
AutoCAD® 2013

2D Tutorials

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Chapter 1

Introduction

1.1 Launching AutoCAD

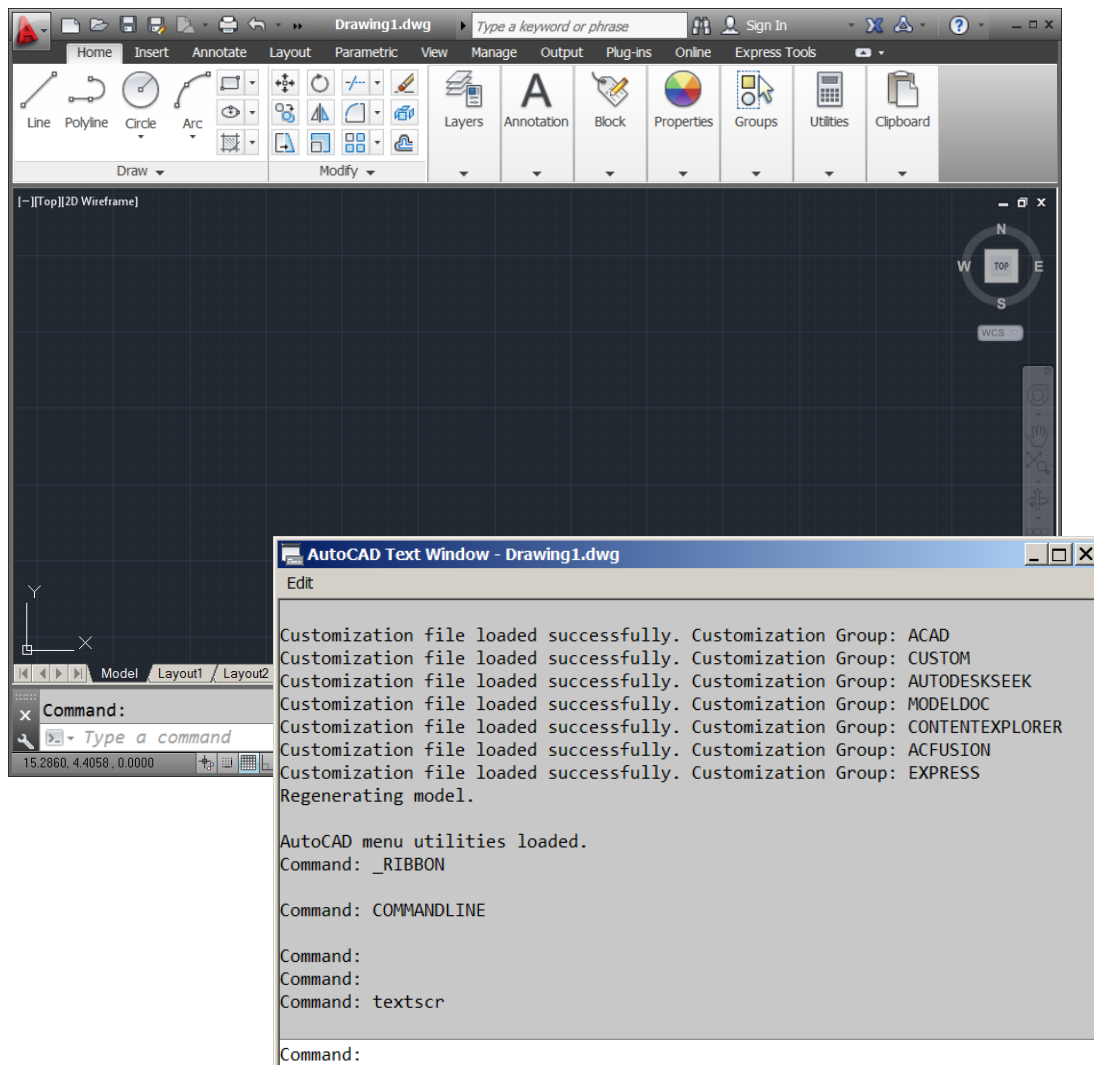
1. **Choose** Start from the Windows program manager.
2. **Choose** All Programs > **Autodesk** > **AutoCAD 2013**.
3. **Click** the AutoCAD 2013 for Windows icon.



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 or
2. **Type** **TEXTSCR** at the command prompt.



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.

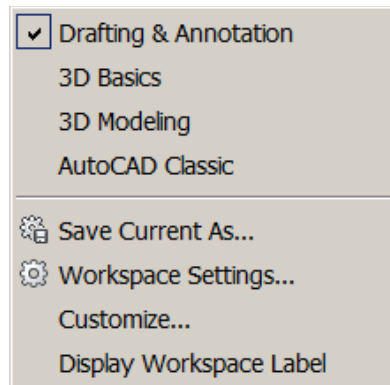
1.3 Workspaces

You can switch between the workspaces from the menu browser

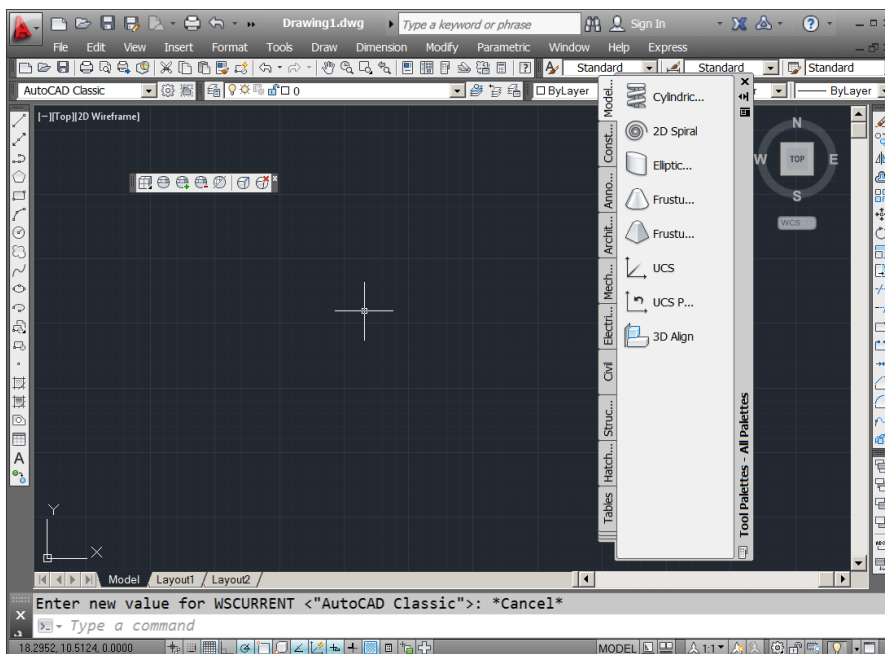
1. **Click** the Workspace switching icon in the lower right corner of the screen.



2. **Click** on one of the following workspace options.



AutoCAD classic workspace



1.4 Status Bar and Command Prompt

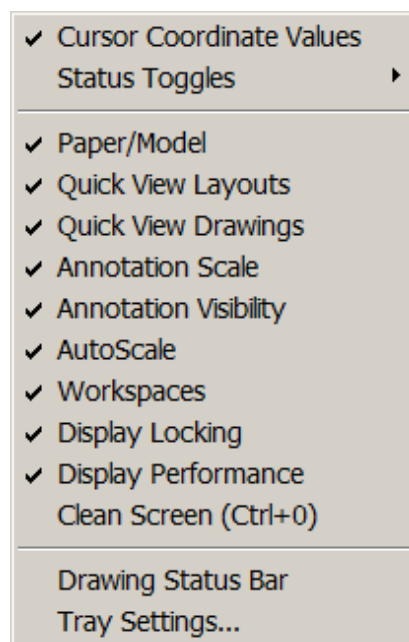
The Status Bar is the area below the command line that shows messages as well as coordinates, modes, and the current time.

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



TIP:

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



1.5 AutoCAD Commands

Typing Commands

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 or the command's alias at the command prompt.

 Command : **LINE**

 Command: **L**
2. **Press** **ENTER** on the keyboard.
3. **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.

Canceling a Command

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

TIP:

- Pressing ESC twice clears nested commands.

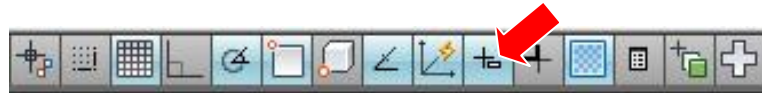
1.6 Dynamic Input

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.

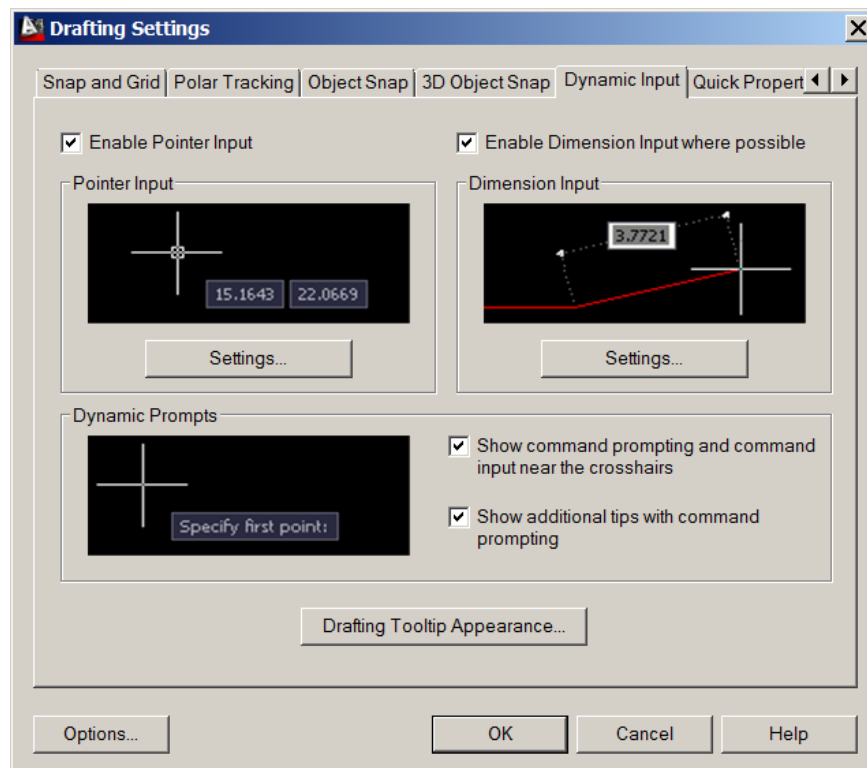
Turning Dynamic Input ON/OFF

1. Click the **Dynamic Input** icon on the status bar or press **F12**.



TIP:

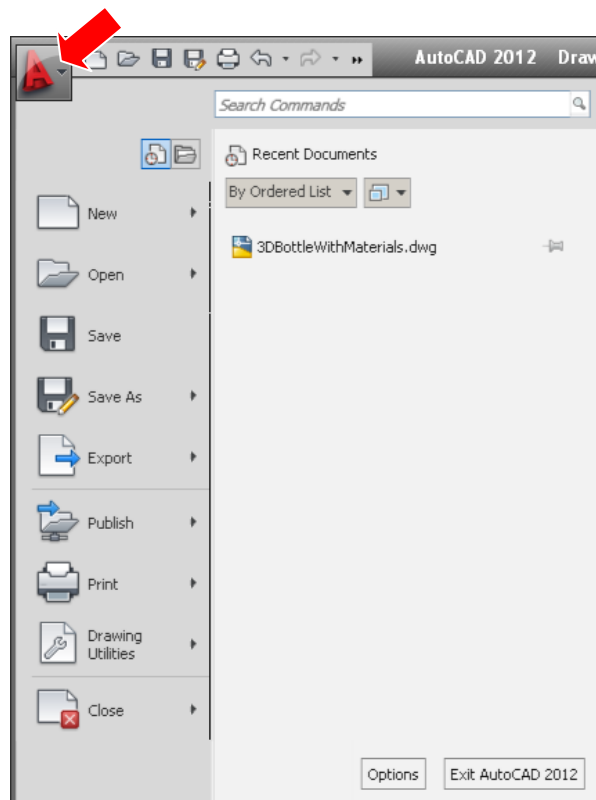
- Right-click the Dynamic Input icon and click Settings to control what is displayed by each component when Dynamic Input is on.



1.8 Menus, Ribbons, and Toolbars

Menu Browser

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



Quick Access Toolbar

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

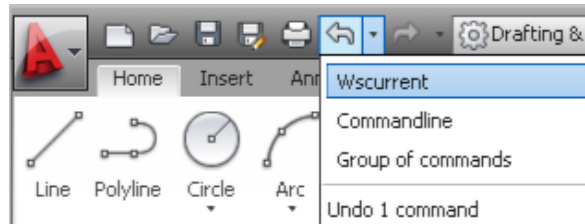


TIPS:

- 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.
- The menu bar provides quick access to most AutoCAD commands. It is a good idea to show these menus, especially when first learning AutoCAD. Many commands in this training manual are accessed by these menus.

Sub-menus

1. **Click** the dropdown button beside the quick access commands.



Pulldown Menus



Info Center

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

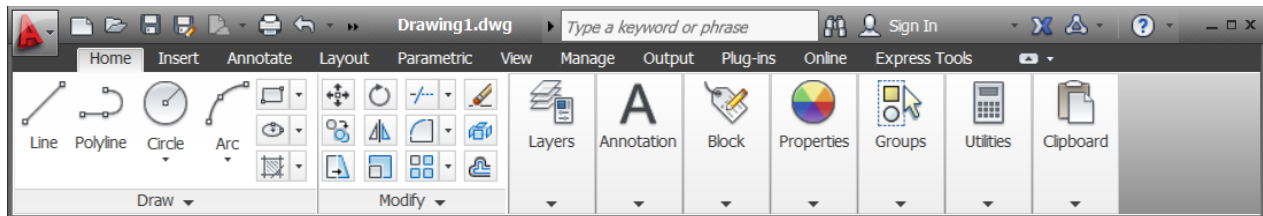


Ribbons

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.



AutoCAD Classic Toolbars

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

To Float a Toolbar:

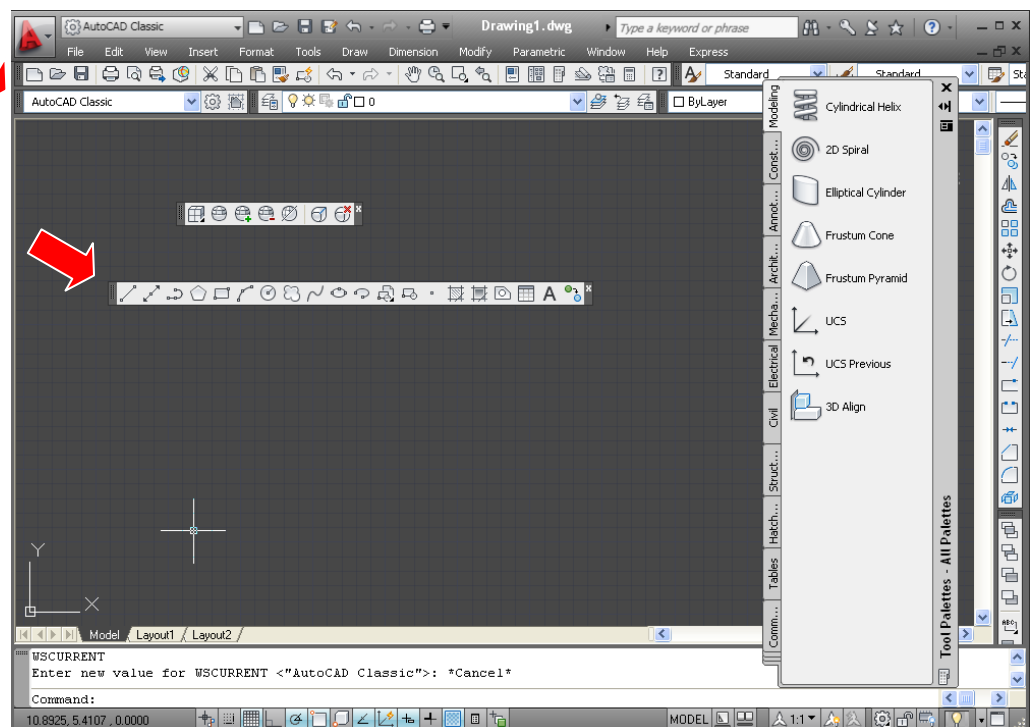
1. **Choose** the gray border surrounding each tool.
2. **Drag** the toolbar to any area on the screen.

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.

Docked toolbars

Floating toolbars



TIP:

- Holding the CTRL key while dragging will prevent docki

Loading Toolbars

1. Right-click

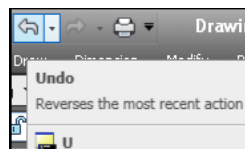
an icon in any toolbar.
This will show a list of all available toolbars.



Help Tooltips

1. Move

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

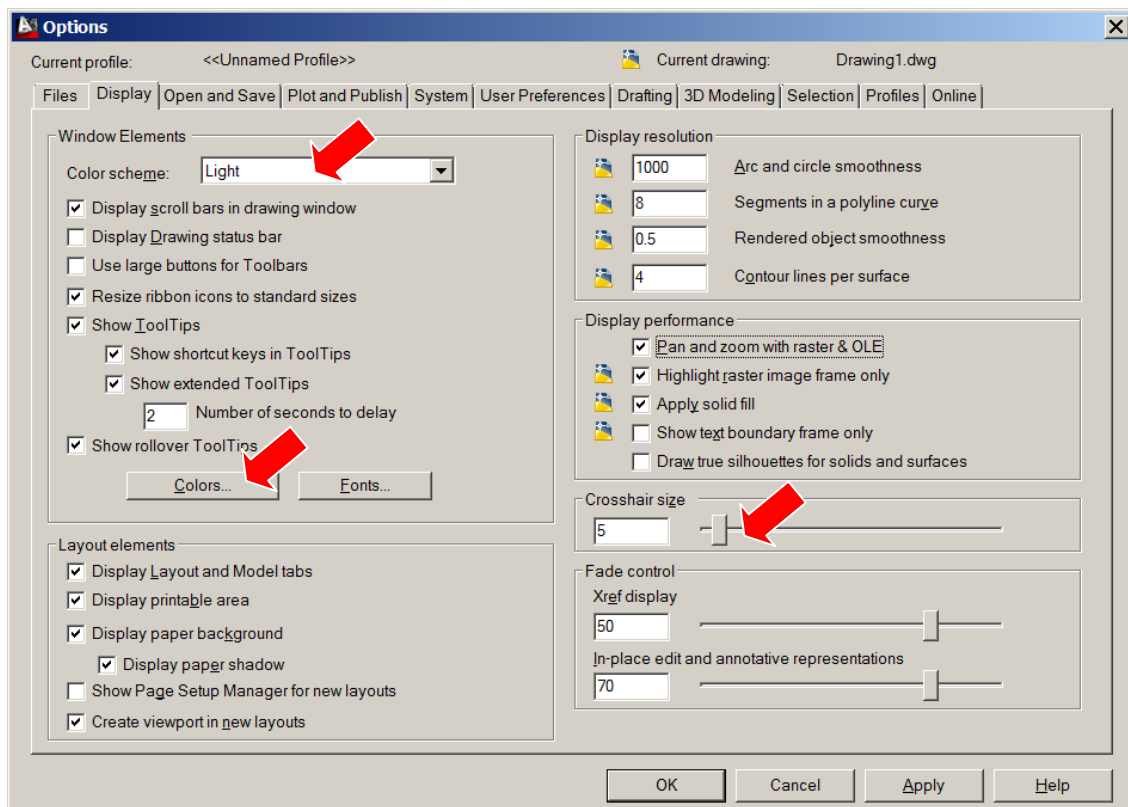


1.9 Cursor and Colors

The Options dialog box 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%.

This dialog box also sets the colors for the application interface.

1. **Type** **OPTIONS** at the command prompt.
 Command: **OPTIONS**
2. **Click** the Display tab.
3. **Click** **Light as the color scheme.**
4. **Drag** the slider bar in under Crosshair size to set the cursor size.
5. **Click** the Colors button under Window Elements to change software interface colors.



1.10 Pointing Device

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).

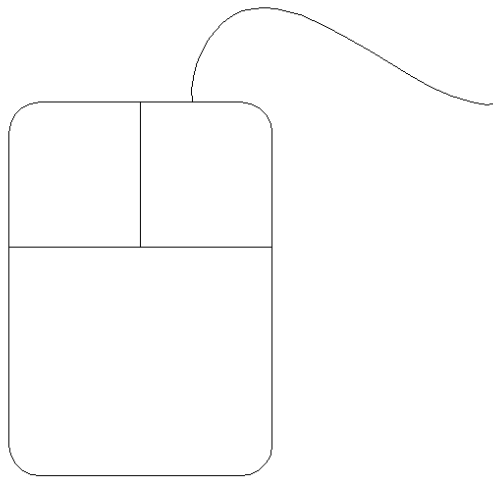
Right Mouse Button

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

1. **Click** the right mouse button.

Mouse Wheel

Some point devices such as a mouse have a middle wheel that can be used to zoom in and out of the drawing window.



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

Undo

Reverses the last action.

1. **Click** the Undo icon.



or

Press **CTRL + Z.**

or

Type U at the command prompt to undo the last command.

Command: **U**

Redo

Reverses the effects of the UNDO or U command.

1. **Click** the Redo icon.



or

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 Keyboard Shortcuts

Active	Hyperlink...	CTRL+K
Active	Toggles Infer Constraints	CTRL+SHIFT+I
Active	Toggle HIDEPALETTES	CTRL+SHIFT+H
Active	Toggle Coords	CTRL+I
Active	Toggle Dynamic UCS	CTRL+D
Active	Toggle Ortho	CTRL+L
Active	Toggles Quick Properties	CTRL+SHIFT+P
Active	CTRL+R	CTRL+R
Active	Select All	CTRL+A
Active	Copy Clip	CTRL+C
Active	New...	CTRL+N
Active	Open...	CTRL+O
Active	Plot...	CTRL+P
Active	Save	CTRL+S
Active	Save As...	CTRL+SHIFT+S
Active	Exit	CTRL+Q
Active	Paste	CTRL+V
Active	Copy with Base Point	CTRL+SHIFT+C
Active	Paste as Block	CTRL+SHIFT+V
Active	Cut	CTRL+X
Active	Redo	CTRL+Y
Active	Undo	CTRL+Z
Active	Clean Screen	CTRL+0
Active	Properties	CTRL+1
Active	DesignCenter	CTRL+2
Active	Tool Palettes	CTRL+3
Active	Sheet Set Manager	CTRL+4
Active	dbConnect	CTRL+6
Active	Markup Set Manager	CTRL+7
Active	QuickCalc	CTRL+8
Active	Command Line	CTRL+9

Active	VBA, Macros...	ALT+F8
Active	VBA, Visual Basic Editor	ALT+F11
Active	CTRL+H	CTRL+H
Active	Subobject Selection, No Filter	SHIFT+F1
Active	Subobject Selection, Vertex	SHIFT+F2
Active	Subobject Selection, Edge	SHIFT+F3
Active	Subobject Selection, Face	SHIFT+F4
Active	Solid History	SHIFT+F5
Active	Toggles Orthogonal Mode	SHIFT
Active	Toggles Object Snap Mode	SHIFT+A
Active	Toggles Polar Mode	SHIFT+X
Active	Enables Object Snap Enforcement	SHIFT+S
Active	Toggles Object Snap Tracking Mode	SHIFT+Q
Active	Object Snap Override : Endpoint	SHIFT+E
Active	Object Snap Override : Midpoint	SHIFT+V
Active	Object Snap Override : Center	SHIFT+C
Active	Disable All Snapping and Tracking	SHIFT+D
Active	Toggles Object Snap Mode	SHIFT+'
Active	Toggles Polar Mode	SHIFT+.
Active	Enables Object Snap Enforcement	SHIFT+;
Active	Toggles Object Snap Tracking Mode	SHIFT+]
Active	Object Snap Override : Endpoint	SHIFT+P
Active	Object Snap Override : Midpoint	SHIFT+M
Active	Object Snap Override : Center	SHIFT+,
Active	Disable All Snapping and Tracking	SHIFT+L
Active	Toggles Object Snap Mode	F3
Active	Toggles Orthogonal Mode	F8
Active	Toggles Snap Mode	F9
Active	Toggles Polar Mode	F10
Active	Toggles Object Snap Tracking Mode	F11
Active	Suppress Dynamic Input	F12
Active	Toggles Dynamic UCS Mode	F6
Active	Toggles Dynamic UCS Mode	SHIFT+Z
Active	Toggles Dynamic UCS Mode	SHIFT+/-

1.13 On-Line Help

1. **Choose** Help, AutoCAD Help.

or

Click the Help icon.



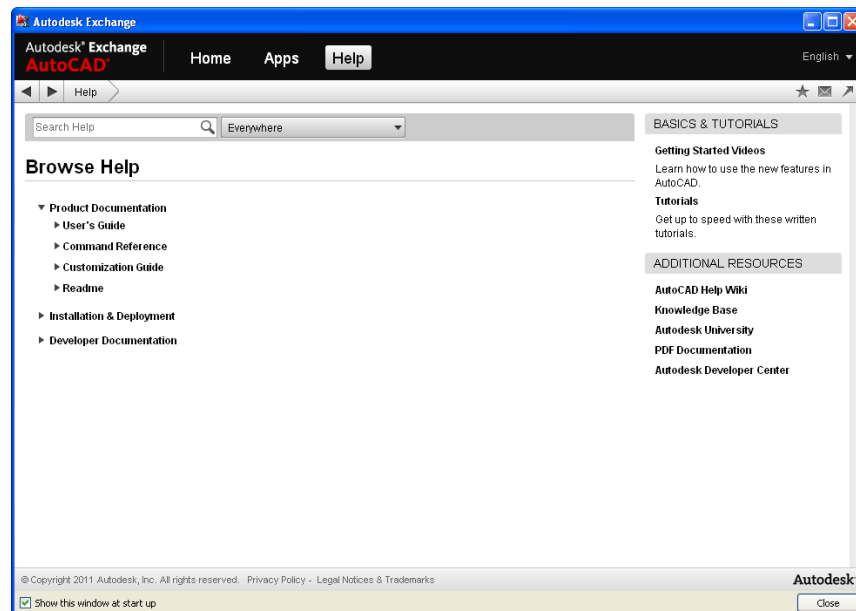
or

Type HELP at the command prompt

Command: **HELP**

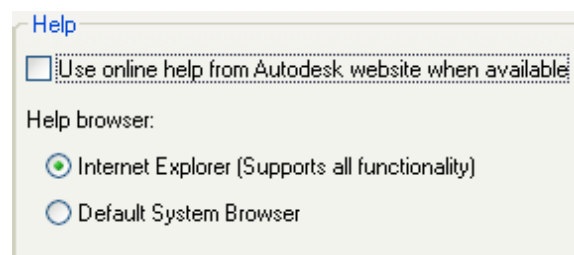
or

Press Function Key F1



TIP:


- If you do not have an Internet connection you can change the help settings using the OPTIONS command. Click the System TAB and turn off “Use online help from Autodesk website when available.”

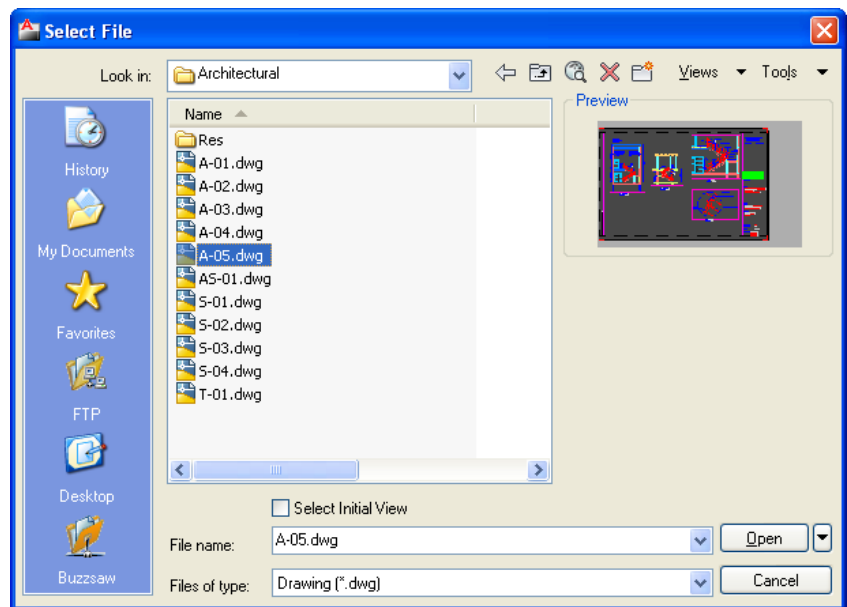


Chapter 2

Introduction to Commands

2.1 Open Existing Drawings

1. **Click** the **A** icon and **OPEN**
or
Press **CTRL + O**.
or
Click the **OPEN** icon. 
or
Type **OPEN** at the command prompt.
Command: **OPEN**
2. **Press** **ENTER** on the keyboard.
3. **Double Click** the desired directory to find the drawing to open.
4. **Click** the drawing name to open.
5. **Click** The OK button.




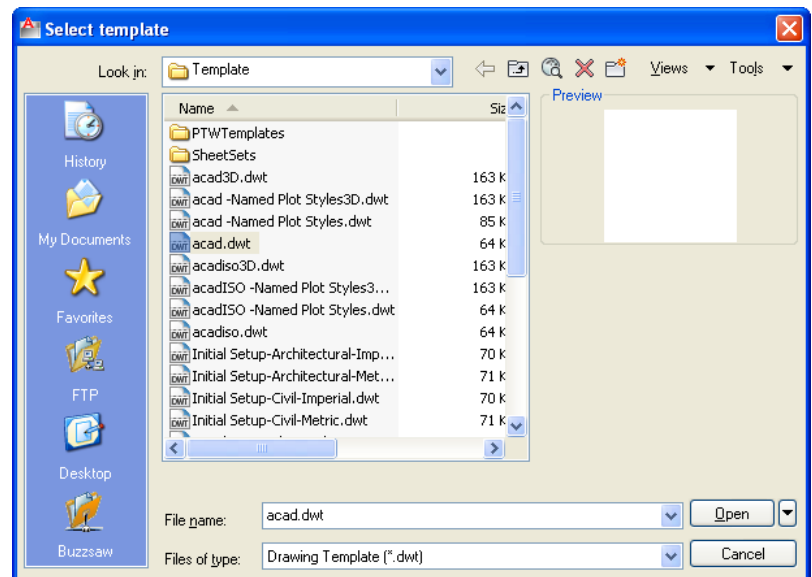
TIP:

- 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

Creates a new drawing file.

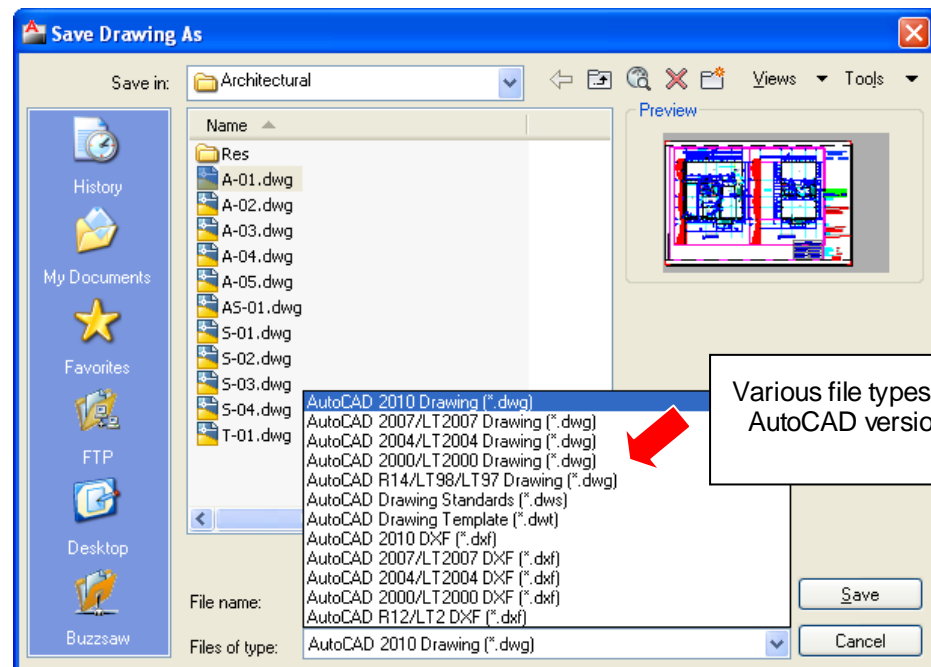
1. **Click** the **A** icon and New.
or
Press CTRL + N
or
Click the New icon. 
or
Type NEW at the Command prompt.
Command: **NEW**
2. **Choose** One of the options for creating a new drawing.
3. **Click** The OK button.
4. **Save** the drawing as another name.



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



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

Click the Save icon.

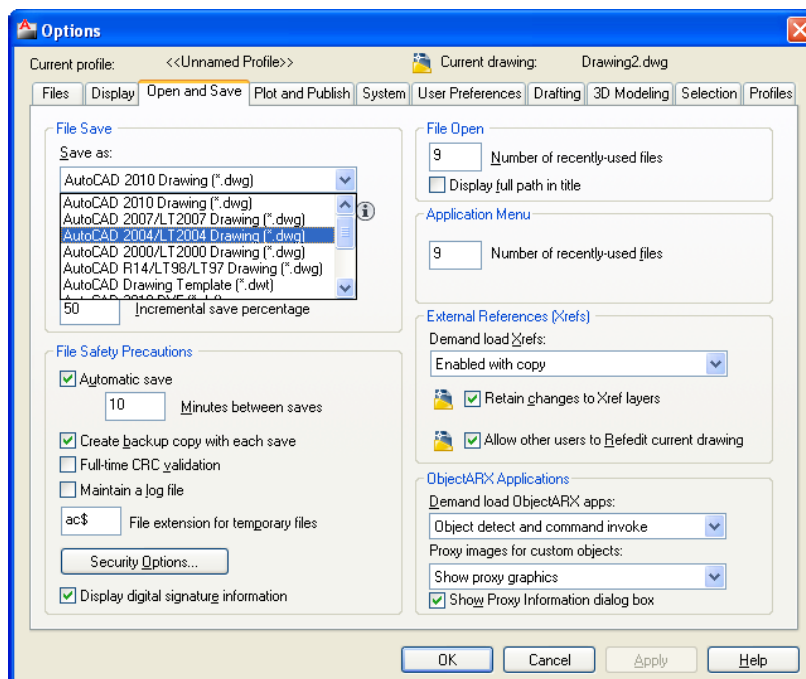


or

Type QSAVE at the command prompt,
Command: **QSAVE**

TIP:

- Drawings can be saved as different versions of AutoCAD using the 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.

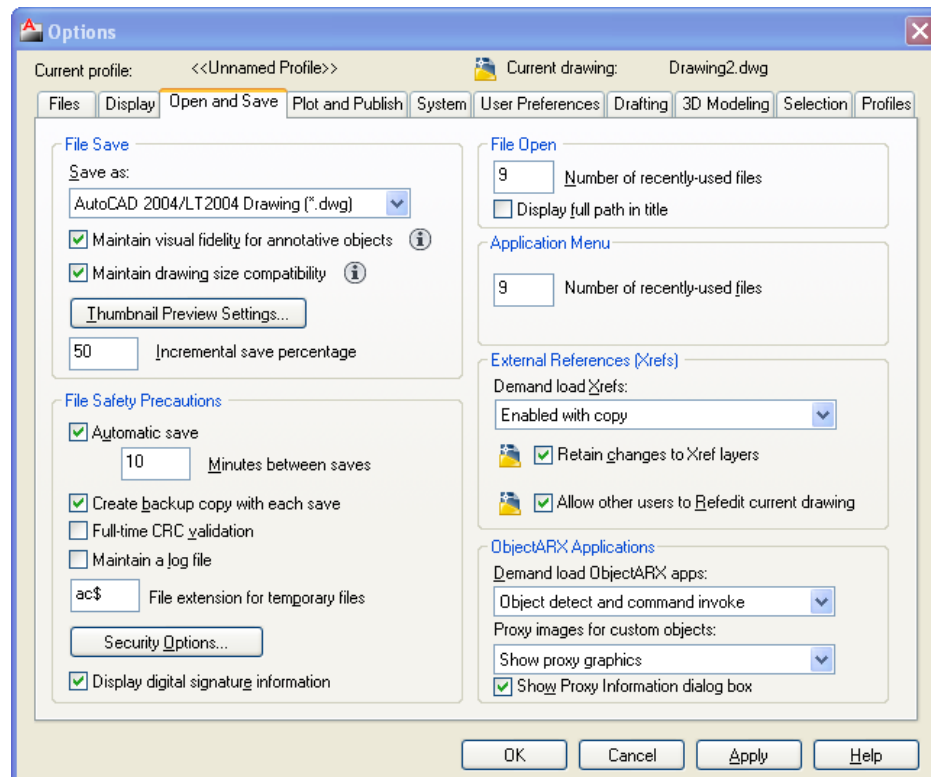
Temporary Files

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.

1. **Click** AutoSave and SV\$ under **Tools, Options....**, Open and Save

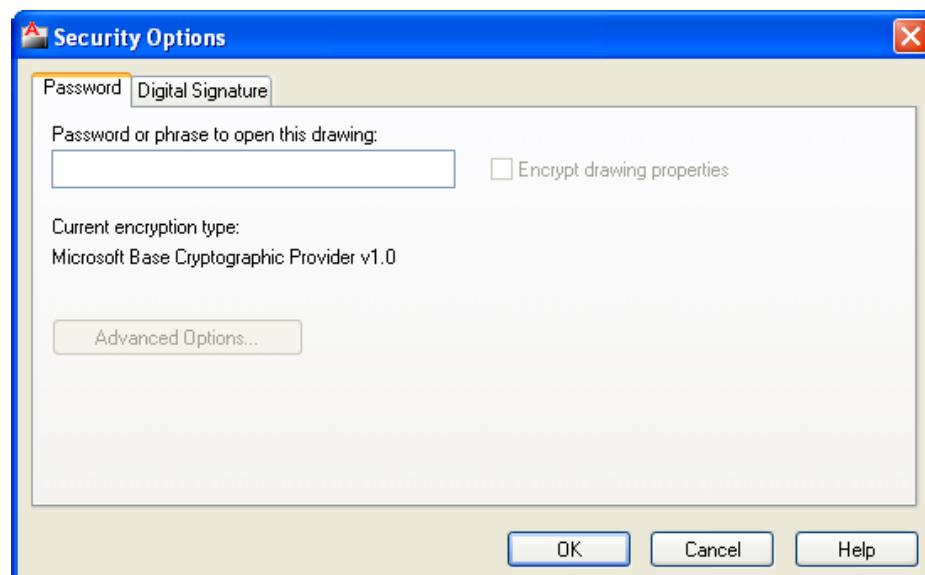
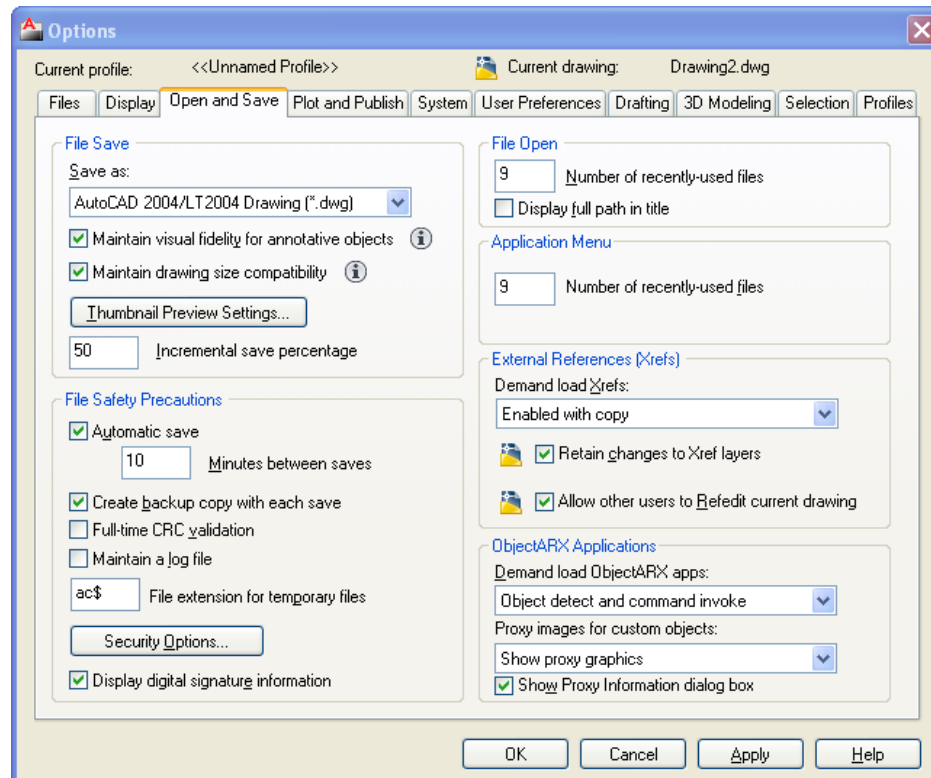


TIP:

- AutoCAD creates .BAK files that can be renamed to .DWG file

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. **Click** the **A** icon, 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



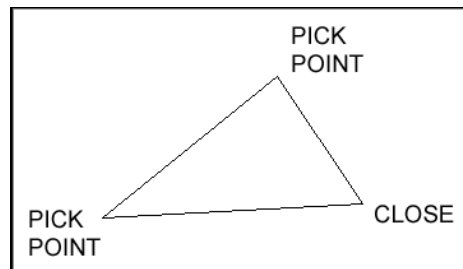
1. **Choose** Draw, Line.
or
Click the Line icon.
or
Type LINE from the command prompt

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

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

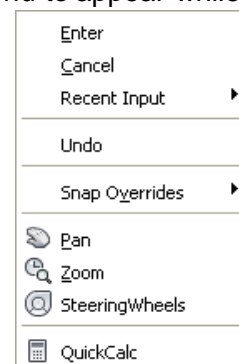
- or
- Type** U to undo the last segment
To point: **U** (undo)
or

7. **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

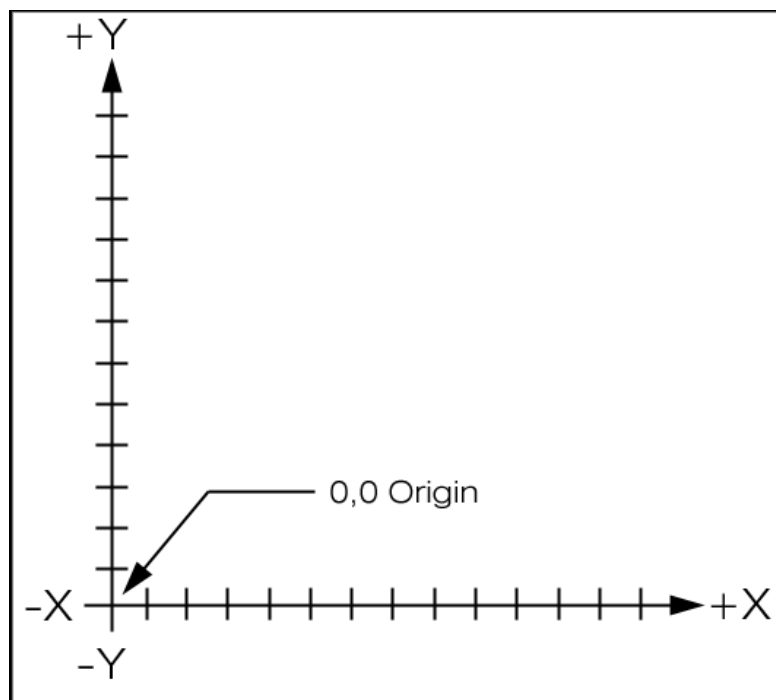


3.2 Cartesian Coordinate System

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

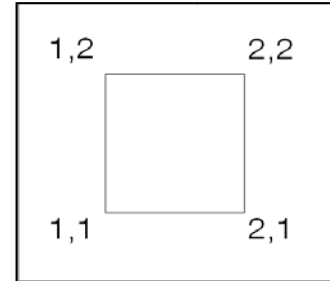
x,y coordinate when AutoCAD asks for a point.
From point: **1,1**

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

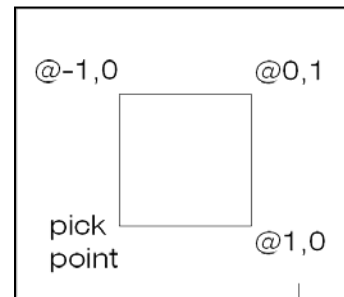
@deltax,deltay when AutoCAD asks for a point. From point pick point

To point: **@1,0**

To point: **@0,1**

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

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



Polar Coordinates

1. Type

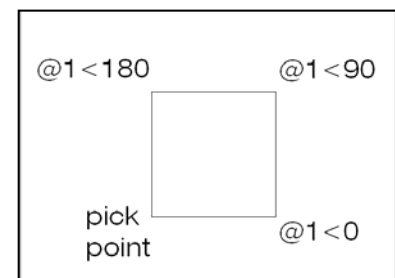
@distance<angle when AutoCAD asks for a point.
From point: pick point

To point: **@1<0**

To point: **@1<90**

To point: **@1<180**

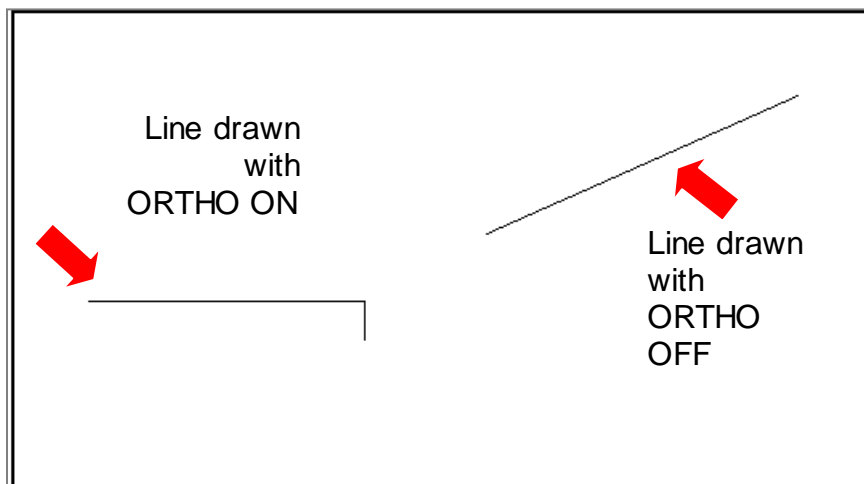
To point: **@1<270**



3.4 Orthogonal Lines

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

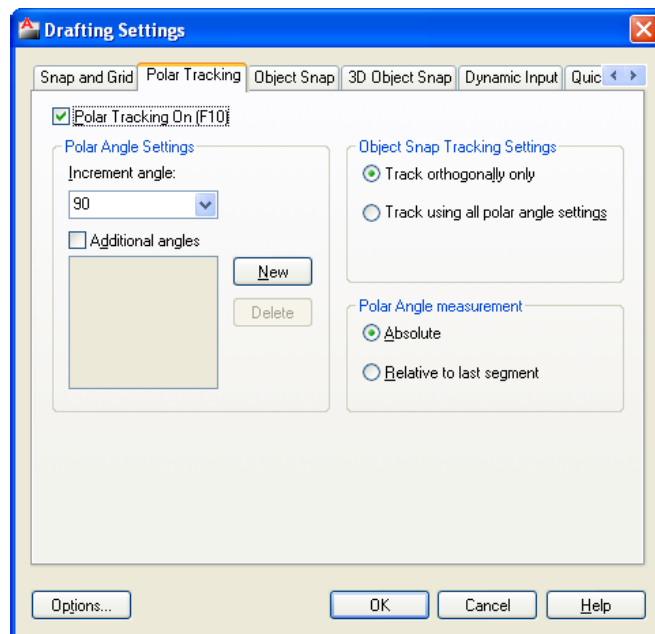
- | | |
|---------------------|----------------------------|
| 1. Press | Function Key F8 |
| | or |
| Double-click | ORTHO from the Status Bar. |
| | or |
| Press | CTRL + L. |



3.5 Polar Tracking

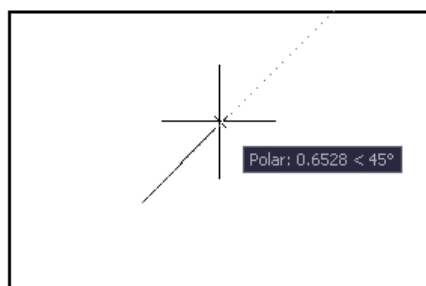
Polar Snaps work independently from snaps. With Polar Snaps on, AutoCAD shows the distances and angles being displayed as the cursor moves.

