut.

   or

2. **Type** Donut at the command prompt.

   Command: **DONUT**

3. **Type** A value for the inside diameter.

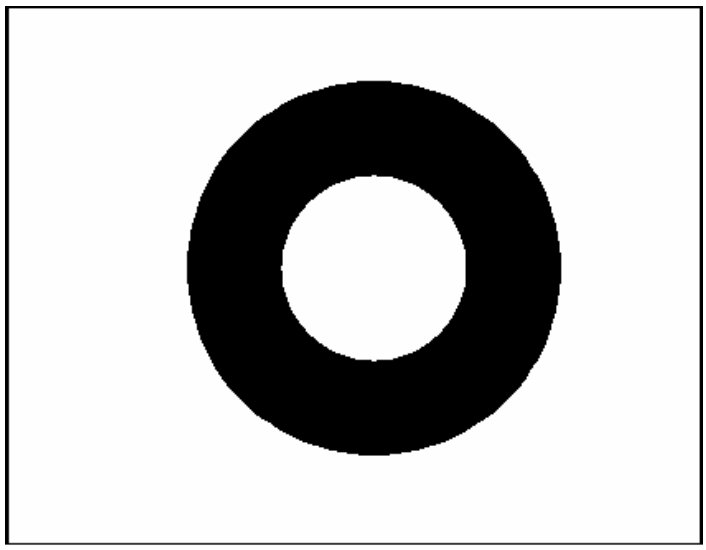
   Inside diameter <last>: **.5**

4. **Type** A value for the outside diameter.

   Outside diameter <last>: **1**

5. **Pick** A point for the center of the donut.

   Center of doughnut: (**point**)



Ellipse 16.7

Creates an ellipse or an elliptical arc.

1. **Choose** Draw, Ellipse.

   or

2. **Choose** the Ellipse or Partial Ellipse icon. 

   or

3. **Type** ELLIPSE at the command prompt

   Command: **ELLIPSE**

4. **Type** One of the following options: Arc/Center/Isocircle /<Axis endpoint 1>:

Ellipse options:

Axis endpoint 1 Defines the first axis by two specified endpoints. The angle of the first axis determines the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

Axis endpoint 2: <Other axis distance> / Rotation: Specify a point or enter a distance

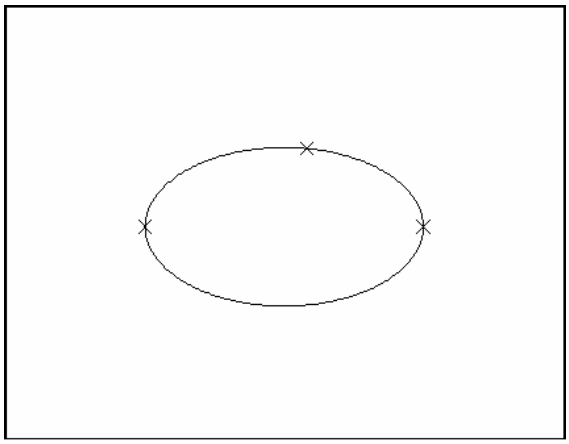
Arc Creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis of the elliptical arc.

Center Creates the ellipse by a specified center point.

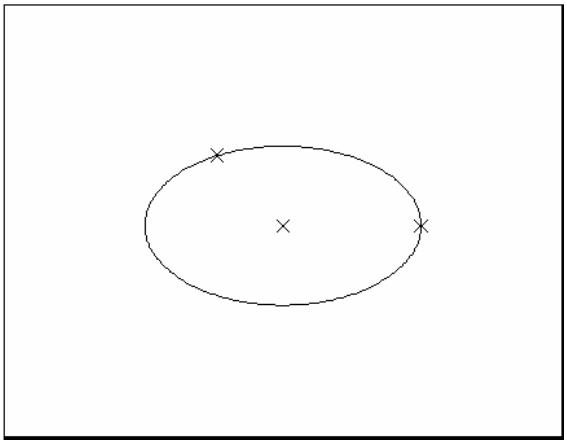
Isocircle Creates an isometric circle in the current isometric drawing plane.

Rotation The major axis is now treated as the diameter of a circle that will be rotated a specified amount around the axis. You enter an angle between 0 and 89.4 degrees.

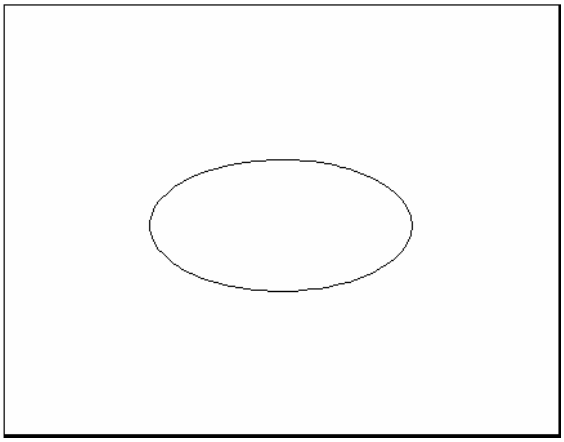
ELLIPSE,
Axis , Eccentricity (Axis Endpoint, Axis Endpoint, Other Axis Distance)



ELLIPSE,
Center, Axis, Axis



ELLIPSE,
Axis Endpoint, Axis Endpoint, Rotation=60

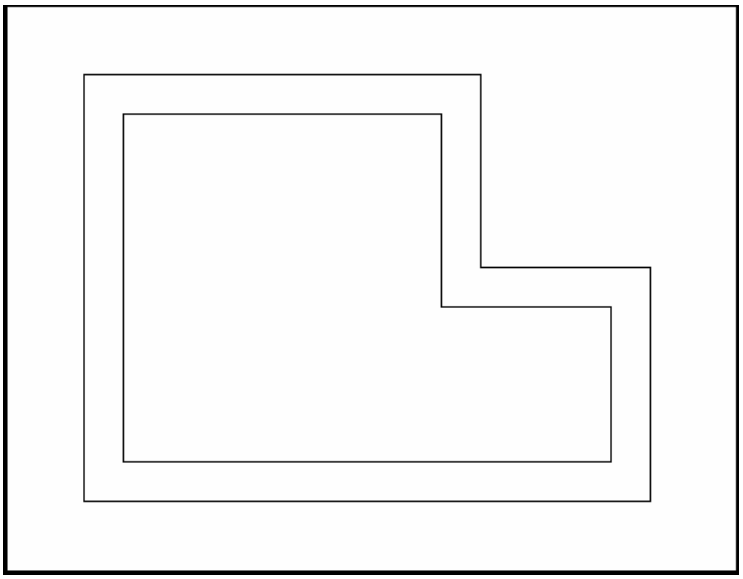


Multilines 16.8

MLINE Command

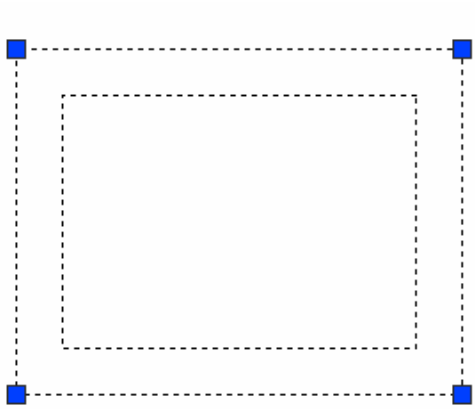
Creates multiple parallel lines.

1.	**Choose**	Draw, Multiline.
		or
2.	**Type**	MLINE at the command prompt.
		Command: **MLINE**
3.	**Pick**	A point to start the multiline.
		Justification/Scale/STyle/<From point>: pick point
4.	**Pick**	A second point to continue the multiline.
		<To point>: **pick point**
5.	**Pick**	The next point to continue drawing multilines.
		Undo/<To point>: pick point
6.	**Press**	ENTER to end the mulitline
		Close/Undo/<To point>: press enter or
7.	**Type**	C to close the multiline back to the first point.
		Close/Undo/<To point>: **c**

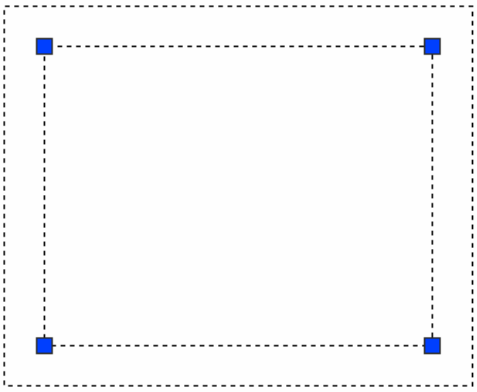


Multiline Justifications

Top Justification



Bottom Justification



Zero Justification

Multiline Styles 16.9

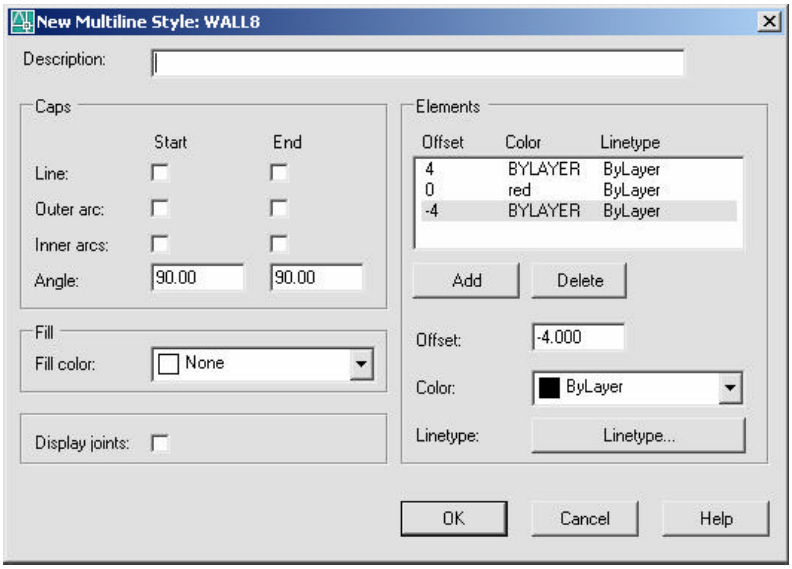
1. **Choose** Format, Multiline Style...

   or

2. **Type** MLSTYLE at the command prompt.

   Command: **MLSTYLE**

3. **Rename** The existing style called STANDARD to your newstyle.
4. **Choose** ElementProperties to change the appearance of the multilines.
5. **Choose** ADD to create the newmultiline.



Editing Multilines 16.10

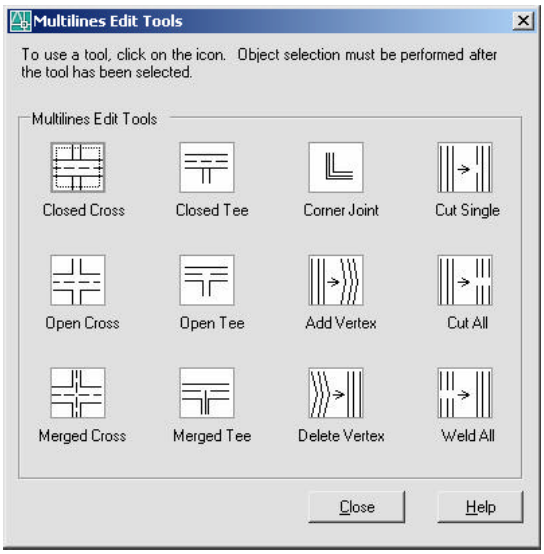
1. **Choose** Modify, Multiline...

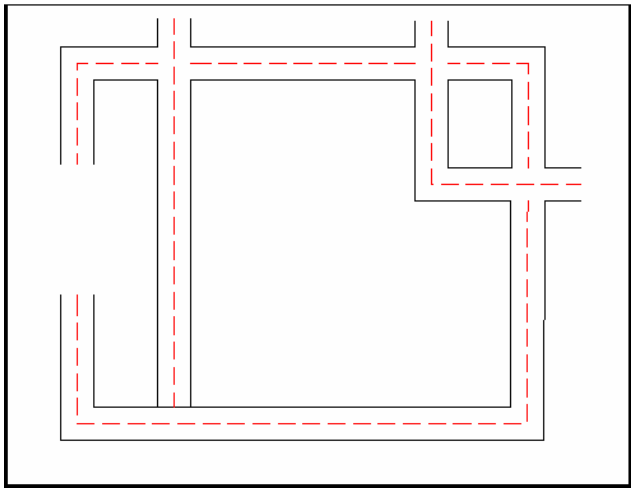
   or

2. **Type** MLEDIT at the command prompt

   Command: **MLEDIT**

3. **Choose** From one of the mledit options:





Construction Line 16.11

Creates an infinite line.

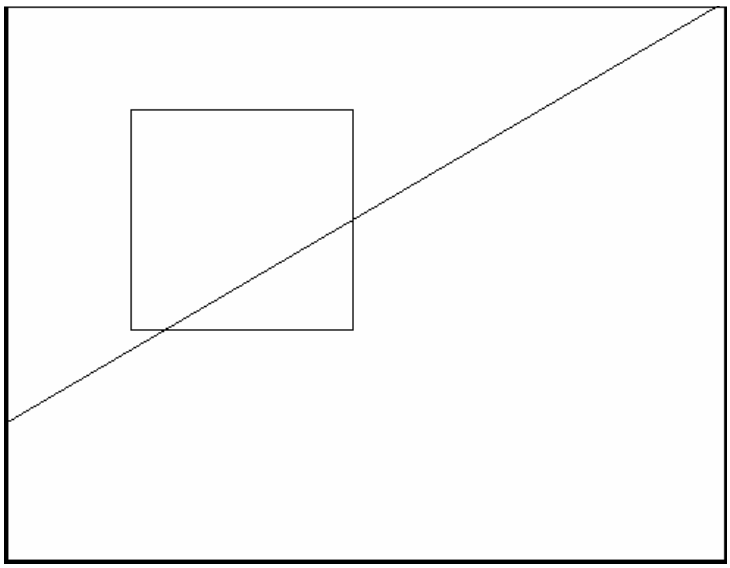
1. **Choose** Draw, ConstructionLine
   or
2. **Choose** the XLINE icon. 
   or
3. **Type** XLINE at the command prompt.

   Command: **XLINE**

   Specify a point or [Hor/Ver/Ang/Bisect/Offset]:

XLINE Options

HOR	Creates a horizontal xline passing through a specified point.
VER	Creates a vertical xline passing through a specified point
ANG	Creates an xline at a specified angle.
BISECT	Creates an xline that passes through the selected angle vertex and bisects the angle between the first and second line
OFFSET	Creates an xline parallel to another object.



Ray Command 16.12

Creates an infinite line in one direction.

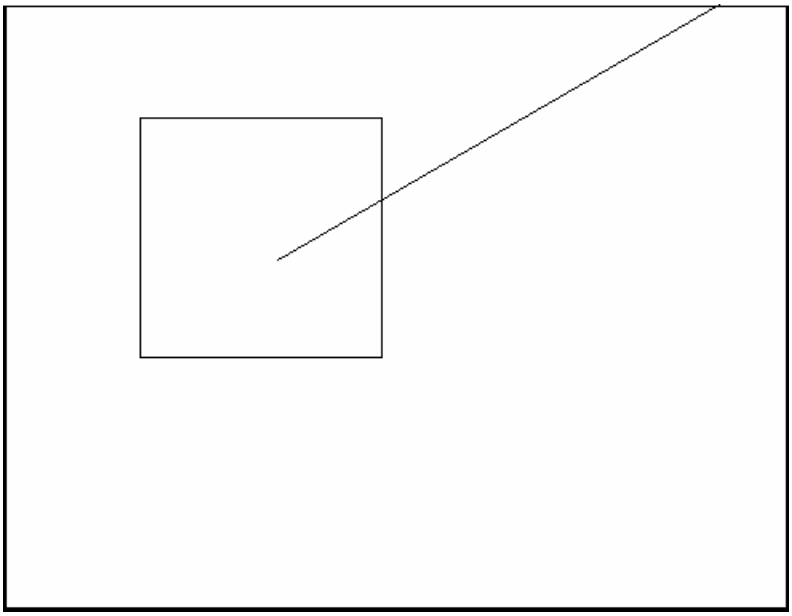
1. **Choose** Draw, RAY

   or

2. **Type** RAY at the command prompt.

   Command: **RAY**

   Specify a point : **(pick through point)**



Chapter 17
Crosshatching

BHATCH Command 17.1

1. **Choose** Draw, Hatch...

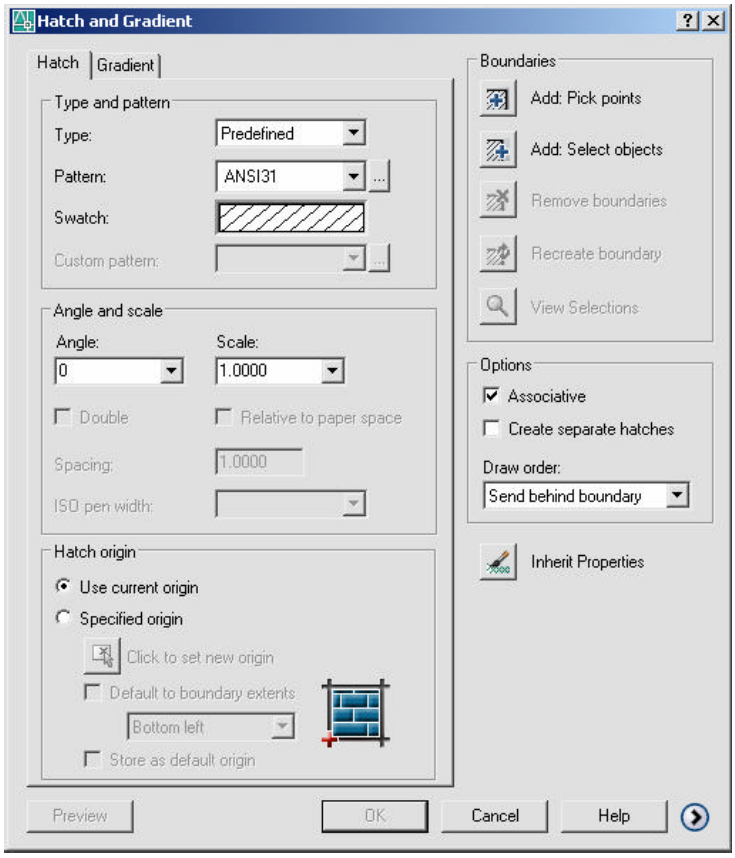
or

2. **Click** the Hatchicon. 

or

3. **Type** BHATCH at the command prompt

Command: **BHATCH**

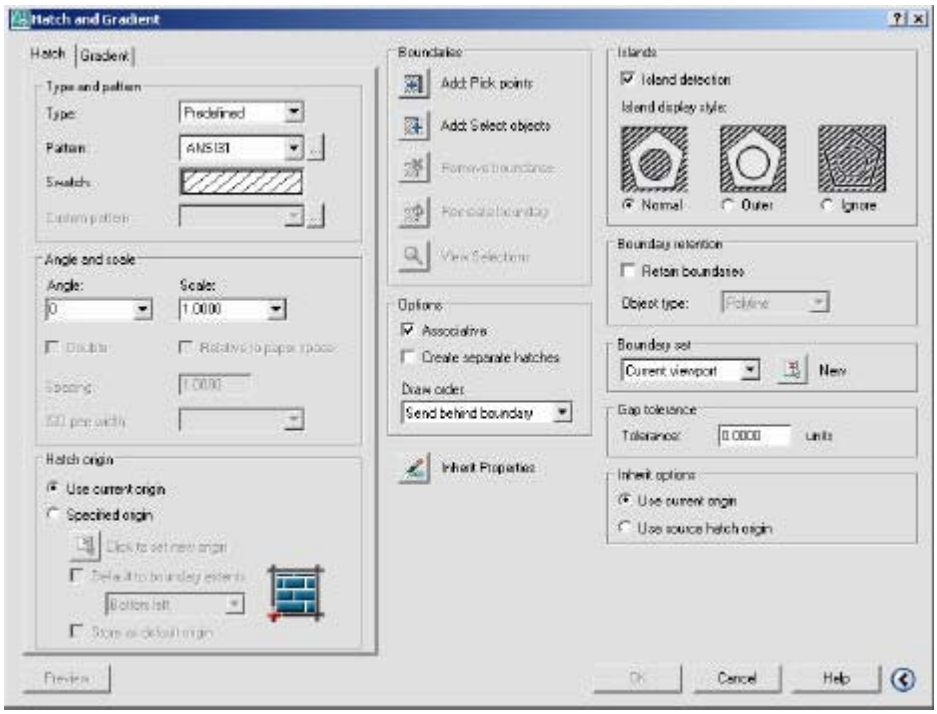


BHATCHoptions:

Pattern Type	Sets the current pattern type by using AutoCAD's predefined patterns or user defined patterns.
Pattern Properties	Sets the current pattern, scale, angle, and spacing. Controls if hatch is double spaced or exploded.
Pick Points	Constructs a boundary from existing objects that form an enclosed area.
Select Objects	Selects specific objects for hatching. The Boundary Hatch dialog box disappears and AutoCAD prompts for object selection.
Inherit Properties	Applies the properties of an existing associative hatch to the current Pattern Type and Pattern Properties options.
Preview Hatch	Displays the hatching before applying it. AutoCAD removes the dialog box and hatches the selected areas.
Associative	Controls associative hatching.
Apply	Creates the crosshatching in the boundary.

Advanced Hatch Options 17.2

1. **Choose** the **Advanced...** TAB from the BHATCH dialog.
2. **Choose** one of the following advanced options:



Define Boundary Set

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

Hatchstyle

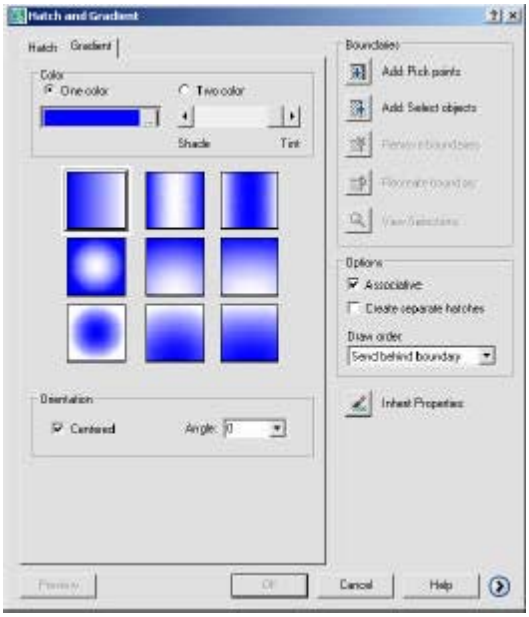
Specifies the method used to hatch objects within the outermost hatch bound- ary. If there are no internal objects selected, specifying a hatching style has no effect.

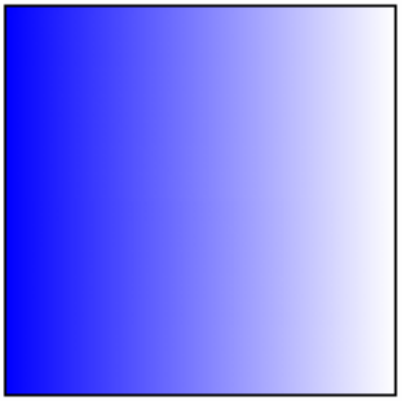
Boundary Options

Specifies whether or not the temporary boundary objects will be added to the drawing.

Gradient Hatch 17.3

1. **Choose** the **Gradient...** TAB from the BHATCH dialog.
2. **Choose** one of the following advanced options:





HATCHEDIT 17.4

1. **Choose** Modify, Hatch...

   or

2. **Click** the Hatch Edit icon from the Modify II toolbar. 

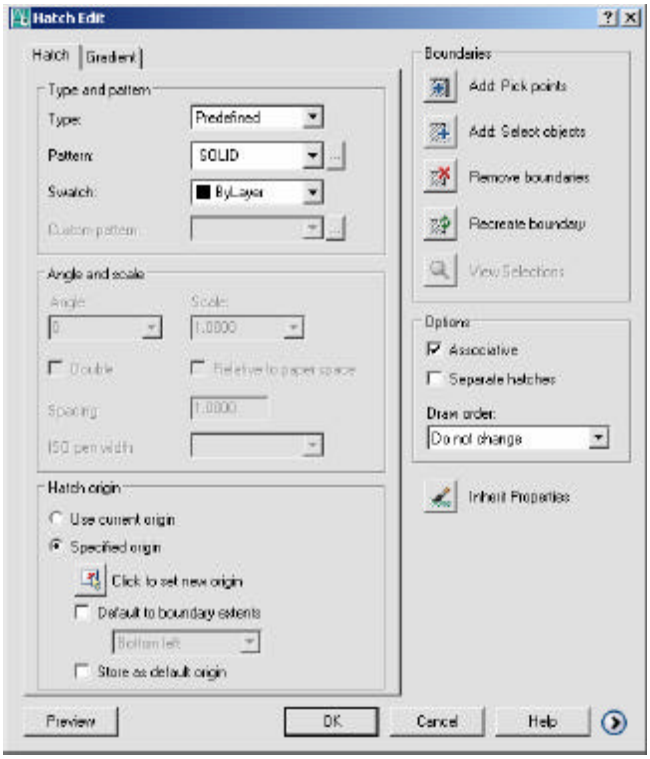
   or

3. **Type** HATCHEDIT at the command prompt.

   Command : **HATCHEDIT**

4. **Choose** One of the BHATCH options to modify.

5. **Pick** The OK button.



Inherit Hatch 17.5

1. **Choose** Draw, Hatch...

   or

2. **Click** the Hatchicon. 

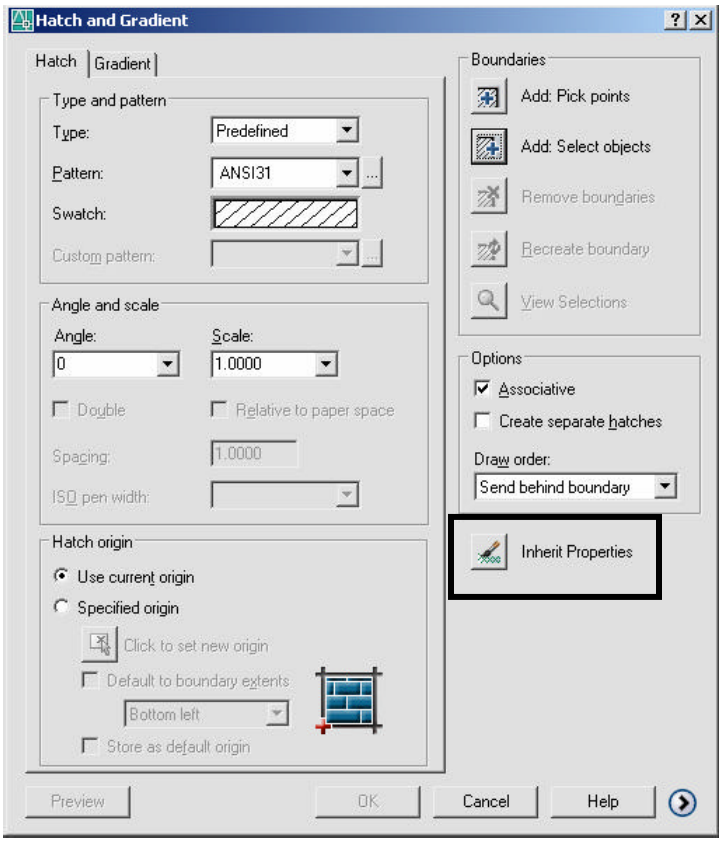
   or

3. **Type** BHATCH at the command prompt

   Command: **BHATCH**

4. **Choose** InheritProperties.

5. **Pick** the crosshatchofanexisting associative hatchto make the current Pattern Type and Pattern Properties options. Preview Hatch Displays the hatching before applying it.



Chapter 18
Regions and Boundaries

Boundary Command 18.1

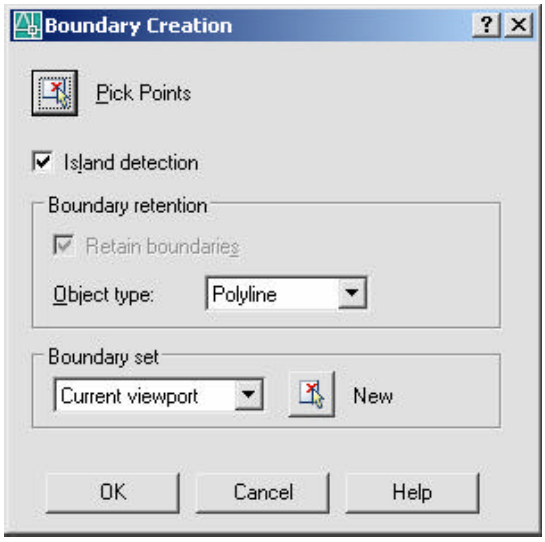
Defines the object type, boundary set, and island detection method for defining boundaries from points you specify.

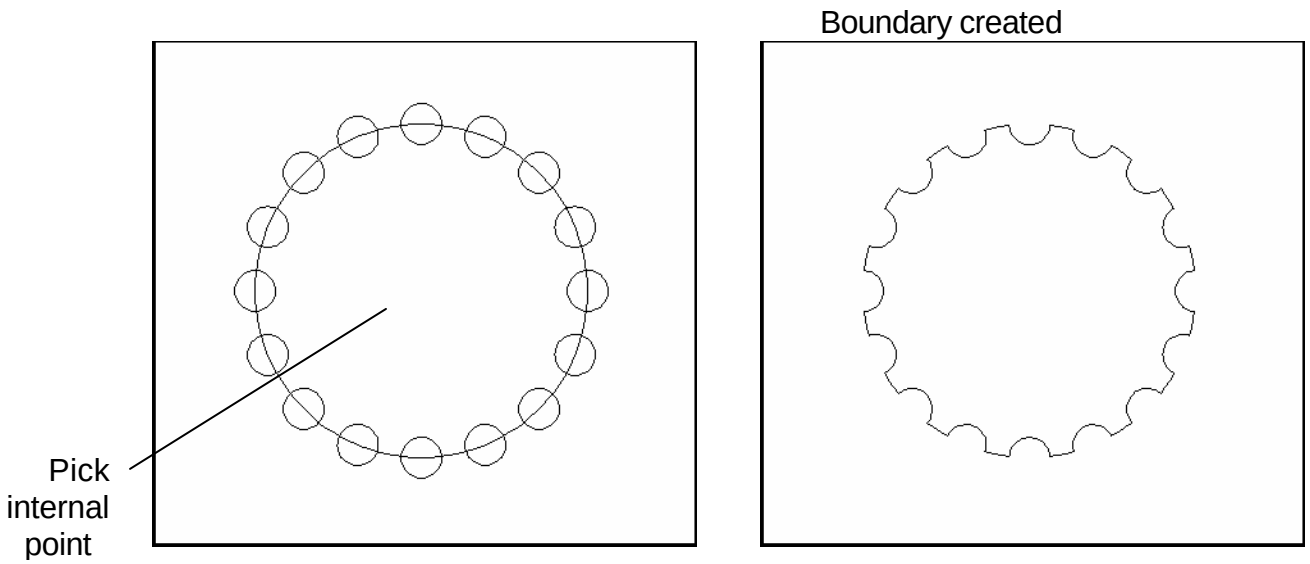
1. **Choose** Draw, Boundary

or

2. **Type** BOUNDARY at the command prompt.

Command: **BOUNDARY**





Region Command 18.2

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.

1. **Choose** Draw, Region
2. **Type** REGION at the command prompt.

Command: **REGION**

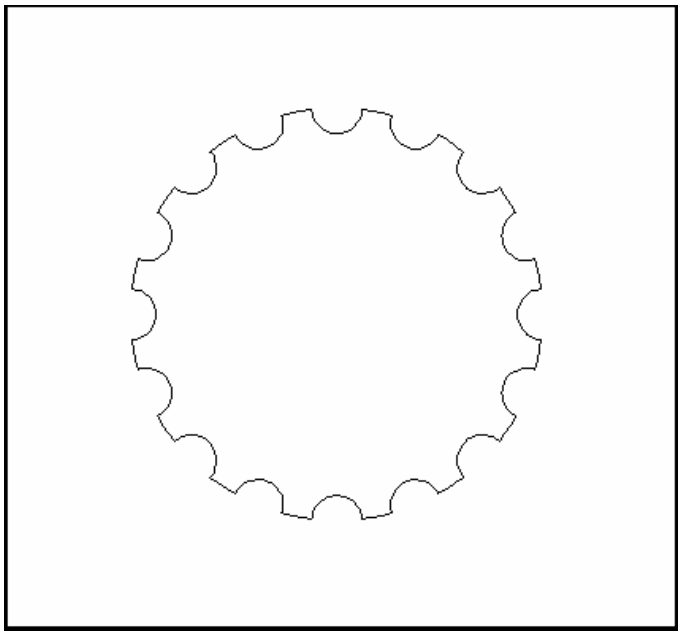
Select objects: **(pick boundary)**

Select objects:1 found

1 loop extracted.

1 Regioncreated.

Object created as a region



Mass Properties 18.3

Calculates the mass properties of regions or solids.

1. **Choose** Tools, Inquiry, Region/Mass Properties
2. **Type** MASSPROP at the command prompt.

Command: **MASSPROP**

Select objects: **(pick region)**

---------- REGIONS ----------

Area:	11.1328
Perimeter:	16.3734
Bounding box:	X: 3.1508 -- 7.1352
	Y: 2.8950 -- 6.8942
Centroid:	X: 5.1508
	Y: 4.8946
Moments of inertia:	X: 276.6983
	Y: 305.3510
Product of inertia:	XY: 280.6701
Radii of gyration:	X: 4.9854
	Y: 5.2372

Principal moments and X-Y directions about centroid:
I: 9.9891 along [1.0000 0.0000] J: 9.9891 along [0.0000 1.0000]

Write analysis to a file? [Yes/No] <N>:

Chapter 19
Blocks and Attributes

Creating Local Blocks (BMAKE) 19.1

1. **Choose** Draw, Block, Make.

or

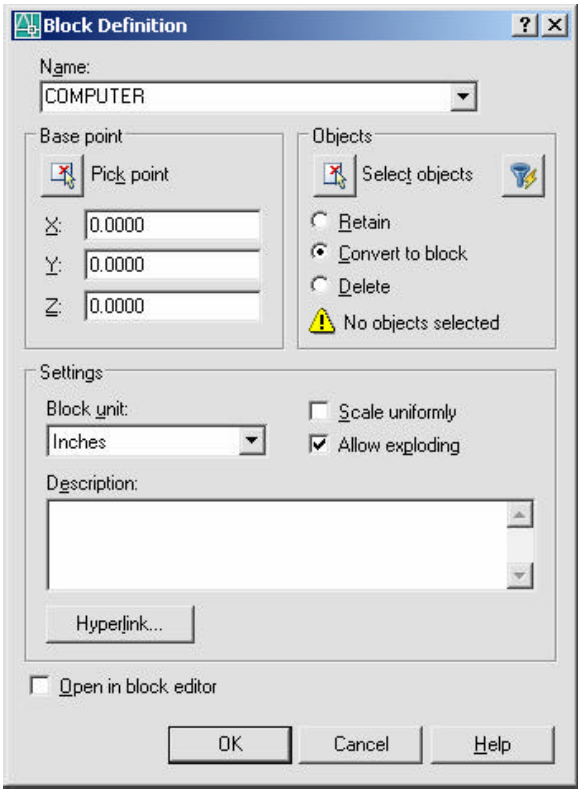
2. **Click** the Make Block icon. 

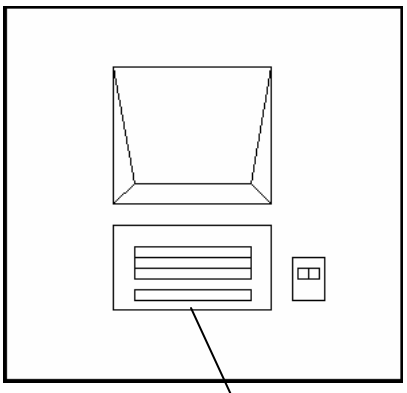
or

3. **Type** BMAKE at the command prompt.

   Command: BMAKE or BLOCK

4. **Type** the name of the block.
5. **Pick** an insertion point.
6. **Select** objects to be included in the block definition.
7. **Click** OK.



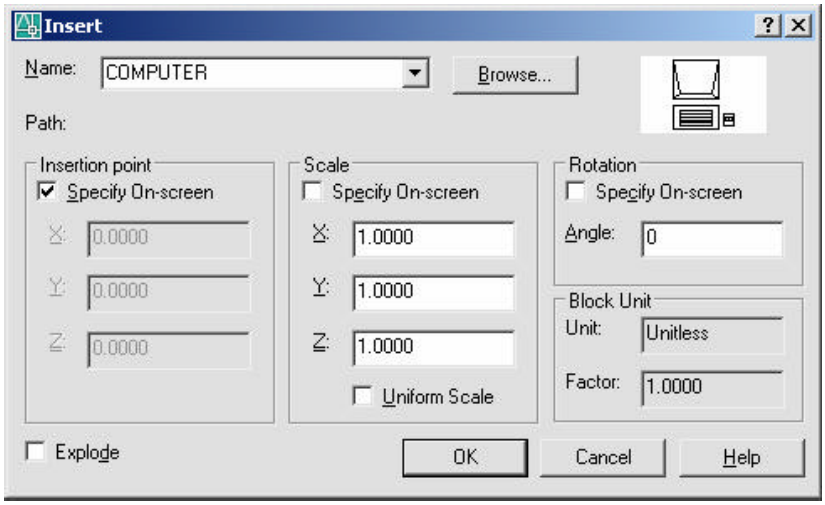


Note You cannot use DIRECT, LIGHT, AVE_RENDER, RM_SDB, SH_SPOT, and OVERHEAD as valid block names.

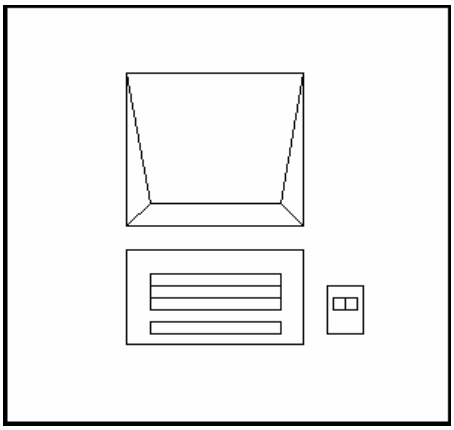
Inserting Blocks 19.2

1. **Choose** Insert, InsertBlock
   or
2. **Click** the Insert icon from the INSERT toolbar. 
3. **Type** INSERT at the command prompt.

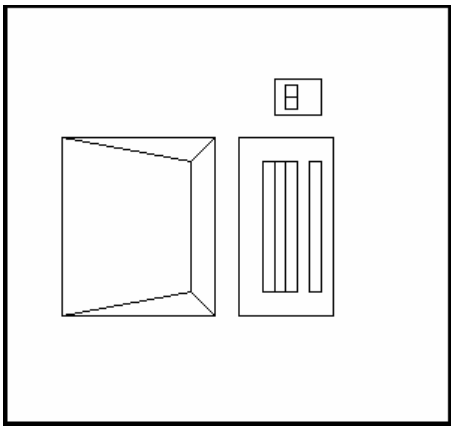
   Command: **INSERT**
4. **Choose** the name to insert a local block and Browse...to insert a Wblock.
5. **Choose** the insertion point, scale, and rotation of the block.



Block Inserted with a zero degree rotation angle



Block Inserted with a ninety degree rotation angle



Typing Insert (-INSERT)

1.	**Type**	- INSERT at the command prompt. Command: **-INSERT**
2.	**Type**	Block name to insert. Insert block name or (?) **type name**
3.	**Pick**	An insertion point. Insertion point: **pick point**
4.	**Press**	ENTER to keep the same x scale factor as the original block. X scale factor <1>Corner / XYZ:
5.	**Press**	ENTER to keep the same x scale factor as the original block. Y scale factor (default=X):
6.	**Press**	ENTER to keep a rotation angle of zero. Rotation angle <0>: **or**
7.	**Pick**	A rotation angle.

Control the Color and Linetype of Blocks 19.3

The objects in an inserted block can retain their original properties, can inherit properties from the layer on which they are inserted, or can inherit the properties set as current in the drawing.

You have three choices for how the color, linetype, and lineweight properties of objects are treated when a block reference is inserted.

- Objects in the block do not inherit color, linetype, and lineweight properties from the current settings. The properties of objects in the block do not change regardless of the current settings.
- For this choice, it is recommended that you set the color, linetype, and lineweight properties individually for each object in the block definition: do not use BYBLOCK or BYLAYER color, linetype, and lineweight settings when creating these objects.
- Objects in the block inherit color, linetype, and lineweight properties from the color, linetype, and lineweight assigned to the current layer only.
- For this choice, before you create objects to be included in the block definition, set the current layer to 0, and set the current color, linetype, and lineweight to BYLAYER.
- Objects inherit color, linetype, and lineweight properties from the current color, linetype, and lineweight that you have set explicitly, that is, that you have set to override the color, linetype, or lineweight assigned to the current layer. If you have not explicitly set them, then these properties are inherited from the color, linetype, and lineweight assigned to the current layer.
- For this choice, before you create objects to be included in the block definition, set the current color or linetype to BYBLOCK.

If you want objects in a block to	Create objects on these layers	Create objects with these properties
Retain original properties	Any but 0 (zero)	Any but BYBLOCK or BYLAYER
Inherit properties from the current layer	0 (zero)	BYLAYER
Inherit individual properties first, then layer properties	Any	BYBLOCK

Wblock Command 19.4

Writes objects to a new drawing file.

1. **Type** WBLOCK at the command prompt

   Command: **WBLOCK**

2. **Type** A drawing name (and location).

3. **Type** A block name if a local block already exists. Block name: **name**

   or

4. **Press** ENTER to create a block.

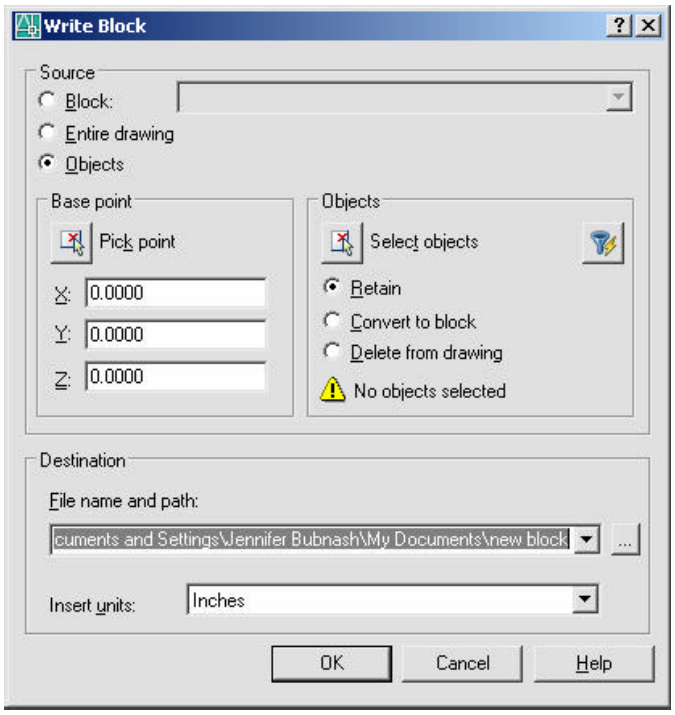
5. **Pick** An insertion point on the object

   Insertion base point: **pick a point**

6. **Pick** Objects to create the block.

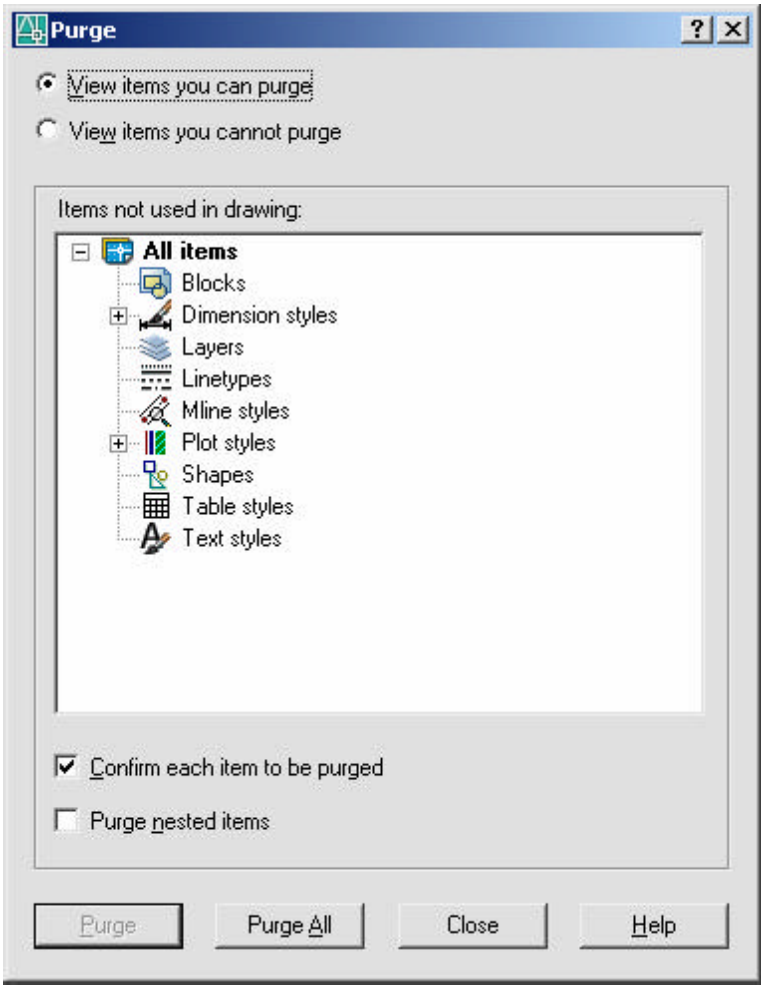
   Select objects: **pick objects**

7. **Press** ENTER to end the selection set.



Purge 19.5

1. **Choose** File, Drawing Utilities, Purge.
   or
2. **Type** PURGE at the command prompt
   Command: **PURGE**
3. **Choose** One of the following purge options:
   Purge unused
   Blocks/Dimstyles/LAyers/ LTypes/
   SHapes/STyles/Mlinestyles/All:

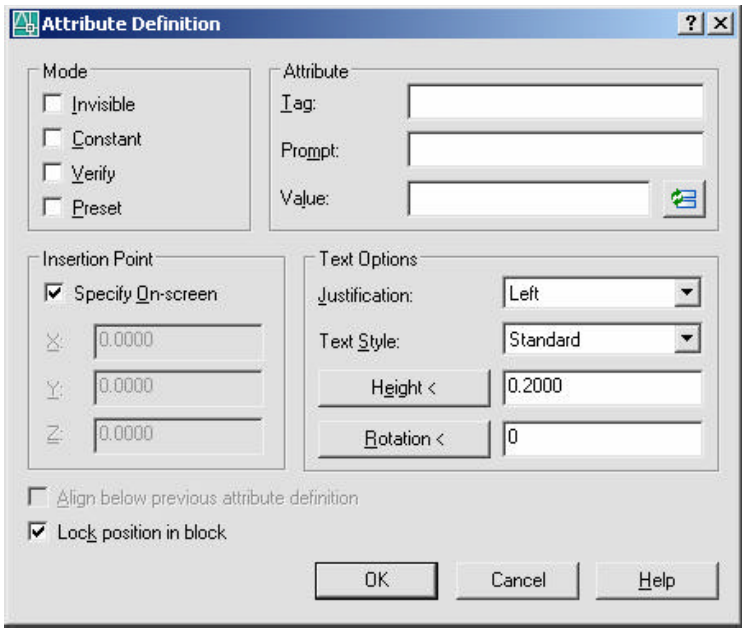


Defining Attributes 19.6

1. **Choose** Draw, Block, Define Attributes...

or

2. **Type** ATTDEF at the command prompt. Command: **ATTDEF**
3. **Choose** TAG to specify each attribute occurrence in the drawing.
4. **Choose** Prompt to fill in the prompt that the user sees when placing the attribute.
5. **Choose** Value to fill in a default value.
6. **Pick** An insertion point for each attribute
7. **Create** A block which includes the new attributes.



Toggle the following mode settings to on or off.

Invisible Does not display, but allows extraction.

Constant All occurrences of this Attribute have the same value.

Verify Issues extra prompts to verify a proper value.

Preset Does not prompt for this Attribute during Block insertion.

Editing Attributes 19.7

1. **Choose** Modify, Object, Attribute, Single...

or

2. **Click** the Edit Attribute icon from the Modify II Toolbar. 

