

Birzeit University

Faculty of Engineering

Mechanical Engineering Department

Mechanical Drawing ENME 223

AutoCAD ENGLISH 2004

For more information

<http://www.alqudspub.com>

<http://www.cadmagazine.net/>

<http://www.cadtutor.net/>

<http://www.dailyautocad.com>

What is CAD?

Computer Aided Design (CAD) is simply, **design** and **drafting** with the aid of a computer. **Design** is creating a real product from an idea. **Drafting** is the production of the drawings that are used to document a design. CAD can be used to create 2D or 3D computer models. A CAD drawing is a file that consists of numeric data in binary form that will be saved onto a disk.

Why should you use CAD?

Traditional drafting is repetitious and can be inaccurate. It may be faster to create a simple "rough" sketch by hand but larger more complex drawings with repetitive operations are drawn more efficiently using CAD.

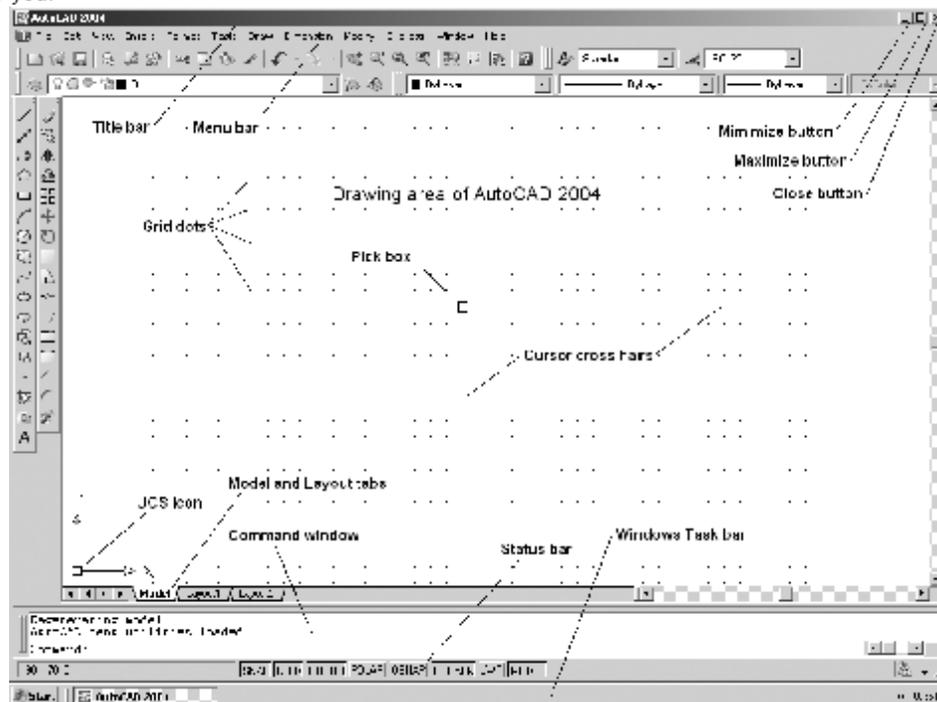
Why use AutoCAD?

AutoCAD is a computer aided design software developed by Autodesk Inc. AutoCAD was first introduced in 1982 the year it is estimated that there were over 4million AutoCAD users worldwide.

What this means to you is that many employers are in need of AutoCAD operators. In addition, learning AutoCAD will give you the basics for learning other CAD packages because many commands, terms and concepts are used universally.

GETTING FAMILIAR WITH THE AUTOCAD WINDOW

Before you can start drawing you need to get familiar with the AutoCAD window. In the following lessons, I will be referring to all of the areas described below. So it is important for you to understand each of them. But remember, this page will always be here for you.



Toolbars

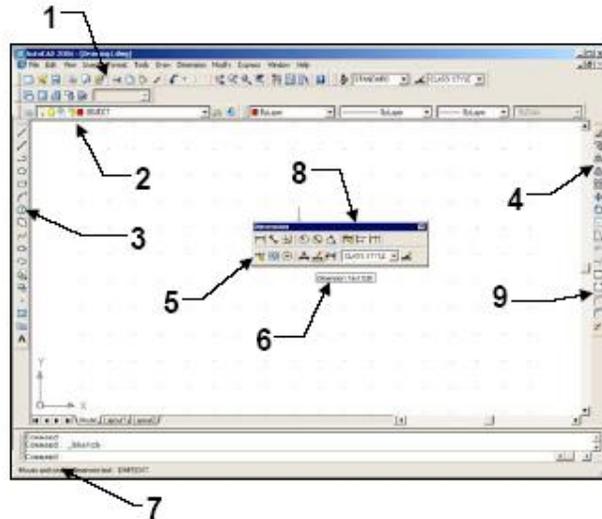
Autocad provides several toolbars to access frequently used commands.

The (1) **Standard**, (2) **Object Properties**, (3) **Draw**, and (4) **Modify** toolbars are displayed by default.

Toolbars contain **Icon Buttons** (5).

Tool Tip (6)

Help Message (7)



Toolbars can be **Floated** or **Docked**. **Floating** (8) and **Docked** (9) **Floating** toolbars (8) move freely in the drawing area and can be resized.

Docked toolbars (9) are locked into place along the top, bottom or sides of the Autocad window.

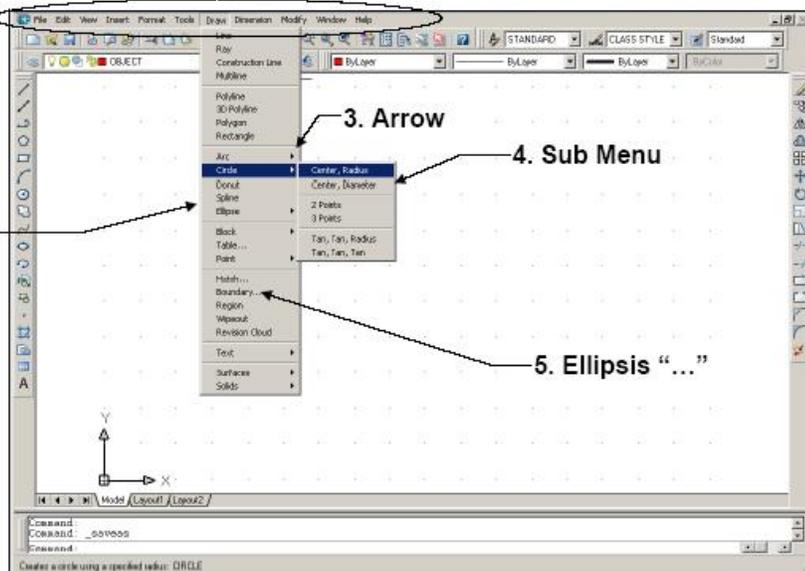
PULL-DOWN “MENU BAR”

(1) The pull-down “MENU BAR” is located at the top of the screen.



(1) Pull-down “MENU BAR”

(2) Pull-down Menu



The Function Keys

Many of the modes described above can be controlled quickly using the keyboard function keys. In most cases this is quicker than using a pull-down or the command line. The function keys are arranged along the top of your keyboard. AutoCAD uses function keys F1 to F11. Their use is described below.

F1	Help	The F1 key on your keyboard brings up the "AutoCAD Help
F2	Flipscreen	The F2 key is used to toggle (turn on and off) the AutoCAD text window.
F3	Osnap	The F3 toggles running Object Snaps on and off.
F4	Tablet	The F4 key on your keyboard toggles tablet mode on and off.
F5	Isoplanes	The F5 key cycles through the Isoplanes. The options are Left, Top and Right.
F6	Coordinate Display	The F6 key is a three way toggle which changes the coordinate reading in the status bar.
F7	Grid	The F7 key is used to toggle grid mode on and off.
F8	Ortho	The F8 key on your keyboard can be used to toggle Ortho (orthogonal) mode on and off.
F9	Snap	The F9 key can be used to toggle Snap mode on and off.
F10	Polar	The F10 key is used to switch polar tracking off and on.
F11	Otrack	The F11 key toggles object snap tracking on and off.

SPECIAL KEY FUNCTIONS

Escape Key Cancels the current command, menu or Dialog Box.

Enter Key Ends a command, or will repeat the previous command if the command line is blank.

Space Bar Same as the Enter Key, except when entering text.

Cautions

Never work for a long time before saving a drawing.

Always save a drawing file on more than one disk.

Q. How to convert a .BAK file to a .DWG?

A. Just Rename the backup file to a drawing file with the **.DWG** extension.

DRAWING LIMITS

A4

Command: **LIMITS** **8**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: **8**

Specify upper right corner <420.0000,297.0000>: **297,210** **8**

A3

Command: **LIMITS** **8**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: **8**

Specify upper right corner <297.0000,210.0000>: **420,297** **8**

Command: **Z** **8**

ZOOM

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **A** **8**

Regenerating model.

Units & Scales

Introduction

You cannot start creating sensible drawings with AutoCAD until you are familiar with scale, units and the commands you use to control them. This tutorial discusses these concepts, starting with the two most commonly asked questions in this subject area.

At what scale should I draw?

As a general rule, everything you draw with AutoCAD will be drawn full size. This often comes as quite a surprise to those who are new to CAD and have spent a number of years working on a drawing board. When you start drawing with AutoCAD you do not have to decide upon a drawing scale as you do when using a drawing board. When drawing on paper you must decide do draw at say, 1:20 or 1:200 depending upon the size of the object that you are drawing so that your scaled drawing will fit on the drawing sheet, be that A3 or A1. **In AutoCAD you do not need to decide upon a drawing scale until you come to print the drawing and because the scaling of your drawing takes place at the printing stage, you can create drawings at a scale of 1:1.**

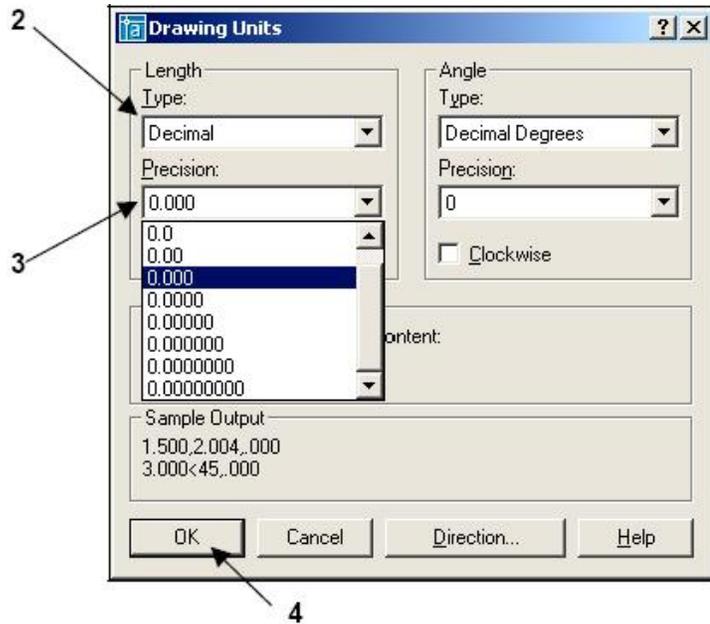
Am I drawing in metres or millimetres?

Most people who use AutoCAD, draw using **decimal drawing units**. What these drawing units represent is entirely up to the individual. However, you must decide what units you will use before you start drawing. One drawing unit could represent one millimetre, one **centimetre**, one **metre**, **kilometre**, **mile**, furlong or fathom. It is entirely up to you.

To change the unit type, you must use the Drawing Units dialogue box.

Units Control

You now need to select what unit of measurement you want to work with. Such as **Decimal** (0.0000) or **Architectural** (0'- 0"). Next you should select how precise you want the measurements. This means, do you want the measurement rounded to a 3 place decimal or the nearest 1/8".



Dialog box "Drawing Units"

Unit Precision

How to set the **units** and **Precision**.

1- Select the units command using one of the following:

Type **UNITS** 8
Pulldown > Format >Units

And the dialog box "Drawing Units" will appear

2- Select the appropriate Type such as: **Decimal** or **Architectural**.

3- Select the appropriate **Precision** associated with the "type".

4- Select the **OK** button to save your selections.

Drawing Lines,Circles and RECTangles by using the curser only

Lines are probably the most simple of AutoCAD objects. Using the Line command, a line can be drawn between any two points picked within the drawing area. Lines are usually the first objects you will want to draw when starting a new drawing. A line can be one segment or a series of connected segments. Each segment is an individual object.



One segment



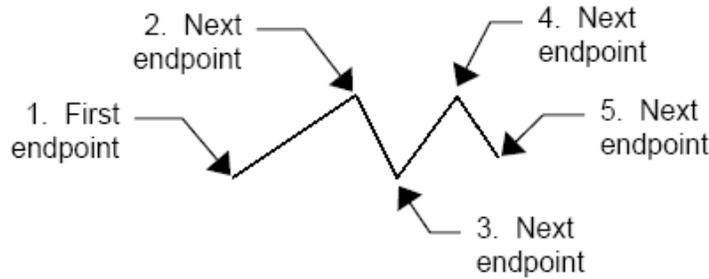
Series of connected segments

Lines are drawn by specifying the locations for the endpoints.

Move the curser to the location of the "first" endpoint then press the left mouse button.

Move the curser again to the "next" endpoint press the left mouse button.

Continue locating "next" endpoints until you want to stop.