1. **Type** DSETTINGS at the command prompt. Command : **DSETTINGS**
2. **Choose** the Polar tracking TAB from the dialog box.
3. **Select** the desired incremental angle from the dropdown list
4. **Click** OK to exit the dialog box.
5. **Draw** a LINE using the Polar Snap references.

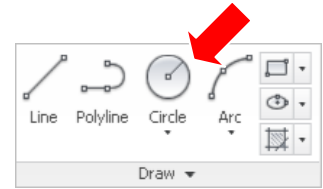


TIP:

- You can type a new angle in the dialog box.



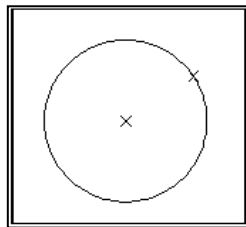
3.6 Circles



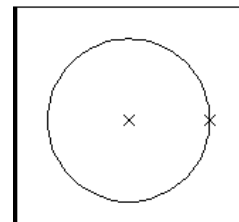
Circle Command

1. **Choose** Draw, Circle.
or
Click the Circle icon.
or
Type CIRCLE at the command prompt.
Command: **CIRCLE**
2. **Type** One of the following options:
3P/2P/TTR/⟨center point⟩:
or
Pick A center point.
3. **Type** A radius or diameter.
or
Pick A radius or diameter
Diameter/⟨radius⟩

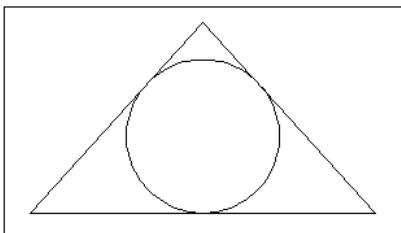
Circle, Center Radius



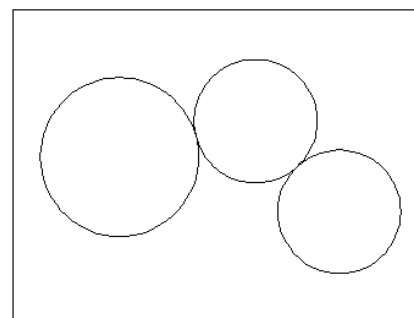
Circle, Center Diameter



Circle, Tangent, Tangent, Tangent



Circle, Tangent, Tangent Radius



TIP:


- To create circles that are the same size, press ENTER when asked for the circle radius.

3.7 Arc Command

1. **Choose**

Draw, Arc.

Click

or
the Arc icon. 

or

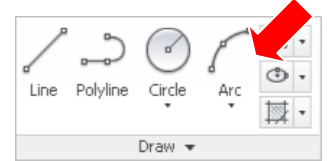
Type

ARC at the command prompt

Command: **ARC**

2. **Draw**

One of the arcs.

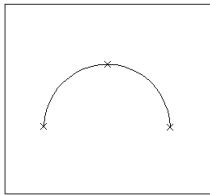


TIP:

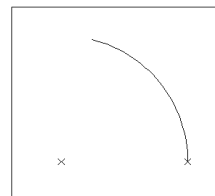
- Except for 3 point arcs, arcs are drawn in a COUNTERCLOCKWISE direction.

Arc Examples

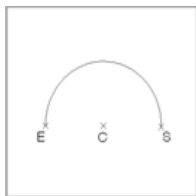
3 point arc



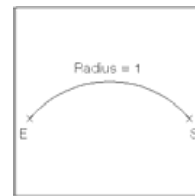
Start ,center, chord length



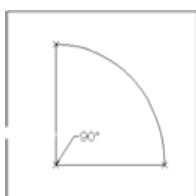
Start, center, end



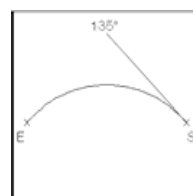
Start, end, radius



Start , center, included angle



Start, end, direction

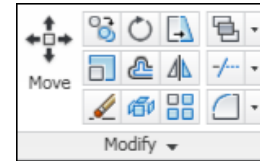



Chapter 4

Erase and Selection Sets

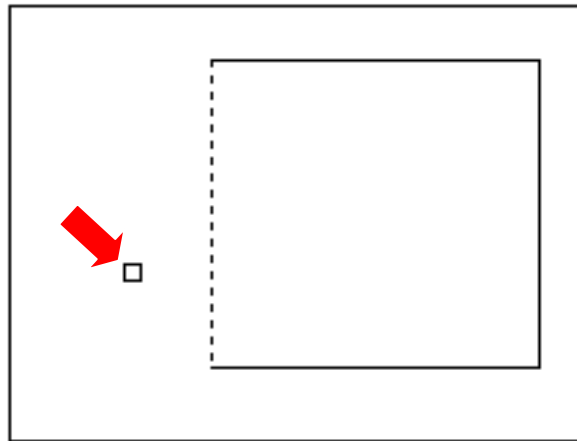
4.1 Erase Command

Deletes objects from a drawing.



1. **Choose** Modify, Erase.
 or
Click the Erase icon. 
 or
Type ERASE at the command prompt.
 Command : **ERASE** or **E**
2. **Pick** Object at the select object prompt.
 Select objects: (**pick object**)
3. **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

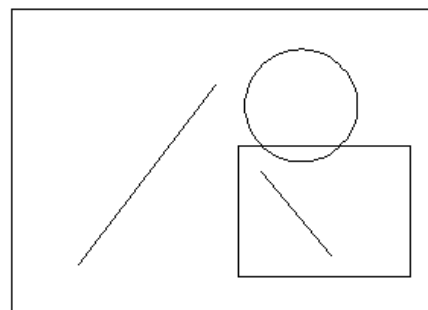
Type one of the following options at the “Select objects:” prompt

ALL	All objects within the drawing are selected unless they are on frozen or locked layers.
Multiple	Multiple objects selected without highlighting (faster edits).
Last	Last object drawn.
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

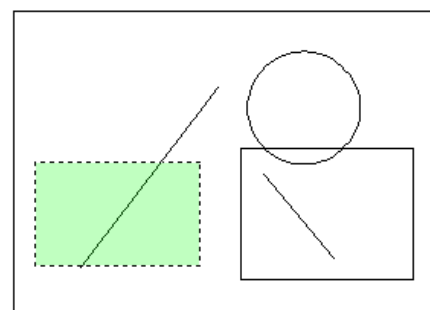
Window

Objects fully enclosed within Window.



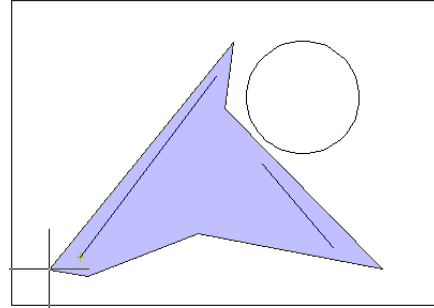
Crossing

Objects within or Crossing a window.

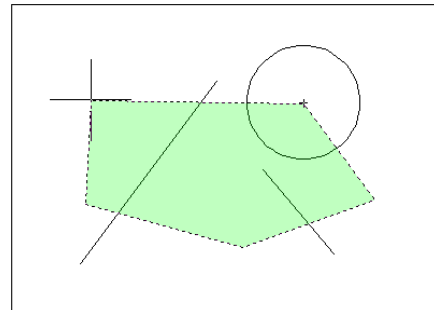


WPolygon

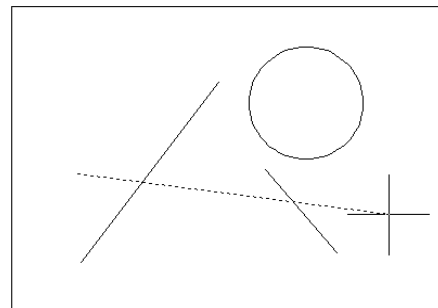
All entities within the boundaries of a polygon created by inputted points.

**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 Command

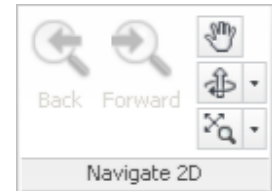
Reinserts the last erased set of objects or block even if it was not the last command issued. Otherwise Oops acts like UNDO.

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

Chapter 5

Basic Display Commands

5.1 ZOOM



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

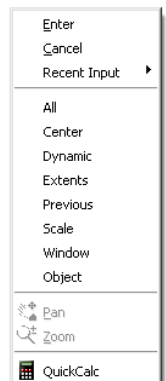
1. **Choose** View, Zoom.
or
2. **Type** ZOOM at the command prompt.
Command: **Zoom** or **Z**
3. **Type** One of the following zoom options:

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 the menu to the right.




5.2 PAN

Shifts the location of a view.

1. **Choose** View, Pan.

or

2. **Click** the Pan icon. 

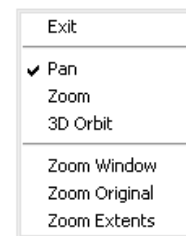
or

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.

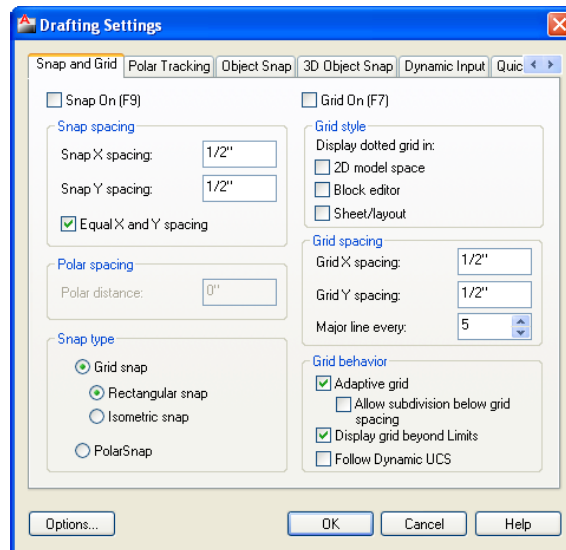
1. **Type** REGEN at the command prompt.
Command: **REGEN** or **RE**

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

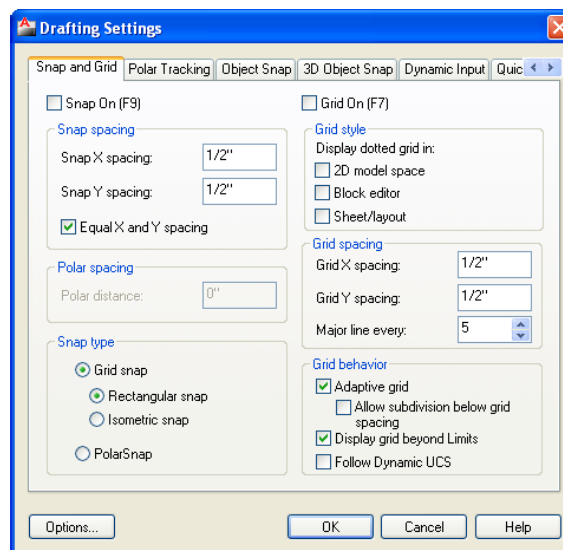
3. **Press** Function Key **F9** to turn the snap ON/OFF.
or
4. **Double Click** SNAP on the Status Bar.
or
5. **Press** CTRL + B.

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.

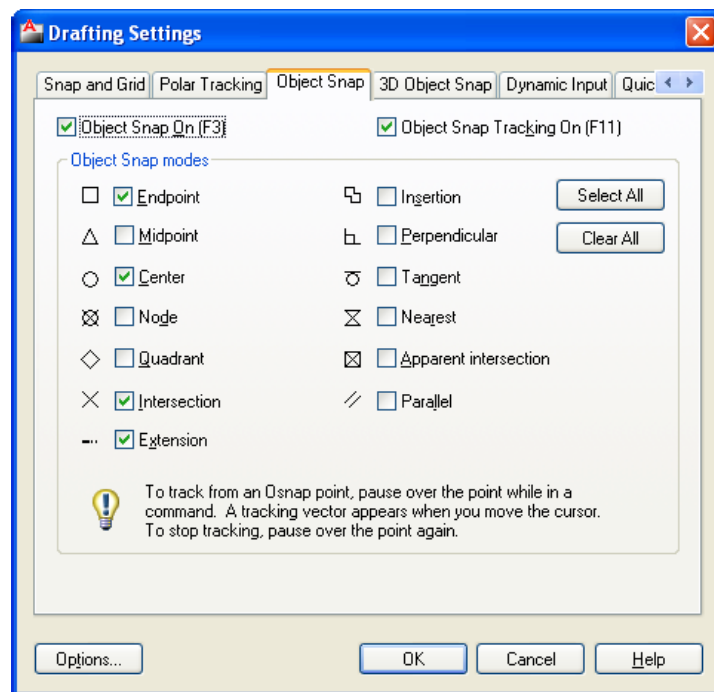
Chapter 7

Object Snapping

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.

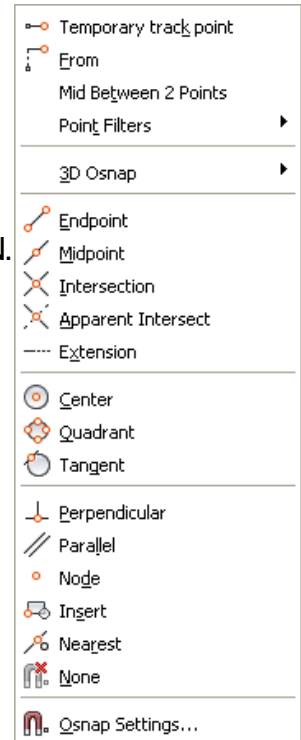
1. **Type** DDOSNAP at the command prompt
 Command: **DDOSNAP**
 or
 Click OSNAP on the Status Bar.
 Right Click the Object Snap TAB.
2. **Choose** an object snap to turn ON/OFF from the dialog box.



7.2 Case by Case (Temporary Mode)

1. **Press**

SHIFT + the RIGHT MOUSE BUTTON.



or

Click

one of the object snaps located Object Snap toolbar icon.



or

Type

The object snap at the prompt line.
Command: Line

From pt: **ENDP**

To pt: **MID**

To pt: **CEN**

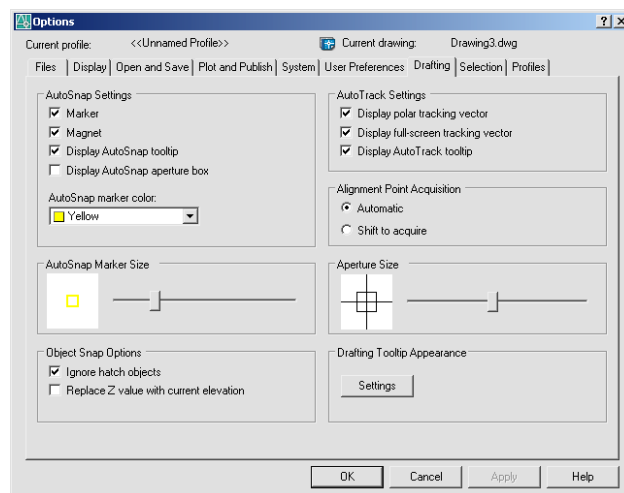
TIP:

- 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. **Type** Options at the command prompt.
Command: **OPTIONS**
2. **Select** the Drafting tab.
3. **Change** settings and choose OK.



The following are object snap modes. In bold caps are shortcut abbreviations to type.

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 definition 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

Chapter 8

Introduction to Edit Commands

8.1 Move Command

1. **Choose** Modify, Move.

or

Click the Move icon.



or

Type MOVE at the command prompt

Command: **MOVE** or **M**

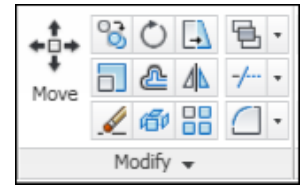
2. **Pick** Objects to move
Select objects: (**select**)

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

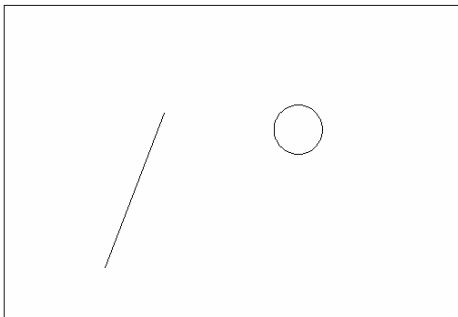
Base point or displacement: (**pick point**)

4. **Pick** A point to move to

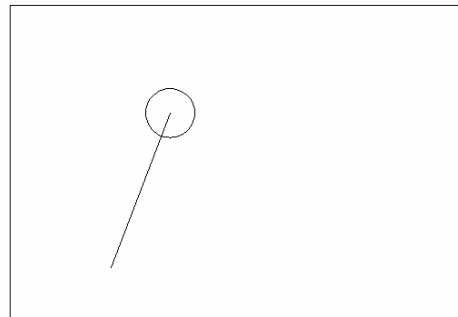
Second point of displacement: (**pick point**)



Circle before move



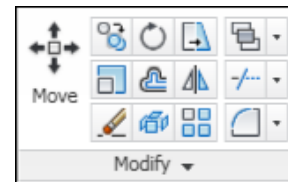
Circle after move



TIP:

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

8.2 Copy Command



1. **Choose** Modify, Copy.

or

Click

the Copy icon.



or

Type

COPY at the command prompt. Command:
COPY or CP

2. **Pick** Objects to copy.

Select objects: (**select**)

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

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

4. **Pick** A point to copy to.

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

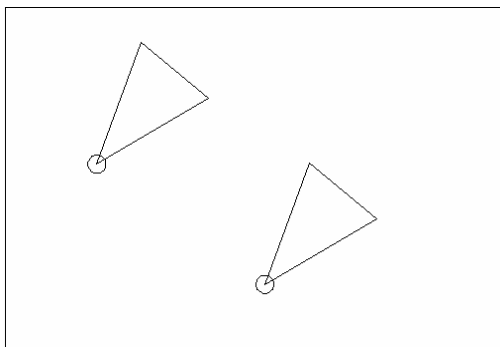
or

Type

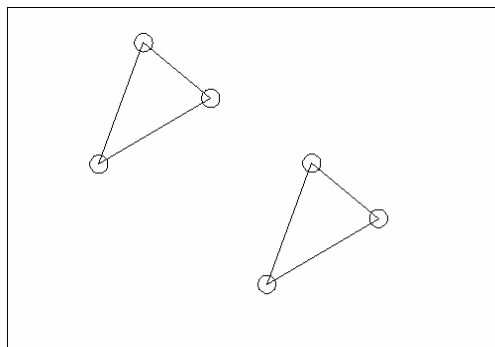
A point to copy to.

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

8.3 Previous Selection

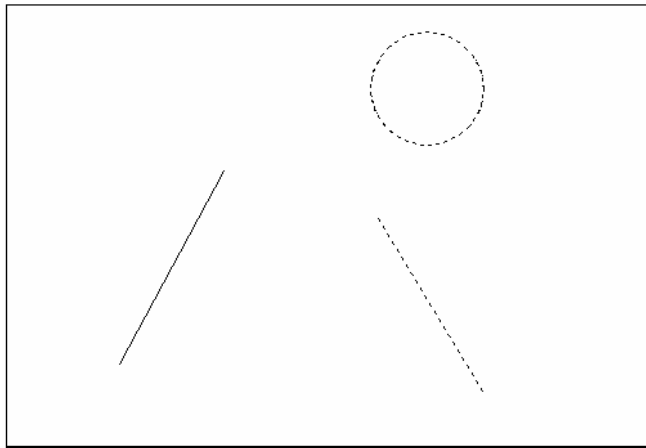
Places selected objects in the Previous selection set

1. **Choose** Modify, Move.
or
Click the Move icon.

or
Type MOVE at the command prompt.
Command:**MOVE or M**
2. **Pick** Objects to move. Select objects: **(P)**



Previous selection set is 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.

8.4 Offset Command

Offset Distance

To offset a specified distance:

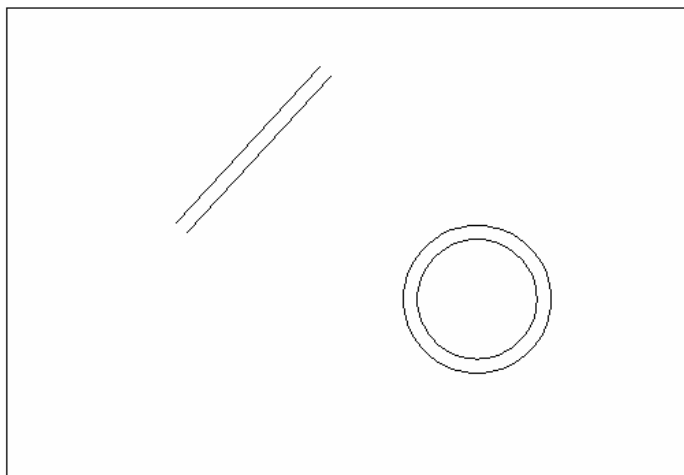
1. **Choose** Modify, Offset.
or
2. **Choose** the Offset icon. 
or
3. **Type** OFFSET at the command prompt. Command:
 OFFSET or O
4. **Type** The distance to offset.

 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.

Offset objects by specifying a distance



Offset Through Point


To offset through point :

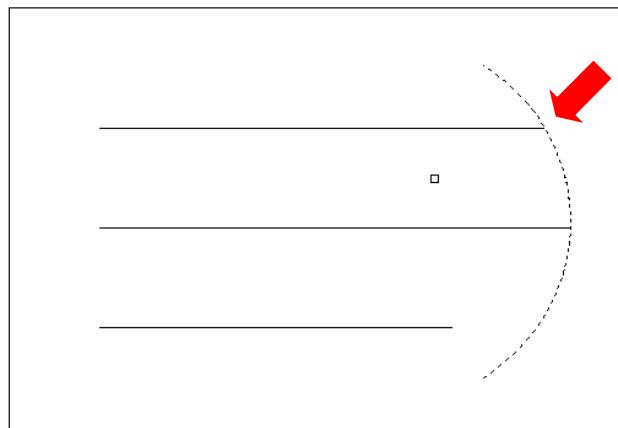
1. **Type** OFFSET at the command prompt
 Command: **OFFSET**
2. **Type** T to specify a through point
 Offset distance or <Through point>: (**T**)
3. **Pick** A point to offset through (HINT: use object
 snaps) Select object to offset: (**pick**)
 Through point: (**select object**)

Offset through a point



8.5 EXTEND

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)*

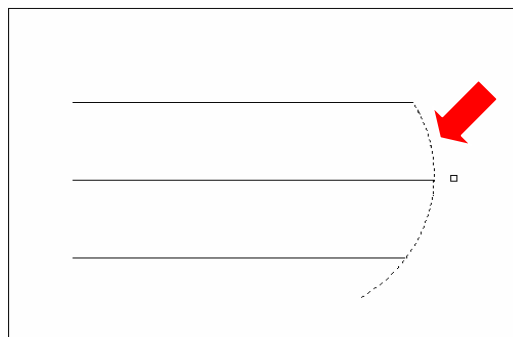
TIP:

- Use the object selection option FENCE to choose multiple objects

8.6 TRIM

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:

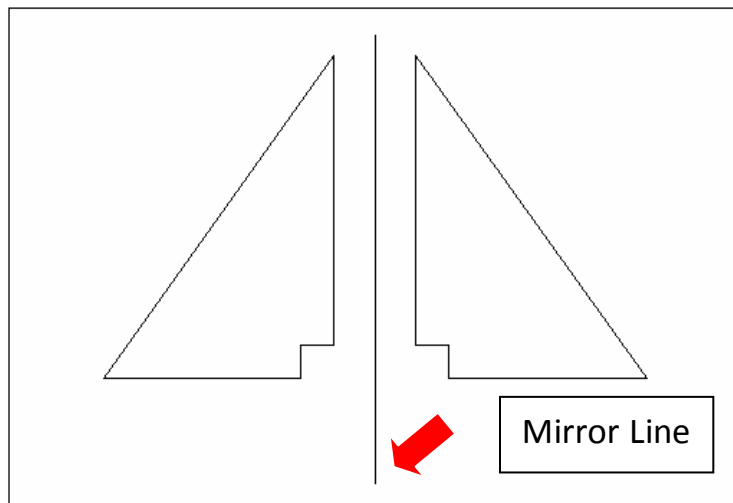
- Hold the SHIFT key to interactively extend instead of trim.

Edgemode

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

8.7 MIRROR

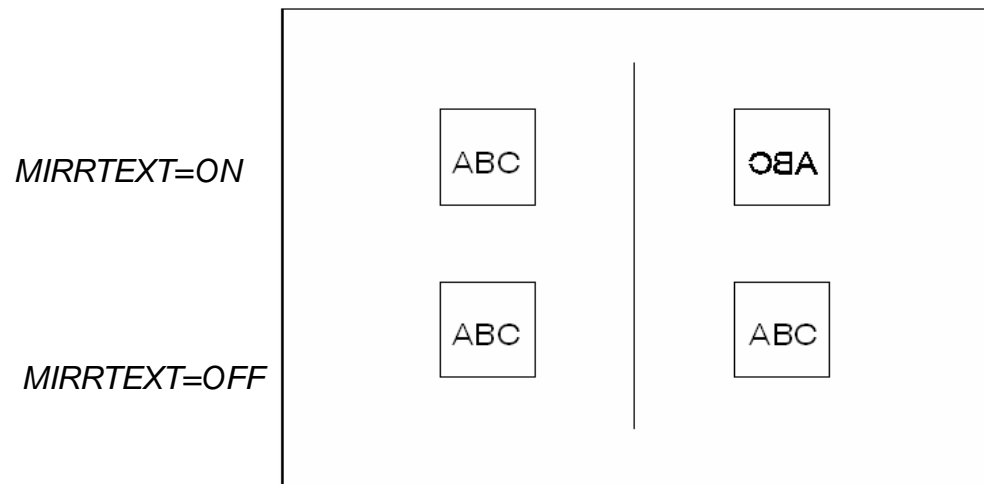
1. **Choose** Modify, Mirror.
or
2. **Click** the Mirror icon. 
or
3. **Type** MIRROR at the command prompt. Command:
MIRROR
4. **Pick** Objects to mirror.
Select objects:(**select**)
5. **Pick** First point of mirror line: (**point**)
6. **Pick** Second point: (**point**)
7. **Type** Yes to delete the original objects and
No to keep them.
Delete old objects? **Y** or **N**




Mirrtext

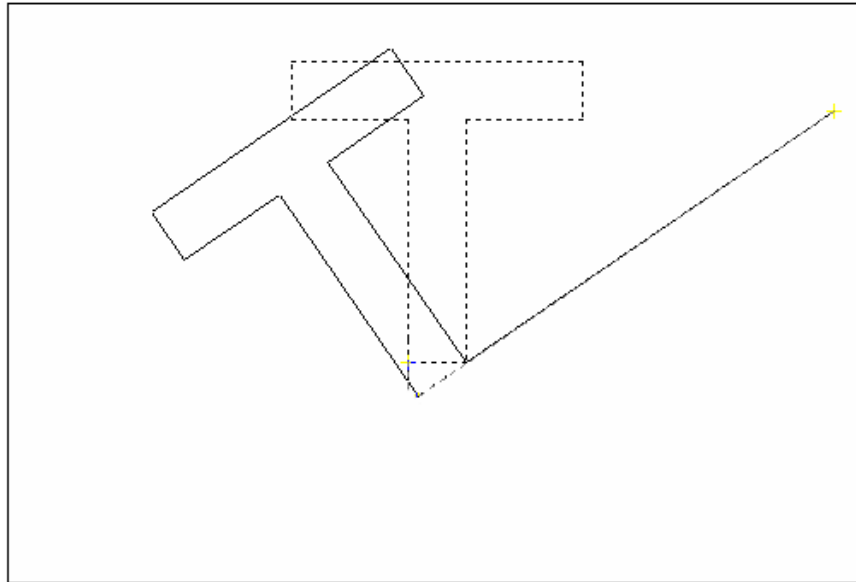
Mirror reflects (mirrors) text if 1, retains text direction if 0.

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**



8.8 ROTATE

1. **Choose** Modify, Rotate.
or
2. **Click** the Rotate 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:

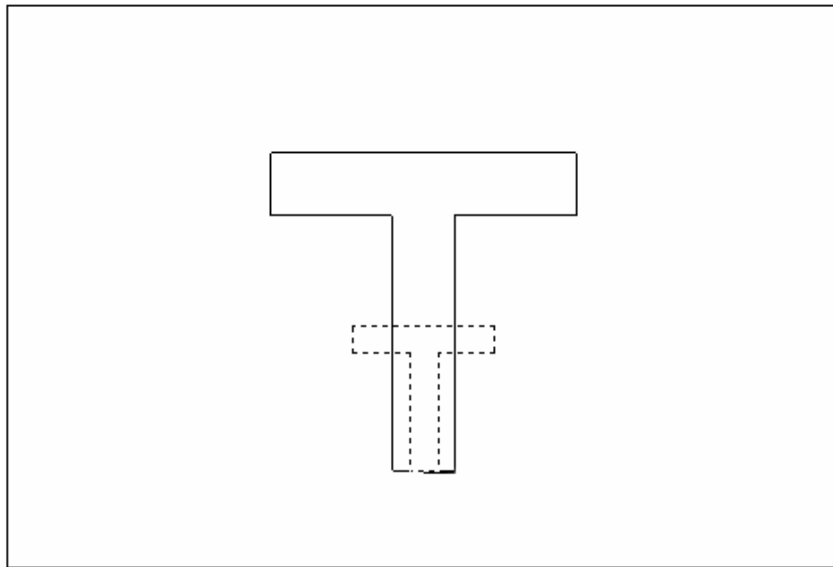
1. **Type** R for a rotation angle<Rotation angle>/Reference:
 (R)
2. **Choose** An existing rotation angle Rotation angle:
 (number or points)
3. **Choose** A new rotation angle New angle:
 (number or points)

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.

8.9 SCALE

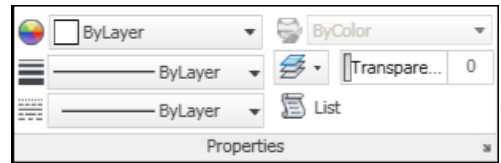
1. **Choose** Modify, Scale.
or
2. **Click** the Scale icon. 
or
3. **Type** SCALE at the command prompt
Command: **SCALE**
Select objects: (**select objects**)
4. **Pick** A pivot point to scale about Base point: (**point**)
5. **Type** A rotation angle<Scale factor>/Reference:(**number**)
or
6. **Pick** A scale factor<Scale factor>/Reference:
(**point**)
Scale factor/Reference: (**points**)



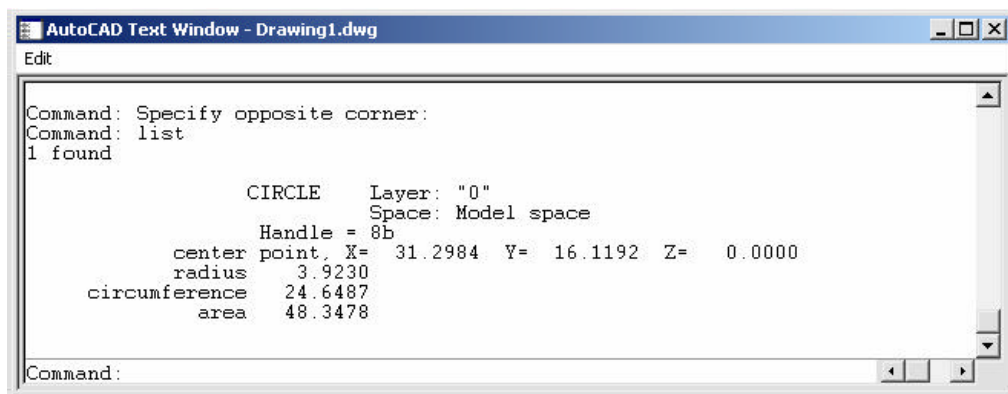
Chapter 9

Setting Up a Drawing

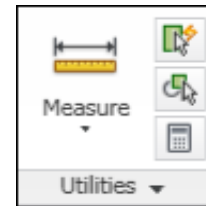
9.1 List Command



1. **Choose** Tools, Inquiry, List.
or
Click the List icon from the Inquiry Toolbar.
or
Type LIST at the command prompt.
Command: **LIST** or **LI**
2. **Pick** The object or objects to list.
Select objects: (select)
3. **Press** ENTER when you are finished choosing objects:



9.2 Measuring Distances



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

or

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



or

- Type** DIST at the command prompt

Command: **DIST**

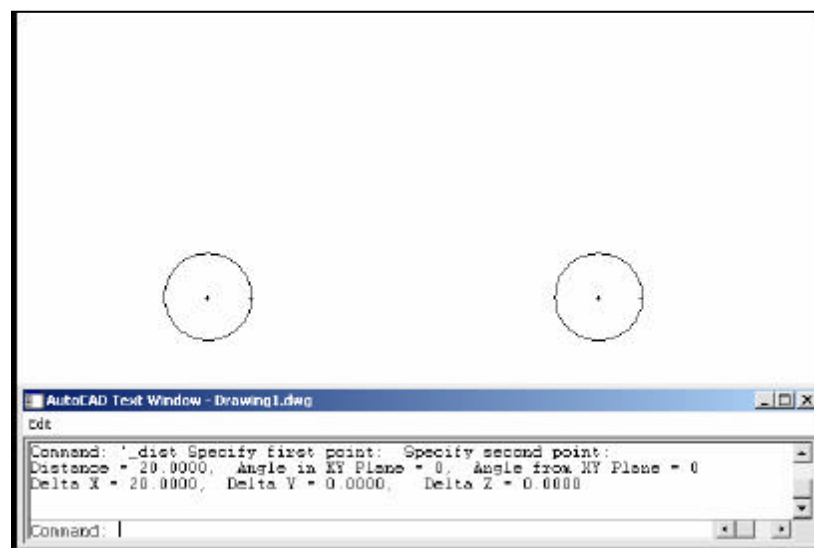
2. **Pick** The first point to measure from

First point: **pick point**

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

Second point: **pick point**


Distance between circle centers

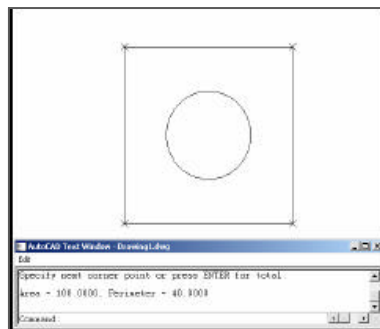


TIP:

- Be sure to use Object Snaps with the MEASURE command.

9.3 Calculating Areas

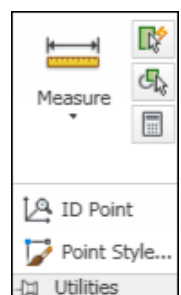
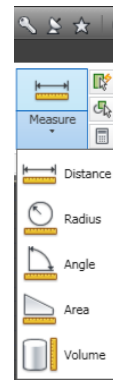
1. **Choose** Tools, Inquiry, Area.
or
Click the Area icon. 
or
Type AREA at the command prompt
Command: **AREA**
2. **Pick** The first point for area calculation
<First point>/Object/Add/Subtract: **pick**
3. **Pick** Next point: **pick**
4. **Pick** Next point: **pick**
5. **Press** ENTER when you 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.

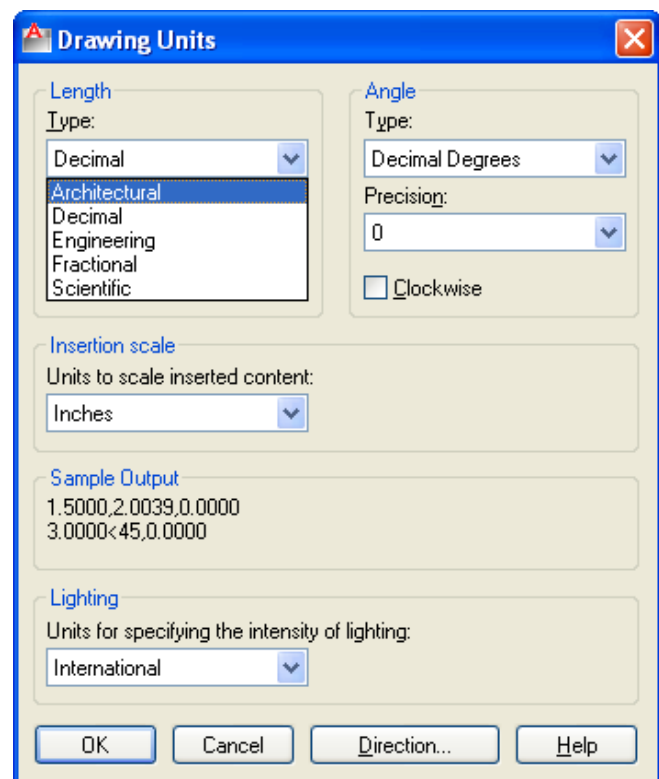
TIPS:

- 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.



9.5 UNITS Command

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



9.6 Drawing Limits

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 |
| Type | LIMITS at the command prompt |
| | Command: LIMITS |
| 2. Type | One of the following options
On/Off/Lower left corner <.000,0.000>: 0,0 |
| 3. 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:

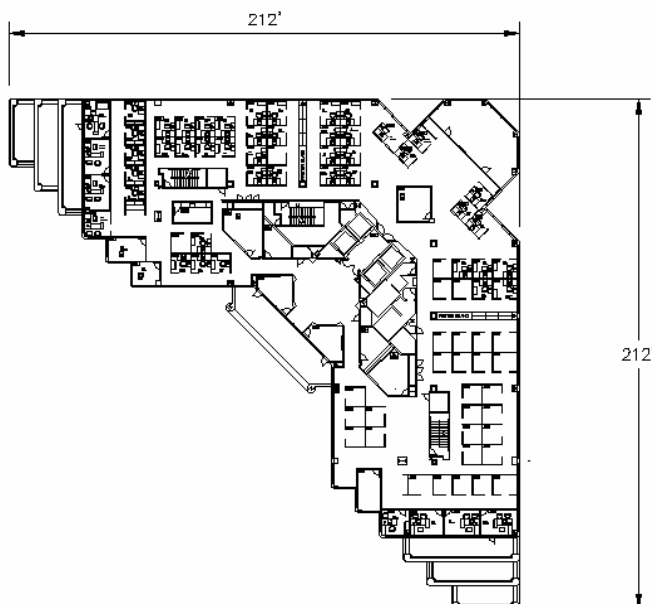
- You can also pick points to define the limits.
- The limcheck variable controls whether or not you can draw outside the limits that are set. A setting of 0 (off) indicates that you can draw outside the limits and a setting of 1(on) indicates that you cannot

9.7 Plot Scales and Paper Sizes

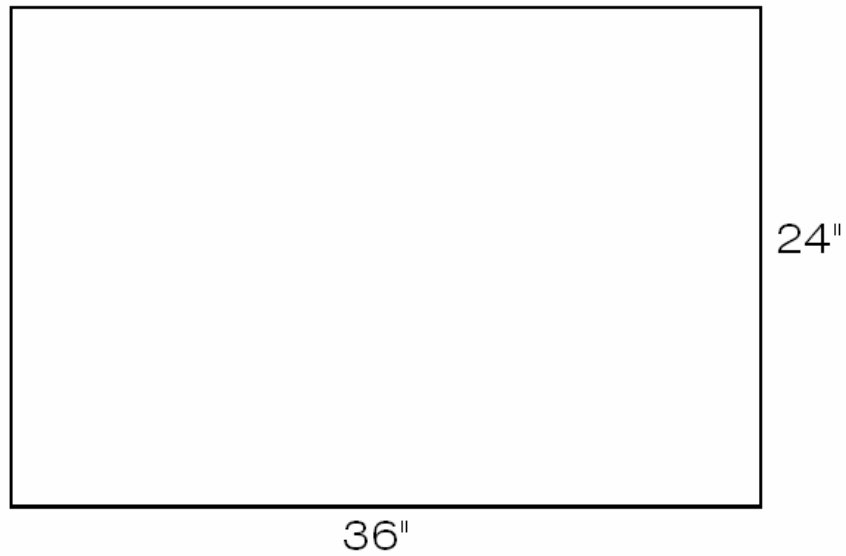
The following is an example of setting up an AutoCAD drawing for a D size sheet of paper (36 x 24) with a scale of 1/16=1'.