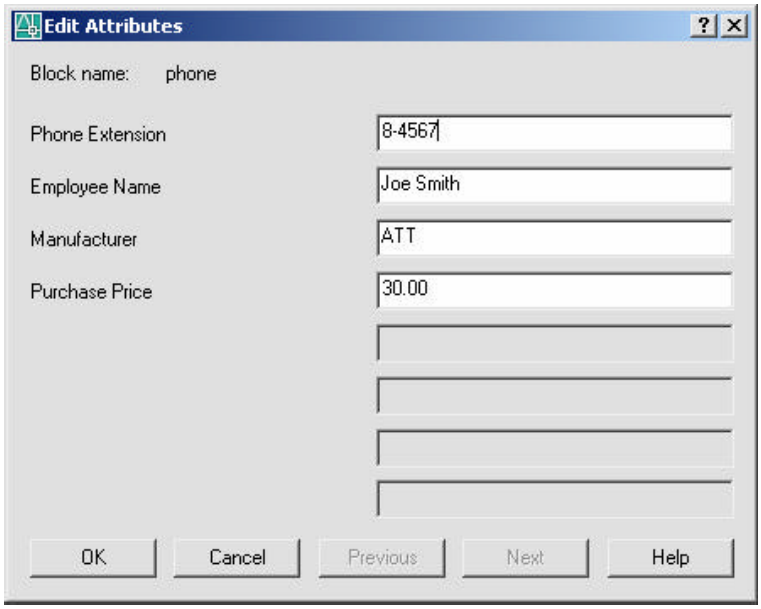
or

3. **Type** ATTEDIT at the command prompt

   Command: **ATTEDIT**

4. **Pick** The block to edit

   Select block: **pick**



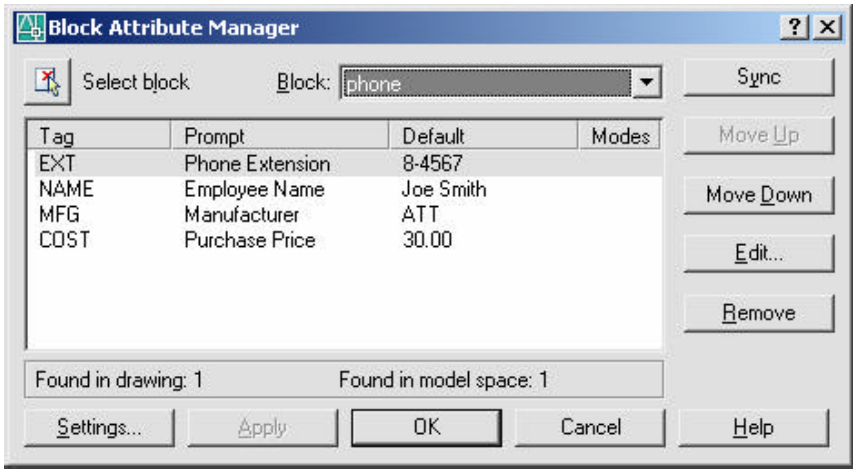
Block Attribute Manager 19.8

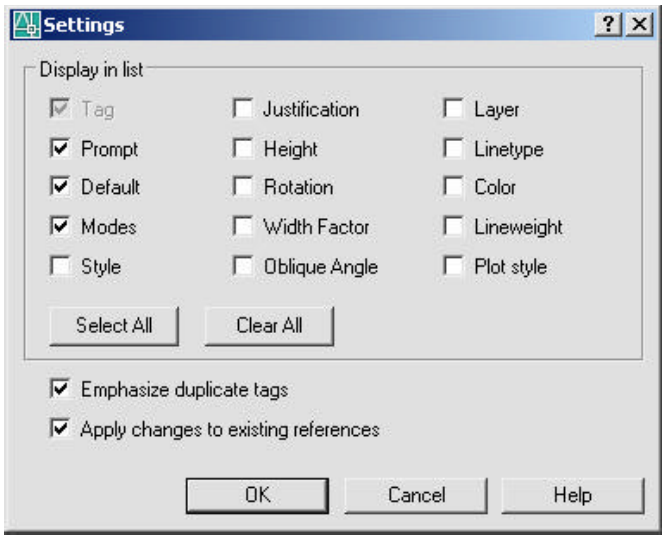
1. **Open** a drawing with attributes.
2. **Type** BATTMAN at the command prompt.

   Command: **BATTMAN**

   or
3. **Choose** Modify, Object, Attribute, Block Attribute Manager

   or
4. **Click** the Block Attribute Manager icon from the Modify II Toolbar 





Synchronize Attributes 19.9

Updates all instances of a specified block with the current attributes defined for the block

1. **Open** a drawing with attributes.
2. **Type** ATTSYNC at the command prompt.
   Command: **ATTSYNC**
3. **Click** the Block Attribute Manager icon from the Modify II toolbar. 
4. **Press** Enter an option [?/Name/Select] <Select>:
   enter and select a block with attributes.
5. **Press** ATTSYNC block computer? [Yes/No]<Yes>:
   enter to synchronies. ATTSYNC complete.

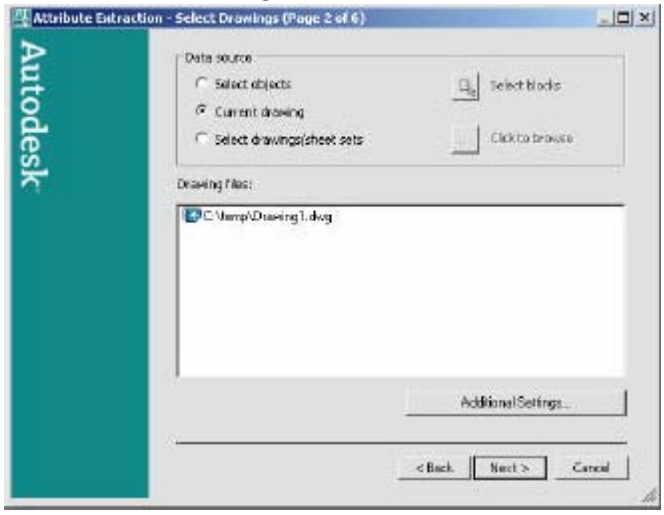
Enhanced Attribute Extract 19.10

The Editor enables you to edit the attributes in an individual block as opposed to the Block Manager which is used to Block Definitions.

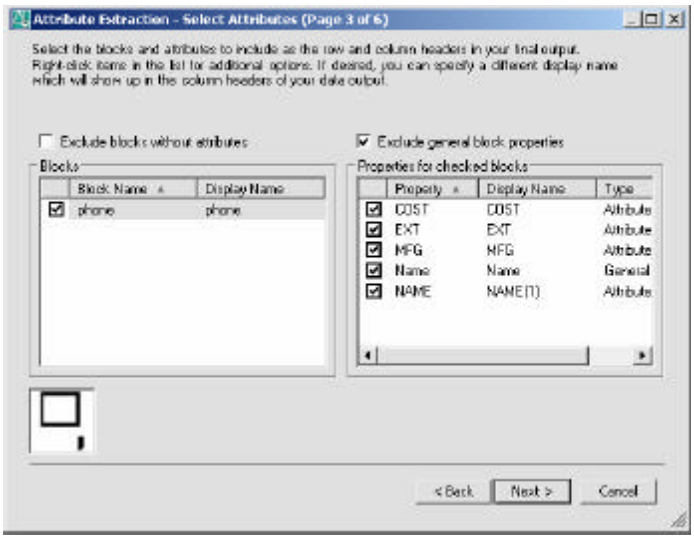
1. **Open** a drawing with block attributes.
2. **Choose** Tools, Attribute Extraction...

or

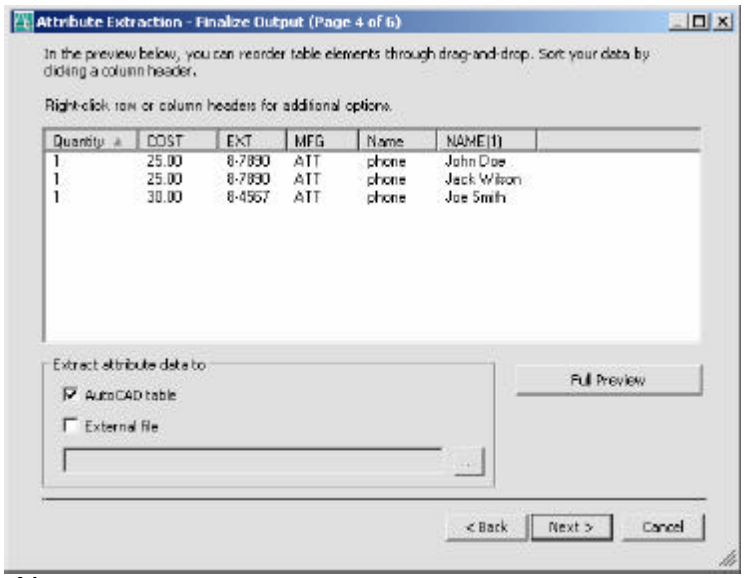
3. **Click** the Attribute Extract icon from the Modify II toolbar. 
4. **Choose** the Current Drawing radio button and Next.



5. **Click** Next to choose Xrefs and nested blocks from the Settings window.
6. **Click** Next to not choose an existing template.
7. **Choose** the Blocks and Attributes to extract and Next.



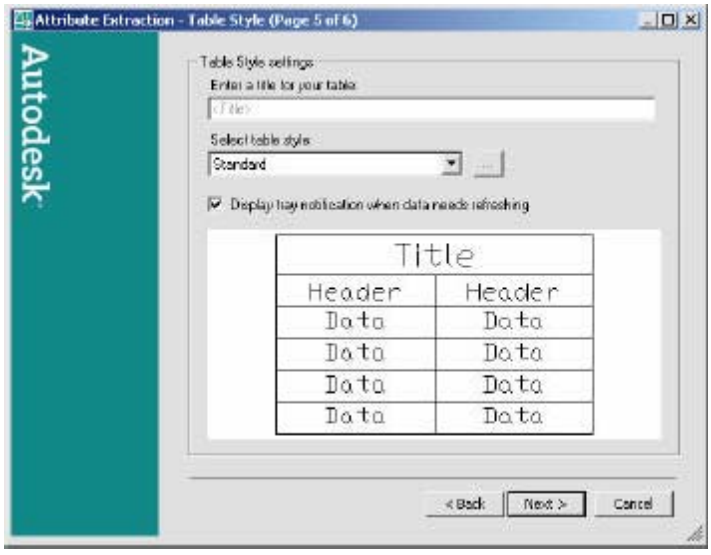
8. **Choose** the View output.

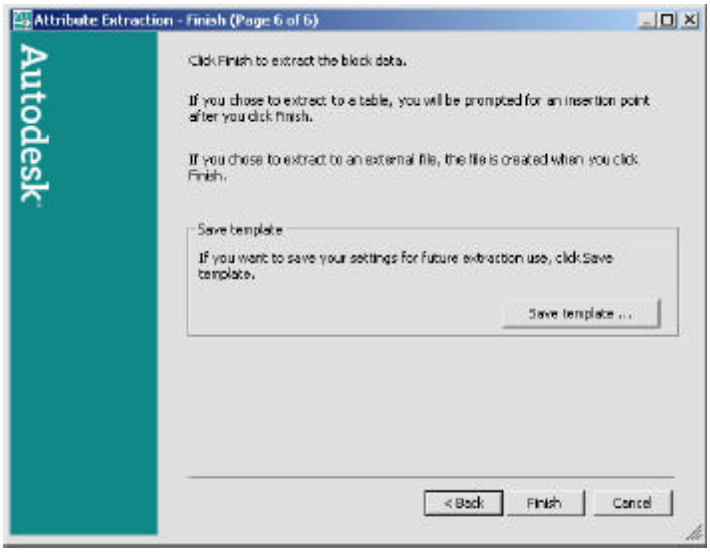


9. **Click** Next.

10. **Save** the file phone.xls

11. **Open** the template in Excel.





Chapter 20
Design Center and Tool Palettes

Design Center Overview 20.1

The AutoCAD DesignCenter finds and tranfers blocks, text styles, layers, dimensionstyles, etc from drawings, WEB

1. **Choose** Tools, AutoCAD DesignCenter.

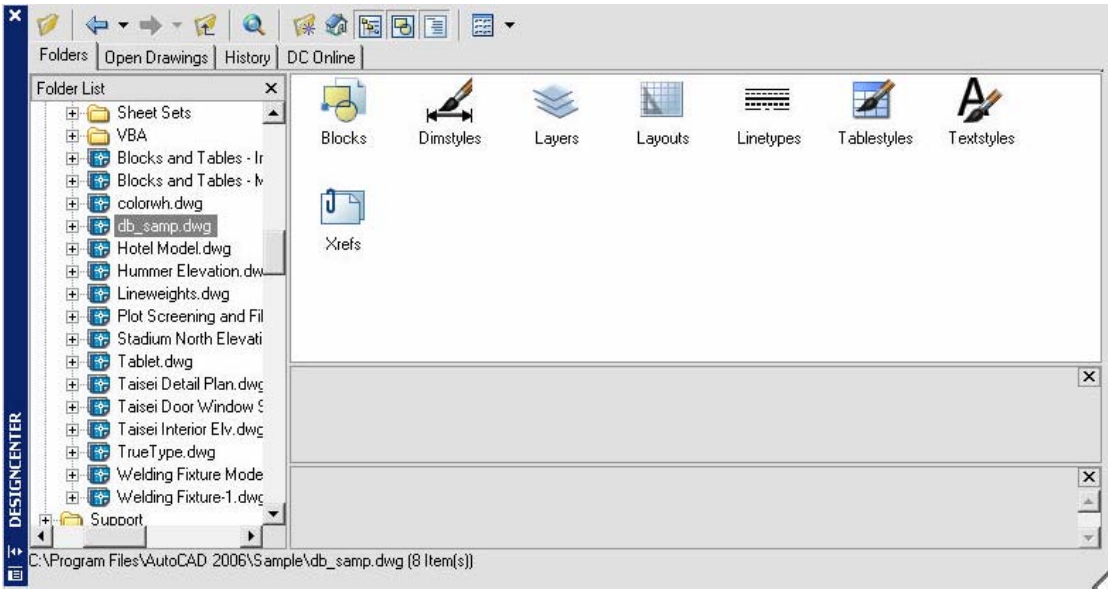
   or

2. **Press** **CTL+2** on the keyboard.

   or

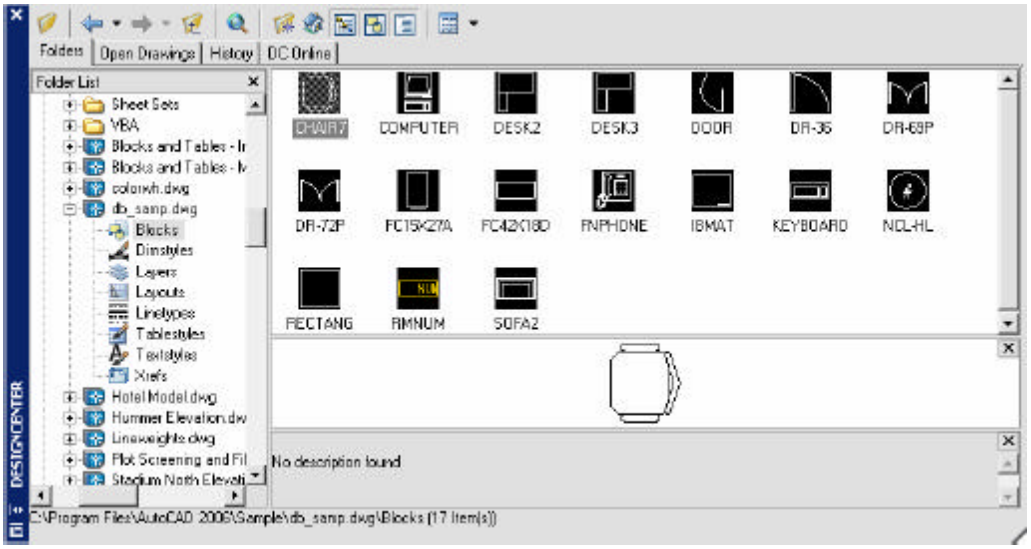
3. **Type** ADCENTER at the command prompt.

   Command: **adcenter**



Design Center Blocks 20.2

1. **Choose** Blocks  from one of the Design Center menus.
2. **Drag** and drop a block from the Design Center into a drawing.

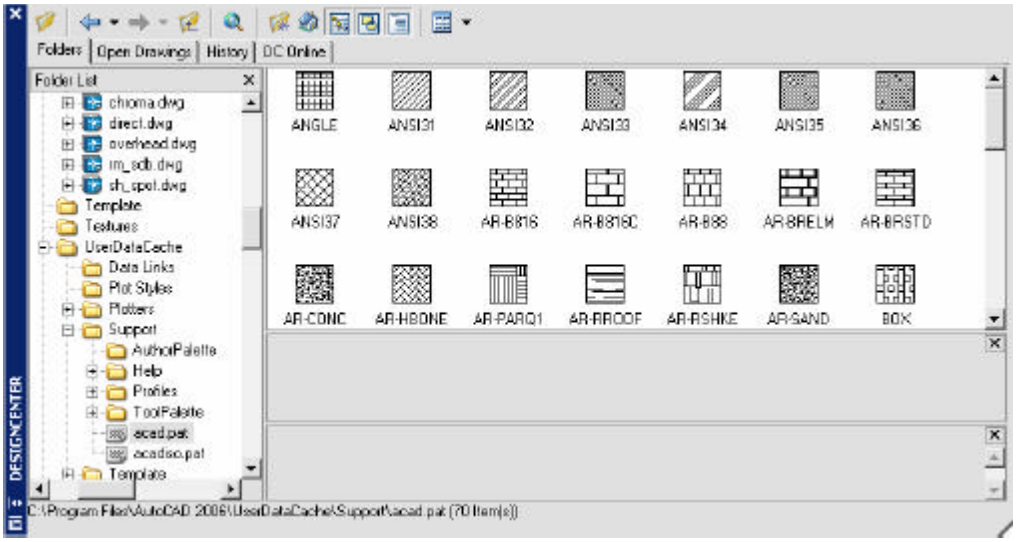


TIP:

- Blocks with attributes will be prompted as they are inserted into the drawing

Hatching from the Design Center 20.3

1. **Choose** a crosshatch pattern from the following AutoCAD directory \AutoCADxxxx\Support\acad.pat or \AutoCADxxxx\Backup
2. **Drag** and drop a pattern into a drawing.



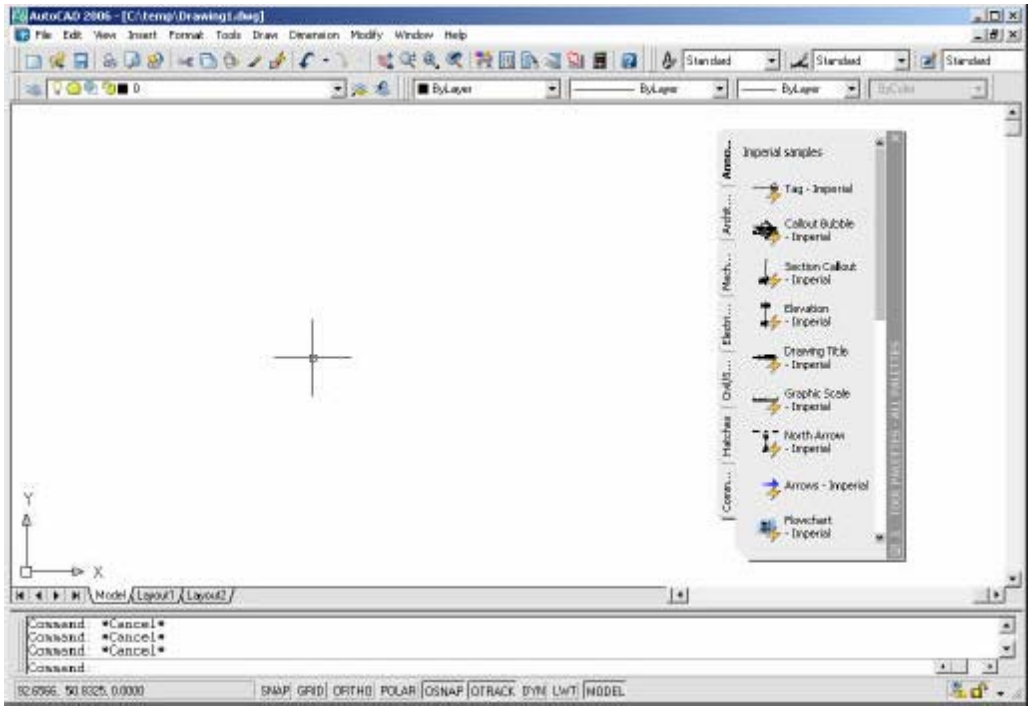
TIP:

- Be sure the HPSCALE is set before dropping a hatch pattern into a drawing.

20.4 Tool Palettes

1. **Choose** Tool, ToolPalettes Window

2. **Choose** Palettes icon from the Standard Toolbar 



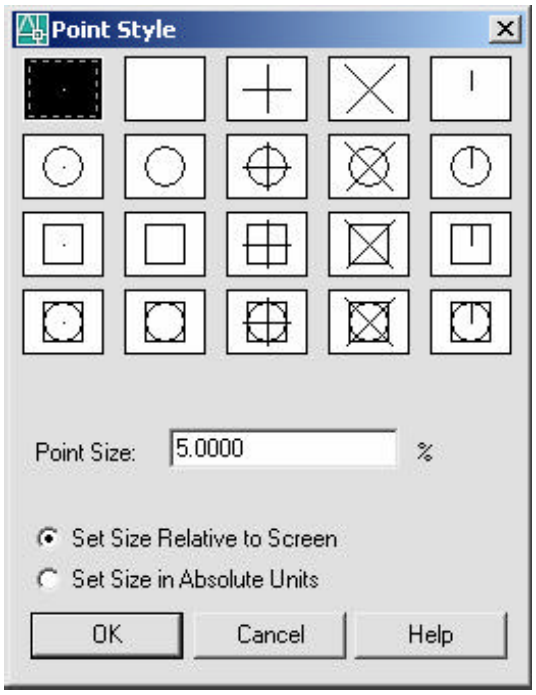
Chapter 21
Point, Divide, Measure

Point Styles 21.1

Changes the appearance of points and point sizes.

1. **Choose** Format, Point Style...
   or
2. **Type** DDPTYPE at the command prompt.

   Command : **DDPTYPE**



Point Command 21.2

1. **Choose** Draw, Point, Single or Multiple Point.

or

2. **Click** the Point icon. 

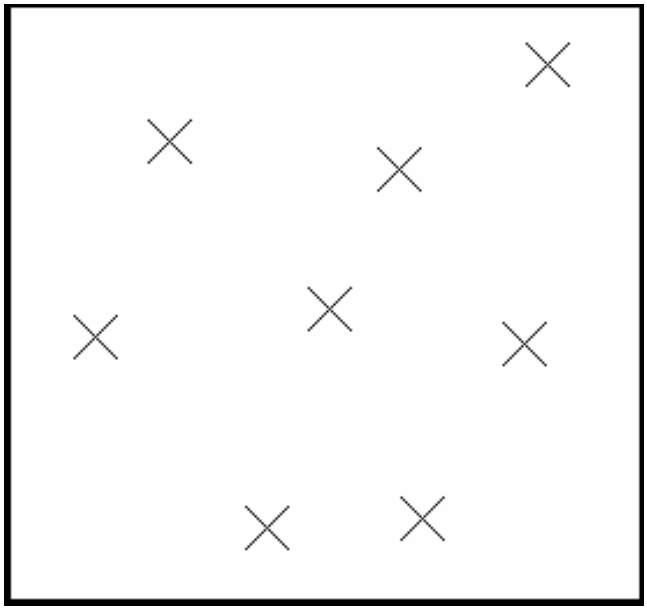
or

3. **Type** POINT at the command prompt.

   Command : **POINT**

4. **Pick** A point on the drawing.

   Point (**point**)



Divide 21.3

1. **Choose** Draw, Point, Divide.

or

2. **Type** DIVIDE at the command prompt

Command: DIVIDE

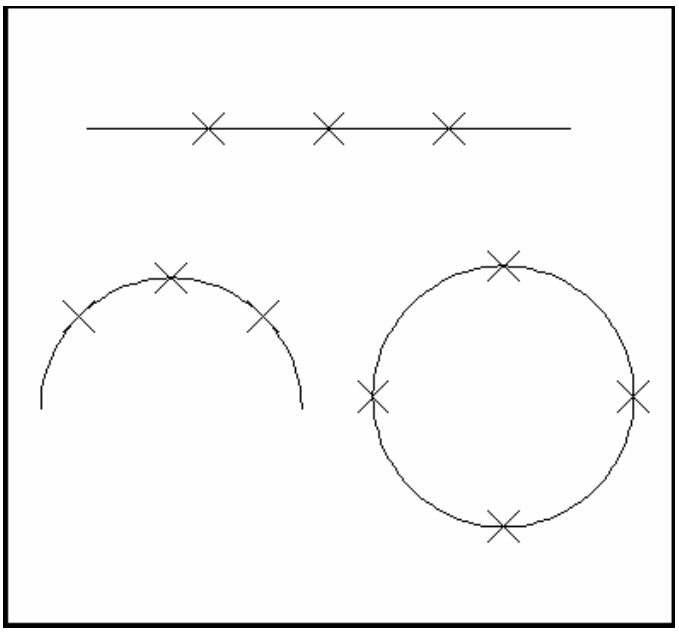
3. **Pick** Object to divide

Select object to divide: (**pick one object**)

You can select a single Line, Arc, Circle, or polyline. If you enter a segment count between 2 and 32,767, Point entities will be placed along the object to divide it into that number of equal segments.

4. **Type** The number of equal segments to divide the object into<Number of segments>/Block: (**number**)

Objects divided using points



or

5. **Type** B to specify a block instead of a point to insert.
6. **Type** The name of the block to insert

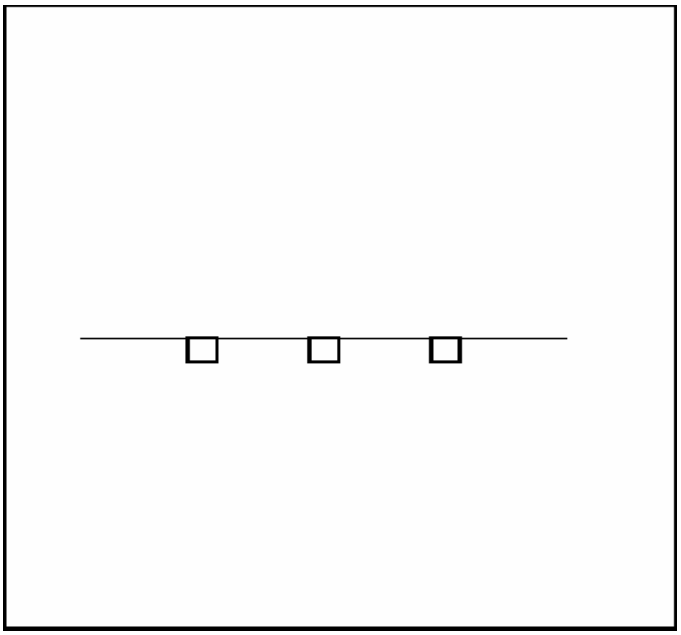
   Block name to insert: (**name**)
7. **Type** Yes or No to align the block with an object

   Align block with object? **Y** or **N** Number of

   segments:
8. **Type** The number of equal segments to divide the object into<Number of segments>/Block: (**number**)

TIP: The Block must currently be defined within the drawing. If you answer yes to the Align block? prompt, the Block will be rotated round its insertion point so that it is drawn tan- gent to the object being divided.

Objects divided using block symbols



Measure 21.4

1. **Choose** Draw, Point, Measure.

   or

2. **Type** MEASURE at the command prompt. Command: **MEASURE**

3. **Pick** Object to measure: Select object to measure: (**pick one object**)

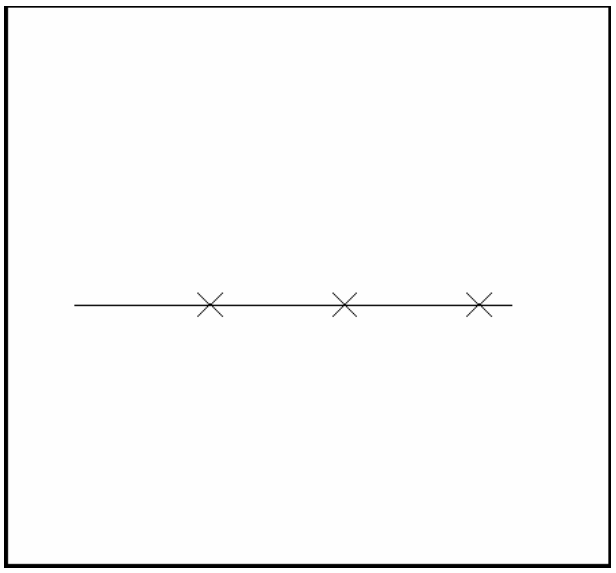
4. **Type** The length of each segment along the object.

   <Length of segment>/Block:(**number**)

   or

5. **Type** B to specify a block instead of a point to insert.

Points placed along measured distance
(remaining length is on the right side of the line)



Chapter 22
Grips

Grips Overview 22.1

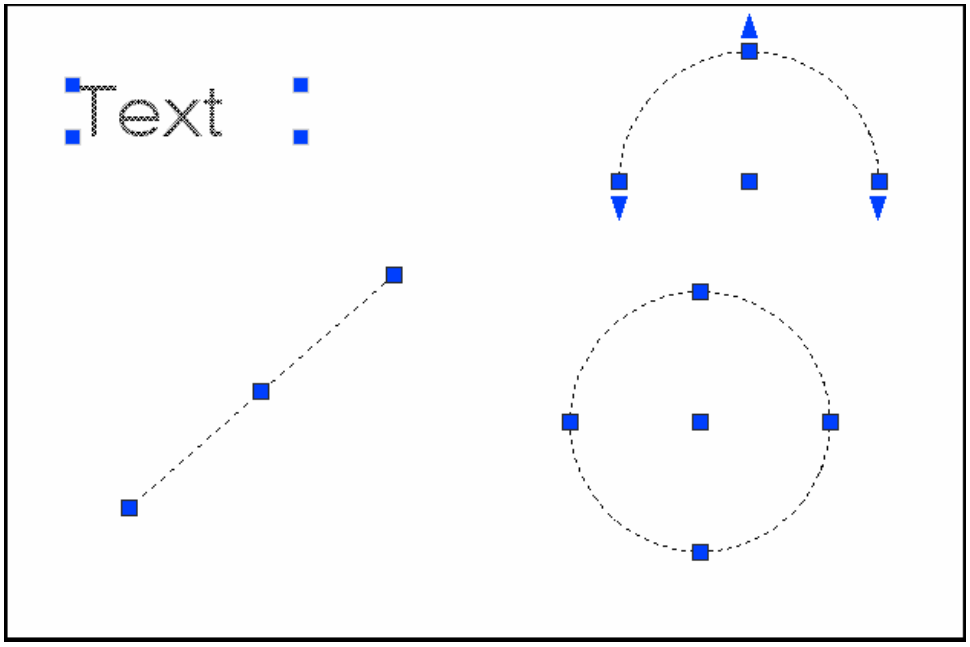
Entity Grips

Entity grips allow AutoCAD drawings to be edited in an entirely new way. Without entering any edit commands, you can stretch, move, copy, rotate, scale, and mirror entities. You can also snap to geometric features such as endpoints, midpoints, centers, quadrants without entering object snaps.

Grips are the small squares that appear when objects are selected with the crosshairs directly from the command prompt.

Unselected Grip

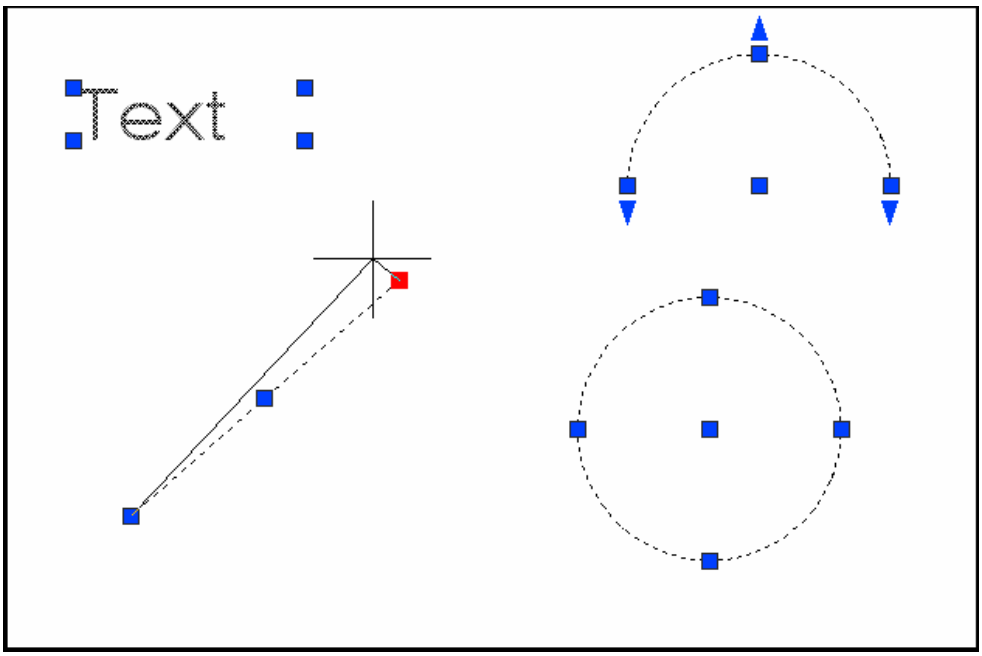
An unselected grip is one that has not yet been picked with the cursor, but is an item in the current selection set (it is highlighted). Pick an object to see grips.



Selected Grip

A selected grip is the grip box that you select with the cursor to define the base point to edit from. It has a solid filled color and is the grip location that editing is done from.

The red grip at the line endpoint is the selected grip



Cancelling Grips

1. **Press** ESC to clear GRIPS.

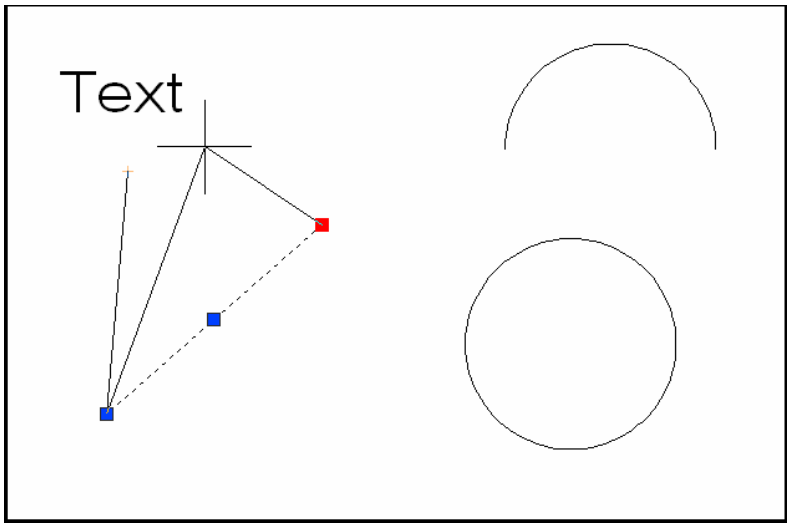
TIP: If grips are visible on an object, pressing the DEL key or ERASE will delete the selected object.

How To Use Grips 22.2

1. **Pick** The objects you want to edit.
2. **Pick** One of the grips to use as the base grip.
3. **Press** the SPACE BAR, or RIGHT MOUSE BUTTON to cycle through the grip modes.

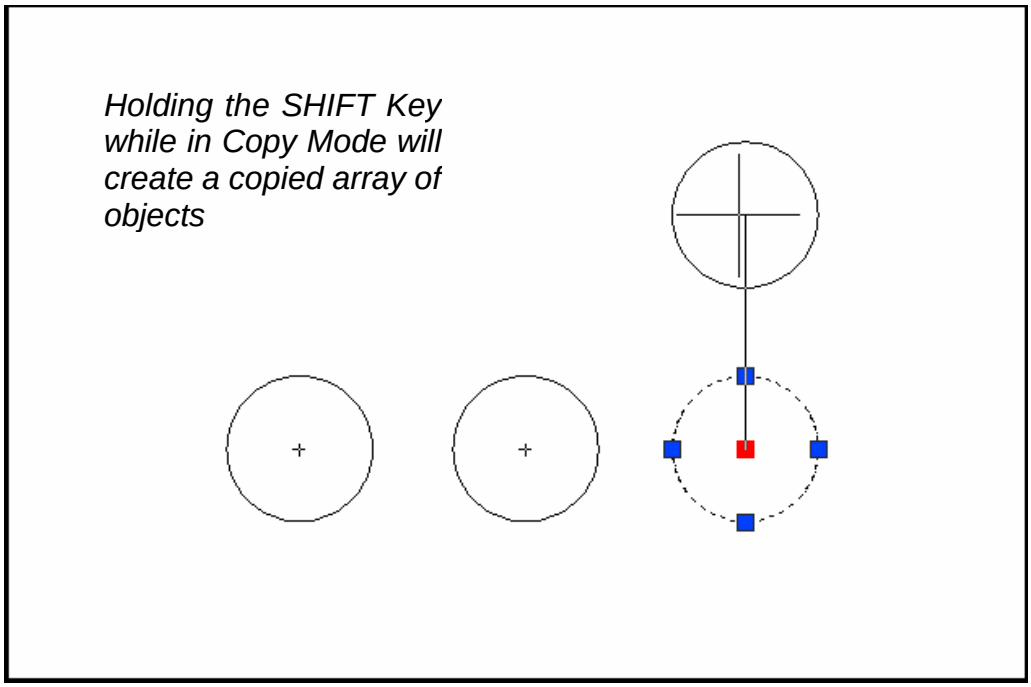
or

4. **Type** The keyword for the mode you want: Stretch (ST) Stretch the objects. Move (MO) Move the objects. Rotate (RO) Rotate the objects. Scale (SC) Scale the objects. Mirror (MI) Mirror the objects.
5. **Drag** The mouse to perform the operation.
6. **Type** C to create a new copy of the selection set enter C.
7. **Type** X to exit Grip mode.



Copy Multiple with Grips 22.3

If you use the COPY option with any one of the edit commands, a temporary auxiliary snap grid is created. To invoke the grid, hold the SHIFT key after specifying the location of the first copy. AutoCAD then uses the X and Y offsets from the original entity to define the snap, grid, and rotation of the remaining entities.



Grips Settings (DDGRIPS Command) 22.4

1. **Choose** Tools,Options...

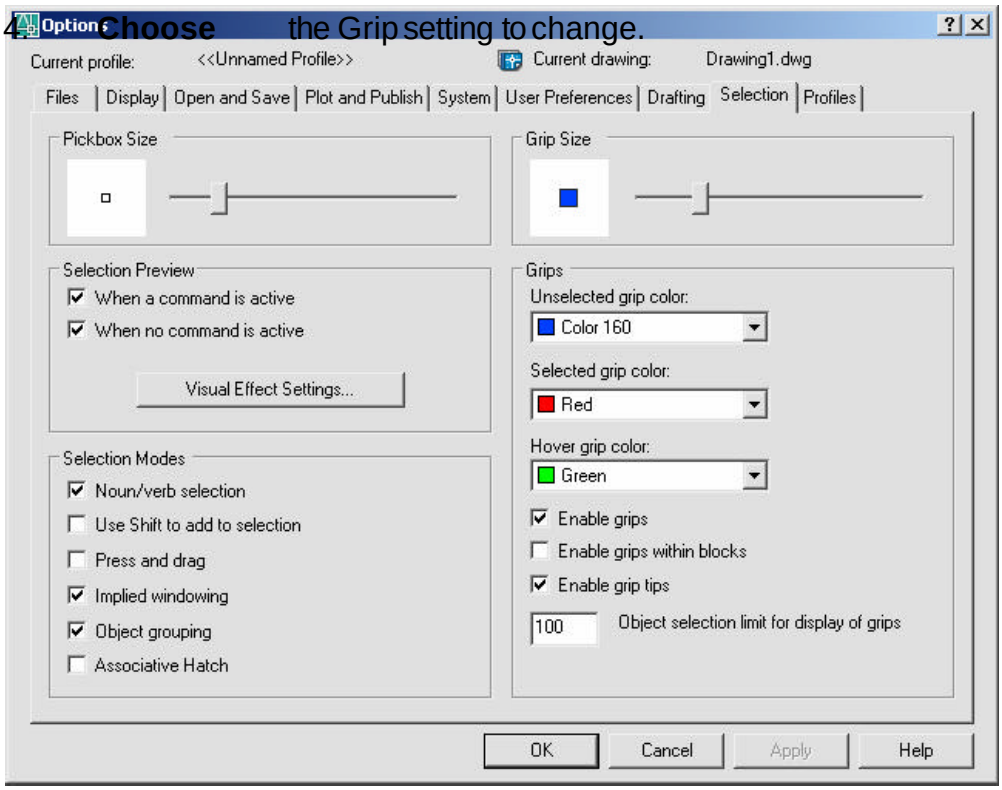
or

2. **Type** DDGRIPS at the command prompt.

   Command: **DDGRIPS**

3. **Choose** the Selection tab from the dialog box.

4. **Choose** the Grip setting to change.



Enable Grips

Enables the display of grips. AutoCAD stores this setting in the GRIPS system variable.

Enable Grips Within Blocks

Enables the display of grips on objects within blocks. If you disable this setting (but have Enable Grips selected), blocks are assigned one grip at their insertion point. Disable this setting to work on blocks with many objects. AutoCAD stores this setting in the GRIPBLOCK system variable.

Unselected

Sets the color of unselected (unfilled) grips. Choosing this button displays the Select Color dialog box, in which you set the grip color. AutoCAD stores the color in the GRIPCOLOR system variable.

Selected

Sets the color of selected (filled) grips. Choosing this button displays the Select Color dialog, in which you set the grip color. AutoCAD stores the color in the GRIPHOT system variable.

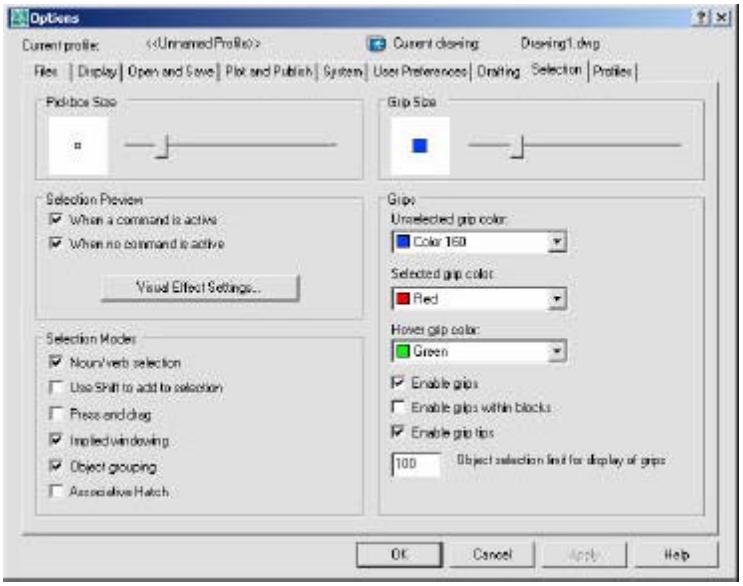
Grip Size

Changes the size of grips. To adjust the size of grips, move the slider box left or right. AutoCAD stores the pixel size (1-255) of the grips in the GRIPSIZE system variable. Changes the size of grips. To adjust the size of grips, move the slider box left or right. AutoCAD stores the pixel size (1-255) of the grips in the GRIPSIZE system variable.

Chapter 23
Advanced Selection Commands

Selection Modes 23.1

1. **Choose** Tools, Options...
2. **Choose** the Selection TAB from the following dialog.



3. **Change** the settings as desired.

Noun/Verb Selection

Allows you to select an object before starting a command. The command affects the previously selected object or objects. You can also set this option by using the PICKFIRST system variable.

Use Shift to Add to Selection

Adds or removes an object to the selection set when you press SHIFT and select an object.

Press and Drag

Draws a selection window by selecting a point and dragging the pointing device to a second point.

Implied Windowing

Initiates the drawing of a selection window when you select a point outside an object.

Object Grouping

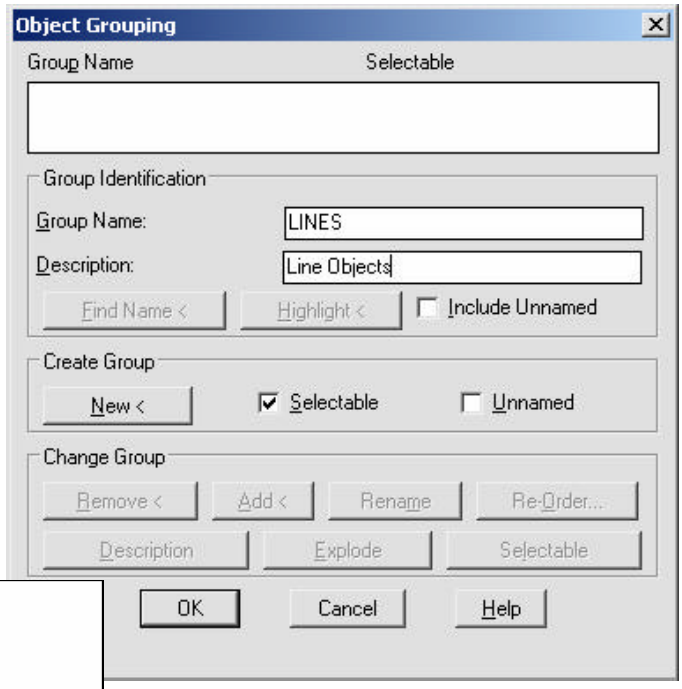
Selects all objects in a group when you select one object in that group. With GROUP you can create and name a set of objects for selection.

Associative Hatch

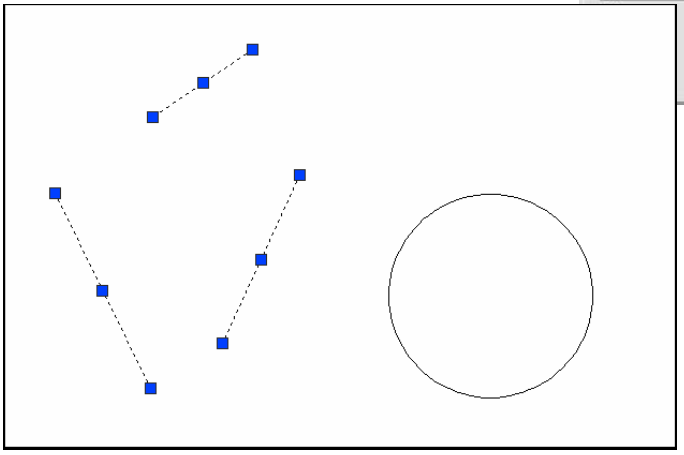
Determines which objects are selected when you select an associative hatch. If this option is selected, boundary objects are also selected when you select an associative hatch.

Groups 23.2

1. **Type** GROUP at the command prompt.
   Command: **GROUP**
2. **Type** a name for a new group.
3. **Choose** the New button under Create Group.
4. **Pick** objects to be included in the group.
5. **Press** ENTER when done choosing objects.
6. **Pick** OK.



Lines in a group



Turning Groups ON/OFF

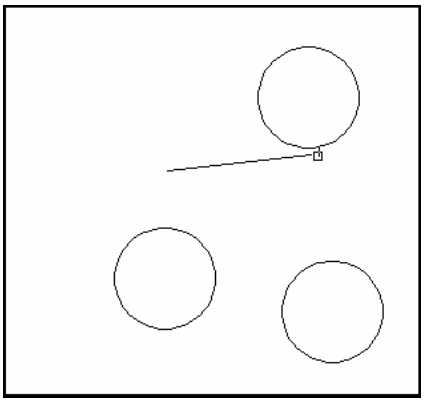
At any time, toggle group selection on and off by pressing either CTRL+H or SHIFT+CTRL+A.

Object Selection Cycling 23.3

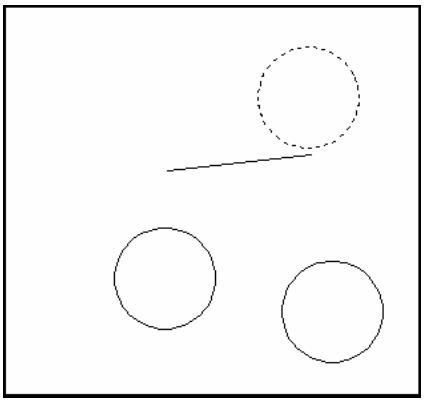
It is difficult to select objects that are close together or lie directly on top of one another. The following example shows two lines and a circle that all lie within the selection pickbox

1. **Press** the CTRL key before choosing objects at the Select Objects prompt.
2. **Pick** repeatedly in the area where multiple objects are located. AutoCAD will cycle through all objects that were touching the pickbox.
3. **Press** ENTER when the desired object highlights.
4. **Press** ENTER again.

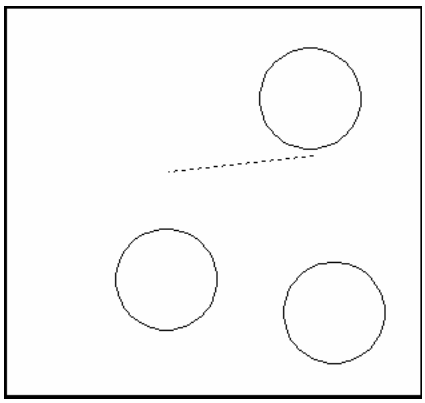
Objects touching pickbox



First object selected highlights

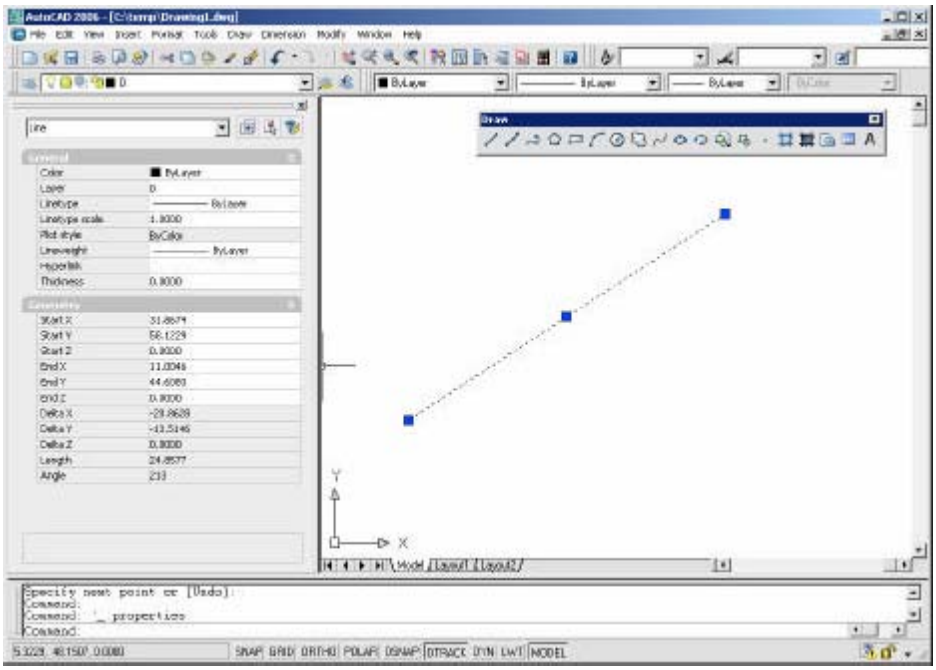


Second object selected



Double-Click Edit 23.4

1. **Double-Click** an object to edit.