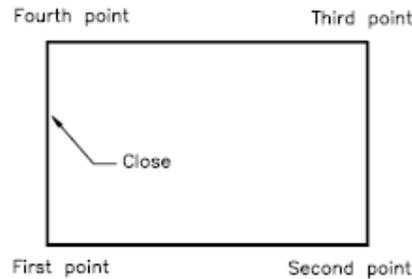
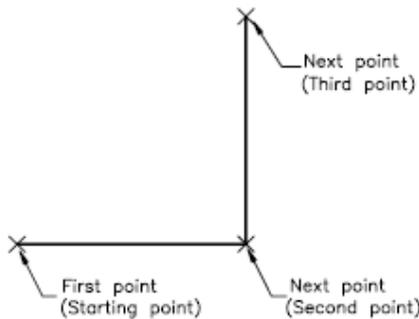
Drawing Lines Using Continue

To draw a line segment between any two points **P₁** and **P₂**

Command: **LINE** ⌨
Specify first point: (pick **First point**) ⌨
Specify next point or [Undo]: (pick **Second point**) ⌨
Specify next point or [Undo]: (pick **Third point**) ⌨
Specify next point or [Undo]: ⌨ (to end)

To draw a rectangle using the **LINE** command

Using Continue and Close
Command: **LINE** ⌨
Specify first point: (pick **First point**) ⌨
Specify next point or [Undo]: (pick **Second point**) ⌨
Specify next point or [Undo]: (pick **Third point**) ⌨
Specify next point or [Undo]: (pick **Fourth point**) ⌨
Specify next point or [Undo]: **C** ⌨ (Joins the fourth point with the first point)



Drawing lines using the **LINE** command with Continue and Close.

You can continue picking points and AutoCAD will draw a straight line between each picked point and the previous point. Each line segment drawn is a separate *object* and can be moved or erased as required. To end this command, just hit the **↵** key on the keyboard.

There are 3 ways to **Stop** drawing line:

- Press **ENTER** key
- Press **SPACE BAR**
- Press the **RIGHT MOUSE BUTTON** then select **ENTER** from the short cut menu.

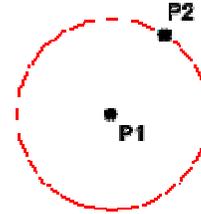
To draw perfectly **Horizontal** or **Vertical** lines select the **ORTHO** mode by clicking on the **ORTHO** button on the **Status Bar** or pressing **F8**

The **Circle** command is used to draw circles. There are a number of ways you can define the circle. The default method is to pick the centre point and then to either pick a second point on the circumference of the circle or enter the circle radius at the keyboard.

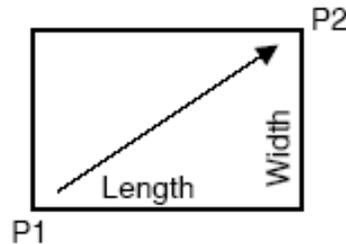
Command: **Circle** ⌨

Specify center point for circle or [3P/2P/Ttr (tan tan, radius)]: (pick **P₁**) ⌨

Specify radius of circle or [Diameter]: (pick **P₂** or enter the exact radius) ⌨



The **Rectangle** command is used to draw a rectangle whose sides are vertical and horizontal. The position and size of the rectangle are defined by picking two diagonal corners. The rectangle isn't really an AutoCAD object at all. It is, in fact, just a closed polyline which is automatically drawn for you.



Command: **REC** ⌨

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: (pick **P₁**) ⌨

Specify other corner point or [Dimensions]: (pick **P₂**) ⌨

SAVING A DRAWING

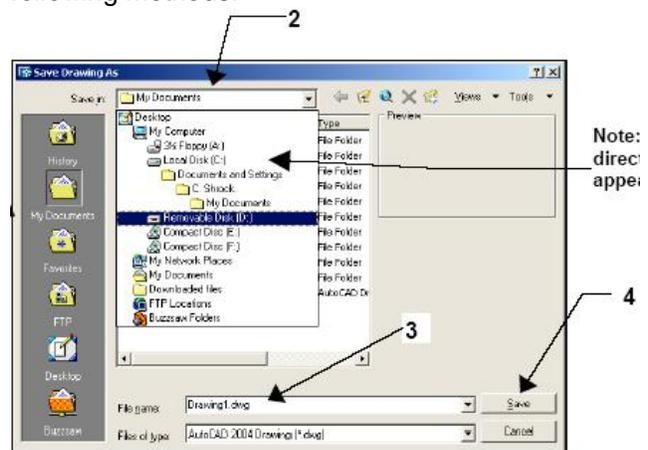
After you have completed a drawing, it is very important to save it. Learning how to save a drawing correctly is almost more important than making the drawing. If you can't save correctly, you will lose the drawing and hours of work.

There are two command for saving a drawing. **Save** and **Save As**. I prefer to use **Save As**. The **Save As** command always pauses to allow you to choose where you want to store the file and what name to assign to the file. This may seem like a small thing, but it has saved me many times from saving a drawing on top of another drawing by mistake. The **Save** command will automatically save the file either back to where you retrieved it or where you last saved a previous drawing. Neither may be the correct destination. So play it safe, use **Save As** for now.

1. Start the command by using one of following methods:

Type **Save As** ⌨
Pulldown > File > SaveAs
Toolbar: No icon for Save As, only for Save (Don't use it)

And the dialog box " **Save As**" will appear



2. Select the appropriate drive and directory from the “**Save As**” box.
This is where your drawing will be saved)
3. Type the new drawing file name in the “**File Name**” box.
4. Select the “**Save**” button.

Coordinate Systems

1- Absolute coordinates

Draw a line **AB**, **A(20,10)**, and **B(30,40)**.

Command: **LINE** **8**

Specify first point: **20,10** **8**

Specify next point or [Undo]: **30,40** **8**

LINE I

Specify next point or [Undo]: **8**

2- Relative Coordinates

Relative rectangle coordinates

Draw a line **AC**, **A(20,10)**, **C** has a distance $x = 30$, and $y = 40$ from point **A**.

Command: **LINE** **8**

Specify first point: **20,10** **8**

Specify next point or [Undo]: **@30,40** **8**

LINE II

Specify next point or [Undo]: **8**

Relative polar coordinates

Draw a line **AD**, **A(20,10)**, **D** has a distance **50** from point **A** and 30° slope from horizontal.

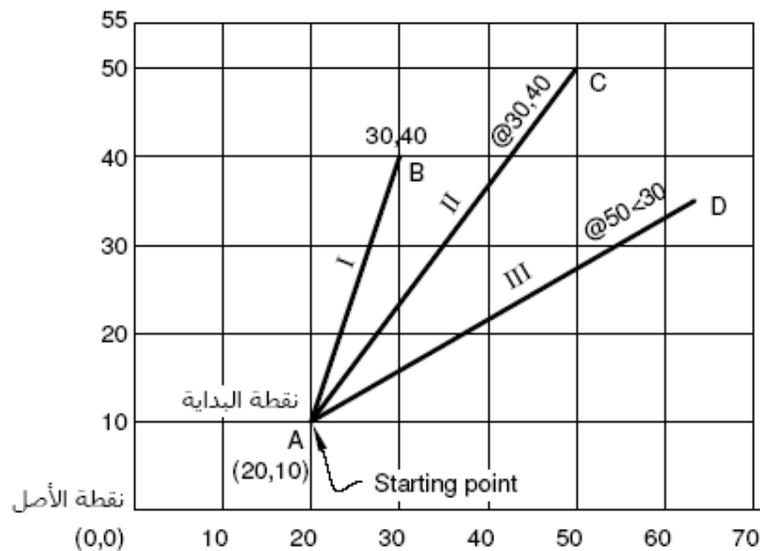
Command: **LINE** **8**

Specify first point: **20,10** **8**

Specify next point or [Undo]: **@50<30** **8**

LINE III

Specify next point or [Undo]: **8**



COORDINATE SYSTEMS

Drawing Aids

Introduction

Drawing with AutoCAD is really just like drawing on a drawing board. Most newcomers to Computer Aided Design assume that they will need to learn how to draw all over again. In fact, many of the drawing aids that AutoCAD provides are analogous to traditional drafting tools. Just as you have a parallel motion and set squares to help you draw horizontal and vertical lines on a drawing board, AutoCAD has similar drawing aids which can help you to draw horizontal and vertical lines on a computer. This means that in many respects, the drawing techniques are very similar. If you ever get stuck, think how you would complete a task on a drawing board and then look for a similar way to do it with AutoCAD.

Ortho Mode

Status Bar 

Pull-down None

Keyboard : **ORTHO** or **F8**

Ortho is short for **Orthogonal**, which means either vertical or horizontal. Like the other options on the status bar, Ortho is not really a command; it is a drawing *mode* which can either be turned on or off. Ortho mode can be *toggle*d on or off in one of three ways.

Ortho can also be toggle'd on and off using the **F8** Function key.

The quickest way is just to click on the ORTHO button on the status bar. The appearance of the button tells you whether Ortho is currently turned on or turned off. When Ortho is turned on, the ORTHO button appears pressed in. You can see how this appears by looking at the status bar illustration below. In the illustration, Ortho is turned on but Grid and Snap are turned off.



Finally, you can also type ORTHO at the command prompt as in the command sequence below. Using Ortho is the equivalent of using your parallel motion and set square on a drawing board. With Ortho mode turned on you can only draw lines which are either vertical or horizontal. Turn Ortho mode on now and draw some lines to get a feeling for how it works.

Command: **ORTHO** ↵

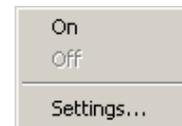
Enter mode [ON/OFF] <OFF>: (type **ON** or **OFF** then press ↵)

The Drawing Grid

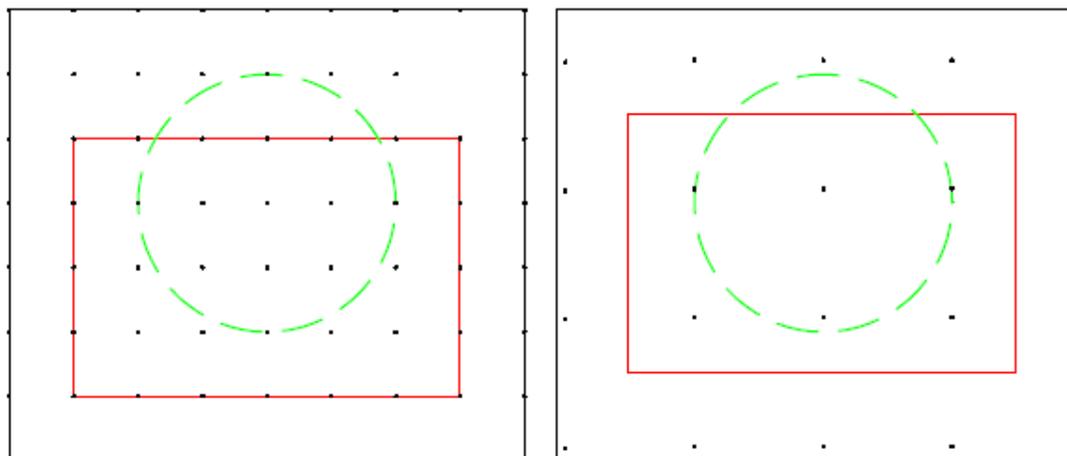
Status Bar  (right-click for settings)

Pull-down None

Keyboard **GRID** or **F7**



The drawing grid is a regular pattern of dots displayed on the screen which acts as a visual aid, it is the equivalent of having a sheet of graph paper behind your drawing on a drawing board. You can control the grid spacing, so it can give you a general idea about the size of drawn objects. It can also be used to define the extent of your drawing.



Grid spacing set to 10x10 drawing units
Snap ON

Grid spacing set to 20x20 drawing units.
Snap OFF

Command: **Grid** ↵

Specify grid spacing(X) or [ON/OFF/Snap/Aspect] <10>: ↵

Although you can use the command line to control the visibility of the grid by using the "ON" and "OFF" options this is more easily achieved using the **F7** key or, better still, by clicking the **GRID** button on the **status bar**.

Command: **Grid** ↵

Specify grid spacing(X) or [ON/OFF/Snap/Aspect] <10>: **ON** ↵

Grid mode and X/Y spacing can also be set using the **Drafting Settings dialogue box**. You can access grid Grid mode and X/Y spacing can also be set using the **Drafting Settings dialogue box**. You can access grid settings by right-clicking the Grid button on the status bar and selecting Settings... from the menu. You can also do this from the pull-down menu,

By using **Relative Rectangle Coordinates** Draw above **TEMPLATE 1** starting from **P₁** if the **absolute coordinate** of **P₆(0,0)**.

Specify next point or [Undo]: **@0,5 8**

Command: **Snap** ↵

settings by right-clicking the Grid button on the status bar and selecting Settings... from the menu. You can also do this from the pull-down menu,

Tools>Drafting Settings...

and click on the "Snap and Grid" tab.

Snap Mode

Status Bar **SNAP** (right-click for settings)

Pull-down None

Keyboard **SNAP** or **F9**



Snap mode takes AutoCAD one step further than the drawing board. With Snap mode turned on AutoCAD only allows you to pick points which **lie on a regular grid**. The Snap grid is completely independent of the display grid. However, the Grid spacing and Snap spacing are usually set to the same value to avoid confusion.

Command: **Snap** ↵

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <10>: ↵

When Snap mode is turned on and the Grid is displayed, the Snap and Grid spacings are the same and the crosshairs will jump from one grid point to another as you move across the screen. This makes it very easy to draw objects which have a regular shape. The Snap command is used to set the snap spacing and to toggle Snap mode.

Command: **Snap** ↵

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <10>: **ON** ↵

The Snap option allows you to automatically set the grid spacing to the current snap spacing. You can also change the aspect ration of the grid. By default, the X and Y spacing of the Grid are the same, resulting in a regular square matrix of grid points. But you can display a grid with different X and Y spacing by using the "Aspect" option.