1. **Size** the object you're drawing.
2. **Choose** a **border size** (e.g. 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. **Set** the drawing **limits** (e.g. lower left limit 0,0. and upper right limit 576', 384').
4. **Set** a **text height** (e.g. for 1/8 notes, multiply by 192 which is the reciprocal of the plot scale 1/8 plotted, 24" drawn).
5. **Set** a **hatch scale** for patterns other than architectural. Hatch Scale = 192
6. **Set** a **dimension scale** (e.g. dimscale = 192)
7. **Set** a **linetype scale** (e.g. ltscale = 96)

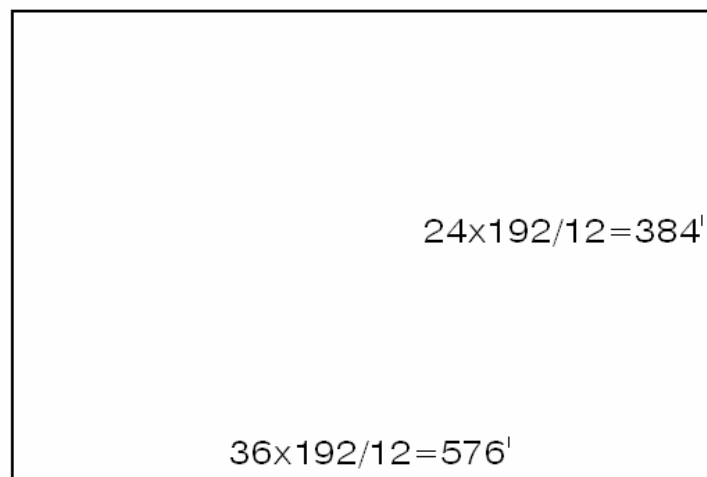
Determine your object size



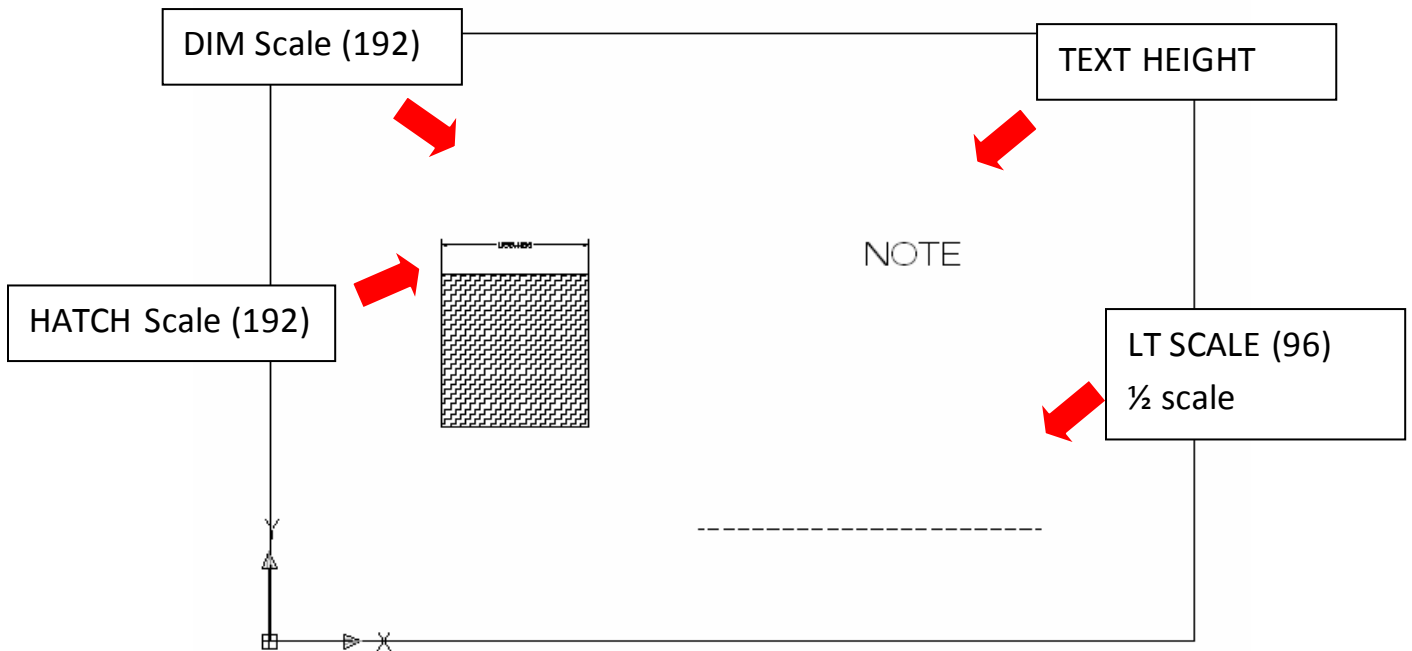
Decide Border (Paper) Size



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



Chapter 10


Text

10.1 Text Command

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.

10.2 Text Justification

1. **Type** JUSTIFYTEXT at the command prompt
Command: **JUSTIFYTEXT**
or
2. **Pick** the Justify Text icon from the Text Toolbar.



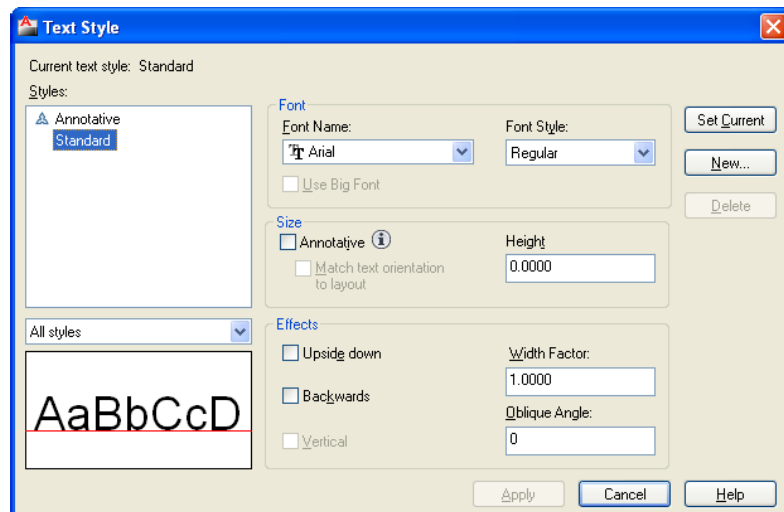
Text Justifications

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.

10.3 Text Styles

Style Command

1. **Choose** Format, Text Style...
or
2. **Type** STYLE at the command prompt. Command:
 STYLE
3. **Choose** a style from the menu or create a NEW style.
4. **Choose** a font file.
5. **Type** a height for the text (set to zero to vary heights)
6. **Type** a width factor for each character. Width factor
 <1>: (**enter**)
7. **Type** an obliquing (slant) angle.
 Obliquing angle <0>: (**angle or enter**)
8. **Type** Yes or No to place characters
 backwards. Backwards? (**Y or N**)
9. **Type** Yes or No to draw characters upside
 down. Upside down? (**Y or N**)
10. **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

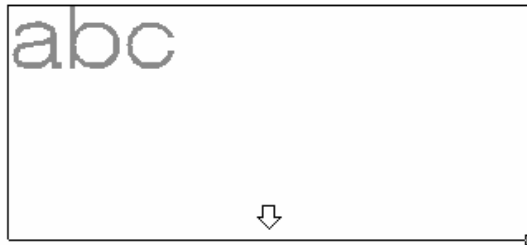
TIP:

- 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.

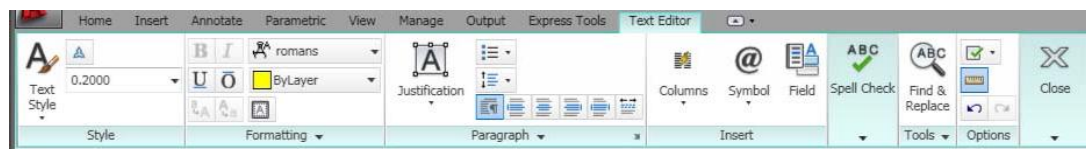
10.4 Multiline Text

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.



7. **Close** the Text Editor when done.

10.5 Editing Text

DDEDIT

1. **Choose** Modify, Text...
or
2. **Type** DDEDIT at the command prompt.
Command: **DDEDIT** or **ED**
3. **Pick** The text to edit.
Select objects: (pick text)
4. **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



10.6 Special Control Codes

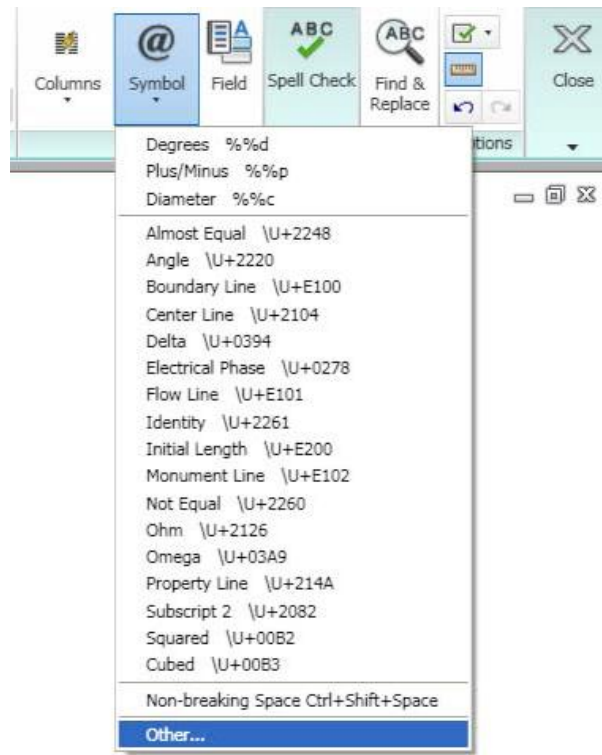
AutoCAD provides special control codes to return drafting symbols when using text.

1. Type

The following characters to return equivalent symbol:

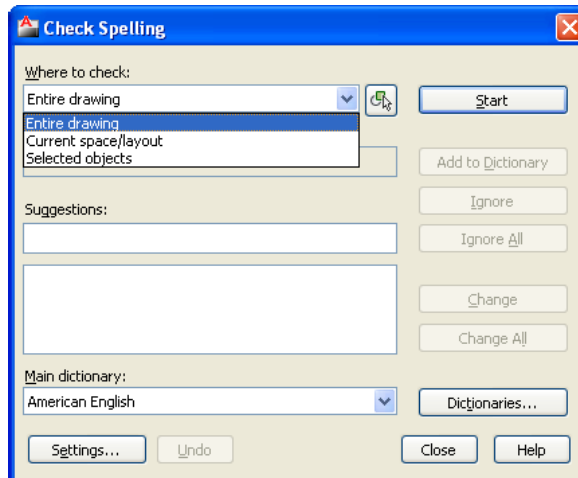
%%d	degree symbol (°)
%%c	diameter symbol (Ø)
%%p	plus minus symbol (±)
%%u	to start and stop underlining (NOTE)
%%o	to start and stop overscoring (NOTE)

The MTEXT command has additional symbols that can be accessed from the Insert tab.

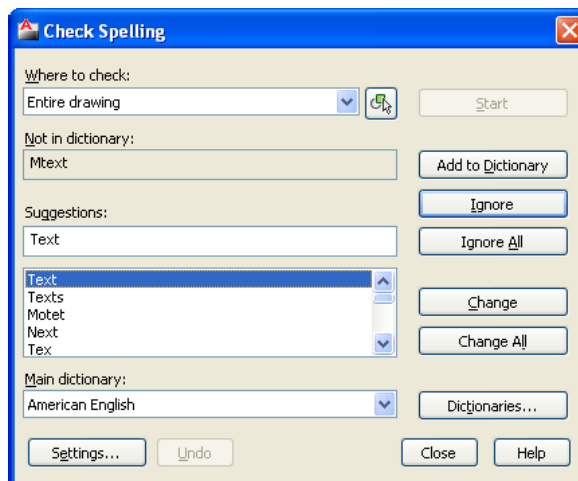


10.7 Spell Check

1. **Choose** Tools, Spelling
or
2. **Type** SPELL at the command prompt. Command: **SPELL**



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



5. **Click** Close when finished.

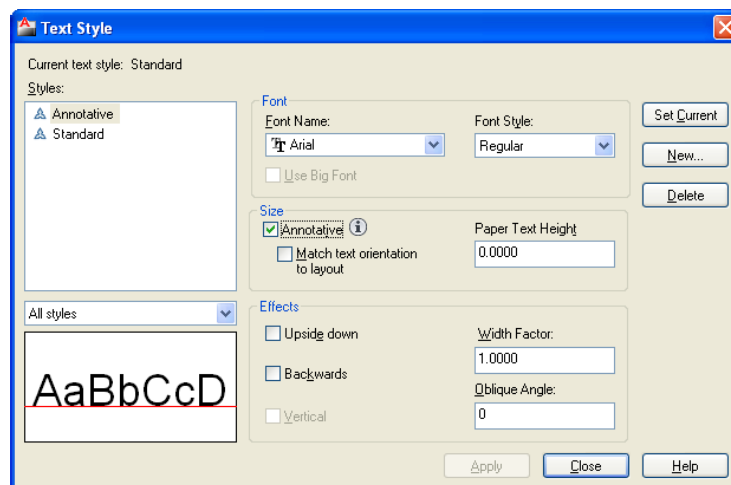
10.8 Annotative Text

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. Type **STLYE** at the command prompt
Command: **STYLE**
2. Click **Annotative** as the text size.



Chapter 11

Layers, Linetypes, Colors

11.1 Introduction to Layers and Layer Dialog Box

1. **Choose**

The Layer Properties icon.

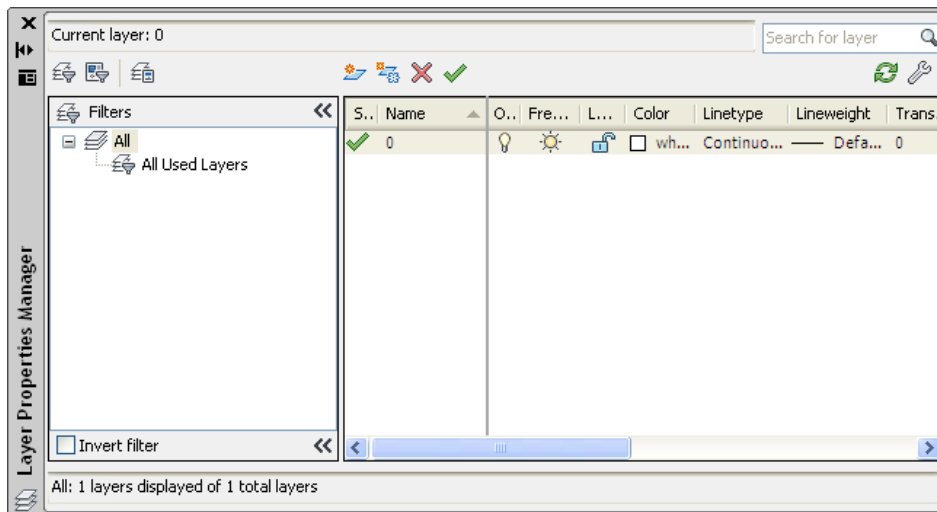


or

2. **Type**

LAYER at the command prompt.

Command: **LAYER (or LA)**



11.2 Layer Options

?	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**

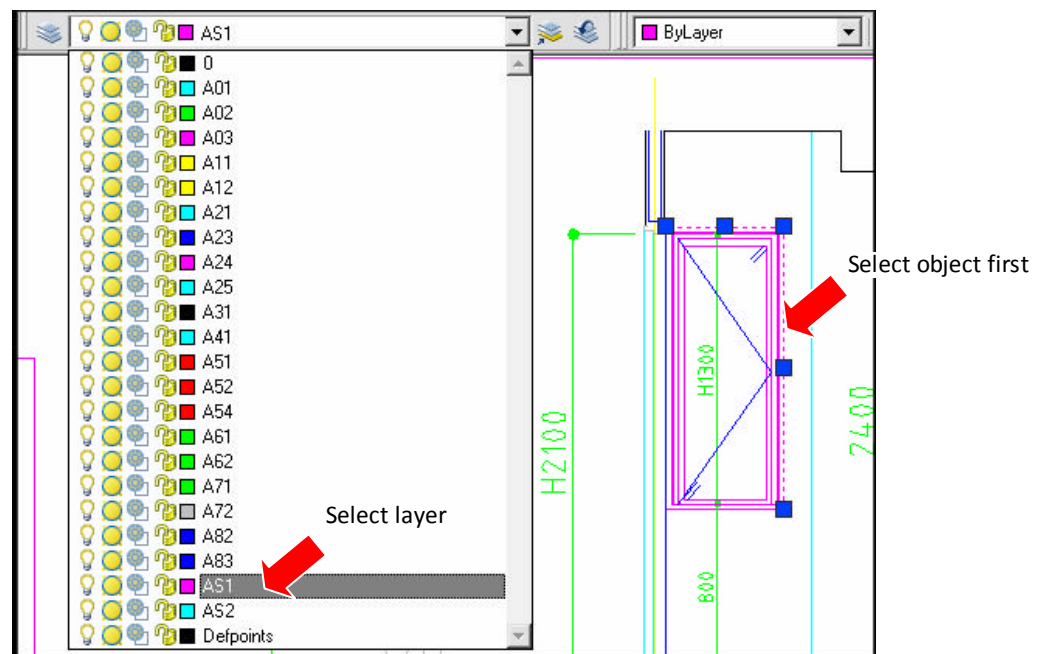
2. **Type** One of the following layer options
 ?/Make/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw:

11.3 Layer Shortcuts


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



11.4 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

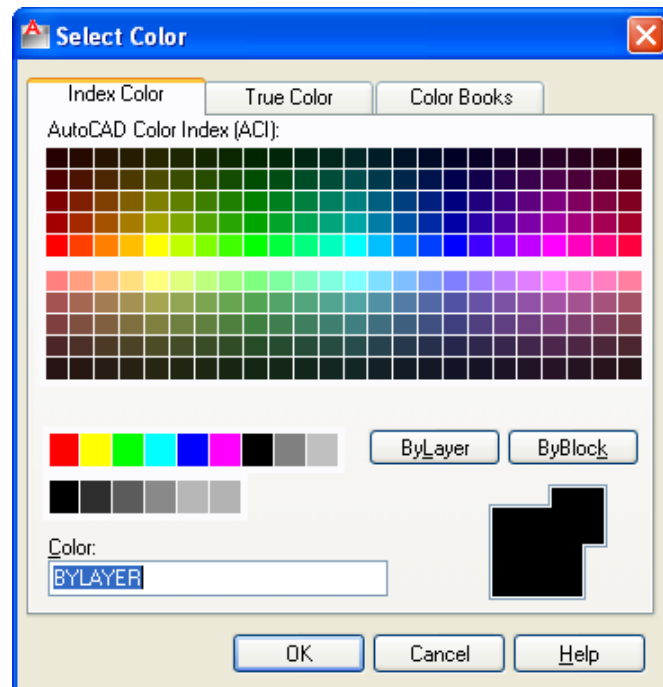
1. **Choose** Modify, Match Properties.
or
2. **Click** the Match Properties Icon from the Layer toolbar. 
or
3. **Type** Command : **MATCHPROP** or **MA**
4. **Select** the object whose properties you want to copy (1).
5. **Select** the objects to which you want to apply the properties (2).

11.5 Color Command

1. **Type** COLOR at the command prompt.
Command: **COLOR or COL**

or

2. **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.

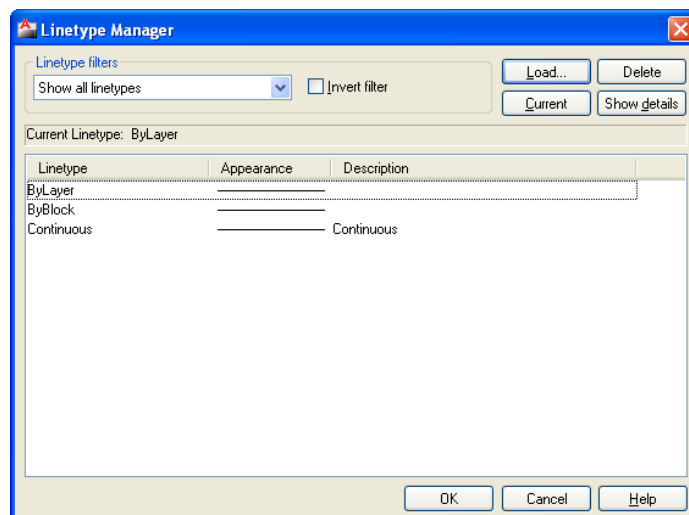
By Block

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.

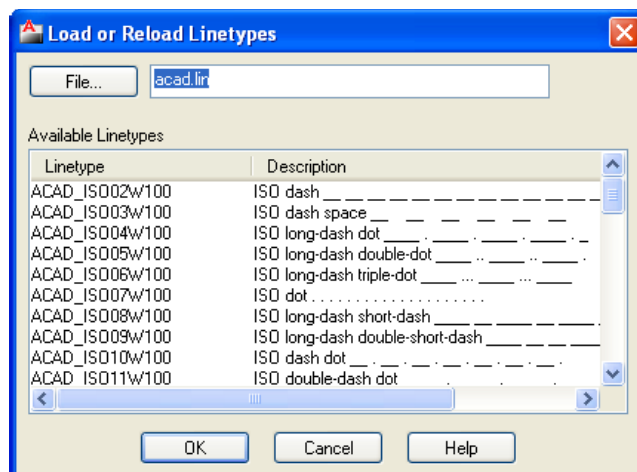
11.6 Linetypes

Loading and Changing Linetypes

1. **Type** LTYPE at the command prompt. Command: **LTYPE or LT**
2. **Choose** Load... to see a list of available linetypes.



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

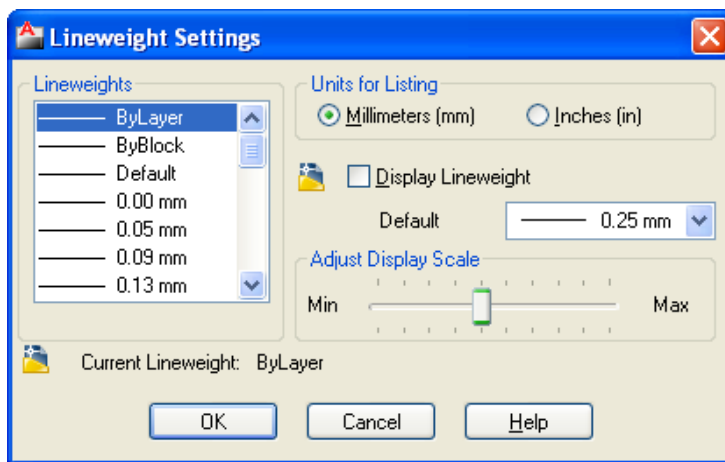


4. **Click** OK.

11.7 Lineweights

Loading and Changing Lineweights

1. **Type** LINEWEIGHT at the command prompt.
Command: **LINEWEIGHT** or **LWEIGHT**



TIPS:

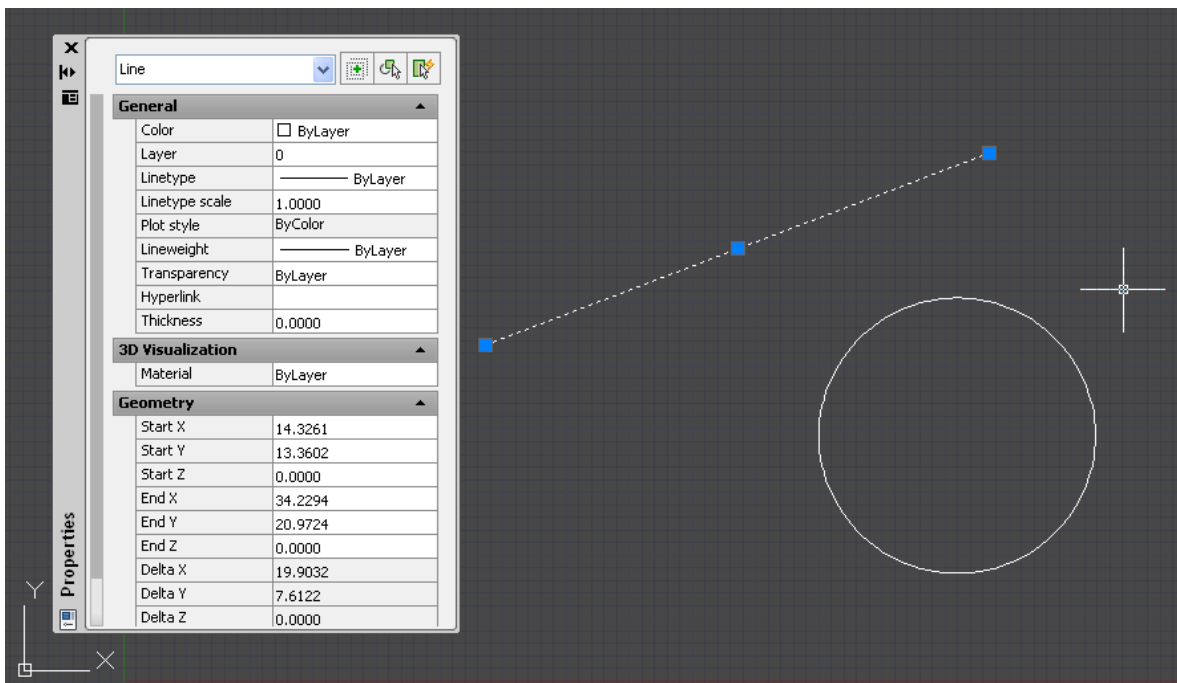
- Lineweights can also be assigned to layers.
- The Display Lineweights feature can be turned on/off on the status bar to show or not show lineweights in the drawing, thus making regenerations faster.



- 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.

11.8 Object Properties


1. **Type** DDCHPROP or DDMODIFY at the command prompt. Command: **DDCHPROP** (CH) or **DDMODIFY** (MO)
2. **Pick** Objects whose properties you want to change Pick a window for DDCHPROP, single object for DDMODIFY.
Select objects:(select)
3. **Press** ENTER to accept objects.
Select objects: (press enter)
4. **Choose** One of the following properties to change.

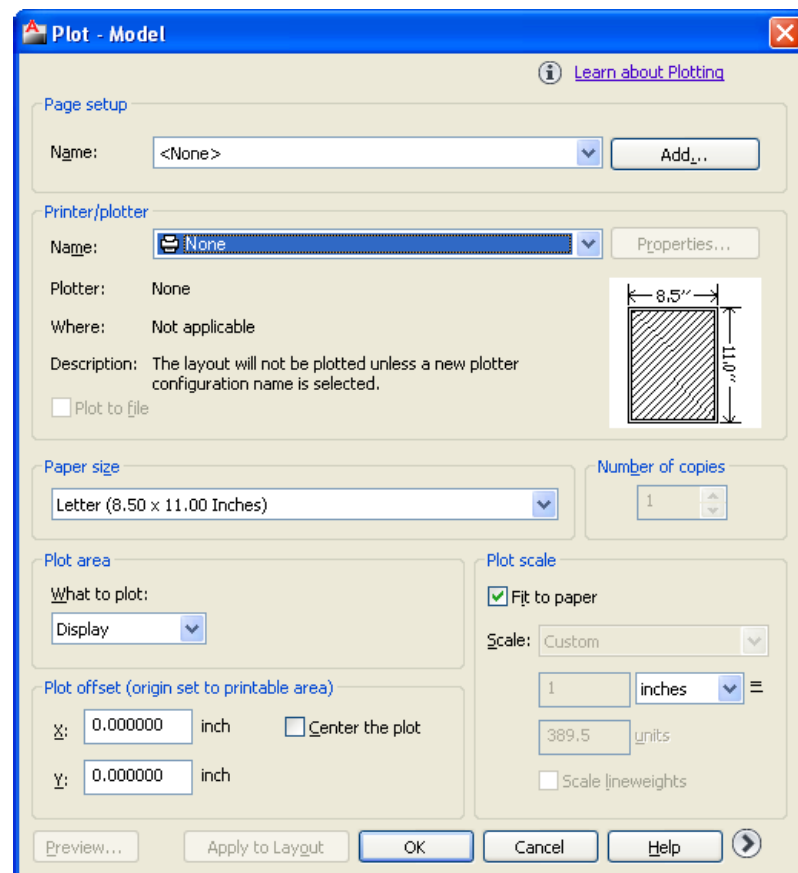


Chapter 12

Plotting

12.1 Plot Command

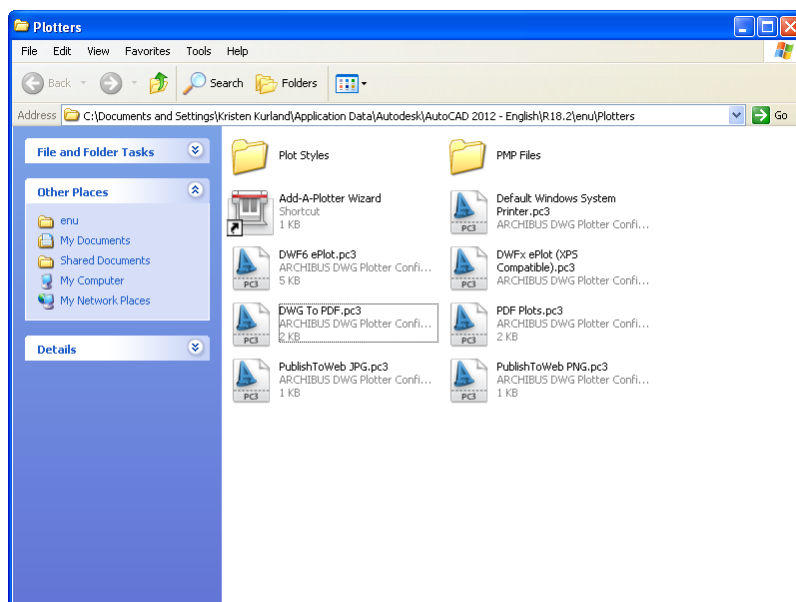
1. **Choose** File, Plot.
or
Click the Plotter icon. 
- or
Type PLOT at the command prompt.
Command: **PLOT** or **PRINT**
or
Press CTRL + P



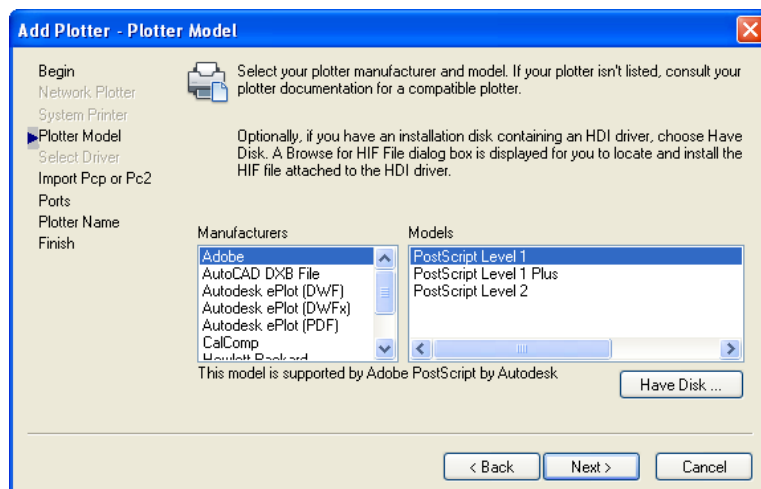
12.2 Adding a Plotter

Plotter Manager Wizard

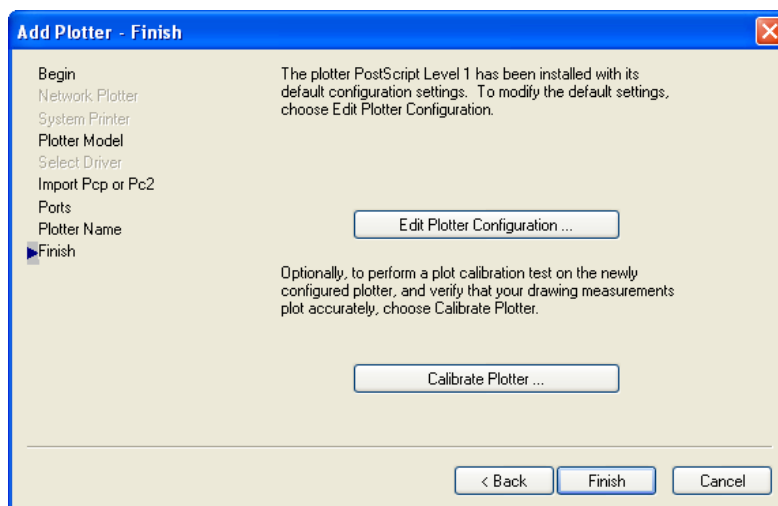
1. **Choose** File, Plotter Manager
2. **Double-Click** the Add a Plotter Wizard icon. AutoCAD adds a plotter configuration to a saved plot file. You can then load from this file later.



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. **Choose** one of the Autodesk Plotter model 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.



6. **Click** Next > until the Finish prompt appears. You may need to assign a port that the plotter is assigned to.



7. **Click** the Finish button.

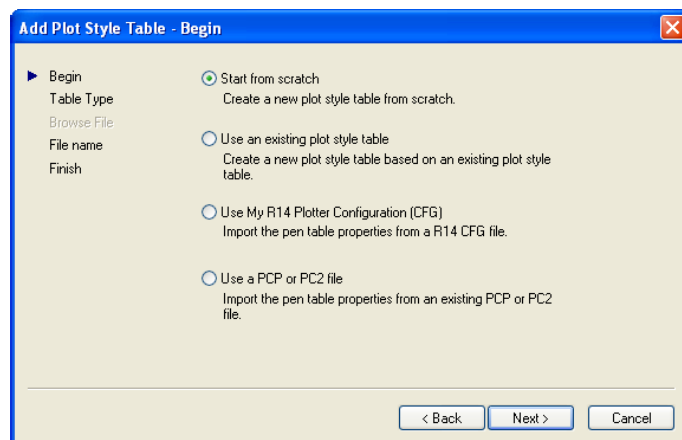
12.3 Plot Styles

Add a Plot Style

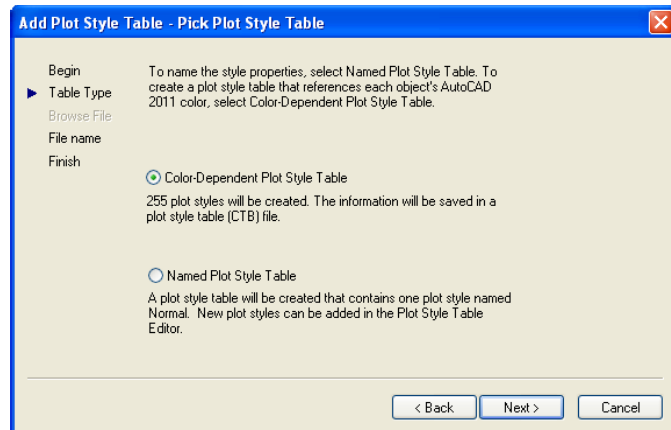
A plot style controls how an object or layer is plotted by determining plotted properties such as lineweight, color, and fill style. Plot style tables collect groups of plot styles. The Plot Style Table Manager is a window that shows all the plot style tables available in AutoCAD.

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.
or
2. **Type** STYLESMANAGER at the command prompt.
Command: **STYLESMANAGER**
3. **Double-Click** Add a Plot Style Table Wizard icon.
4. **Click** Next >
5. **Choose** Start from Scratch to create a new Plot Style.
6. **Click** Next >



7. **Choose** Color-Dependent Plot Style Table
8. **Click** Next >

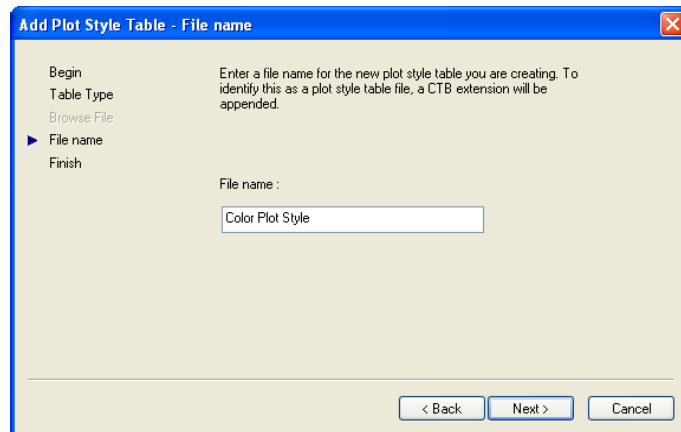


9. **Type**

a name for the plot style table.

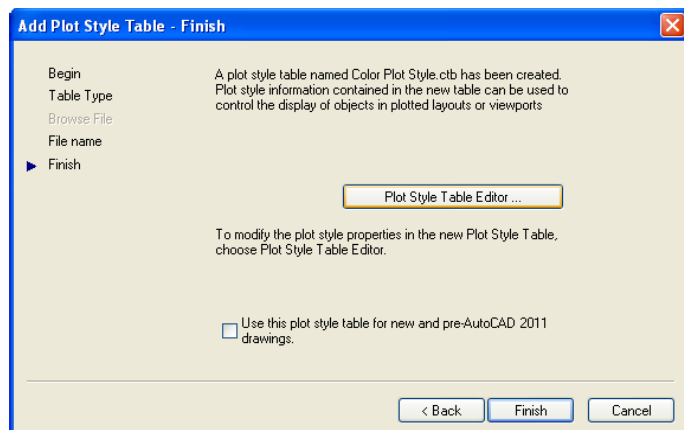
10. **Click**

Next>



11. **Choose**

“Plot Style Table Editor...”

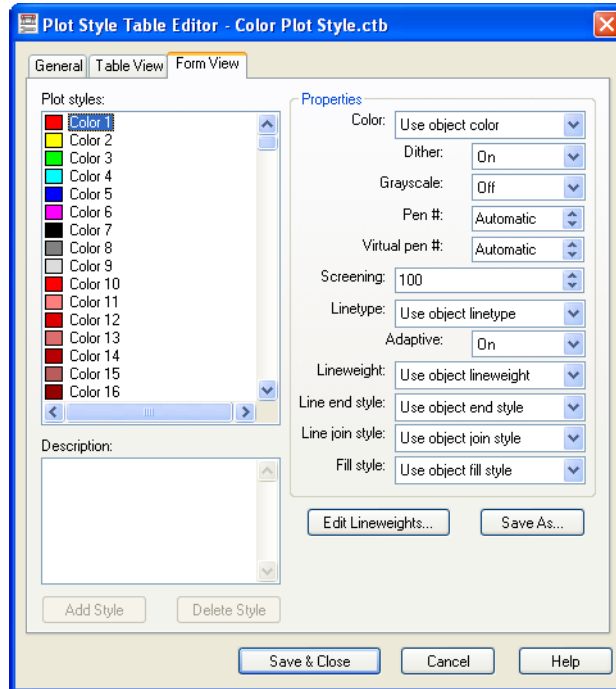


12. **Pick**

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.

13. **Choose** Save and Close

14. **Choose** Finish




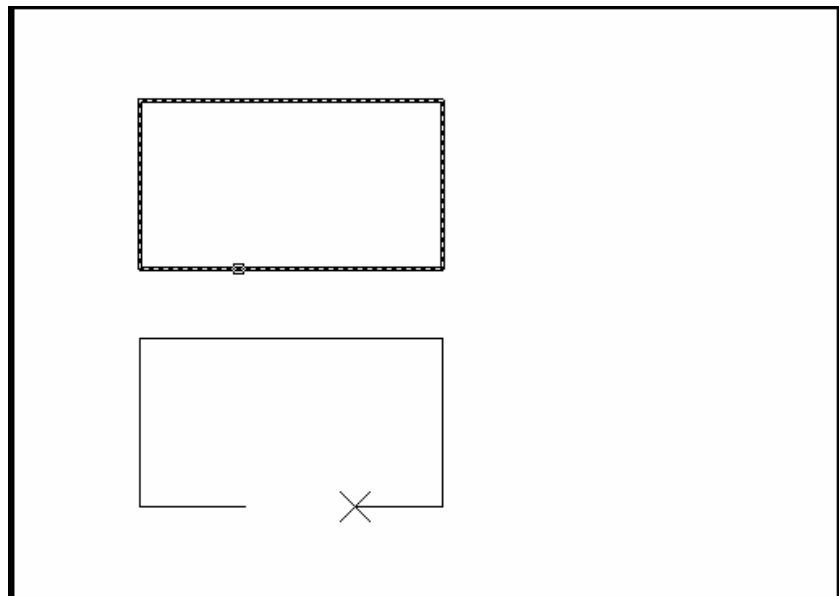
AutoCAD will save the file called COLORPLOTSTYLE.CBT

Chapter 13

More Edit Commands

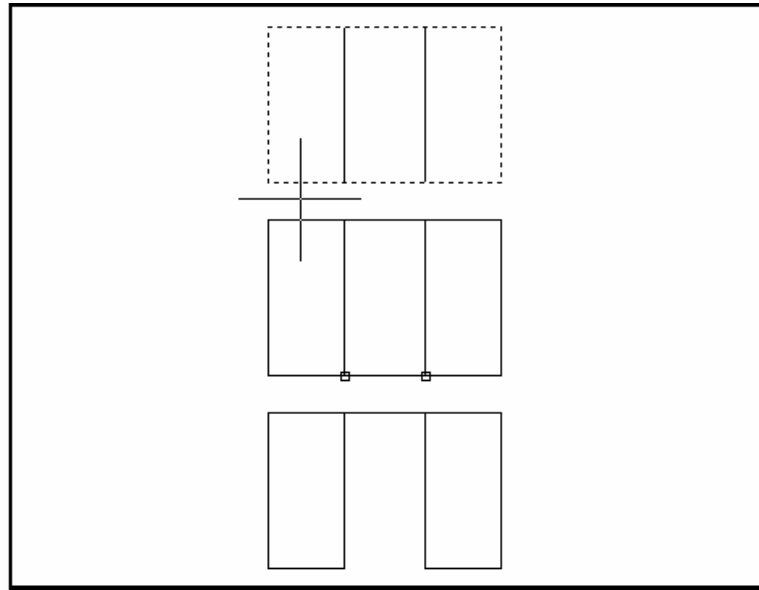
13.1 Break

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)
7. **Pick** The first break point on the object
Enter first point: (**point**)
8. **Pick** A second break point


**TIPS:**

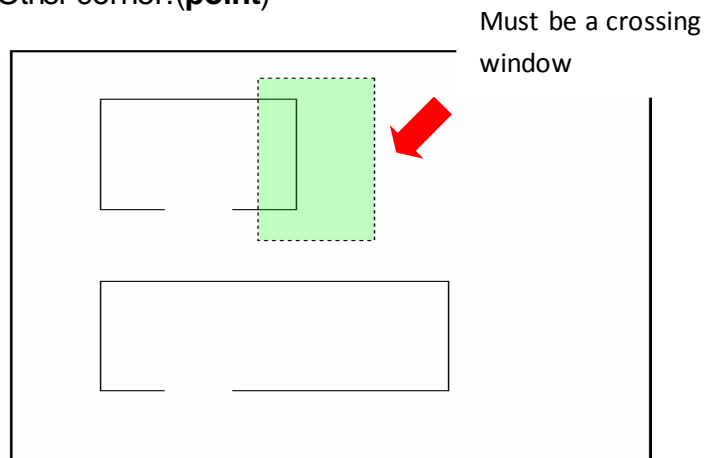
You can also type coordinates instead of picking a break point. Enter second point (or F for first point) @3'<0

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.