Draw Order 23.5

1. Choose Tools, DisplayOrder.

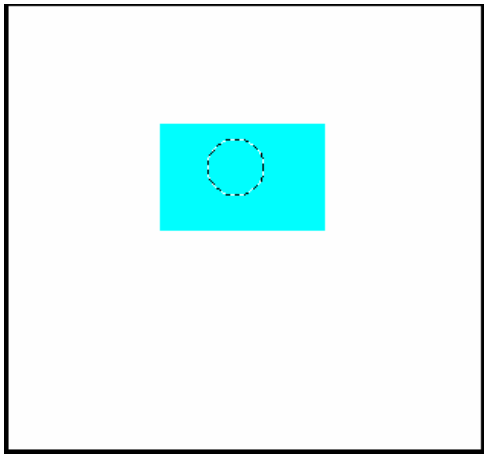
or

2. Click the Draworder Icon from the Modify II Toolbar.

or

3. Type DRAWORDER at the command prompt.
   Command: **DRAWORDER**

   Select objects: pick an object.



Object Filters 23.6

1. **Type** FILTER at the Command prompt.
2. **Select** Line in the Object Selection Filters dialog box under Select Filter.
3. **Choose** Add to List.
4. **Choose** Apply.
5. **Type** ALL at the Select Objects prompt or select a window.

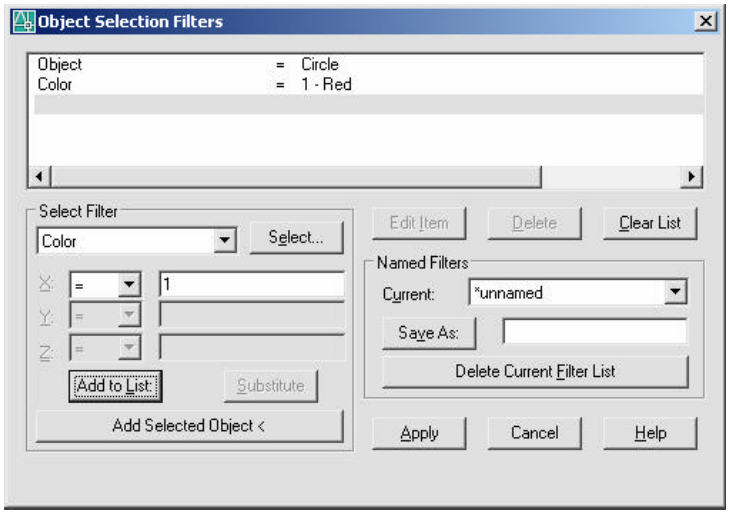
   Command:

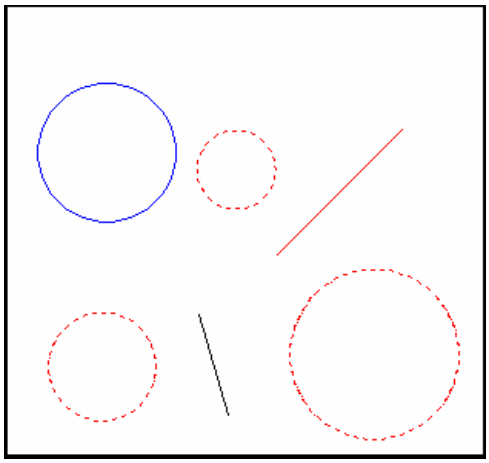
   FILTER Applying filter to selection.

   Select objects: **all**

   6 found

   3 were filtered out.

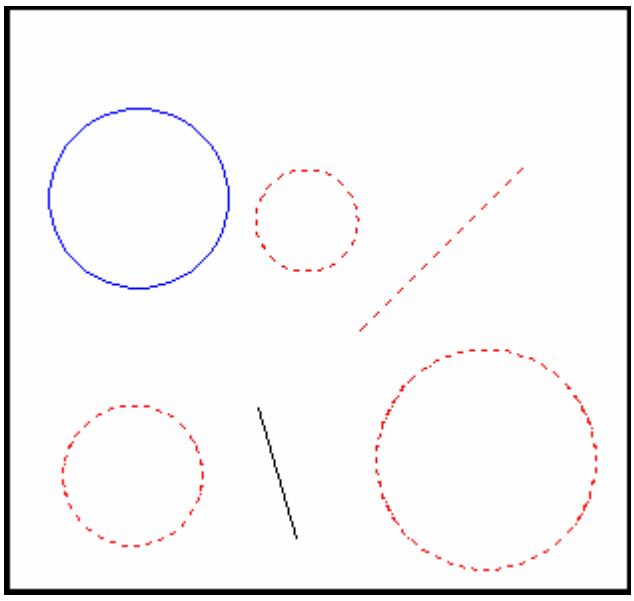


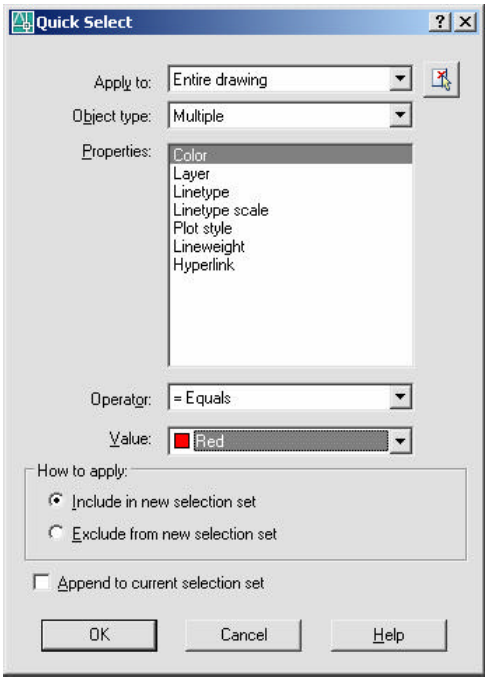


Quick Select 23.7

1. **Type** QSELECT at the command prompt.
   Command: **QSELECT**
2. **Enter** the selection criteria (i.e. TEXT HEIGHT < .5)
3. **Choose** OK.

TIP: AutoCAD puts those object(s) into a selection set. Use the "P" previous option to select these objects.



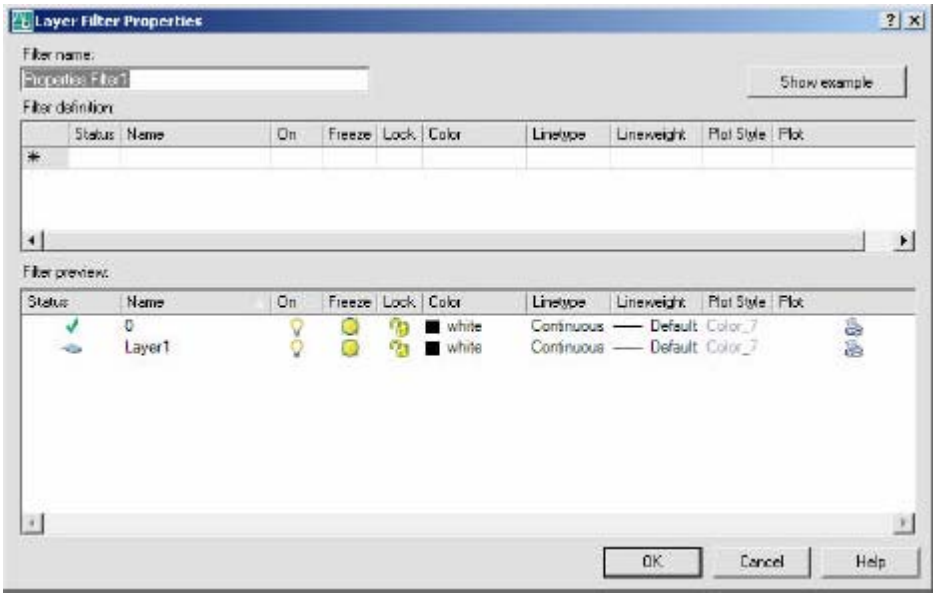


Layer Filters 23.8

1. **Choose** the layer dialog box.

2. **Choose** the three dots (...) to invoke the filter dialog for layers.

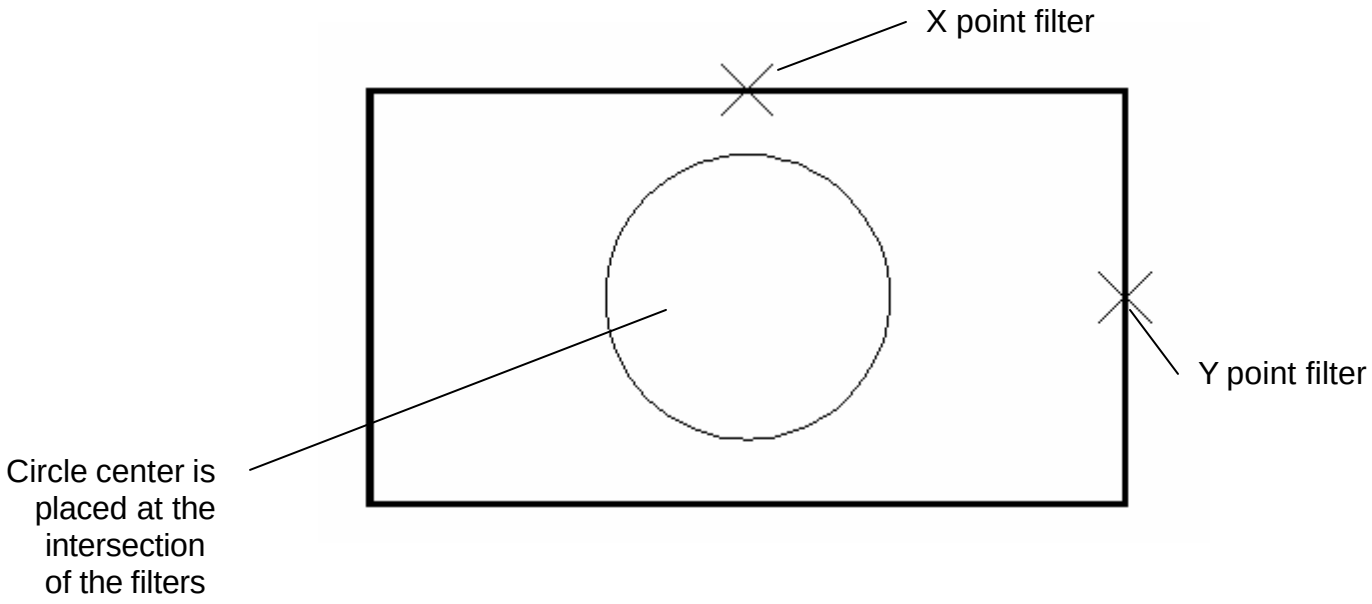
3. **Choose** the type of filter you would like to use (e.g. layer name = 1st*)



Point Filters 23.9

AutoCAD point filters allow the user to specify one coordinate, such as the X, with one pick and a second coordinate, such as the Y, with another pick. The point filters are .X, .Y, .XY,.XZ and .YZ. Only .X and .Y are used for two dimensional drawings.

1. **Type** A command that asks for a point.
   Command:**CIRCLE**
2. **Type** .X when AutoCAD asks for a point.
   3P/2P/TTR/<center point>: **.X**
3. **Pick** The point to filter (HINT: use osnaps)
   of **MID of point on x axis**
4. **Pick** The next point to filter
   (need YZ)
   MID of point on y axis
5. **Pick** A diameter or radius
   Diameter/<radius>: Pick or type a diameter



Chapter 24
External References

External Reference Files Overview

Attaches, overlays, lists, binds, detaches, reloads, unloads, renames, and modifies paths to external references (xrefs) in the current (or host) drawing.

1. **Choose** Insert, ExternalReference...

or

2. **Click** the Xref Iconfrom the Reference Toolbar. 

or

3. **Type** XREF at the command Prompt.

Command: **XREF**

The following are Xref characteristics:

-An external reference file is known as an “Xref”.

-Current drawing contains only a “pointer”, the path and filename, to the Xref.

-The current drawing does not increase much in size when it contains an Xref.

-The Xref is reloaded each time the current drawing is loaded, thus always showing the latest revision of the Xref.

-Xrefs import their linetypes, layers, text styles, dimstyles, views, ucs’s, vports, and blocks into their current drawing.

-Each Xref named object is prefixed with the xref drawing name and a pipe “|” symbol.

(i.e. HOUSE|A-WALL for drawing house.dwg and layer a-wall)

- Xdep stands for external reference dependent objects.

- Xref’s layers can be turned on/off in the current drawing.

- Layer zero(0) resides on layer zero(0) of the current drawing.

- Xrefs can be bound to the current drawing, in which case they become blocks.

-Xref layers cannot be made current the the drawing they are xreferenced into.

- Xrefs can be snapped to.

-Xref entities cannot be individually modified in the current drawing.

-Xrefs can be plotted.

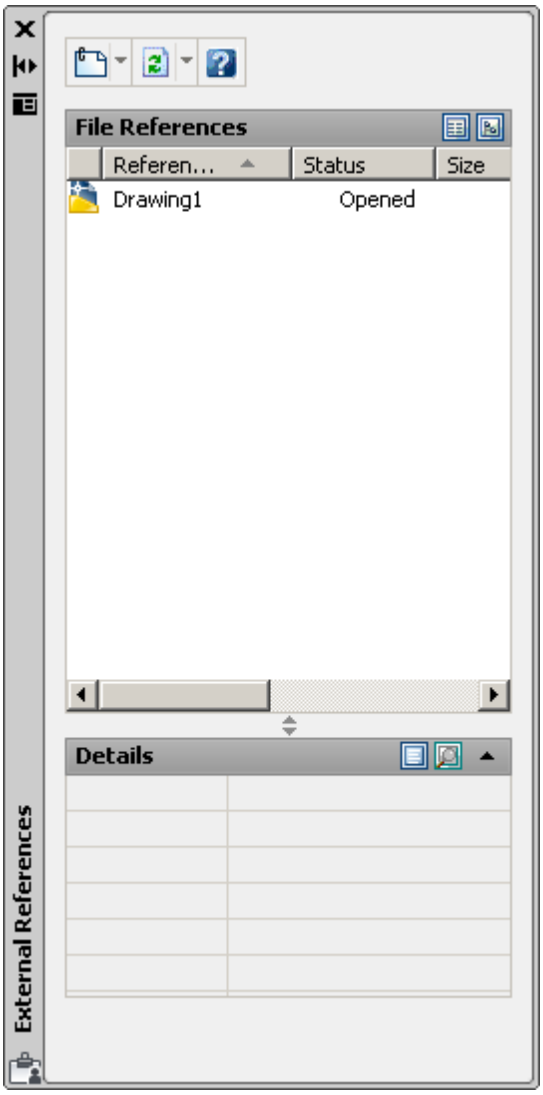
-Xrefs can be detatched from the current drawing and will disappear.

-The current drawing pointer, file and pathname can be changed.

-Xrefs can be re-loaded during the current drawing session.

-Xrefs can be nested.

-Xrefs can be clipped to show parts of the reference files.



Attaching Xrefs 24.1

Attaches, overlays, lists, binds, detaches, reloads, unloads, renames, and modifies paths to external references (xrefs) in the current (or host) drawing.

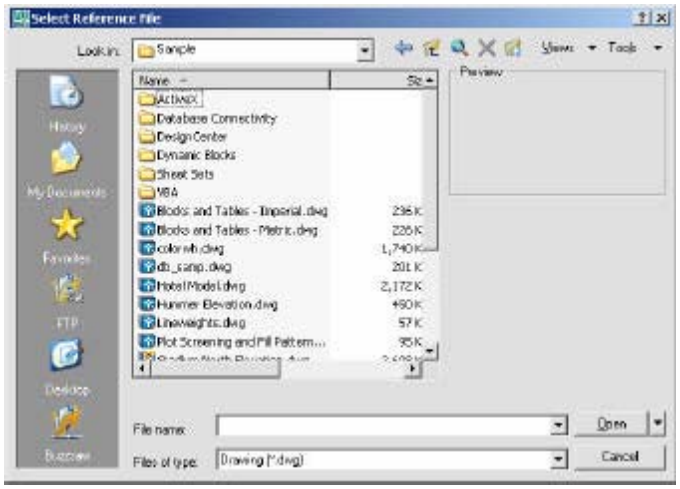
1. **Choose** Insert, ExternalReference

or

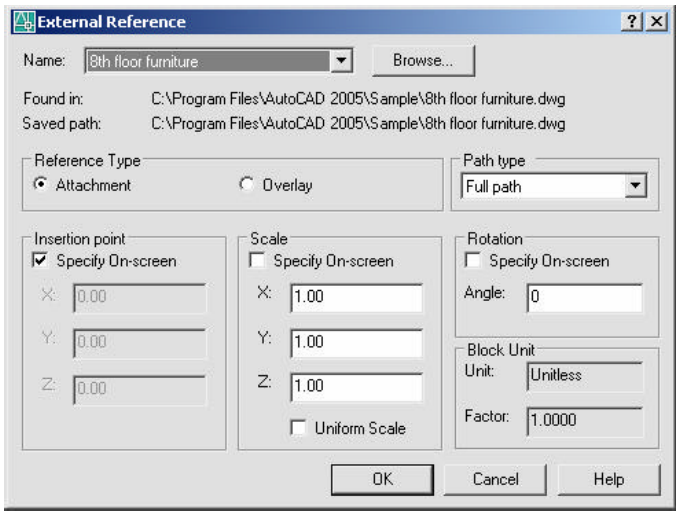
2. **Type** XATTACH at the command prompt.

or

3. **Click** the Xref Attach Icon from the Reference Toolbar. 
4. **Choose** a drawing name to attach.



5. **Specify** the insertion parameters.



Xref Layers 24.2

Layer Dialog Box

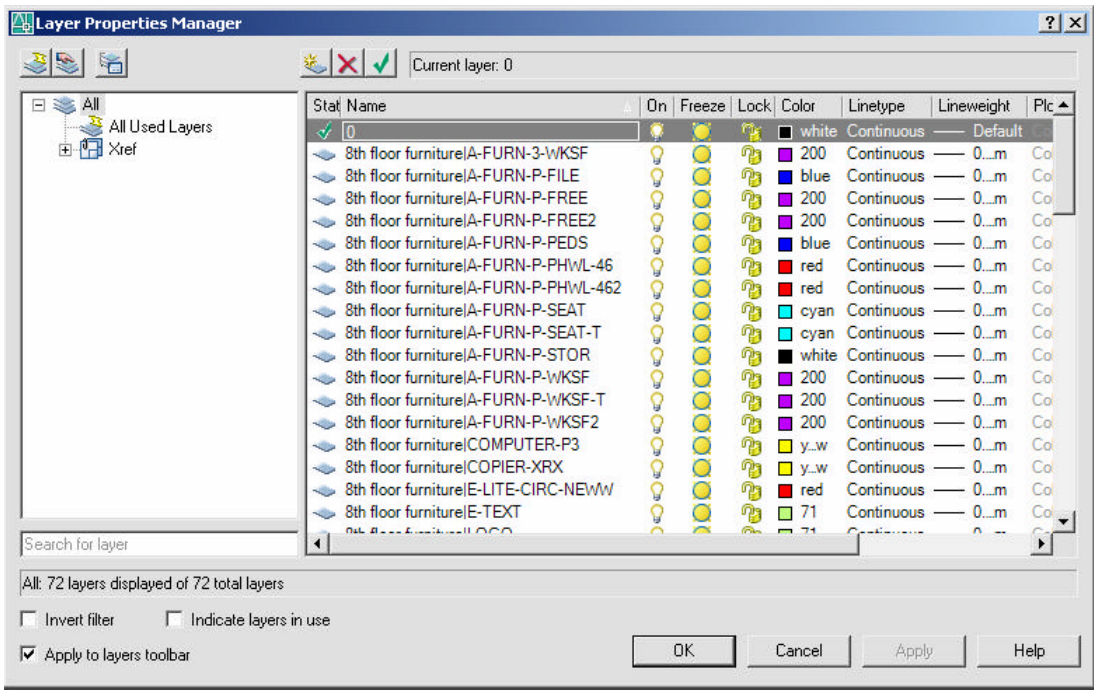
-Each Xref named object is prefixed with the xref drawing name and a pipe "|" symbol. (i.e. HOUSE|A-WALL for drawing house.dwg and layer a-wall)

-Xdep stands for external reference dependent objects.

-Xref's layers can be turned on/off in the current drawing.

-Layer zero(0) resides on layer zero(0) of the current drawing.

-Xref layers cannot be made current the the drawing they are xreferenced into.



Xclip 24.3

Defines an xref or block clipping boundary and sets the front or back clipping planes.

1. **Choose** Modify, Clip, XRef

   or

2. **Click** the Xclip Iconfrom the Reference Toolbar. 

   or

3. **Type** XCLIP at the command prompt.

   Command: **XCLIP** Select objects:

   Other corner: 1 found Select objects:

   pick reference file

   ON/OFF/Clipdepth/Delete/generate Polyline/

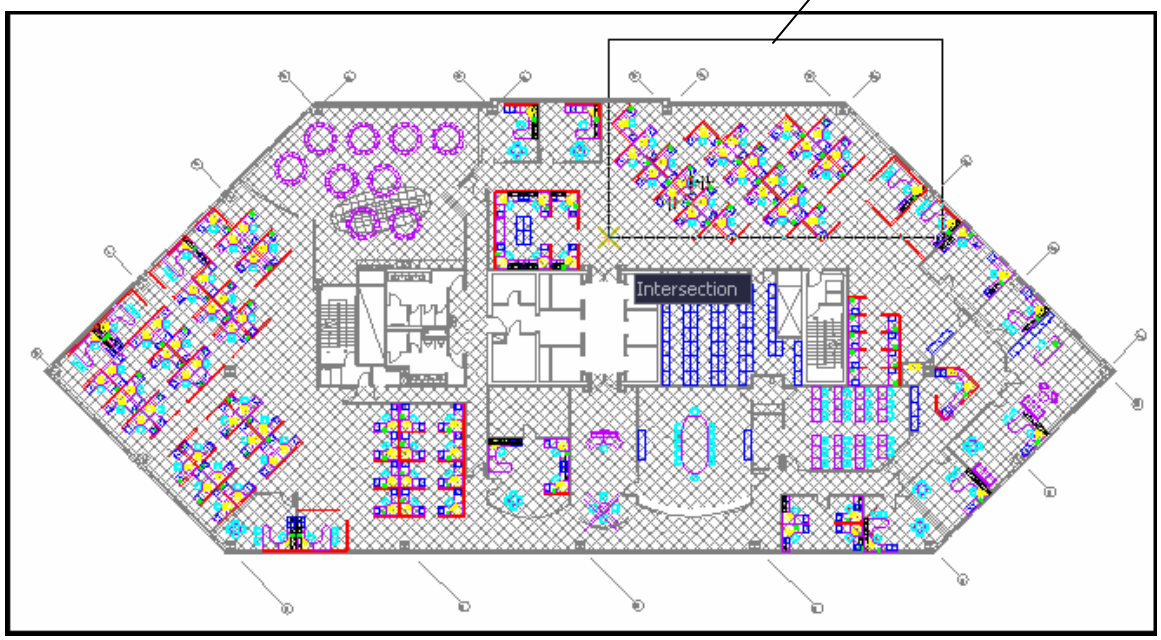
   <New boundary>:

   Specifyclipping boundary:

   Select polyline/Polygonal/<Rectangular>: First corner:

   Other corner: **pick corners**

Defining a Clipping Boundary



Xclipframe 24.4

Controls visibility of xref clipping boundaries.

1. **Choose** Modify, Object, ExternalReference.

or

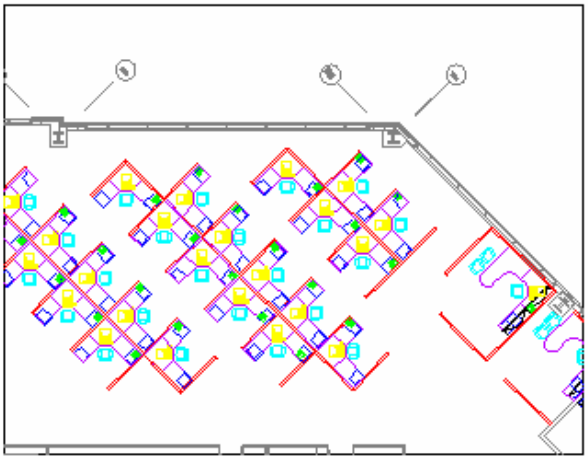
2. **Type** XCLIPFRAME at the command prompt.

Command: **XCLIPFRAME**

0 Clipping boundary is not visible

1 Clipping boundary is visible

Turning xclip frame on/off

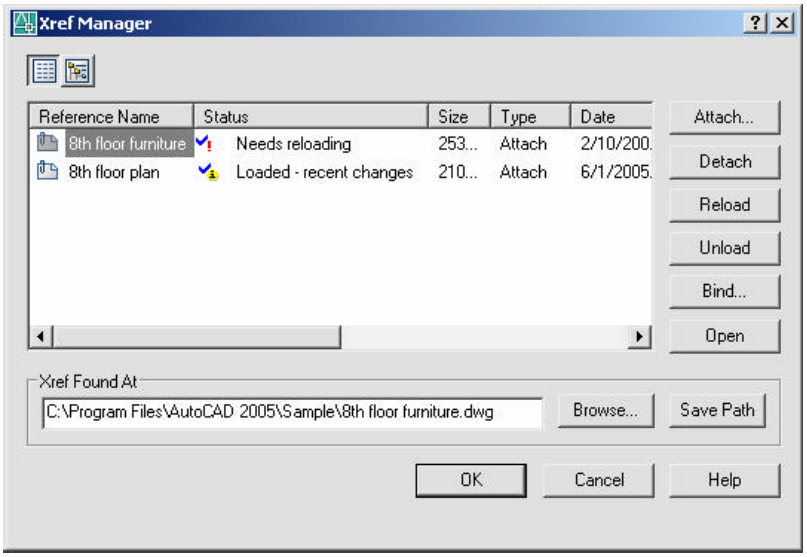


Binding an Xref 24.5

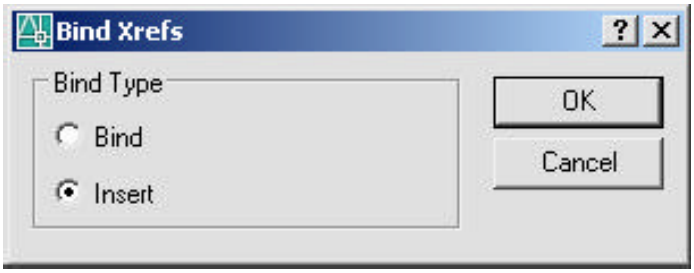
Converts Xreference files to block definitions in the current drawing. Associ- ated objects change names (i.e. the layer called HOUSE|A-WALL becomes HOUSE0A-WALL). Binding looses the connection to the referenced file. Xrefs can also be inserted like Wblocks.

To Bind an Xref:

1. **Choose** Insert,Xref Manager
2. **Choose** an xref name.
3. **Choose** the Bind option.



4. **Choose** Bind or Insert.
5. **Choose** OK.



Xbind 24.6

The XBIND command is used to bind layers, blocks, linetypes, styles, and dimstyles of an attached xref without binding the entire xref.

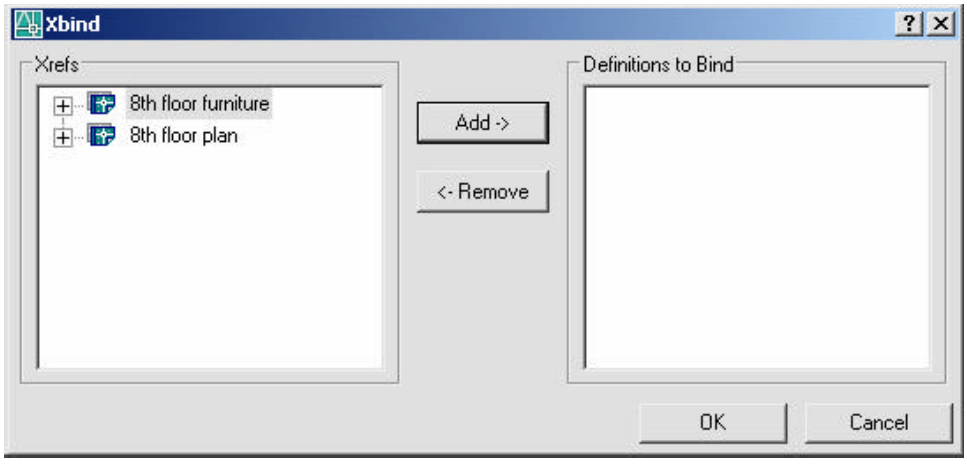
1. **Choose** Modify, Object, External Reference, Bind...

   or

2. **Click** the Xbind Icon from the Reference Toolbar. 

3. **Type** XBIND at the command prompt.

   Command: **XBIND**
   Block / Dimstyle / LAyer / LType / Style:



Editing Xrefs (Xref Manager) 24.7

Detaching Xrefs

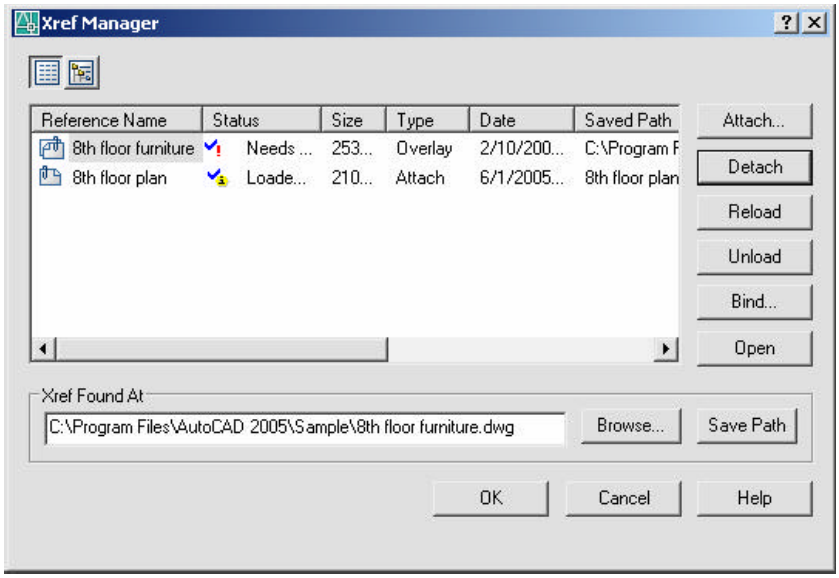
Detaches one or more xrefs from your drawing, erasing all instances of a specified xref and marking the xref definition for deletion from the symbol table.

1. **Choose** a drawing name to detach from the Xref Dialog.
2. **Choose** the Detachoption.

Unload an Xref

Unloads one or more xrefs. Unloaded xrefs can be easily reloaded. Unlike detach,unloading does not remove the xref permanently. It merely sup- presses the display and regeneration of the xref definition, to help current session editing and improvement of performance.

1. **Choose** a drawing name to unload from the Xref Dialog.
2. **Choose** the Unload option.



Reload an Xref

Marks one or more xrefs for reloading. This option re-reads and displays the most recently saved version of the drawing.

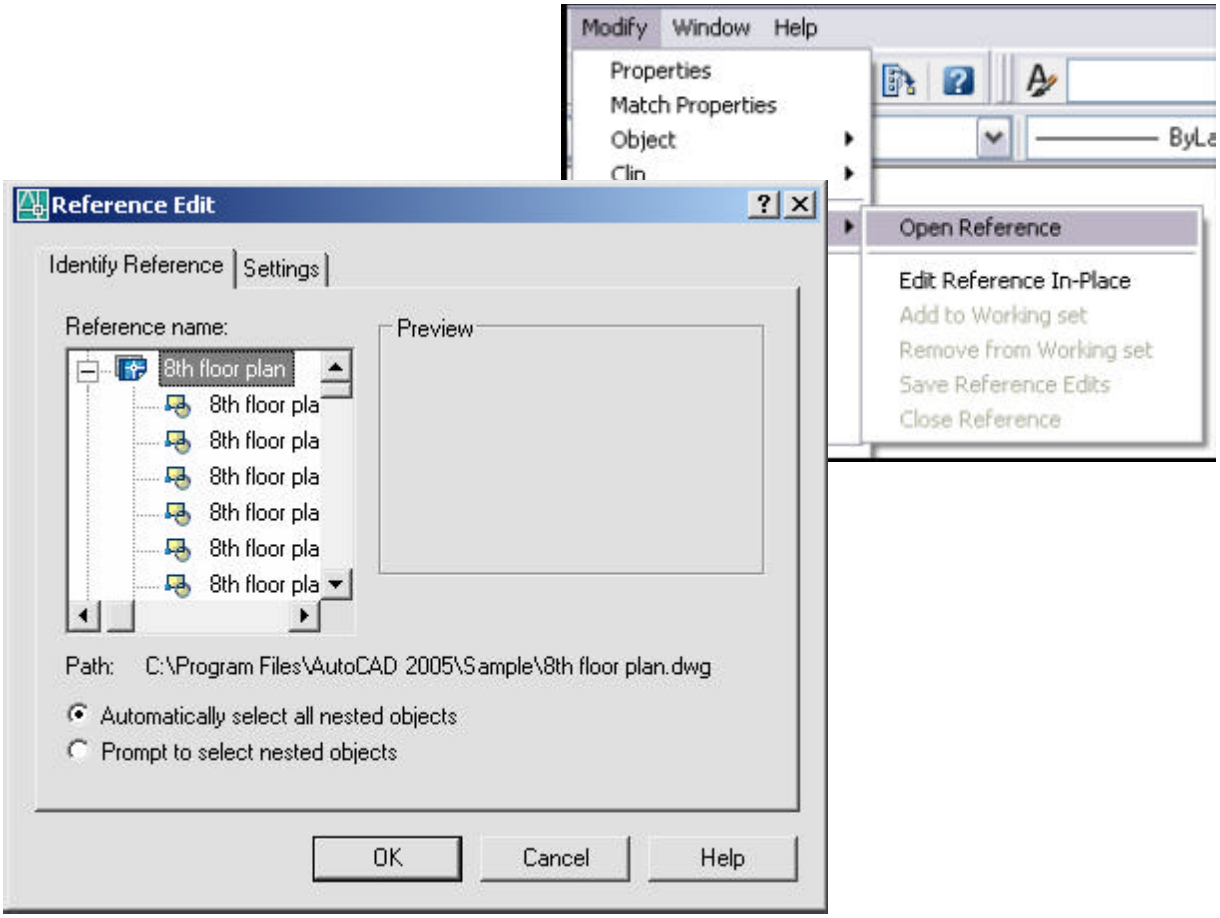
1. **Choose** a drawing name to reload from the Xref dialog.
2. **Choose** the Reload option.

Opening Xrefs to Edit

1. **Choose** Modify, Xref and Block Editing, Open Reference.

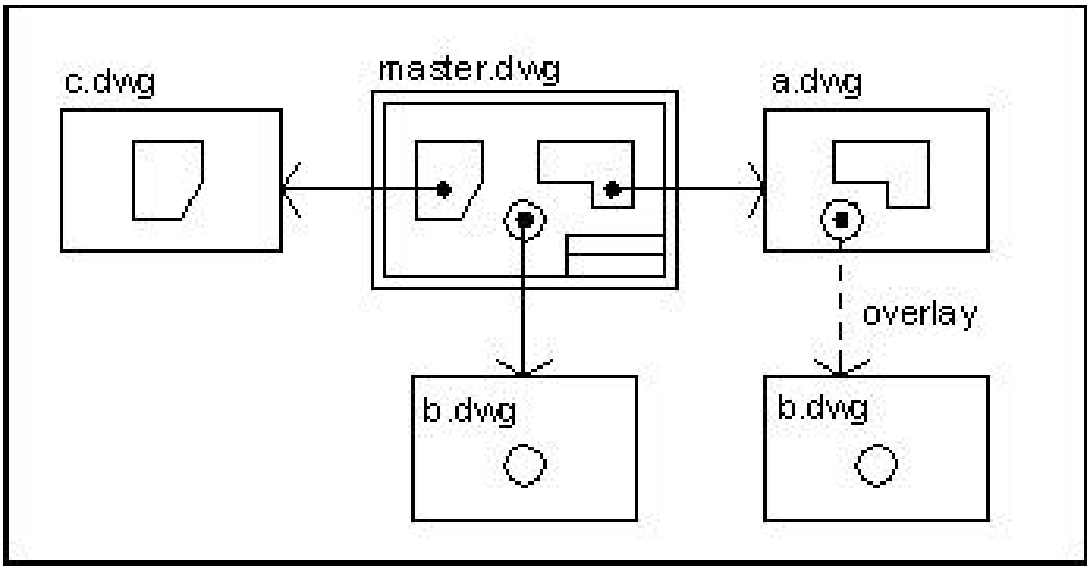
or

2. **Choose** Modify, Xref and Block Editing, Edit Xreference in Place.
3. **Edit** the objects as desired.
4. **Save** the edits with the following icons.



Overlay an Xref 24.8

Overlays are typically used when you need to view another drawing's geometry temporarily, but don't plan to plot using that data. In the following illustration, several people are working on drawings referenced by master.dwg. The person working on a.dwg needs to see the work being completed by the person working on b.dwg, but does not want to xref b.dwg because it would then appear twice in master.dwg. Instead, the person overlays b.dwg, which is not included when a.dwg is referenced by master.dwg.

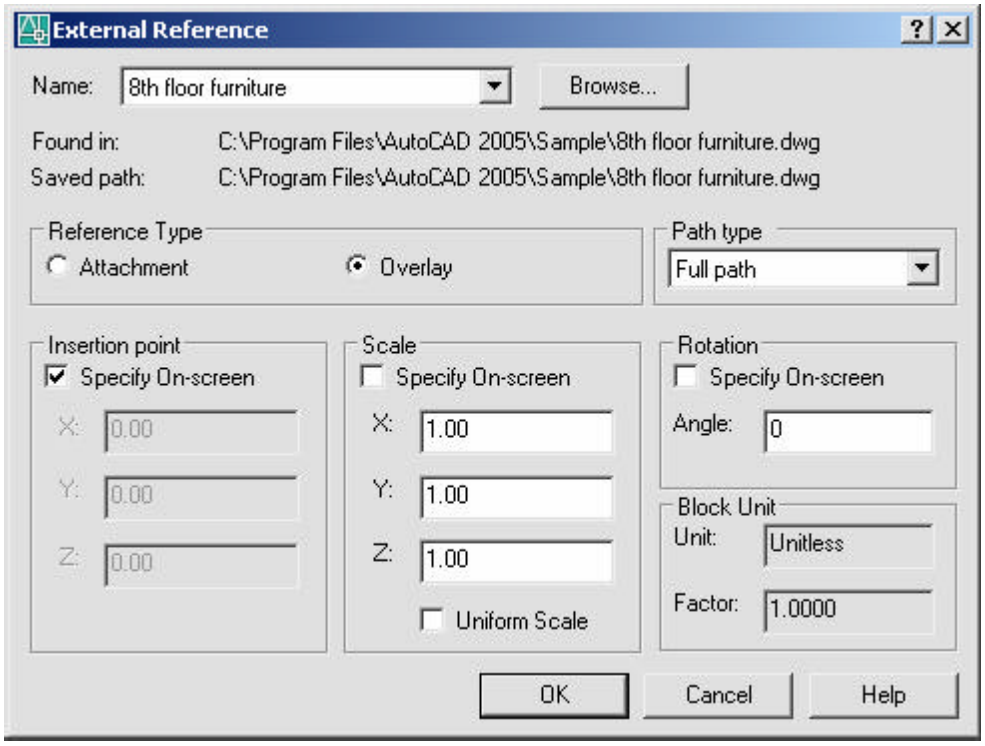


1. **Choose** the Attach option from the Xref Dialog.

or

2. **Click** the Xref Attach Icon from the Reference Toolbar.
3. **Choose** a drawing name to attach.
4. **Choose** Overlay in the Attach Xref dialog box under Reference Type.

5. **Specify** the insertionparameters.



Chapter 25
Raster Images

Inserting Images 25.1

1. **Choose** Insert, Raster Image...
2. **Choose** a raster image file to insert.
3. **Specify** a locationto insert the image.

or

4. **Type** IMAGEATTACHat the command prompt.

   Command: **IMAGEATTACH**

or

5. **Click** the Image iconfrom the Reference Toolbar. 

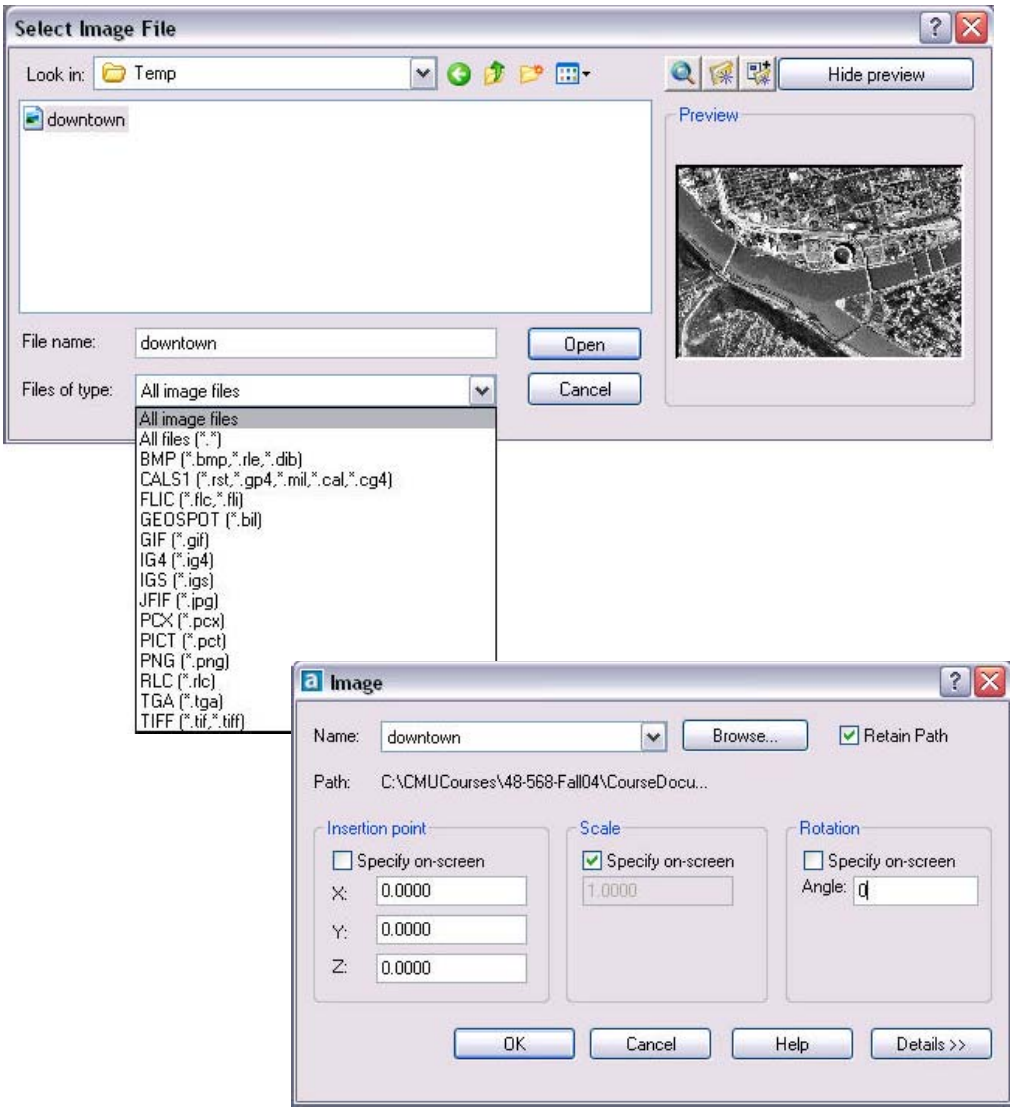


Image Appearance 25.2

Adjusting Image Appearance

1. **Choose** Modify, Object, Image,Adjust...
2. **Choose** a raster image file edit.
3. **Choose** options from the dialog box to adjust.

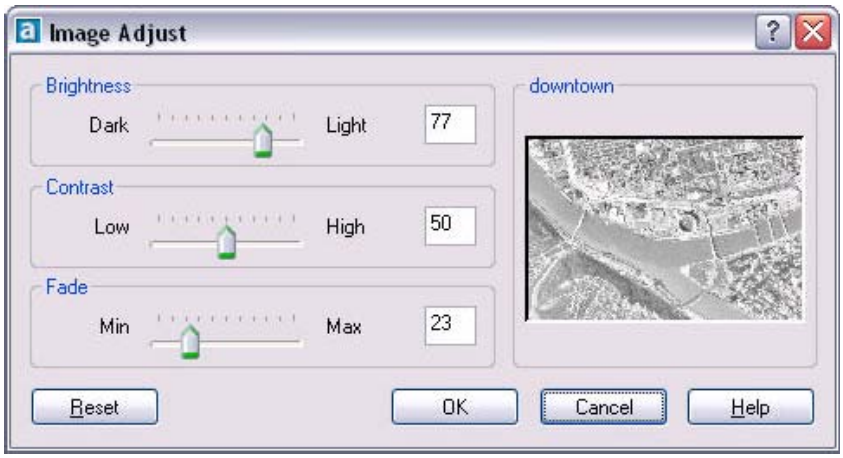


Image Transparency

1. **Choose** Modify, Object, Image, Transparency.
2. **Choose** a raster image file edit.
3. **Type** ON or OFF to turn an image's transparency on or off.

Erasing Images

1. **Click** once on the image to remove.
2. **Choose** Edit, Cut.

Imagequality

1. **Type** IMAGEQUALITY at the command prompt.
   Command: **imagequality**
   Enter image quality setting [High/Draft] <High>:

Clipping Images 25.3

1. **Choose** Modify, Clip, Image.

Chapter 26
Dimensioning

Linear Dimensions 26.1

1. **Choose** Dimension, Linear.

or

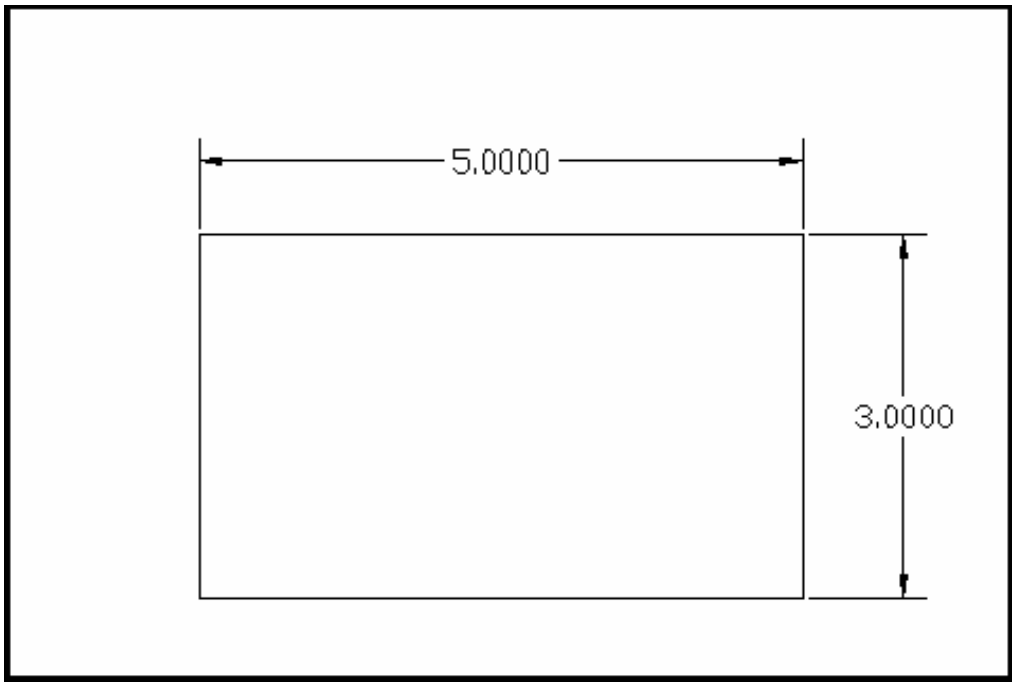
2. **Click** the Linear Dimension command from the toolbar.

or

3. **Type** DIM at the command prompt.

Command: **DIM**

Dim: HOR or VER



Aligned Dimensions 26.2

1. **Choose** Dimension, Aligned.

or

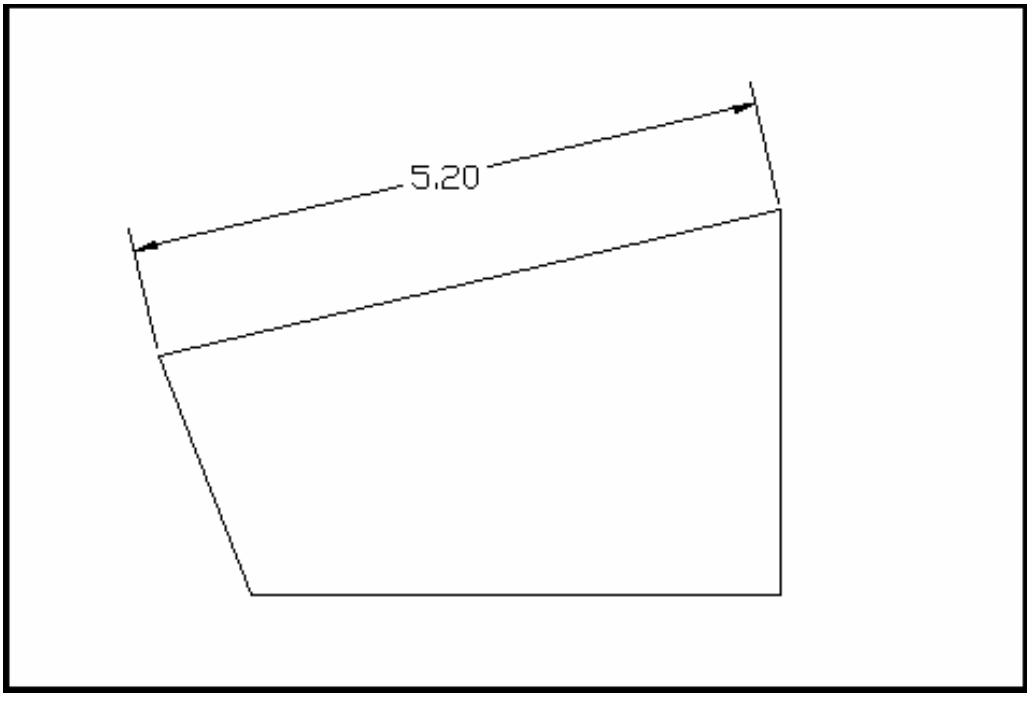
2. **Click** the Aligned Dimension command from the toolbar.

or

3. **Type** DIM at the command prompt.