Example 1

Command: **LINE** 8

Specify first point: **Pick P₁**

Specify next point or [Undo]: **@7,0** 8

Specify next point or [Undo]: **@0,5** 8

Specify next point or [Close/Undo]: **@-3,0** 8

Specify next point or [Close/Undo]: **@0,2** 8

Specify next point or [Close/Undo]: **@3,0** 8

Specify next point or [Close/Undo]: **@4,0** 8

Specify next point or [Close/Undo]: **u** 8

Specify next point or [Close/Undo]: **@0,4** 8

Specify next point or [Close/Undo]: **@-3,3** 8

Specify next point or [Close/Undo]: **@-2,-1** 8

Specify next point or [Close/Undo]: **@-2,1** 8

Specify next point or [Close/Undo]: **@-3,-3** 8

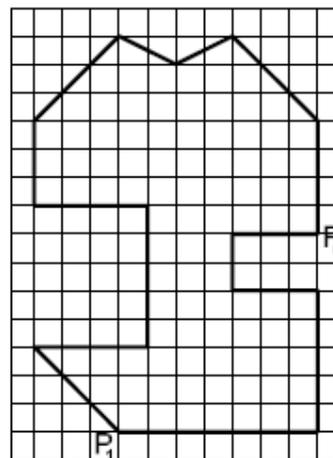
Specify next point or [Close/Undo]: **@0,-3** 8

Specify next point or [Close/Undo]: **@4,0** 8

Specify next point or [Close/Undo]: **@0,-5** 8

Specify next point or [Close/Undo]: **@-4,0** 8

Specify next point or [Close/Undo]: **C Close** 8



TEMPLATE 1

Problem 1

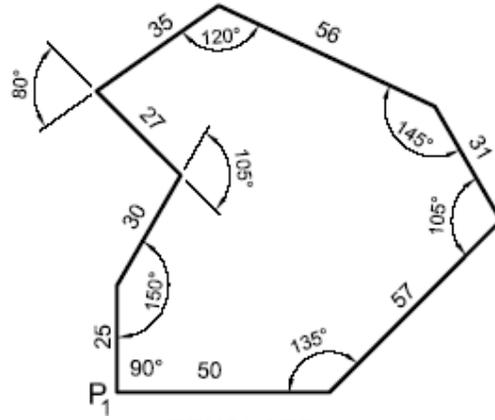
By using **Relative Rectangle Coordinates** Draw above **TEMPLATE 1** starting from **P₁** if the **absolute coordinate** of **P₆(0,0)**.

P₁		P₄		P₇		P₁₀		P₁₃	
P₂		P₅		P₈		P₁₁		P₁₄	
P₃		P₆	0.0	P₉		P₁₂		P₁₅	

Example 2

By using **Relative Polar Coordinates** Draw a **TEMPLATE 2** starting from **P₁(10,0)** in a clockwise direction.

Command: **L** ↵
 Specify first point: **10,0** ↵
 Specify next point or [Undo]: **@50,0** ↵
 Specify next point or [Undo]: **@57<45** ↵
 Specify next point or [Close/Undo]: **@31<120** ↵
 Specify next point or [Close/Undo]: **@56<155** ↵
 Specify next point or [Close/Undo]: **@35<-145** ↵
 Specify next point or [Close/Undo]: **@28<-45** ↵
 Specify next point or [Close/Undo]: **@30<-120** ↵
 Specify next point or [Close/Undo]: **C** ↵



TEMPLATE 2

Problem 2:

By using **Relative Polar Coordinates** Draw **TEMPLATE 2** starting from **P₁(10,0)** in anti clockwise direction.

P₁		P₄		P₇	
P₂		P₅		P₈	
P₃		P₆			

Polar Tracking

Draw a line **BC** starting from point **B**. **B** has a horizontal distance of 100 mm from **A**, **Fig 1**.

Command: **ID** ↵
 Specify point: Pick point **A** ↵
 X = 221 Y = 185 Z = 0

Now, **A** become as a point (0,0) in Cartesian coordinates.

Command: **LINE** ↵
 LINE Specify first point: **@100<0** ↵
 Specify next point or [Undo]: **@75<90** ↵
 Specify next point or [Undo]: ↵

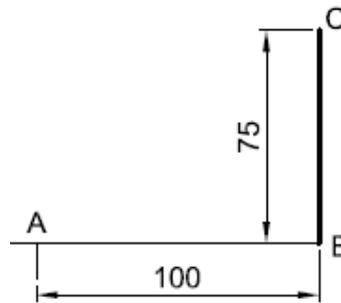


Fig. 1

Draw a line **EF** starting from point **B** to point **F** as shown in **Fig. 2**

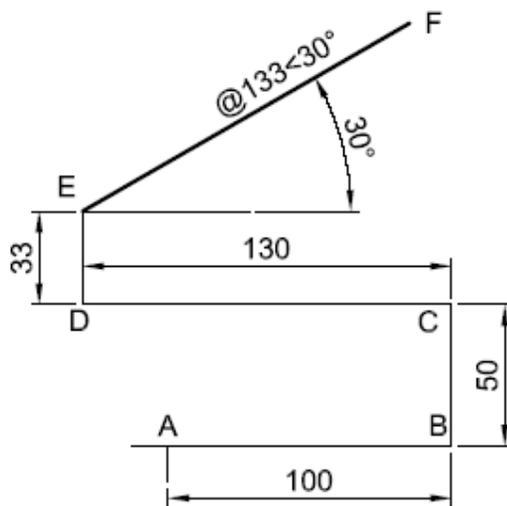


Fig. 2

Command: **LINE 8**

Specify first point: **TRACKING** ↵

First tracking point: **Pick point A** ↵

Next point (Press ENTER to end tracking): **@100<0** ↵

Next point (Press ENTER to end tracking): **@50<90** ↵

Next point (Press ENTER to end tracking): **@130<180** ↵

Next point (Press ENTER to end tracking): **@33<90** ↵

Next point (Press ENTER to end tracking): ↵

Specify next point or [Undo]: **@133<30** ↵

Specify next point or [Undo]: ↵

Direct Distance Entry DDE

Direct distance entry **DDE** enables you to draw an object, such as a line, by pointing in a particular direction with the cursor and entering a distance at the command line.

How does it work?

Example 3: Draw a series of connected lines Shown in **geometry** starting with a horizontal line from point **A**.

- 1- Start the Line command. Pick any point **A**. Or press F10 on Polar Tracking.
- 2- Use the **POLAR** button on the status bar to turn on **Polar Tracking**.
- 3- Move your cursor to the right of the first pick point.
Hold your cursor in this position and simply enter **29** at the keyboard.
When you hit the **ENTER** key, a line segment is drawn, **29** units long and in the direction you were pointing.

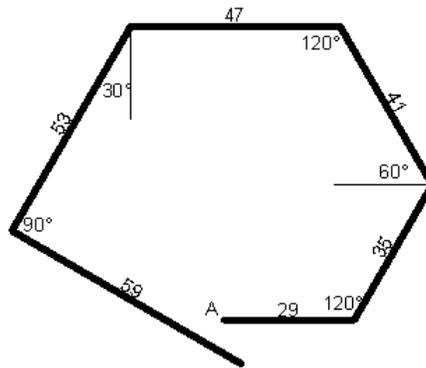
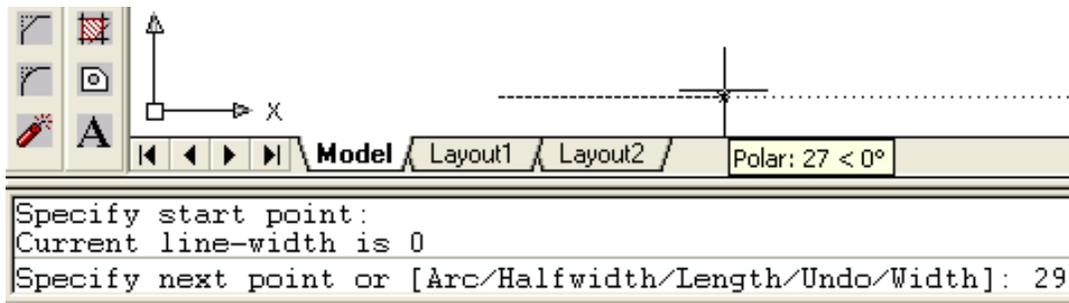


Fig 3



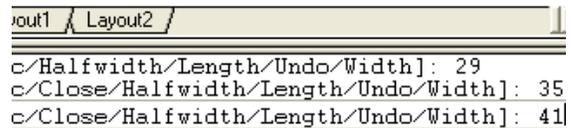
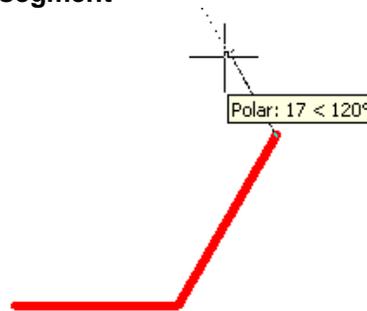
Drawing First segment

Obviously, you could vary this sequence to get different effects.

Use the **POLAR** button on the status bar to turn on **Polar Tracking** with angle of 15° .

If you only want to draw horizontal or vertical lines, you could use **Ortho** rather than **Polar Tracking**.

You might even want to turn both Ortho and Polar Tracking off and use free angles.



Drawing Second and Third segment

The Polyline Command

Toolbar 

Pull-down: **Draw** ▶ **Polyline**

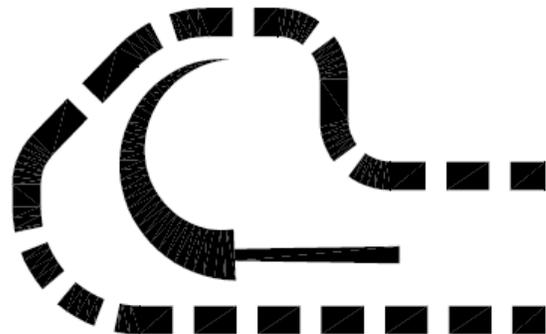
Keyboard: **PLINE** short-cut: **PL**

The **Polyline** or **Pline** command is similar to the line command except that the resulting object may be composed of a number of segments which form a single object. In addition to the two ends a polyline is said to have vertices (singular vertex) where intermediate line segments join. In practice the Polyline command works in the same way as the Line command allowing you to pick as many points as you like. Again, just hit **↵** to end. As with the Line command, you also have the option to automatically close a polyline end to end. To do this, type **C** to use the close option instead of hitting **↵**. Follow the command sequence below to see how this works.

Options

[Arc/Halfwidth/Length/Undo/Width]

Try experimenting with options such as **Arc** and **Width** and see if you can create polylines like the ones in the illustration. The Undo option is particularly useful. This allows you to unpick polyline vertices, one at a time so that you can easily correct mistakes.



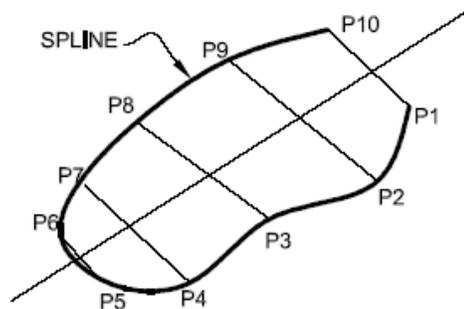
Polylines can be edited after they are created to, for example, change their width. You can do this using the **PEDIT** command, **Modify>Object>Polyline** from the pull-down menu.

The Spline Command

Toolbar 

Pull-down: **Draw>Spline**

Keyboard: **SPLINE** short-cut: **SPI**



Command: **Spline** **↵**

Specify first point or [Object]: (**Pick P1**)

Specify next point: (**Pick P2**)

Specify next point or [Close/Fit tolerance] <start tangent>:(**Pick P3**)

Specify next point or [Close/Fit tolerance] <start tangent>:(**Pick P10**) **↵ ↵ ↵**

You can create linear approximations to splines by smoothing polylines with the **PEDIT** command, **Modify>Polyline** from the pull-down menu. However, you can also turn polylines into true splines using the Object option of the Spline command.

Problem 3

Draw a **Hexagon** (Edge = 39 units) by using the **Polar Tracking** with angle of 15° .

Problem 4

Draw a **Hexagon** (Edge = 14 units) by using the **Polar Tracking** with angle of 60° .

Problem 5

Draw an **Octagon** (Edge = 39 units) by using the **Polar Tracking** with angle of 45° .

The Erase Command

Toolbar: 

Pull-down: **Modify>Erase**

Keyboard: **ERASE** or **E**

The **Erase** command is one of the simplest AutoCAD commands and is one of the most used. The command erases (deletes) any selected object(s) from the drawing.

Command: **ERASE** ↵

Select objects: (pick an object to erase)

Select objects: ↵ (to end the selection and erase the object)

**You can use the DELETE ICON to delete any object on the screen.
Pick objects then click a DELETE ICON.**

Object Selection

Before you start to use the AutoCAD Modify commands, you need to know something about selecting objects. All of the Modify commands require that you make one or more object selections. AutoCAD has a whole range of tools which are designed to help you select just the objects you need.

Selecting Objects by Picking

The most obvious way to select an object in AutoCAD is simply to pick it. When you start a Modify command such as ERASE, two things happen. First, the cursor changes from the usual *crosshairs* to the pickbox and second, you will then see the "Select objects" prompt on the command line.

Select objects:

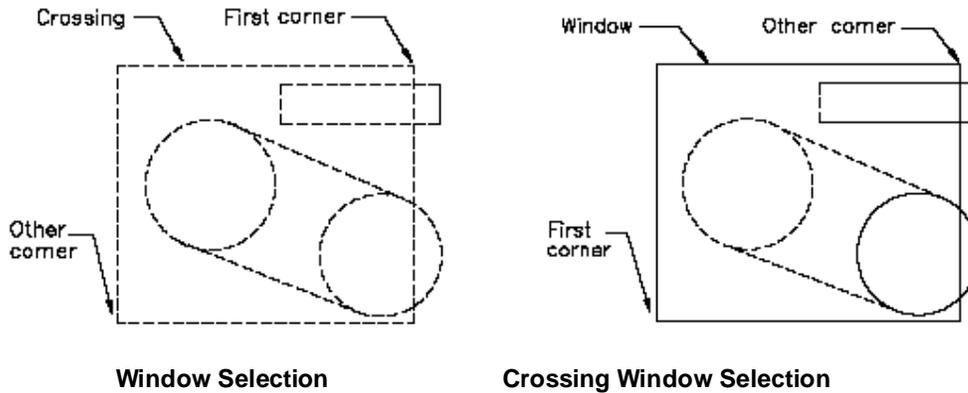
To select an object, place the pickbox over a part of the object and left-click the mouse. When the object has been picked it is *highlighted* in a dashed line to show that it is part of the current selection and the command line reports "1 found". You will now see the "Select objects" prompt on the command line again. At this point you can continue adding more objects to the current selection by picking them or you can press ↵ or the Space Bar to complete the selection.