13.2 Stretch

1. **Choose** Modify, Stretch.
or
2. **Click** the Stretchicon. 
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)**




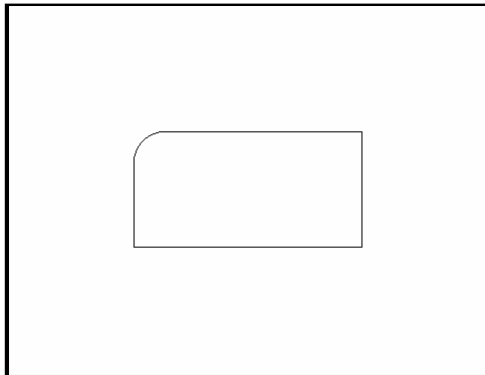
7. **Press** ENTER to accept objects to stretch.
8. **Pick** A base point to stretch from Base point: **(point)**
9. **Pick** A point to stretch to Newpoint: **(point)**
or
10. **Type** A distance to stretch. New point: **@1<0**

TIP:

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

13.3 Fillet


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 trimmode (trim cuts the fillet corner and no trim keeps the fillet corner)



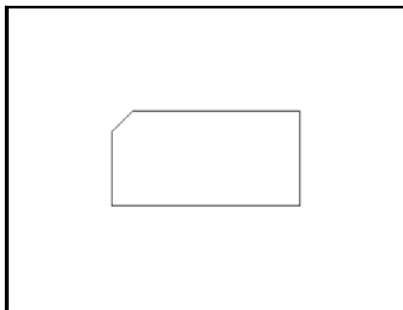
TIP:

- 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

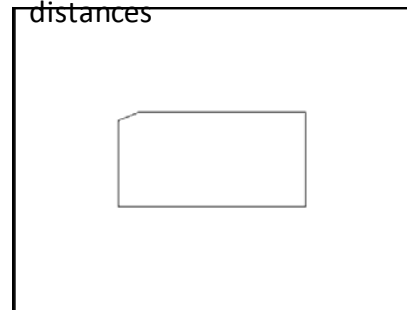
13.4 Chamfer

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 trim mode
 - M** Sets the method to distance or angle.

Chamfer with equal distances




Chamfer with different distances

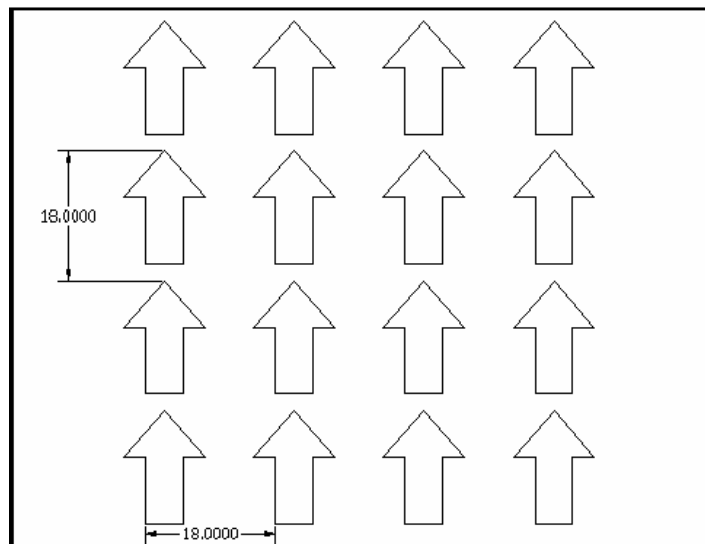


13.5 Array

Rectangular Array


To draw rectangular array:

1. **Choose** Modify, Array.
 or
 the Array icon. 
2. **Click** or
3. **Type** ARRAY at the command prompt.
 Command : **ARRAY**
4. **Pick** Objects to array. Select
 objects : (**select**)
5. **Type** R for a rectangular array.
 Enter array type [Rectangular/PAth/POLar] <Rectangular>: **R**
 Type = Rectangular Associative = Yes
 Select grip to edit array or [ASsociative/Base
 point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
6. **Type** **S** to change the spacing distance between each
 column and/or row.
7. **Type** **COL** to change the number of columns.
8. **Type** **R** to change the number of rows.



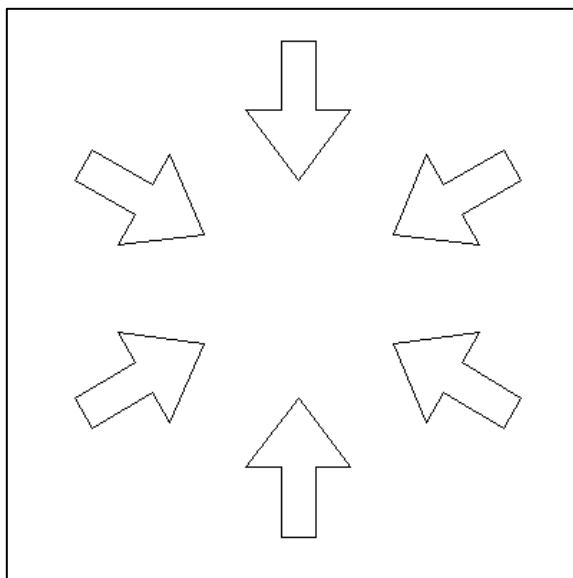
Polar Array

To draw a polar array:

1. **Choose** Modify, ARRAY.
2. **Click** **or**
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. Enter array type
[Rectangular/PAtch/POLar] <Rectangular>: PO
(R/P): **PO**
6. **Pick** A center point for the array. Center point of array:
pick point
7. **Type** one of the following options or press **ENTER**


Select grip to edit array or [ASsociative/Base
point/Items/Angle between/Fill
angle/ROws/Levels/ROtate items/eXit]:



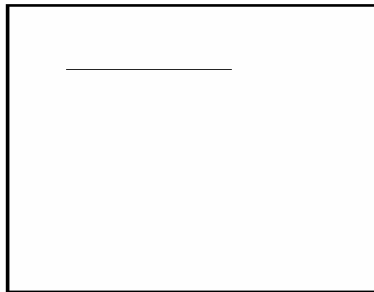
13.6 Lengthen

1. **Choose** Modify, LENGTHEN.

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

or
3. **Choose** the lengthen icon. 
4. **Select** an object to change or [Undo]: **pick object**
5. **Choose** one of the following lengthen options:
[Delta/Percent/Total/ Enter delta
 length or [Angle] <0.0000>:

Object before lengthen



Object after lengthen



Chapter 14

Advanced Display Commands

14.1 Transparent Commands

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**

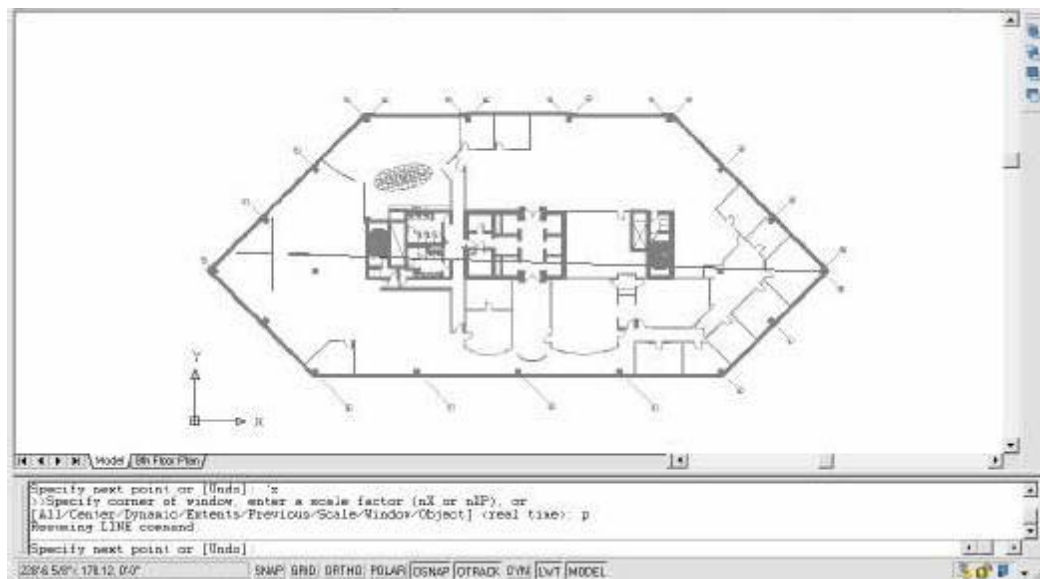
Specify first point: (**pick point**)

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**)



Resuming LINE command.

Specify next point or [Undo]: (**pick point**)

TIP:

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

14.2 Multiple Command

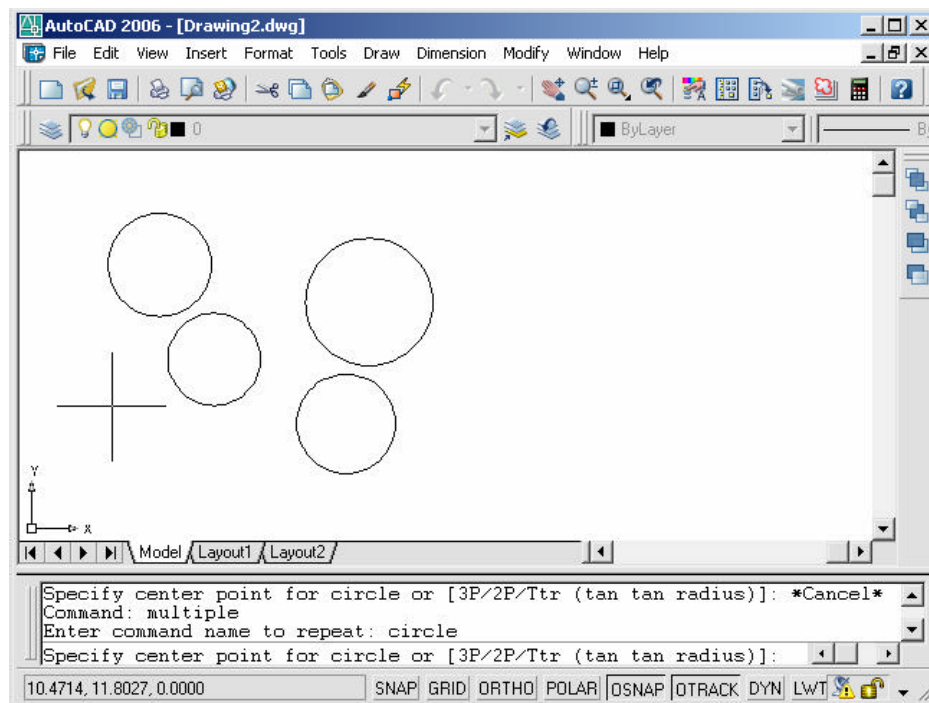
Multiple repeats the specified command until canceled

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**



14.3 Calculator (CAL Command)

Evaluates mathematical and 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

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

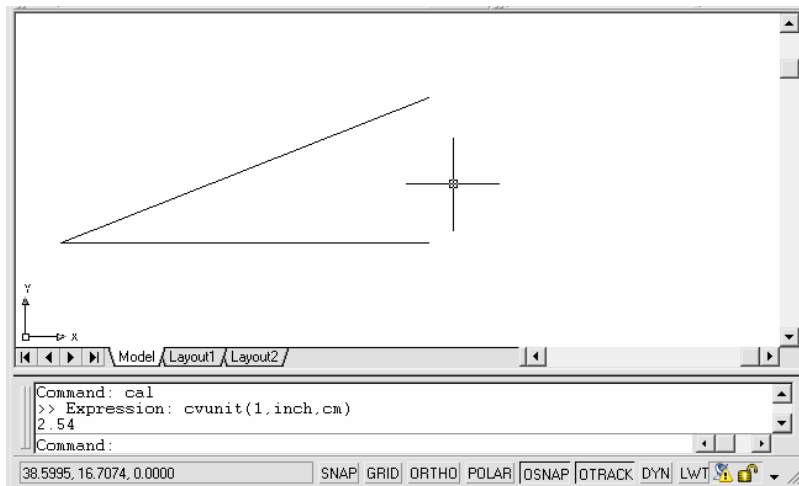
+, - Adds, subtracts vectors (points)

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

Convert units of measure

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

Initializing...>> Expression: **cvunit (1,inch,cm)**
2.54



Determine Angles

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


Initializing...>> Expression: **ang(end,end,end)**
45

Chapter 15

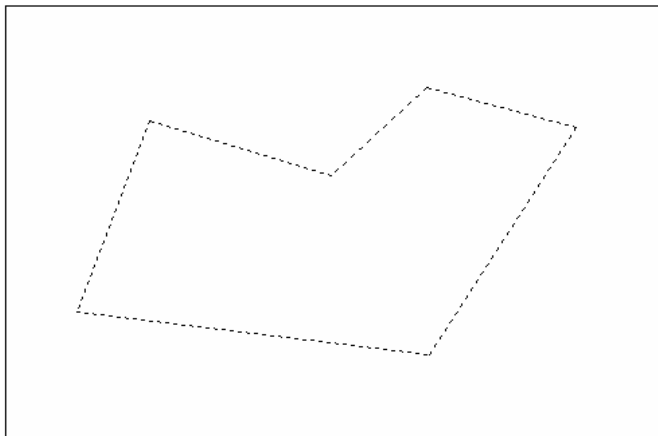
Polylines

15.1 Pline Command

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

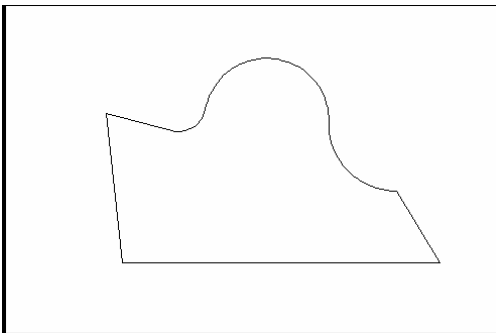
Polyline as one segment



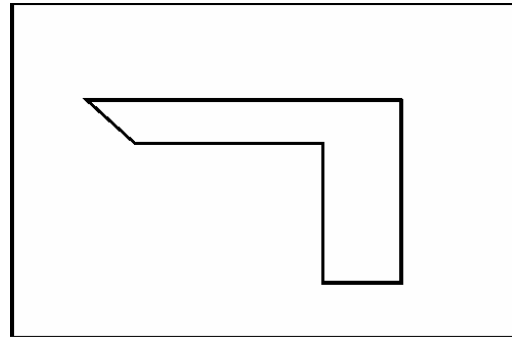
PLINE options:

Arc	Toggles to arc mode and you receive the following:Angle/CEnter/CLose/Direction/ Halfwidth/Line/ Radius /SecondPt/Undo/Width/<endpt of arc>:
Close	Closes a polyline as it does in the linecommand.
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.

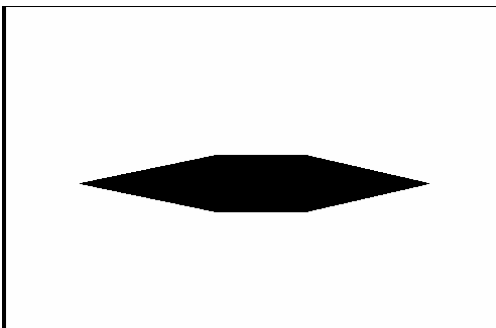
Polyline with arcs



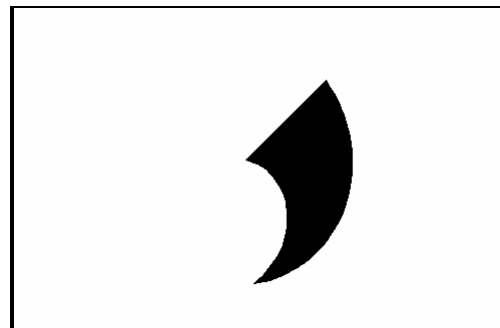
Polyline with width .125




Tapered width polyline



Tapered width arc polyline



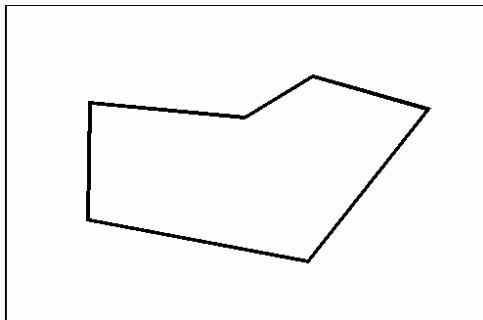
15.2 Editing Polylines

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

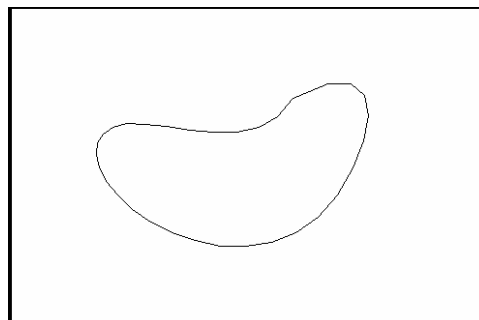
PEDIT options:

- 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. |

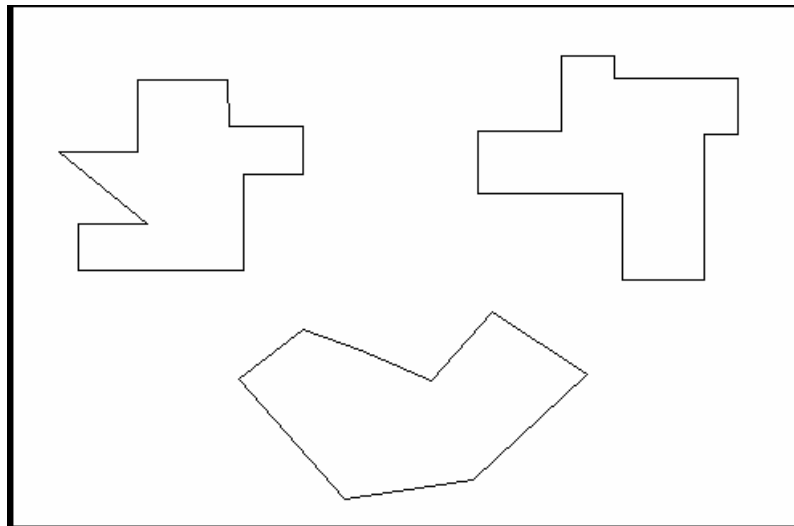
Edit Vertex Options

- | | |
|-------------------|---|
| 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. |


15.3 Editing Multiple Polylines

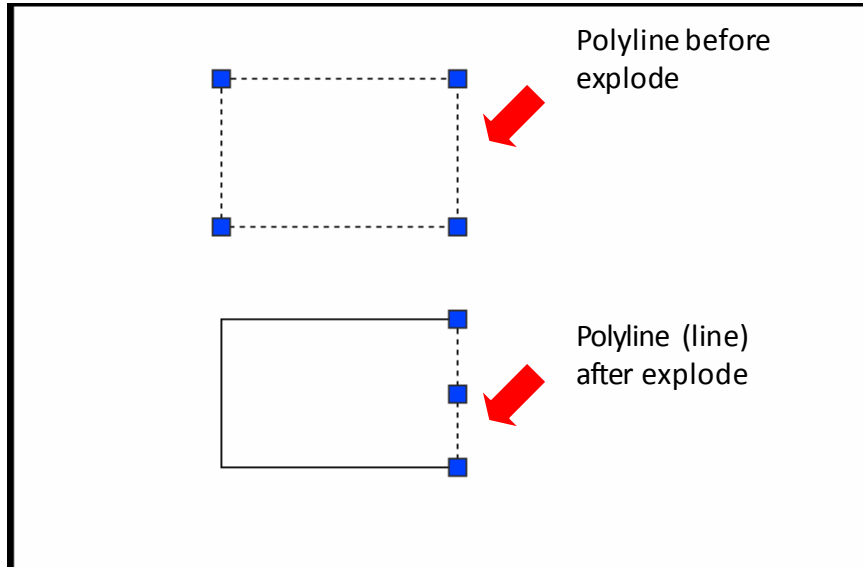
1. **Type** the PEDIT at the command prompt. Command:
PEDIT

Select polyline or [Multiple]:**M**
Pick multiple polylines to edit.



15.4 Explode Command

1. **Choose** Modify, Explode.
or
Pick the Explode icon. 
- Type** EXPLODE at the command prompt.
Command: **EXPLODE**
or
2. **Pick** The object to explode. Select objects: **(pick)**



15.5 Turning Lines into Polylines

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

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

Object selected is not a polyline

Do you want to turn it into one? <Y> **Y**
Enter an option [Close/Join/Width/Edit
vertex/Fit/Spline/Decurve/Ltype
gen/Undo]: **j**


TIP:

- Lines and Arcs must have a common end point to join them together.

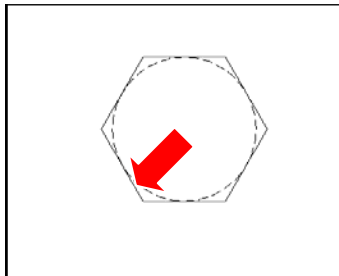
Chapter 16

More Draw Commands

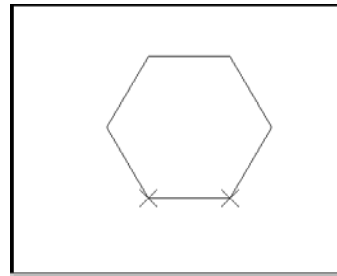
16.1 Polygon

1. **Choose** Draw, Polygon.
or
2. **Click** the Polygon icon. 
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):

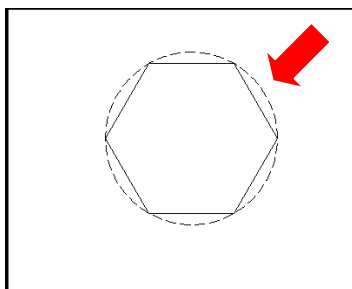
Polygon Inscribed in an imaginary circle




Polygon drawn with an edge

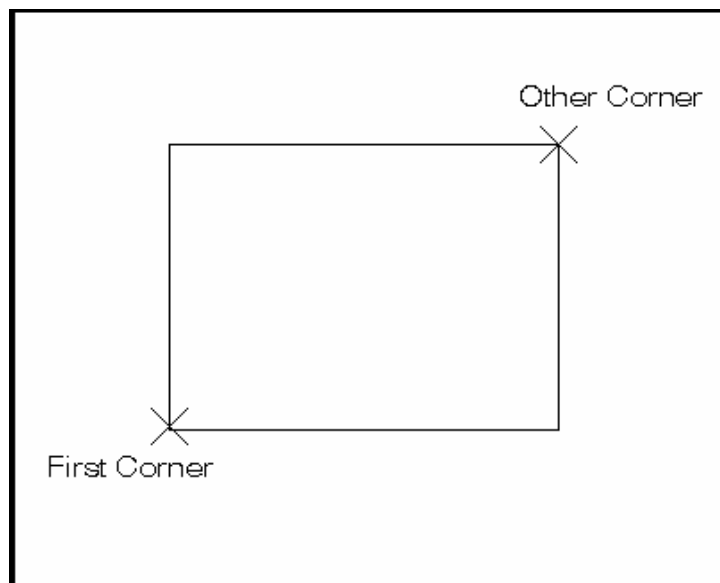


Polygon circumscribed around an imaginary circle



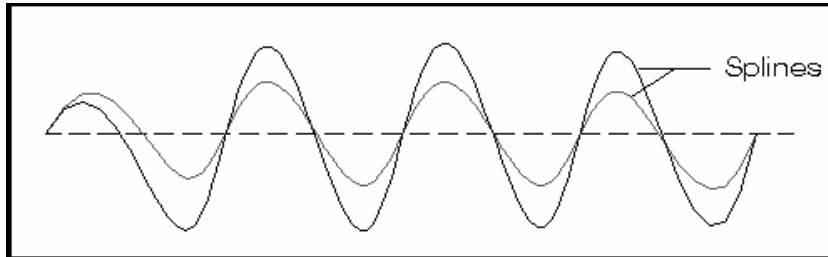
16.2 Rectangle

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**).



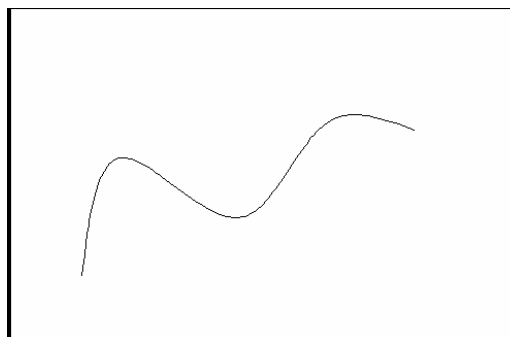
16.3 Spline

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**
Specify first point or
[Method/Knots/Object]:

4. **Pick** A start point for the spline
Enter next point or [end
Tangency/toLerance/Undo/Close]:
(pick point)
5. **Pick** Points until done drawing splines
(pick points)
6. **Press** Enter or close to complete the spline



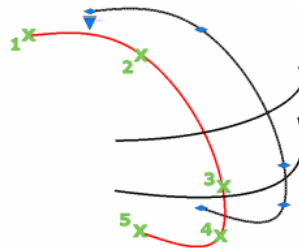
Spline options:

Method Controls whether the spline is created with fit points or control vertices. The options differ depending on whether you select Options for Splines with Fit Points or Options for Splines with Control Vertices.

Fit

Draw a spline by specifying fit points.

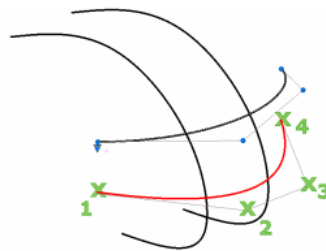
Changing the Method updates the SPLMETHOD system variable.



Control Points

Draw a spline by specifying control vertices.

This is the preferred method if you are creating geometry to use with 3D NURBS surfaces. Changing the Method updates the SPLMETHOD system variable.



Knot Specifies the knot parameterization which affects the shape of the curve as it passes through a fit point. (SPLKNOTS system variable)


- **Chord** - Numbers edit points with decimal values representing their location on the curve.
- **Square Root** - Numbers edit points based on the square root of the chordal length between consecutive knots.
- **Uniform** - Numbers edit points with consecutive integers.

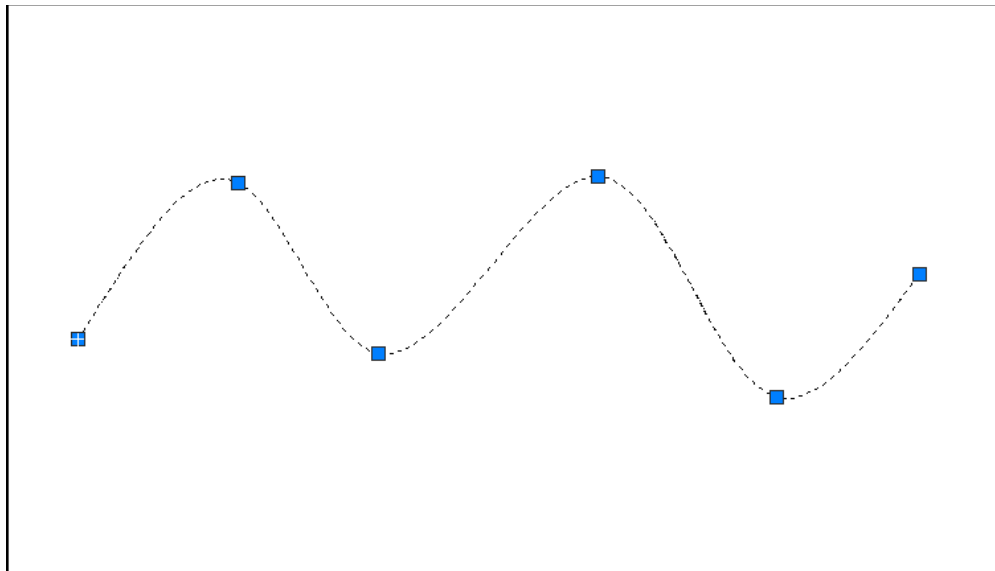
Object Converts 2D or 3D quadratic or cubic spline-fit polylines to equivalent splines and (depending on the setting of the DELOBJ system variable) deletes the polylines.

TIP:

- Refer to AutoCAD online help topic for more information on spline options

16.4 Editing Splines

1. **Choose** Modify, Object, Spline.
or
2. **Choose** the spline edit icon. 
or
3. **Type** SPLINEDIT at the command prompt.
Command: **SPLINEDIT**

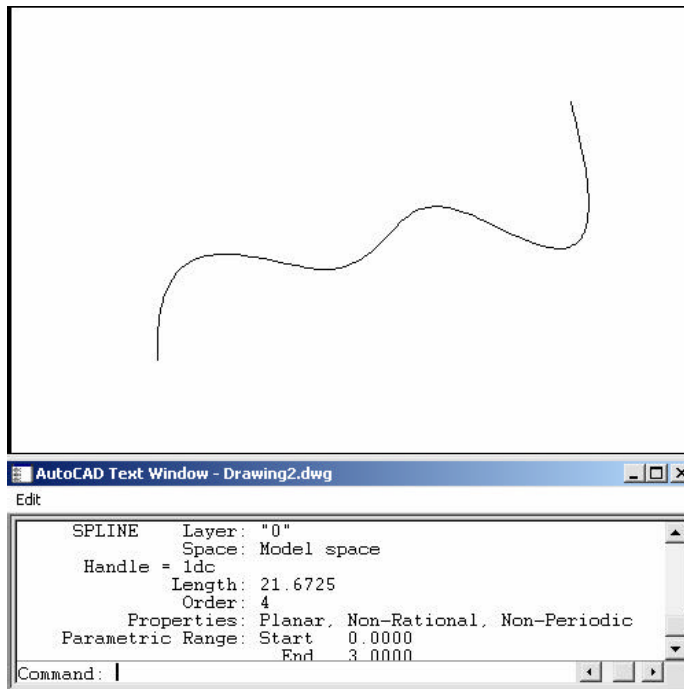


TIP:

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

16.5 Covert PLINE to Spline

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




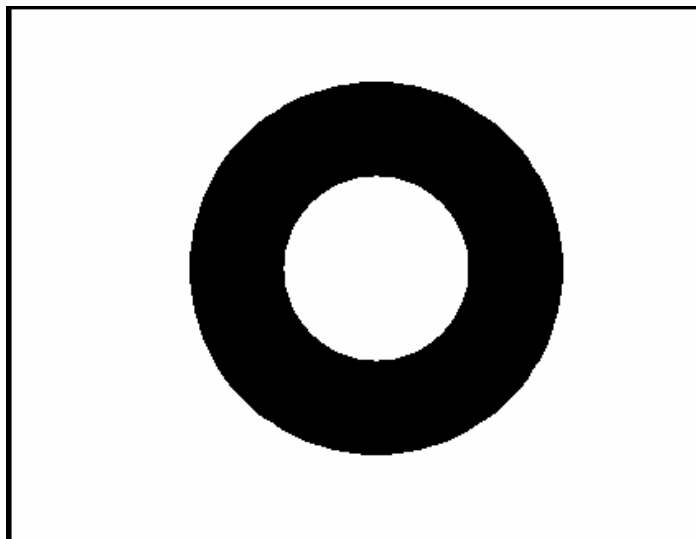
TIP:

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

16.6 Donut


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**
or
3. **Choose** the donut icon. 
4. **Type** A value for the inside diameter. Inside diameter <last>: **.5**
5. **Type** A value for the outside diameter. Outside diameter <last>: **1**
6. **Pick** A point for the center of the donut. Center of doughnut: **(point)**



16.7 Ellipse

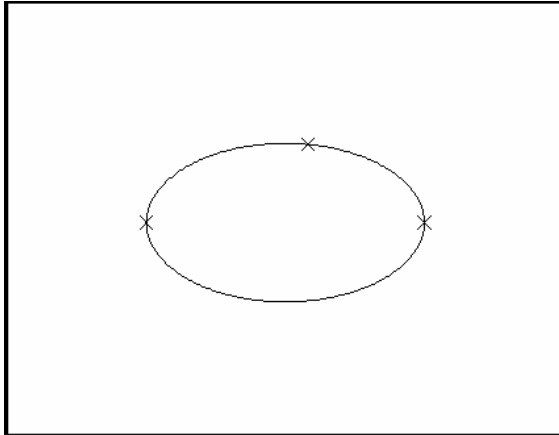
Creates an ellipse or an elliptical arc.

1. **Choose** Draw, Ellipse.
or
2. **Choose** the 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>:

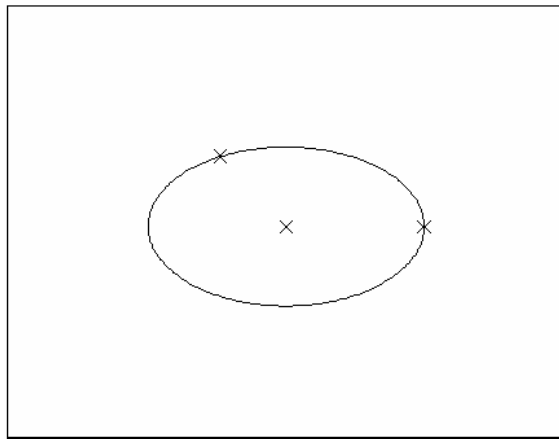
Ellipse options:

- | | |
|-------------------------|--|
| 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. |

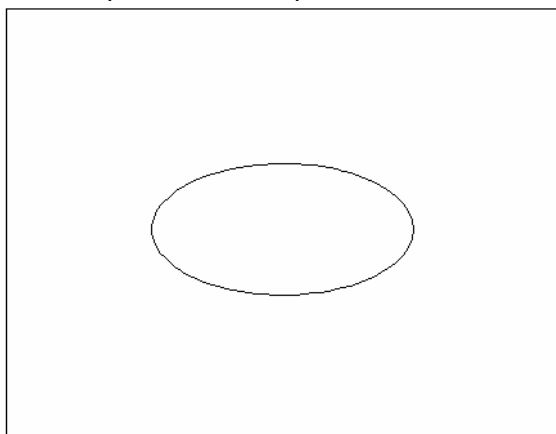
*ELLIPSE,
Axis, Eccentricity (Axis Endpoint, Axis Endpoint, Other Axis Distance)*



*ELLIPSE,
Center, Axis, Axis*



*ELLIPSE,
Axis Endpoint, Axis Endpoint, Rotation=60*



16.8 Multilines

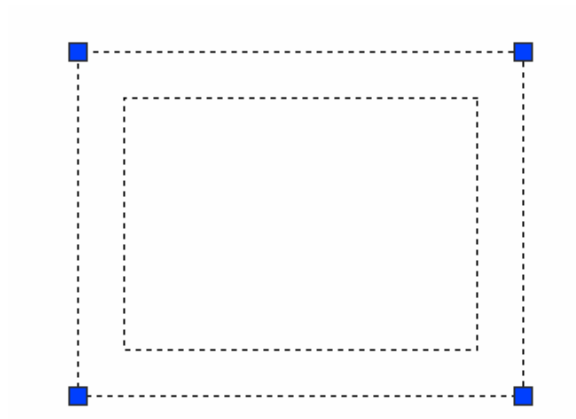
MLINE Command

Creates multiple parallel lines.

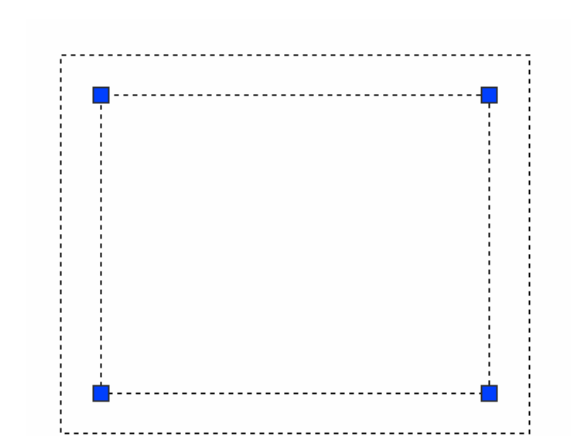
1. **Choose** Draw, Multiline.
or
2. **Type** MLINE at the command prompt.
Command: **MLINE**
3. **Pick** A point to start the multiline. Justification/Scale/
STyle/<From point>: pick point
4. **Pick** A second point to continue the multiline.
<To point>: **pick point**
5. **Pick** The next point to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

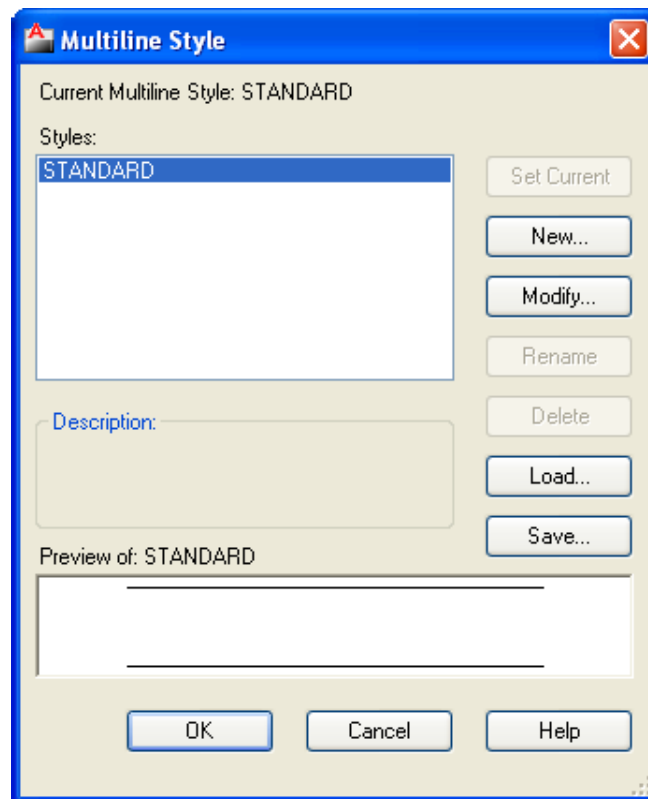


Zero Justification



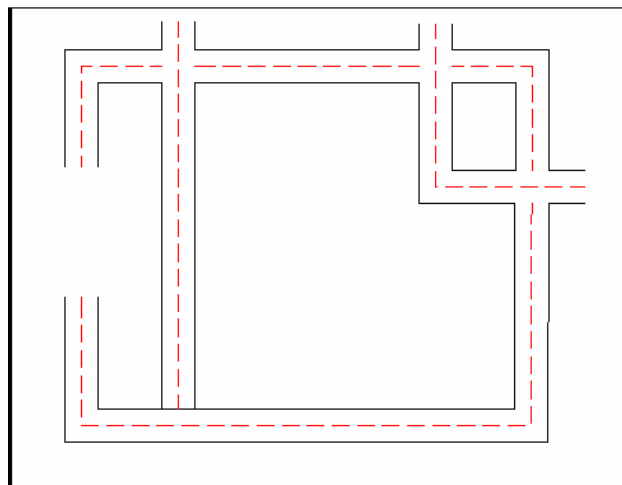
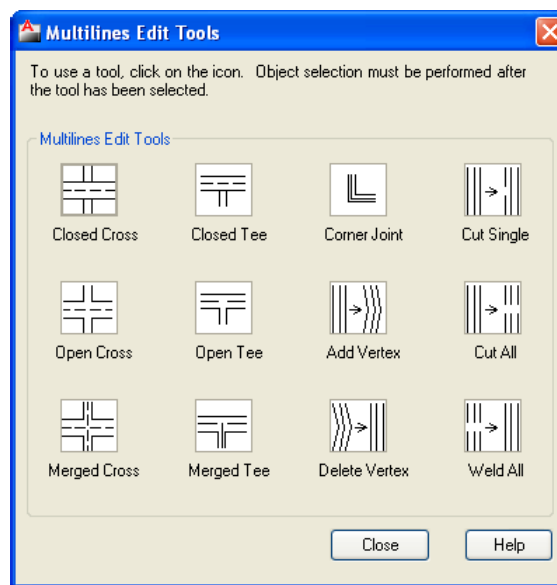
16.9 Multiline Styles

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** Element Properties to change the appearance of
 the multilines.
5. **Choose** ADD to create the new multiline.



16.10 Editing Multilines

1. **Choose** Modify, Multiline...
- or
2. **Type** MLEDIT at the command prompt
Command: **MLEDIT**
3. **Choose** From one of the mledit options:




16.11 Construction Line

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**

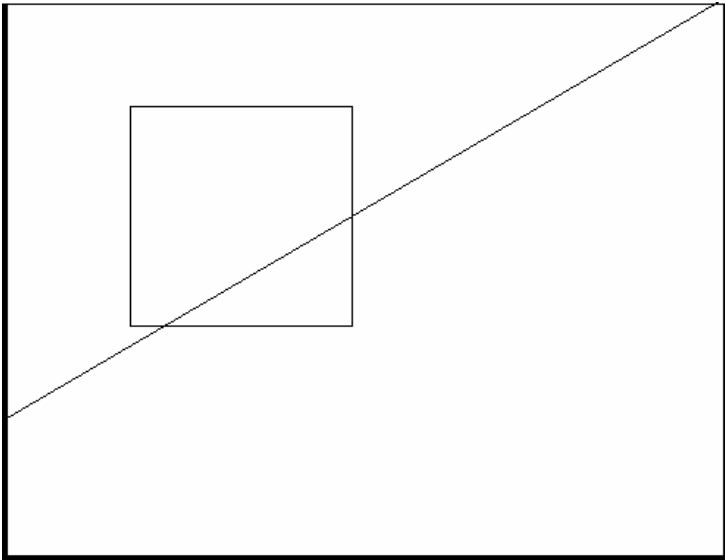
Creates a horizontal xline passing through a specified point.
- VER**

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.



16.12 Ray Command

Creates an infinite line in one direction.

1. **Choose** Draw, Ray
or
2. **Choose** the Ray icon. 
or
3. **Type** RAY at the command prompt.
Command: **RAY**
Specify a point : **(pick through point)**

Chapter 17

Crosshatching

17.1 HATCH Command

Choose Draw, Hatch...

or

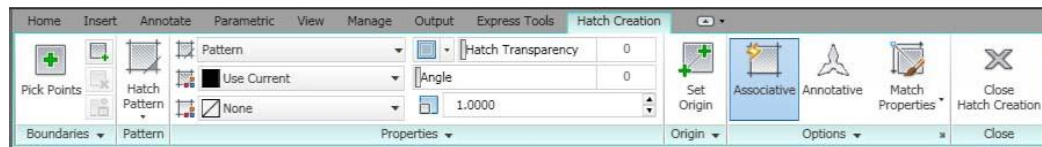
Click the Hatch icon 

or

Type HATCH at the command prompt

Command: **HATCH**

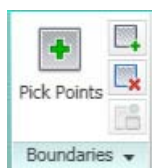
AutoCAD's ribbon changes to show hatch options



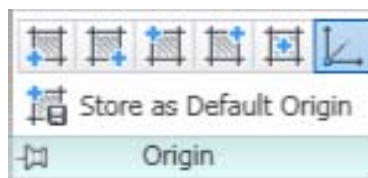
Hatch Patterns



Boundaries



Origin



Hatch Options

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.