Command: **DIM**

Dim: **ALIGNED**



Radial Dimensions 26.3

1. **Choose** Dimension, Radius or Diameter.

or

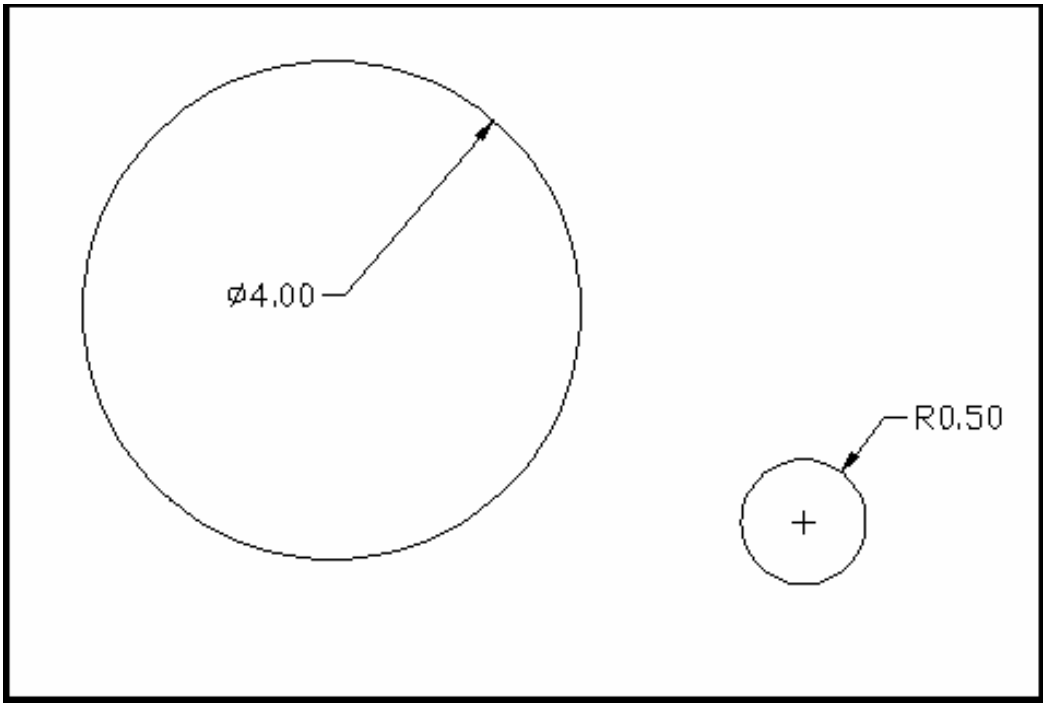
2. **Click** the Radial Dimensions command from the toolbar.

or

3. **Type** DIM at the command prompt.

Command: **DIM**

Dim: **RADIUS or DIAMETER**



Angular Dimensions 26.4

1. **Choose** Dimension, Angular.

or

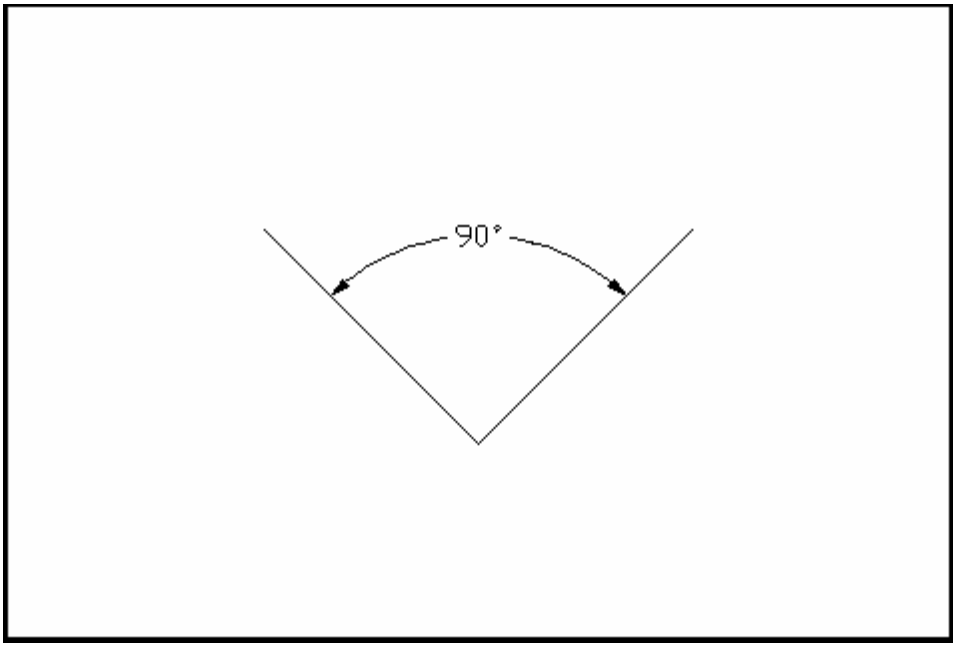
2. **Click** the Angular Dimensions command from the toolbar.

or

3. **Type** DIM at the command prompt.

Command: **DIM**

Dim: **ANGULAR**



Continued and Baseline Dimensions 26.5

1. **Choose** Dimension, Continue or Baseline.

or

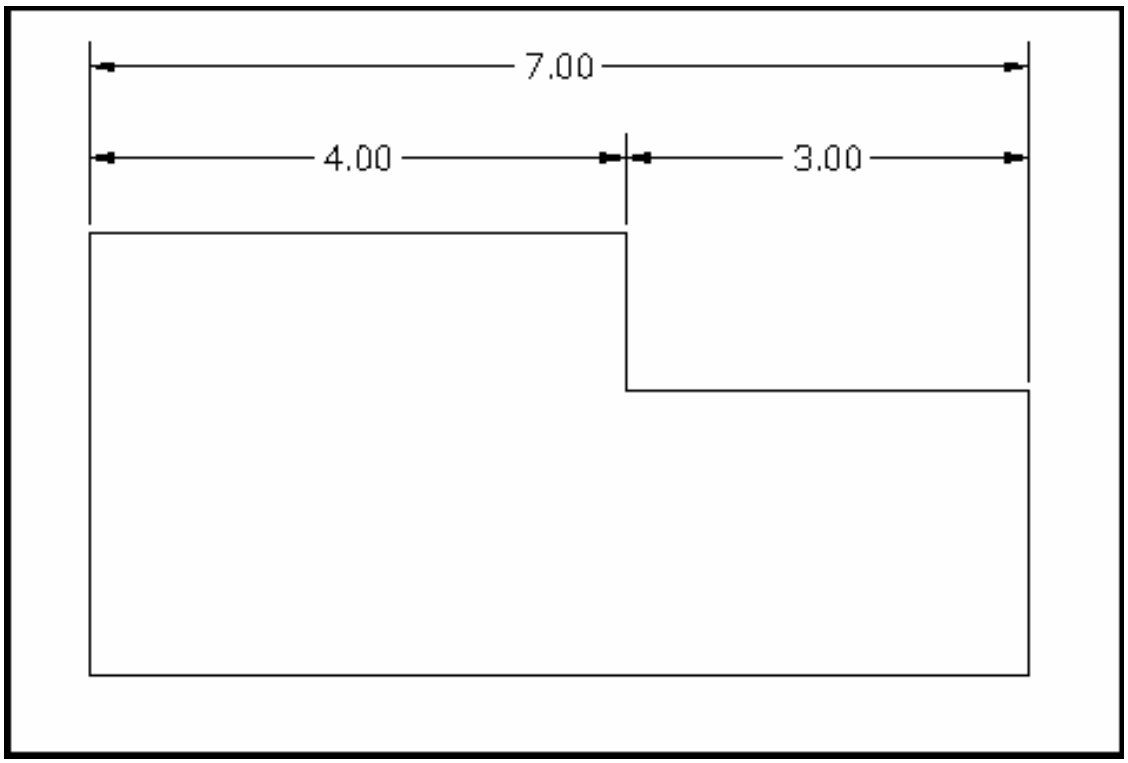
2. **Click** the Continue or Baseline Dimensions command from the toolbar.

or

3. **Type** DIM at the command prompt.

   Command: **DIM**

   Dim: **CONTINUE or BASELINE**



Leaders 26.6

1. **Choose** Dimension, Leader...

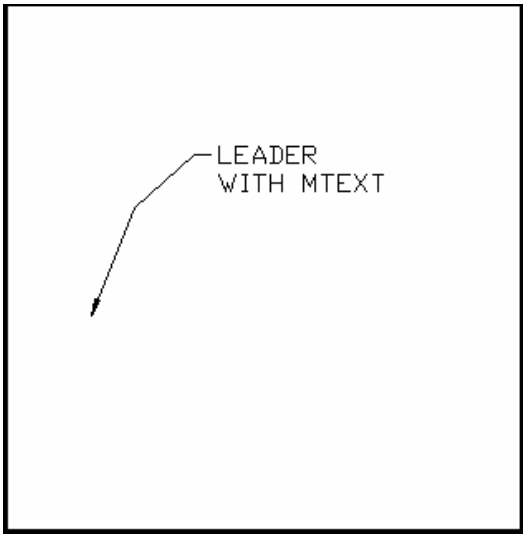
or

2. **Click** the Leader icon from the Dimension toolbar.

or

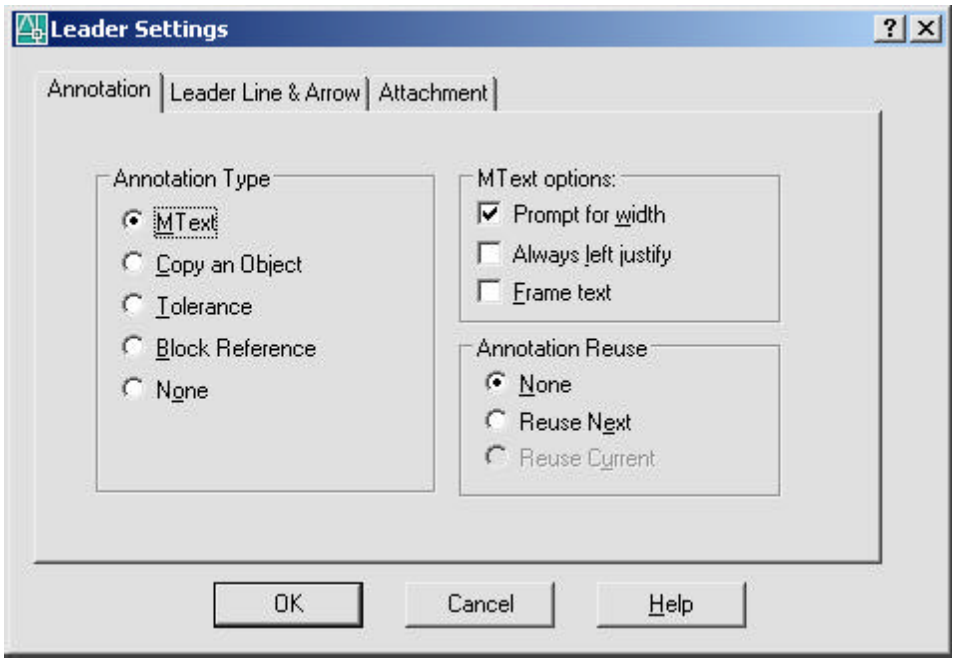
3. **Type** QLEADER at the command prompt.

Command: **QLEADER**



Leader Settings

1. **Type** QLEADER at the command prompt.
   Command: **QLEADER**
2. **Type** "S" at the QLEADER prompt to change the leader settings.
3. **Choose** a setting from the following dialog box.



Quick Dimensions 26.7

Quickly creates dimension arrangements from the geometry you select.

1. **Choose** Dimension, QDIM.

or

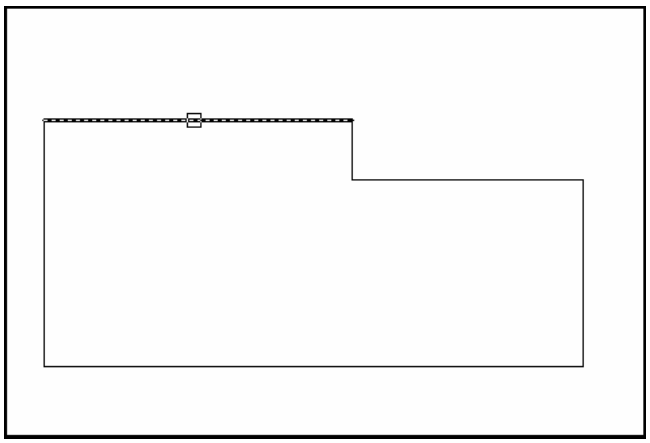
2. **Click** the Quick Dimension icon from the Dimensions toolbar.

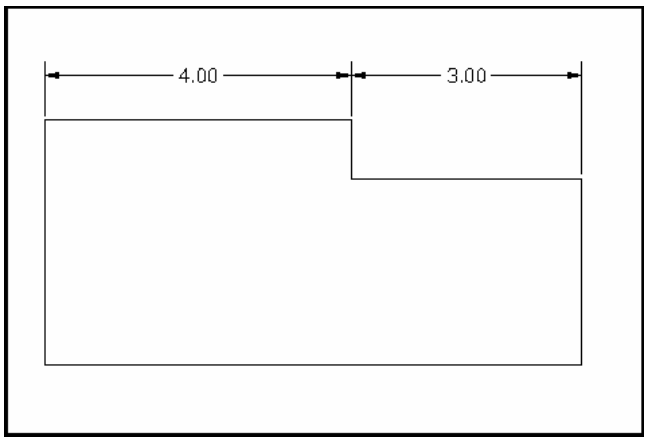
or

3. **Type** QDIM at the command prompt.

   Command: **QDIM**

4. **Pick** the objects to dimension.



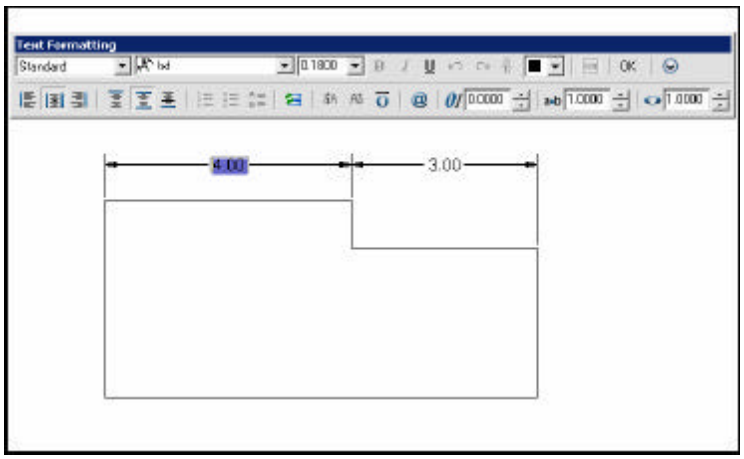


Modifying Dimensions 26.8

DDEDIT

1. **Choose** Modify, Object, Text.
2. **Choose** the dimension text to modify.

TIP: The actual dimension is placed in brackets <>. Text can be placed in front of or behind these brackets. If text is placed between the brackets, the dimension loses its associative properties.



Stretching Dimensions

1. **Choose** Modify, Stretch.
2. **Choose** a crossing window around the area to stretch. Be sure to include the dimension endpoints.

DIMTEDIT

Moves and rotates dimension text

1. **Choose** Dimension, Align Text. or
2. **Type** DIMTEDIT at the command prompt.

Command: **DIMTEDIT**

Select dimension: select object

Enter text location (Left / Right / Angle):

Dimension Edit Commands

HOMetext	Moves the Dimension text back to its home (default) position.
NEWtext	Modifies the text of the Dimensions.
Rotate	Rotates dimension text.
OBlique	Sets the obliquing angle of Dimension extension lines.
OVerride	Overrides a subset of the Dimension variable settings.
UPdate	Redraws the Dimensions as directed by the current settings of all dimensioning variables.

Ordinate Dimensions 26.9

1.	**Choose**	Dimension, Ordinate or
2.	Type	DIMORDINATE at the command prompt. Command: Dimordinate

Chapter 27
Dimension Styles

Creating Dimension Styles 27.1

1. **Choose** Format, Dimension Style...

or

2. **Choose** Dimension, Style.

or

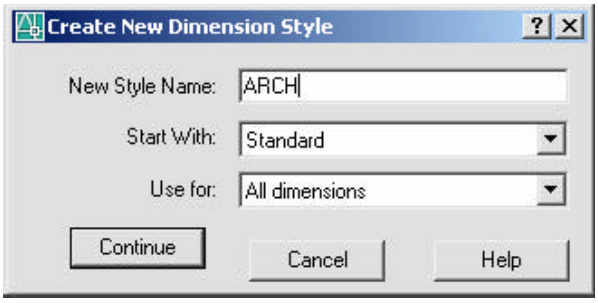
3. **Choose** Dimension Style icon from the Dimension Style toolbar.

4. **Type** DDIM at the command prompt

   Command:**DDIM**

5. **Choose** New... from the dialog box.

6. **Create** a new style from the existing styles.



7. **Click** the Continue button.

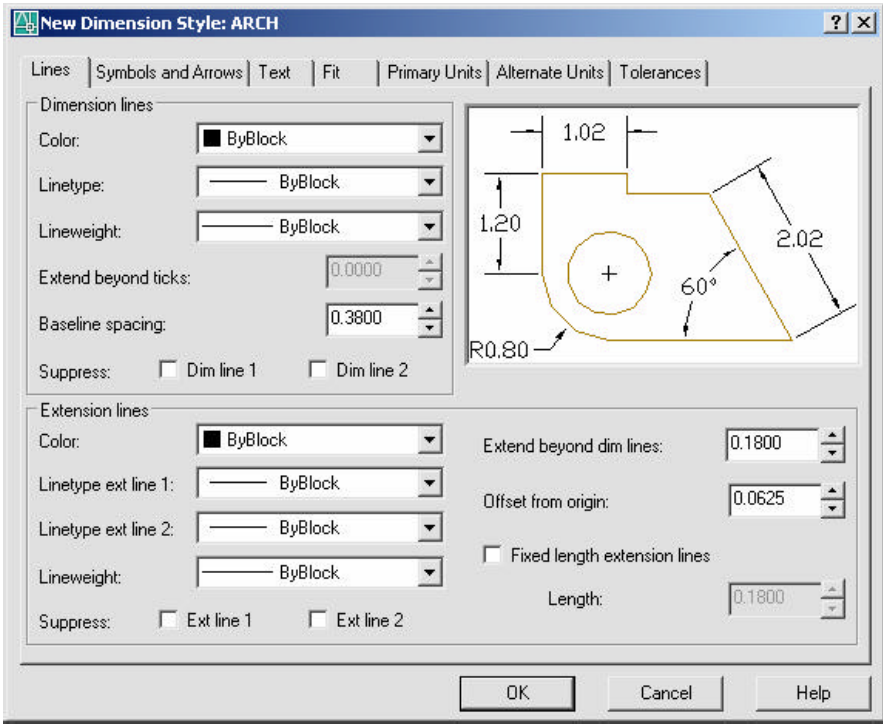
TIP:

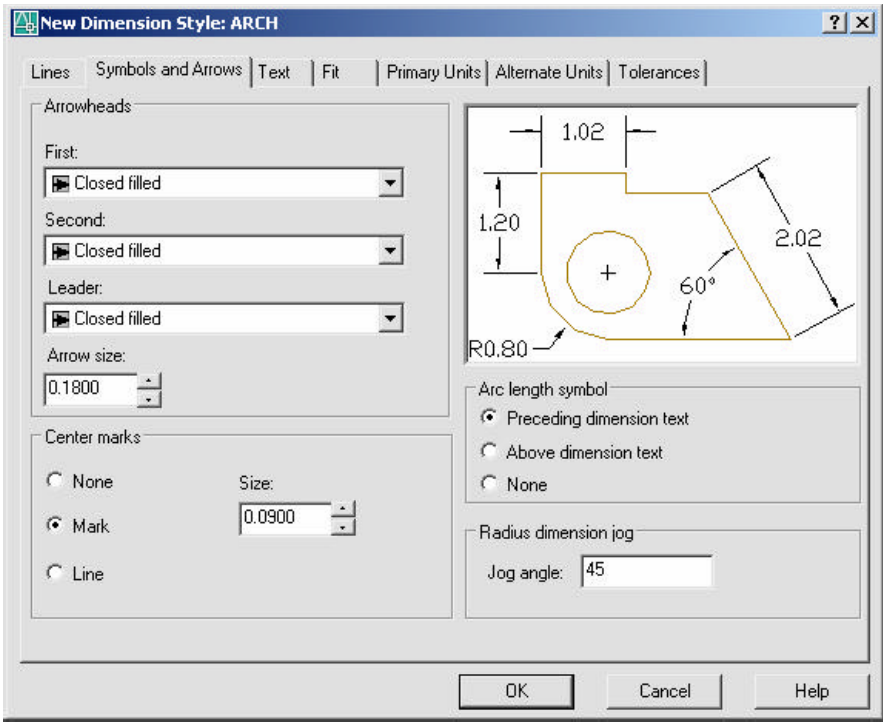
All dimension variables except for DIMSHO and DIMASO can be saved as a style.

Lines and Arrows 27.2

Edits Dimension Lines, Extension Lines, and Arrows.

1. **Pick** the Lines and Arrows tab from the Dimension Variables and Styles dialog box.

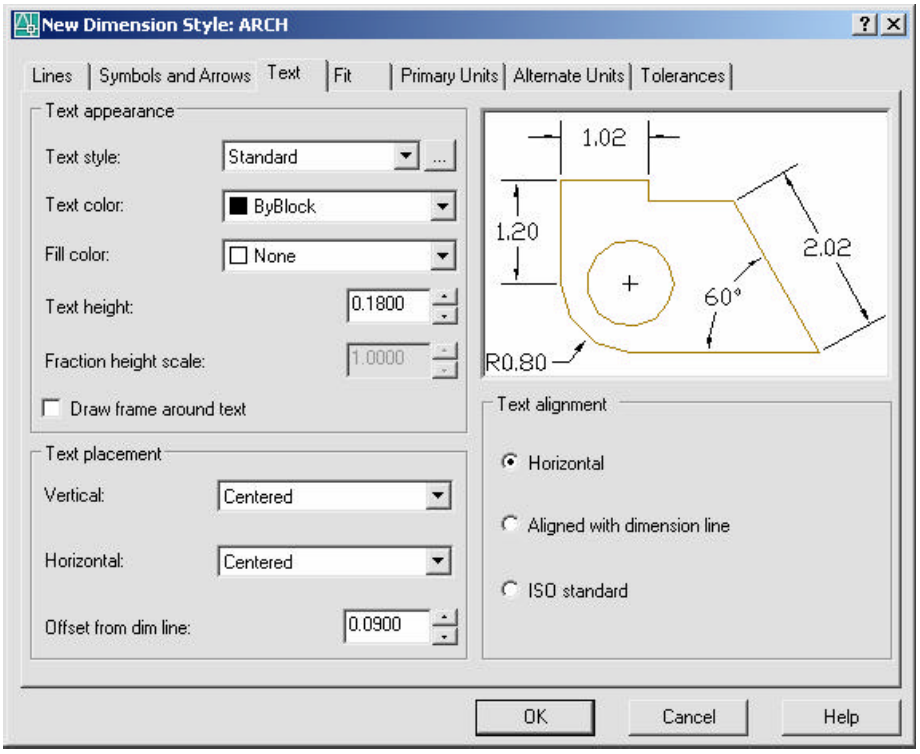




Text 27.3

Edits Text Appearance, Text Placement and Text Alignment.

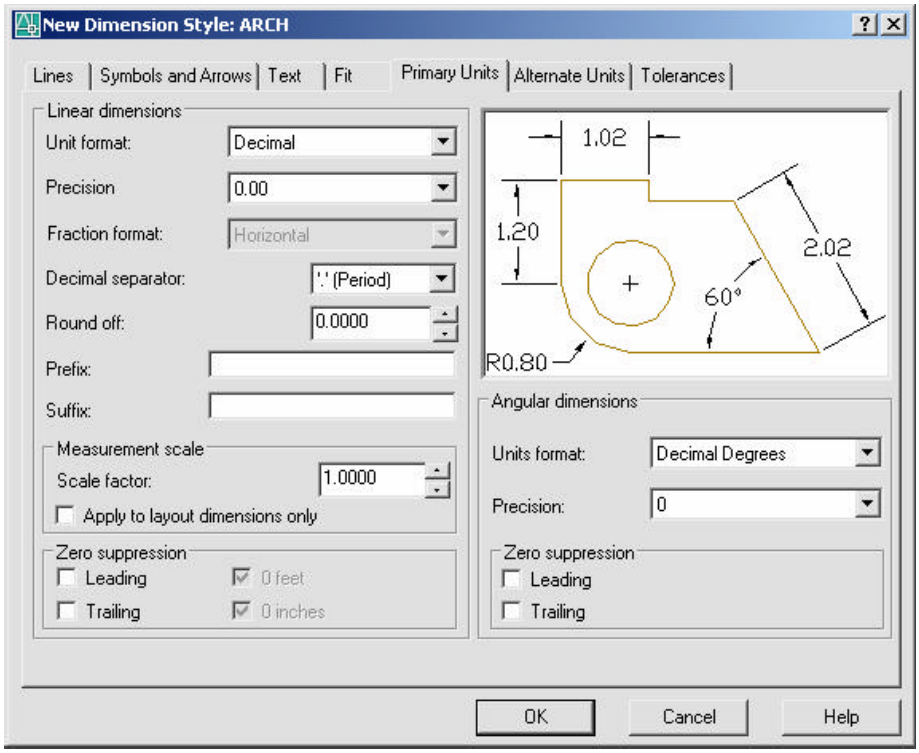
1. **Pick** the Text tab from the Dimension Variables and Styles dialog box.



Primary Units 27.4

Edits Unit options for dimension's primary units.

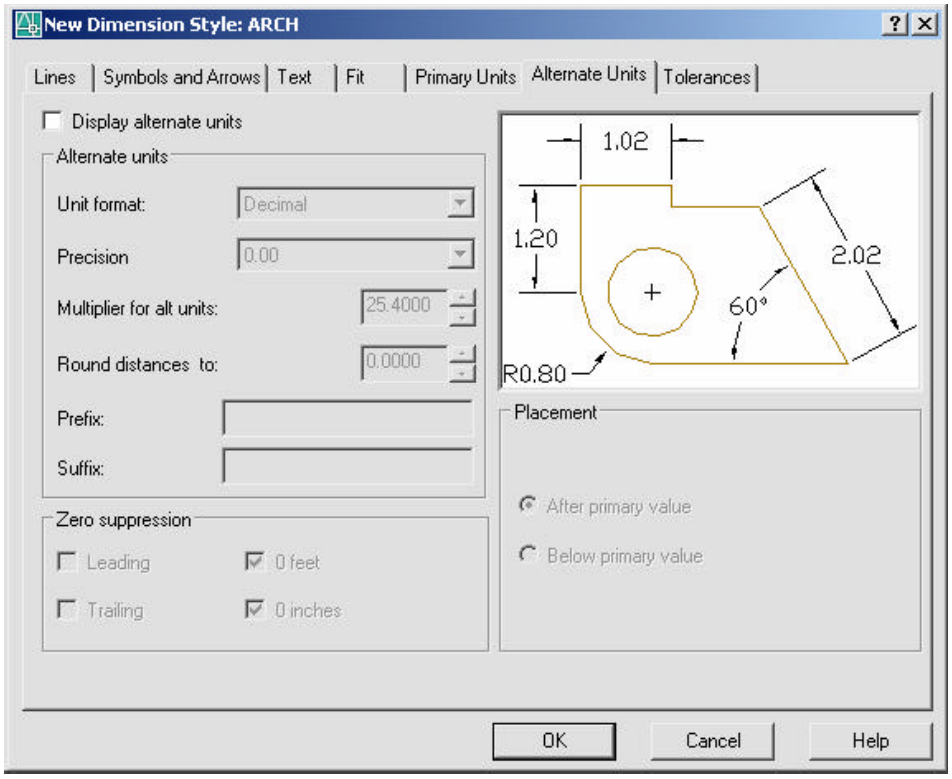
1. **Pick** the PRIMARY UNIT tab from the Dimension Variables and Styles dialog box.



Alternate Units 27.5

Edits Unit options for dimension's alternate units.

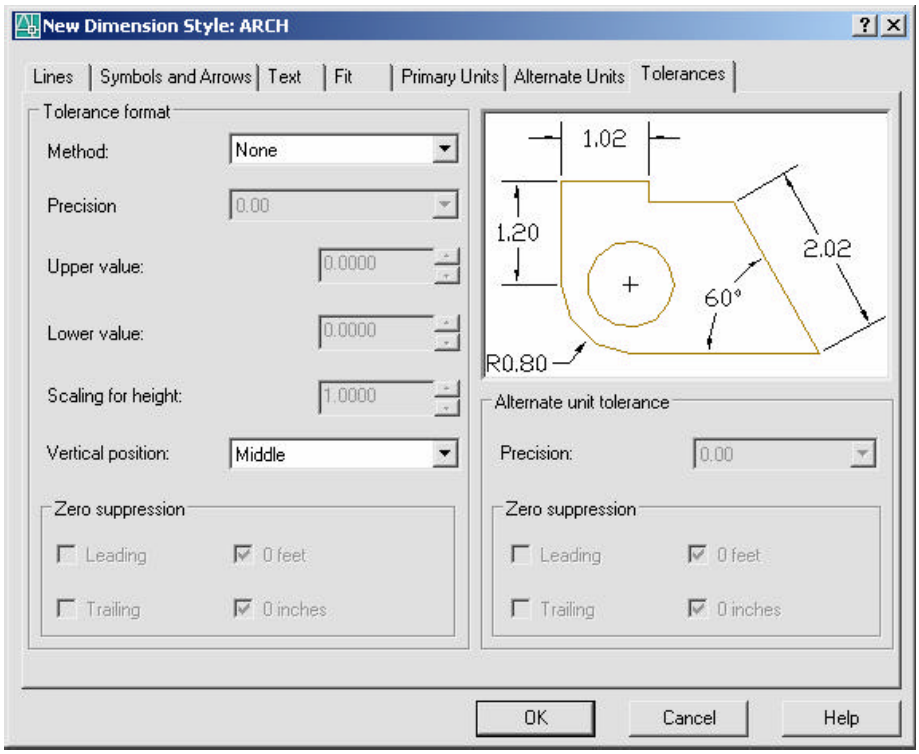
1. **Pick** the ALTERNATE UNIT tab from the Dimension Variables and Styles dialog box.



Tolerances 27.6

Edits Unit options for tolerances.

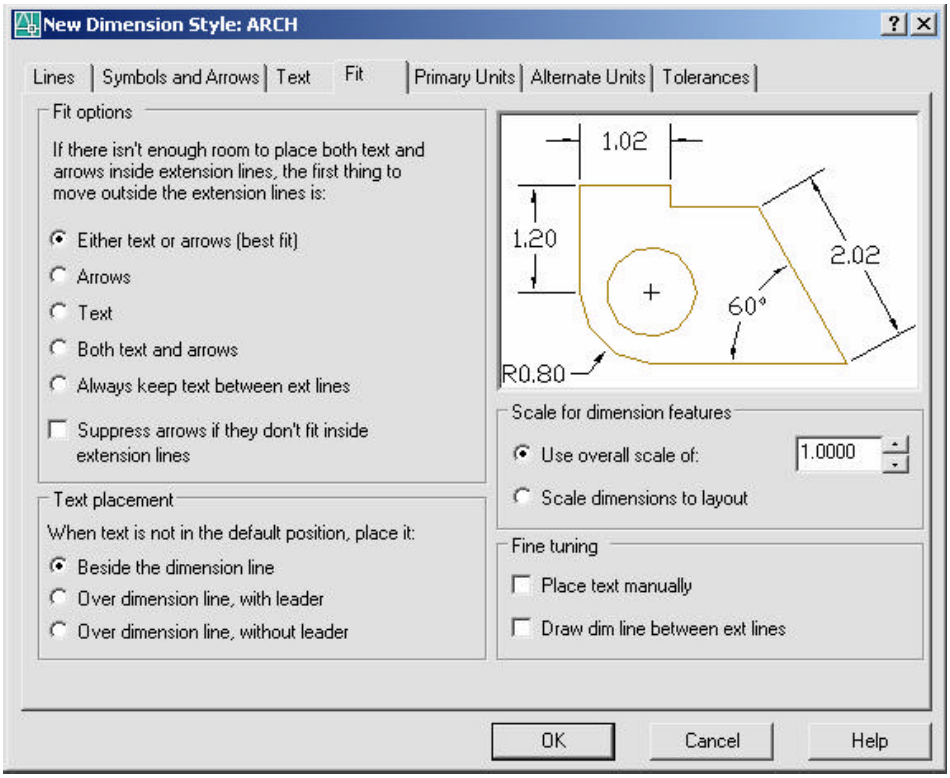
1. **Pick** the TOLERANCES tab from the Dimension Variables and Styles dialog box.



Fit 27.7

Edits Unit options for fitting dimensions and dimension scales.

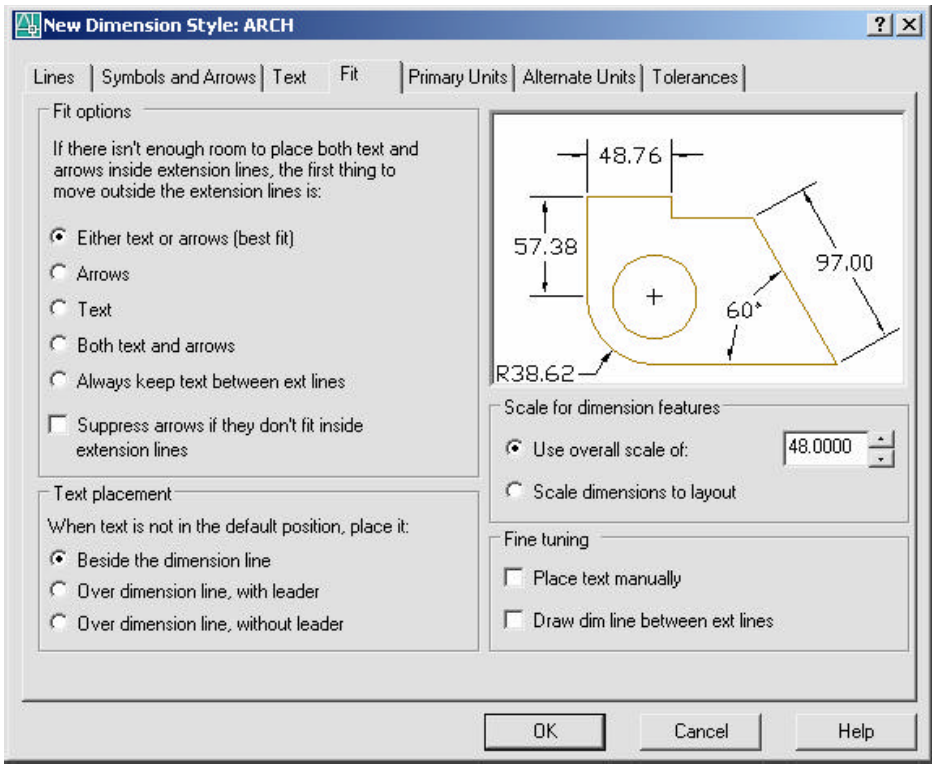
1. **Pick** the FIT tab from the Dimension Variables and Styles dialog box.



Dimscale 27.8

Edits Unit options for fitting dimensions and dimension scales.

1. **Pick** the FIT tab from the Dimension Variables and Styles dialog box.



Dimension Overrride 27.9

1. **Choose** Dimension, Override.
2. **Type** a dimension setting to change (i.e. DIMSE1 which suppresses the first extension line).

   Command: **_dimoverride**

   Enter dimension variable name to override or [Clear overrides]: **dimse1**
3. **Set** the new value.

   Enter new value for dimension variable <Off>: **on**
4. **Press** enter.
5. **Pick** the dimension to override.

Dimension Variables 27.10

1. **Type** **SETVAR at the command prompt.**

Command: **SETVAR**
Enter variable name or [?]: **?**
Enter variable(s) to list <*>: **dim***

DIMADEC 0
DIMALT OFF
DIMALTD 2
DIMALTF 25.4000
DIMALTRND 0.0000
DIMALTTD 2
DIMALTTZ 0
DIMALTU 2
DIMALTZ 0
DIMAPOST ""
DIMASO ON
DIMASSOC 1
DIMASZ 0.1800
DIMATFIT 3
DIMAUNIT 0
DIMAZIN 0
DIMBLK "ArchTick"
DIMBLK1 ""
DIMBLK2 ""
DIMCEN 0.0900
DIMCLRD 0
DIMCLRE 0
DIMCLRT 5
DIMDEC 1
DIMDLE 0.1250
DIMDLI 0.5000

```
DIMDSEP         "."
DIMEXE          0.1800
DIMEXO          0.1250
DIMFIT          3
DIMFRAC         0
DIMGAP          0.0900
DIMJUST         0
DIMLDRBLK       "Open90"
DIMLFAC         1.0000
DIMLIM          OFF
DIMLUNIT        4
DIMLWD          -2
DIMLWE          -2
DIMPOST         ""
DIMRND          0.0000
DIMSAH          OFF
DIMSCALE        1.0000
DIMSD1          OFF
DIMSD2          OFF
DIMSE1          OFF
DIMSE2          OFF
DIMSHO          ON
DIMSOXD         OFF
DIMSTYLE        "ARCH"                  (read only)
DIMTAD          1
DIMTDEC         1
DIMTFAC         1.0000
DIMTIH          ON
DIMTIX          OFF
DIMTM           0.0000
```

DIMTMOVE 0
DIMTOFL OFF
DIMTOH ON
DIMTOL OFF
DIMTOLJ 1
DIMTP 0.0000
DIMTSZ 0.0000
DIMTVP 0.0000
DIMTXSTY "Standard"
DIMTXT 0.1800
DIMTZIN 0
DIMUNIT 4
DIMUPT OFF
DIMZIN 0

Chapter 28
Views and Viewports

Named Views 28.1

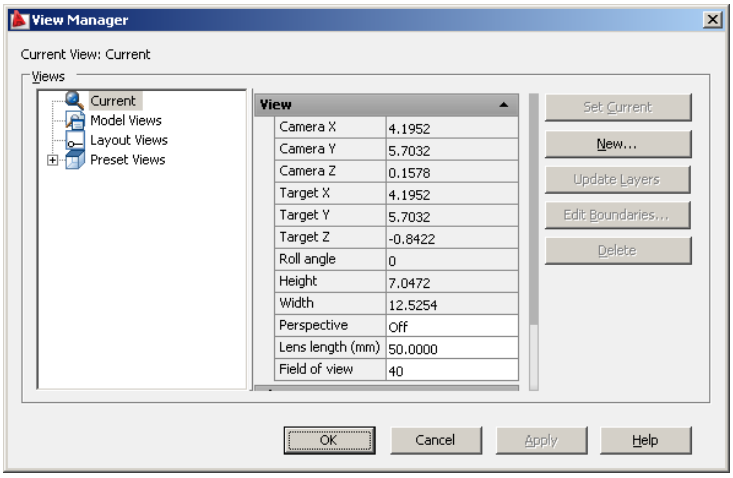
Ddview Command

1. **Choose** View, Named Views...

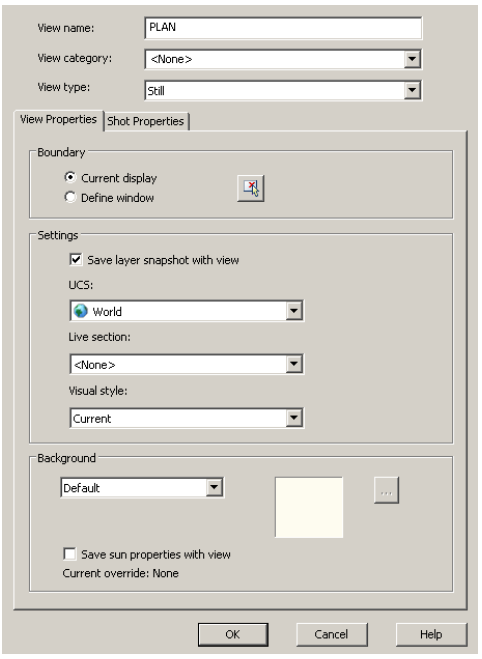
or

2. **Click** the Named View icon from the View toolbar. 
3. **Type** DDVIEW at the command prompt.

   Command: **DDVIEW**



4. **Choose** the NEW button.
5. **Type** a view name.
6. **Choose** Current display or Define Window.



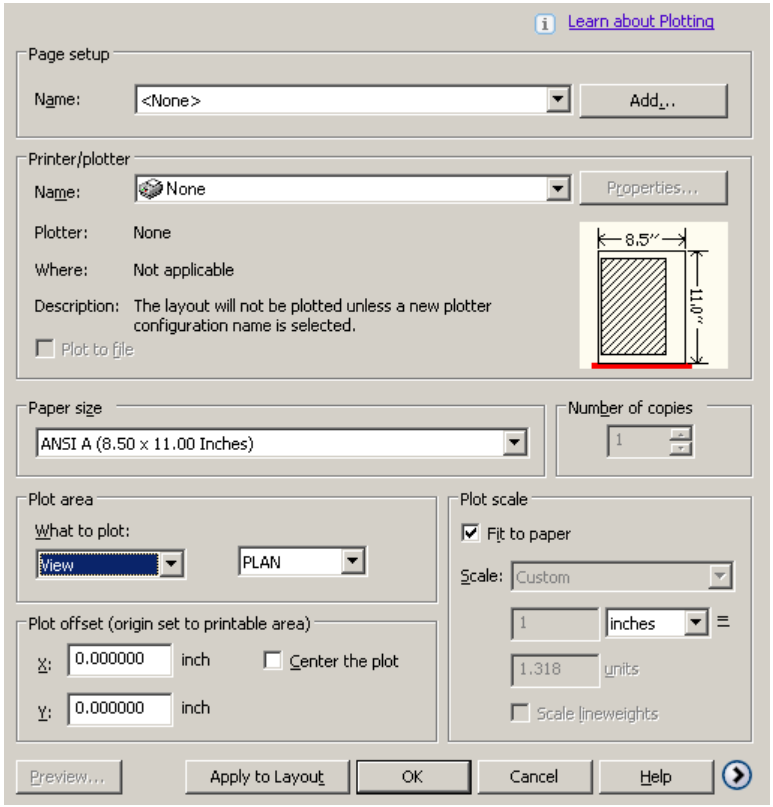
Typing the View Command

1. **Type** View at the command prompt. Command: **-VIEW**
2. **Type** One of the following view options:
   ?/Delete/Restore/Save/Window:

View options:

?	Lists the named views for this drawing
Delete	Deletes the named view
Restore	Displays the specified view
Save	Attaches a name to the current view of the drawing
Window	Attaches a name to specified window

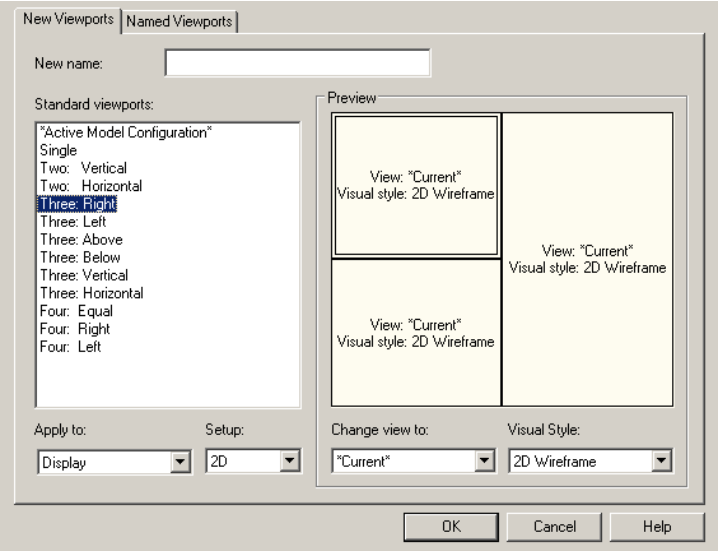
Plotting Named Views



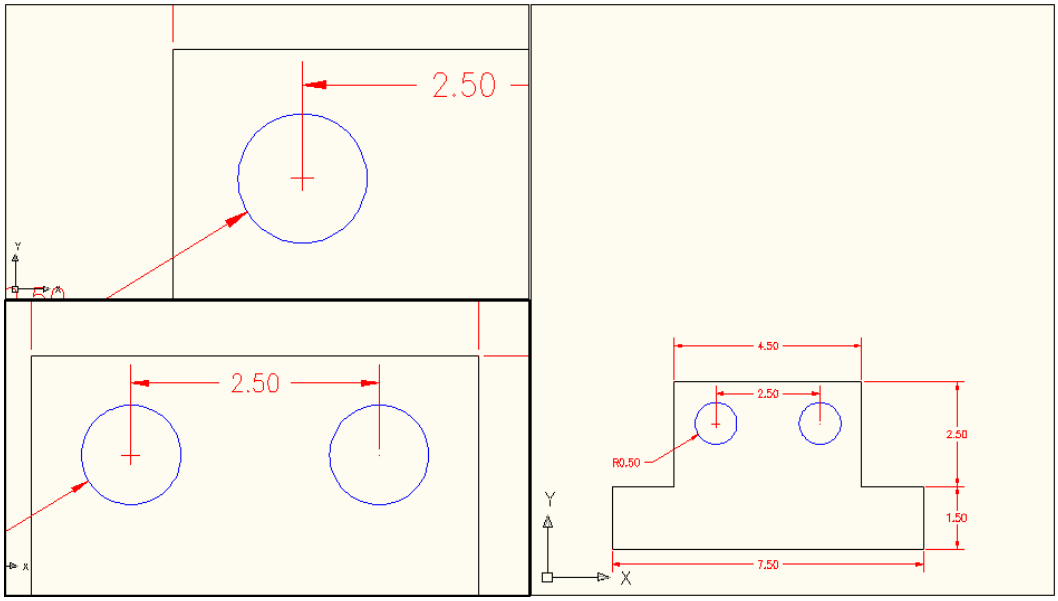
Viewports 28.2

Vports Command

1. **Choose** View, Viewports, New Viewports...
2. **Choose** one of the viewports configurations
3. **Click** OK.



4. **Click** once in each vport to make it active.
5. **Type** a ZOOM option in each viewport.



Viewport options

New Name	Gives a name to a viewport
Restore	Restores an original viewport
Delete	DEL deletes a viewport
Join	Joins two viewports together
SIngle	Creates on viewport in the drawing

TIPS:

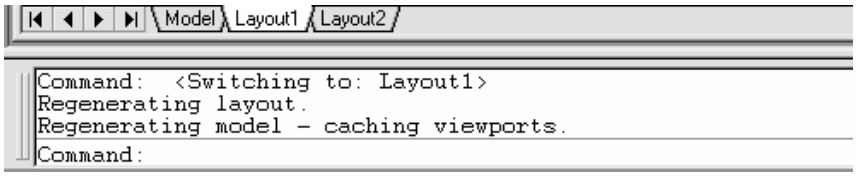
Viewports can be named and restored later.

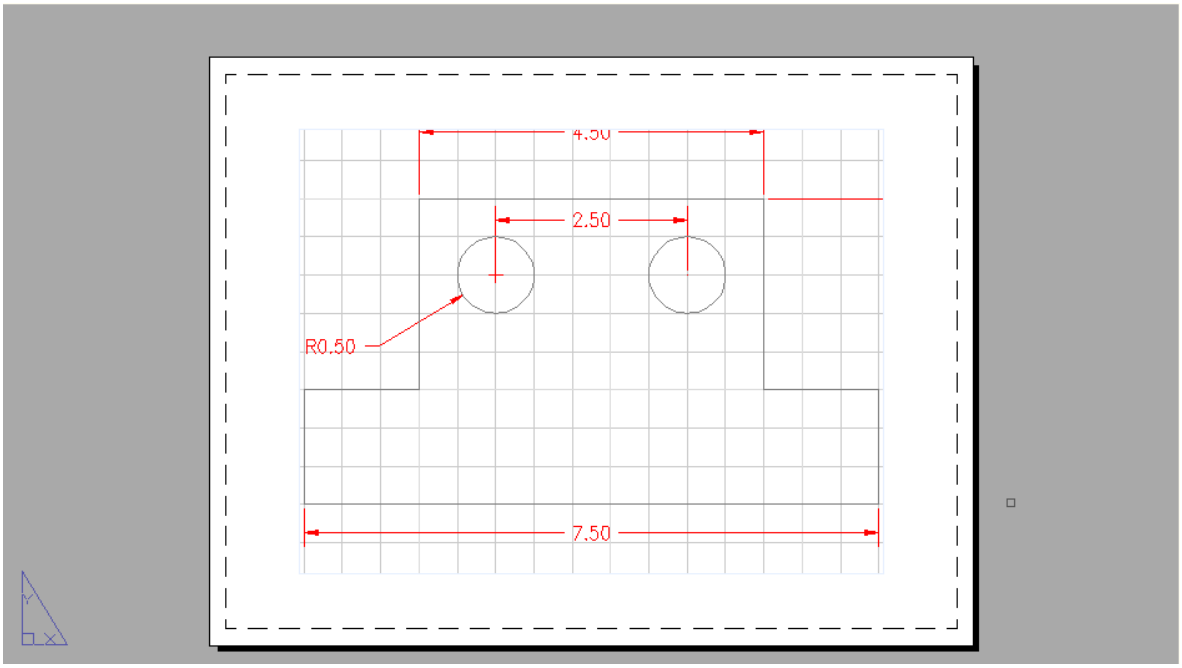
AutoCAD plots only the current vport.