Window Selection

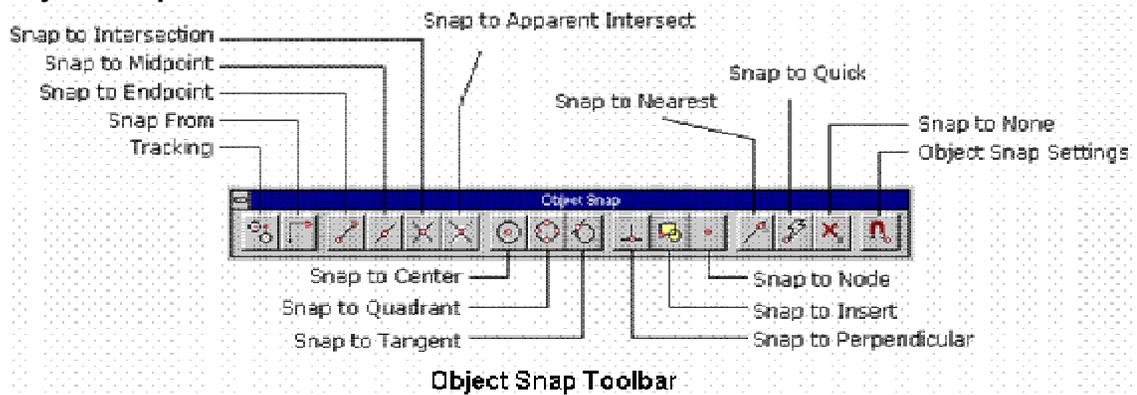
The Window option is invoked by typing **W** in response to the "Select objects" prompt. Window allows you to define a rectangle using two points in exactly the same way as the RECTANGLE command. Once the window is defined, all objects which lie entirely within the window will be selected.

Crossing Window Selection

The Crossing Window option is invoked by typing **C** at the "Select objects" prompt and is a variation of the Window command. The command sequence is exactly the same but objects are selected which lie entirely within the window and those which cross the window border.



Object Snap



Object Snaps

Object snaps (Osnaps) are one of the handiest tools in AutoCAD. Without them, you would have a lot of trouble doing even the simplest dimension accurately. Before discussing how they are used, first think about what they are. Their name, "object snaps" means that they snap to objects, or more precisely, they snap to specific parts of an object. A line for example has 3 points that you can snap to: a midpoint and two endpoints. A circle has 5 points: a center and 4 quadrants. To use Osnaps effectively, you must know what points an object has that you are able to snap to.

There are also some Osnaps that are not precise. These allow you to snap to (sometimes) arbitrary points along an object. These would include nearest, tangent, perpendicular and to some extent, intersection.

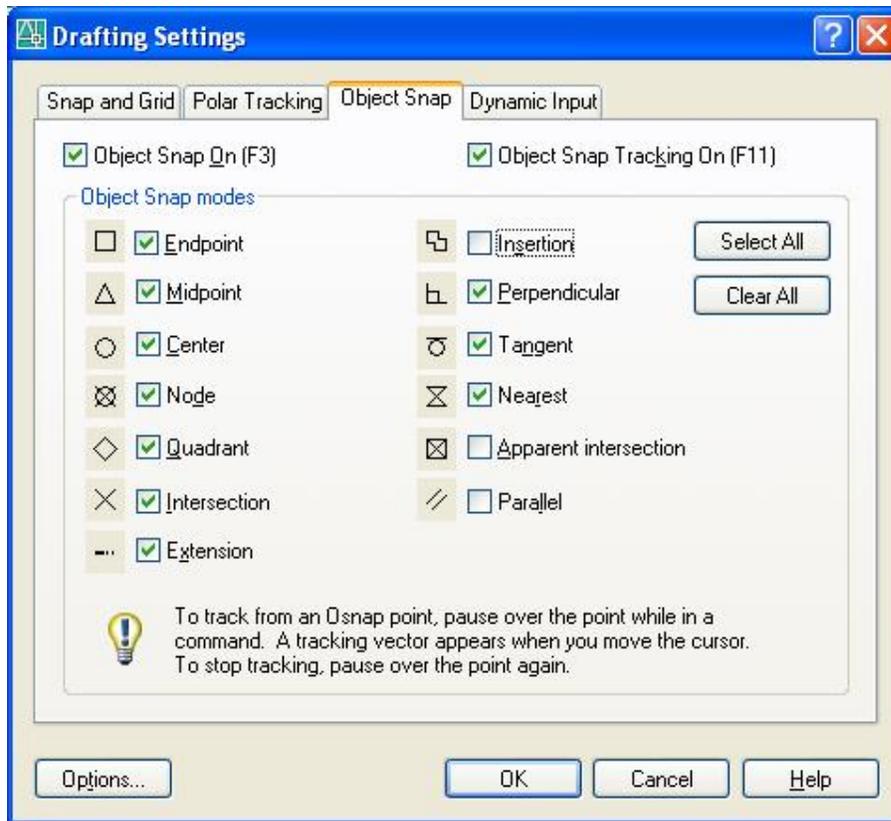
There are two different ways of working with Osnaps; in running mode or invoking them individually as required. Running Osnaps mean that certain ones are turned on and running in the background. This means that they are available when needed in the middle of a command.

Sometimes you may need one Osnap that isn't one you have selected in running mode. In this case you can turn it on for one-time use. There are a few ways of doing this: you can select the icon for the Osnap you need, you can type in the 3 letter shortcut, or you can bring up the Osnap dialog box and turn it on ('OS), or even shift+right-click and pick from the

menu. I recommend learning the 3 letter shortcuts and typing them in when required. This is generally the fastest way to get your Osnap on, plus it doesn't leave it on if you don't need it afterwards. Having the icons up your screen works, but takes up valuable screen area. Try each method and see which one works best for you.

In most cases you want to turn on all the osnaps while drawing as you could accidentally snap to a point you don't want. For example, the endpoint and a perpendicular point could be very close - so you need to be careful when using perpendicular.

Running object snaps can be configured using the Object Snap tab of the Drafting Settings dialogue box, illustrated below. This tabbed dialogue box also includes settings for Snap and Grid and for Polar Tracking



Object Snap Dialogue Box

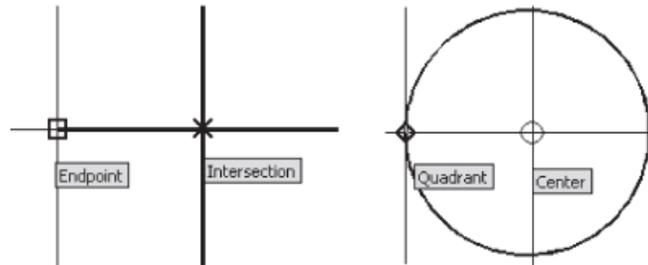
Example: Draw a Line from END of a line thru INTERsection of two lines

Command: **LINE** ↵

LINE Specify first point: **END** ↵ (Pick END of line ↵)

Specify next point or [Undo]: **INT** ↵ (Pick the intersection of two lines)

Specify next point or [Undo]: ↵



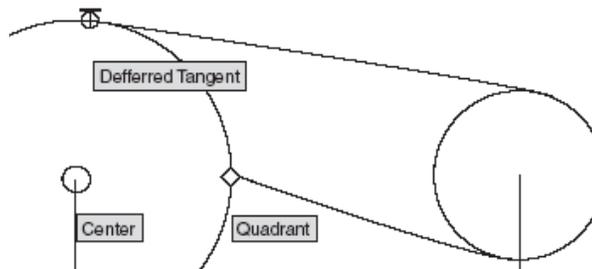
Example: Draw a Line tangent to two circles.

Command: **line** ↵

Specify first point: **tan** ↵ (Pick a point nearest to the tangent on circle)

Specify next point or [Undo]: **tan** ↵ (Pick a point nearest to the tangent on other circle)

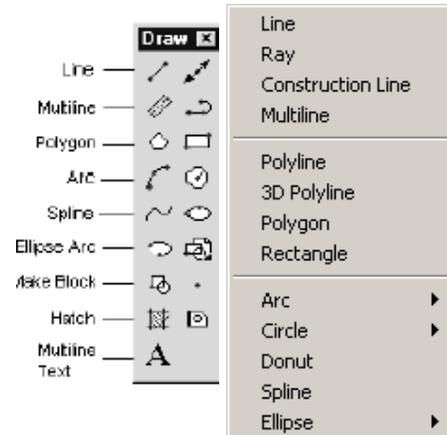
Specify next point or [Undo]: ↵



Draw Toolbar

This tutorial is designed to show you how some of the AutoCAD Draw commands work. Not all of the Draw commands that appear on the Draw toolbar are covered in this tutorial. Blocks, Hatch and Text for example are all tutorial topics in their own right!

The Draw commands can be used to create new objects such as lines and circles. Most AutoCAD drawings are composed purely and simply from these basic components. A good understanding of the Draw commands is fundamental to the efficient use of AutoCAD.



The Draw toolbar The sections below cover the most frequently used Draw commands such as **Line**, **Polyline**, **Circle**, **Polygon**, **Rectangle**, as well as the more advanced commands like **Multiline** and **Multiline Style**.

In common with most AutoCAD commands, the Draw commands can be started in a number of ways. Command names or short-cuts can be entered at the **keyboard**, commands can be started from the **Draw pull-down menu**, shown on the right or from the **Draw toolbar**.

Drawing Lines

The Line Command

Toolbar 

Pull-down **Draw > Line**, short-cut **L**

Command: **Line** ↵ (**L** ↵)

Draw a **Square**, length of side = **30** mm from the vertex **100,100**

Command: **LINE** ↵

Specify first point: **100,100** ↵

Specify next point or [Undo]: **@30<0** ↵ (OR **@30, 0** ↵)

Specify next point or [Undo]: **@30<90** ↵ (OR **@0, 30** ↵)

Specify next point or [Close/Undo]: **@30<180** ↵ (OR **@-30, 0** ↵)

Specify next point or [Close/Undo]: **C** ↵

Draw an **equilateral triangle**, length of side = **40** mm from the vertex **60,70**.

Command: **LINE** ↵

Specify first point: **60,70** ↵

Specify next point or [Undo]: **@40<0** ↵ (OR **@40, 0** ↵)

Specify next point or [Undo]: **@40<120** ↵

Specify next point or [Close/Undo]: **C** ↵

The Construction Line Command

Toolbar 

Pull-down **Draw > Xline**, short-cut **XI**

Command: **Xline** ↵ (**XI** ↵)

Command: **XI** ↵

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

Specify through point: **pick a second point**

Specify through point: **pick a point to position the line**

Drawing horizontal construction lines

Command: **XLINE** ↵

Hor/Ver/Ang/Bisect/Offset/<From point>: **H** ↵

Through point: **pick a point to position the line**

Through point: **to end or pick a point for another horizontal line** ↵

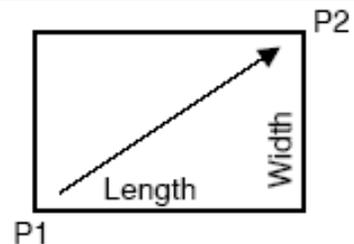
The Rectangle Command

Toolbar 

Pull-down **Draw > Rectangle**

Keyboard **RECTANGLE**

short-cuts **REC, RECTANG**



The Rectangle command is used to draw a rectangle whose sides are **vertical and horizontal**. The position and size of the rectangle are defined by picking two diagonal corners. The rectangle isn't really an AutoCAD object at all. It is, in fact, just a closed polyline which is automatically drawn for you.

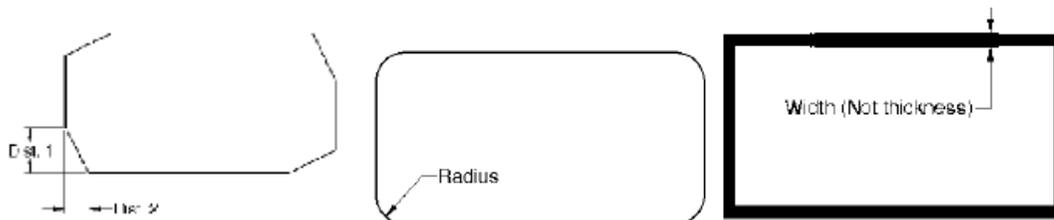
Notice that, instead of picking a second point to draw the rectangle, you have the option of entering dimensions. Say you wanted to draw a rectangle 20 drawing units long and 10 drawing units wide. The command sequence would look like this:

Command: **RECTANG** ↵
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: (pick a point)
Specify other corner point or [Dimensions]: **D** ↵
Specify length for rectangles <0.0000>: **20** ↵
Specify width for rectangles <0.0000>: **10** ↵
Specify other corner point or [Dimensions]: (pick a point to fix the orientation)

Options for drawing RECTangle:

[Chamfer/Elevation/Fillet/Thickness/Width]:

The Rectangle command also has a number of options. **Width** works in the same way as for the Polyline command. The **Chamfer** and **Fillet** options have the same effect as the **Chamfer** and **Fillet** commands, see the Modifying Objects tutorial for details. **Elevation** and **Thickness** are 3D options.