17.2 HATCHEDIT

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

or

Click

the Hatch Edit icon.



or

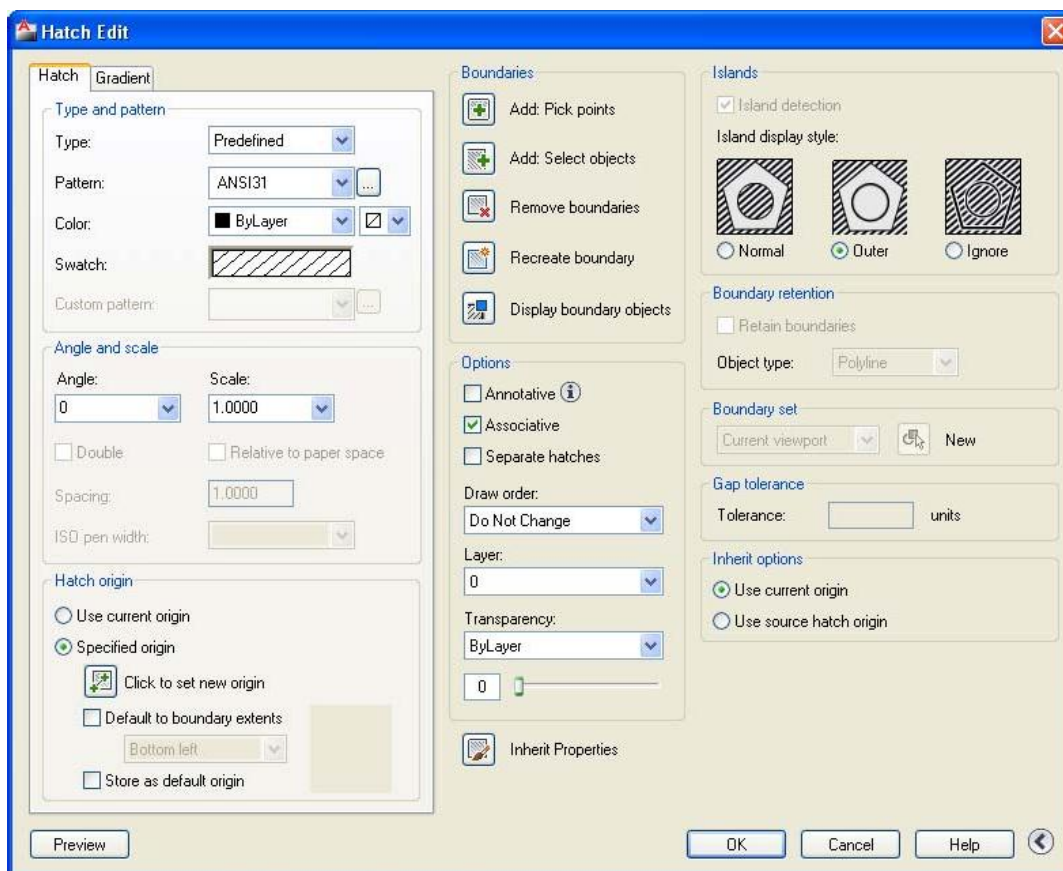
Type

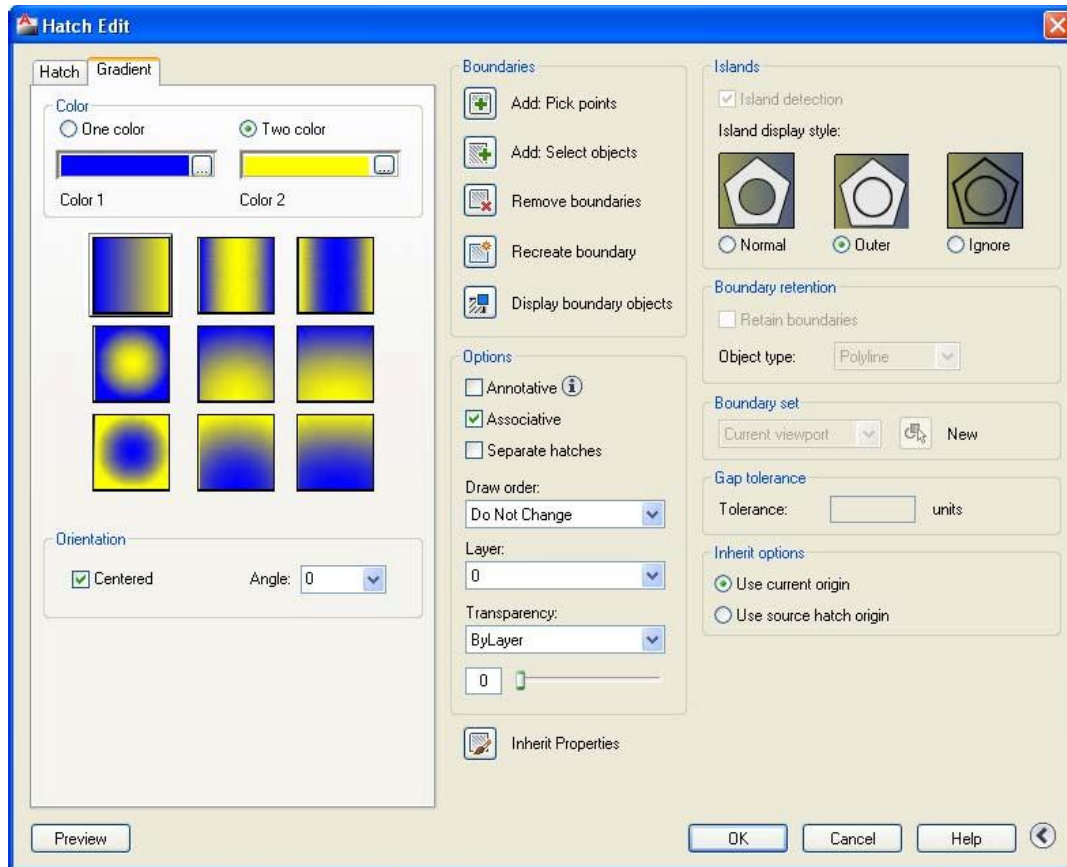
HATCHEDIT at the command prompt.

Command :**HATCHEDIT**

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

3. **Pick** The OK button.





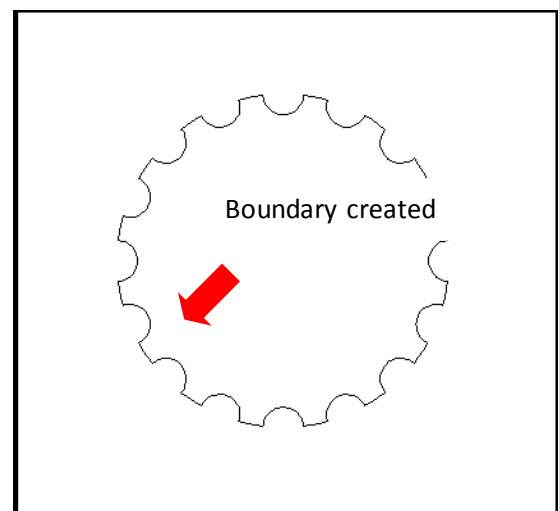
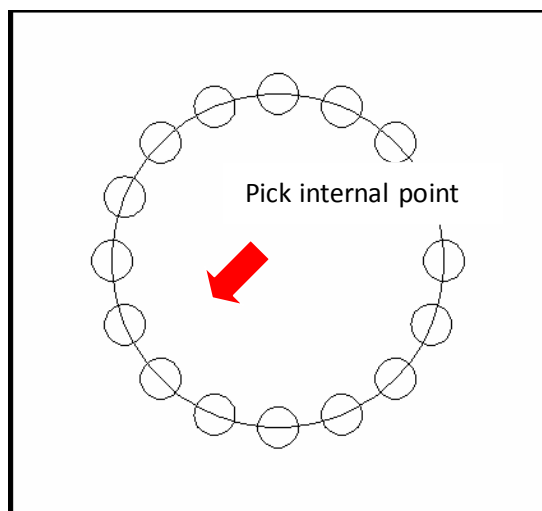
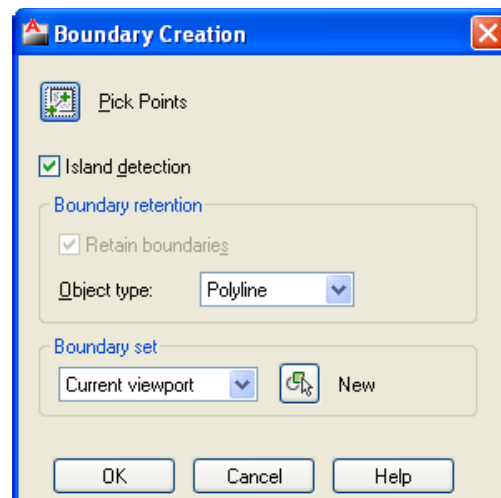
Chapter 18

Regions and Boundaries

18.1 Boundary Command

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**



18.2 Region Command

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**

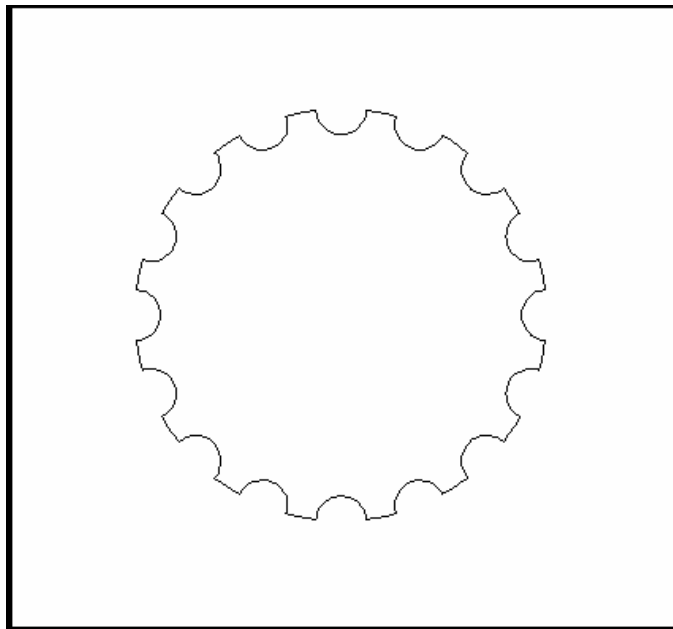
Select objects: **(pick boundary)**

Select objects: 1 found

1 loop extracted.

1 Region created.

Object created as a region



18.3 Mass Properties

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]

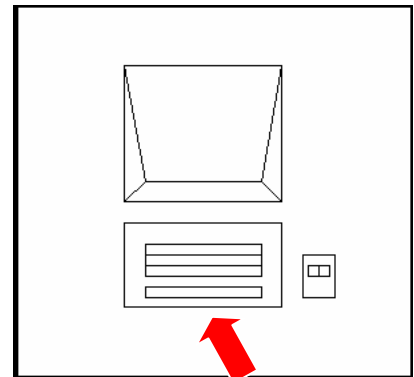
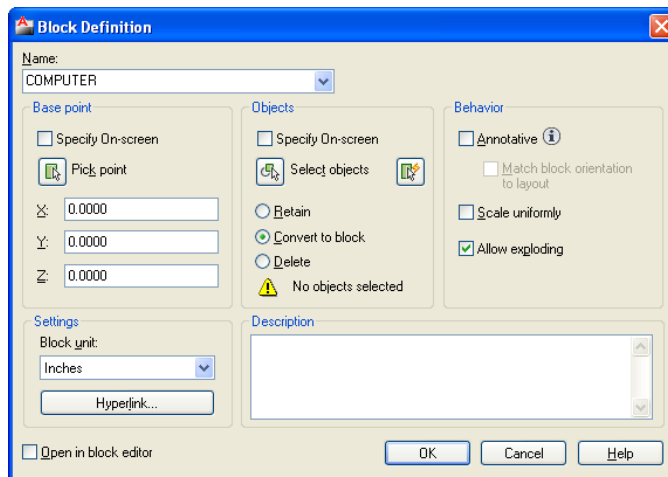
Write analysis to a file? [Yes/No] <N>:

Chapter 19

Blocks and Attributes

19.1 Creating Local Blocks (BMAKE)

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.




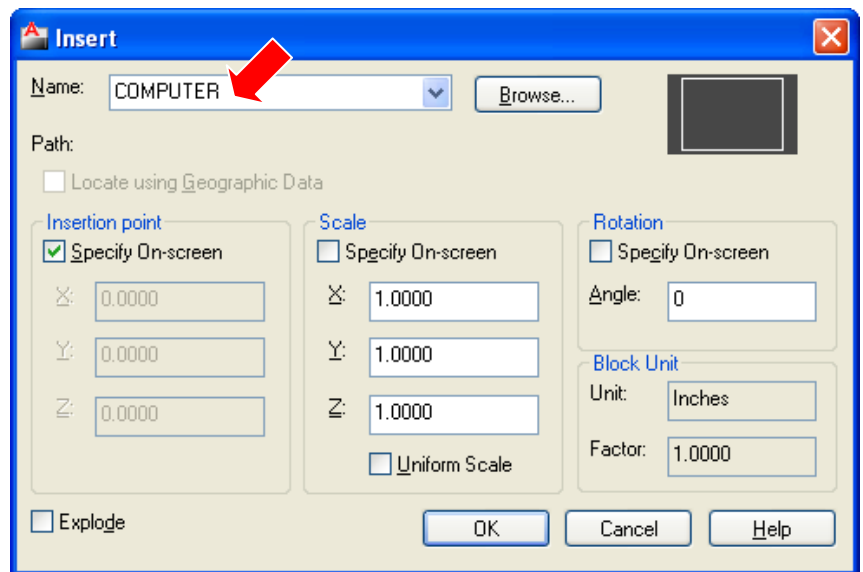
Pick insertion point

TIP:

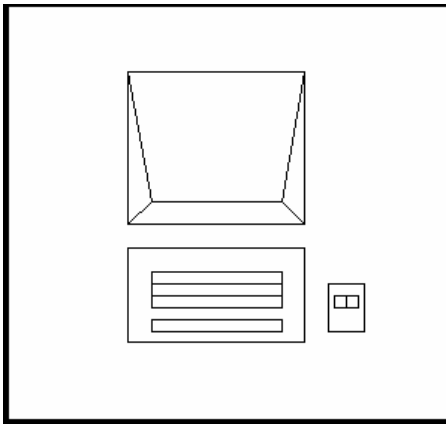
- You cannot use DIRECT, LIGHT, AVE_RENDER, RM_SDB, SH_SPOT, and OVERHEAD as valid block names. These are used in lights and rendering objects.

19.2 Inserting Blocks

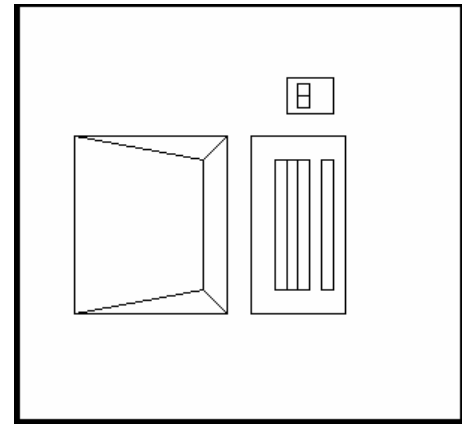
1. **Choose** Insert, InsertBlock
 or
Click the Insert icon from the INSERT toolbar. 
 or
Type INSERT at the command prompt.
 Command: **INSERT**
2. **Choose** the name to insert a local block and Browse...to insert a Wblock.



3. **Choose** the insertion point, scale, and rotation of the block.



*Block Inserted with
zero degree rotation angle*



*Block Inserted with a 90 degree a
rotation angle*

Typing Insert (-INSERT)

By typing a dash (-) before the insert command you will suppress the insert dialog box and can type each of the prompts at the command line. This is useful if you wish to insert a block from a macro or toolbar.

1. Type INSERT at the command prompt.
Command: **-INSERT**
2. Type the Block name to insert and follow the remaining insert prompts..

19.3 Control the Color and Linetype of Blocks

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

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`.

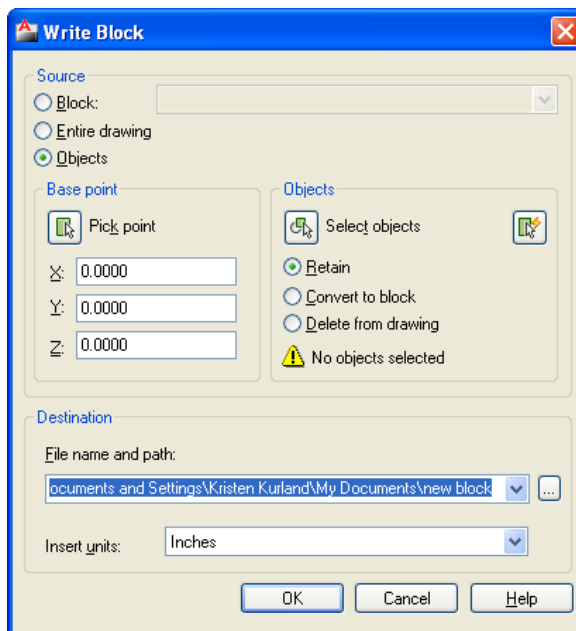
19.4 Wblock Command

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

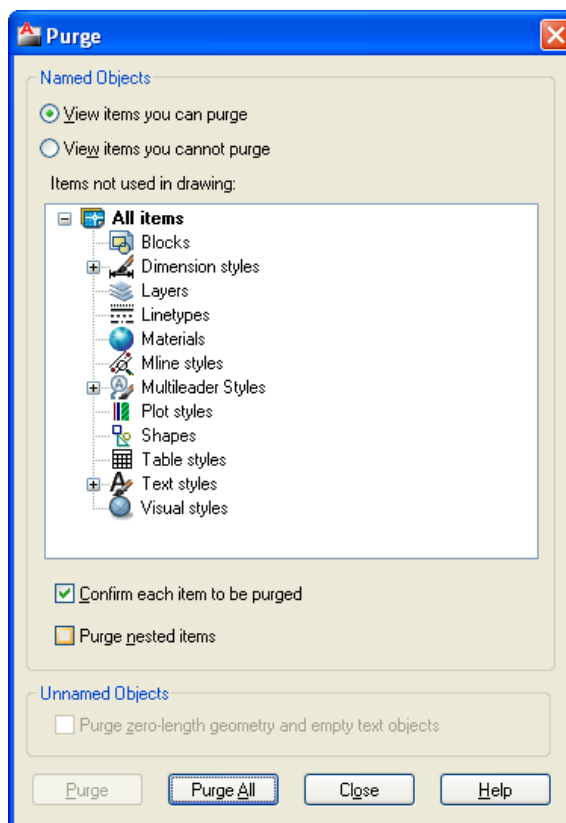
 Insertion base point: **pick a point**
6. **Pick** Objects to create the block. Select
 objects: **pick objects**
7. **Press** ENTER to end the selection set.



19.5 Purge

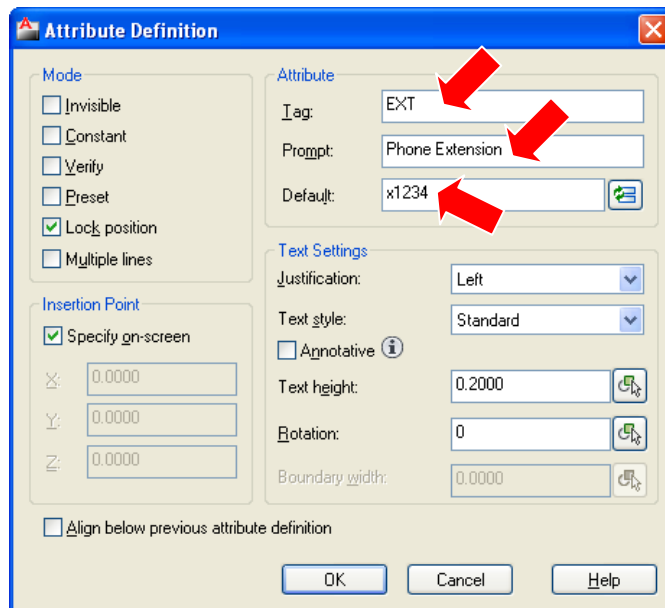
The purge command deletes unused objects in an AutoCAD drawing.

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/STypes/Mlinestyles/All:



19.6 Defining Attributes

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. |

19.7 Editing Attributes

1. **Choose** Modify, Object, Attribute, Single...
or
Click the Edit Attribute icon from the Modify II Toolbar.
or
Type ATTEDIT at the command prompt
Command: **ATTEDIT**
2. **Pick** The block to edit

Select block: **pick block with attributes**



Edit Attributes

Block name: PHONE

Purchase price: \$100.00

Manufacturer: ATT

Employee Name: John Doe

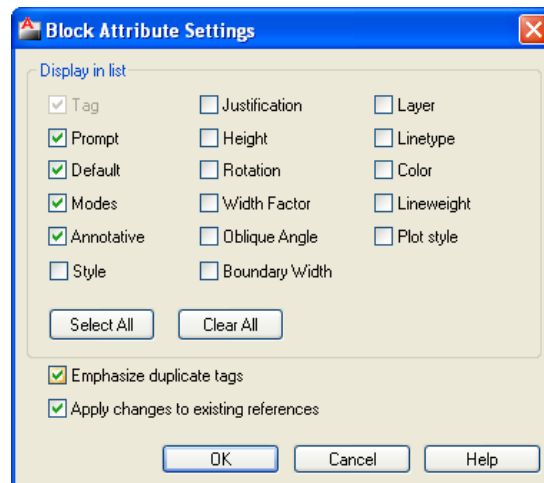
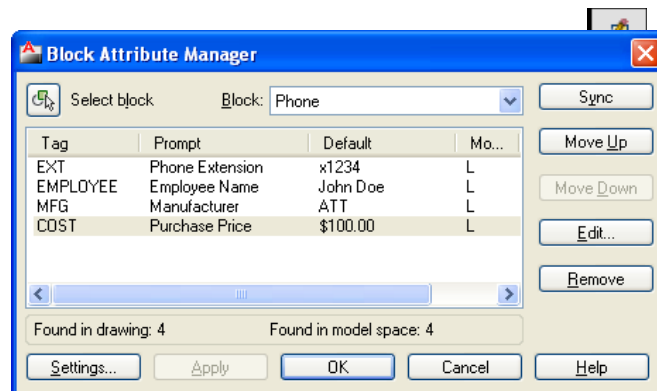
Phone Extension: x1234

OK Cancel Previous Next Help

19.8 Block Attribute Manager


Manages the attribute definitions for blocks in the current drawing.

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



19.9 Synchronize Attributes

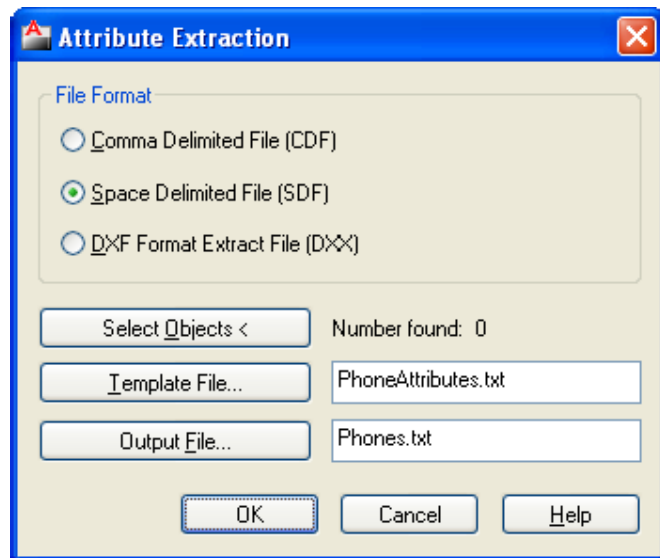
Updates all instances of a specified block with the current attributes defined for the block

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 a option [?/Name/Select] <Select>:
ENTER and select a block with attributes.
5. **Press** ATTSYNC block computer? [Yes/No]<Yes>:
ENTER to synchronies. ATTSYNC complete.

19.10 Attribute Extraction

Enables you to extract value of an attribute in a drawing to an external file.

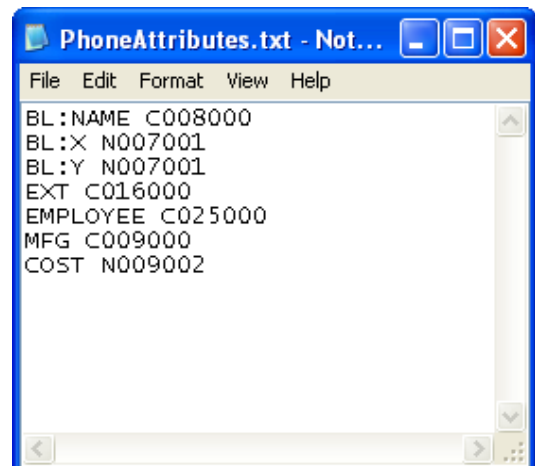
1. **Type** ATTEXT at the command prompt.
Command: **ATTEXT**
2. **Choose** Select objects and choose blocks with attributes defined.
3. **Choose** A template file.



4. **Click** OK.

TIP:

- The following is an example of an attribute template file. Be sure to use spaces and not the
- TAB key when entering fields and values.

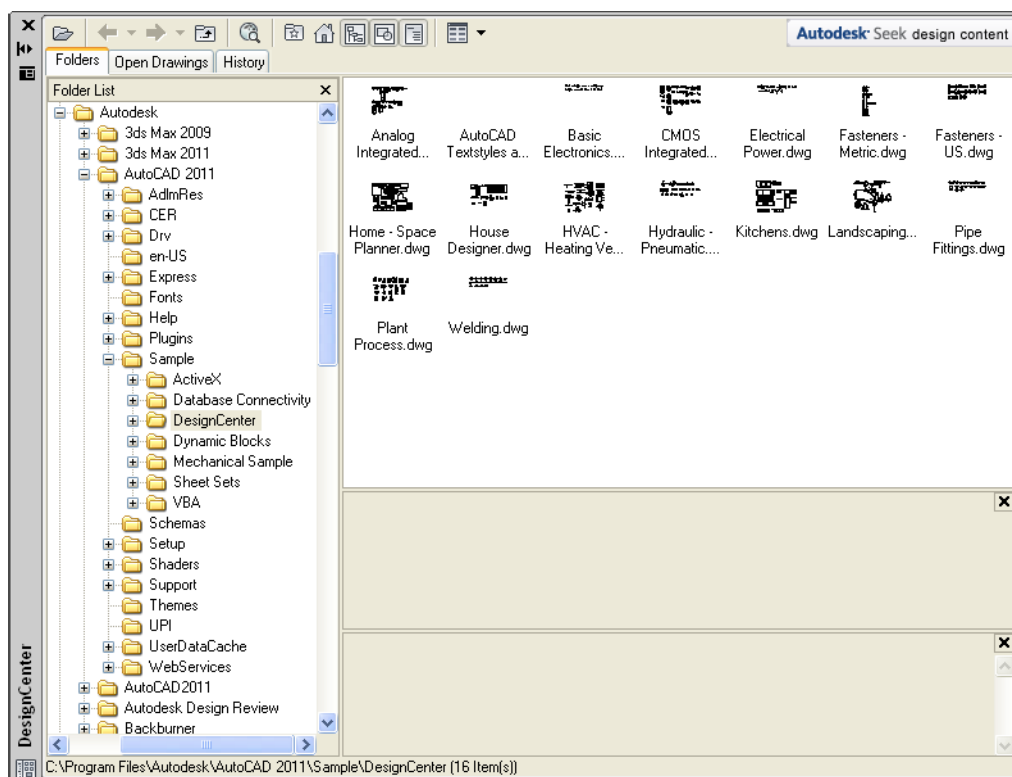


Chapter 20
Design Center and Tool
Palettes

20.1 Design Center Overview

The AutoCAD Design Center finds and transfers blocks, text styles, layers, dimension styles, etc from drawings, WEB

1. **Choose** Tools, AutoCAD Design Center.
or
2. **Press** **CTRL+2** on the keyboard.
or
3. **Type** **ADCENTER** at the command prompt.
Command: **ADCENTER**



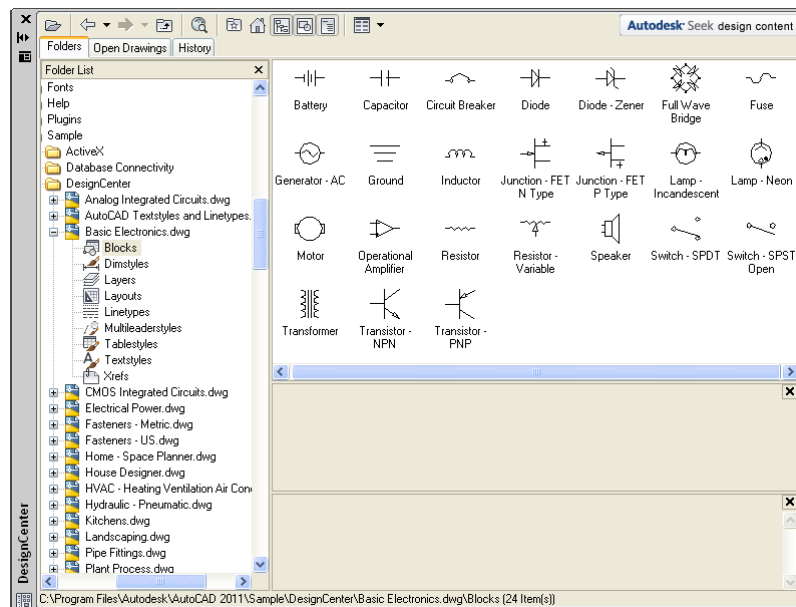
20.2 Design Center Blocks

1. **Choose** the blocks icon from one of the Design Center menus.



Blocks

2. **Drag** and drop a block from the Design Center into a drawing.

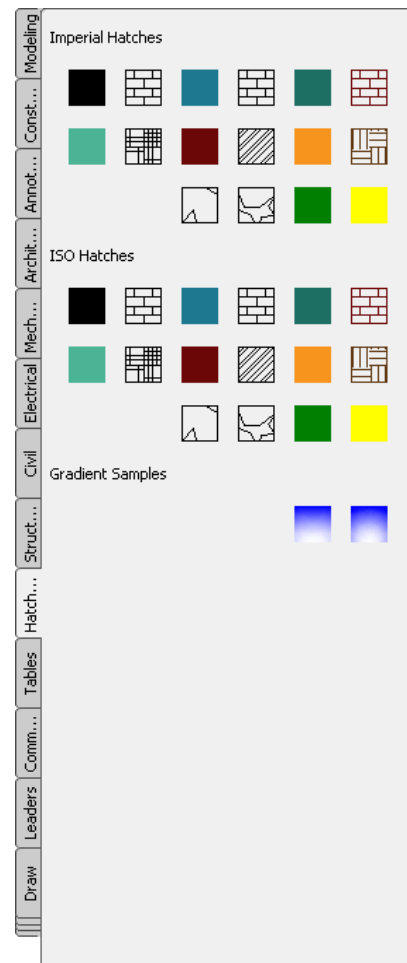
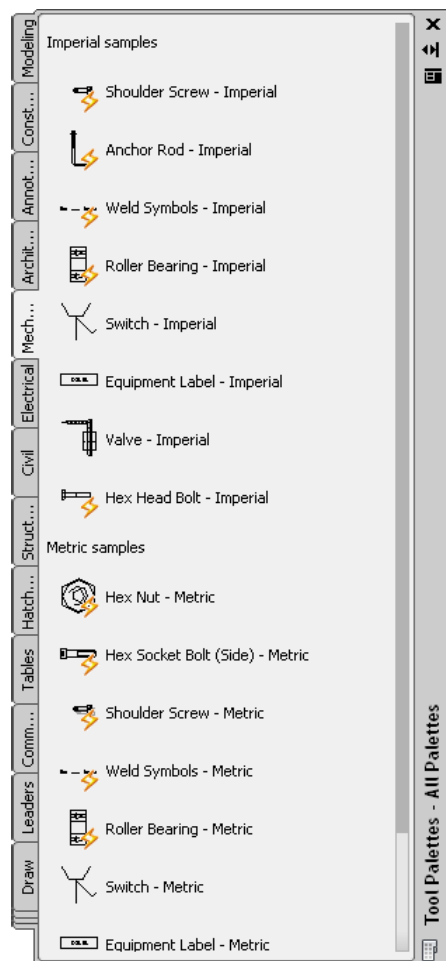


TIP:

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

20.3 Tool Palettes

1. **Choose** Tool, Tool Palettes Window
Or
2. **Press** CTRL +3 on the keyboard.



Chapter 21

Point, Divide, Measure

21.1 Point Styles

Changes the appearance of points and point sizes.

1. **Choose**

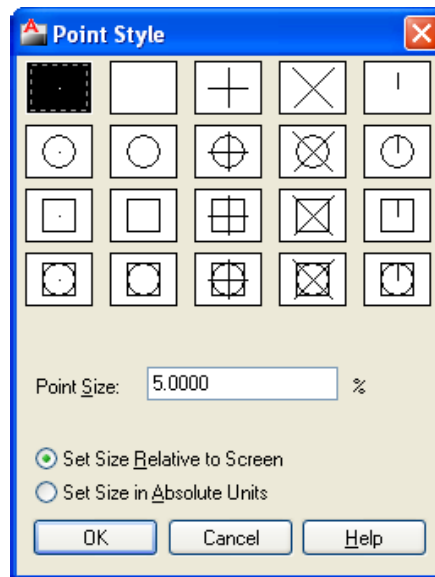
Format, Point Style...

or

Type

DDPTYPE at the command prompt.

Command : **DDPTYPE**



21.2 Point Command

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

or

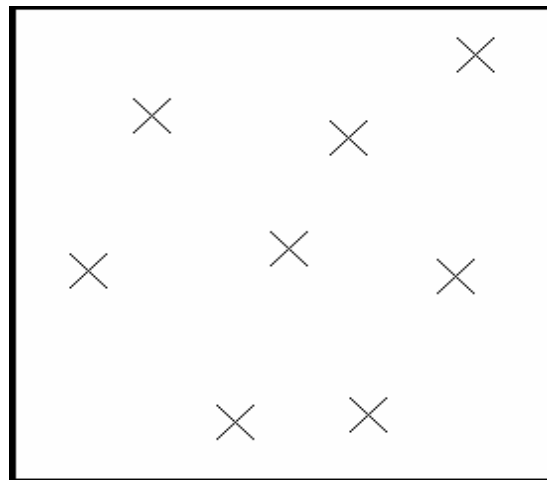
2. **Click** the Point icon.

or

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

Command: **POINT**

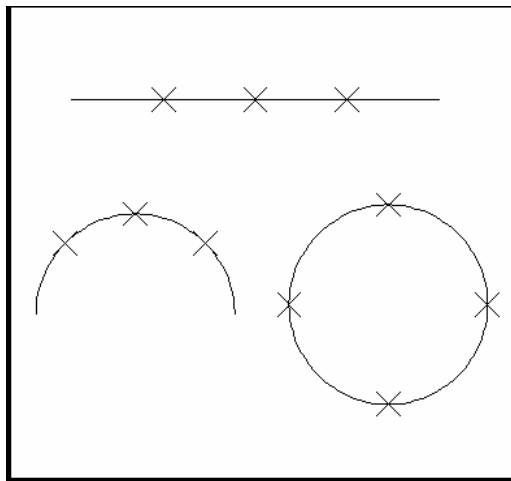
4. **Pick** A point on the drawing. Point (**point**)



21.3 Divide

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

Objects divided using points



TIP:

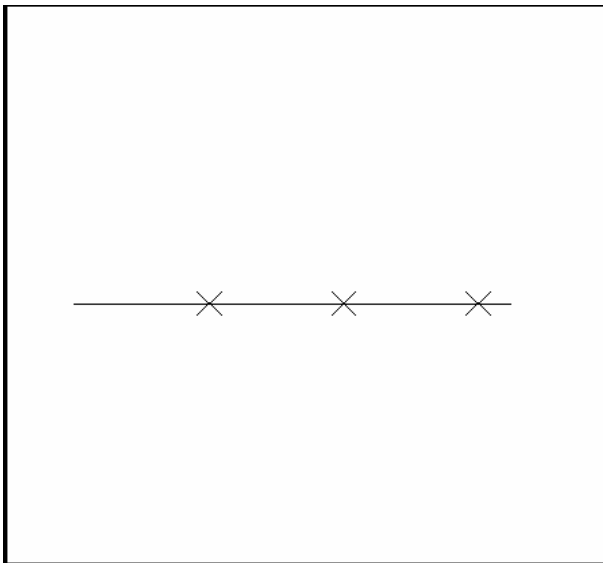
- Block symbols can be used in place of a point. The Block must currently be defined within the drawing. If you answer yes to the Align block? prompt, the Block will be rotated round its insertion point so that it is drawn tangent to the object being divided.

21.4 Measure

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

22.1 Grips Overview

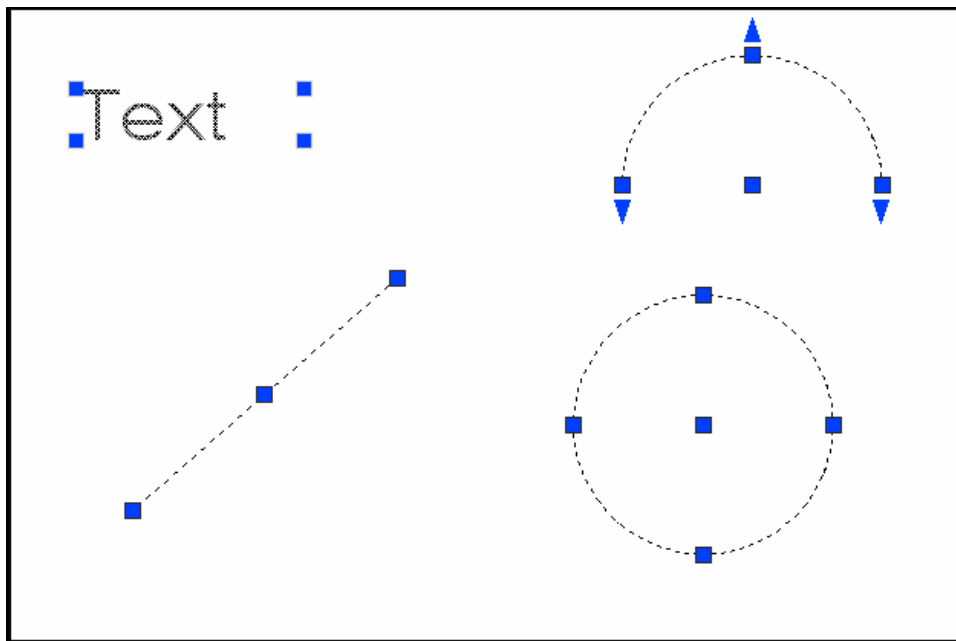
Entity Grips

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

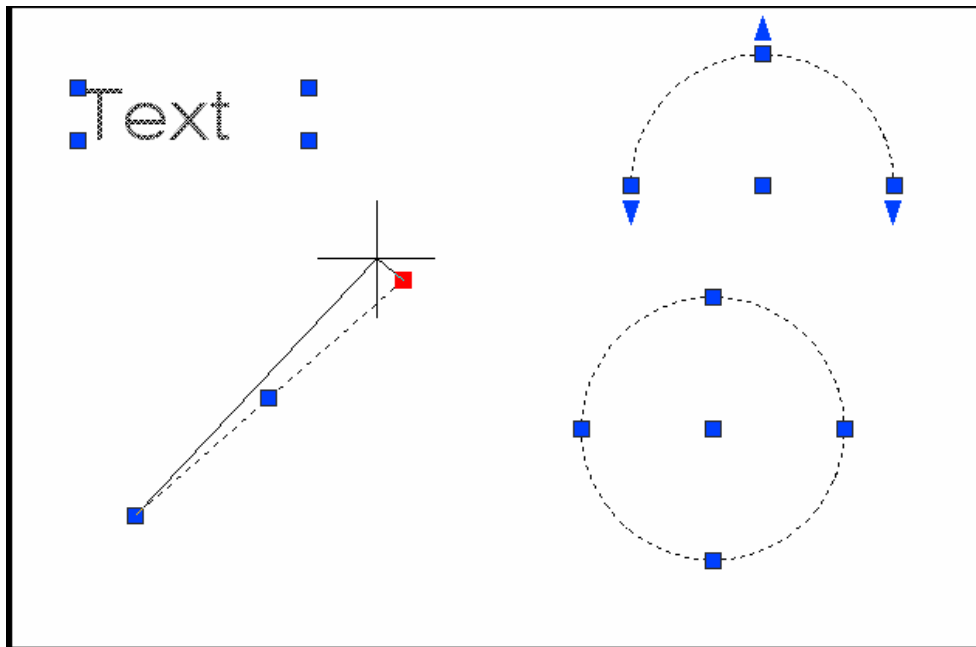
An unselected grip is one that has not yet been picked with the cursor, but is an item in the current selection set (it is highlighted). Pick an object to see grips.



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

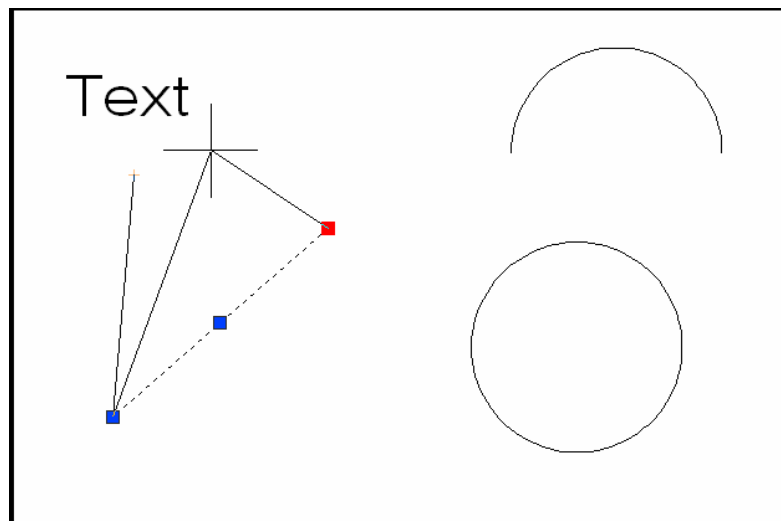
1. **Press** ESC to clear GRIPS.

TIP:

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

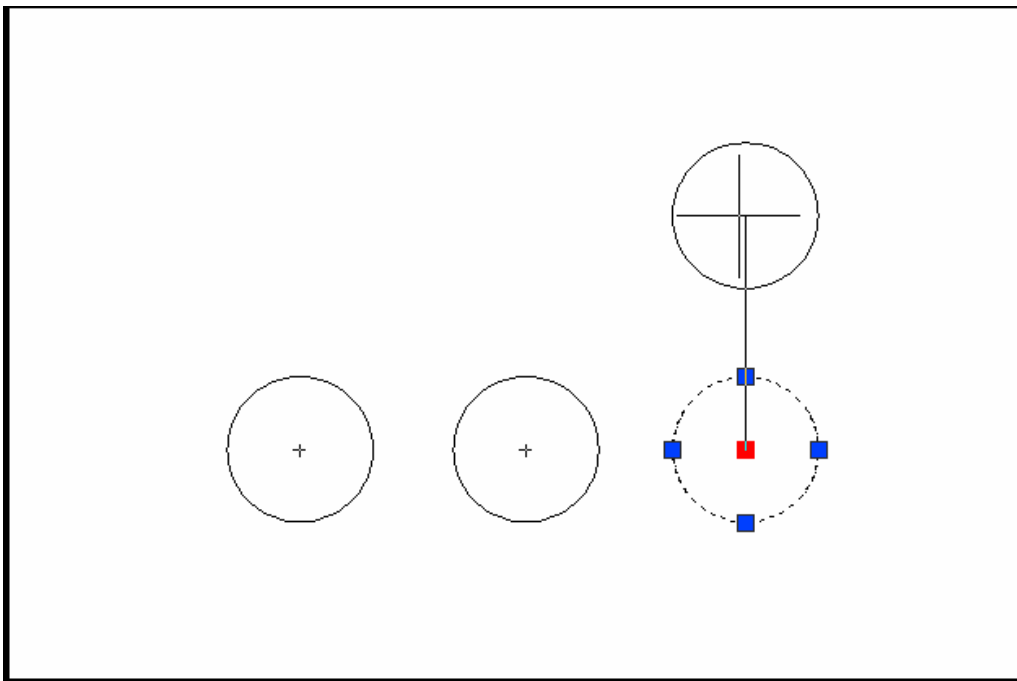
22.2 How To Use Grips

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
7. **Type** Grip mode.
enter C. X to exit



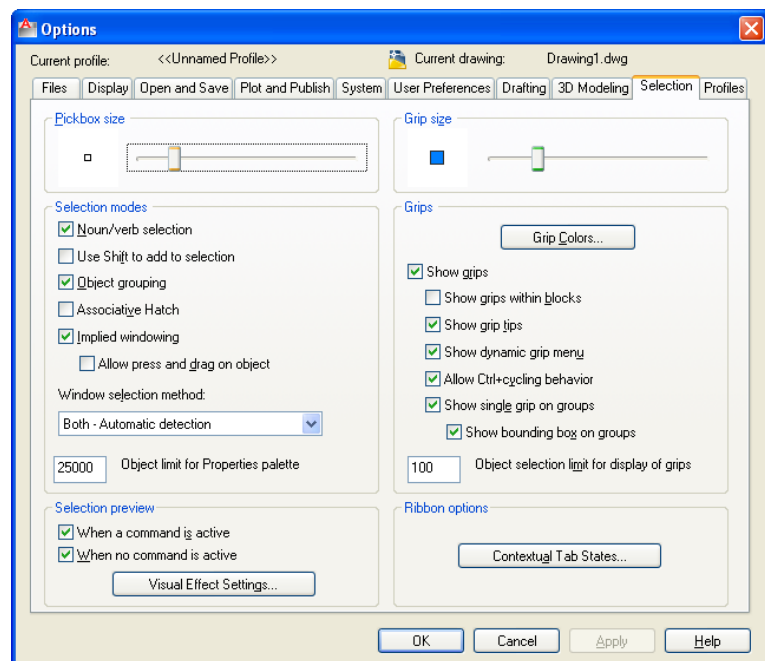
22.3 Copy Multiple with Grips

If you use the COPY option with anyone of the edit commands,a temporary auxiliarsnapgrid is created. To invoke the grid, hold the SHIFT keyafter 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.