Chapter 29
Model Space and Paper Space

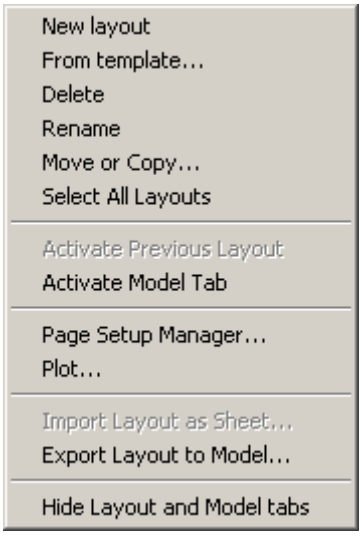
Creating a Layout 29.1

1. **Choose** the Layout1 TAB at the bottom of the screen.



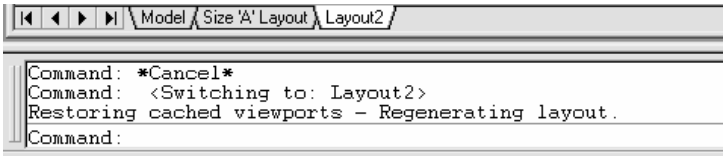


2. **Right-click** Layout 1 to change the name and other properties of a layout

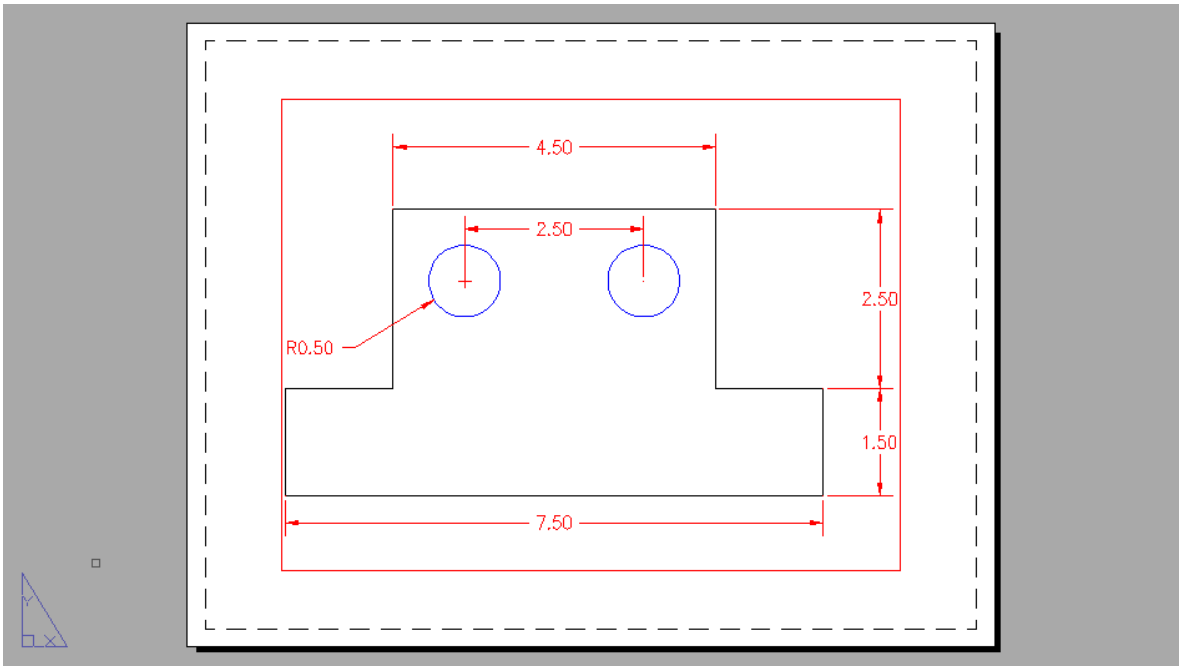


Creating Multiple Layouts 29.2

1. **Choose** the Layout2 TAB at the bottom of the screen.



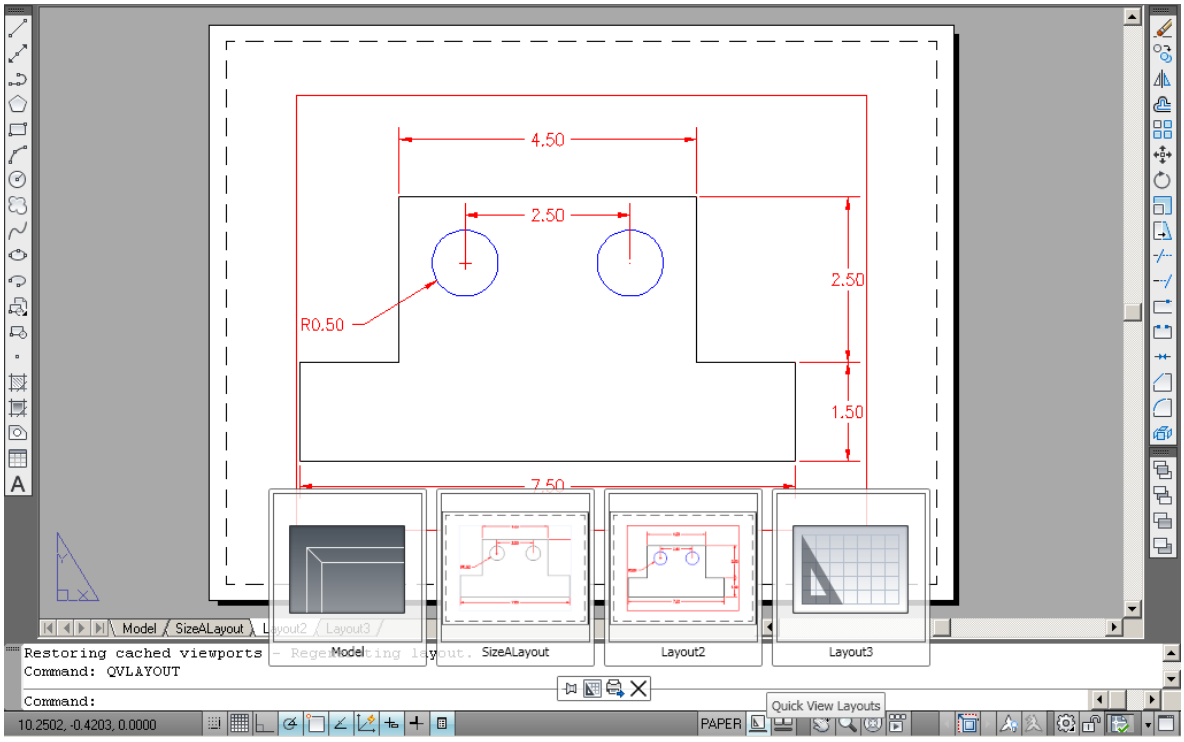
2. **Change** the name of the layout.
3. **Change** the remaining Layout options for page setup and plots.



Quick View Layouts 29.3

1. **Choose** Quick View Layouts from the status bar.

2. Choose a layout to view.



Mview Command 29.4

- The MVIEW command controls the size and position of the mview viewports (from now on called mviews). Mview is to tilemode = 0 as vports is to tilemode = 1.
- Use mview when you would like to see a view of the model.
- Pspace mode must be active to use mview. AutoCAD will automatically switch to pspace when you issue the mview command.
- The default mview option is "<<first point>>" To use this option, pick a point which represents one corner of the mview. At the "other corner" prompt, pick a point which represents the opposite corner of the mview.

1. **Choose** View, Viewports, 1 Viewport.

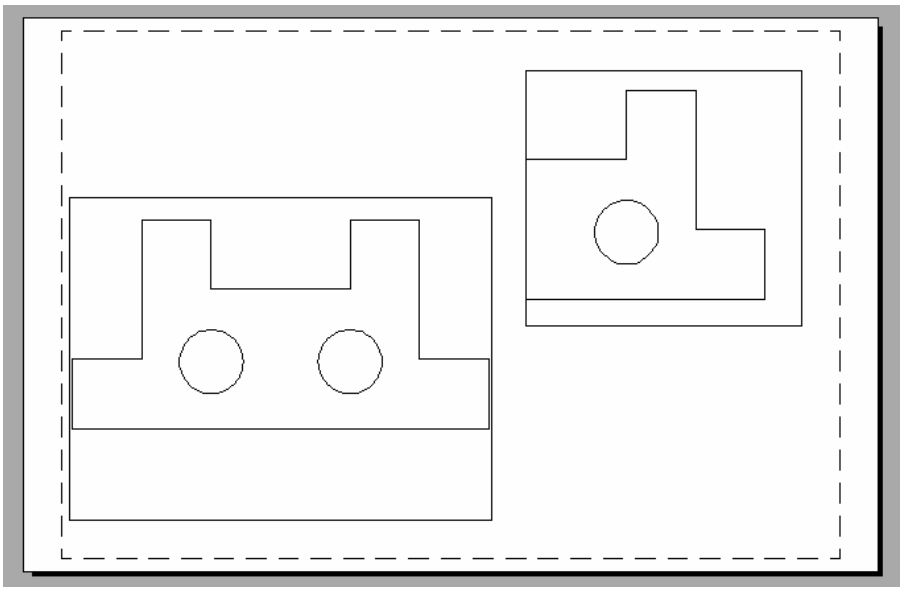
   or

2. **Type** Type MVIEW at the command prompt.

   Command: **MVIEW or MV**

   ON/OFF/Hideplot/Fit/2/3/4/Restore/<<First Point>>: **P1**

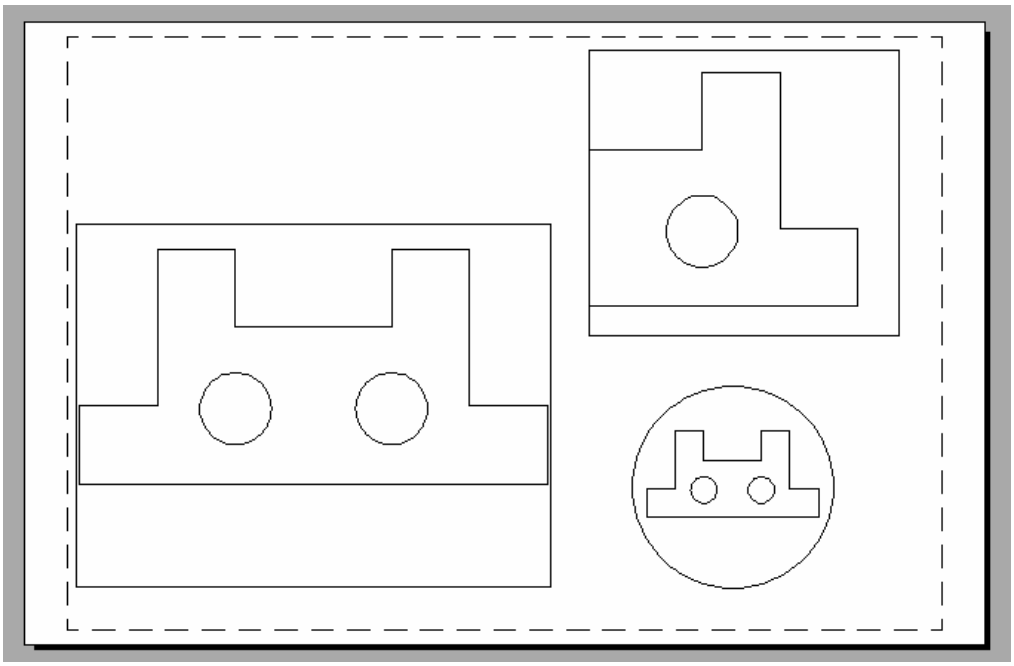
   Other corner: **P2**



TIP: Mviews should be created on their own layers in order to be turned ON/OFF.

Irregular Shaped Viewports 29.5

1. **Draw** a shape in Paper Space (e.g. circle, polygon, ellipse)
2. **Choose** View, Viewports, Object
3. **Choose** the object to make a viewport.



Model Space 29.6

MSPACE (model space) can only be activated if there is at least one mview. To enter model space mode use "MSPACE".

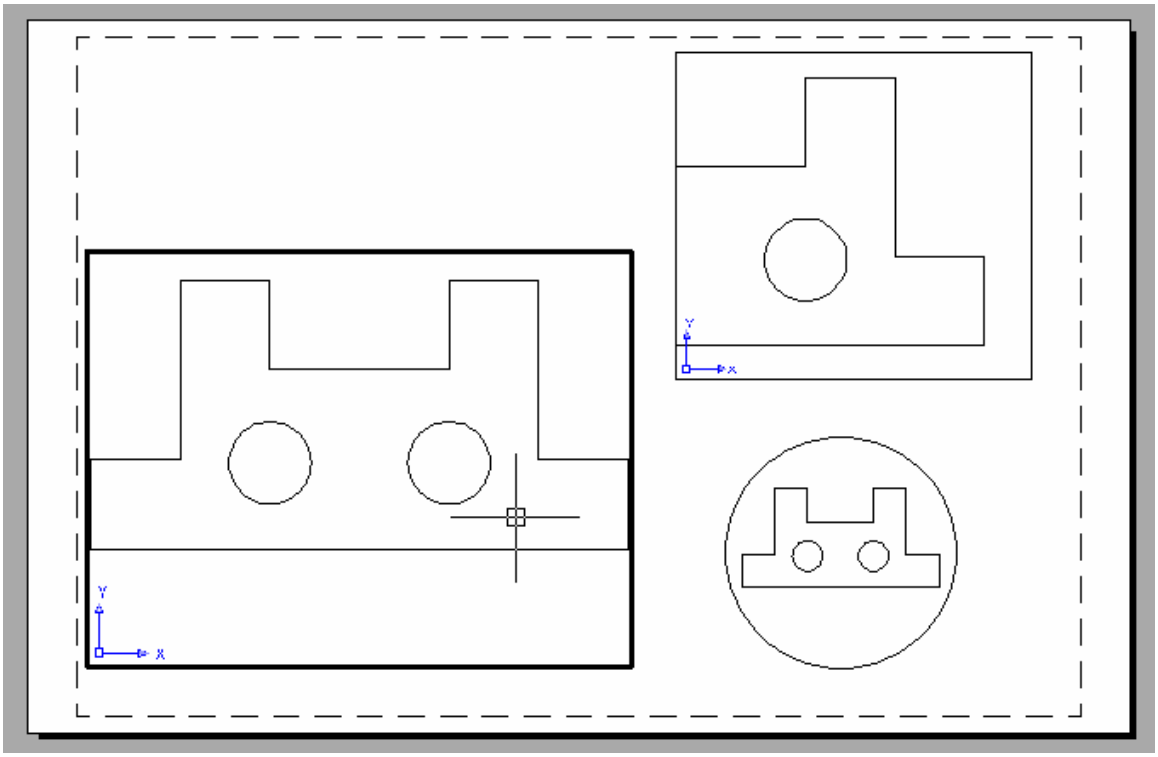
1. **Type** MSPACE at the command prompt.

   Command: **MSPACE or MS**

   or

2. **Double-Click** the word “PAPER” on the Status Bar to toggle to model space.

Notice the ucsicon will appear in each of the mviews when you enter model space.



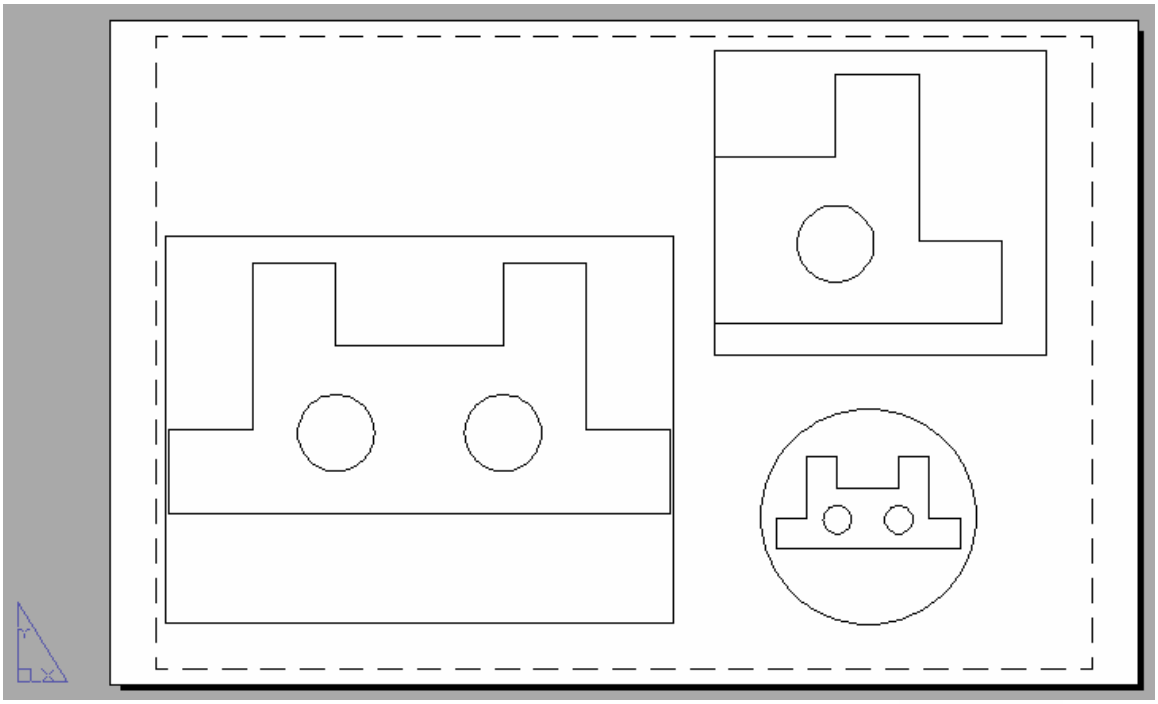
Paper Space 29.6

PSPACE mode should be entered to create a border, a title, mviews, and annotations only. This environment is used to lay out a 2 dimen- sional working drawing suitable for plotting. When you plot from pspace, you should plot 1=1.

1. **Type** PSPACE at the command prompt.
   Command: **PSPACE or PS**

   or

2. **Double-Click** the word “MODEL” on the Status Bar to toggle to paper space.



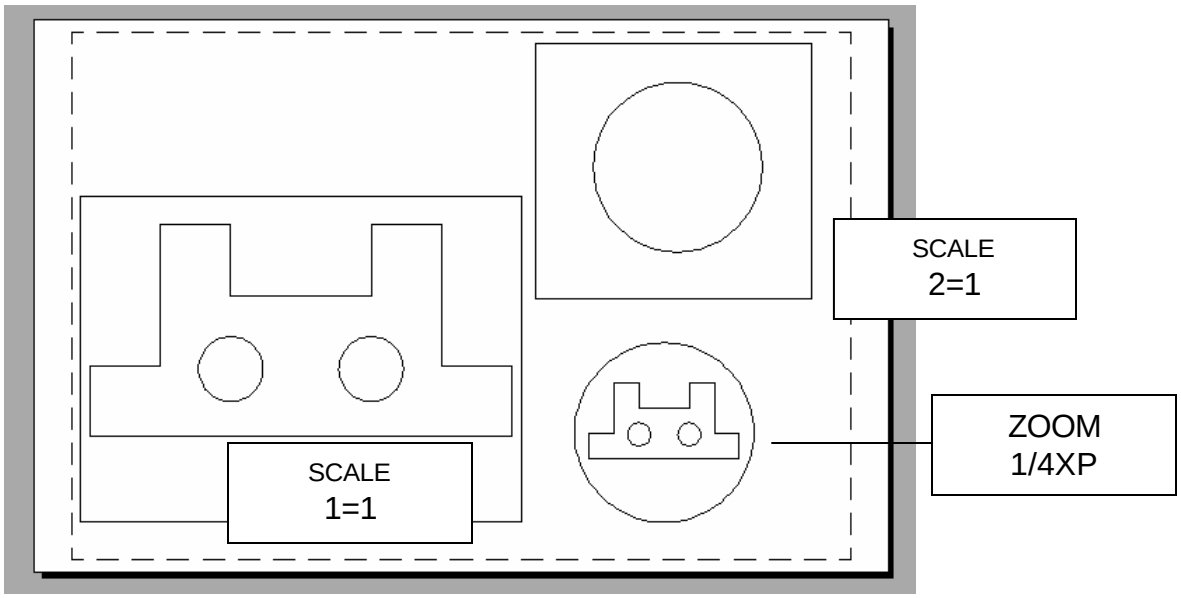
Notice the "Paper" in the status line and the pspace icon.

Scales - Zooming in Model Space 29.7

- Use ZOOM "XP" to zoom the model a certain factor of the paper.
- If you enter a value followed by xp, AutoCAD specifies the scale rela- tive to paper space units. For example, entering .5xp displays model space at half the scale of paper space units.
- If you w ant to plot the m odel at 1/4”=1’, typeZOOM 1/48XP . If you want to plot a part at 3 times, type ZOOM 3XP.
- Views can also be shown in 3D by using the VPOINT command.

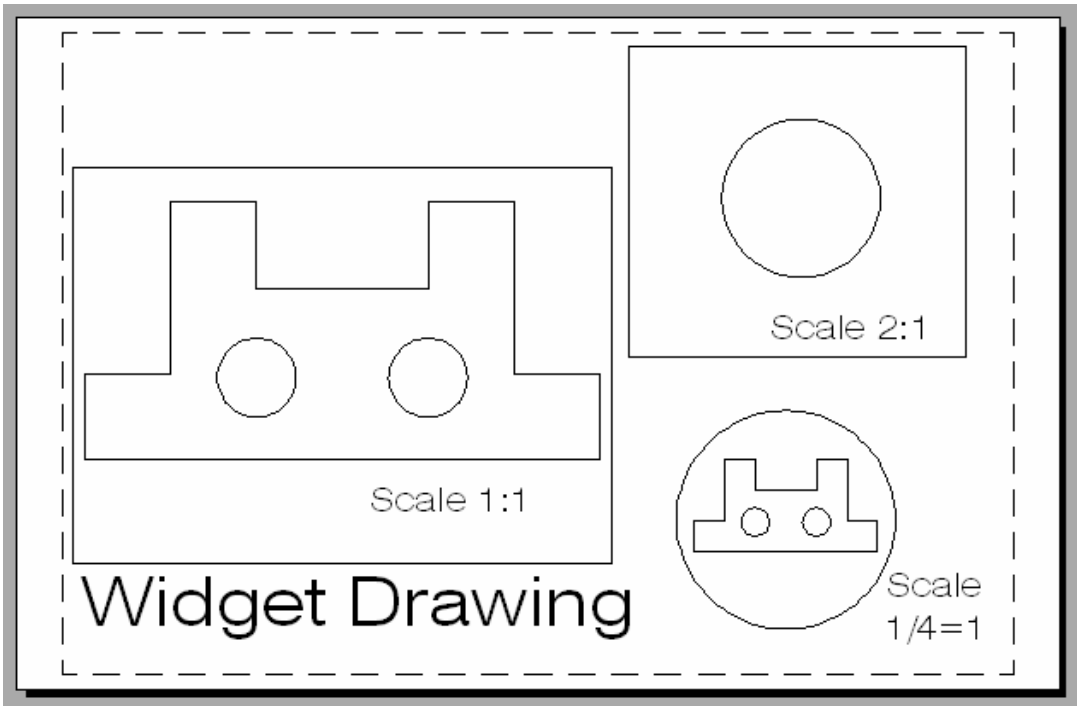
1. **Type** MS at the command prompt to enter Model Space for each individual viewport.
2. **Type** ZOOM at the command prompt. Command: ZOOM All/Center/Dynamic/Extents/Previous/ Scale(X/XP)/ Window/<Realtime>: **3XP**

You can also change the scale from the Viewports Toolbar.



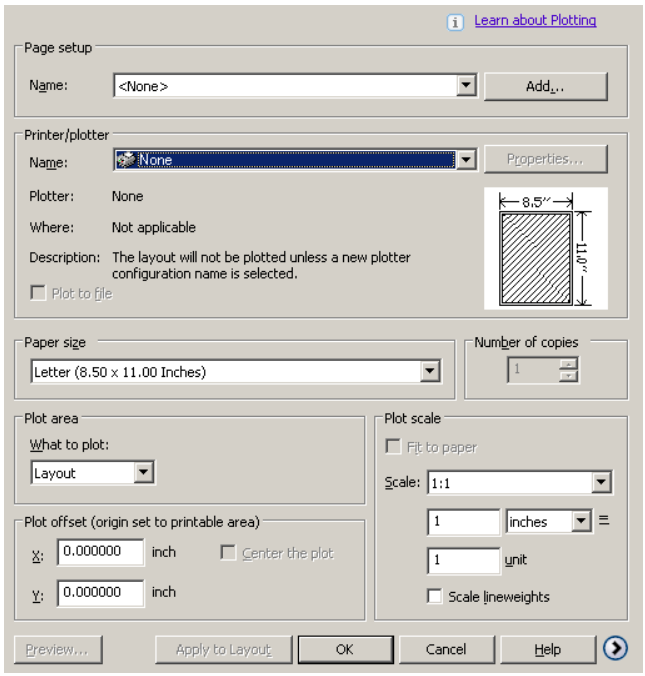
Adding Text in Paper Space 29.8

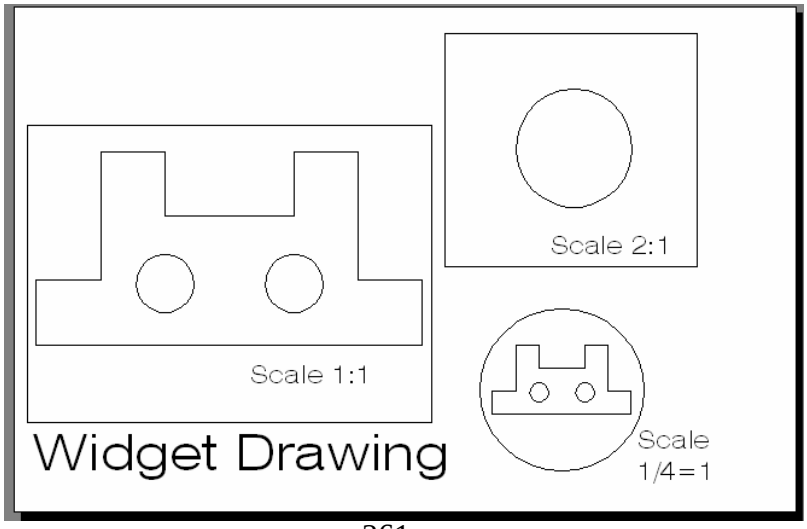
Title block text and miscellaneous text can be added in Paper Space.



Plotting in Paper Space 29.9

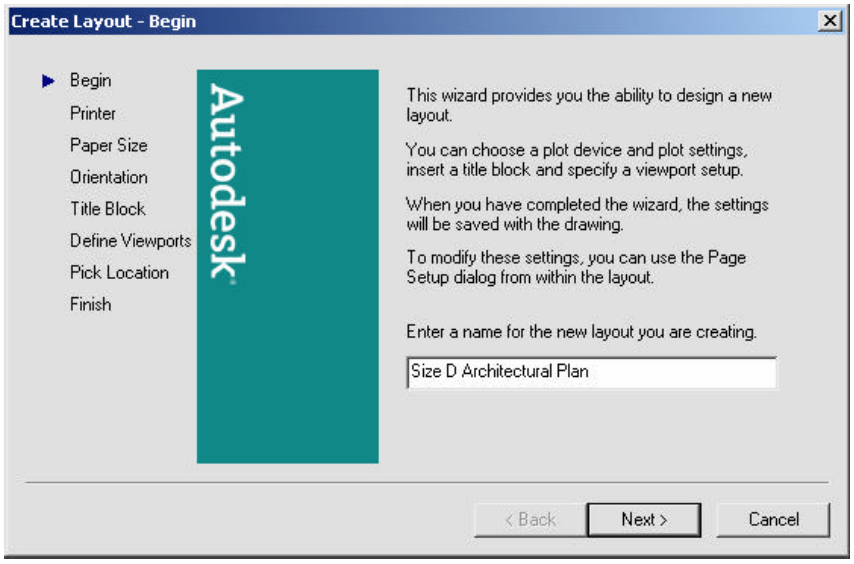
- Plotting all MVIEWS should be done from Paper Space not from Model Space.
- When you plot from pspace, you should plot1=1.
- For hidden line removals, remember to use the HIDEPLOT option in the MVIEW command.
- Once a ZOOM SCALE has been defined, do not zoom again before plotting. You can change the display with the PAN command.

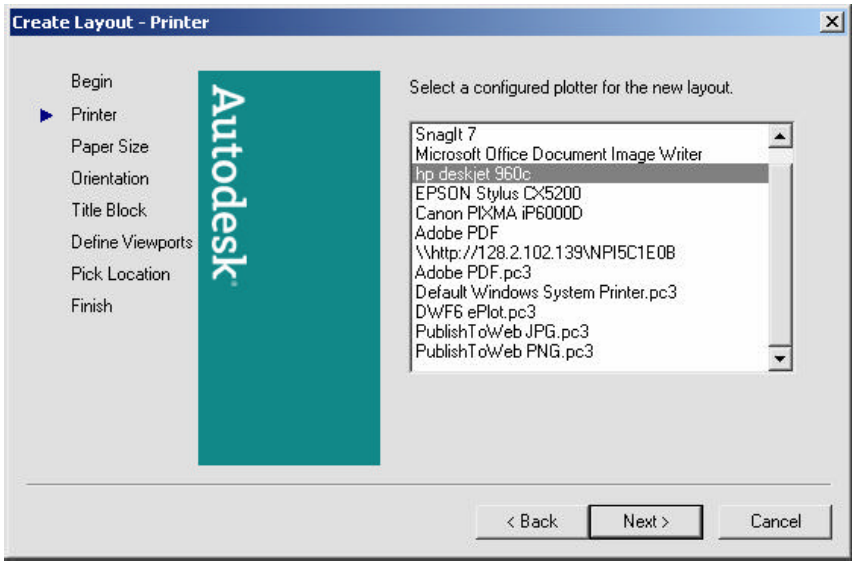


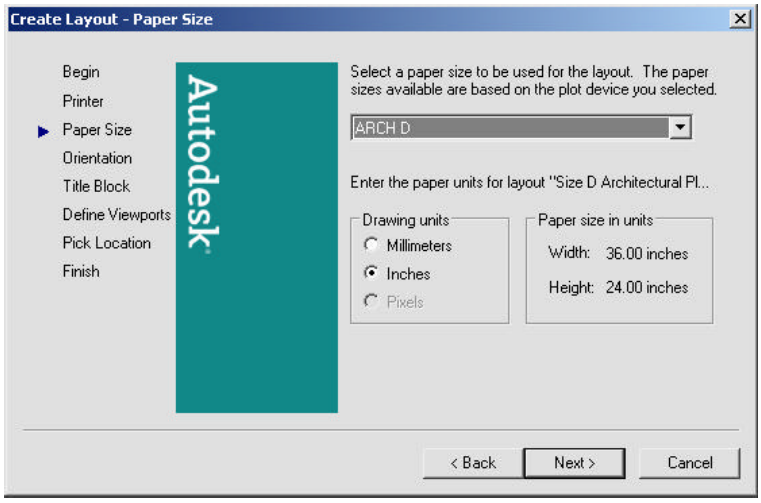


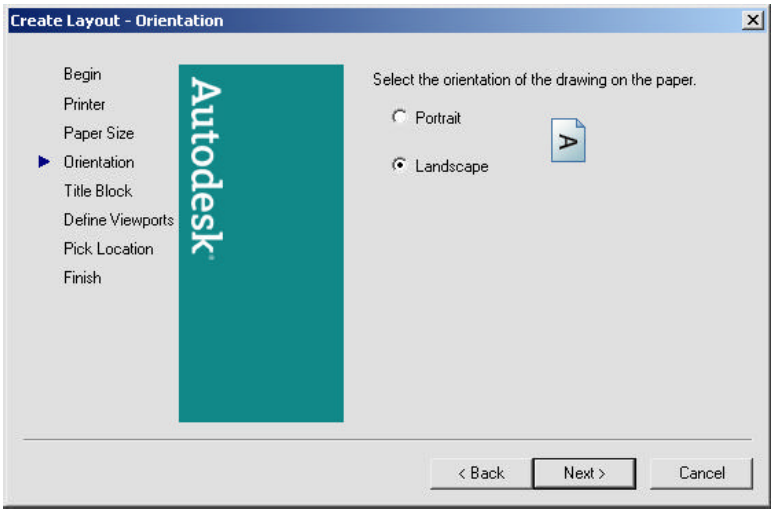
Layout Wizard 29.10

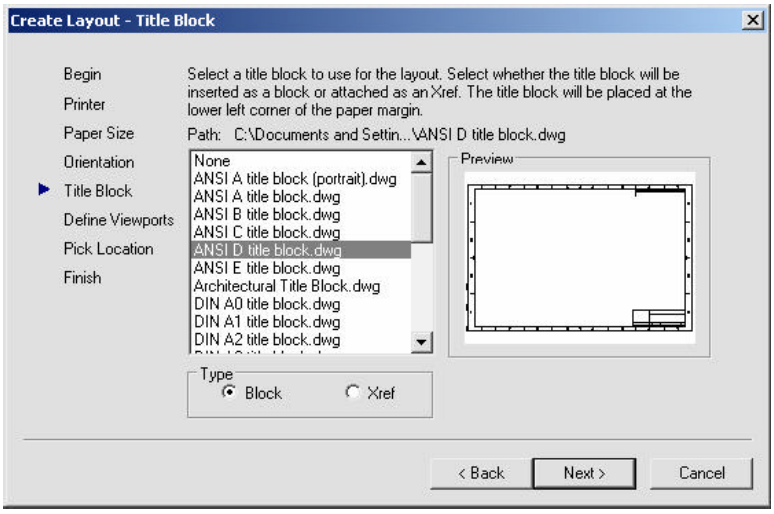
1. **Choose** Insert, Layout, Layout Wizard
2 **Change** the various options in the Layout Wizard.

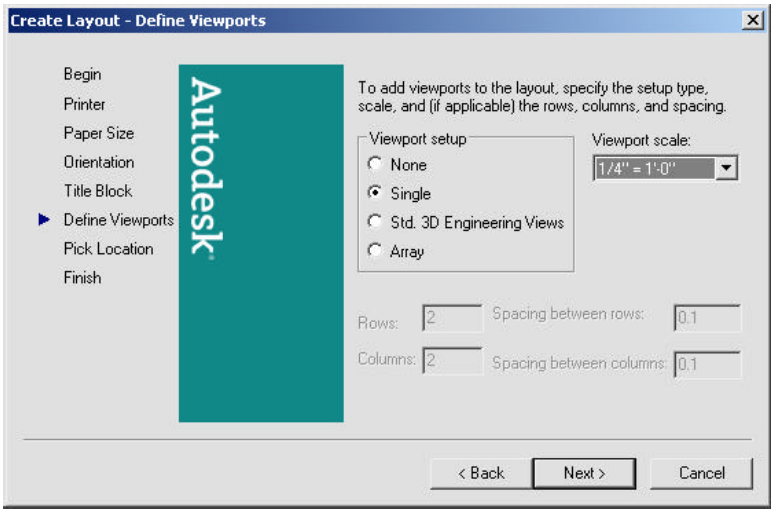


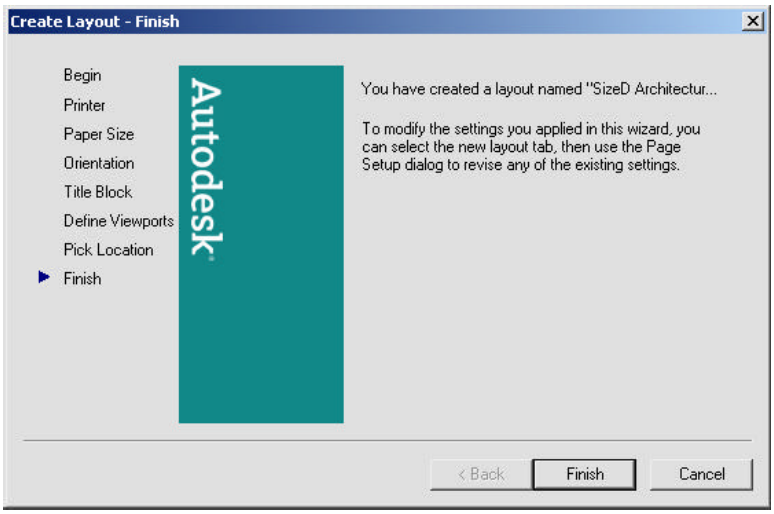


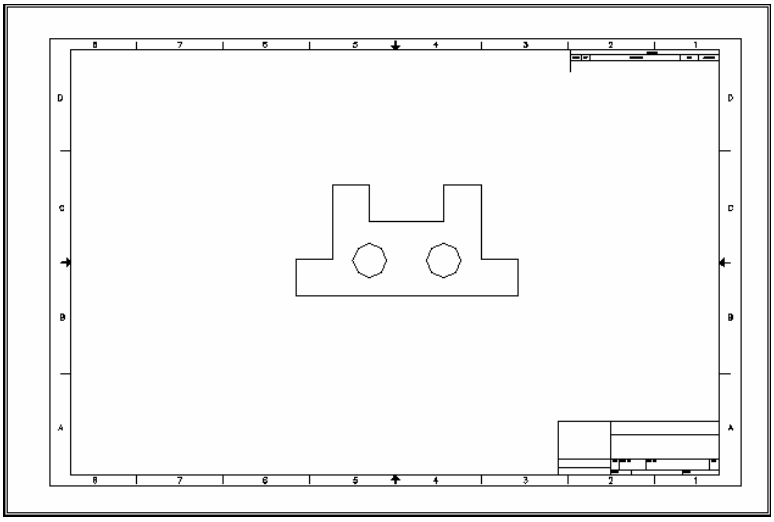

Create Layout - Paper Size
Begin
Printer
Paper Size
Orientation
Title Block
Define Viewports
Pick Location
Finish
Autodesk
Select a paper size to be used for the layout. The paper sizes available are based on the plot device you selected.
ARCH D
Enter the paper units for layout "Size D Architectural Pl...
Drawing units
Millimeters
Inches
Pixels
Paper size in units
Width: 36.00 inches
Height: 24.00 inches
< Back
Next >
Cancel


Create Layout - Orientation
Begin
Printer
Paper Size
Orientation
Title Block
Define Viewports
Pick Location
Finish
Autodesk
Select the orientation of the drawing on the paper.
Portrait
Landscape
A
< Back
Next >
Cancel


Create Layout - Title Block
Begin
Printer
Paper Size
Orientation
Title Block
Define Viewports
Pick Location
Finish
Select a title block to use for the layout. Select whether the title block will be inserted as a block or attached as an Xref. The title block will be placed at the lower left corner of the paper margin.
Path: C:\Documents and Settin...\ANSI D title block.dwg
None
ANSI A title block (portrait).dwg
ANSI A title block.dwg
ANSI B title block.dwg
ANSI C title block.dwg
ANSI D title block.dwg
ANSI E title block.dwg
Architectural Title Block.dwg
DIN A0 title block.dwg
DIN A1 title block.dwg
DIN A2 title block.dwg
Preview
Type
Block
Xref
< Back
Next >
Cancel


Create Layout - Define Viewports
Begin
Printer
Paper Size
Orientation
Title Block
Define Viewports
Pick Location
Finish
Autodesk
To add viewports to the layout, specify the setup type, scale, and (if applicable) the rows, columns, and spacing.
Viewport setup
None
Single
Std. 3D Engineering Views
Array
Viewport scale:
1/4" = 1'-0"
Rows:
2
Spacing between rows:
0.1
Columns:
2
Spacing between columns:
0.1
< Back
Next >
Cancel


Create Layout - Finish
Begin
Printer
Paper Size
Orientation
Title Block
Define Viewports
Pick Location
Finish
Autodesk
You have created a layout named "SizeD Architectur...
To modify the settings you applied in this wizard, you can select the new layout tab, then use the Page Setup dialog to revise any of the existing settings.
< Back
Finish
Cancel

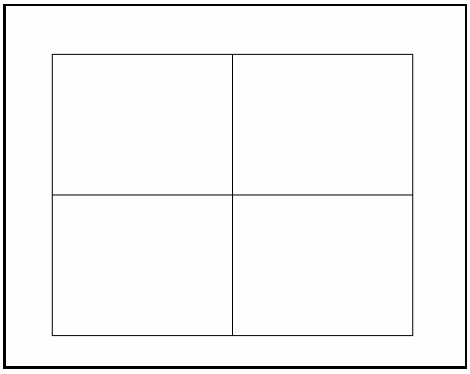


Tilemode 29.11

Tilemode is an AutoCAD system variable which can be set to 0 or 1. When tilemode is set to "1". viewports act as they traditionally did, like floortiles. Each viewport butts up against the next. The viewports fill the screen. They can only be plotted one as a time. These traditional viewports are known as "tiled areas of the screen". This is the default setting.

1. **Type** TILEMODE at the command prompt.
   Command: **TILEMODE**

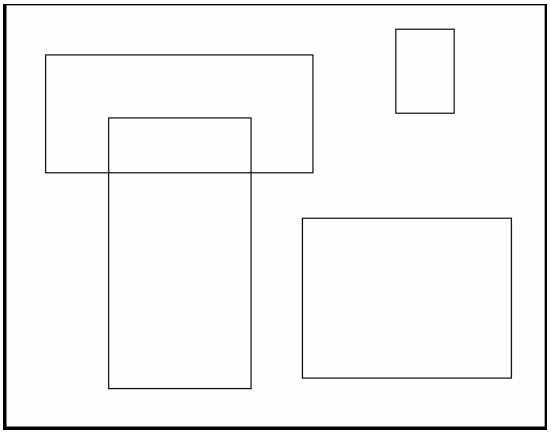
   New value for tilemode <1> : **Press ENTER**



When tilemode is set to "0". The new metaview (mview) viewports can be used. Mviews can be any size or place on the screen. They may overlap. More than one mview can be plotted at a time. Each mview can be quickly turned on or off.

1. **Type** Command: TILEMODE

   New value for tilemode <1> : 0

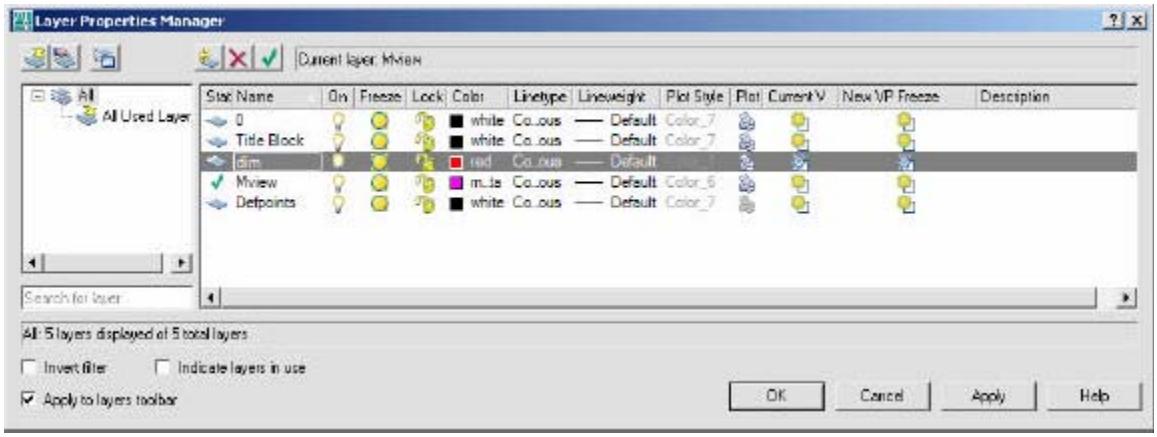


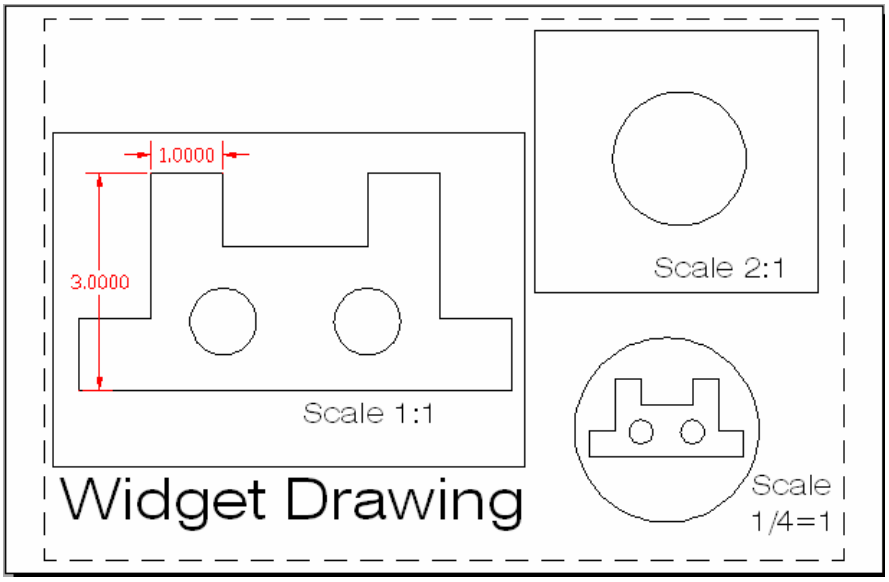
Viewport Layers 29.12

VPLAYER (viewport layer) controls layers on and off, and freeze and thaw, for each mview. Layer controls the on and off, freeze and thaw, globally. Layers must be on and thawed globally before they can be effected per mview with vplayer.

1. **Click** in the viewport to change layer status.
2. **Choose** the layer dialog box.
3. **Highlight** the layer to freeze or thaw in the current or new viewport.

The layer dialog box also allows control of layers for each viewport.

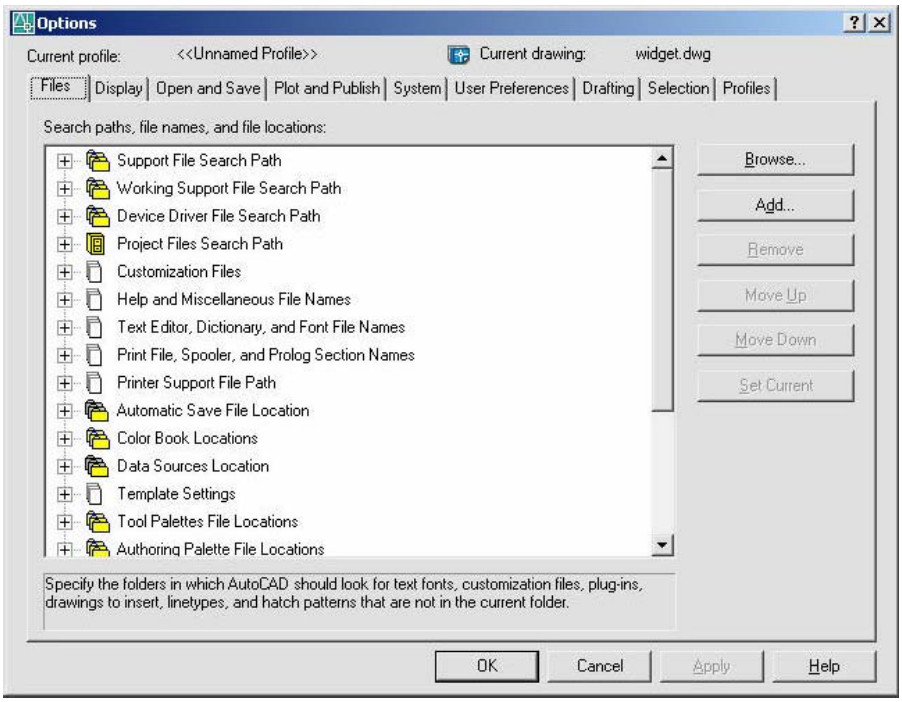




Chapter 30
Options Menu

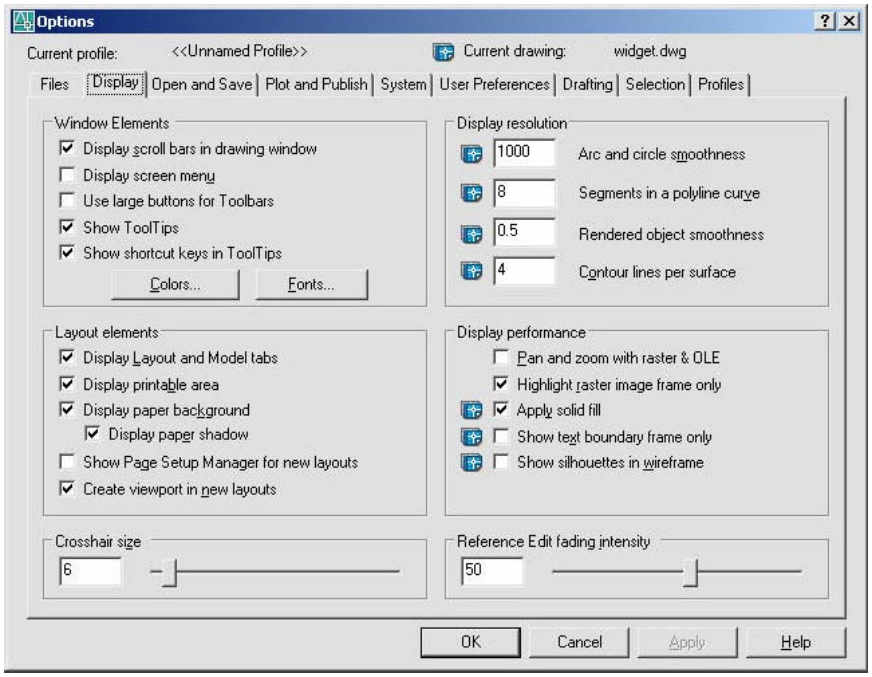
Files 30.1

1. **Choose** Tools, Options...
2. **Click** the Files TAB.