The Polygon Command

Toolbar 

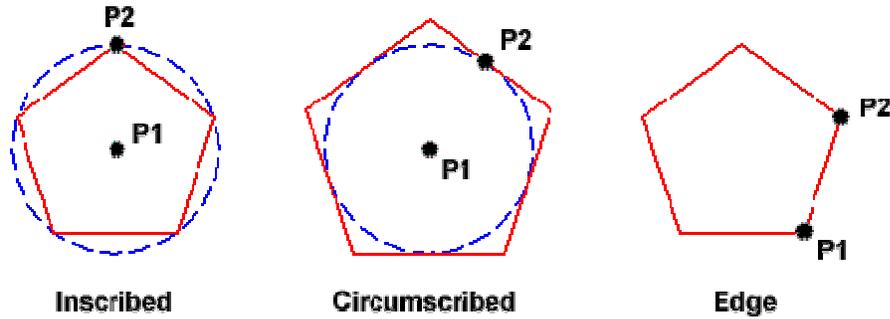
Pull-down: **Draw>Polygon**

Keyboard: **POLYGON**, short-cut **POL**

The Polygon command can be used to draw any regular polygon from 3 sides up to 1024 sides. This command requires four inputs from the user, the number of sides, a pick point for the centre of the polygon, whether you want the polygon inscribed or circumscribed and then a pick point which determines both the radius of this imaginary circle and the orientation of the polygon. The polygon command creates a closed polyline in the shape of the required polygon.

Command Sequence

Command: **POLYGON** ↵
Enter number of sides <4>: **5** ↵
Specify center of polygon or [Edge]: pick P1
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: ↵ (to accept the inscribed default or type **C** for circumscribed)
Specify radius of circle: (pick P2 or enter exact radius)



This command also allows you to define the polygon by entering the length of a side using the Edge option. You can also control the size of the polygon by entering an exact radius for the circle. Follow the command sequence below to see how this command works.

Example: Draw a Pentagon, length of side = 30 mm from the vertex 100,100

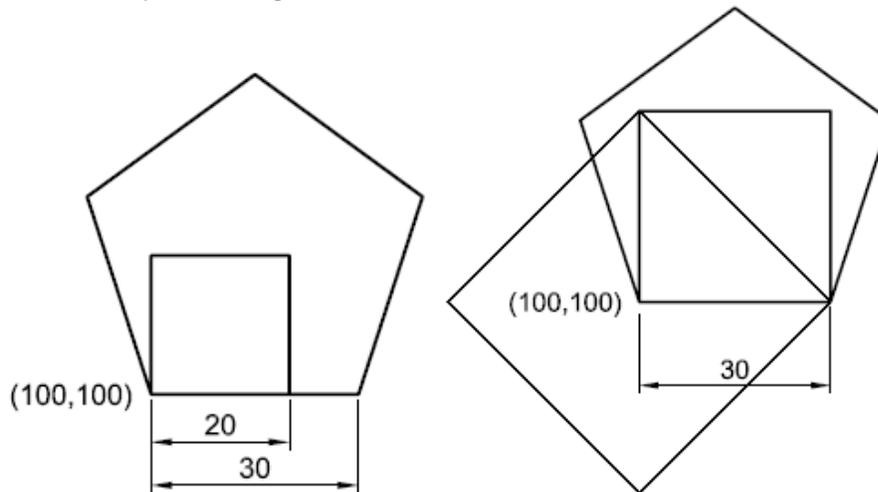
Command: **Polygon** ↵

Enter number of sides <3>: **5** ↵

Specify center of polygon or [Edge]: **E** ↵

Specify first endpoint of edge: **100,100** ↵

Specify second endpoint of edge: **@30<0** ↵



Draw a Square, length of side = 20 mm from the vertex 100,100

Command: **Polygon** ↵

Enter number of sides <5>: **4** ↵

Specify center of polygon or [Edge]: **E** ↵

Enter an option [Inscribed in circle/Circumscribed about circle]

<I>: **100,100** ↵

Specify radius of circle: **@20<0** ↵

The Circle Command

Center, Radius
Center, Diameter
2 Points
3 Points
Tan, Tan, Radius
Tan, Tan, Tan



Toolbar: **Draw** ▶ **Circle** ▶ **Center, Radius**

Keyboard: **CIRCLE** short-cut **C**

The Circle command is used to draw circles. There are a number of ways you can define the circle. The default method is to pick the centre point and then to either pick a second point on the circumference of the circle or enter the circle radius at the keyboard.

Draw a circle which the center is at absolute coordinate **(0,10)** and radius is **53** unit.

Command: **CIRCLE** ↵

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **0, 10** ↵

Specify radius of circle or [Diameter] <50.0195>: **53** ↵

Draw a circle which the center is at absolute coordinate (15,32) and Diameter is 18 unit.

Command: **CIRCLE** ↵

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **15, 32** ↵

Specify radius of circle or [Diameter] <50.0195>: **D** ↵

Specify Diameter of circle <100.390>: **18** ↵

The **circle** command gives you three extra options to define a circle.

3P which uses any three points on the circumference,

2P which uses two points on the circumference to form a diameter and

TTR which stands for **Tangent Tangent Radius**.

3 Points

Command: **CIRCLE** ↵

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

Specify radius of circle or [Diameter] <50.0195>: **Pick P1**

Specify radius of circle or [Diameter] <50.0195>: **Pick P2**

Specify radius of circle or [Diameter] <50.0195>: **Pick P3**

2 Points

Command: **CIRCLE** ↵

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

Specify radius of circle or [Diameter] <50.0195>: **Pick P1**

Specify radius of circle or [Diameter] <50.0195>: **Pick P2**



Tangent Tangent Radius TTR

Command: **CIRCLE** ↵

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **TTR** ↵

Specify radius of circle or [Diameter] <50.0195>: **Pick P1**

Specify radius of circle or [Diameter] <50.0195>: **Pick P2**

Specify radius of circle <49.315>: **15** ↵

Tangent Tangent Tangent

Command: **CIRCLE** ↵

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

Specify radius of circle or [Diameter] <50.0195>: (**Type TAN** ↵) **Pick P1**

Specify radius of circle or [Diameter] <50.0195>: (**Type TAN** ↵) **Pick P2**

Specify radius of circle or [Diameter] <50.0195>: (**Type TAN** ↵) **Pick P3**

Draw a circle tangent to a given triangle.

Draw a circle through a three point of triangle.

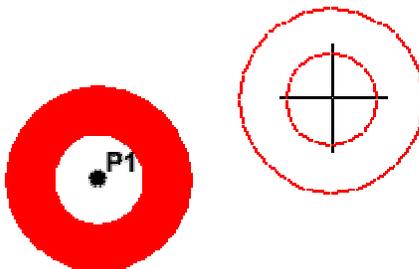
The Donut Command

Toolbar custom 

Pull-down **Draw** ▶ **Donut**

Keyboard **DONUT** short-cut **DO**

This command draws a solid donut shape, actually it's just a closed polyline consisting of two arc segments which have been given a width. AutoCAD asks you to define the inside diameter i.e. the diameter of the hole and then the outside diameter of the donut. The donut is then drawn in outline and you are asked to pick the centre point in order to position the donut.



then drawn in outline and you are asked to pick the centre point in order to position the donut. You can continue picking centre points to draw more donuts or you can hit ↵ to end the command. Surprisingly, donuts are constructed from single closed polylines composed of two arc segments which have been given a width. Fortunately AutoCAD works all this out for you, so all you see is a donut.

Command Sequence

Draw a **donut**, inside diameter =1.5 and outside diameter 2.25.

Command: **DONUT** ↵

Specify inside diameter of donut <0.5000>: **1.5** ↵

Specify outside diameter of donut <1.5000>: **2.25** ↵

Specify center of donut or <exit>: (pick **P1**)

Specify center of donut or <exit>: ↵ (to end or continue to pick for more donuts)

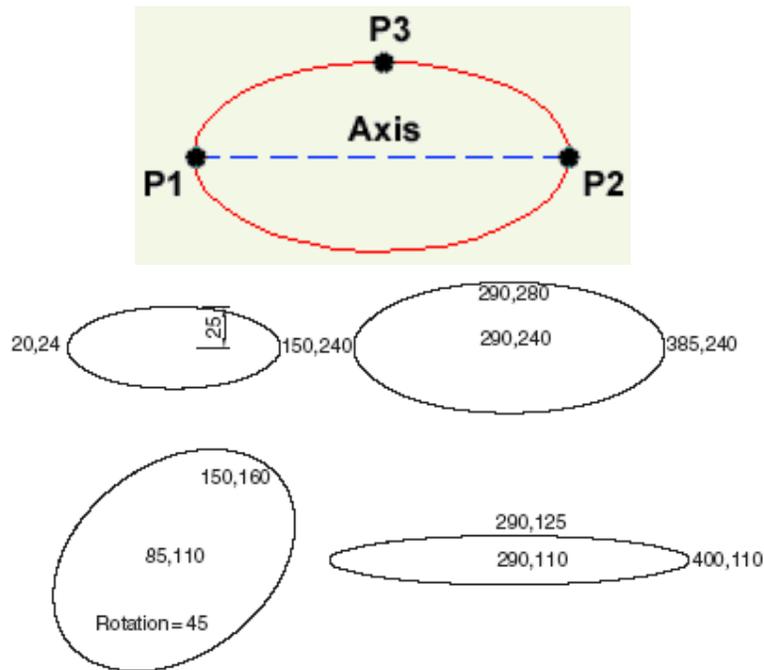
The Ellipse Command

Toolbar: 

Pull-down: **Draw > Ellipse > Axis, End**

Keyboard: **ELLIPSE** short-cut: **EL**

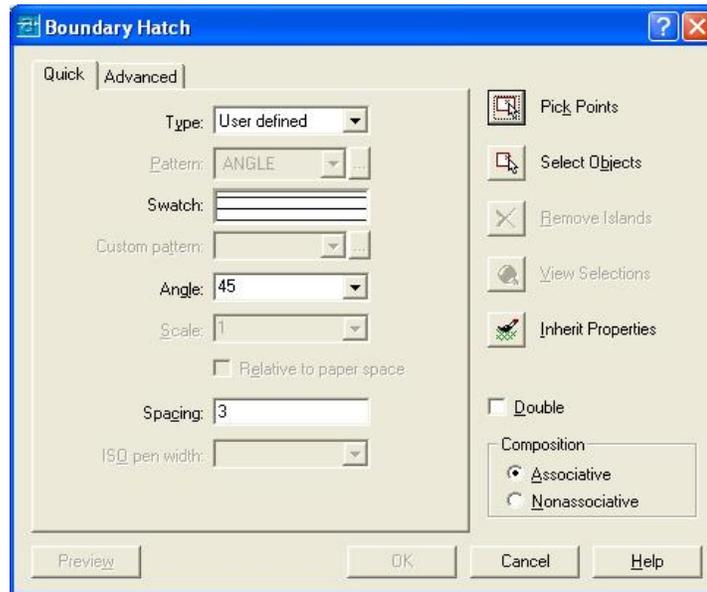
The Ellipse command gives you a number of different creation options. The default option is to pick the two end points of an axis and then a third point to define the eccentricity of the ellipse. After you have mastered the default option, try out the others.



Bhatch

Command: (**Bhatch** or **BH**) 8

.....The **Hatch Dialog Box** will appear



Hatch Dialog Box

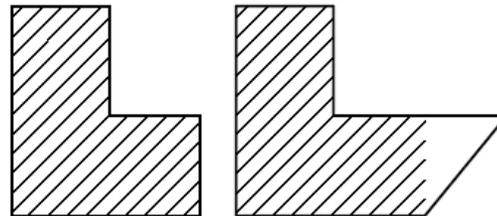
Choose:

Type: User defined

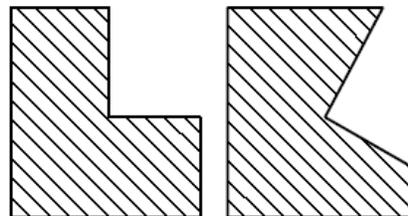
Angle: 45°

Spacing: 3 units

Pick Points: Pick a point inside closed area & Composition: Associative or Nonassociative



NONASSOCIATIVE HATCH



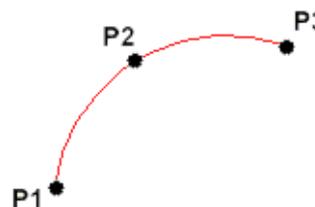
ASSOCIATIVE HATCH

The Arc Command

Toolbar: Draw > 

Pull-down: **Draw > Arc > 3 Points**

Keyboard: **ARC** short-cut: **A**



The Arc command allows you to draw an arc of a circle. There are numerous ways to define an arc, the default method uses three pick points, a start point, a second point and an end point. Using this method, the drawn arc will start at the first pick point, pass through the second point and end at the third point. Once you have mastered the default method try some of the others. You may, for example need to draw an arc with a specific radius. All of the Arc command options are available from the pull-down menu.

Command: **ARC** ←
Specify start point of arc or [Center]: **pick P1**
Specify second point of arc or [Center/End]: **pick P2**
Specify end point of arc: **pick P3**

- 3 Points

- Start, Center, End
- Start, Center, Angle
- Start, Center, Length

- Start, End, Angle
- Start, End, Direction
- Start, End, Radius

- Center, Start, End
- Center, Start, Angle
- Center, Start, Length

- Continue

It is also possible to create an arc by trimming a circle object. In practice, many arcs are actually created this way. See the Trim command on the Modifying Objects tutorial for details.

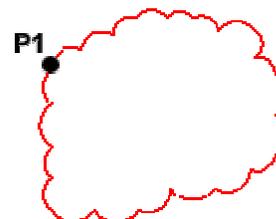
The Revcloud Command

Toolbar: Draw 
Pull-down: **Draw** ▶ **Revision Cloud**
Keyboard **REVCLOUD**

The Revcloud command is used to draw a "freehand" revision cloud or to convert any closed shape into a revision cloud.