22.4 Grips Settings (DDGRIPS Command)

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 ***

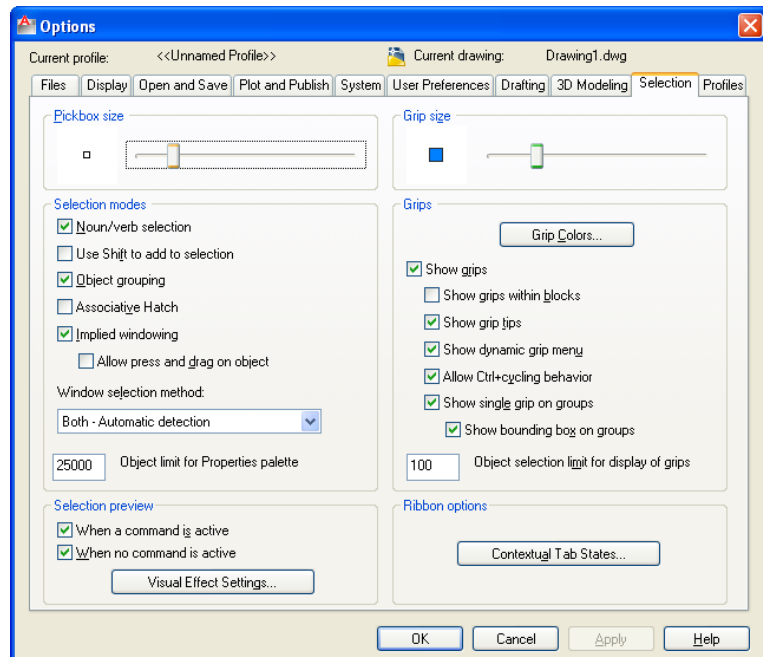
Advanced Selection Commands

23.1 Selection Modes

1. **Choose**
2. **Choose**

Tools, Options...

the SelectionTAB 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.

23.2 Groups

1. Type

GROUP at the command prompt.
Command: **GROUP**

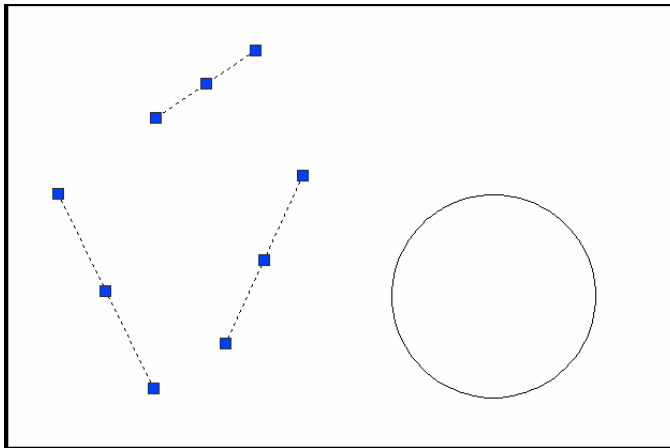
Select objects or [Name/Description]: N

Enter a group name or [?]: **LINES**

Select objects or [Name/Description]: Specify
opposite corner: 3 found, 1 group

Select objects or [Name/Description]: press enter

Lines in a group



Turning Groups ON/OFF

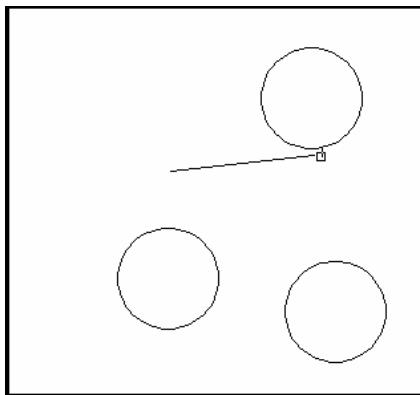
At any time, toggle group selection on and off by pressing either CTRL+H or SHIFT+CTRL+A.

23.3 Object Selection Cycling

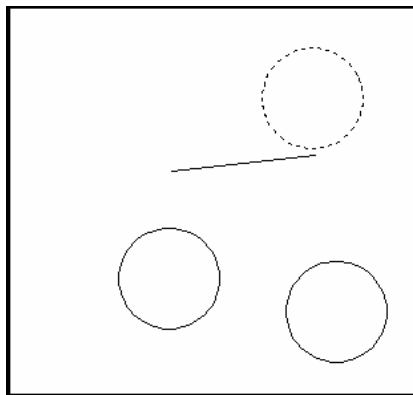
It is difficult to select objects that are close together or lie directly on top of one another. The following example shows two lines and a circle that all lie within the selection pickbox

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.

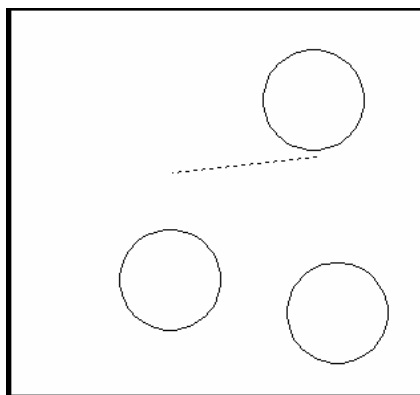
Objects touching pickbox highlights



First object selected

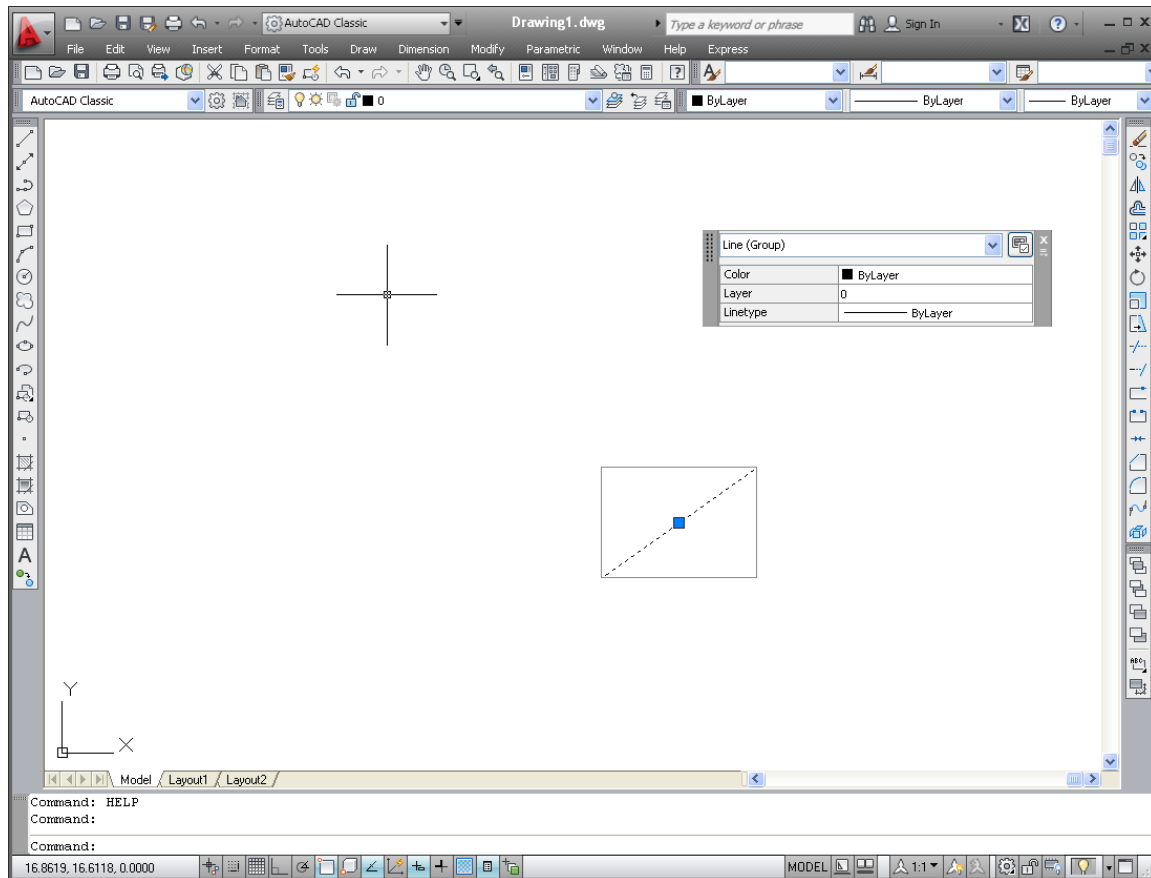


Second object selected



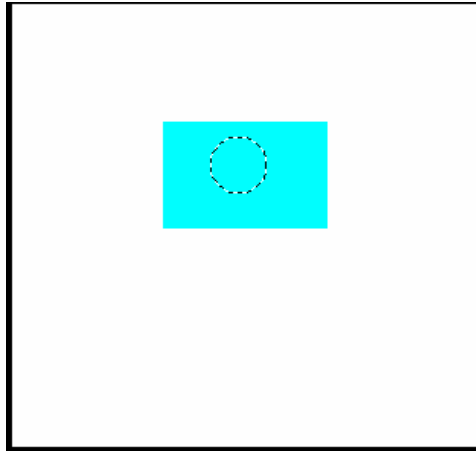
23.4 Double-Click Edit

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



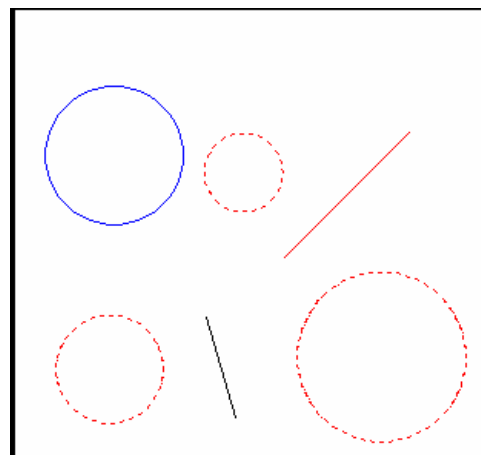
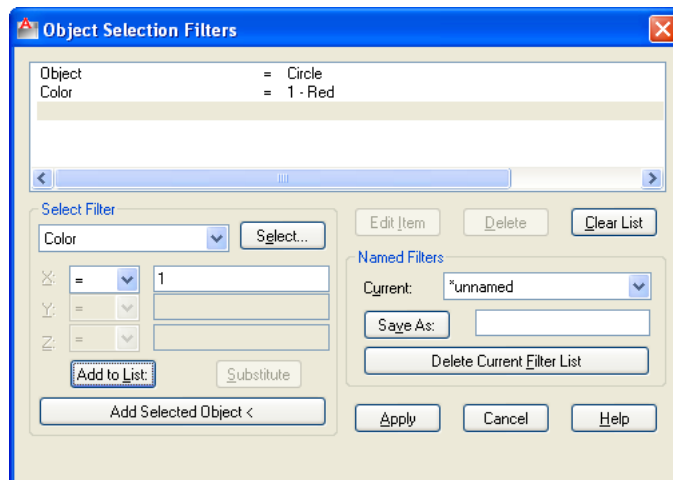
23.5 Draw Order

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.



23.6 Object Filters

1. **Type** FILTER at the Command prompt.
Command: **FILTER**
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.

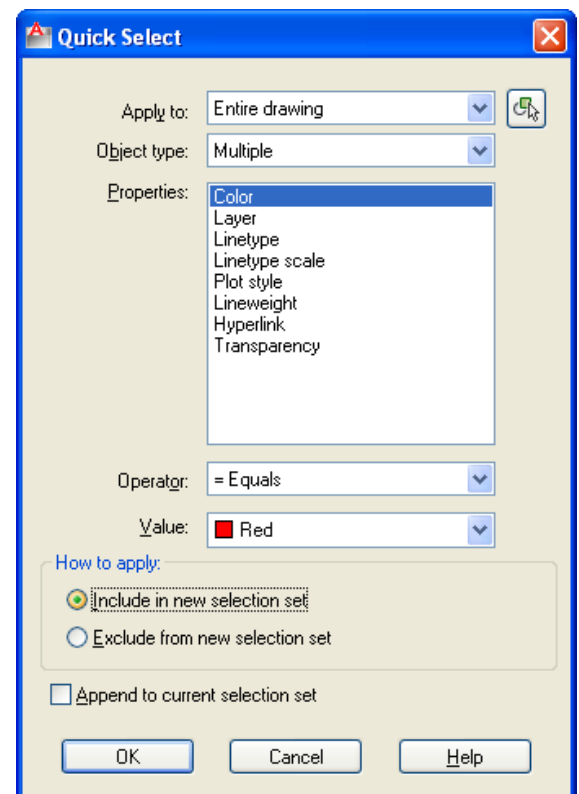
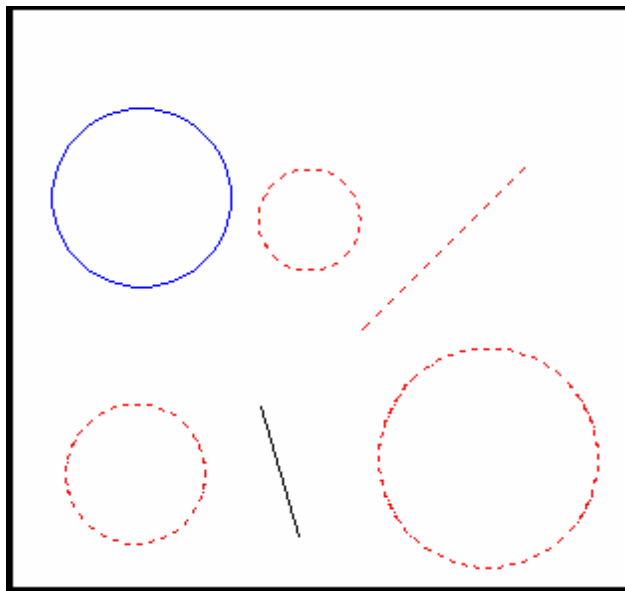


23.7 Quick Select


1. **Type** QSELECT at the command prompt.
Command: **QSELECT**
2. **Enter** the selection criteria.
3. **Choose** OK.

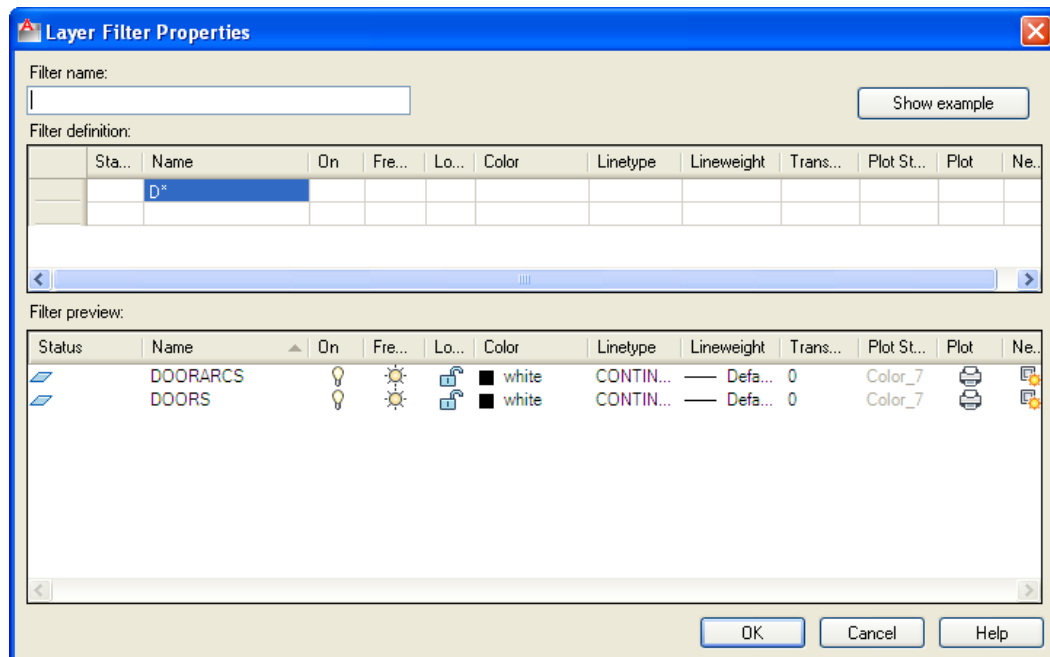
TIP:

- AutoCAD puts those object(s) into a selection set. Use the “P” previous option to select these objects.



23.8 Layer Filters

1. **Open** the layer dialog box.
2. **Choose** the filter icon. 
3. **Choose** the type of filter you would like to use (e.g. Name D*)

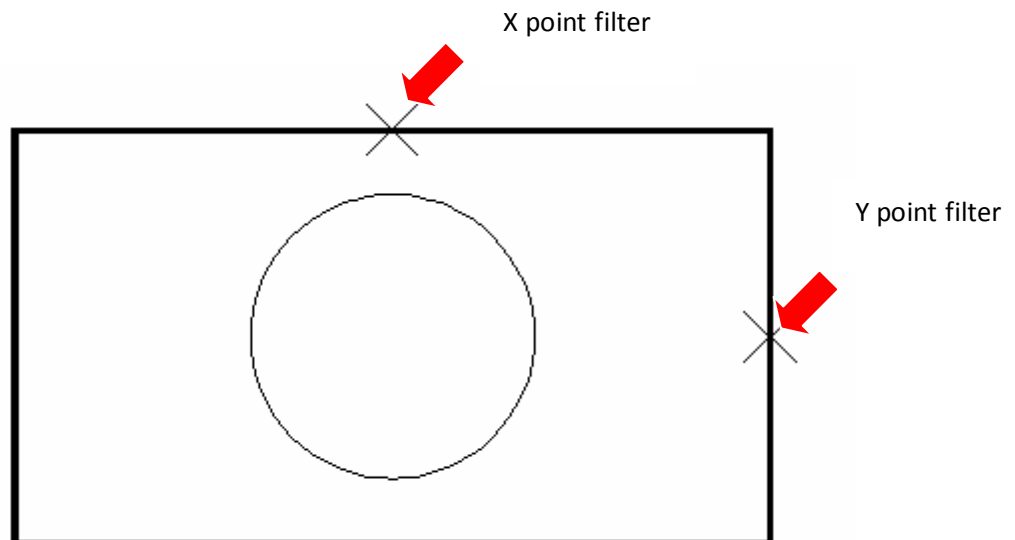


23.9 Point Filters

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

Circle center is placed at the intersection of the filters



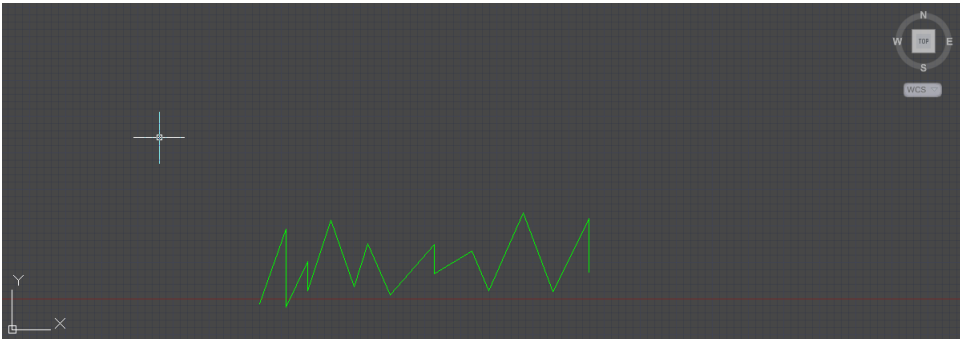
Chapter 24

External References and Underlays

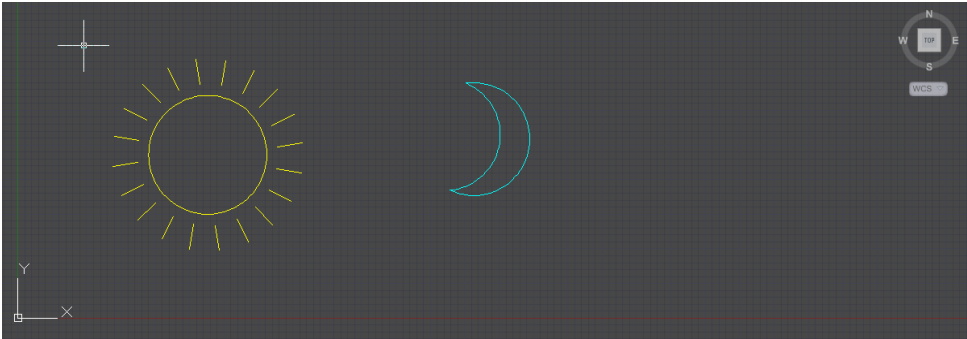
External Reference Files Overview

Note: The following drawings and layers were created for this chapter:

earth.dwg



Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight
	0				white	Continuo...	Defa...
	Grass				green	Continuo...	Defa...




Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight
	0				white	Continuo...	Defa...
	Moon				cyan	Continuo...	Defa...
	Sun				yellow	Continuo...	Defa...

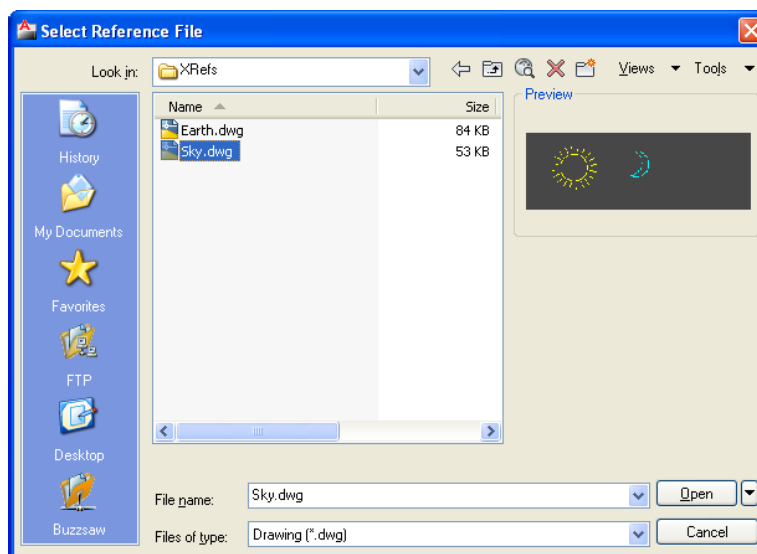
The following are Xref characteristics:

- 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. DWG|LAYER)
- 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 detached 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.

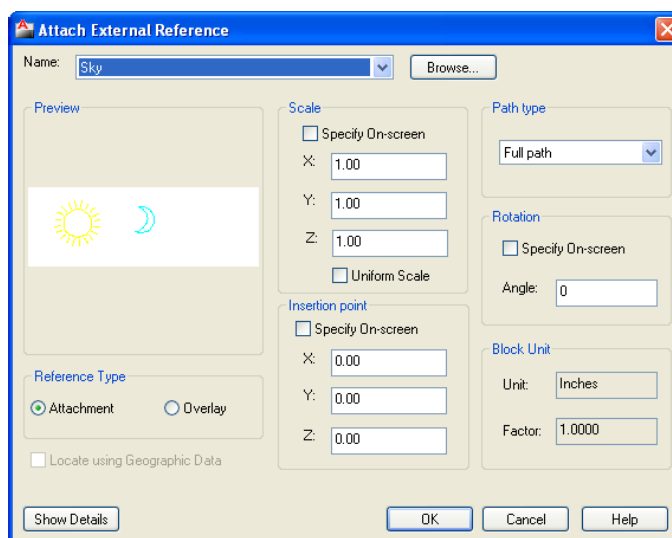
24.1 Attaching Xrefs

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

1. **Open** the drawing “earth.dwg”
2. **Choose** Insert, External Reference
or
3. **Type** XATTACH at the command prompt.
Command: **XATTACH**
or
4. **Click** the Xref Attach Icon from the Reference Toolbar. 
5. **Choose** a drawing name to attach.



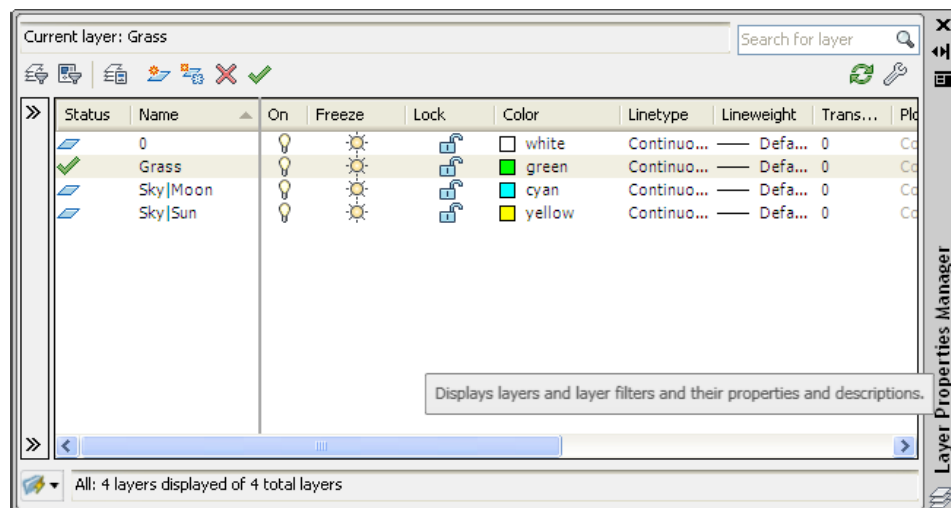
6. **Specify** the insertion parameters.



24.2 Xref Layers

Layer Dialog Box

- Each Xref named object is prefixed with the xref drawing name and a pipe “|” symbol. (i.e. Sky|Sun for drawing sky.dwg and layer SUN)
- Xdep stands for external reference dependent objects.
- Xrref'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.



24.3 Xclip

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 Icon from the Reference Toolbar.
or
3. **Type** XCLIP at the command prompt.
Command: **XCLIP**



Select objects: Other corner: 1 found

Select objects: **pick reference file**

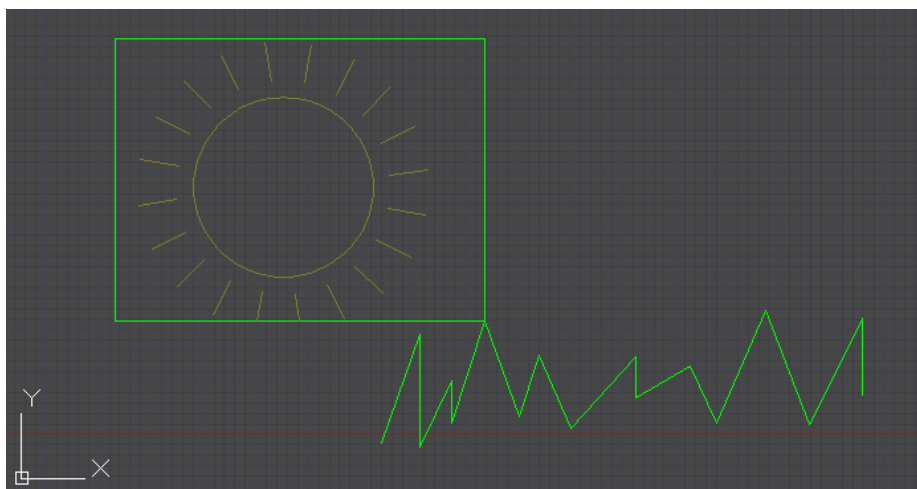
ON/OFF/Clipdepth/Delete/generate Polyline/

<New boundary>:

Specify clipping boundary:

Select polyline/Polygonal/<Rectangular>:

First corner: Other corner: **pick corners**

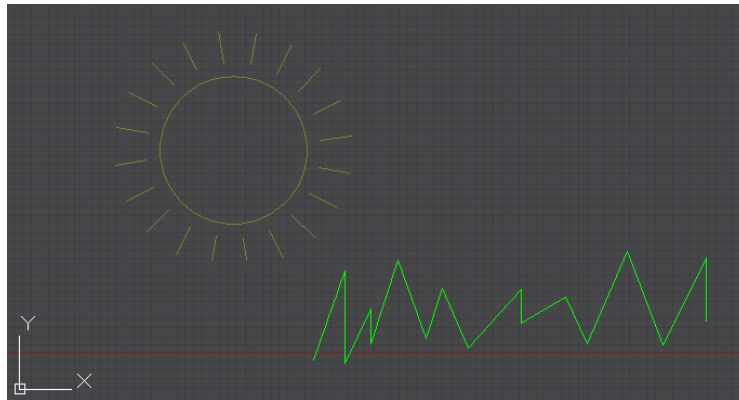


24.4 Xclipframe

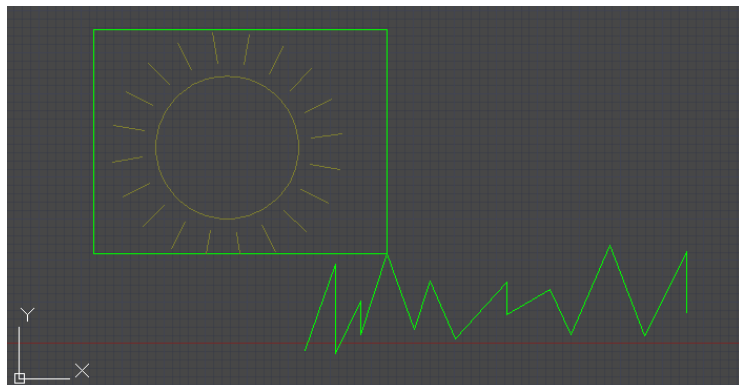
Controls visibility of xref clipping boundaries.

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 |

Xclip frame off



Xclip frame on

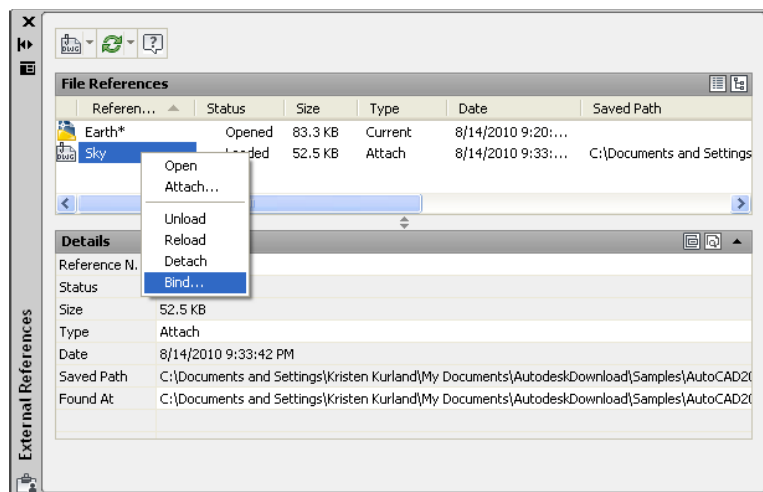


Binding an Xref 24.5

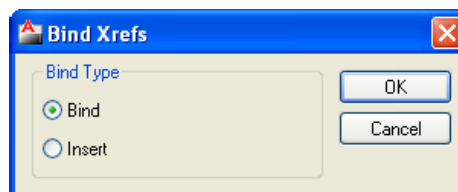
Converts Xreference files to block definitions in the current drawing. Associated objects change names (i.e. the layer called SKY|SUN becomes SKY\$0\$SUN). Binding loses the connection to the referenced file. Xrefs can also be inserted like Wblocks.

To Bind an Xref:

1. **Choose** Insert, ExternalReferences
Or
2. **Type** Externalreferences at the command prompt.
Command: **EXTERNALREFERENCES**
3. **Choose** an xref name.
4. **Right-click** and choose the Bind option.



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



24.6 Xbind

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

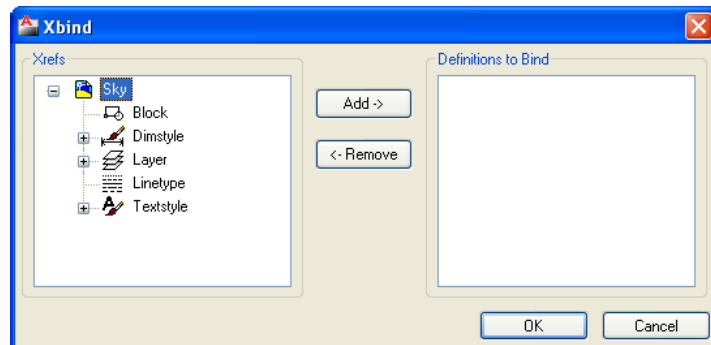
2. **Click** the Xbind Icon from the Reference Toolbar.



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

XBIND

Block/Dimstyle / LAyer/LType /Style:



24.7 Editing Xrefs

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. **Type** ExternalReferences from the command prompt.
Command: **EXTERNALREFERENCES**
2. **Right-click** and choose the Detach option.

Unload an Xref

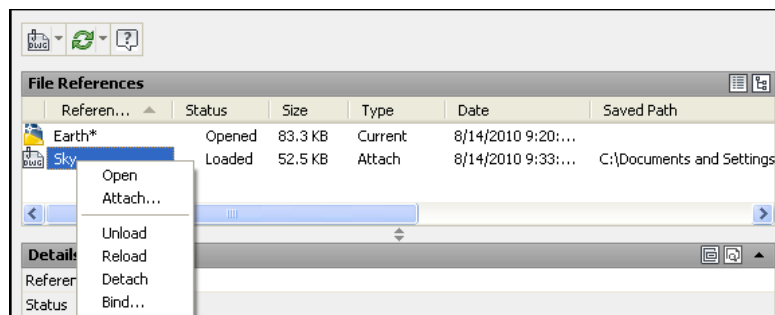
Unloads one or more xrefs. Unloaded xrefs can be easily reloaded. Unlike detach, unloading does not remove the xref permanently. It merely suppresses the display and regeneration of the xref definition, to help current session editing and improvement of performance.

1. **Type** ExternalReferences from the command prompt.
Command: **EXTERNALREFERENCES**
2. **Right-click** and choose the Unload option.

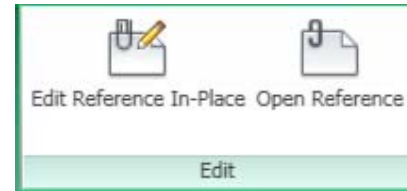
Reload an Xref

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

1. **Type** ExternalReferences from the command prompt.
Command: **EXTERNALREFERENCES**
2. **Right-click** and choose the Reload option.



24.8 Open an Xref to Edit



XOpen

Opens a block or xref in a new window.

1. **Choose** Modify, Xref and Block Editing, Open Reference.
or
2. **Type** Xopen at the command prompt.
Command: **XOPEN**
3. **Edit** the objects as desired.
4. **Save** the edits.

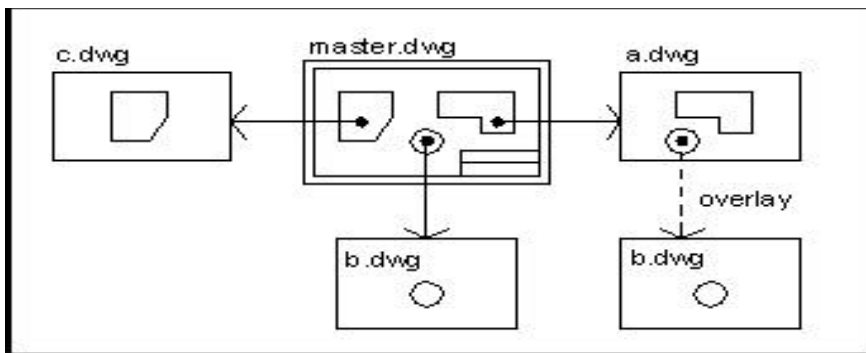
Refedit


Edits a block or external reference file in the current drawing

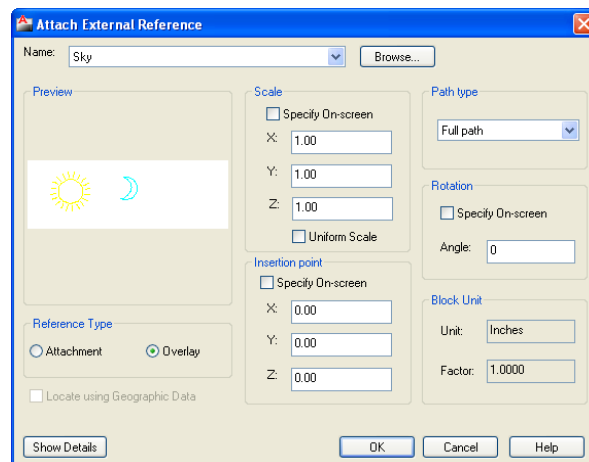
1. **Choose** Modify, Xref and Block Editing, Edit Xreference in Place
or
2. **Type** Refedit at the command prompt.
Command: **REFEDIT**
3. **Edit** the objects as desired.
4. **Save** the edits with the REFEDIT toolbar or REFCLOSE command.

24.9 Overlay an Xref

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. **Type** External references at the command prompt.
Command: **EXTERNALREFERENCES**
2. **Click** the drawing icon. 
3. **Choose** a drawing name to attach.
4. **Choose** Overlay in the AttachXref dialog box under Reference Type.
5. **Specify** the insertion parameters.



24.10 Underlays

Underlays allow you to attach DWF, DWFx, DGN, and PDF files. You can underlay and snap to 2D geometry stored in these files. Underlays are similar to attached raster images in that they provide visual content but also support object snapping and clipping. Unlike external references, underlays cannot be bound to the drawing.

Attaching an Underlay

1. **Type** one of the following commands to attach an underlay: Command: ATTACH, DGNATTACH, DWFATTACH, or PDFATTACH

Detaching an Underlay

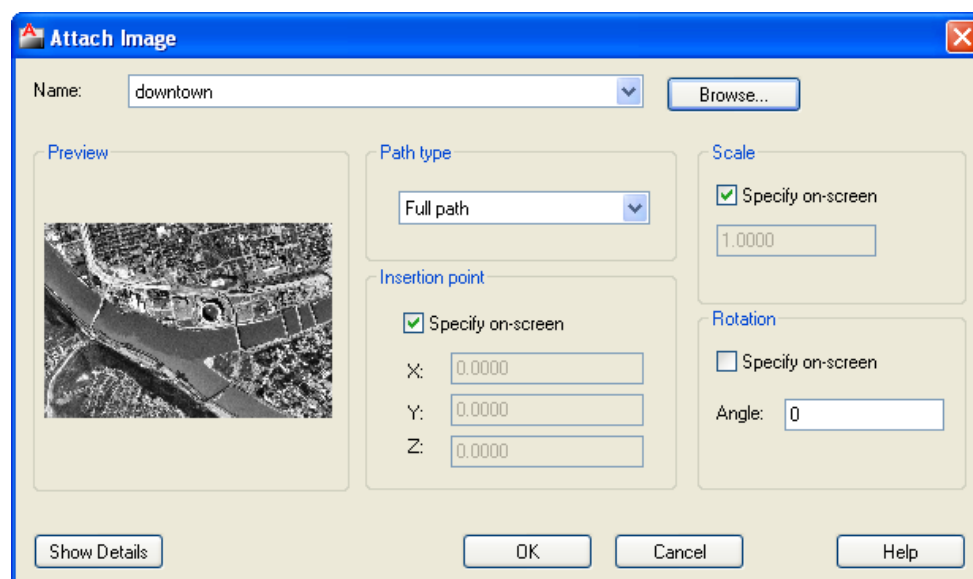
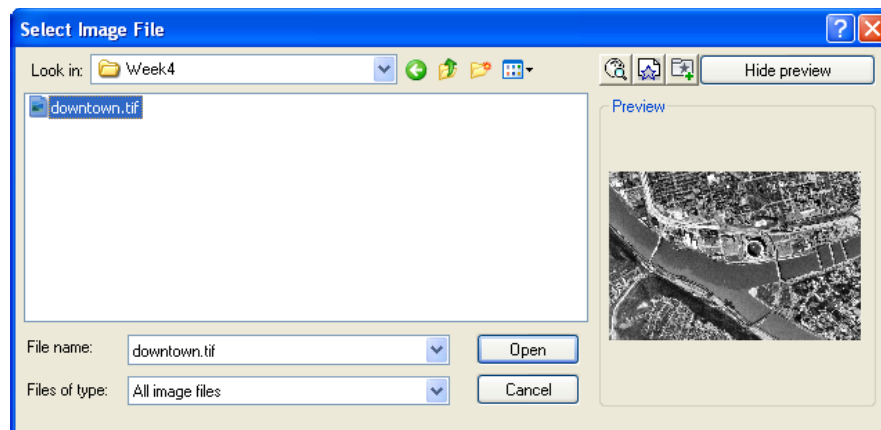
1. **Type** External References from the command prompt.
Command: **EXTERNALREFERENCES**
2. **Right-click** and choose the Detach or Unload option.

Chapter 25

Raster Images

25.1 Inserting Images

1. **Choose** Insert, Raster Image Reference...
 2. **Choose** a raster image file to insert.
 3. **Specify** a location to insert the image.
- or
4. **Type** IMAGEATTACH at the command prompt.
Command: **IMAGEATTACH**
- or
5. **Click** the Image icon from the Reference Toolbar.



25.2 Image Appearance

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.

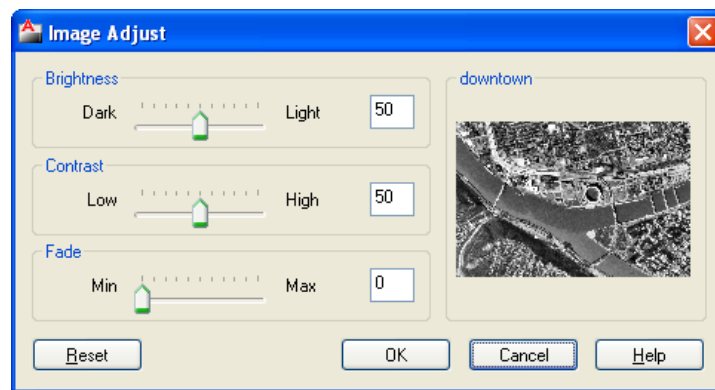


Image Transparency

1. **Choose** Modify, Object, Image, Transparency.
or
2. **Type** TRANSPARENCY at the command prompt.
Command: **TRANSPARENCY**
3. **Choose** a raster image file edit.
4. **Type** ON or OFF to turn an image's transparency on or off.