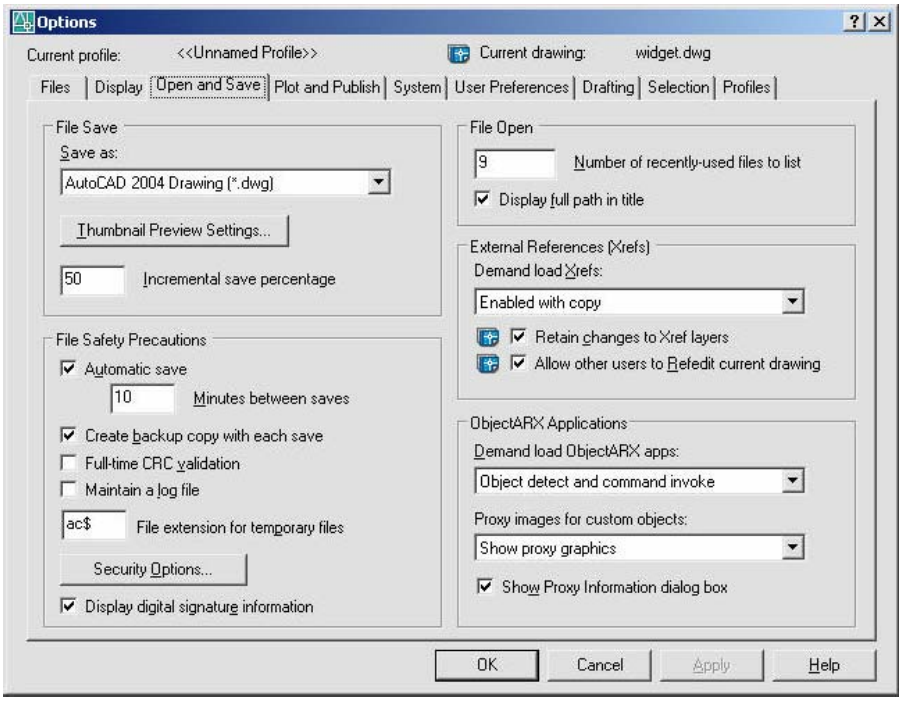
Display 30.2

1. **Choose** Tools, Options...
2. **Click** the Display TAB.



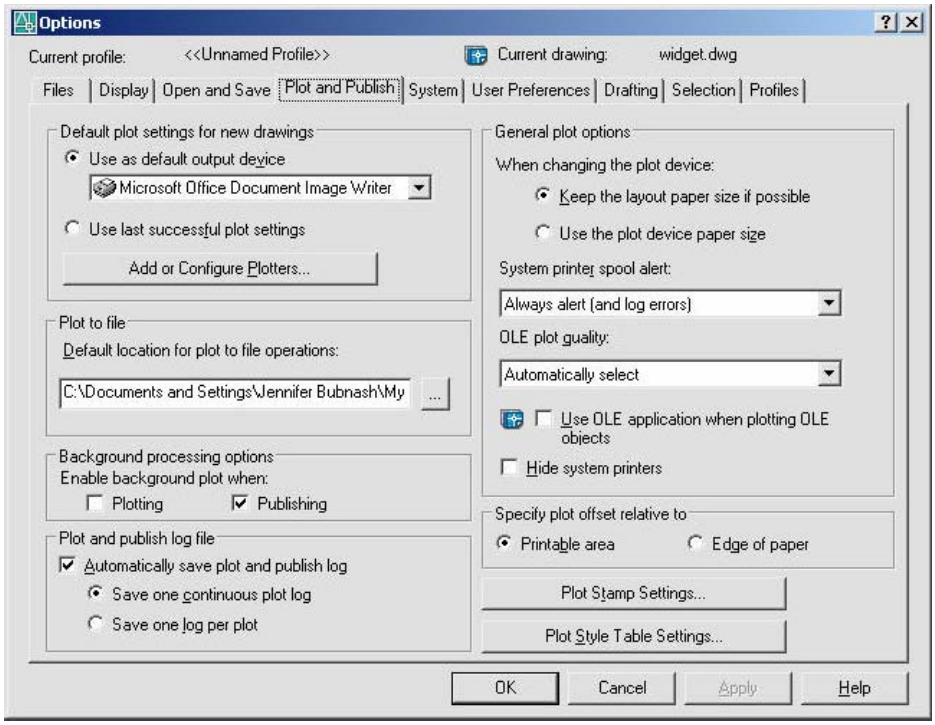
Open and Save 30.3

1. **Choose** Tools, Options...
2. **Click** the Open and Save TAB.



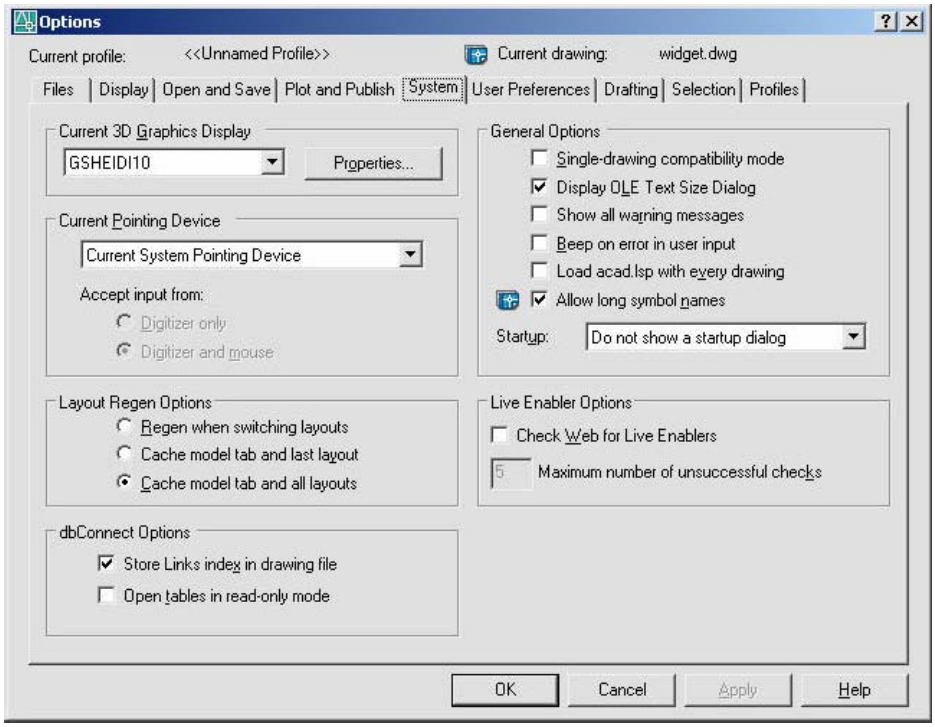
Plotting 30.4

1. **Choose** Tools, Options...
2. **Click** the Plotting and Save TAB.



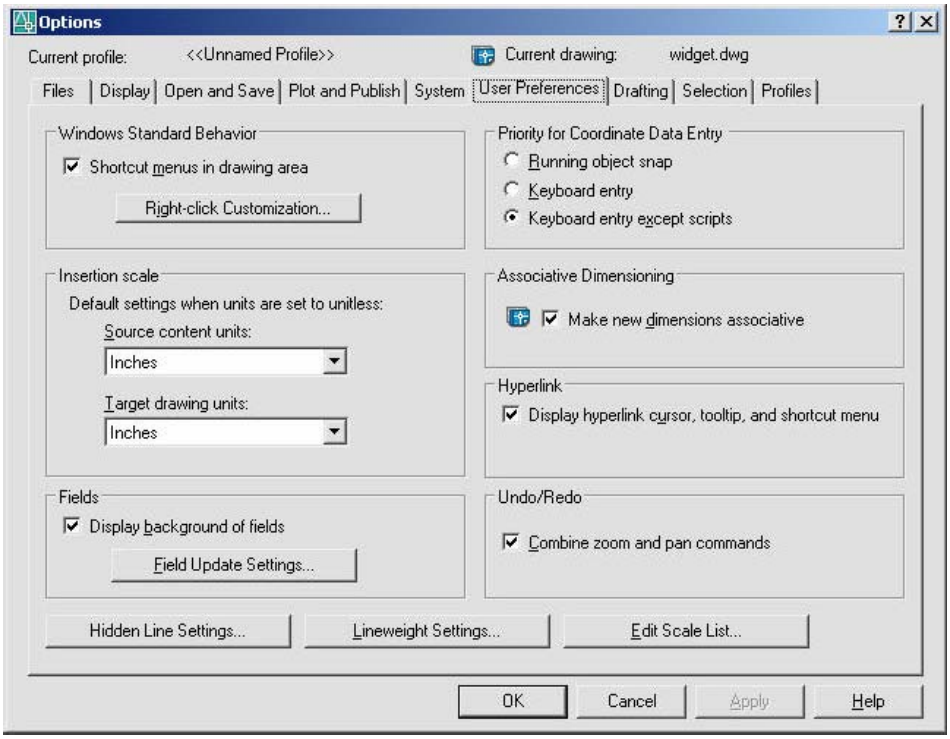
System 30.5

1. **Choose** Tools, Options...
2. **Click** the System TAB.



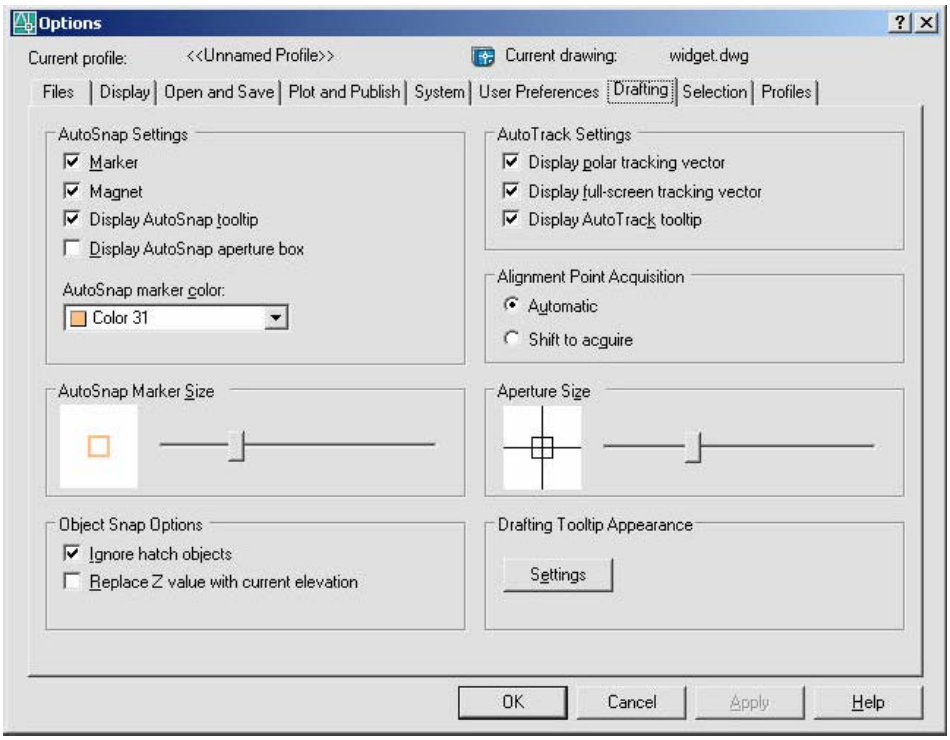
User Preferences 30.6

1. **Choose** Tools, Options...
2. **Click** the User Preferences TAB.



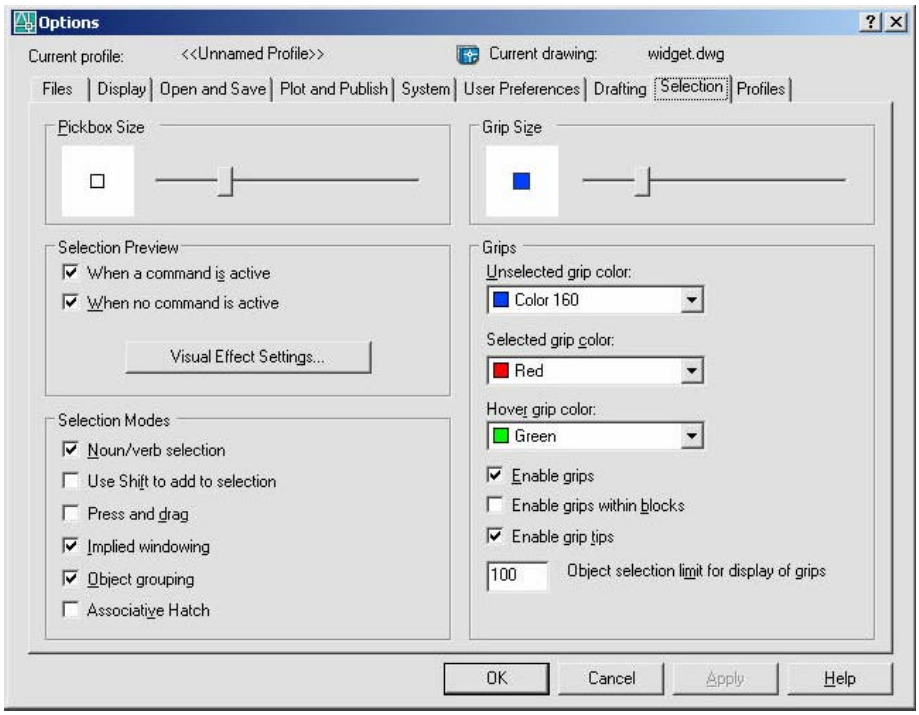
Drafting 30.7

1. **Choose** Tools, Options...
2. **Click** the Drafting TAB.



Selection 30.8

1. **Choose** Tools, Options...
2. **Click** the Selection TAB.



Chapter 31
Drawing Utilities

AUDIT 31.1

1. **Choose** File, Drawing Utilities, Audit.

or

2. **Type** Audit at the command prompt

   Command: **AUDIT**

3. **Type** Yes or No to fix any errors

   Fix any errors detected? <N>: Y or N

If a drawing contains errors that AUDIT can't fix, use the RECOVER command to retrieve the drawing and correct its errors.

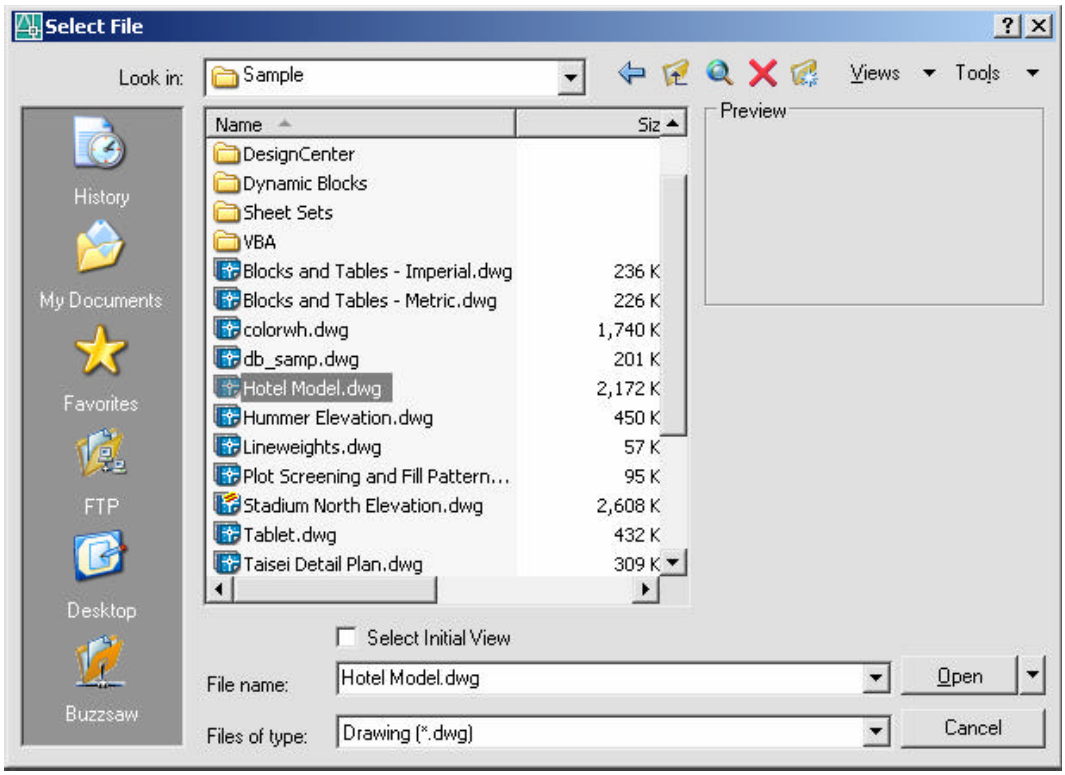
RECOVER 31.2

1. **Choose** File, Drawing Utilities, Recover....

or

2. **Type** RECOVER at the command prompt

Command : **RECOVER**



The RECOVER command performs recoveries or audit operations on DWG files only. Performing a recover on a DXF file will only open the file.

PURGE 31.3

1. **Choose** File, Drawing Utilities, Purge.

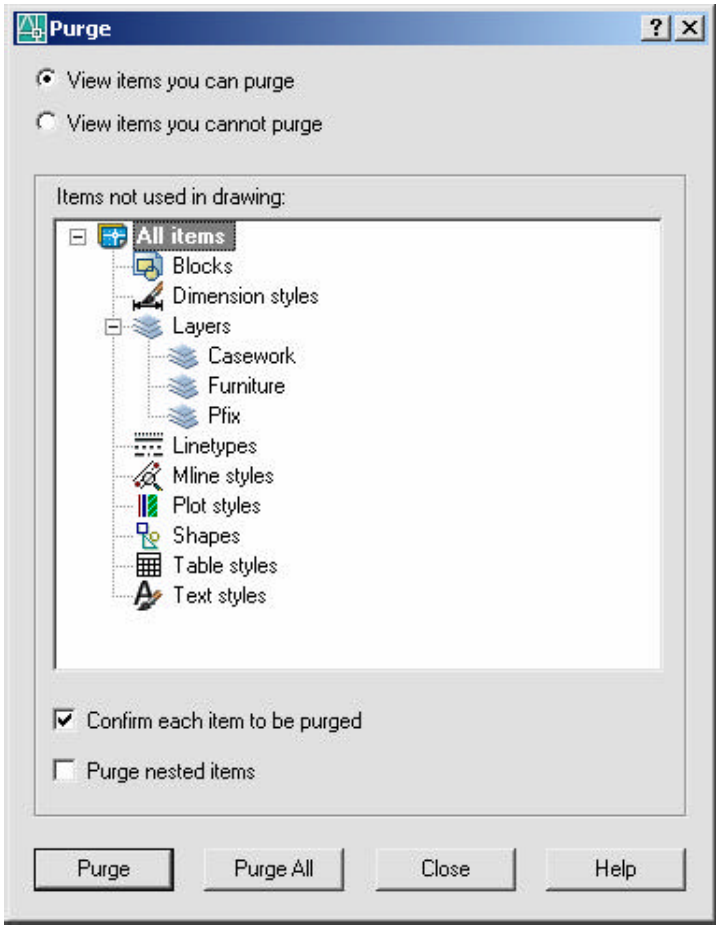
or

2. **Type** PURGE at the command prompt

   Command: **PURGE**

3. **Type** One of the following purge options:

   Purge unused Blocks/Dimstyles/LAyers/LTypes/ SHapes/STyles/Mlinestyles/All:



Rename 31.4

1. **Choose** Format, Rename...

   or

2. **Type** RENAME at the command prompt

   Command: **RENAME**

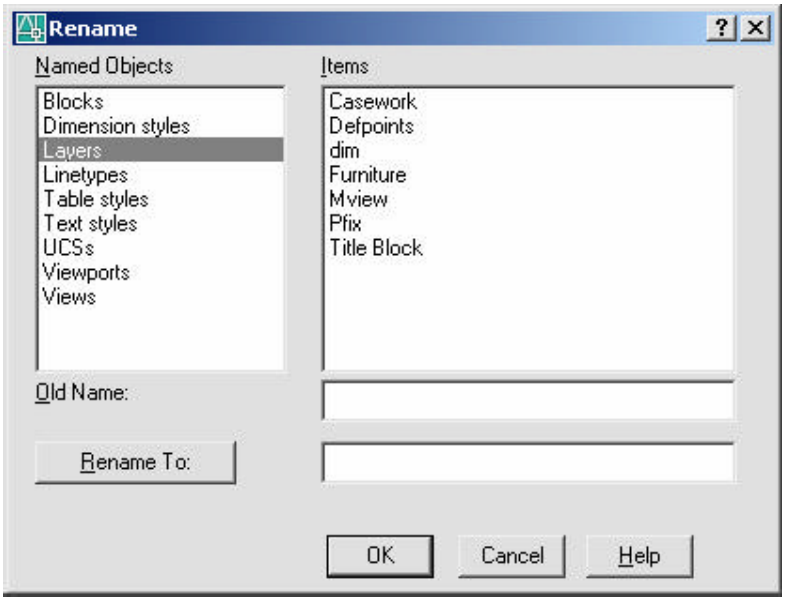
3. **Choose** One of the following options to rename“ Block / Dimstyle / LAyer / LType / Style / Ucs / VIew / VPort:

4. **Type** The old object name

   Old (object) name: **Enter the old name**

5. **Type** The new object name

   New (object) name: **Enter the new name**



Chapter 32
Data Management

Importing Files 32.1

1. **Choose** File, Open

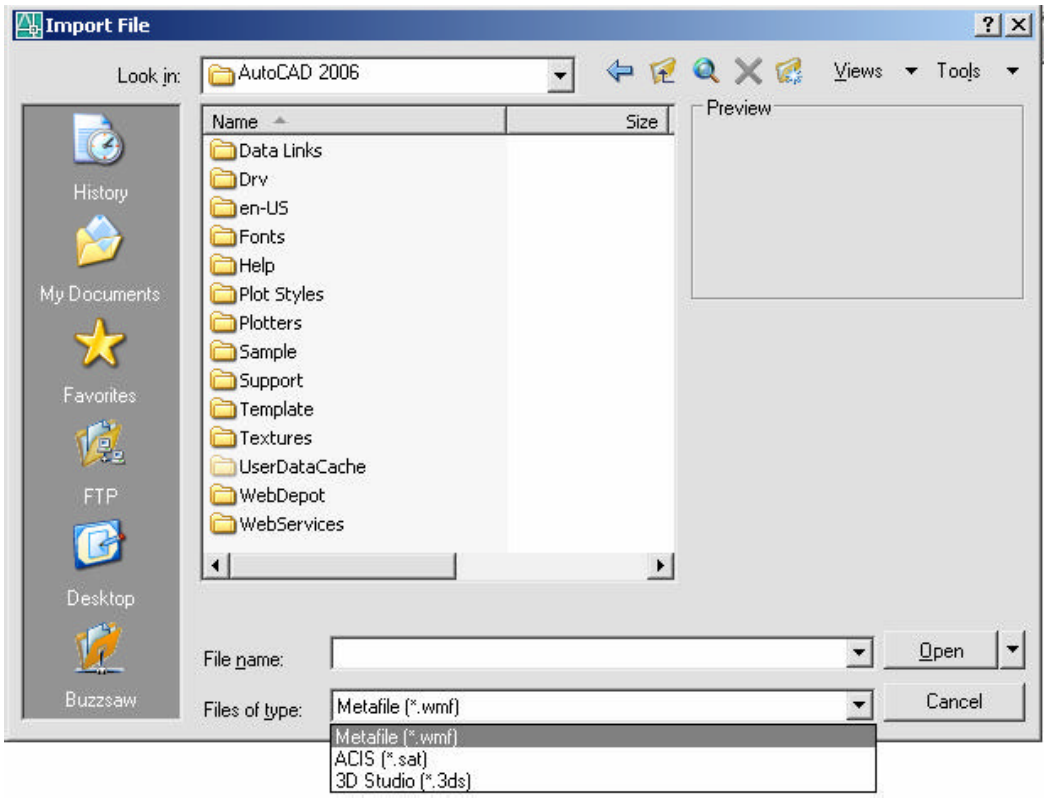
   or

2. **Choose** Insert, 3D Studio, ACIS, DXB, WMF, or ESP

   or

3. **Type** Import at the command prompt.

   Command: **Import**

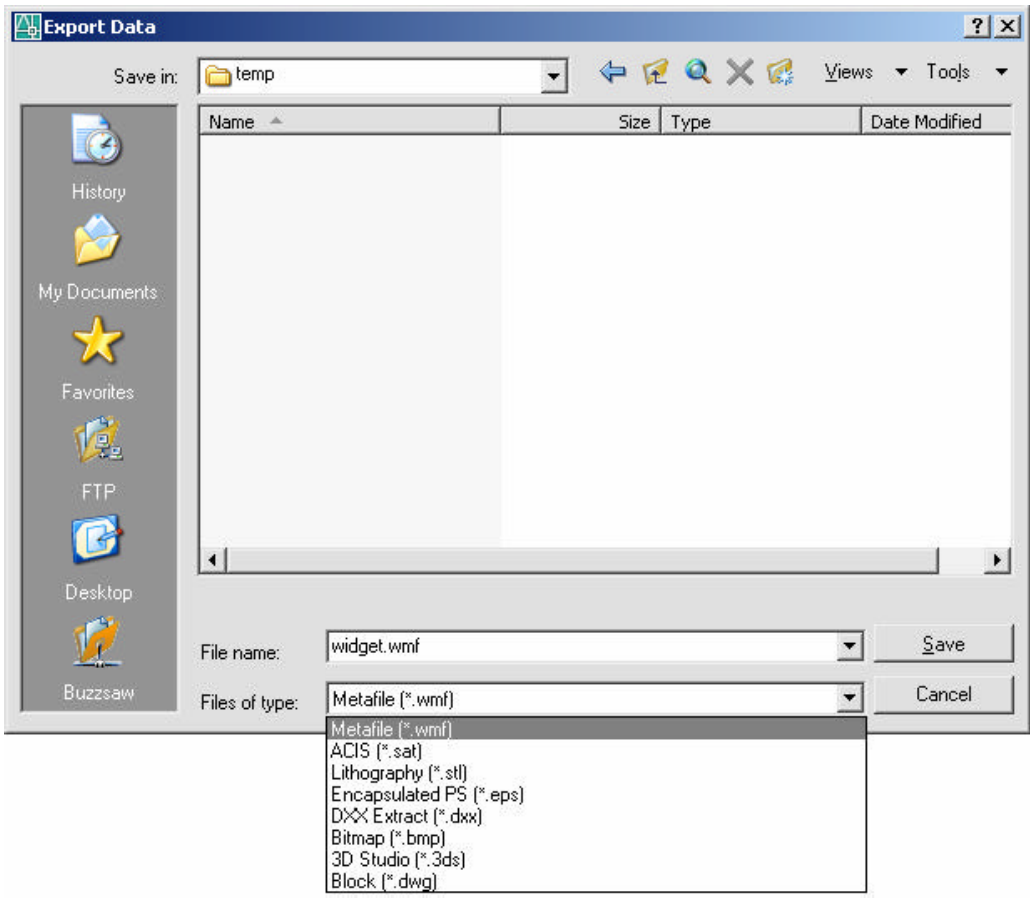


Exporting Files 32.2

1. **Choose** File, Saveas

or

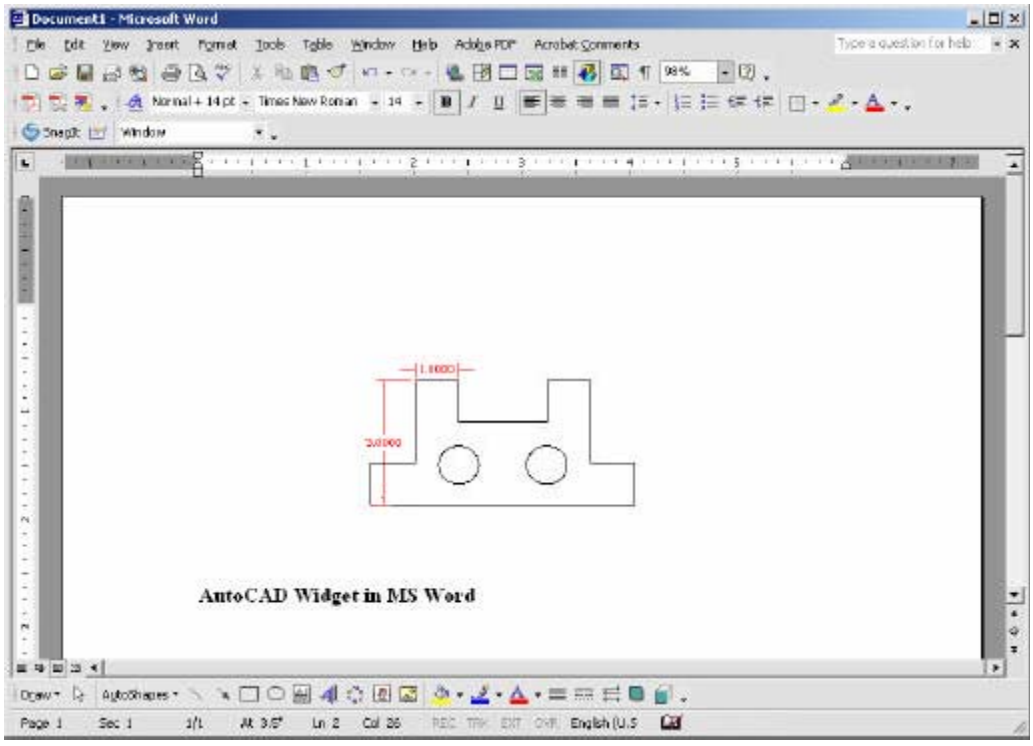
2. **Choose** File, Export



Chapter 33
Object Linking and Embedding

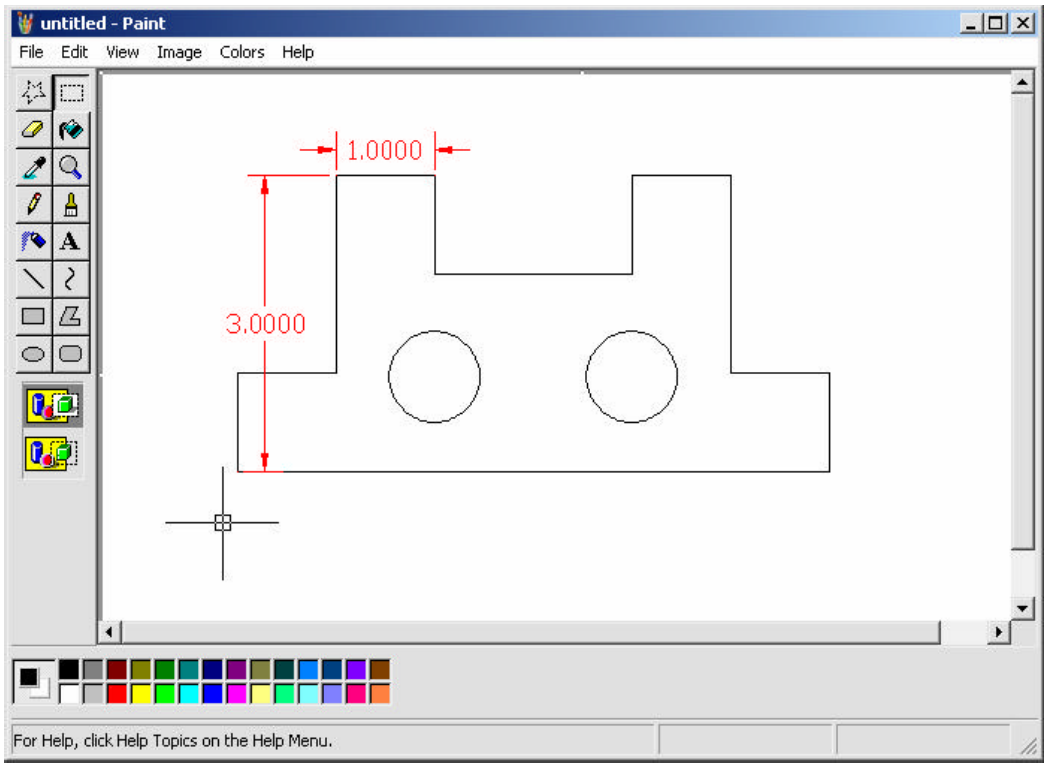
Copying from AutoCAD 33.1

1. **Launch** a Windows program to link to (e.g. Microsoft Word)
2. **Open** an AutoCAD drawing.
3. **Choose** Edit, Copy.
4. **Pick** the AutoCAD objects to copy.
5. **TAB** to the Window's program.
6. **Choose** Edit, Paste.



Print Screen 33.2

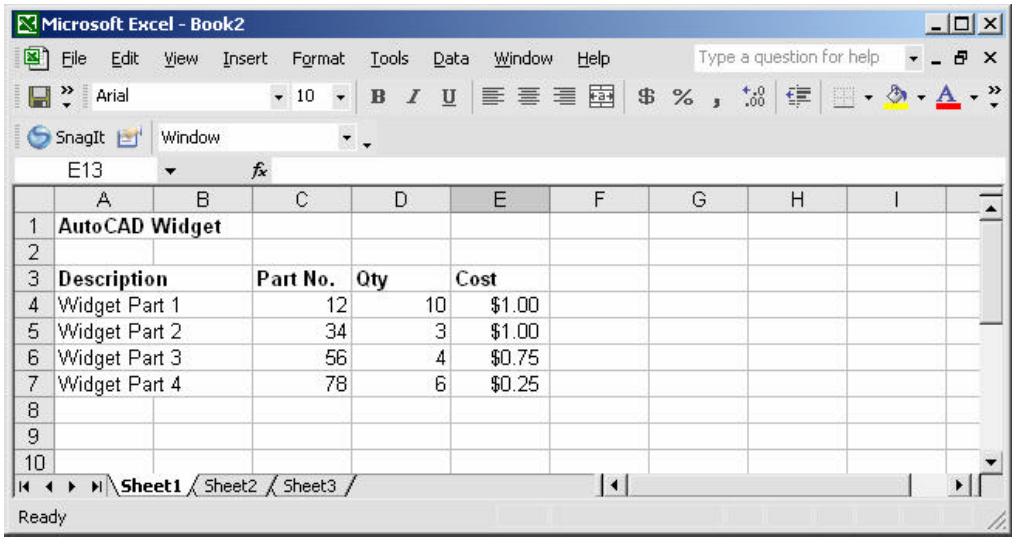
1. **Launch** a Windows program to link to (e.g. Microsoft Word)
2. **Open** an AutoCAD drawing.
3. **Press** PRINT SCREEN on the keyboard.
4. **TAB** to the Window's program.
5. **Choose** Edit, Paste.



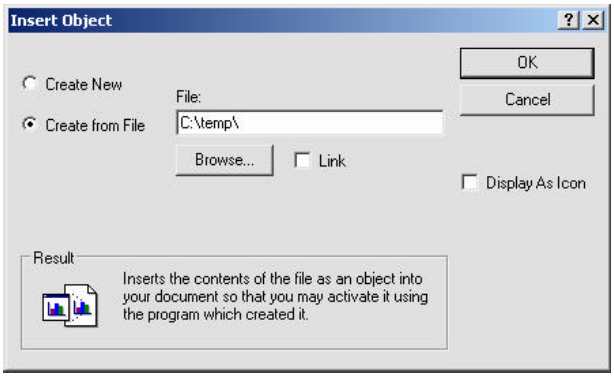
OLE Linking to AutoCAD 33.3

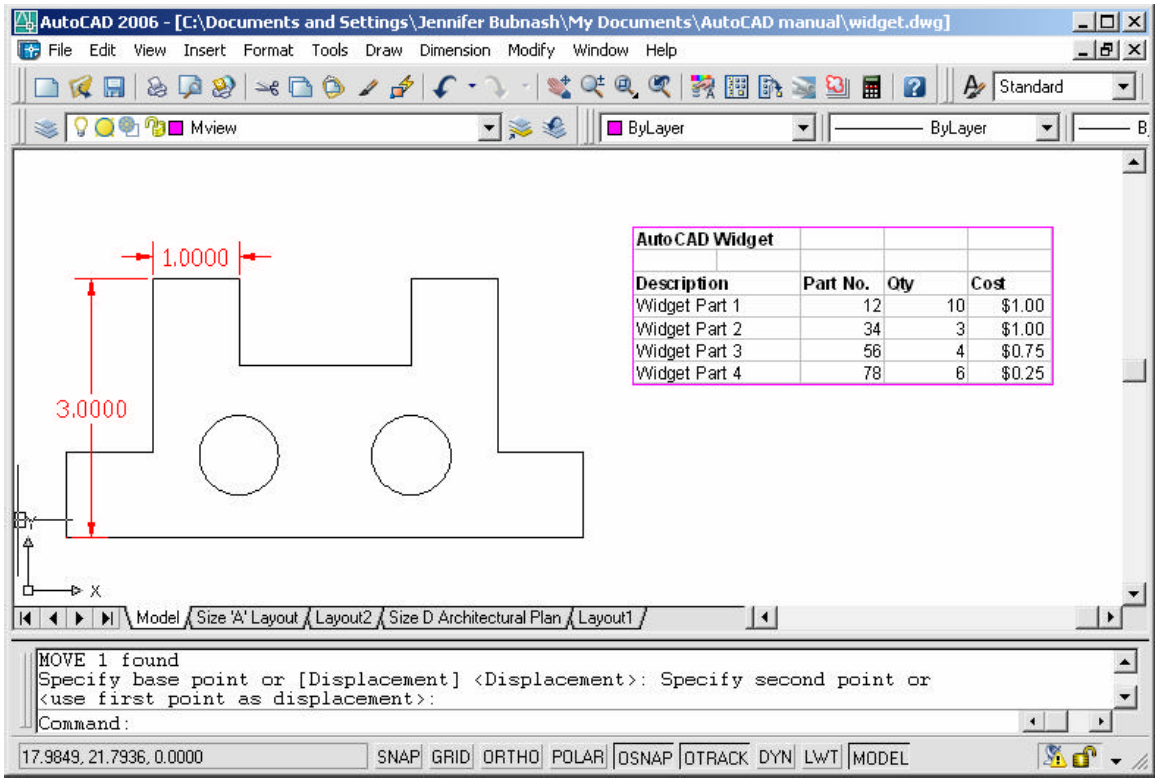
A linked object remains associated with its source file. When you edit a linked object in AutoCAD, the source file changes. When you edit the object in the source file, the linked object in AutoCAD changes.

1. **Launch** a Windows program to link from (e.g. Microsoft Excel)
2. **Create** a spreadsheet to bring into AutoCAD.



3. **Open** an AutoCAD drawing.
4. **Choose** Insert, OLE Object.
5. **Choose** browse to pick a file to link to AutoCAD.
6. **Pick** a location in the drawing to place the OLE object.





7. **Double Click** to edit that object in the original program.

TIPS:

- Spreadsheets that are imported into AutoCAD drawings with OLE are limited in size. If your spreadsheet is too large, you can reduce the column width and row height, reduce the font size, or paste the spreadsheet in separate parts to break the OLE object into smaller OLE objects.
- OLE objects are inserted in an AutoCAD drawing on the current layer. Turn off or freeze a layer to suppress the display of OLE objects on that layer.
- Set the system variable OLEHIDE to display or suppress the display of all OLE objects in paper space, model space, or both.

Hyperlinking 33.4

1. **Choose** Insert, Hyperlink

   or

2. **Press** CTRL + K

   or

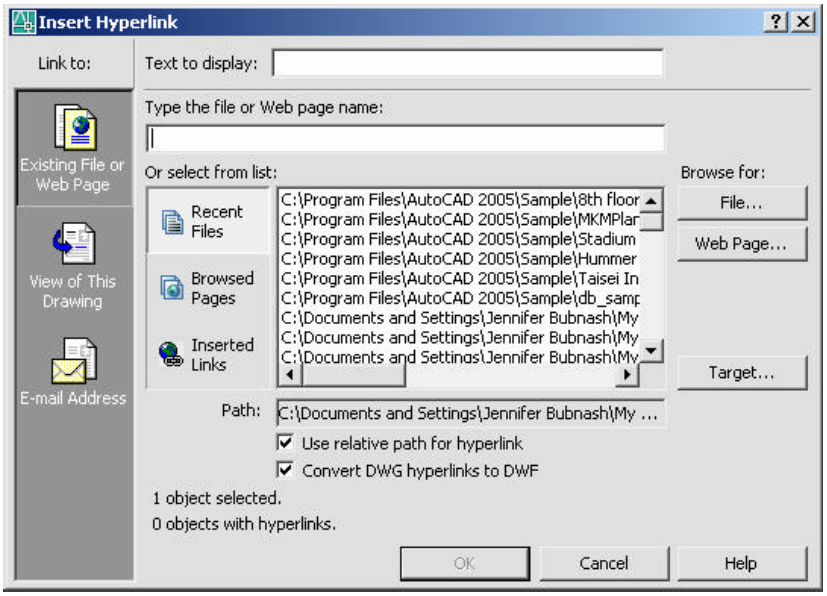
3. **Type** HYPE RLINK at the command prompt

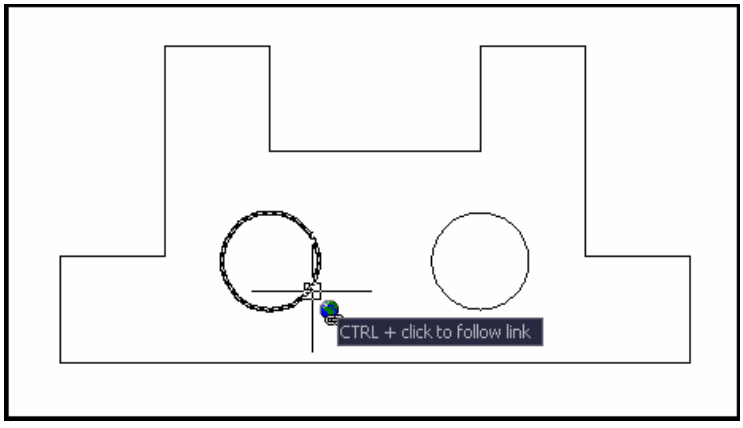
   Command: **hyperlink**

4. **Select** the object to hyperlink

   Select objects: 1 found

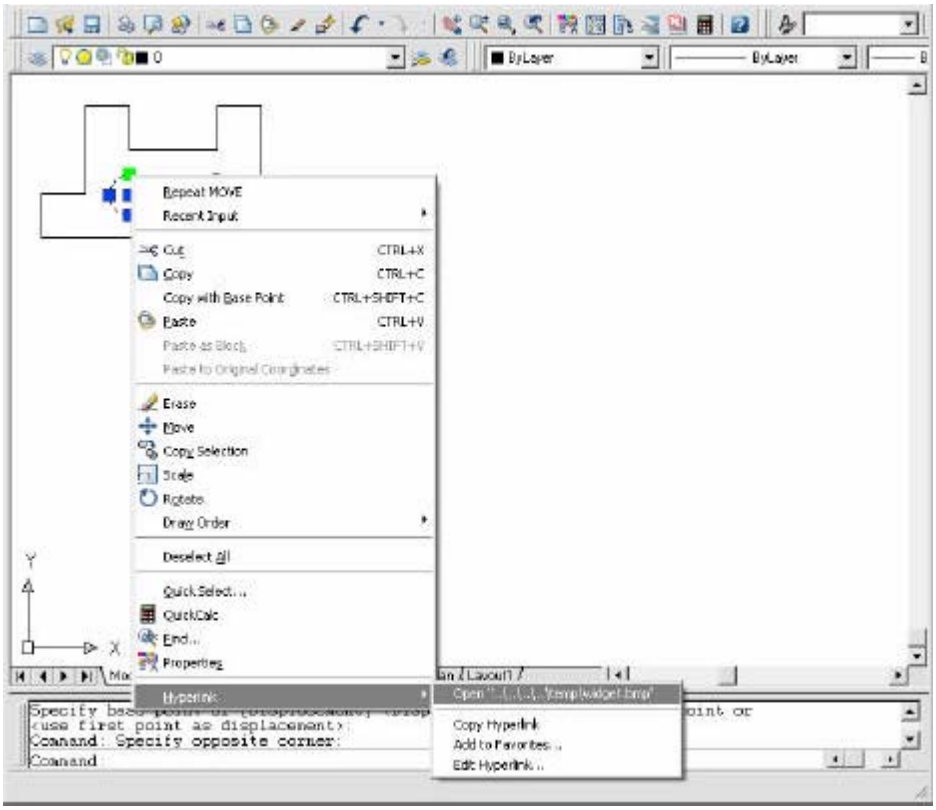
5. **Choose** an option in the hyperlink dialog box.

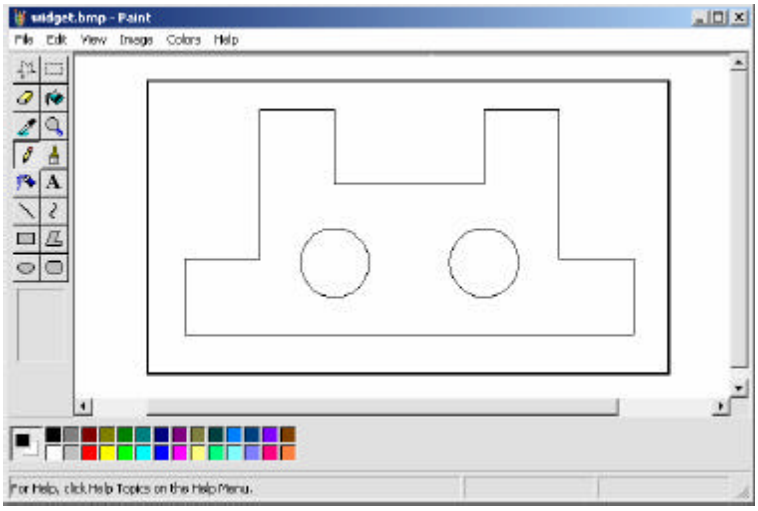




Opening a Hyperlink 33.4

1. **Move** the cursor to the object with the hyperlink.
2. **Click** with your right mouse button.
3. **Choose** Hyperlink from the menu.
4. **Open** the file from the menu.

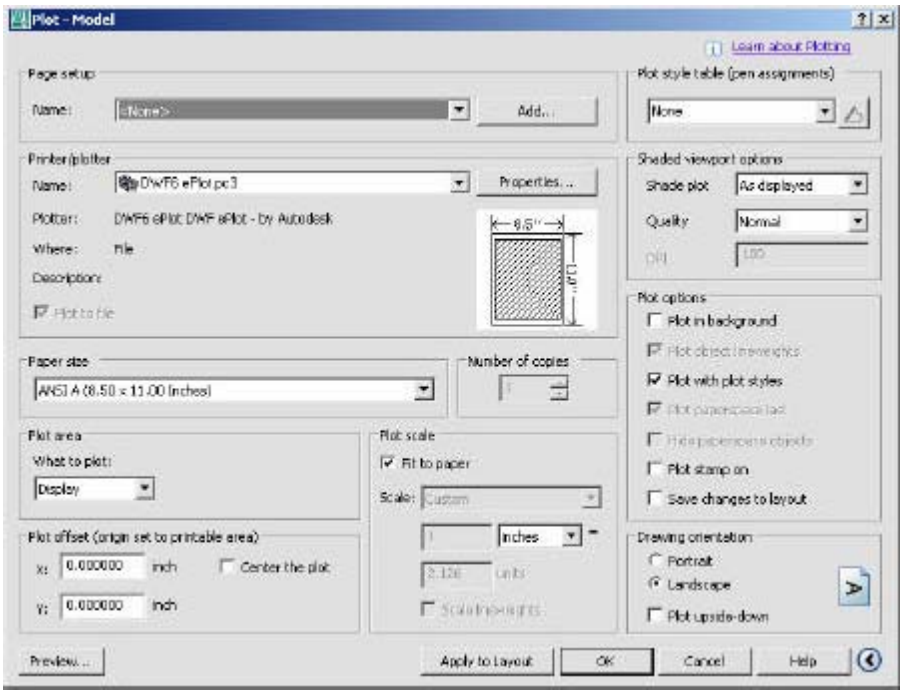




Chapter 34
Communication and Collaboration Tools

34.1 Plotting to the WEB

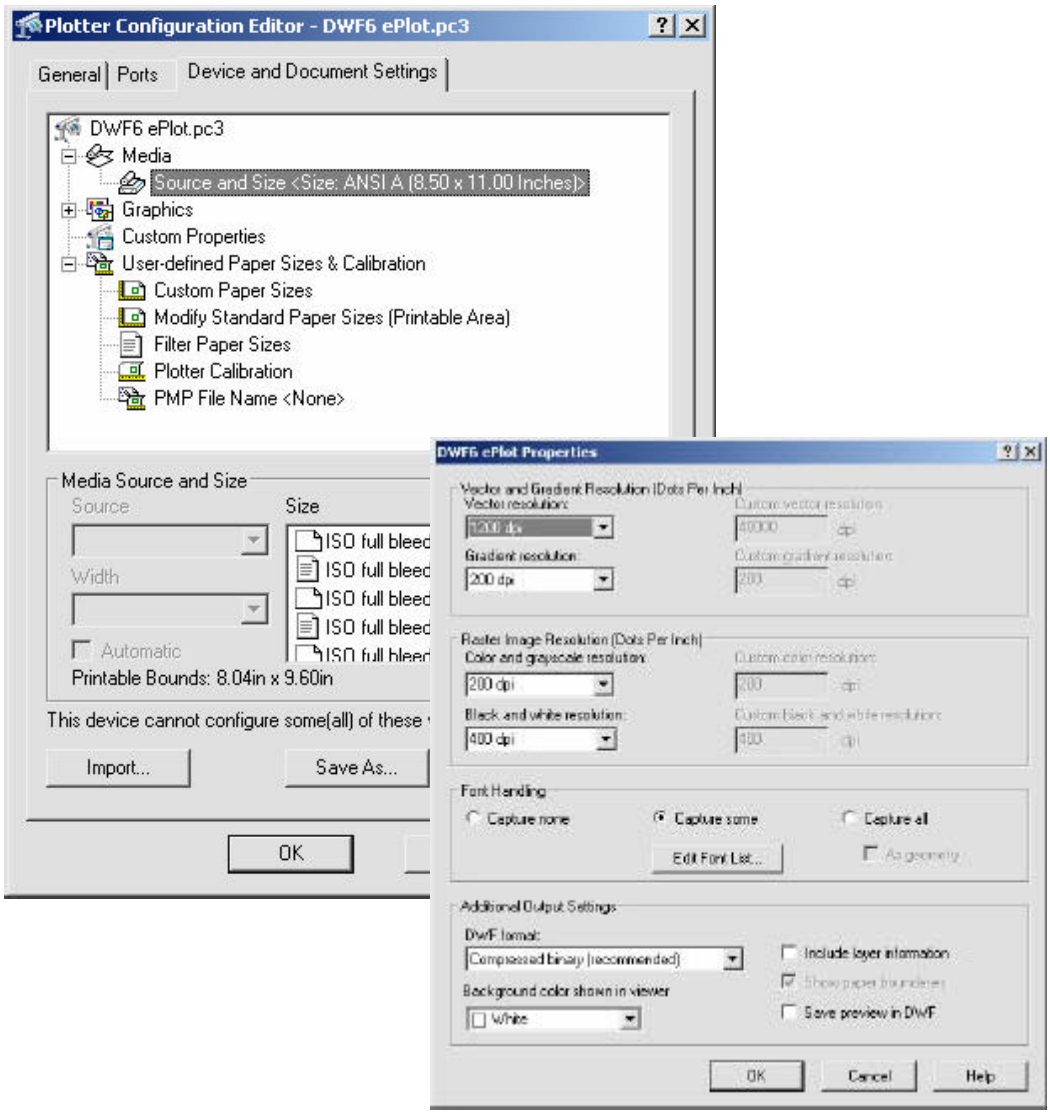
1. **Type** **PLOT** at the command prompt.
   Command: **plot**
2. **Choose** the Plot Device TAB.
3. **Choose** the dropdown list for Plotter Configuration.
4. **Plot** to a .DWF, JPG, or PNG file.