Command Sequence

Command: **Revcloud** ←
Minimum arc length: 5 Maximum arc length: 6 Style: Normal
Specify start point or [Arc length/Object/Style] <Object>: **a** ←
Specify minimum length of arc <5>: **10** ←
Specify maximum length of arc <10>: **12** ←
Specify start point or [Arc length/Object/Style] <Object>:**Pick P1**)
Guide crosshairs along cloud path...
Move the mouse to form a closed shape; the command automatically ends when a closed shape is formed.
Reverse direction [Yes/No] <No>:



The Point Command

Toolbar: Draw 
Pull-down: **Draw** ▶ **Point** ▶ **Single Point**
Keyboard **POINT** short-cut **PO**

The point command will insert a point marker in your drawing at a position which you pick in the drawing window or at any co-ordinate location which you enter at the

keyboard. The default point style is a simple dot, which is often difficult to see but you can change the point style to something more easily visible or elaborate using the point style dialogue box. Points can be used for "setting out" a drawing in addition to construction lines. You can *Snap* to points using the Node object snap. See the [Object Snap](#) tutorial for details.

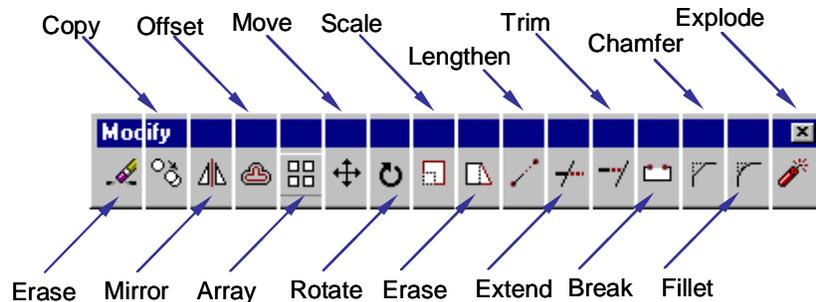
Command Sequence

Command: **POINT** ↵
Current point modes: PDMODE=0 PDSIZE=0.0000
Specify a point: **pick any point**

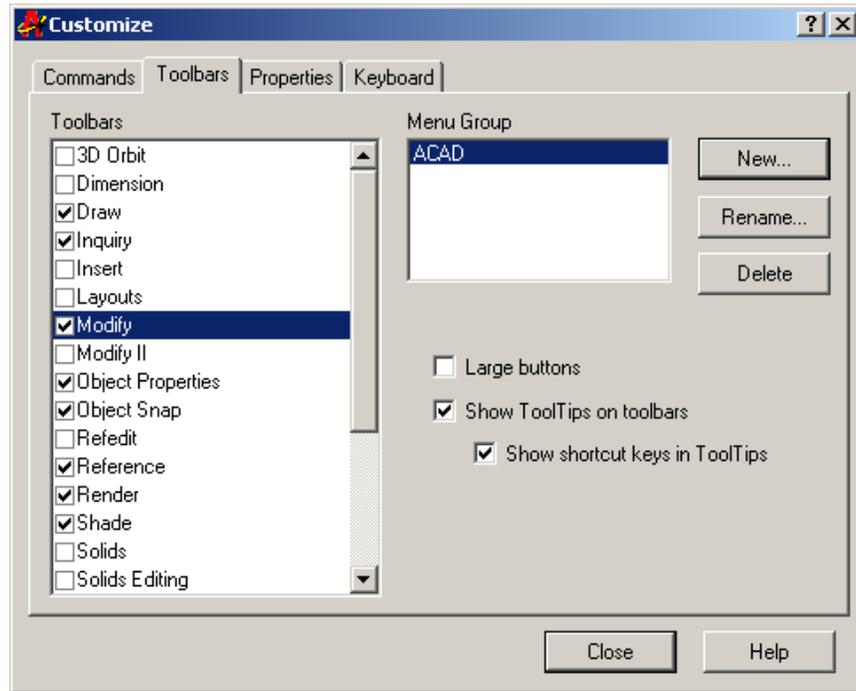
Strangely, in Multiple Point mode (the default for the Point button on the Draw toolbar) you will need to use the escape key (Esc) on your keyboard to end the command. The usual right-click or enter doesn't work.

Modify Toolbar

AutoCAD drawings are rarely completed simply by drawing lines, circles etc. Most likely you will need to *Modify* these basic drawing objects in some way in order to create the image you need. AutoCAD provides a whole range of modify tools such as **Move**, **Copy**, **Rotate** and **Mirror**. As you can see, the command names are easily understandable. However, the way these commands work is not always obvious. This tutorial is designed to show you how all of the Modify commands work.



As is usual with AutoCAD, the Modify tools can be accessed in one of three ways, from the **Keyboard**, from the **Pull-down Menu** and from the **Toolbar**. All of the Modify tools are available from the Modify pull-down and the Modify toolbar. In each section below, the toolbar, pull-down and keyboard options are given. The method you choose is entirely up to you. Ultimately you will use the method that you feel most comfortable with or the one you find most efficient. AutoCAD allows great flexibility and there aren't any right or wrong ways of working. That said, it should be pointed out that the use of toolbars in AutoCAD is almost always quicker than any other method.



Customize dialog box

The Modify toolbar is usually displayed by default but if it is not already displayed, you can display it using the **TOOLBAR** command, **View>Toolbars...** from the **Pull-down Menu**. When the Toolbar dialogue box (shown above) appears, simply check the box next to "Modify" in the toolbars list. Many AutoCAD users work with the Modify toolbar permanently **Docked** on their screen because it gives one-click access to all of the commands, making the drawing process much more efficient.

The Erase Command

Toolbar: 

Pull-down: **Modify>Erase**

Keyboard: **ERASE** or **E**

The Erase command is one of the simplest AutoCAD commands and is one of the most used. The command erases (deletes) any selected object(s) from the drawing. Remember you can always get deleted objects back by typing **U** to undo,  from the Standard toolbar or by using the OOPS command.

Command: **ERASE** **↵**

Select objects: (**pick an object to erase**)

Select objects: **↵**(to end the selection and erase the object)

If you simply want to erase the last object you created you can type **L** at the "Select objects" prompt. The last object will be highlighted and you can then select more objects or **↵** to end the command.

Command: **ERASE** **8**

Select objects: 1 found **8**

Select objects: **8**

Command: **ERASE** **8**

Select objects: all **8**

Select objects: **8**

The Copy Command

Toolbar: 

Pull-down: **Modify>Copy**

Keyboard: **COPY** or **CP**

The Copy command can be used to create one or more duplicates of any drawing object or objects which you have previously created. Copy is a very useful and time-saving command because you can create very complex drawing elements and then simply copy them as many times as you like.

Command: **COPY** ↵

Select objects: (**pick object to copy, P1**)

Select objects: ← (**to end selection**)

Specify base point or displacement, or [Multiple]: (**pick P2 or M for multiple copies**)

Specify second point of displacement or <use first point as displacement>: (**pick P3**)

The multiple option allows you to create additional copies of the selected object(s) by picking as many new points as you like. To end a multiple copy, just hit the ↵ key.

Notice that the "Base point", P_2 and the "Second point", P_3 do not have to be picked on or near the object. The two points are simply used to indicate the distance and direction of the copied object from the original object.

Canceling and Undoing a command:

If you are in a command and you want to cancel or get out of that command, press the ESC (Escape) key on the keyboard.

Similarly, sometimes you unintentionally erase some object from the screen. When you discover such an error, you can correct it by restoring the erased object by means of the OOPS command. The OOPS command restores objects that have been accidentally erased by the previous ERASE command,

The Mirror Command

Toolbar: 

Pull-down: **Modify>Mirror**

Keyboard: **MIRROR**

The Mirror command allows you to mirror selected objects in your drawing by picking them and then defining the position of an imaginary mirror line using two points

Command: **MIRROR** ↵

Select objects: (**pick object to mirror, P1**)

Select objects: ← (**to end selection**)

Specify first point of mirror line: (**pick P2**)

Specify second point of mirror line: (**pick P3**)

Delete source objects? [Yes/No] <N>: ↵ (**for No to keep the original object**)

Notice that in the command sequence above, pressing the ↵ key when asked whether to "Delete source objects?" resulted in a "No" response. This is because "No" is the *default* option. AutoCAD always shows the default option within triangular brackets, in this case "<N>". Many AutoCAD commands offer default options. They are usually the most commonly used and you need only right-click and select "Enter" from the menu or press ↵ on the keyboard to accept them.

The Offset Command

Toolbar: 

Pull-down: **Modify>Offset**

Keyboard: **OFFSET**

Offset is probably one of the most useful commands for constructing drawings. The Offset command creates a new object parallel to or concentric with a selected object. The new object is drawn at a user defined distance (the offset) from the original and in a direction chosen by the user with a pick point. You can offset lines, arcs, circles, ellipses, 2D polylines, xlines, rays and planar splines.

Command: **OFFSET** ↵

Specify offset distance or [Through] <1.0>: (**specify distance, as example 50**) ↵

Select object to offset or <exit>: (**select object, P1**)

Specify point on side to offset: (**pick direction, P2**)

Select object to offset or <exit>: ↵ (**to end or select another object to offset**)

Circles can be offset inside or outside of themselves to create a new circle which is concentric (has the same centre point) with the original circle. In the illustration, a new circle has been created outside of the original by picking a point outside of the original circle. The radius of the new circle is the offset distance "**Offset**" plus the radius of the original circle.

The Array Command

Toolbar: 

Pull-down: **Modify>Array**

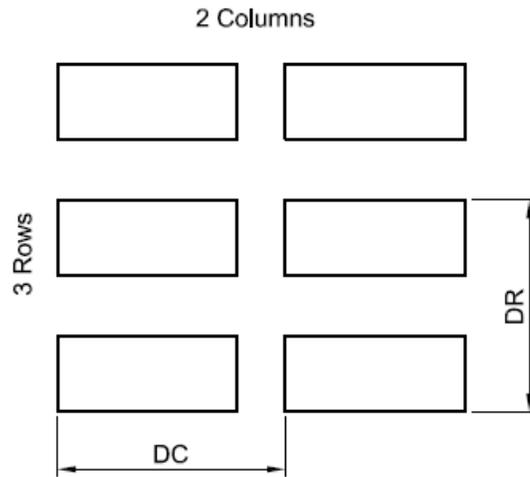
Keyboard: **Array**

The Array command makes multiple copies of selected objects in a rectangular matrix (columns and rows) or a polar (circular) pattern. This command has been completely transformed in AutoCAD 2000i. It is now completely dialogue box driven with the option to see a preview of the array before it is created. You can also now create rectangular arrays at a user specified angle. This constitutes a major improvement in usability.

The Rectangular Array

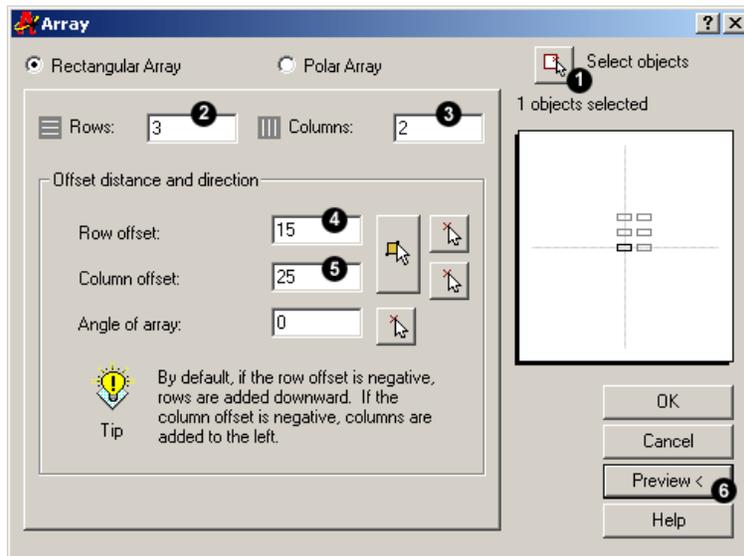
The illustration on the right shows the results of a rectangular array with two columns and three rows. The distance between rows is indicated with the dimension **DR** and between columns with **DC**. When creating rectangular arrays it is important to remember that new rows are created above the original object and new columns are created to the right of the original object. The resulting array is, therefore, always created with the original object in the bottom left hand position with respect to the current co-ordinate system.

Try creating this array yourself. First, draw a rectangle 20 drawing units wide and 10 drawing units tall. To do this, start the rectangle command **Draw>Rectangle** from the pull-down menu. When you are prompted to pick a point, pick somewhere in the lower left quarter of the drawing area. Then, when you are prompted for a second point, enter the relative cartesian co-ordinate **@20,10**. You have now drawn a rectangle at the right size.



Rectangular Array

Now start the Array command. First of all, make sure that the Rectangular Array radio button is selected and that you are looking at the Rectangular Array dialogue box and now follow the steps below:



Rectangular Array dialogue box

1. Click the Select objects button. The dialogue box will temporarily disappear enabling you to select the rectangle you just drew. Press the Return button on your keyboard to complete the selection. You are now returned to the dialogue box and the message immediately below the Select Objects button should read "1 objects selected".
2. Enter the number of rows required in the Rows edit box. For this example, enter the value "3". Notice that the schematic preview on the right hand side of the dialogue box updates to reflect the values you are entering.
3. Enter the number of columns required in the Columns edit box. Enter the value "2".