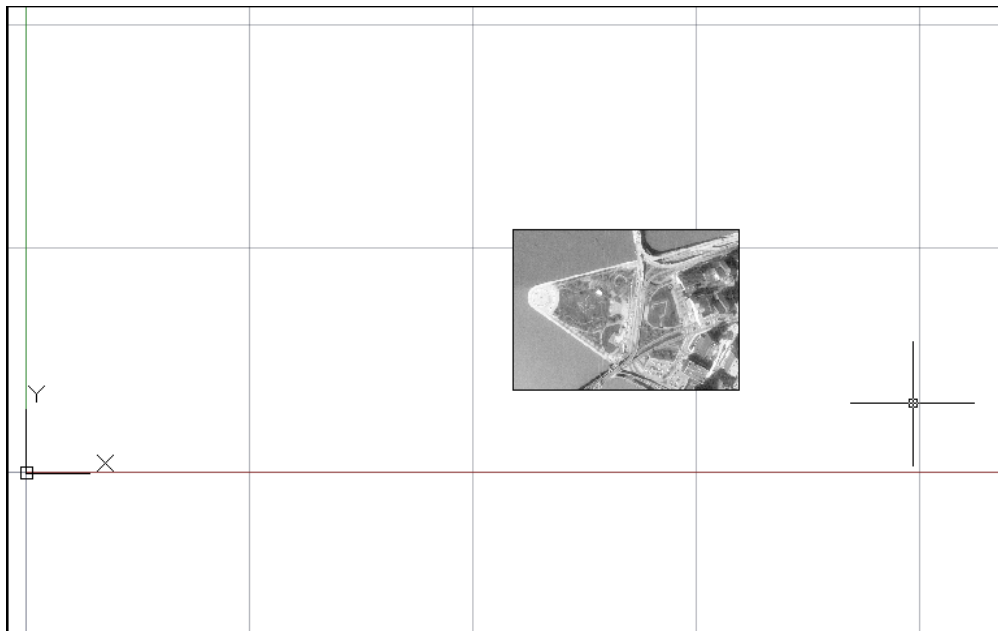
Image Quality

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

25.3 Clipping Images

1. **Choose** Modify, Clip, Image.
or
2. **Type** IMAGECLIP at the command prompt.
Command: **IMAGECLIP**
Select image to clip:
Enter image clipping option

[ON/OFF/Delete/New boundary] <New>:
Outside mode - Objects outside boundary will
be hidden.
Specify clipping boundary or select invert
option:
[Select polyline/Polygonal/Rectangular/Invert



Chapter 26

Dimensions

26.1 Linear Dimensions

1. **Choose** Dimension, Linear.

or

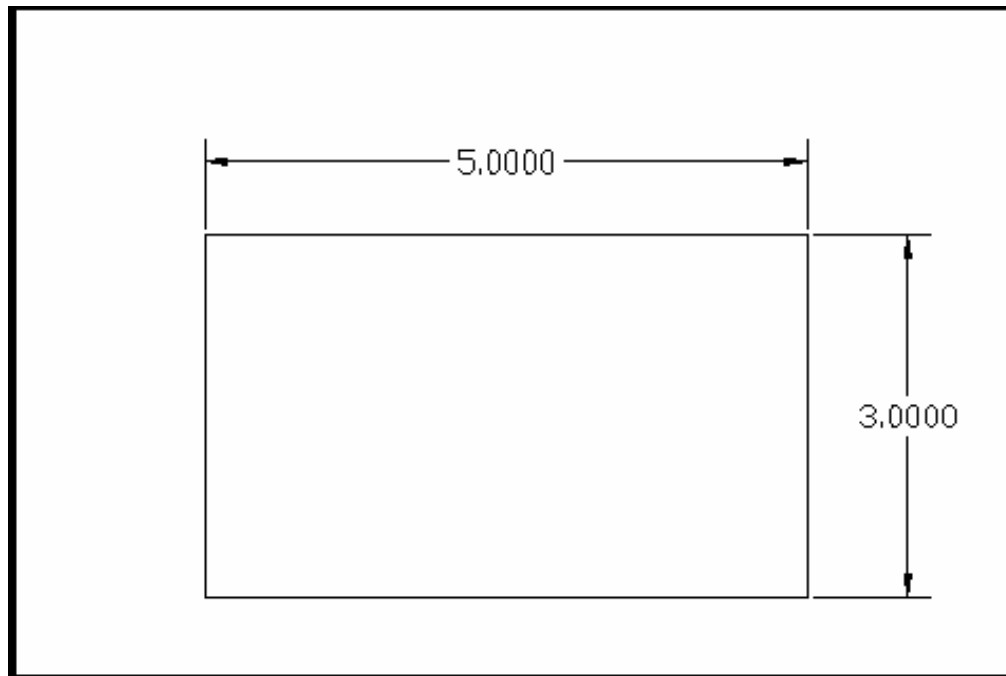
2. **Click** the Linear Dimension command from the toolbar.



or

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

Dim: HOR or VER



26.2 Aligned Dimensions

1. **Choose** Dimension, Aligned.

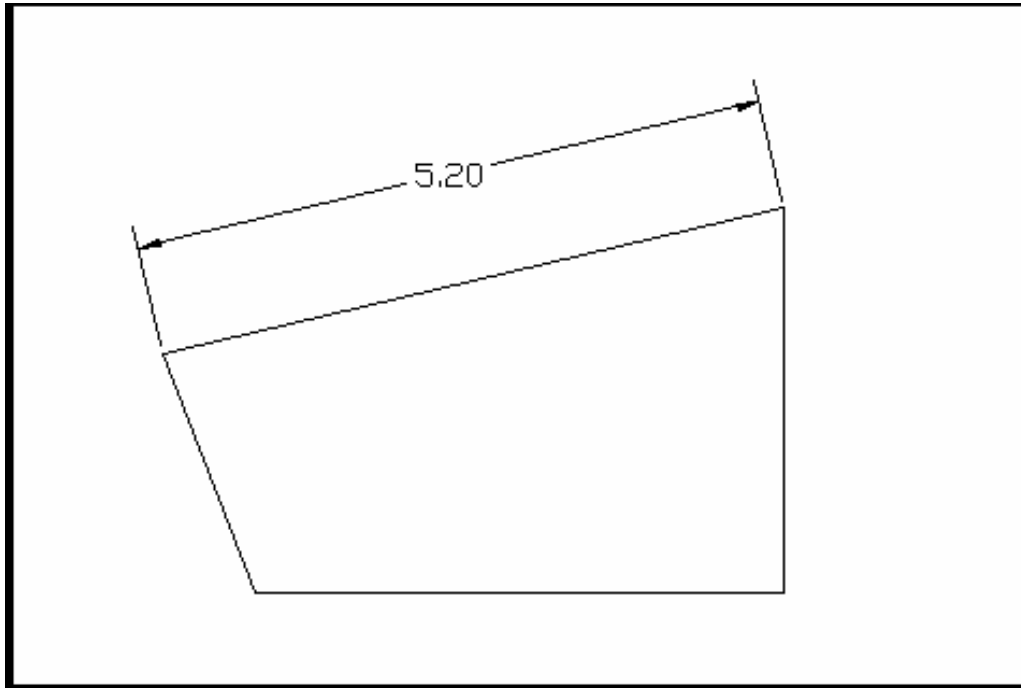
or

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



or

3. **Type** DIM at the command prompt. Command: **DIM**
Dim: **ALIGNED**



26.3 Radial Dimensions

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

or

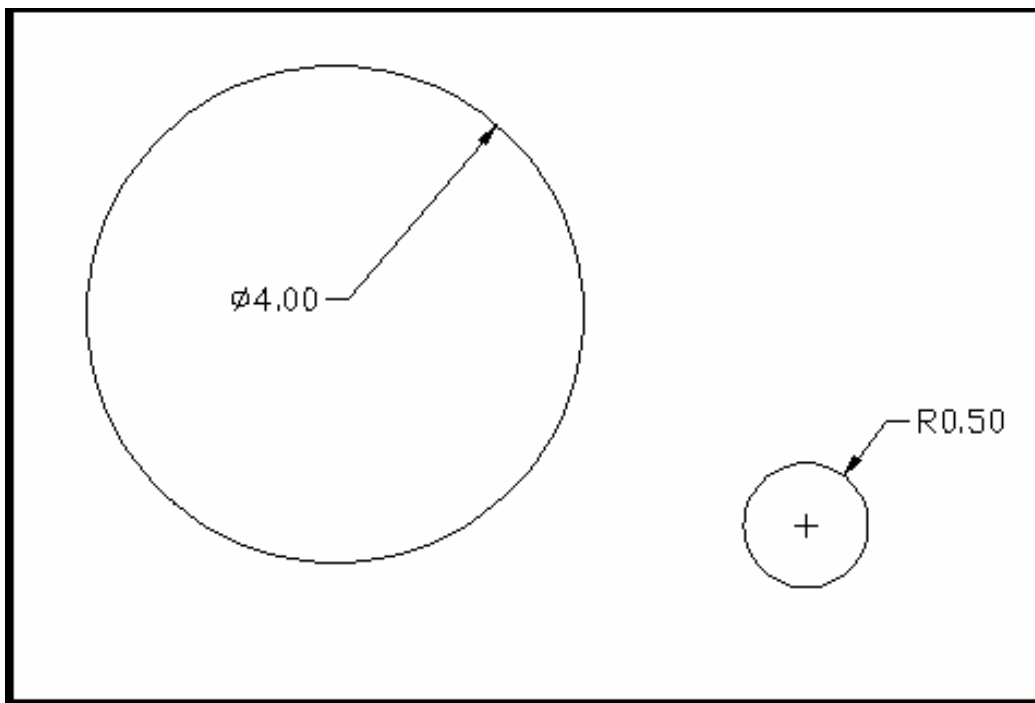
2. **Click** the Radial Dimensions command from the toolbar.



or

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

Dim: **RADIUS** or **DIAMETER**



26.4 Angular Dimensions

1. **Choose** Dimension, Angular.

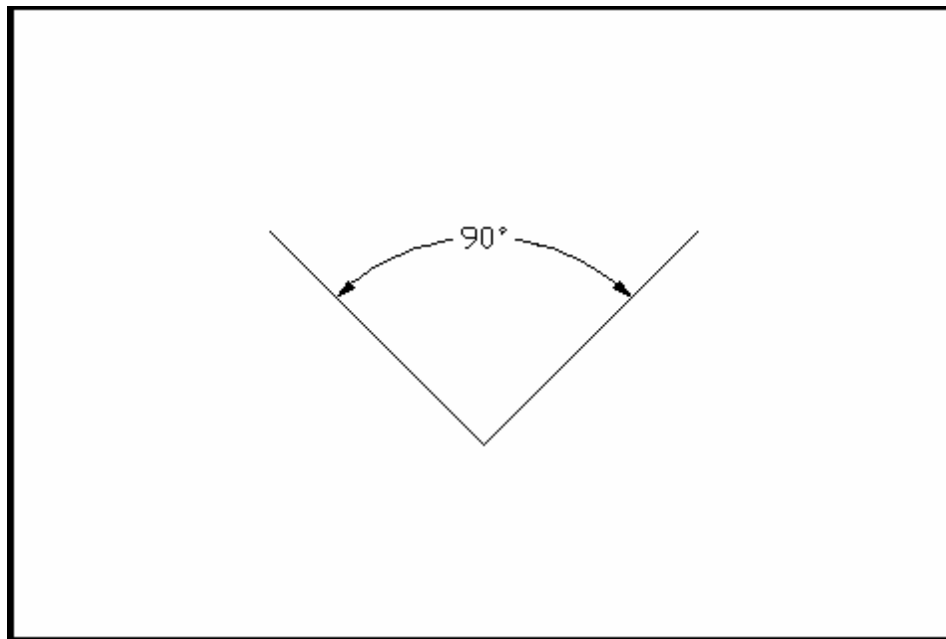
or

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



or

3. **Type** DIM at the command prompt. Command: **DIM**
Dim: **ANGULAR**



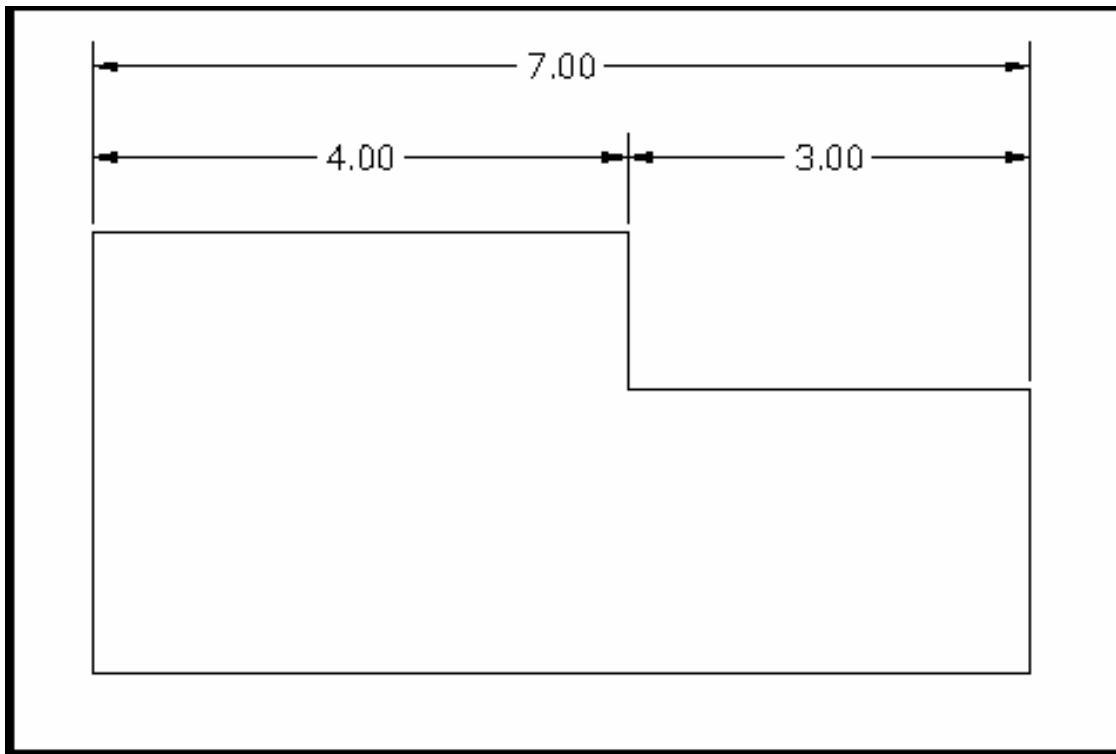
26.5 Continued and Baseline Dimensions

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**



26.6 Leaders

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

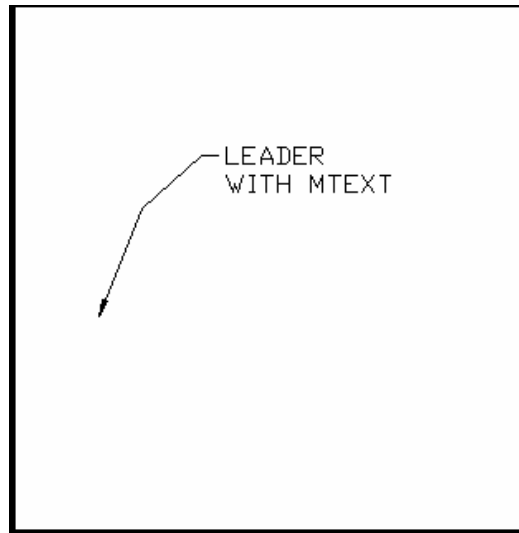
or

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



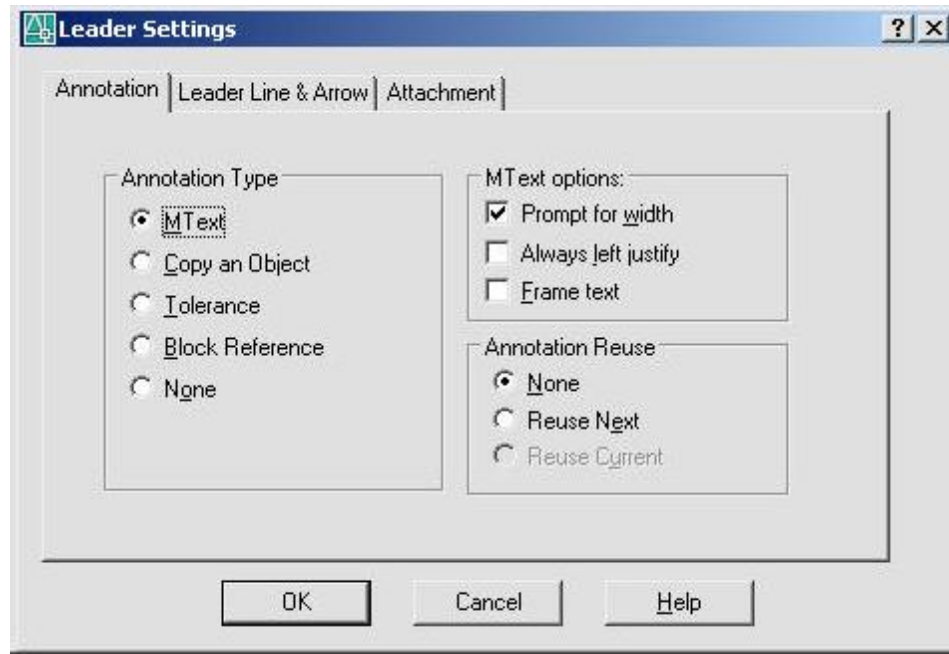
or

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.



26.7 Quick Dimensions

Quickly creates dimension arrangements from the geometry you select.

1. **Choose** Dimension, QDIM.

or

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

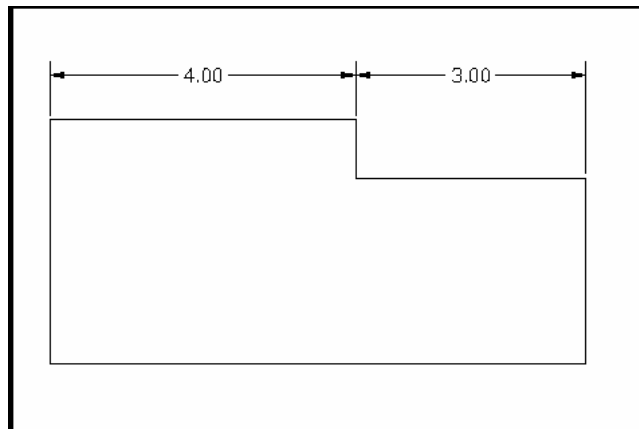
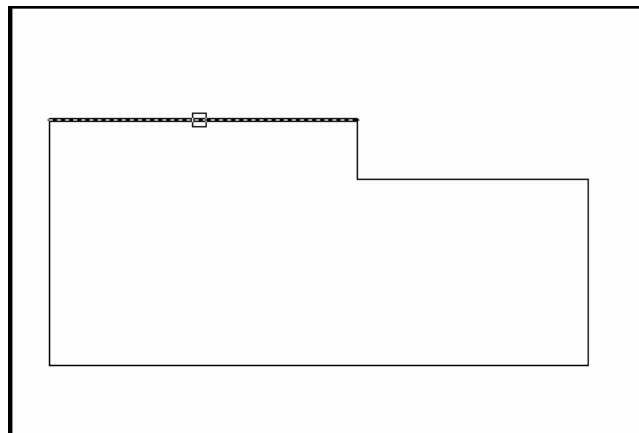


or

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

QDIM

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



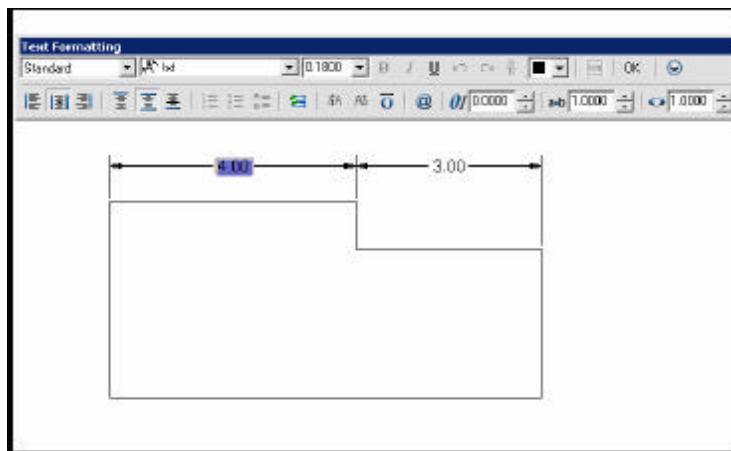
26.8 Modifying Dimensions

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.

26.9 Ordinate Dimensions

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

Chapter 27

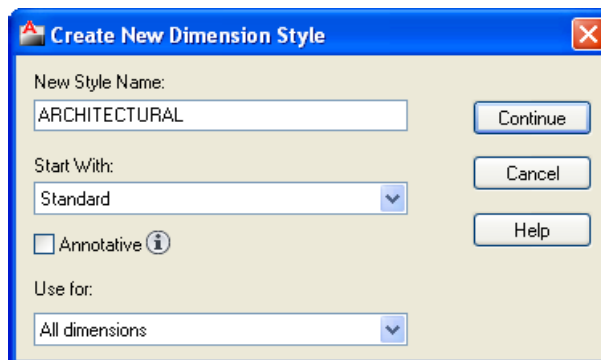
Dimension Styles

27.1 Creating Dimension Styles

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.

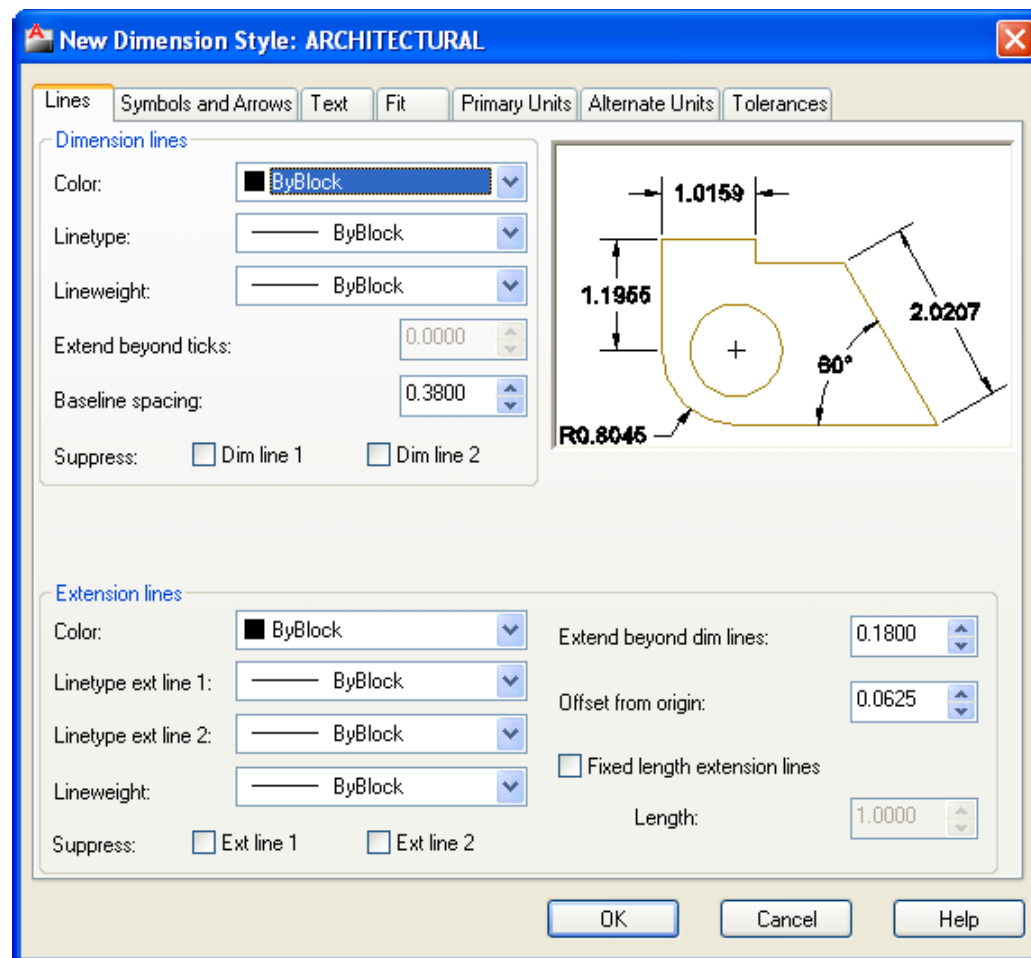
TIP:

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

27.2 Lines

Edits Dimension Lines, Extension Lines.

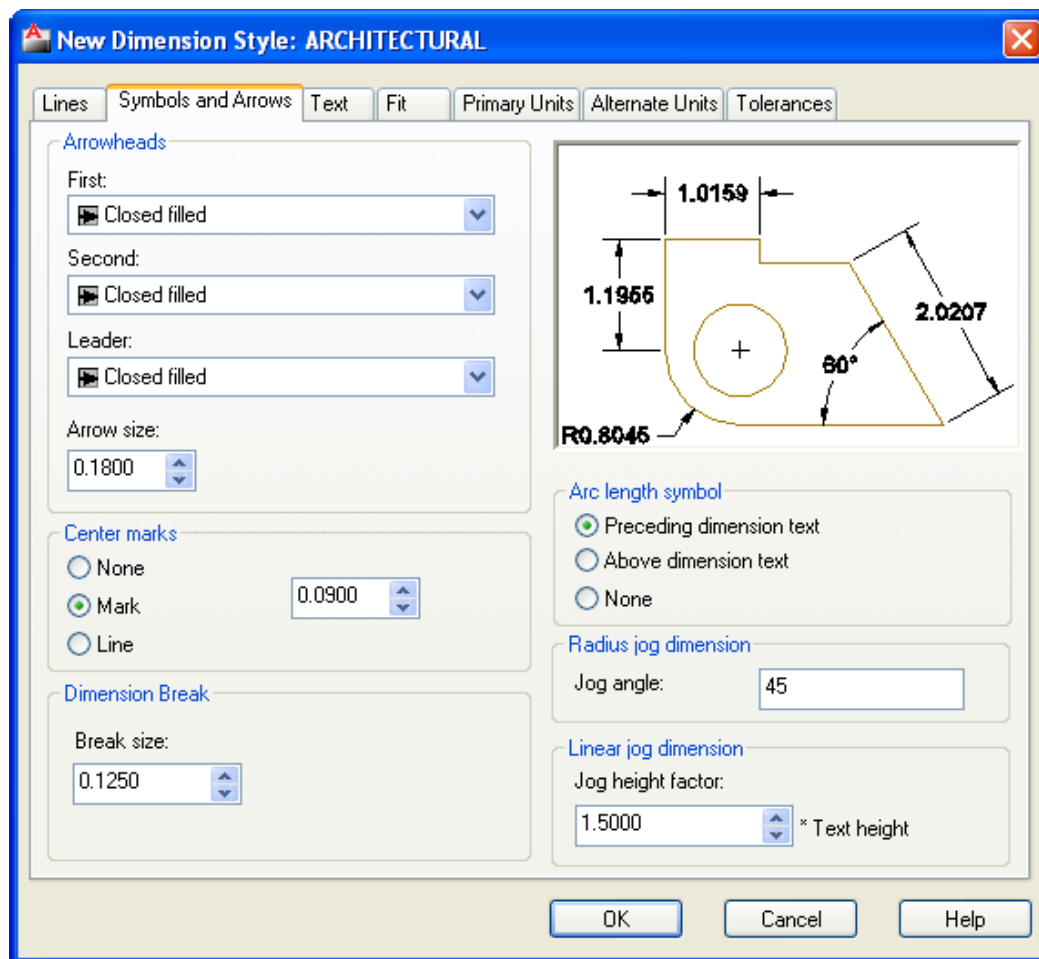
1. **Pick** the Lines tab from the Dimension Variables and Styles dialog box.



27.3 Symbols and Arrows

Edits Symbols and Arrows.

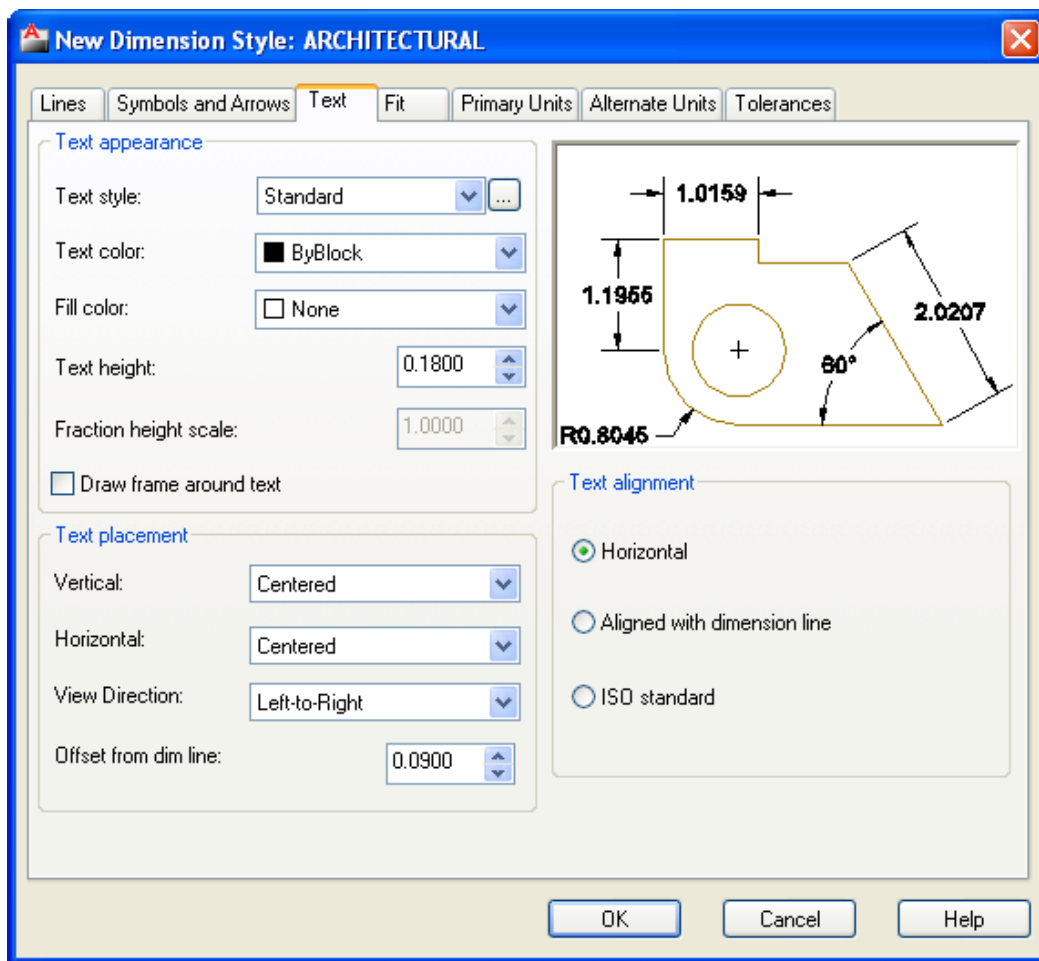
1. **Pick** the Symbols and Arrows tab from the Dimension Variables and Styles dialog box.



27.4 Text

Edits Text Appearance, Text Placement and Text Alignment.

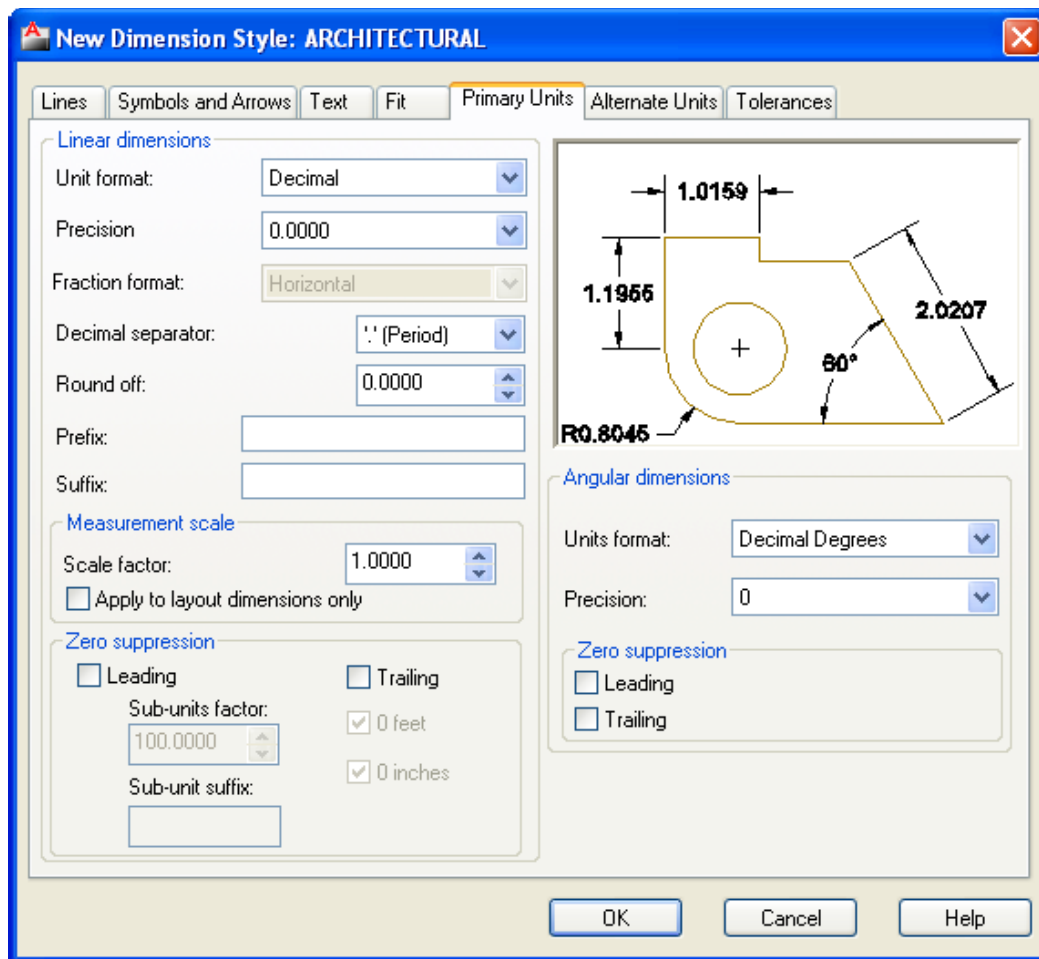
1. **Pick** the Text tab from the Dimension Variables and Styles dialog box.



27.5 Primary Units

Edits Unit options for dimension's primary units.

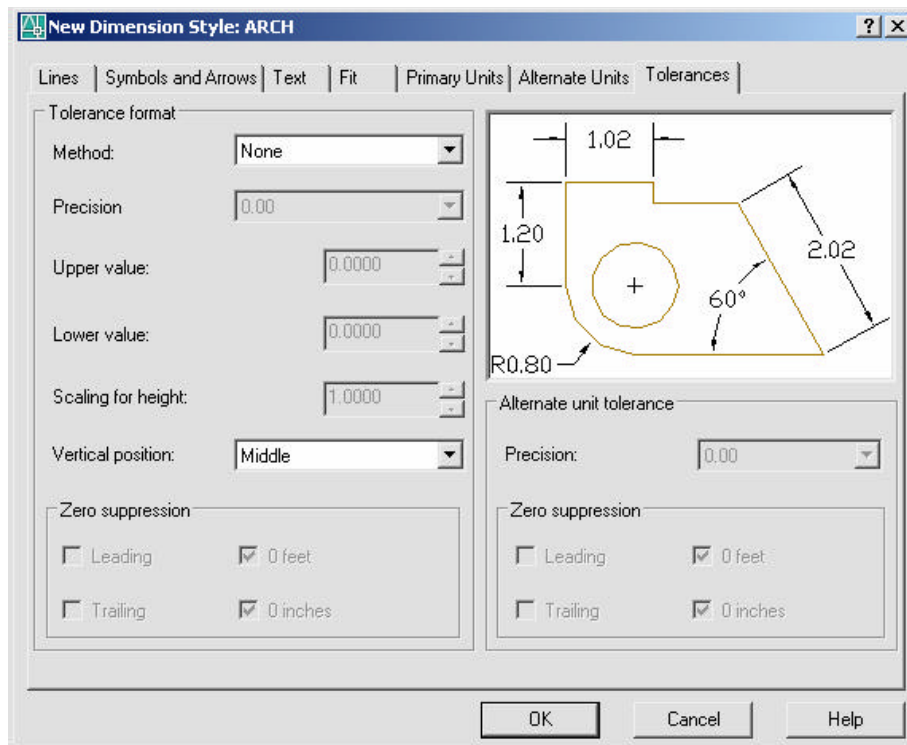
1. **Pick** the PRIMARY UNITS tab from the Dimension Variables and Styles dialog box.



27.6 Alternate Units

Edits Unit options for dimension's alternate units.

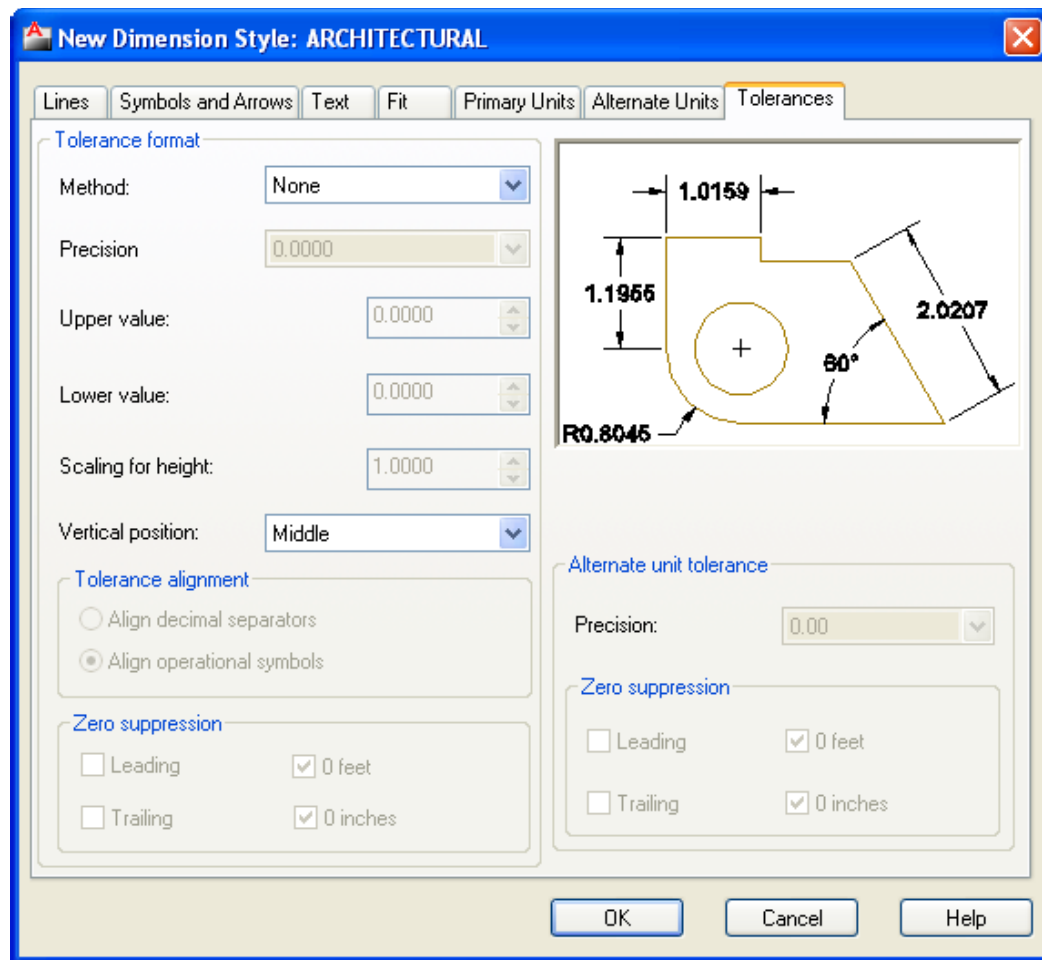
Pick the ALTERNATE UNITS tab from the Dimension Variables and Styles dialog box.



27.7 Tolerances

Edits Unit options for tolerances.

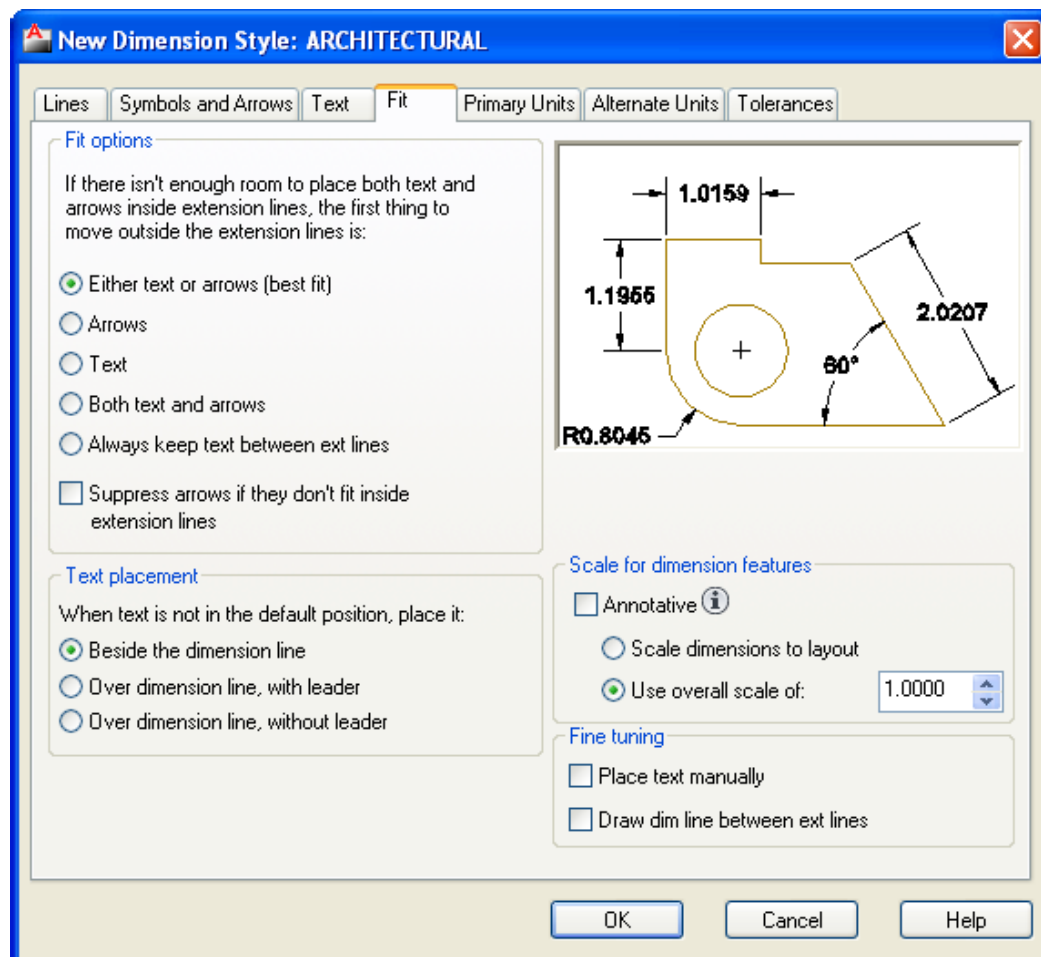
1. **Pick** the TOLERANCES tab from the Dimension Variables and Styles dialog box.



27.8 Fit

Edits Unit options for fitting dimensions and dimension scales.