- DWF format does not compress the drawing file.
- JPEG format uses lossy compression; that is, some data is deliberately discarded to greatly reduce the size of the compressed file.
- PNG (Portable Network Graphics) format uses lossless compression; that is, no original data is sacrificed to reduce the size of the file.

34.2 Configuring DWF Files

1. **Type** PLOT at the command prompt.
   Command: **plot**
2. **Choose** the DWF plot configuration option.
3. **Choose** the **Properties...**button.
4. **Choose** **Custom Properties….**
5. **Slide** the slider bar to extreme for a clearer resolution on the DWF file.



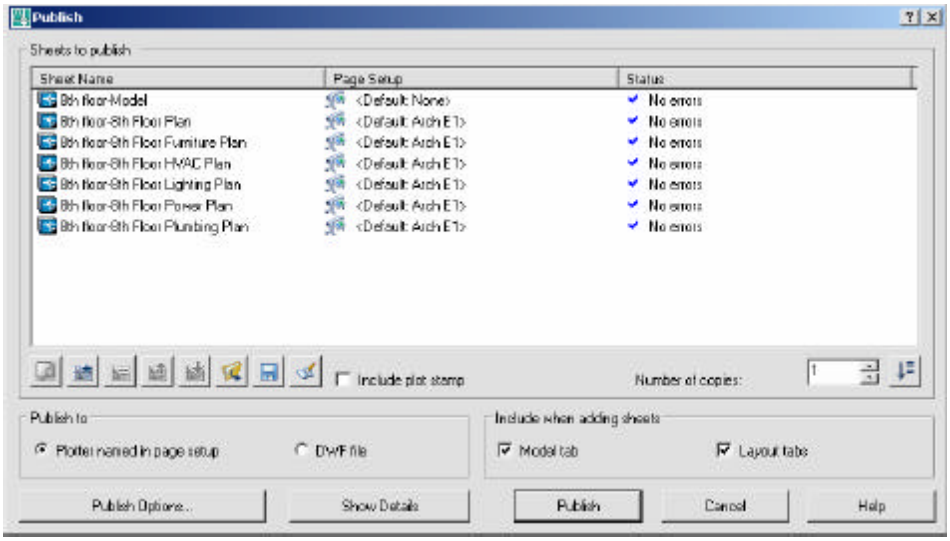
34.3 Publish Command

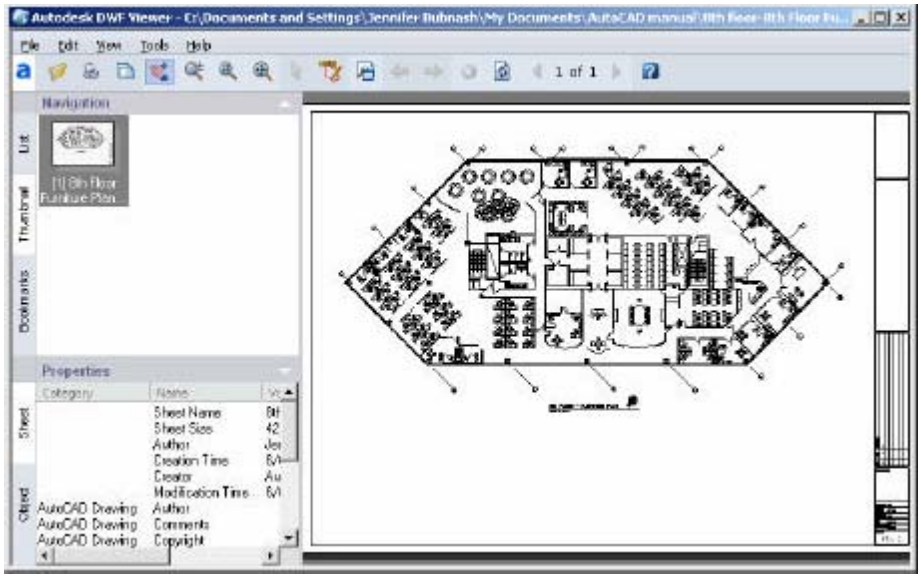
1. **Choose** the Publish to WEB icon from the Standard Toolbar. 

or

2. **Choose** File, Publish
3. **Type** PUBLISH at the command prompt.

   Command: **publish**





34.4 Publishing WEB Pages

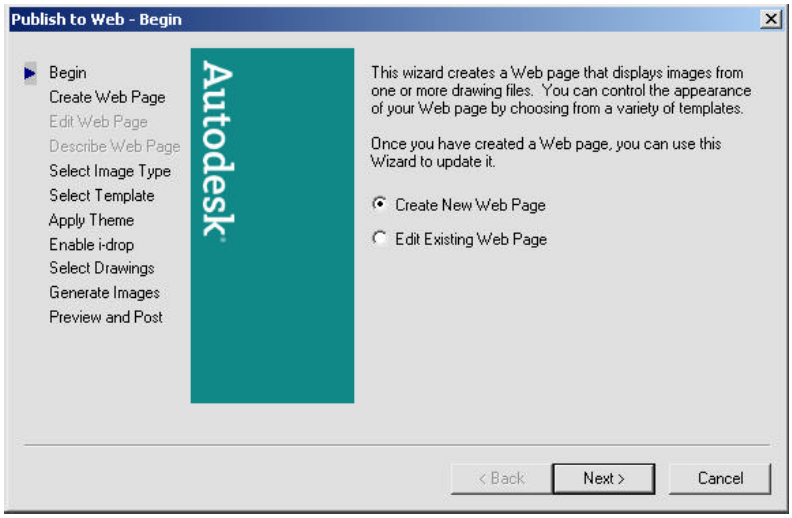
1. **Choose** File, Publish to WEB.

or

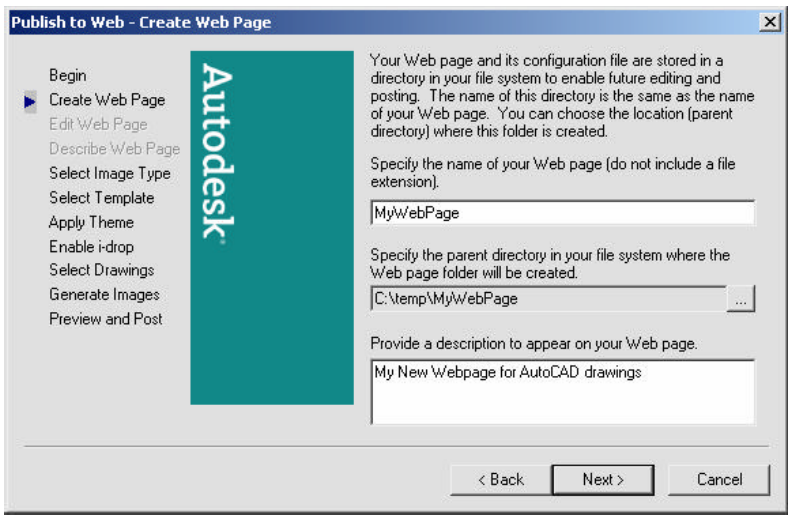
2. **Type** PUBLISHTOWEB at the command prompt.

   Command: **PUBLISHTOWEB**

3. **Click** Create New Web Page and Next.

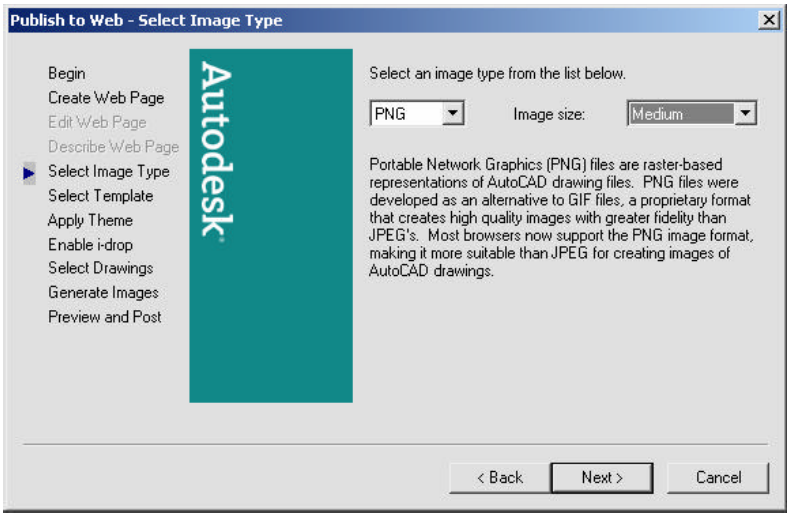


4. **Specify** a name for the WEB page, location and description for the new WEB page and click Next.
   (NOTE: Save the WEB pages to C:\TEMP)

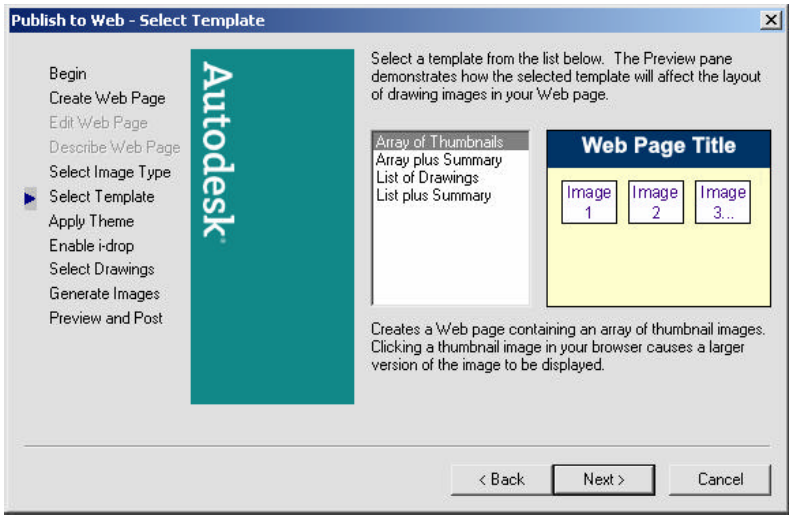


4. **Select** PNG as the image type for the drawings and Image Size “Medium” and click Next.

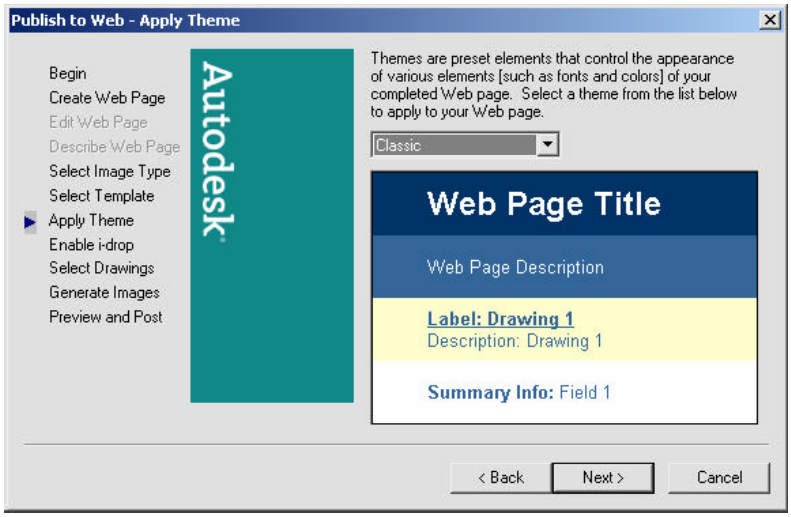
NOTE: If you have the Express Viewer loaded, you can pick DWF files to view. This will allow you to zoom and pan the drawings.



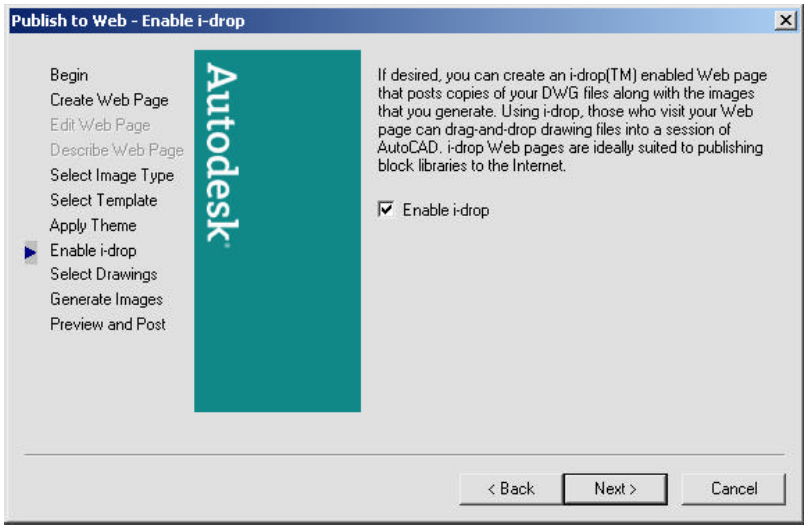
5. **Select** **Array of ThumbNails** as the Template type and click Next.



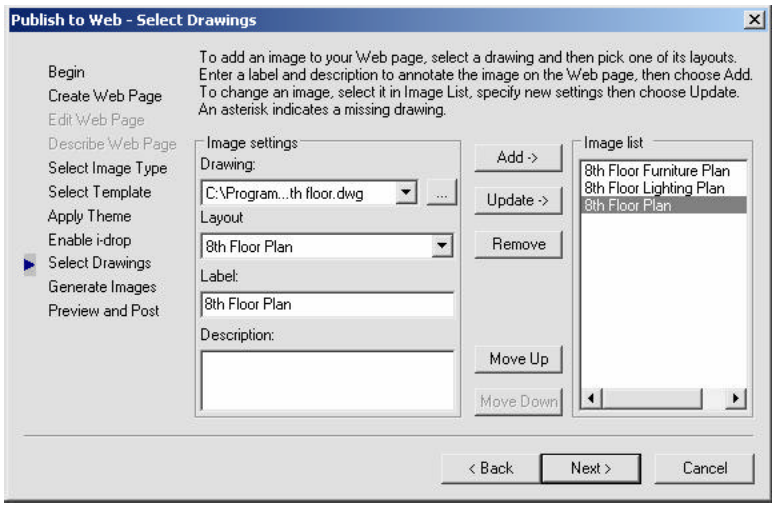
6. **Select** a Theme (color) and click Next.



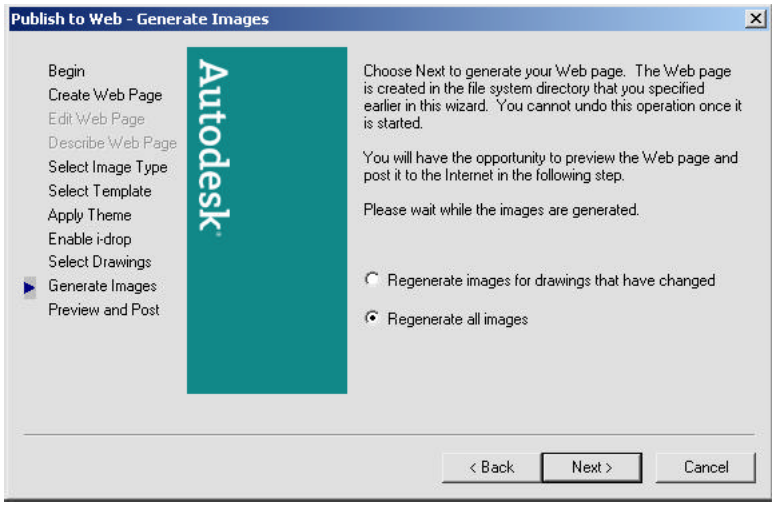
7. **Select** Enable i-drop and Click Next



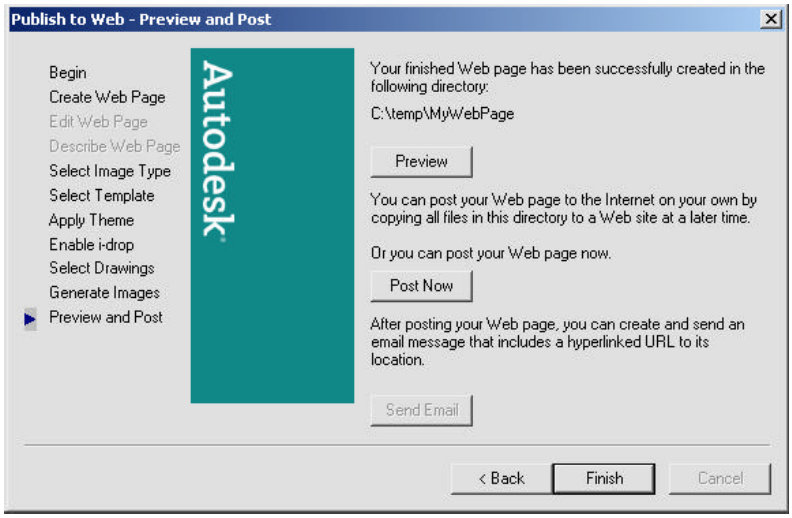
8. **Select** drawings and/or layouts to place on the WEB page.

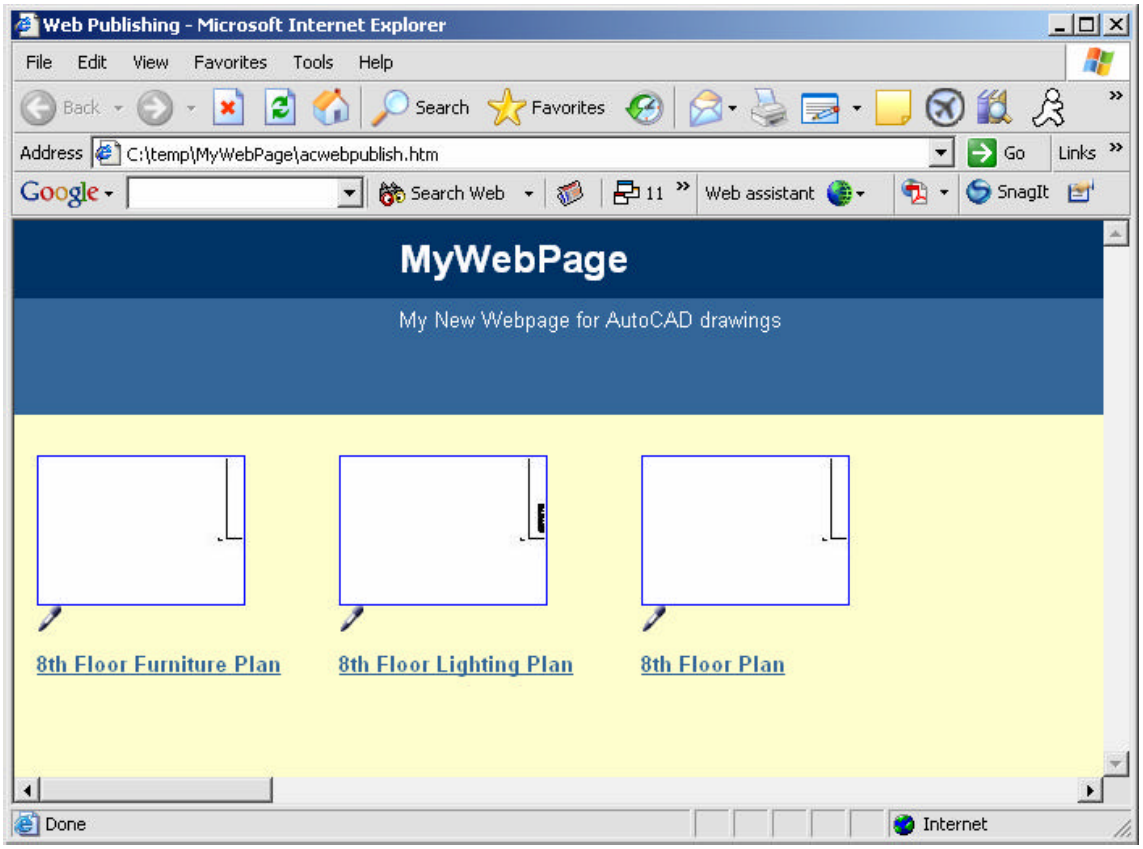


9. **Click** Regenerate all images and Next.



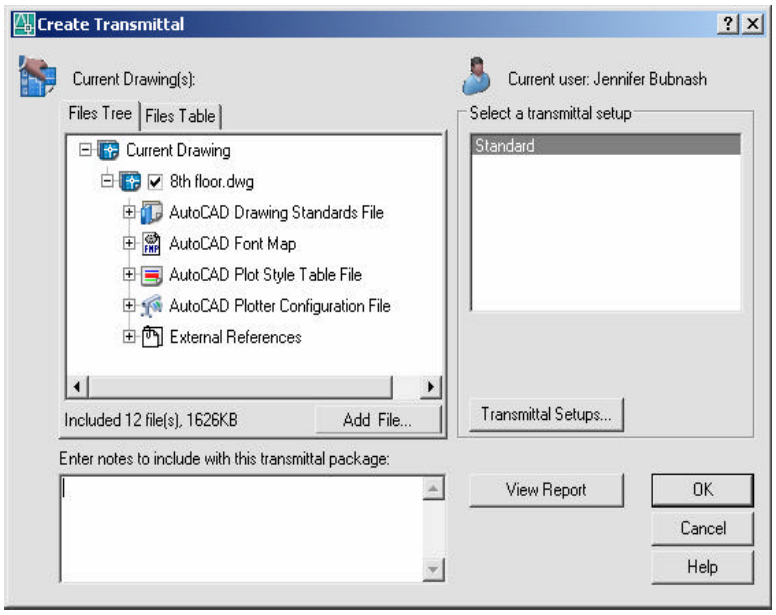
10. **Click** Preview and Finish.





34.4 e-Transmit

1. **Type** **ETRANSIT** at the command prompt.
   Command:**etransmit**



e-Transmit options

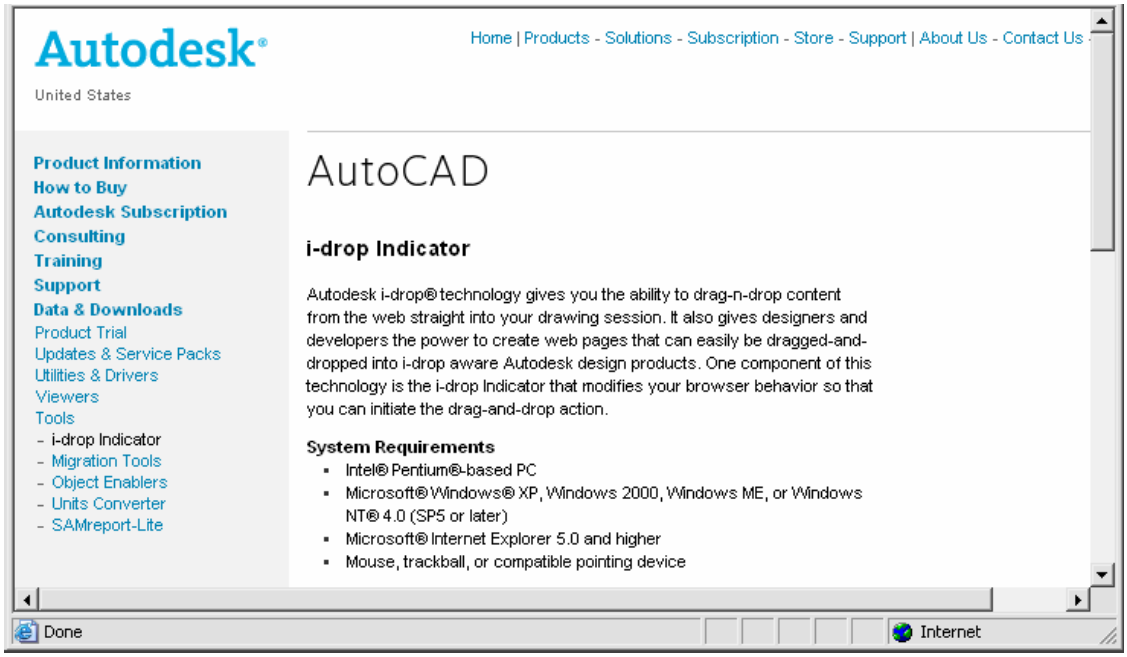
.EXE (files are self extracti ng)

.ZIP (PKZIP or WINZIP is needed to extract files)

34.6 i-Drop

i-drop allows users to drag a drawing from an i-drop handle on an i-drop supported WEB site to an open AutoCAD drawing. Choose the i-drop option when publishing to the WEB to create an i-drop enabled WEB page.

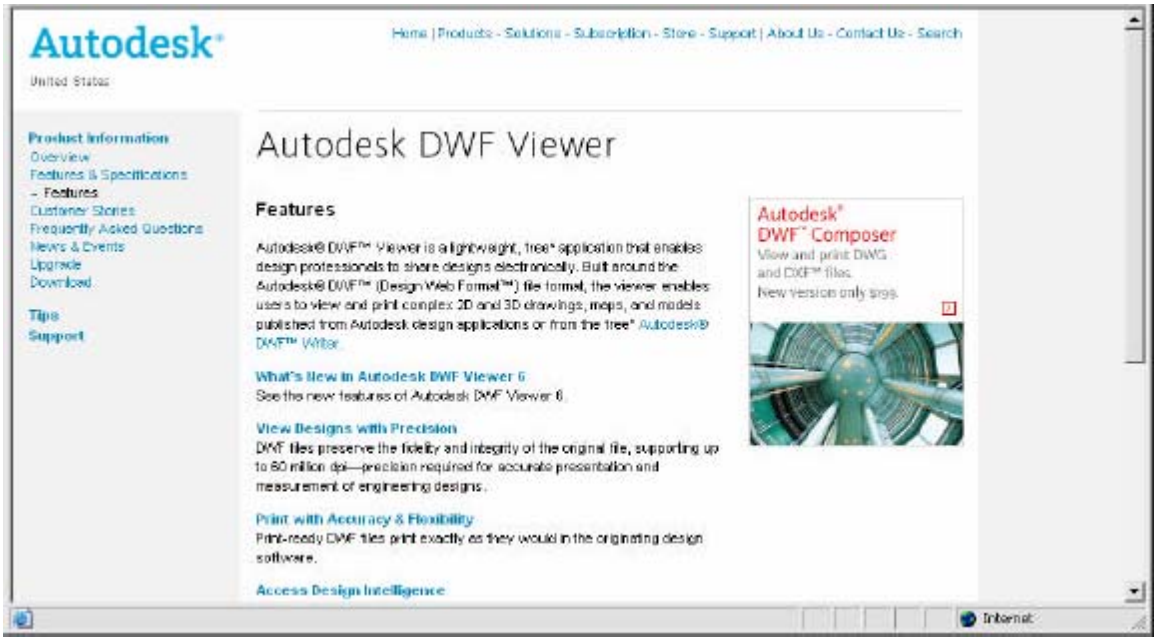
1. **Type** **http://www.autodesk.com/idrop** to learn more about Autodesk's i-drop technology.



Chapter 35
AutoCAD WEB Viewers

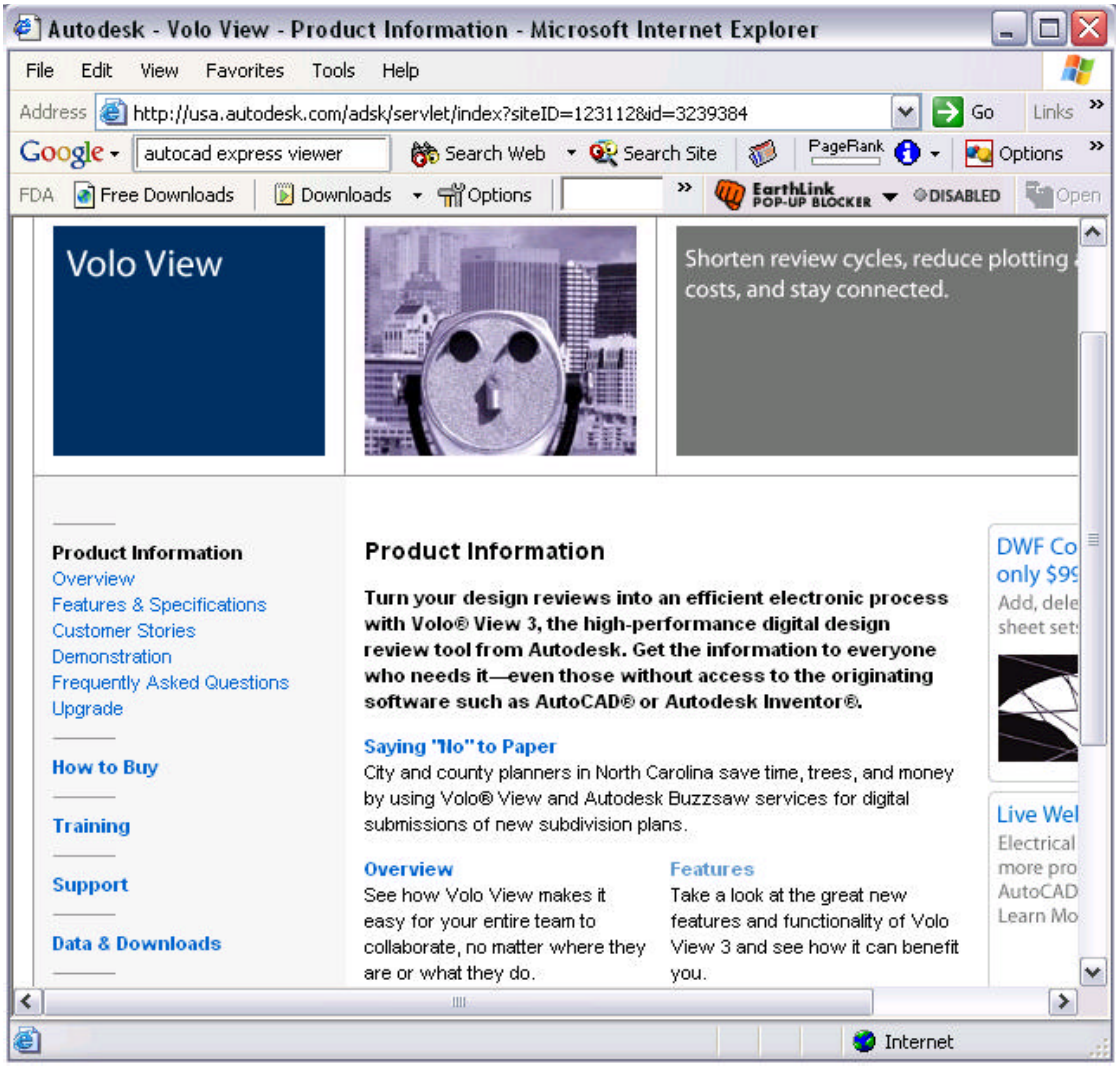
AutoCAD DWF Viewer 35.1

1. **Launch** Your WEB Browser.
2. **Type** http://www.autodesk.com/products
3. **Click** on Autodesk DWF View from the list of Autodesk products.
4. **Browse** the site for information or to download a free copy of the DWF Viewer.



Volo View 35.2

1. **Launch** Your WEB Browser.
2. **Go to** http://www.autodesk.com/products/
3. **Click** Volo View to learn more about the Volo View application.

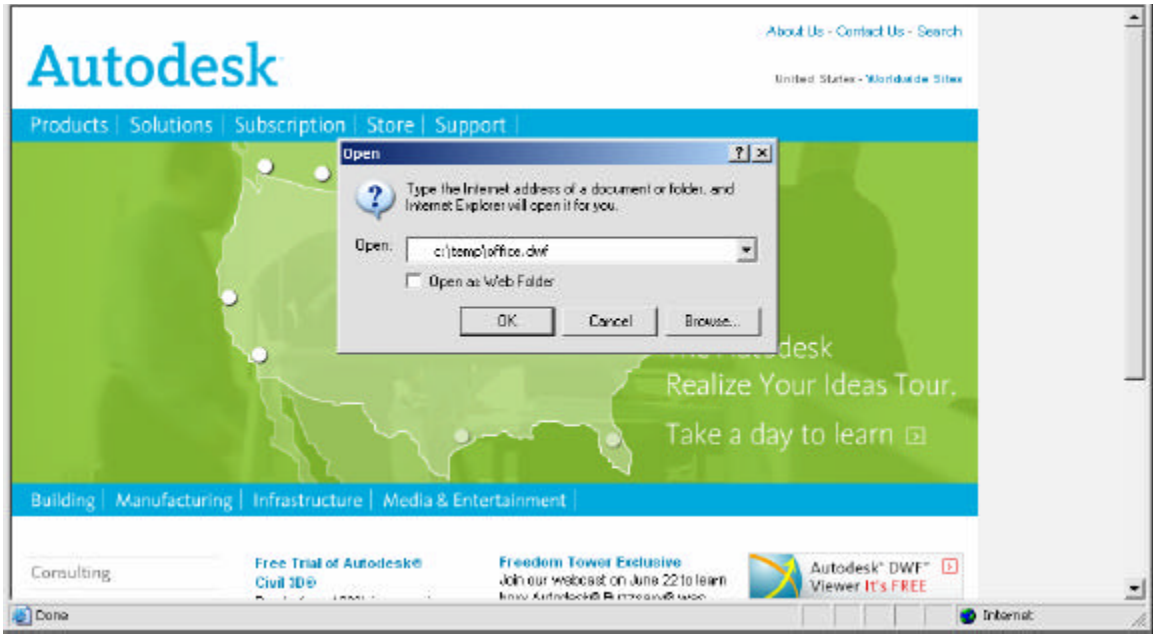


CAD Viewer 35.3

1. **Launch** Your WEB Browser.
2. **Go to** **http://www.cadviewer.com/**

Internet Explorer 35.5

1. **Launch** Internet Explorer 5.0 or later.
2. **Choose** File, Open.
3. **Browse** to C:\TEMP and open a .DWF file to view.



AutoCAD Related WEB Sites 35.6

1. **Launch** Internet Explorer.
2. **Click** on one of the following WEB sites.

or

3. **Search** for AutoCAD related topics on the Internet on an Internet search engine

http://www.cadalog.com

http://www.cadalyst.com/

http://www.cadsoftware.com/

http://www.3dcafe.com/

http://www.mcneel.com

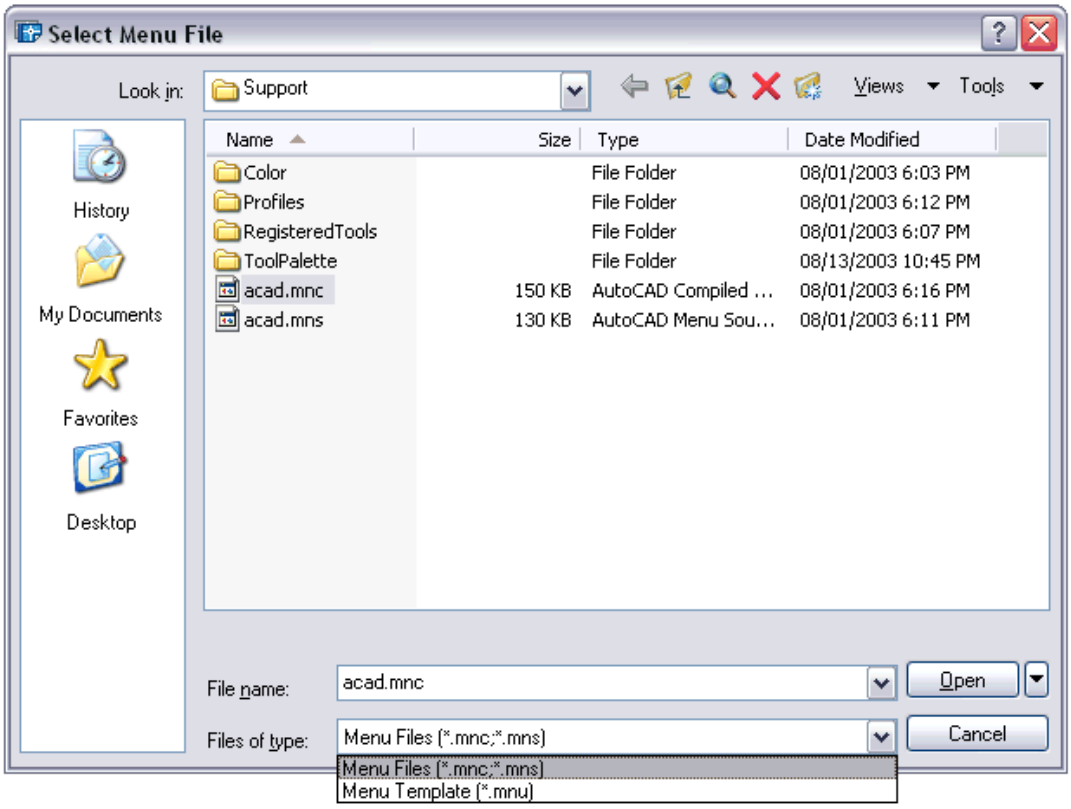
http://www.caddepot.com/

http://www.caddigest.com/

Chapter 36
Customization

Menu Loading 36.1

1. **Type** MENU at the command prompt.
   Command: **Menu**
2. **Choose** a menu (mnu) file to load.

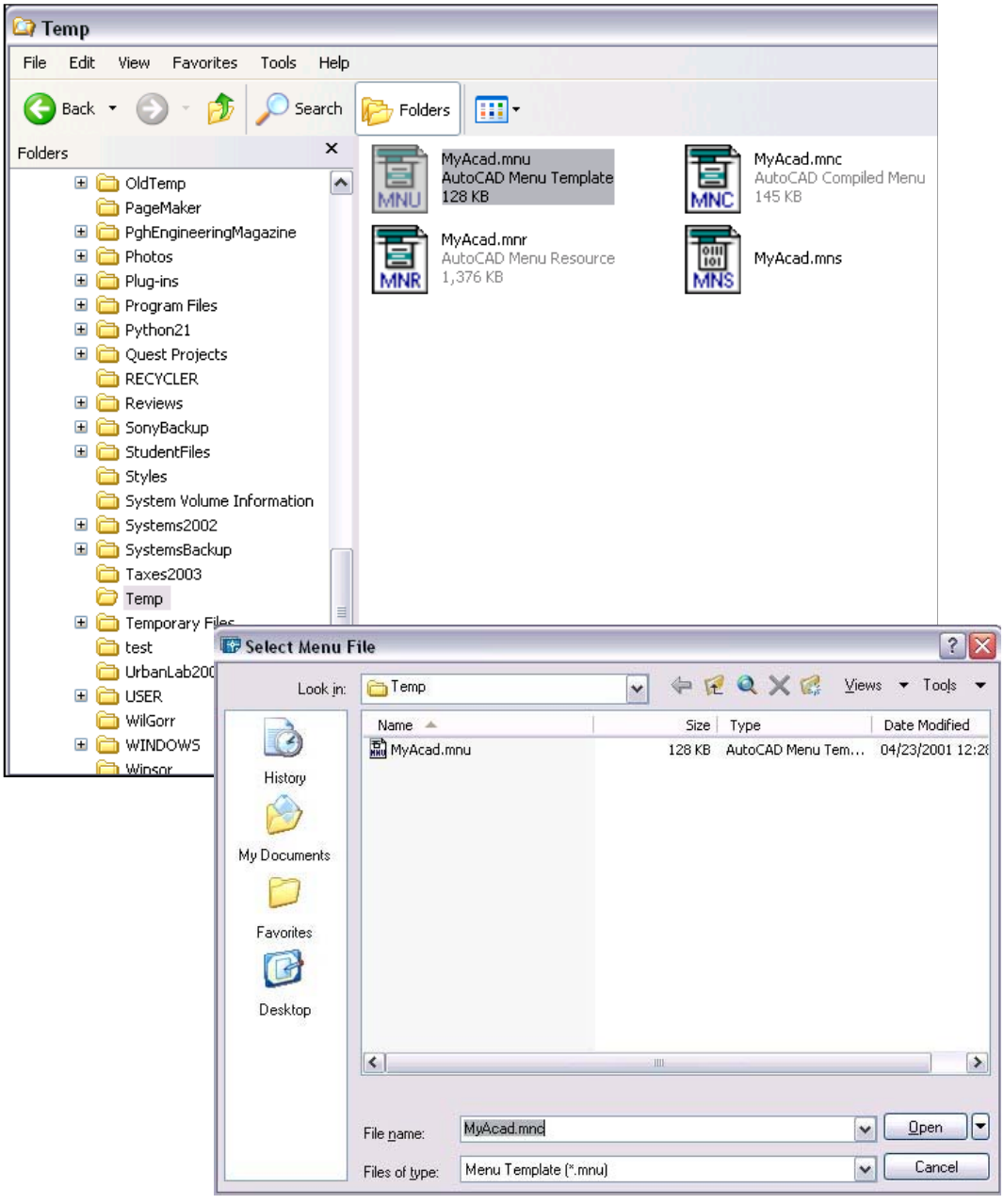


Creating New Menu Files 36.2

1. **Copy** ACAD.MNU to a new file name such as **MYACAD.mnu**

2. **Type** MENU at the command prompt to load the new menu.

Command: MENU



Menuload 36.3

1. **Choose** Tools, Customize, Menus…

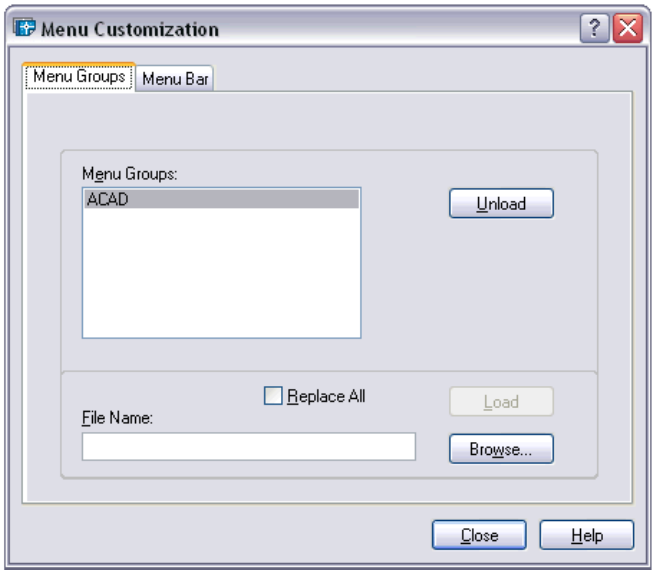
   or

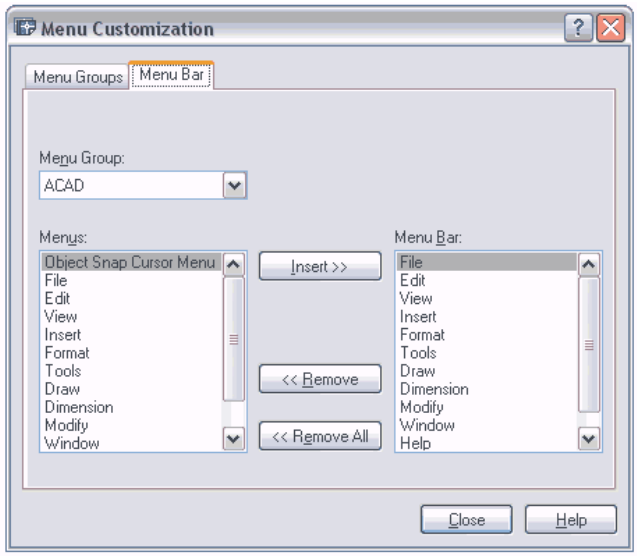
2. **Type** MENULOAD at the command prompt.

   Command: **MENULOAD**

3. **Choose** the Menu Bar Tab.

4. **Choose** Add or Remove to modify the appearance of the Pulldown menu.

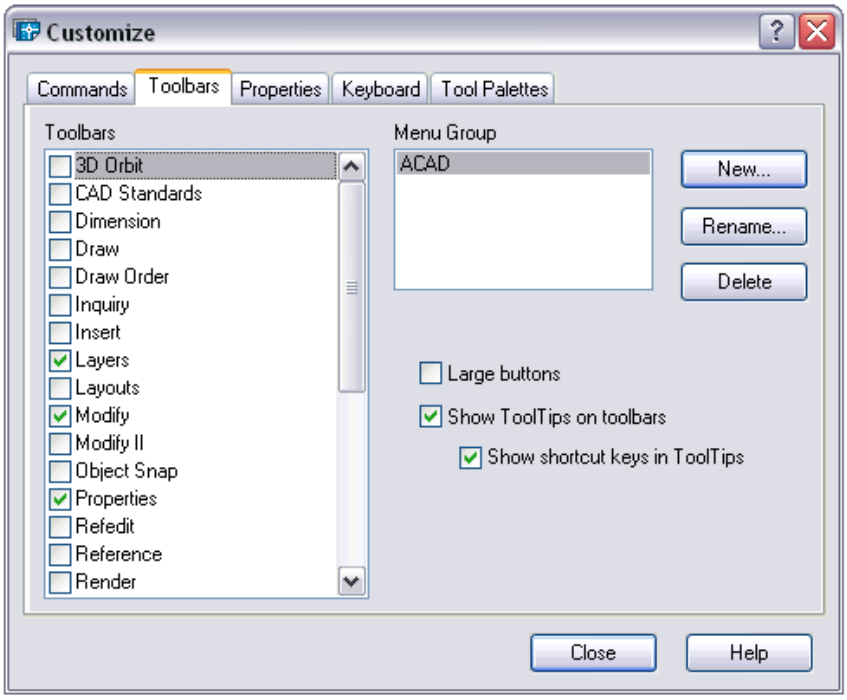




Customizing Toolbars 36.4

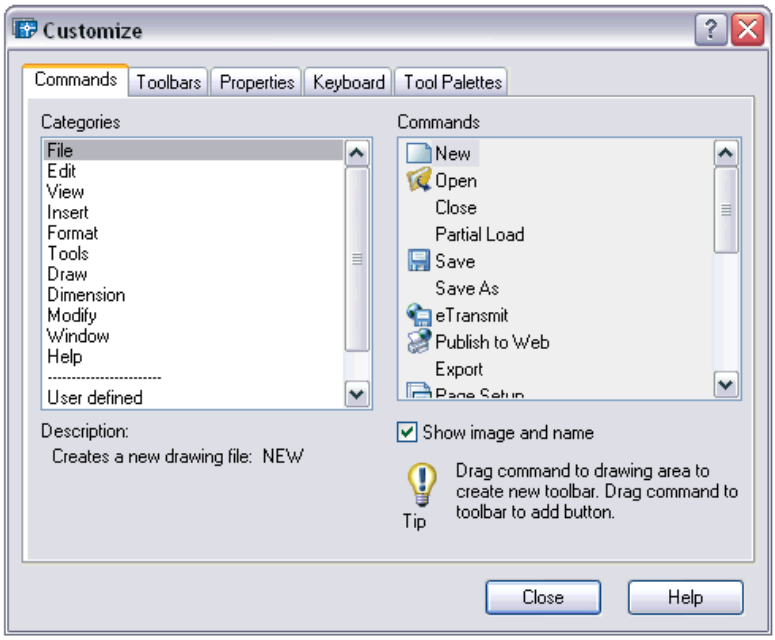
Create New Toolbars

1. **Choose** View, Toolbars... or
2. **Type** TBCONFIG or TOOLBAR at the command prompt.
   Command: **TBCONFIG or TOOLBAR**

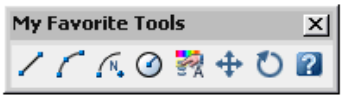


3. **Choose** the **New...**button.
4. **Type** the name of the new toolbar.
5. **Click** OK.

6. Choose the Commands...TAB.



7. **Drag** commands from each menu and drop onto your toolbar.

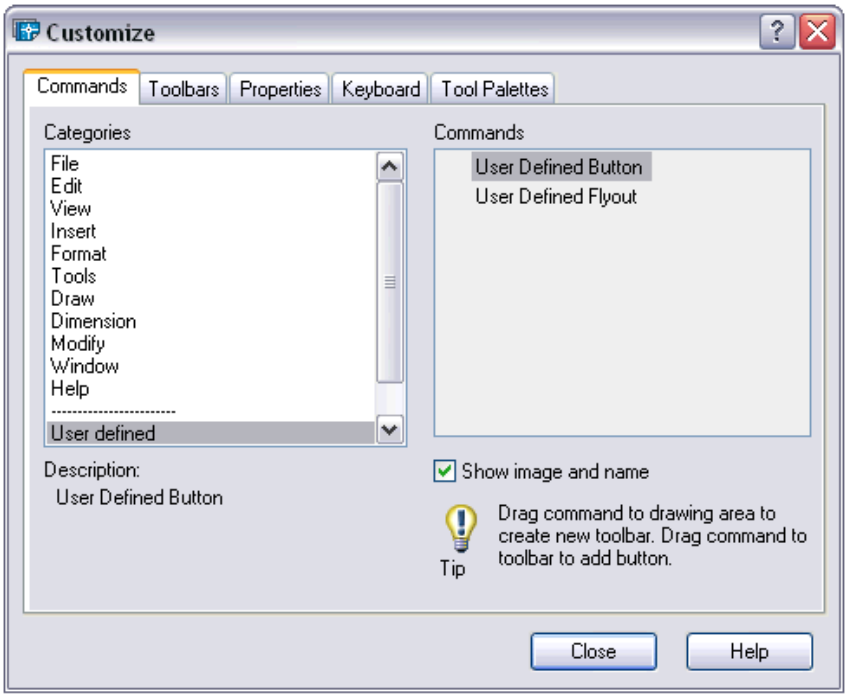


TIP: To copy a tool from another toolbar, press and drag the tool to the new toolbar.