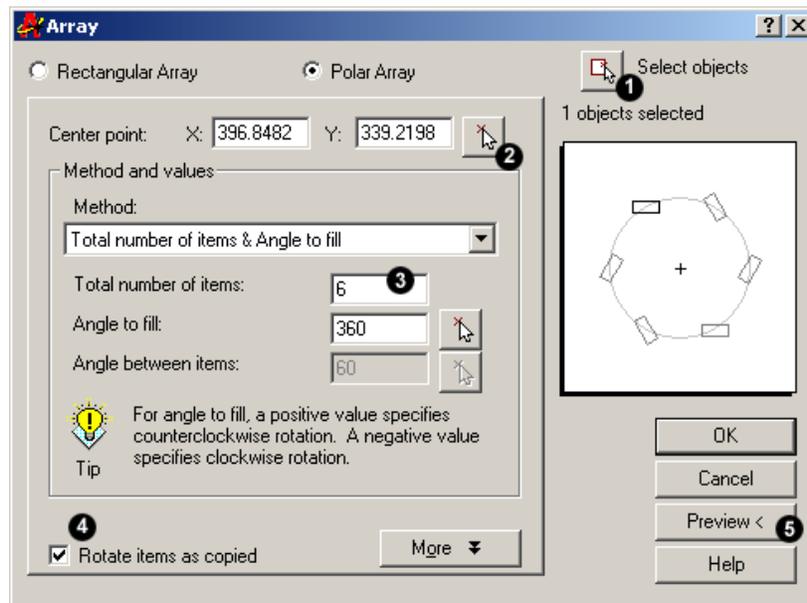
4. Enter the row offset in the Row Offset edit box. This is the distance DR in the illustration above. Note that this is not the distance between rows. In this example, our rectangle is 10 units high and we will enter a row offset of 15. The result will be a 5 unit gap between rectangles.
5. Enter the column offset in the Column Offset edit box. The same parameters apply as for the row offset. Enter a value of 25 to give a 5 unit gap between our rectangles.
6. Click on the Preview button. Once again, the dialogue box disappears and the specified array is temporarily drawn so that we can preview it. We are now offered 3 choices. If the array isn't quite right, click the Modify button to return to the Array dialogue box. If you are happy with the array, click the Accept button, the array will be permanently drawn and the command is ended.



dialogue box: Array – Accept ...

The Polar Array

The polar array works in a similar way to the rectangular array. The main difference is that rather than specifying the number and offset for rows and columns, you must specify a center point and the total number of objects in the array (including the original object).



Rectangular Array dialogue box

You can try this for yourself by drawing another rectangle as described above. This time, though, locate the rectangle in the centre top half of the drawing area. Now, start the Array command, click the Polar Array radio button and follow the steps below:

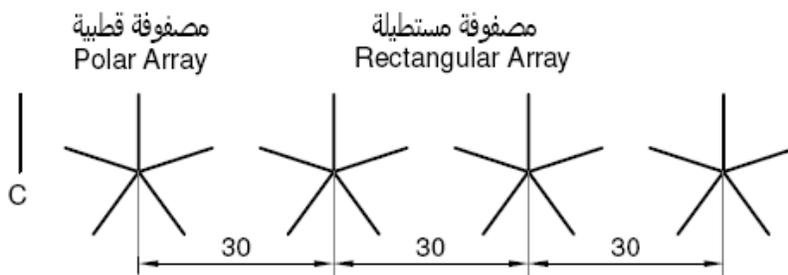
1. Click the Select objects button. The dialogue box will temporarily disappear enabling you to select the rectangle you just drew. Press the Return button on

your keyboard to complete the selection. You are now returned to the dialogue box and the message immediately below the Select Objects button should read "1 objects selected".

2. Specify the center point for the array. This is the point C in the illustration below. You can do this by entering x and y co-ordinates into the appropriate edit boxes if you know what these values should be. However, this is rarely the case and most often you will want to click the Pick Center Point button to pick a point from the drawing area. Pick a point somewhere below the rectangle you have just drawn. Enter a value for the total number of items. For this example, enter the value "6". Notice that once again, the schematic preview updates to reflect the values you have entered.
3. Make sure that the Rotate items as copied checkbox is checked.
4. Click on the Preview button. Once again, the dialogue box disappears and the specified array is temporarily drawn so that we can preview it. We are now offered 3 choices. If the array isn't quite right, click the Modify button to return to the Array dialogue box. If you are happy with the array, click the Accept button, the array will be permanently drawn and the command is ended.

The array you have just drawn should look something like the one illustrated below, left. Take some time to play around with the other options in the Array dialogue box to see what they can do.

Example:



The Move Command

Toolbar:

Pull-down: **Modify> Move**

Keyboard: **MOVE** or **M**

The Move command works in a similar way to the Copy command except that no copy is made, the selected object(s) is simply moved from one location to another.

Command: **MOVE**

Select objects: **(pick object to move, P1)**

Select objects: **(to end selection)**

Specify base point or displacement: **(pick P2)**

Specify second point of displacement or <use first point as displacement>: **(pick P3)**

Note that as with the Copy command, the two pick points, **P2** and **P3** are used only to indicate the distance and direction of movement.

The Rotate Command

Toolbar: 

Pull-down: **Modify> Rotate**

Keyboard: **ROTATE** or **RO**

The Rotate command allows an object or objects to be rotated about a point selected by the user. AutoCAD prompts for a second rotation point or an angle which can be typed at the keyboard.

Command: **ROTATE** ↵

Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0

Select objects: (**pick object to rotate, P1**)

Select objects: ↵(**to end selection**)

Specify base point: (**pick base point, P2**)

Specify rotation angle or [Reference]: (**pick second point, P3 or enter angle**)

Remember, by default, AutoCAD angles start at 3 o'clock and increase in an anti-clockwise direction. The "**ANGDIR**" and "**ANGBASE**" variables remind you of this. If you want to rotate in a clockwise direction you can enter a negative angle by using a minus sign.

The Scale Command

Toolbar: 

Pull-down: **Modify> Scale**

Keyboard: **SCALE** or **SC**

The Scale command can be used to change the size of an object or group of objects. You are prompted for a pick point about which the selection set will be scaled. Scaling can then be completed by picking a second point (not always easy because it can sometimes be difficult to precisely control the scaling) or by entering a scale factor at the keyboard. For example a scale factor of 2 will double the size of the objects in the selection set and a factor of 0.5 will half them.

Command: **SCALE** ↵

Select objects: **W** (or pick the screw)

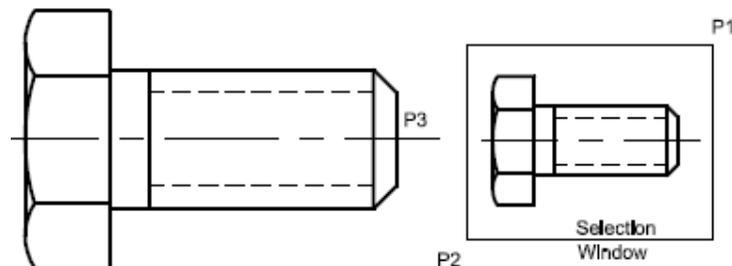
Specify first corner: **P1**

Specify opposite corner: **P2** 1 found

Select objects: : ↵(**to end selection**)

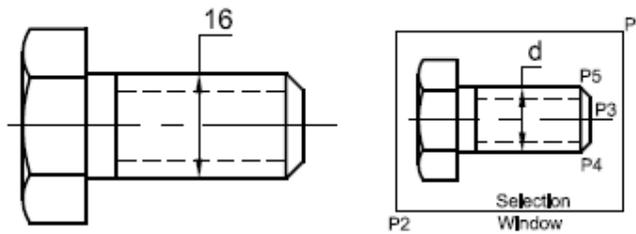
Specify base point: **P3**

Specify scale factor or [Reference]:**2** ↵ (**To double the size of the screw about P3**)



Scaling the object, SCALE FACTOR 2

Command: **SCALE** ↵
Select objects: **W** ↵
Specify first corner: **(Pick P1)**
Specify opposite corner: **(Pick P2)** 1 found
Select objects: : ↵**(to end selection)**
Specify base point: **(Pick P3)**
Specify scale factor or [Reference]: **R** ↵
Specify reference length <1>:**(Pick P4)** Specify second point: **(Pick P5)**
Specify new length: **16** ↵



Scaling the object, SCALE REFERENCE d → 16

In the example shown above, the original tree symbol has been enlarged by dynamically scaling it using pick points to determine the change in scale. If you want to scale an object precisely, it is much easier to enter a scale factor using the keyboard.

Note that the position of the new sized screw is determined by the location of the base point **P3**.

The Trim Command

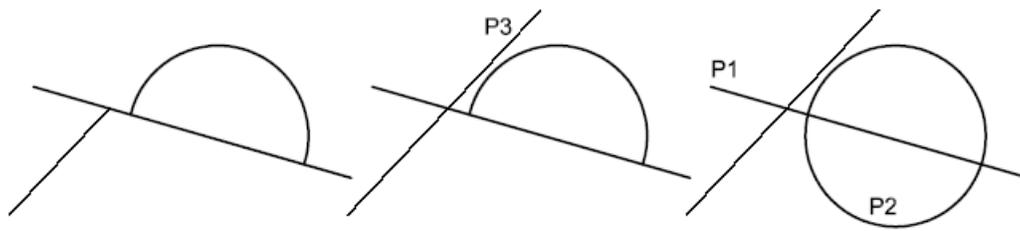
Toolbar: 

Pull-down: **Modify> Trim**

Keyboard: **TRIM** or **TR**

The Trim command can be used to trim a part of an object. In order to trim an object you must draw a second object which forms the "cutting edge". Cutting edges can be **lines, xlines, rays, polylines, circles, arcs** or **ellipses**. Blocks and text cannot be trimmed or used as cutting edges. The Trim command, unlike most other modify commands requires that two separate object selections are made. The cutting edges are selected first (there can be one or more) and then the objects to be trimmed are selected.

Command: **TRIM** ↵
Current settings: Projection=UCS Edge=None
Select cutting edges ...
Select objects: **(select the cutting edge, LINE P1)**
Select objects: ↵**(to end cutting edge selection)**
Select object to trim or shift-select to extend or [Project/Edge/Undo]:
(pick the part of the CIRCLE which you want to trim, P2)
Select object to trim or shift-select to extend
or [Project/Edge/Undo]: **(pick the part of the LINE , P3)**
Select object to trim or shift-select to extend
or [Project/Edge/Undo]: ↵**(to end)**



Trim Sequence from Right to Left

Notice that at each trimming step you are given the option to undo the previous trim. This can be very useful if you inadvertently pick the wrong object.

The Extend Command

Toolbar: 

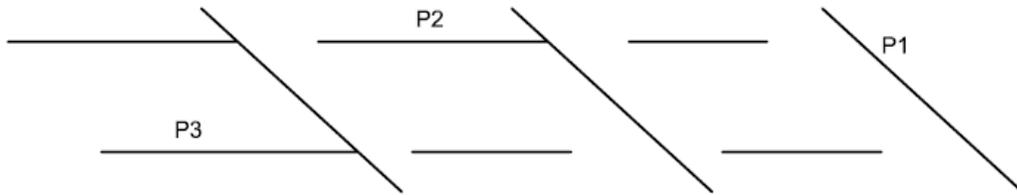
Pull-down: **Modify> Extend**

Keyboard: **EXTEND** or **EX**

This command extends a **line**, **polyline** or **arc** to meet another drawing object (known as the boundary edge). This command works in a similar way to the Trim command, described above. Two selections are made, one for the boundary edge(s) and one for the object(s) to extend.

Lines and other objects can be extended in one of two directions. You can tell AutoCAD in which direction to extend by picking a point to the right or left of the midpoint respectively. AutoCAD does not intuitively know where the boundary edge lies so you must explicitly indicate the direction of extension by picking either one side or other of the midpoint.

Command: **EXTEND** ↵
Current settings: Projection=UCS Edge=None
Select boundary edges ...
Select objects: **(select the boundary edge, LINE P1)**
Select objects: ↵**(to end boundary edge selection)**
Select object to extend or shift-select to trim or [Project/Edge/Undo]:
(pick the object which you want to be extended, LINE P2)
Select object to extend or shift-select to trim or [Project/Edge/Undo]:
(pick another object to be extended, LINE P3)
Select object to extend or shift-select to trim or [Project/Edge/Undo]: ↵**(to end)**



Extend Sequence from Right to Left

Sometimes you may get the message "Object does not intersect an edge" or "No edge in that direction". If this happens it means that you are either picking the wrong end of the object or the object you are trying to extend will not meet the boundary edge. The solution is either to pick near the end you want to extend or to move the boundary edge so that the extended line will intersect with it.

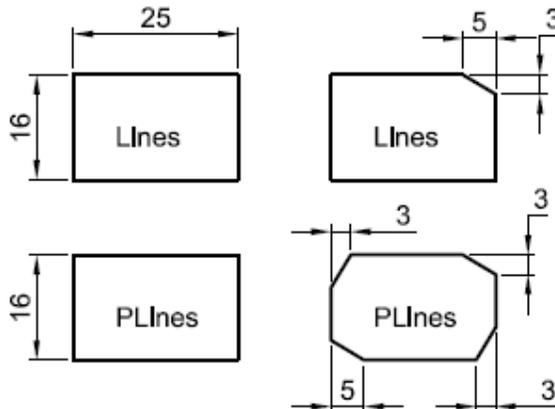
The Chamfer Command

Toolbar: **Modify** > 
Pull-down: **Modify > Chamfer**
Keyboard: **CHAMFER** short-cut: **CHA**

The Chamfer command enables you to create a chamfer between any two non-parallel lines as in the illustration below or any two adjacent polyline segments. Usually, the Chamfer command is used to set the chamfer distances before drawing the chamfer. Follow the command sequence below where the chamfer distances are changed to 3 and 5 before the chamfer is made

Example: Make a chamfer at the 25x16 rectangle drawn in by **Line** command.

Command: **CHAMFER** ↵
(TRIM mode) Current chamfer Dist1 = 10.0000, Dist2 = 10.0000
Select first line or [Polyline/Distance/Angle/Trim/Method]: **D** ↵ (to set distances)
Specify first chamfer distance <10.0000>: **3** ↵
Specify second chamfer distance <20.0000>: **5** ↵
Select first line or [Polyline/Distance/Angle/Trim/Method]: **Pick vertical line**
Select second line: **Pick horizontal line**
The chamfer is made and the command ends.



Example: Make a chamfer at the 25x16 rectangle drawn by Rec command.