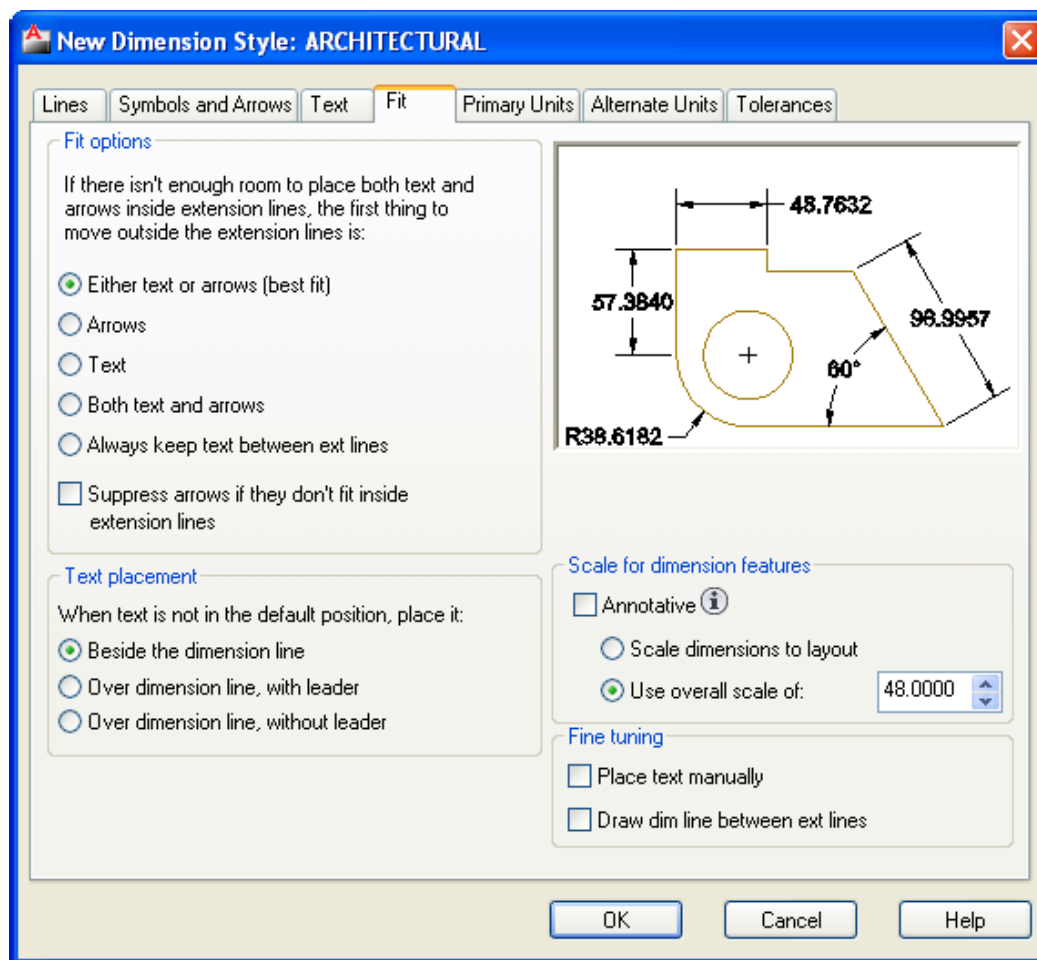
1. **Pick** the FIT tab from the Dimension Variables and Styles dialog box.



27.9 Dimscale

Edits Unit options for fitting dimensions and dimension scales.

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



27.10 Dimension Override

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.

27.11 Dimension Variables

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

28.1 Named Views

Ddview Command

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

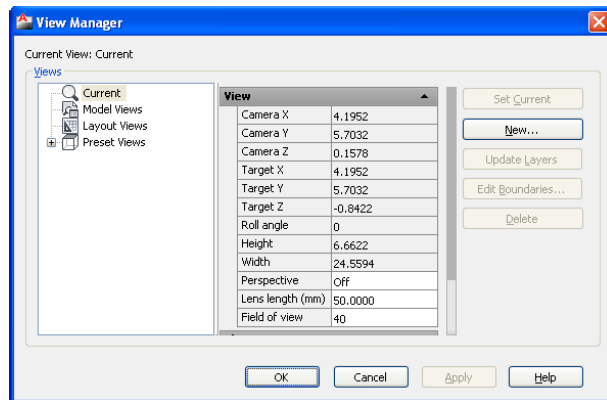
or



2. **Click** the Named View icon from the View toolbar.

or

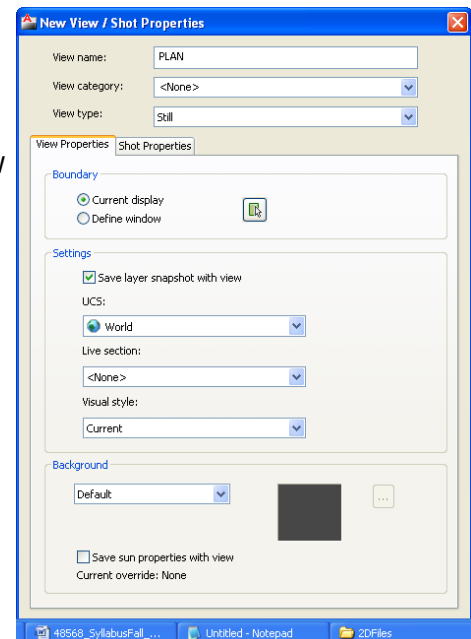
3. **Type** DDVIEW at the command prompt. Command:
DDVIEW



4. **Choose** the NEW button.

5. **Type** a view name.

6. **Choose** Current display or Define Window



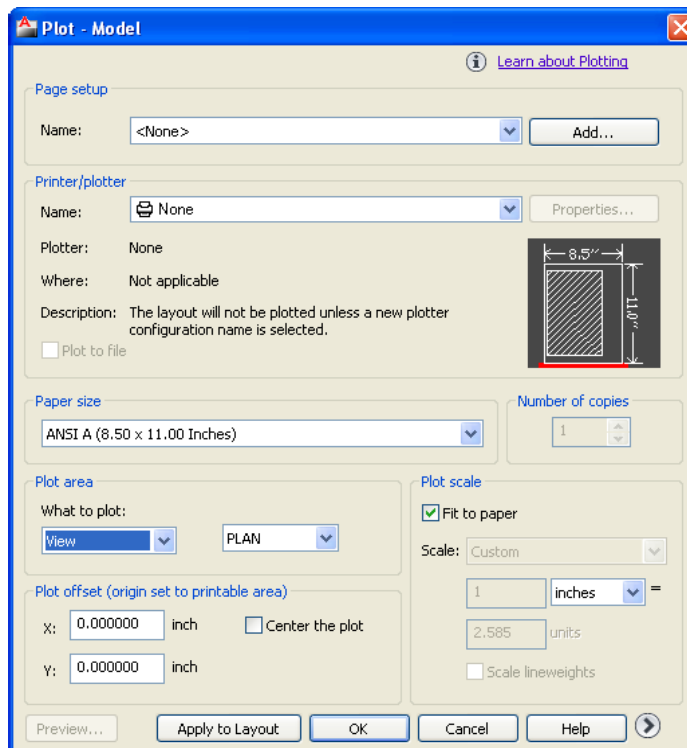
Typing the View Command

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 |

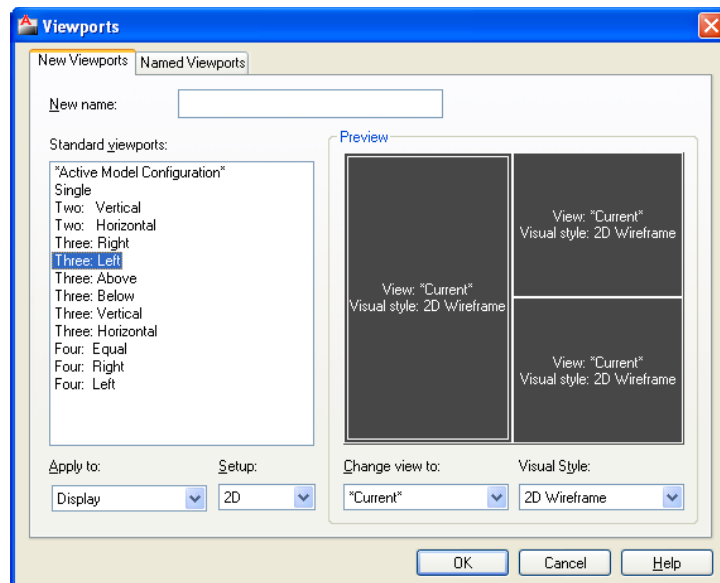
Plotting Named Views



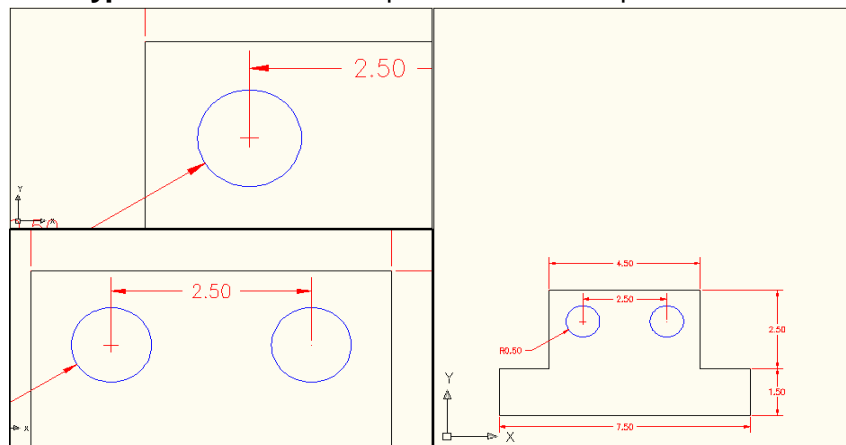
28.2 Viewports

Vports Command

1. **Choose** View, Viewports, New Viewports... or type **VPORTS** at the command prompt.
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	Assigns 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:

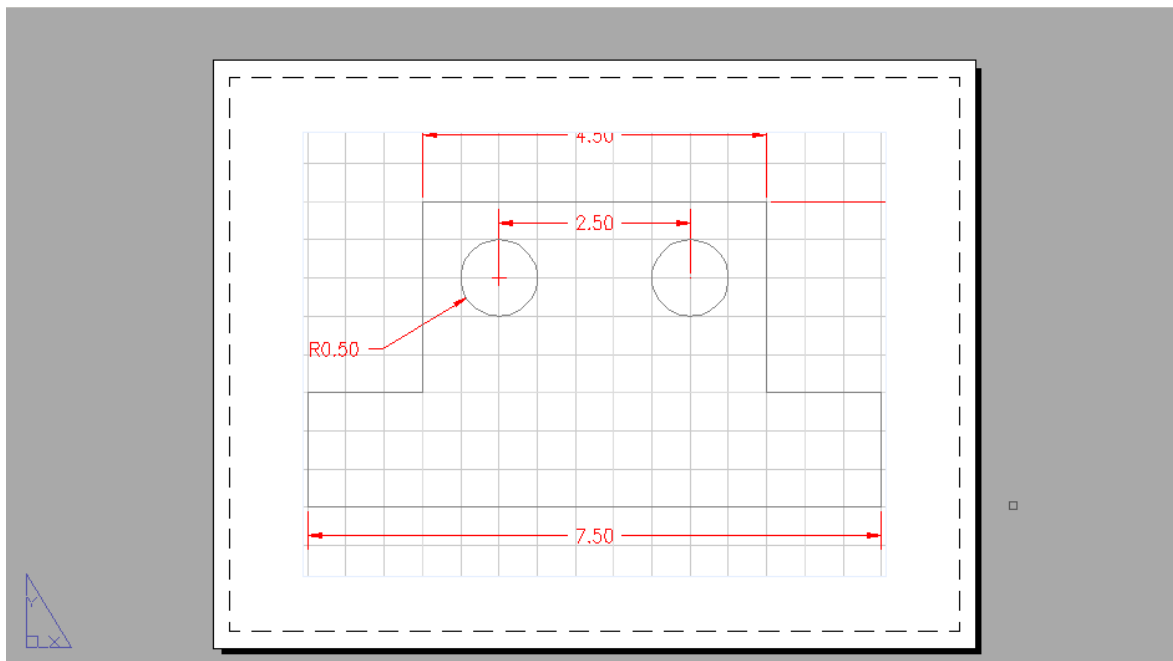
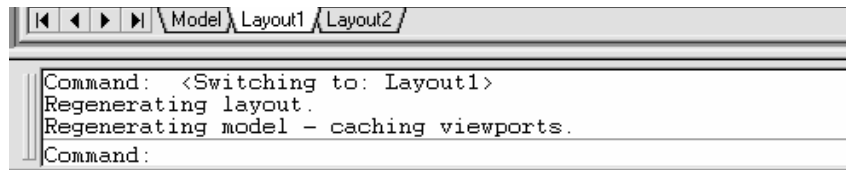
- Viewports can be named and restored later.
- AutoCAD plots only the current vport.

Chapter 29

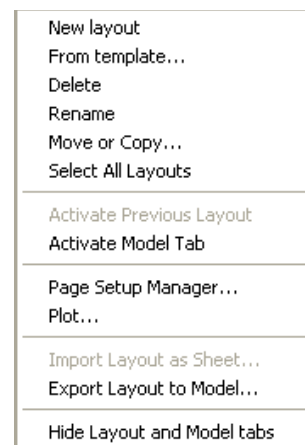
Model Space and Paper Space Layouts

29.1 Creating a Layout

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

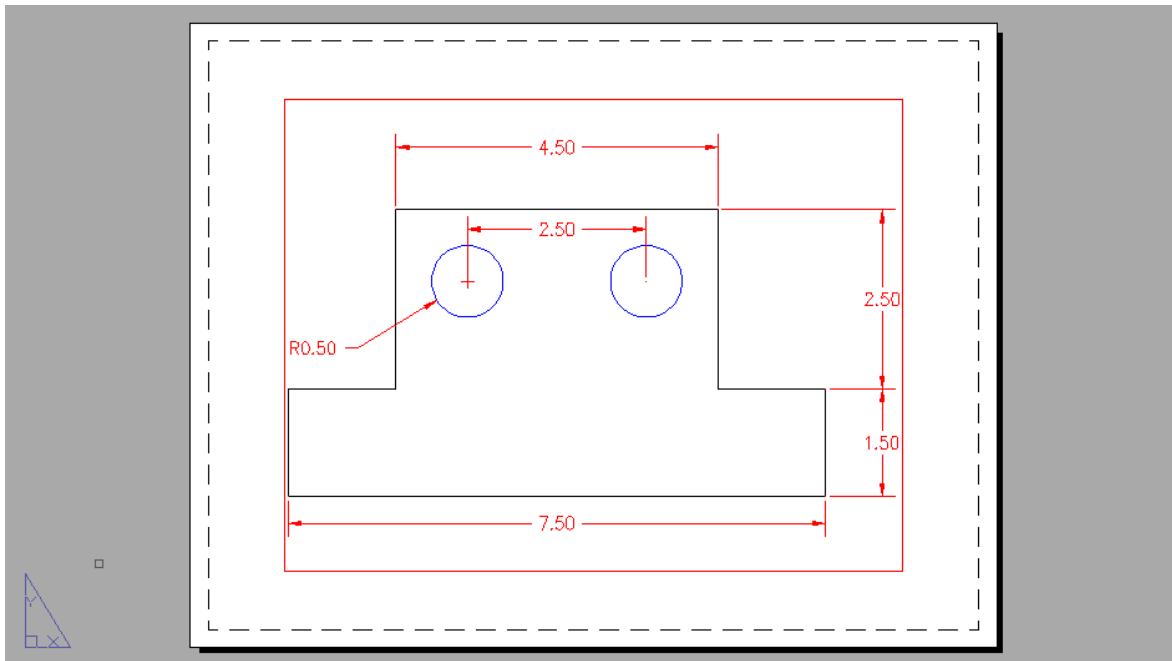


29.2 Creating Multiple Layouts

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

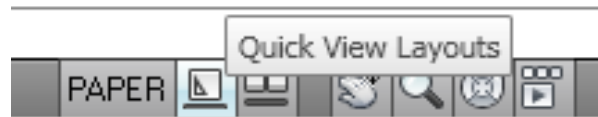


2. **Change** the name of the layout.
3. **Change** the remaining Layout options for page setup and plots.

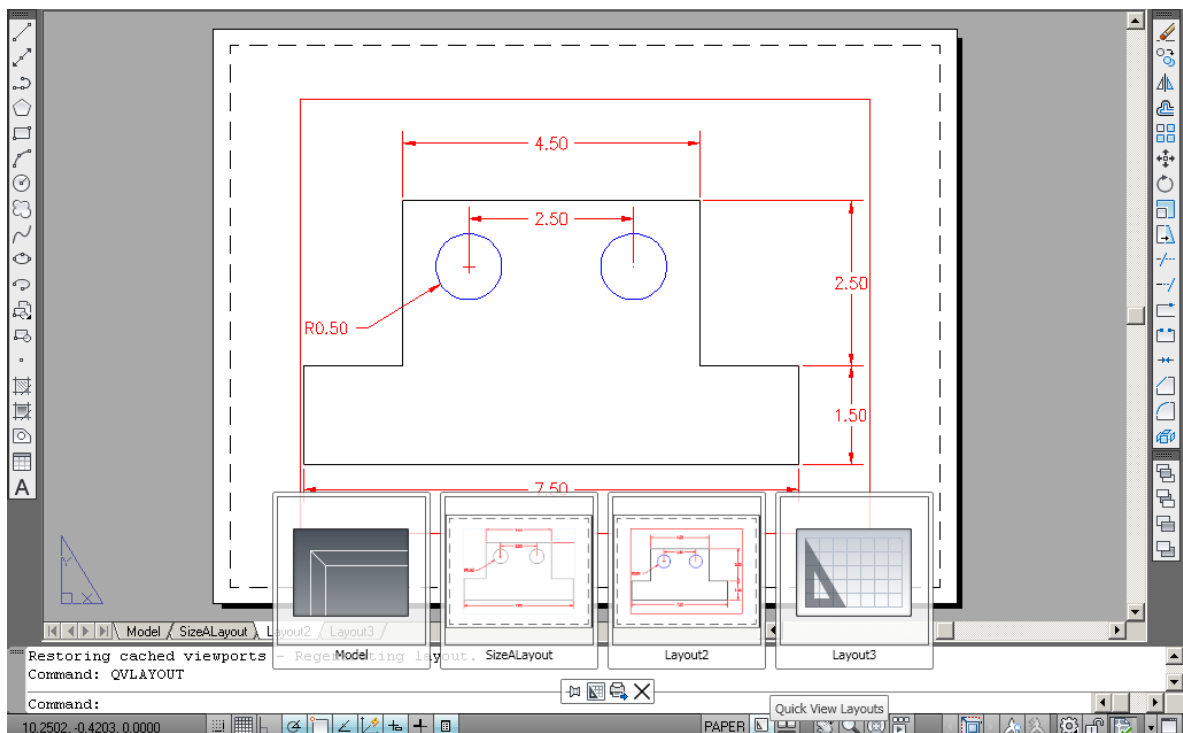


29.3 Quick View Layouts

Choose Quick View Layouts from the status bar.



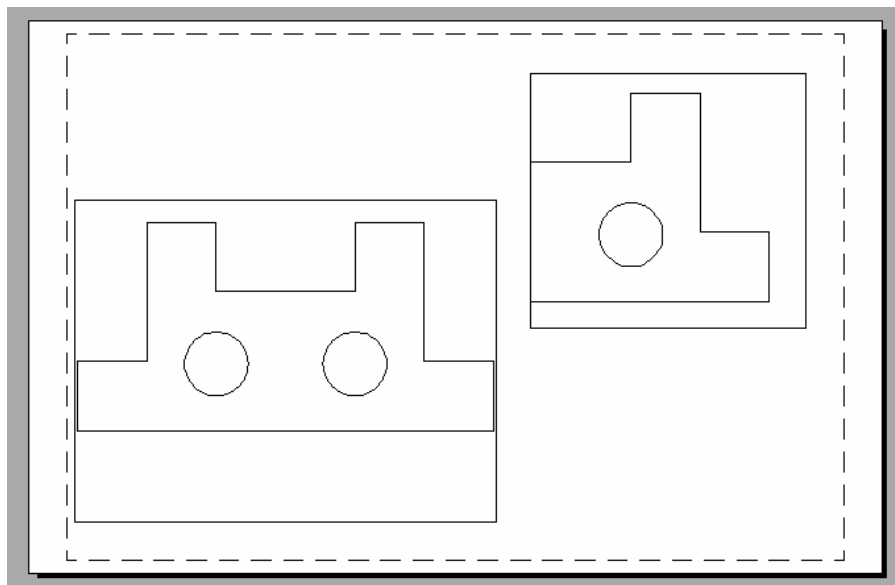
Choose a layout to view.



29.4 Mview Command

- 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.

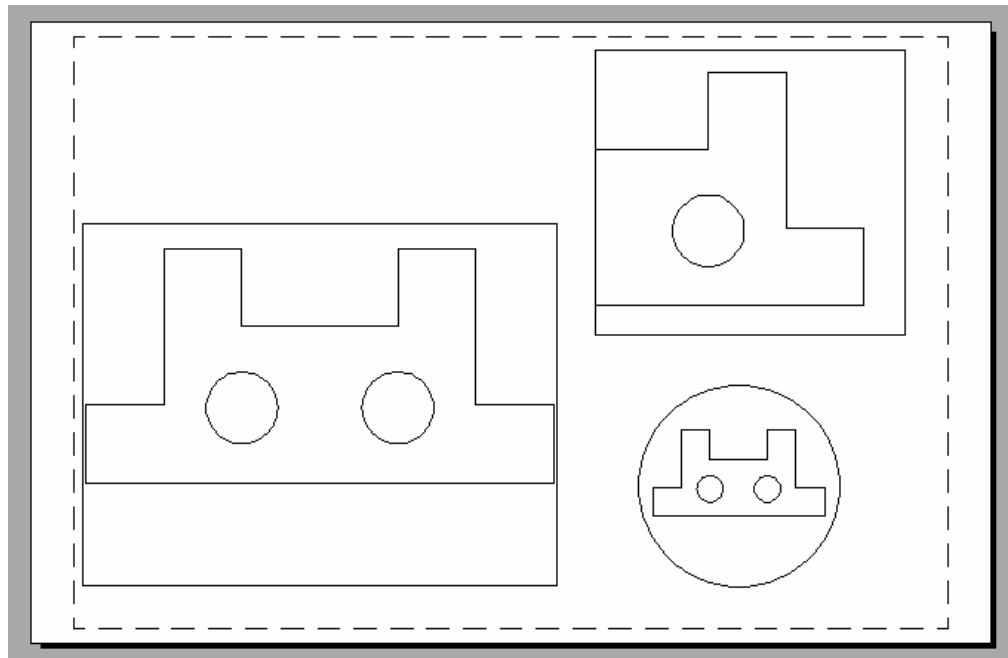
1. **Choose** View, Viewports, 1 Viewport.
or
2. **Type** Type MVIEW at the command prompt.
Command: **MVIEW or MV**
ON/OFF/Hideplot/Fit/2/3/4/Restore/<<First Point>>:**P1**
Other corner: **P2**



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

29.5 Irregular Shaped Viewports

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.



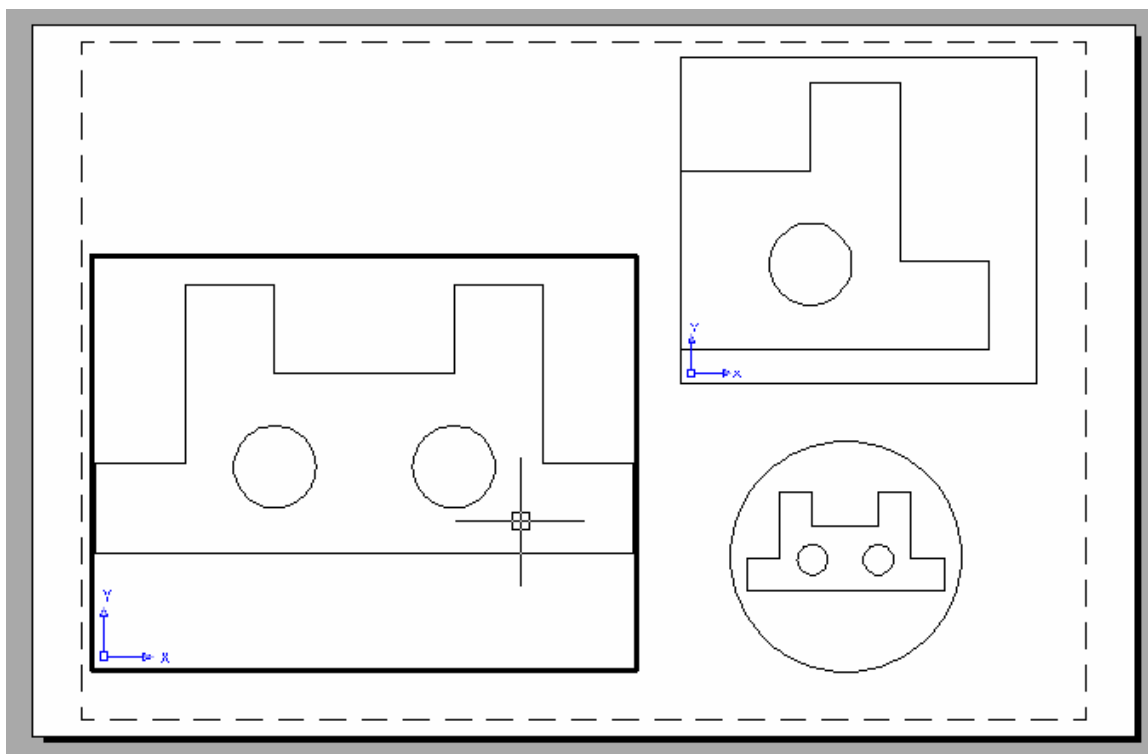
29.6 Model Space

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** "PAPER" on the Status Bar to toggle to model space.



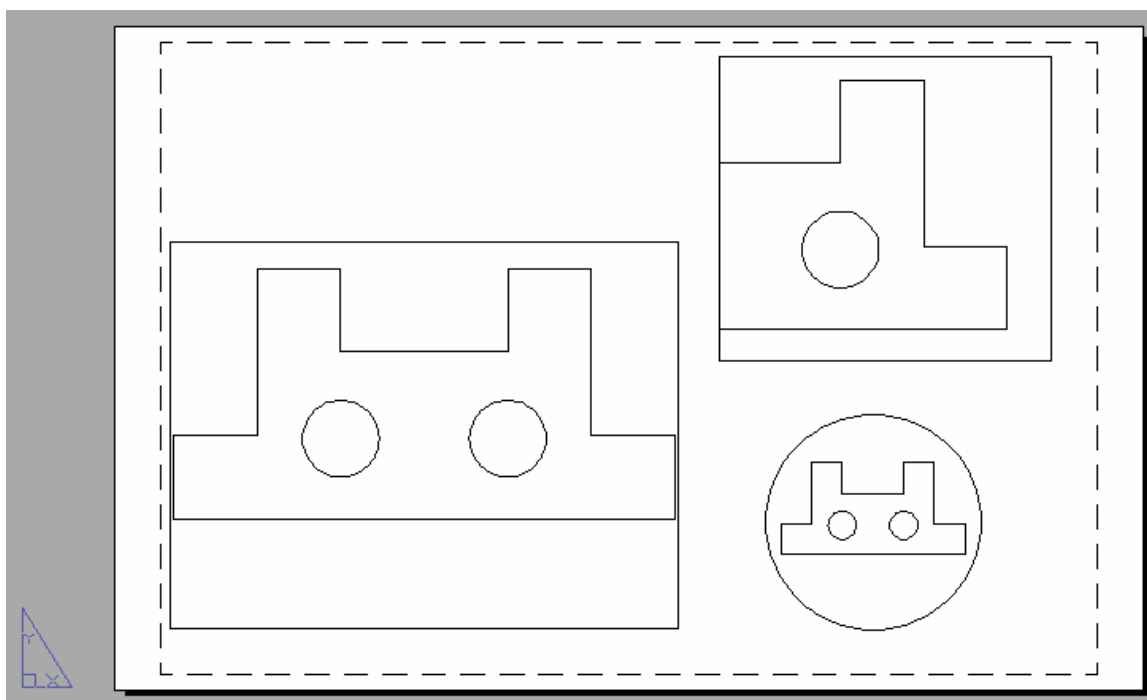
Notice the UCS icon will appear in each of the mviews when you enter model space.



29.7 Paper Space

PSPACE mode should be entered to create a border, a title, mviews, and annotations only. This environment is used to lay out a 2 dimensional 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** "MODEL" on the Status Bar to toggle to paper space.



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



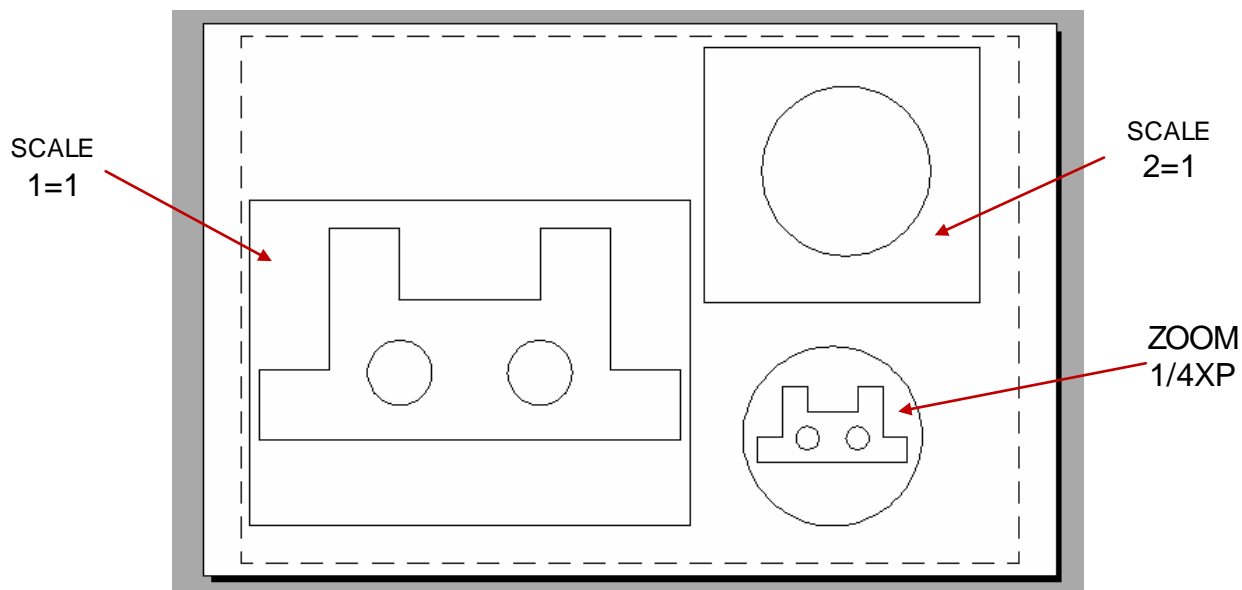
29.8 Scales - Zooming in Model Space

- 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 relative to paper space units. For example, entering .5xp displays model space at half the scale of paper space units.
- If you want to plot the model at 1/4"=1', type ZOOM 1/48XP . If you want to plot a part at 3 times, type ZOOM 3XP.

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**

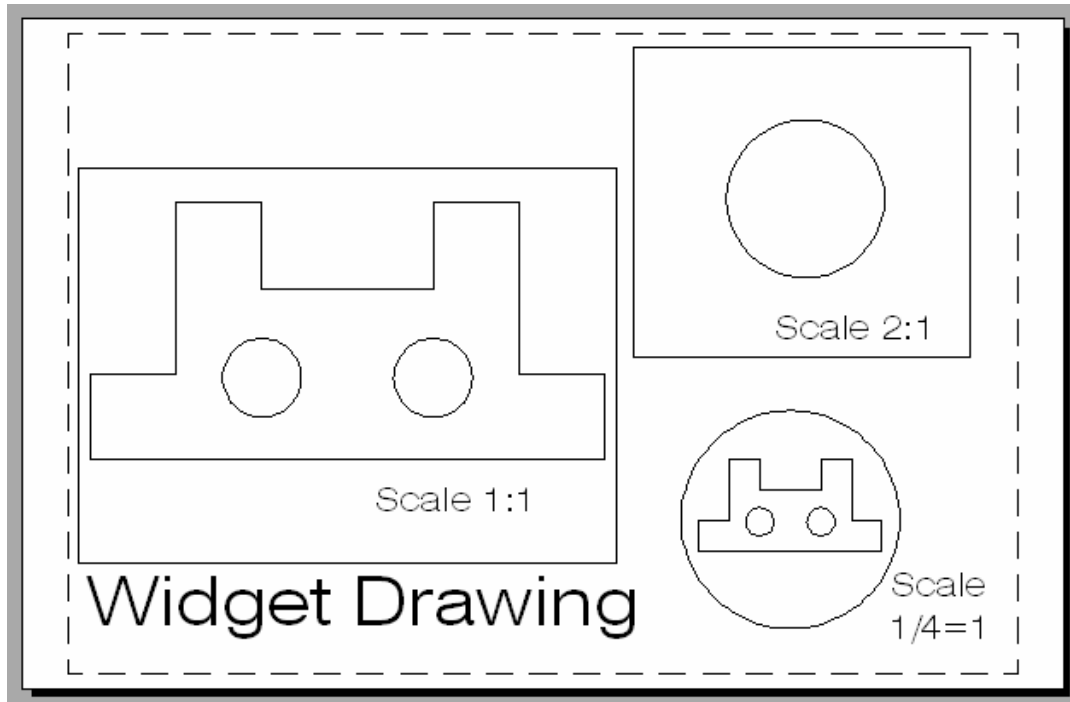


You can also change the scale from the Viewports Toolbar.



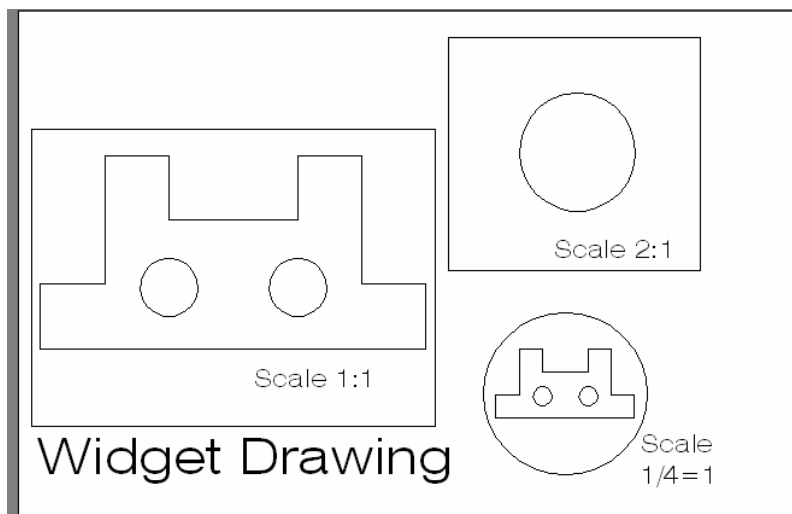
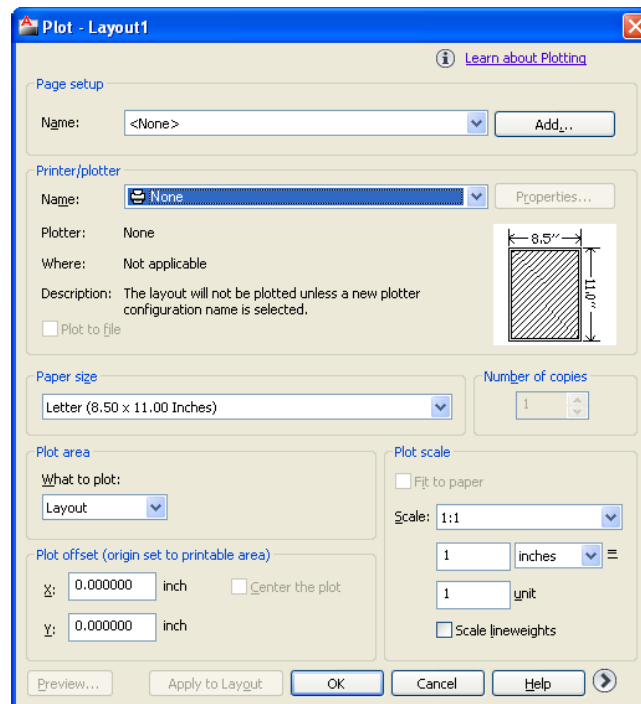
29.9 Adding Text in Paper Space

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



29.10 Plotting in Paper Space

- 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.



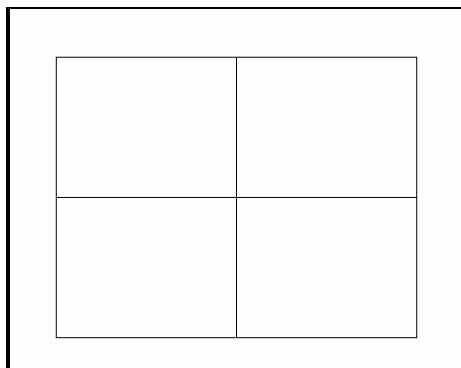
29.11 Tilemode

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 floor tiles. Each viewport butts up against the next. The viewports fill the screen. They can only be plotted one at 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**

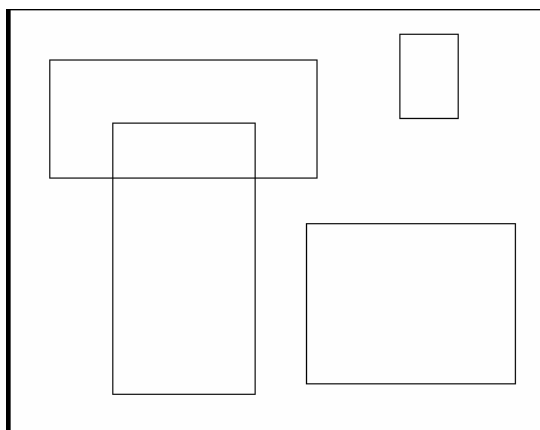


When tilemode is set to "0", the new metaview (mview) viewports can be used. Mviews can be any size or place on the screen. They may overlap. More than one mview can be plotted at a time. Each mview can be quickly turned on or off.

2. Type

Command: TILEMODE

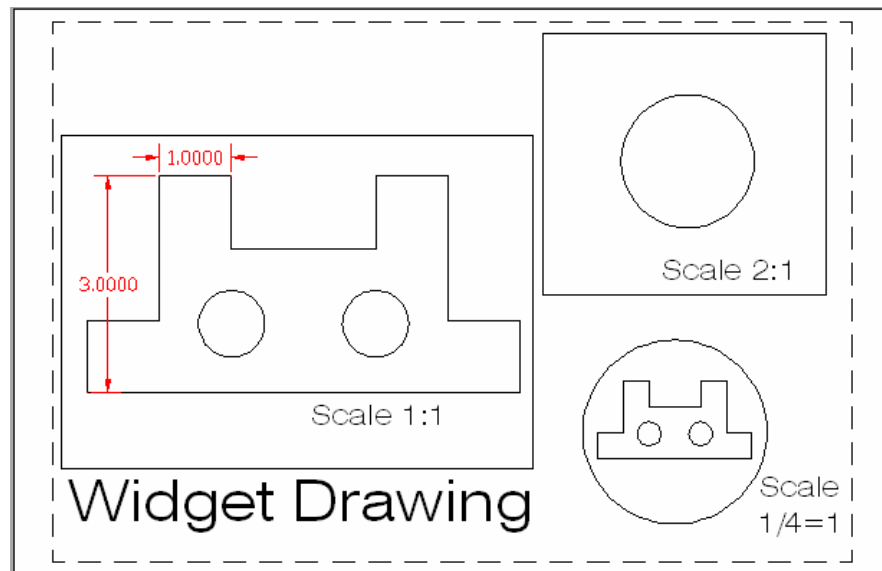
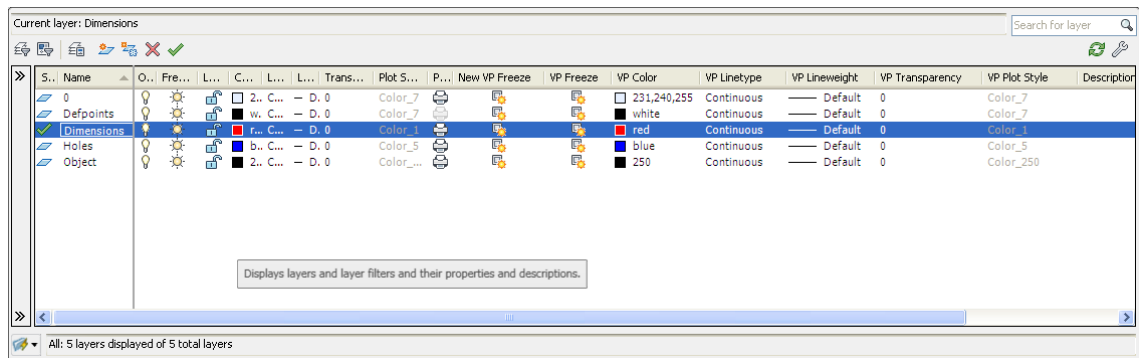
New value for tilemode <1> : 0



29.12 Viewport Layers

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.

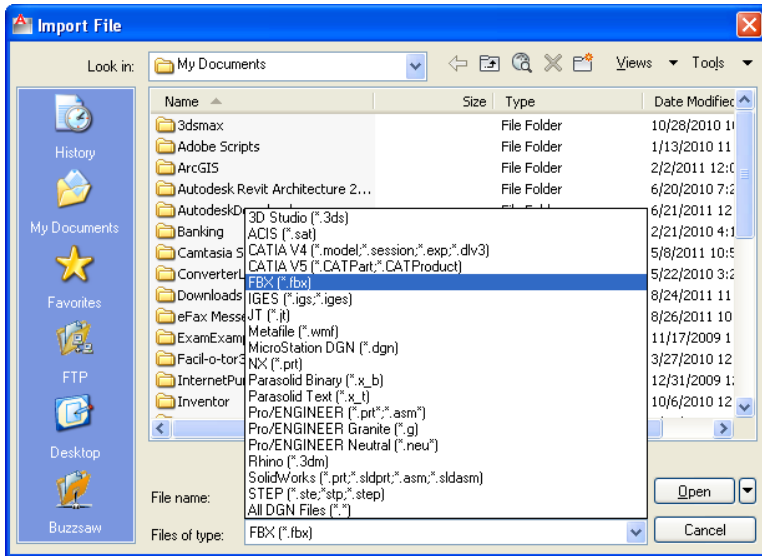


Chapter 30

Data Management

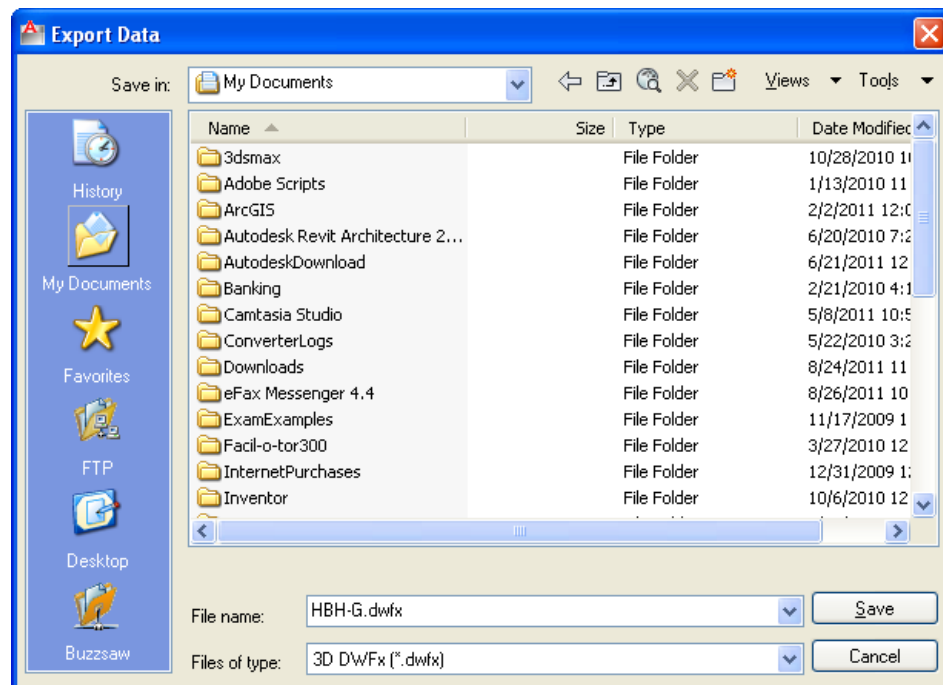
Importing Files 32.1

1. **Choose** File, Import.
or
2. **Type** Import at the command prompt. Command: **Import**



Exporting Files 30.2

1. **Choose** File, Export
2. **Type** Export at the command prompt.
Command: **Export**



3D DWF (*.dwf)
 3D DWFx (*.dwfx)
 FBX (*.fbx)
 Metafile (*.wmf)
ACIS (*.sat)
 Lithography (*.stl)
 Encapsulated PS (*.eps)
 DXX Extract (*.dxx)
 Bitmap (*.bmp)
 Block (*.dwg)
 V8 DGN (*.dgn)
 V7 DGN (*.dgn)
 IGES (*.iges)
 IGES (*.igs)