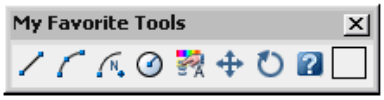
8. **Choose** Close to close the Customize dialog box.

User Defined Buttons 36.5

1. **Choose** View, Toolbars...
2. **Choose** the Commands...TAB.
3. **Click** **User Defined** from the Categories section.

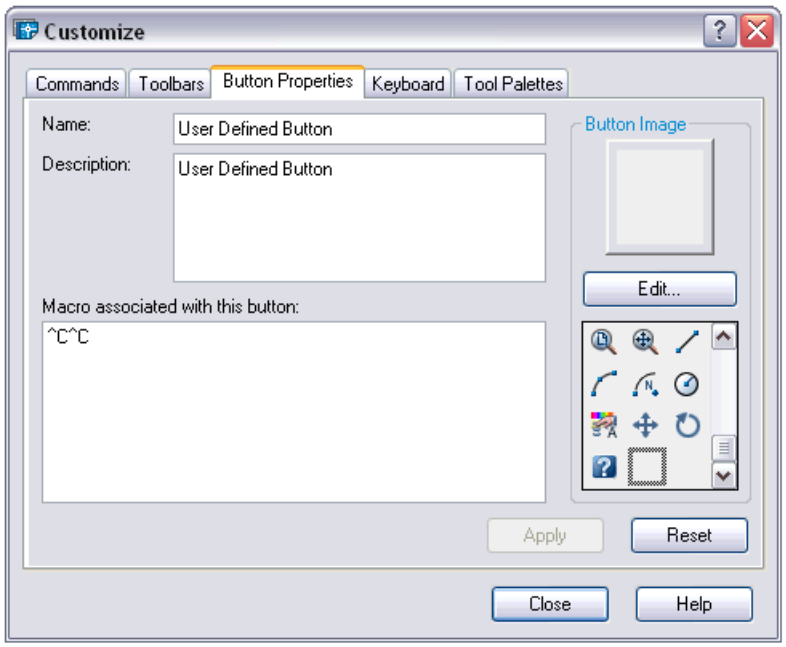


4. **Drag** a user defined button to your toolbox.

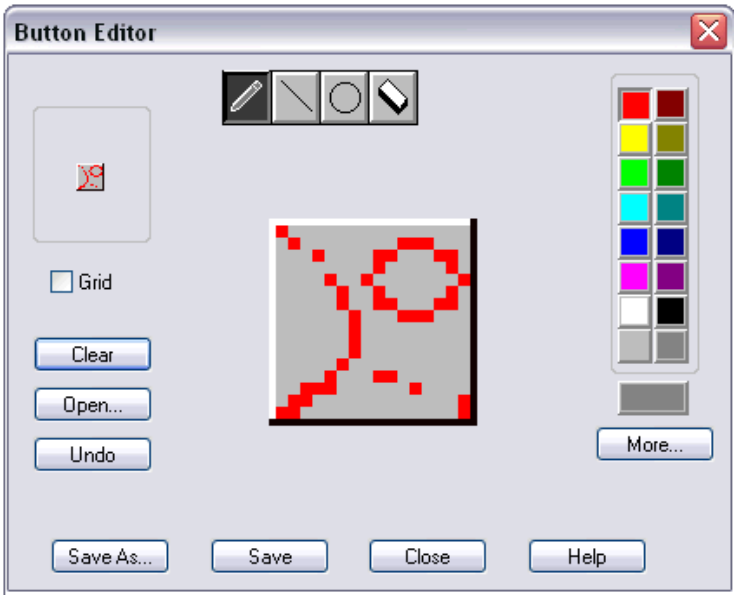


5. **Double Click** the new button.

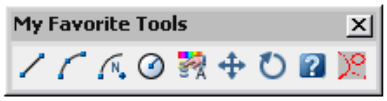
   The Button Properties dialog box will appear.
6. **Choose** Edit...from the Button Properties window.



7. **Draw** the desired symbol by using the draw tools.
8. **Close** the Button Editor (save if necessary).
9. Press Apply.



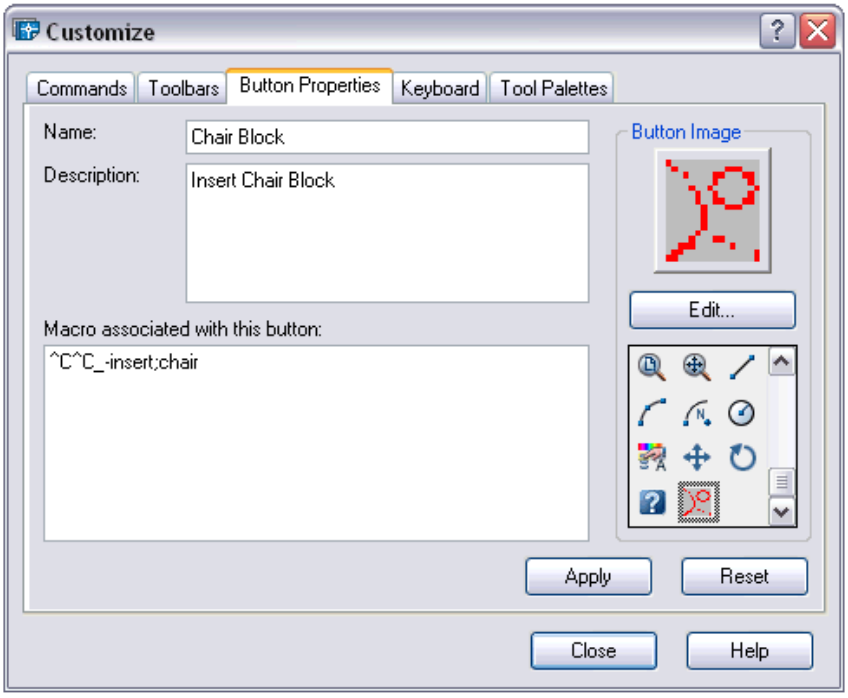
10. **Close** the button properties window by selecting the X in the upper right corner.
11. **Close** the Toolbars window.



Macros 36.6

1. **Click** Your right mouse button on the button you wish to modify or create.
2. **Type** the macro command in the macro window.

Macro that inserts a block called chair



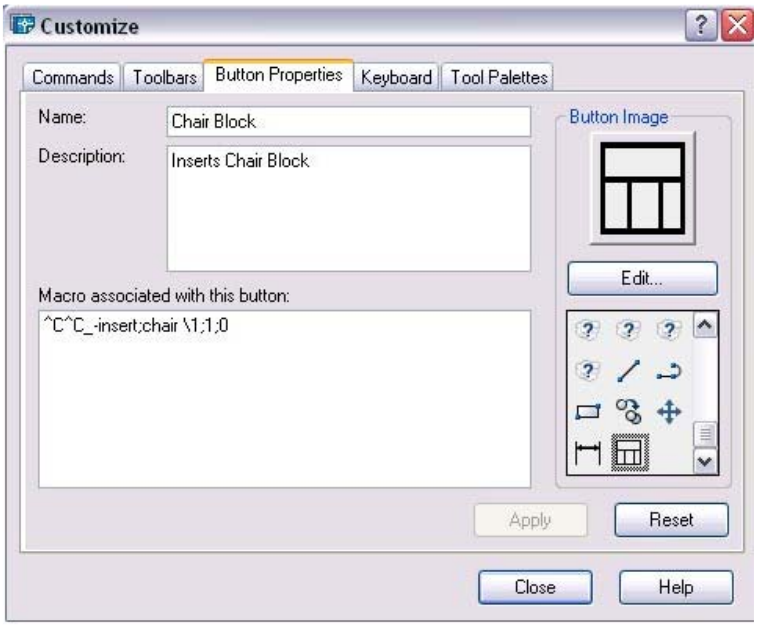
TIPS:

- Use ^C^C to cancel any previous AutoCAD command
- apostrophe (‘) will issue a transparent command
- Use a semicolon (;) to separate a series of commands
- a dash (-) will issue the command without a dialog box.

Characters Used in Macros 36.7

Pausing for User Input Example

To accept input from the keyboard or the pointing device in the middle of a menu macro, place a backslash (\) at the point where you want input.

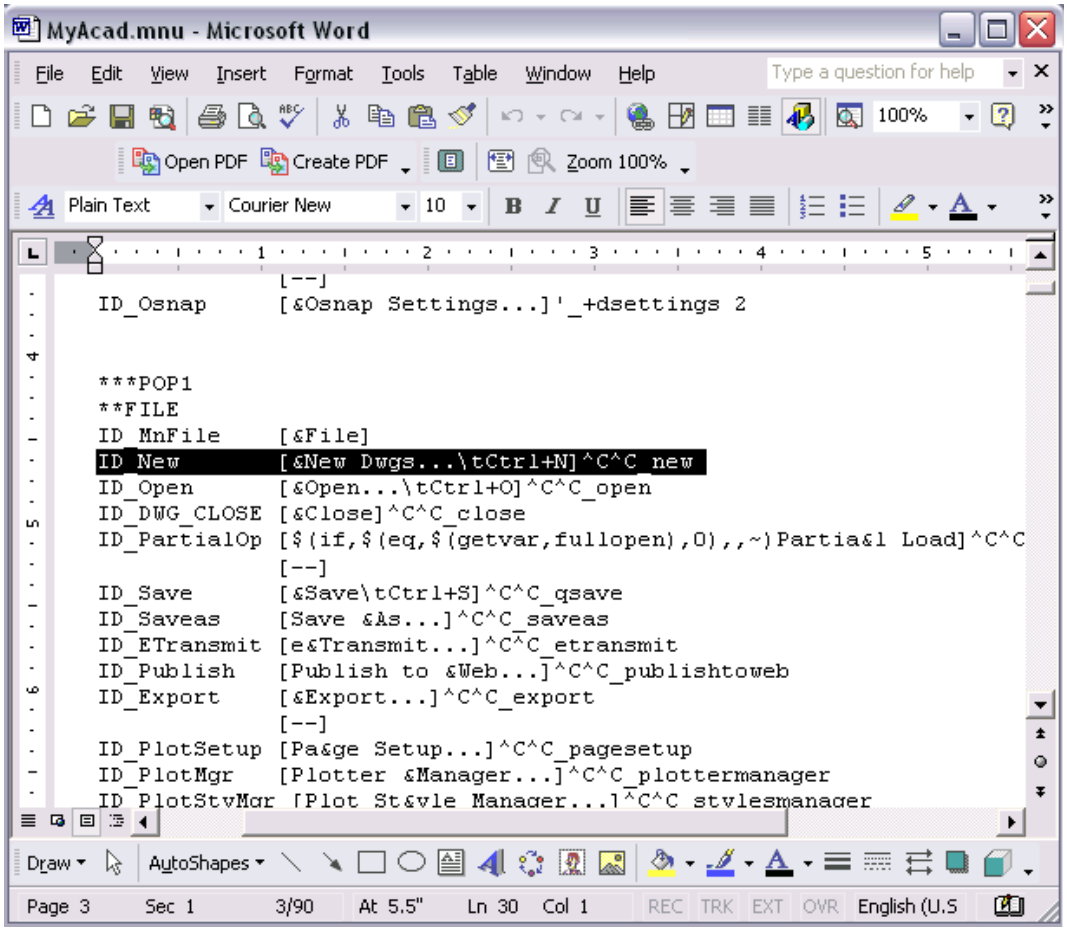


;	Issues ENTER
^M	Issues ENTER
^I	Issues TAB
SPACEBAR	Enters a space; blank space between command sequences in a menu item is equivalent to pressing the SPACEBAR
\	Pauses for user input (cannot be used in the
Accel	erators section)
_	Translates AutoCAD commands and keywords
that	follow
+	Continues menu macro to the next line (if last character)

=*	Displays the current top level image, pull-down, or shortcut menu
*^C^C	Prefix for a repeating item
$ or expression	Special character code that loads a menu section introduces a conditional DIESEL macro ($M=)
^B	Toggles Snap on or off (CTRL+B)
^C	Cancels command (ESC)
^D the	Toggles Coords on or off (CTRL+D) SPACEBAR at end of a menu item
^E	Sets the next isometric plane (CTRL+E)
^G	Toggles Grid on or off (CTRL+G)
^H	Issues backspace
^O	Toggles Ortho on or off (CTRL+O)
^P	Toggles MENUECHO on or off
^Q	Echoes all prompts, status listings, and input to the printer (CTRL+Q)
^T	Toggles tablet on or off (CTRL+T)
^V	Changes current viewport (CTRL+V)
^Z of	Null character that suppresses the automatic addition

Editing Menus in Word 36.8

1. **Open** a menu (.mnu) file in a text editor.
2. **Change** the desired menu.
3. **Type** MENU at the AutoCAD command prompt to compile and use the menu in AutoCAD/



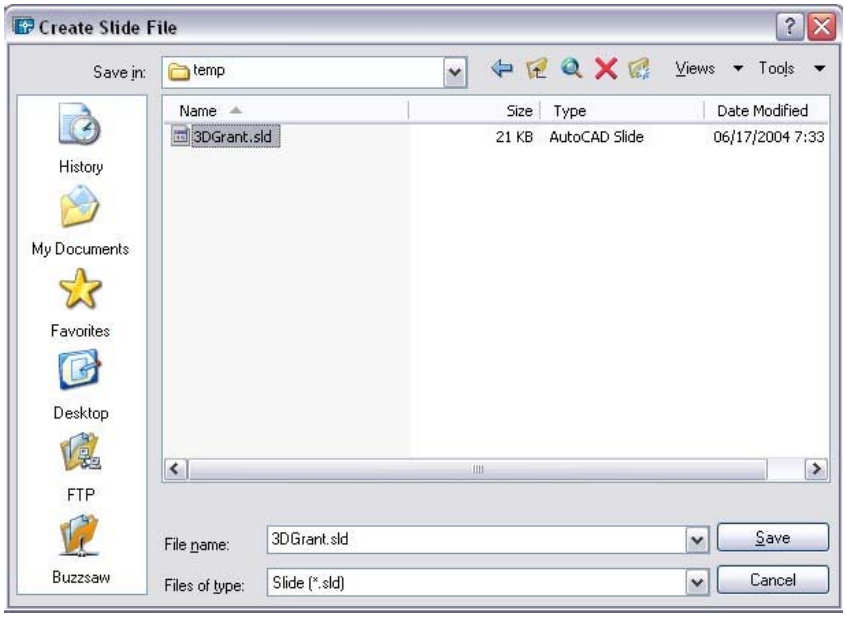
Chapter 38
Slide Shows

Creating Slides 38.1

MSlide Command

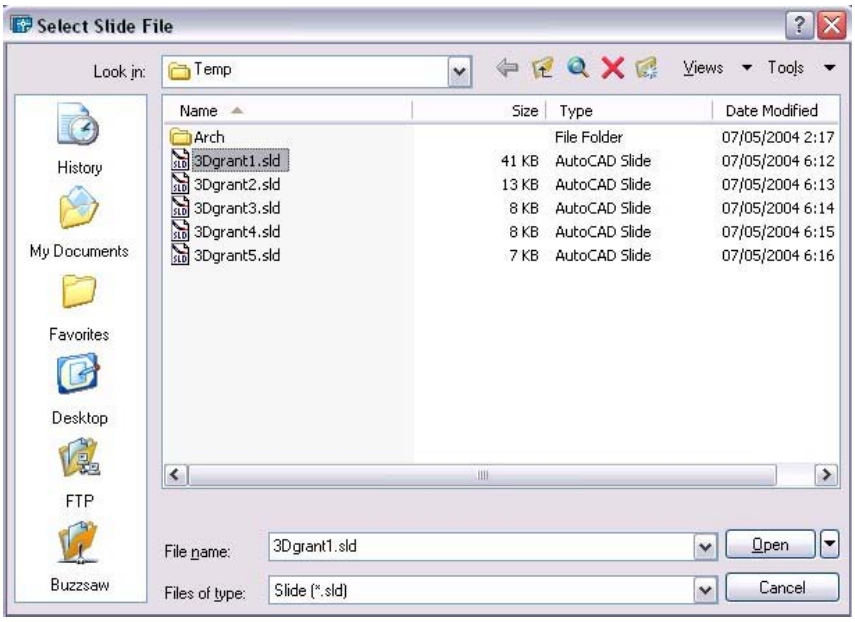
1. **Type** MSLIDE at the command prompt.
   Command: **MSLIDE**

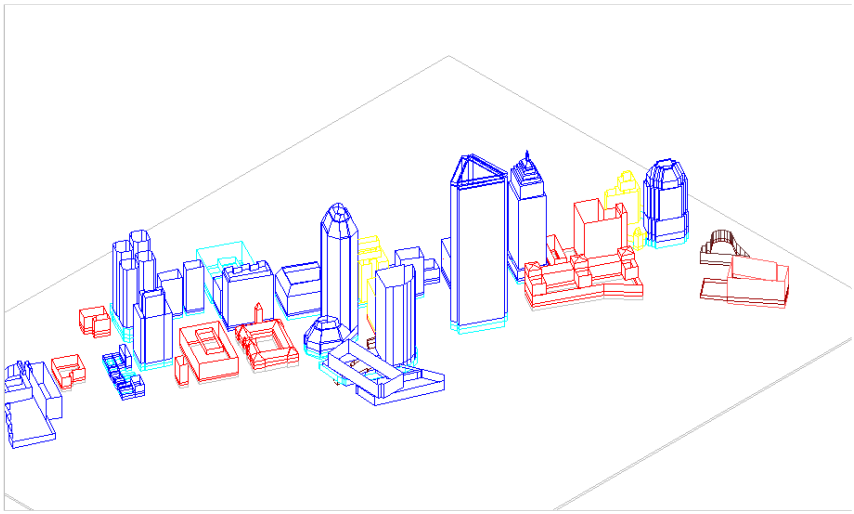
2. **Type** the name of the slide file (and location).



Viewing Slides 38.2

1. **Type** VSLIDEat the command prompt.
   Command: **VSLIDE**
2. **Pick** the name of the slide file (and location).





Slideshows 38.3

Scripts

In a Word Processor, create a series of commands to execute in AutoCAD.

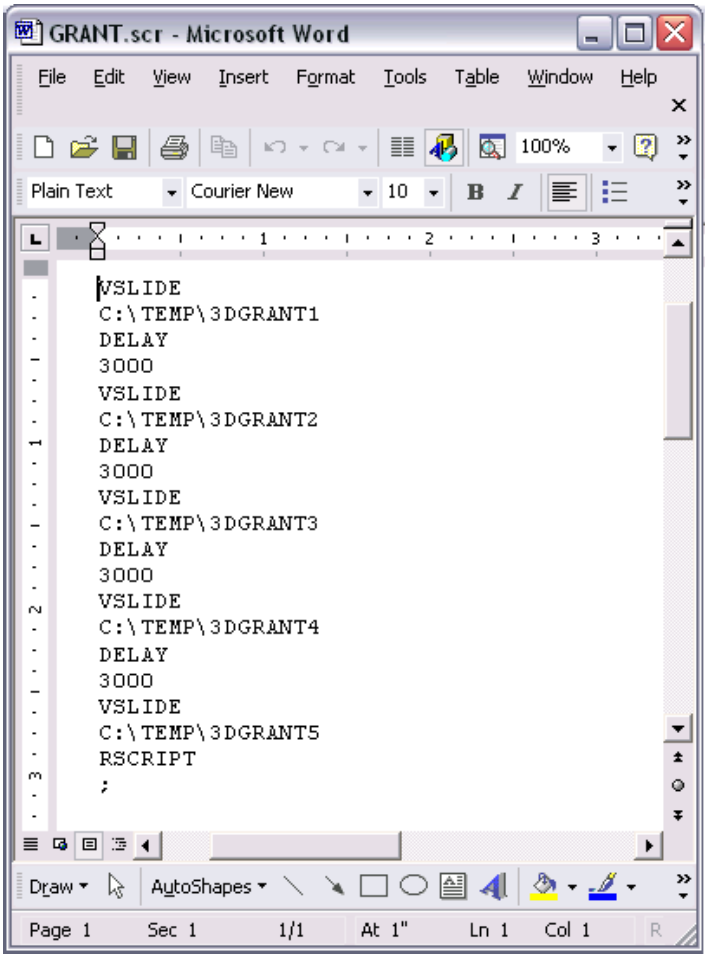
Save the script file with an extension called **.SCR.**

Pausing a Slide

1. **Type** DELAY at the command prompt.
   Command: **DELAY**

   Enter delay time (in milliseconds): **3000**

   NOTE: 3000 milliseconds is 3 seconds



Running a Script in AutoCAD

1. **Type** SCRIPT at the command prompt.
   Command: **SCRIPT**
2. **Pick** the script name to run.

Repeating a Script

1. **Type** RSCRIPT at the command prompt.
   Command: **RSCRIPT**

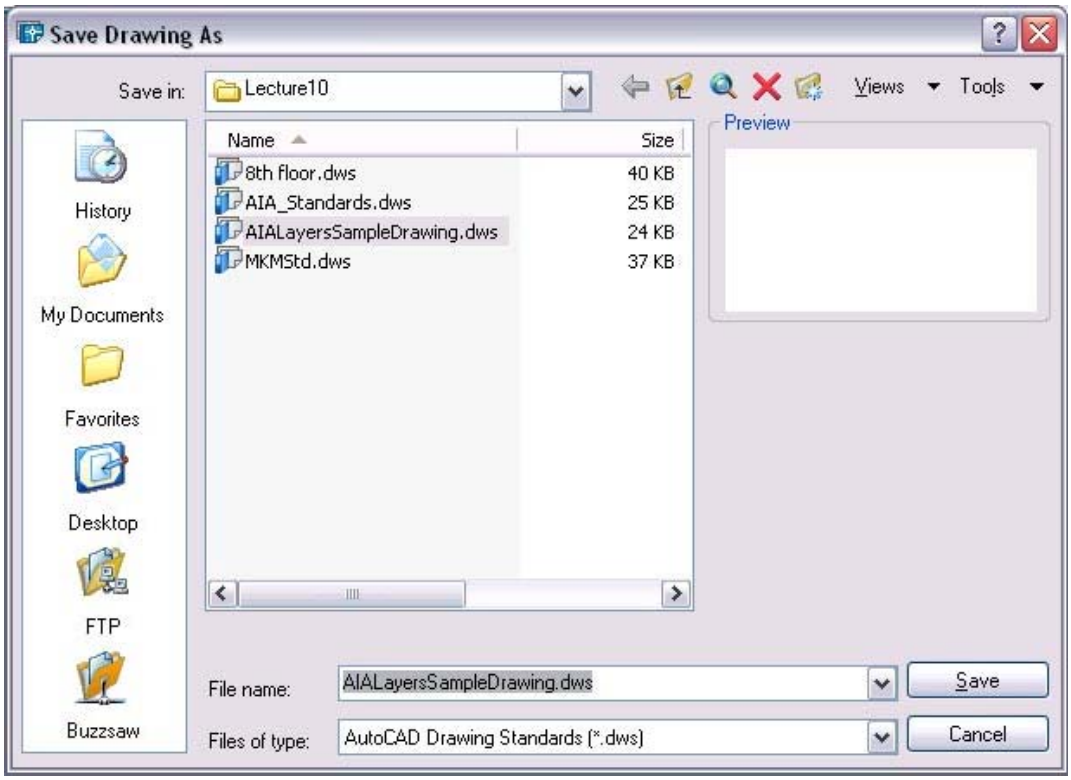
   This will repeat the script command lines continuously.

Chapter 39
CAD Standards

39.1 Drawing Standards (.DWS) Files

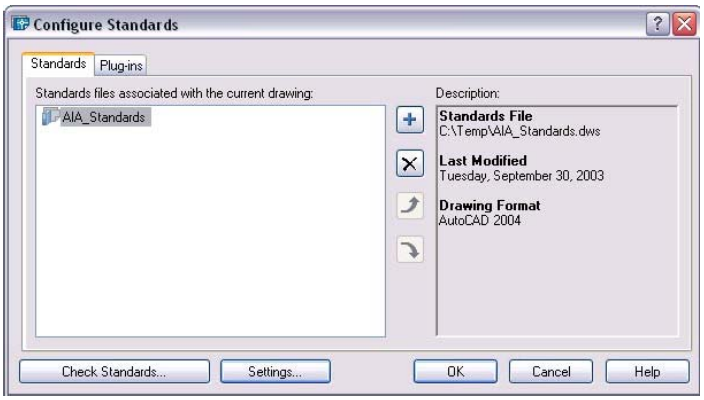
Standards define a set of common properties for named objects such as layers and text styles. You or your CAD manager can create, apply, and audit standards in AutoCAD drawings to enforce consistency. Because standards make it easier for others to interpret drawings, standards are particularly useful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

1. **Open** a drawing with standards defined (i.e. AIALayersSampleDrawing.dwg)
2. **Type** SAVEAS at the command prompt.
   Command: **saveas**
3. **Choose** .DWS as the file type to save.
4. **Save** the drawing standard file.

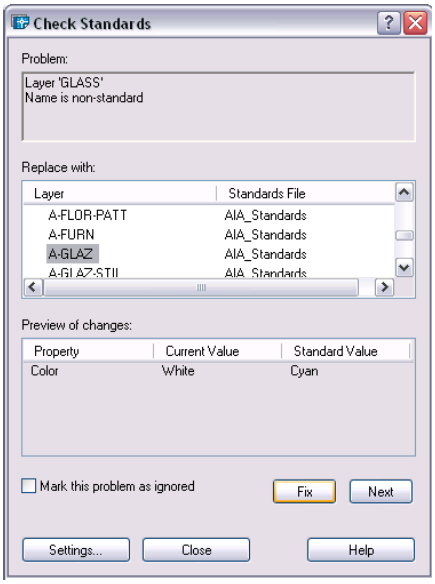


39.2 CAD Standards Manager

1. **Choose** Tools, CAD Standards, Configure... or
2. **Type** STANDARDS at the command prompt.
   Command: **standards**
3. **Choose** Add standards button to add a standards file (aialayer.dws)

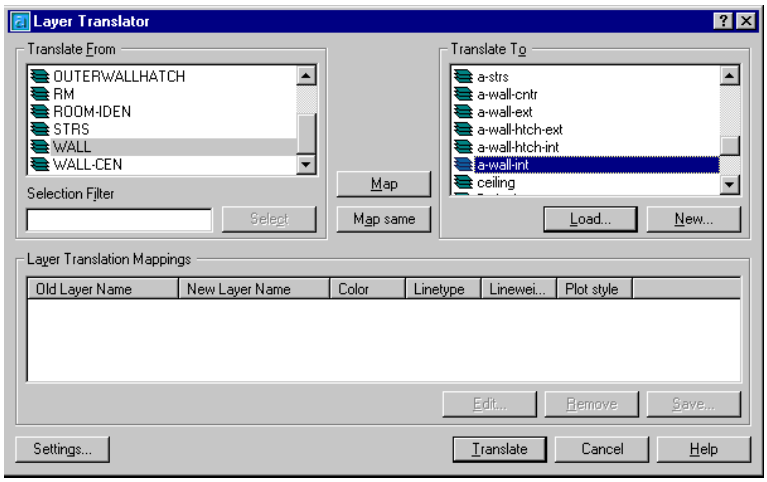


4. **Open** a drawing to check its standards (i.e. HBH-G.dwg)
5. **Load** the CAD standards AIALayer.DWS file.
6. **Choose** Checkstandads... or
7. **Type** CHECKSTANDARDS at the command prompt.
   Command: checkstandards
8. **Choose** the fix button to make changes to the existing drawing.



39.3 Layer Translator

1. **Type** LAYTRANS at the command prompt.
   Command: **laytrans**
   or
2. **Choose** Tools, CAD Standards, Layer Translator.
3. **Choose** Load...to load standards from a .DWS or .DWG file.
4. **Match** the layers in the current drawing to the layers in the .DWG or .DWS file.

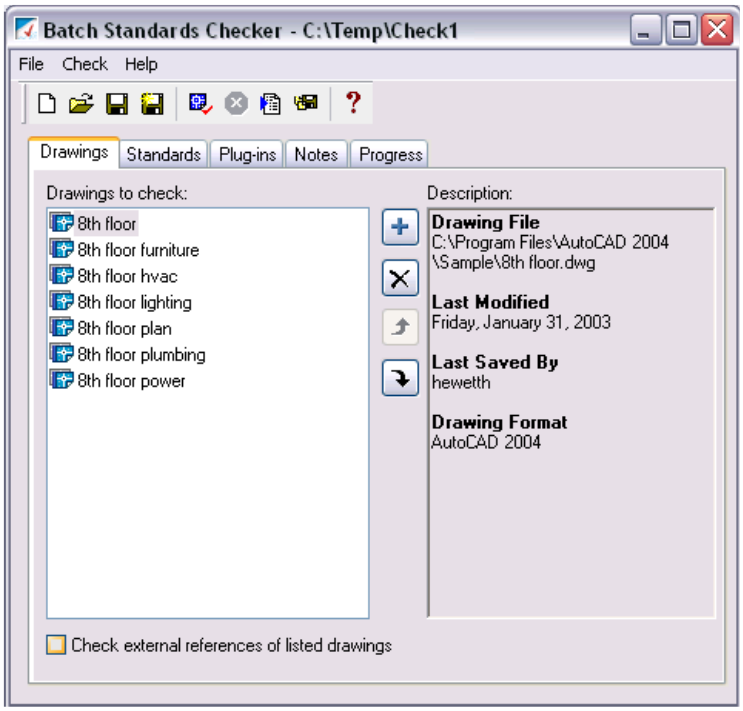


39.4 Batch Standards Checker

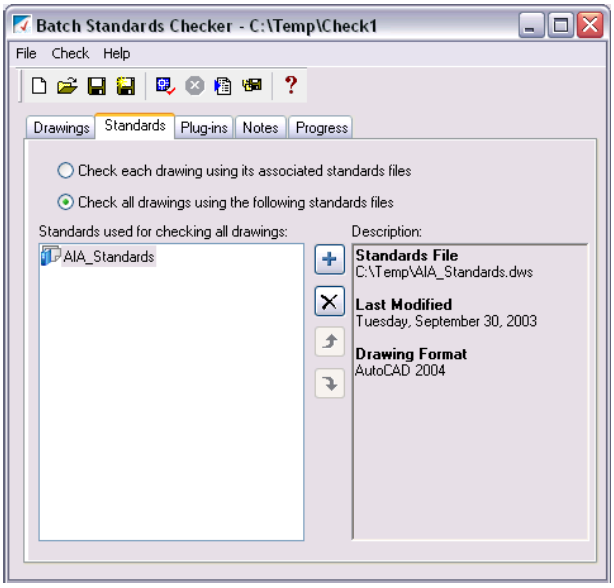
Performs batch checking on multiple drawings outside of AutoCAD's drawing editor.

1. **Choose** Start, Programs, AutoCAD 2000x, Batch Standards Checker.

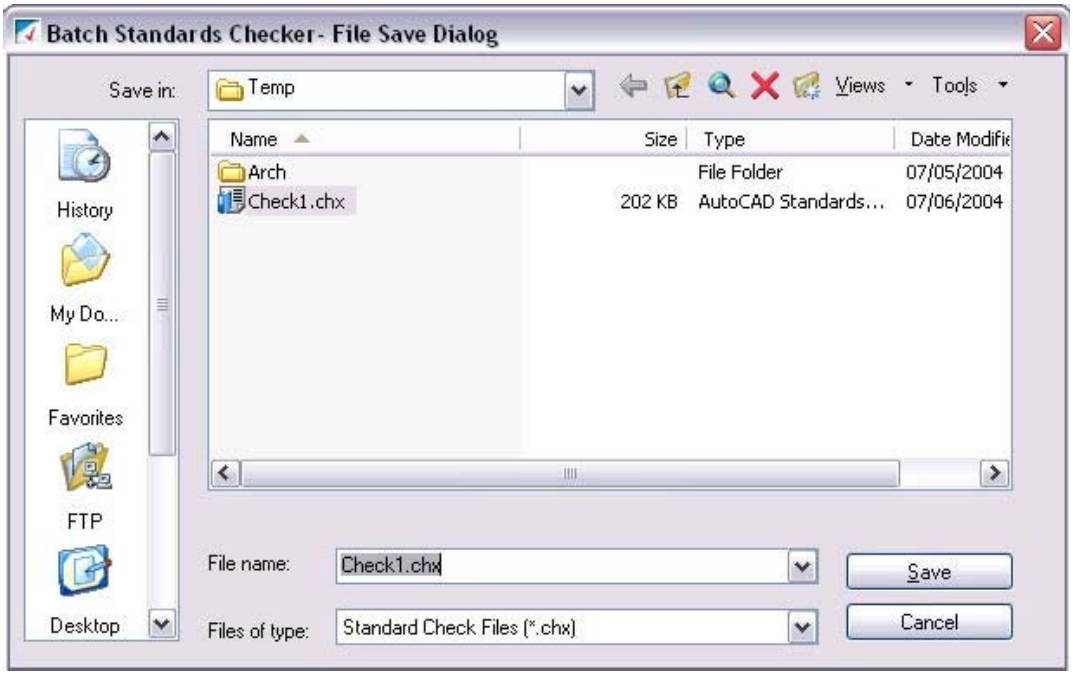
2. **Load** multiple drawings to check.

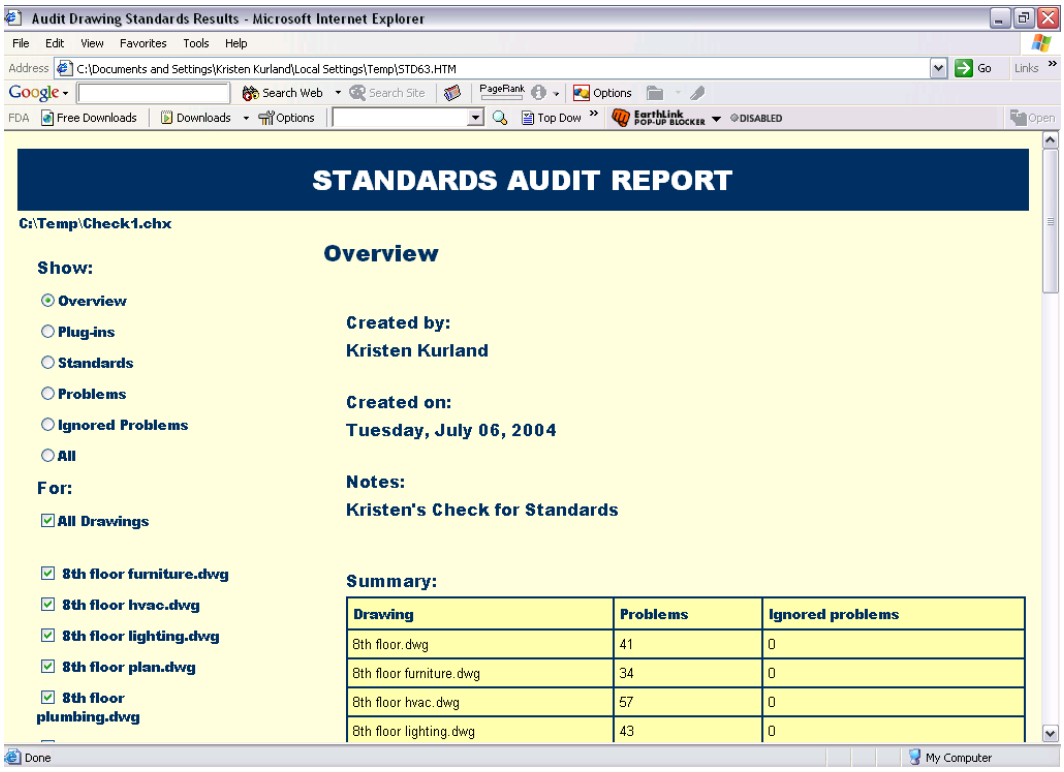


3. **Compare** to an existing standards file (i.e. aialayers.dws).



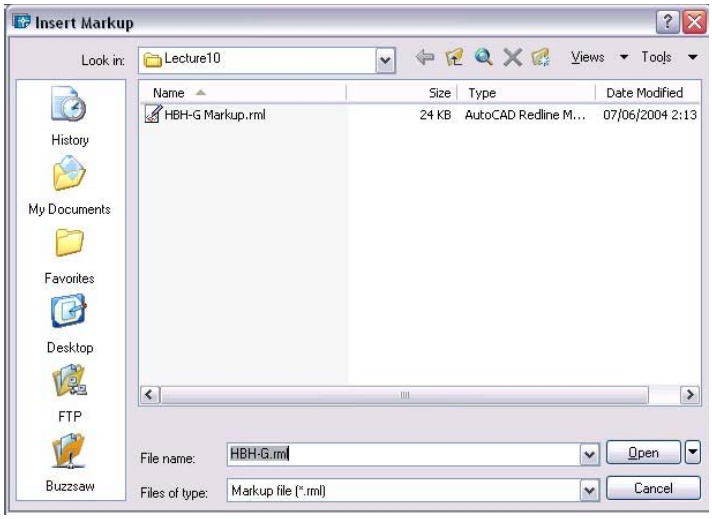
4. **Save** the standards audit as a file (.chx)





39.5 Redline Markup Language

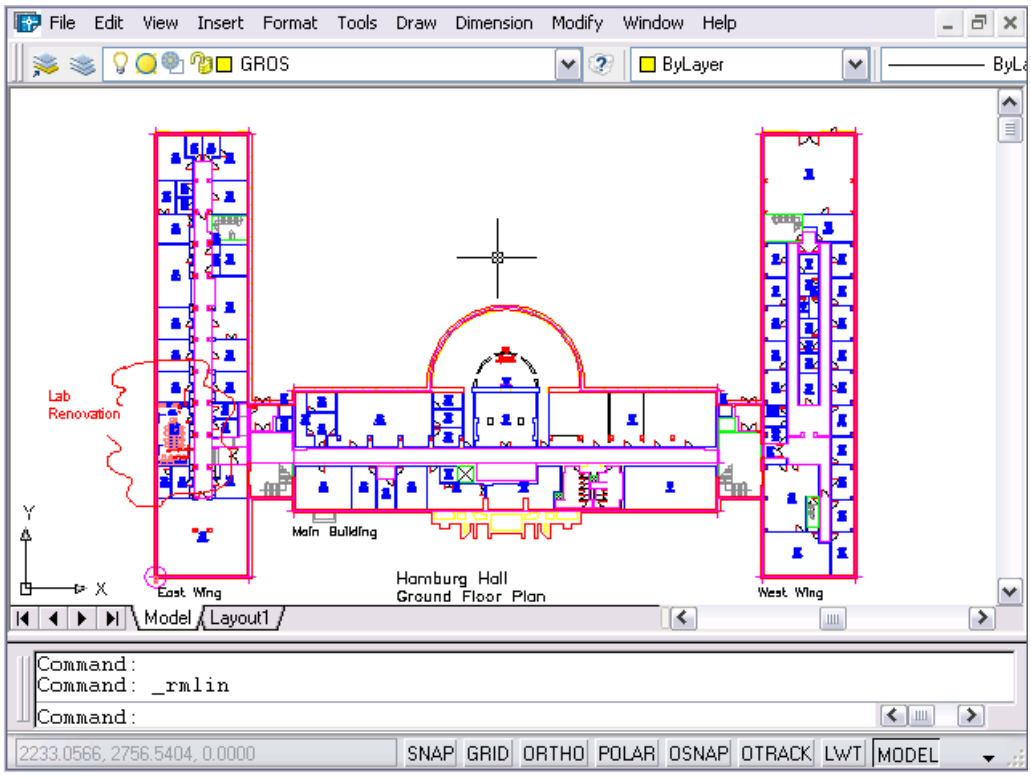
1. **Choose** Insert, Markup...
2. **Pick** a markup language file to insert.



TIPS:

Markups range from hyperlinks to simple boxes and circles. Created in programs such as VoloView

A new "MARKUP" layer is created in the current drawing.



39.6 Revision Cloud

1. **Choose** Draw, Revcloud

or

2. **Type** **REVCLOUD** at the command prompt.

Command: **revcloud**

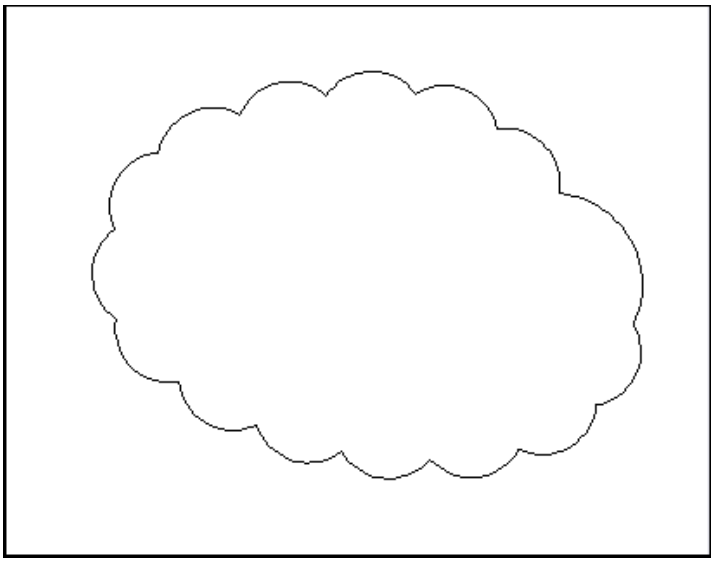
Minimum arc length: 0.5000

Maximum arc length: 0.5000

Specify start point or [Arc length/Object] <Object>: Guide crosshairs along cloud path...
Revision cloud finished.

or

3. **Click** the Revcloud icon from the draw menu.



39.8 Wipeout

Creates a polygonal area that masks underlying objects with the current background color. This area is bounded by the wipeout frame. You can turn on the wipeout frame for editing and turn it off for plotting.

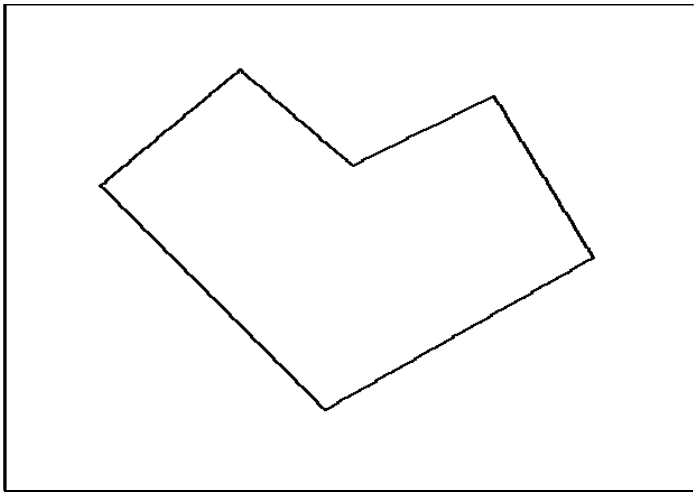
1. **Choose** Draw, Wipeout

or

2. **Type** WIPEOUT at the command prompt.

Command: **_wipeout**

Specify first point or [Frames/Polyline] <Polyline>:
Select a closed polyline:



Chapter 40
Isometrics

Isometric Cursor 40.1

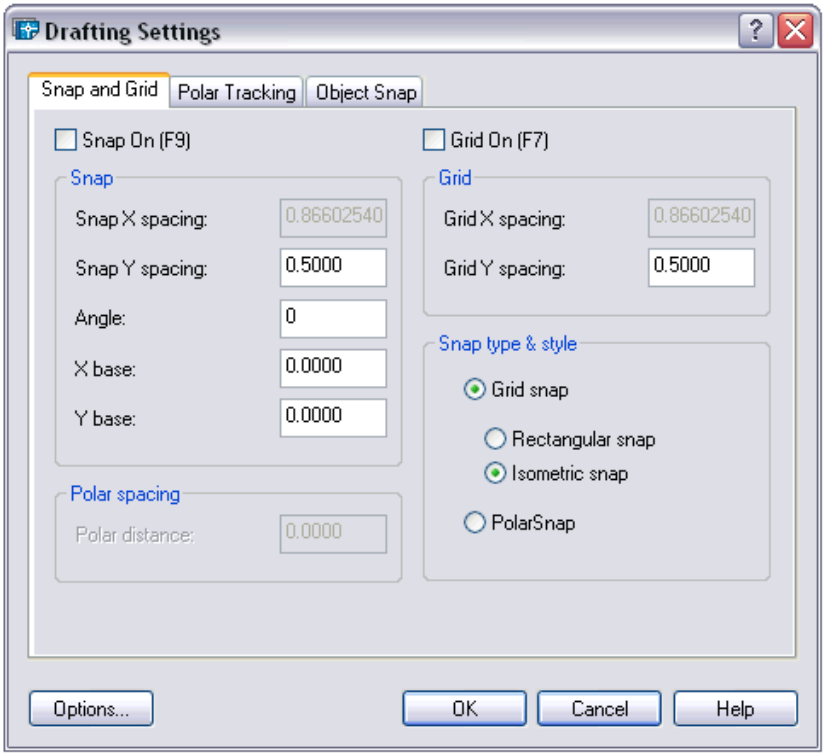
1. **Choose** Tools, Drawing Aids...

   or

2. **Type** DDRMODES at the command prompt.

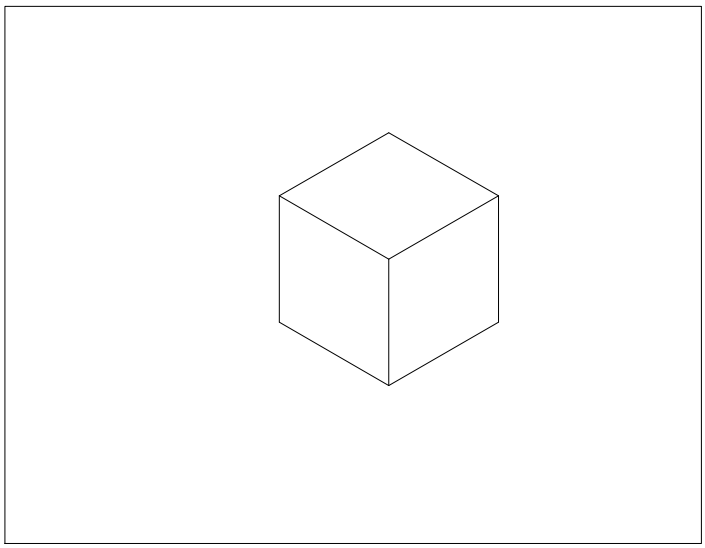
   Command: **DDRMODES**

3. **Toggle** Isometric Snap/Grid to ON.



Isoplane Toggle 40.2

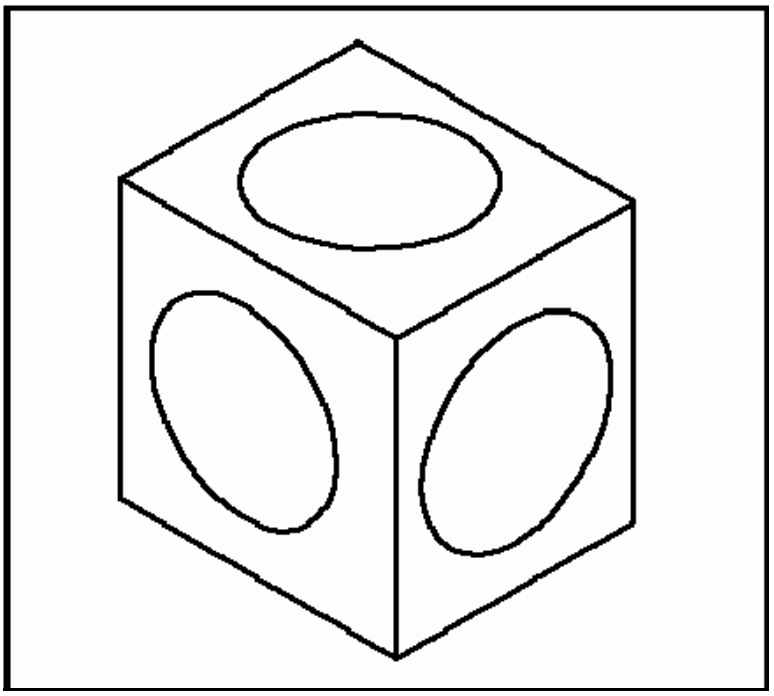
1. **Press** Function Key F5 to toggle
   <Isoplane Top>
   <Isoplane Left>
   <Isoplane Right>
   or
2. **Press** **CTRL + E** to toggle isoplanes.



Isometric Circles 40.3

1. **Type** ELLIPSE at the command prompt.
   Command: **ELLIPSE** or **EL**
   Arc/Center/Isocircle/<Axis endpoint 1>: I Center of circle: <Isoplane Top>

   <Circle radius>/Diameter:



Isometric Text 40.4

1. **Type** STYLE at the command prompt
   Command:**STYLE**
2. **Type** RISO as a style name
   Text style name (or ?) **RISO**
3. **Pick** A font file
   Font file: **Romans.shx**
4. **Type** Zero(0) for the text height
   Height <0>**0**
5. **Type** .85 for the character width factor
   Width factor: **.85**
6. **Type** 30 degrees for an obliquing angle
   Obliquing Angle: **30**
7. **Type** NO to Backwards, Upside Down, and Vertical
8. **Type** DTEXT at the command prompt
   Command:**DTEXT**
9. **Pick** A start point
   Justify/Style/<Start point>: **pick**
10. **Type** 30 for the rotation angle
    Rotation angle:30
11. **Type** A string of text
    Text:(text string)
12. **Press** ENTER to end the text command

.

Style	Width Factor	Oblique Angle	Rotation Angle
Left ISO	.85	-30	-30
Right ISO	.85	30	30
Top ISO	.85	30	-30

Isometric Dimensions 40.5

1. **Type** DIMALIGNED to place an aligned dimension in isometrics
2. **Type** DIMEDIT oblique the angle of the dimension line and rotate the text.

   Oblique Angle = **-30**

   Rotated Text = **30**