Command: **CHAMFER** ↵
(TRIM mode) Current chamfer Dist1 = 5, Dist2 = 3
Select first line or [Polyline/Distance/Angle/Trim/Method]: **D** ↵
Specify first chamfer distance <5>: **3** ↵
Specify second chamfer distance <4>: **5** ↵
Select first line or [Polyline/Distance/Angle/Trim/Method]: **P** ↵
Select 2D polyline: **Select a Rectangle**
4 lines were chamfered

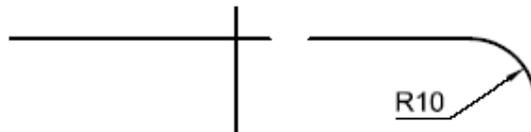
Notice from the command sequence that there are a number of options which can be used to control the way the Chamfer command behaves. The **Polyline** option can be used to chamfer all vertexes of a polyline simultaneously. The **Distance** option allows you to specify the two chamfer distances. Angle allows the angle between the first line and the chamfer to be specified. Trim is used to control whether the original lines are trimmed to the chamfer or remain as they are. Finally, Method is used to toggle the command between Distance and Angle mode. When Angle mode is used, the chamfer is defined using one distance and an angle rather than two distances.

The Fillet Command

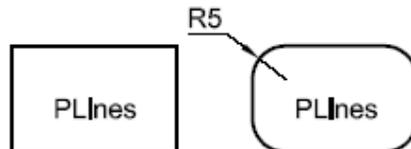
Toolbar: **Modify** > 
Pull-down: **Modify > Fillet**
Keyboard: **Fillet** short-cut: **F**

The Fillet command is a very useful tool which allows you to draw an arc between two intersecting lines or adjacent polyline segments. You first need to use the command to set the required radius and then a second time to select the two lines.

Command: **FILLET** ↵
Current settings: Mode = TRIM, Radius = 10.0000
Select first object or [Polyline/Radius/Trim]: **R** ↵
Specify fillet radius <10.000>: **10** ↵
Select first object or [Polyline/Radius/Trim]: **Pick vertical line**
Select second object : **Pick horizontal line**



The Fillet command can also be used to fillet arcs and circles. The "Polyline" option also allows you to fillet all vertices of a polyline with a single command. It's worth experimenting with this command, it can save you lots of time and enables you to construct shapes which otherwise would be quite difficult.



Command: **FILLET** ←
Current settings: Mode = TRIM, Radius = 5.0000
Select first object or [Polyline/Radius/Trim]: **P** ←
Select 2D polyline: **Pick Rectangle**
4 lines were filleted

The Explode Command

Toolbar: **Modify** > 
Pull-down: **Modify** > **Explode**
Keyboard: **EXPLODE**

The Explode command is used to "explode" single objects back to their constituent parts. In other words, the command is used to return blocks, polylines etc. (which may be composed of a number of component objects) back to their individual component parts. The change has no visible effect.

The Lengthen Command

Toolbar: **Modify** 
Pull-down: **Modify** ▶ **Lengthen**
Keyboard: **LENGTHEN** short-cut: **LEN**

The Lengthen command can often be used instead of either the **Trim** or **Extend** commands. Indeed the end result is exactly the same. The Lengthen command can be used to either lengthen or shorten Lines, Arcs, open Polylines, elliptical Arcs and open Splines without the use of cutting or boundary edges.

Command Sequence

Command: **LENGTHEN** ←
Select an object or [DElta/Percent/Total/DYnamic]: **DY** ←
Select an object to change or [Undo]: (**select a line or arc**)
Specify new end point: (**pick new end point**)
Select an object to change or [Undo]: ←(**to end**)

Select an object to change or [Undo]: The command sequence above demonstrates the use of the Dynamic Lengthen option which is probably the most useful for general purpose drafting. However, the other options are worth getting to know because they can save lots of time and effort.

Command: **LENGTHEN** ←
Select an object or [DElta/Percent/Total/DYnamic]: **T** ←
Specify total length or [Angle] <89.500>: **93** ←
Select an object to change or [Undo]: ←

The **Total** option allows you to change the total length of a line to any value that you specify. The **Percent** option allows you to change a line length using a percentage. For example, a value of 50 will result in a line one half the length of the original and a value of 200 will result in a line twice the length of the original. The **Delta** option can be used to extend or reduce the endpoint of a line by a given distance. The endpoint affected by the change is the one closest to the pick point when the object selection is made.

The Match Properties Command

Toolbar: 

Pull-down: **Modify** ▸ **Match Properties**

Keyboard **MATCHPROP**

Another way to change the properties of an object or objects is to match the properties of any other object using the Match Properties command on the Standard toolbar.

Command Sequence

Command: **MATCHPROP** ↵

Select Source Object: (**pick the object you wish to match**)

Current active settings = color layer ltype ltscale thickness text dim hatch

Settings/<Select Destination Object(s)>: (**pick the objects to inherit properties**)

Settings/<Select Destination Object(s)>: ↵

Creating blocks

Command: **Block** (b) ↵

Block Definition dialogue box will appear

Give **NAME OF BLOCK**, B1 as an

Select objects: Specify opposite corner: 1 found

Select objects: ↵

Specify insertion base point: int Of

Dividing lines into equal parts.

Divide a given line, curve or spline into equal parts.

Command: **divide** ↵(div ↵)

Select object to divide: **Pick an object or line** ↵

Enter the number of segments or [Block]: **5** ↵ (5 parts)

Divide a given line into equal parts by using blocks.

Command: **div** ↵

DIVIDE

Select object to divide: **pick an object**

Enter the number of segments or [Block]: **b**

Enter name of block to insert: **b1** ↵

Align block with object? [Yes/No] <Y>: ↵

Enter the number of segments: **3** ↵

Measure a certain-size parts on line, curve or spline

Draw a 117 line and measure 50 unit equal parts on it.

Command: **measure** ↵

Select object to measure: **Pick an object** ↵

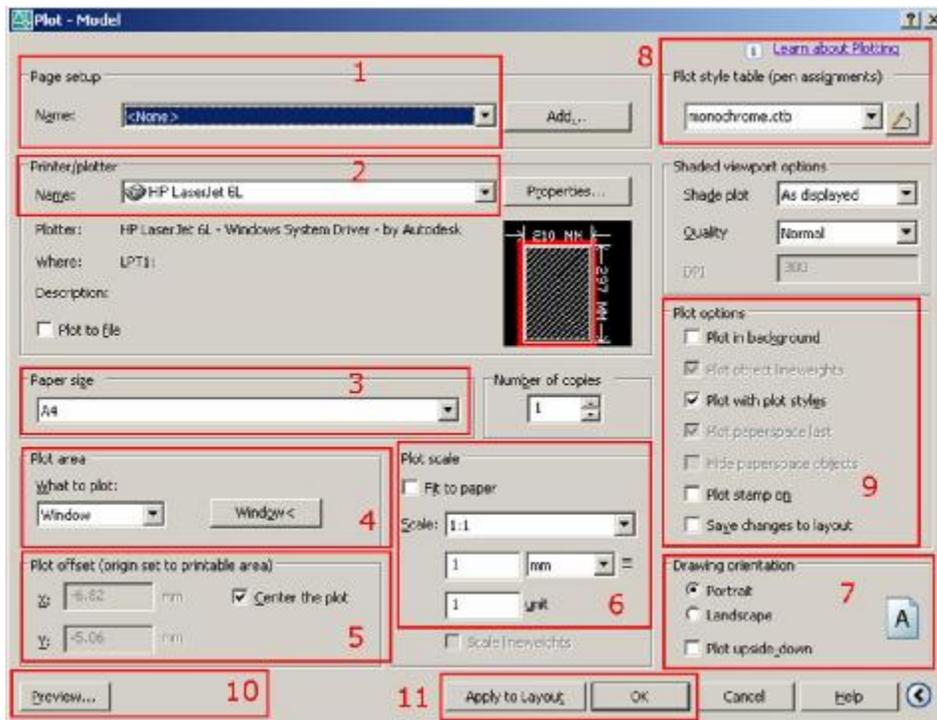
Specify length of segment or [Block]: **50** ↵ (every part equal 50 mm)

Command: **measure** ↵
Select object to measure: Pick a line (object) ↵
Specify length of segment or [Block]: **b** ↵
Enter name of block to insert: **a** ↵
Align block with object? [Yes/No] <Y>: ↵
Specify length of segment: **150** ↵

Plotting

When you have done the assignment, print it out. To do this, bring up the print dialog box using any method explained above (**plot** <enter> will work). Set it up to print as shown below. Follow these steps for a successful plot (see diagram below):

1. Select your printer - laser or inkjet will work fine.
2. Select the paper size - "A4" (297x210) is needed in this case.
3. For the "Plot Area", select "Extents" - that will plot everything you drew.
4. Select the checkbox to "Center the Plot" on your sheet of paper (looks better).
5. If "Fit to Paper" is selected by default, uncheck it and select a scale of 1 inch to 1 paper unit. This will make your printout 'life-sized'.
6. Now Preview your drawing. I strongly recommend that you **preview EVERY drawing** you will ever draw in AutoCAD - a lot can go wrong, so you don't want to waste paper (especially when you're using expensive 3'x4' sheets!). If your preview looks good, cancel out of it by clicking on the large red X icon.
7. If you're sure that everything's ok (this is where good habits begin), press the **OK** button.



Note:

You may have to change the paper size in your printer (Use A4 as an example). You may also have to change the rotation or origin of the plot. Check the **Landscape** radio button in the Drawing Orientation section.

BLOCKS

In this lesson you will be introduced to blocks. By definition, a block is a collection of simple entities (**lines, arcs, circles, text, etc.**) that form a more complex entity that normally represents an object in the real world, e.g. a door, a chair, a window, a computer, valve, transistor, diode and others.

There are many advantages to using blocks, here the major ones:

- Blocks are a single entity. This means that you can modify (move, copy, rotate) a block by selecting only one object in it.
- You can build up a library of blocks consisting of the parts that you require many times in your workday. These blocks can be stored in a separate folder and even on a network so that all drafters have access to them.
- Using blocks can help keep your file size down. AutoCAD stores block definitions in its database. When you insert a block, AutoCAD only stores the name of the block, its location (insertion point), scale and rotation. This can be very noticeable in large drawing.
- If you need to change something, you can redefine a block. For example, you draw a chair and turn it into a block. Later, you're told that the size of the chair has changed. Since you used a block you can redefine the block and all of your chairs are updated automatically. If you had drawn (or copied) 100 chairs in your drawing, you would have to manually change each one.
- Blocks can also contain non-graphical information. This means text objects called **attributes**. For example, you have made blocks of different chairs. You can add information to the block such manufacturer, cost, weight, etc. This information stays with the block, but can also be extracted to a database or spreadsheet. This would be useful for things such as a bill of materials. Attributes can also be visible or invisible in your drawing. Another good use of attributes could be a title block.
- You can even easily add internet hyperlinks to blocks so you can connect a block to a page on a supplier's online catalog.

There are two types of blocks you can create: blocks that are **internal** to your current drawing, and those that are **external**, or saved as a separate file. To create the different types, different commands are used.

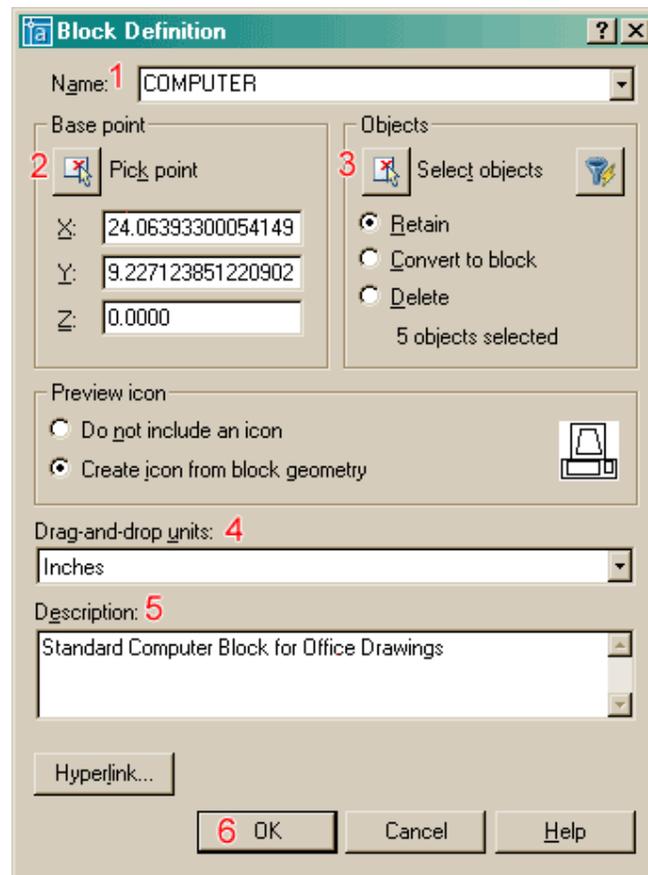
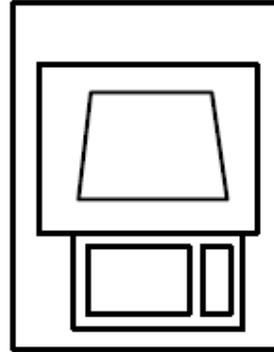
Here are the commands that you will need for using blocks in this lesson:

Command	Keystroke	Icon	Menu	Result
Block	Bmake/ B		Draw > Block > Make	Creates a block from separate entities (internal to current drawing)
Write Block	Wblock / W	None	None	Creates a block and writes it to a file (external)
Insert	Insert / I		Insert > Block	Inserts a block (internal or external)

Draw the computer as shown below. You do not have to dimension it.

Draw the computer as shown below.
You do not have to dimension it.

Start the **BLOCK** command by either typing **B** or using the pull down menu or the icon. You will see a dialog box that looks like the one shown (some versions of AutoCAD will have a slightly different dialog box (with different options). Enter information the same way though.



1 : The first thing that you want to do is give your block a name. Type **COMPUTER** in the edit box beside Block Name.

2 : Now you need to select an insertion/base point. Pick the **Pick Point** button and then pick the midpoint of the bottom line. Make sure that the retain button is selected (this will keep your objects on the screen as individual objects).

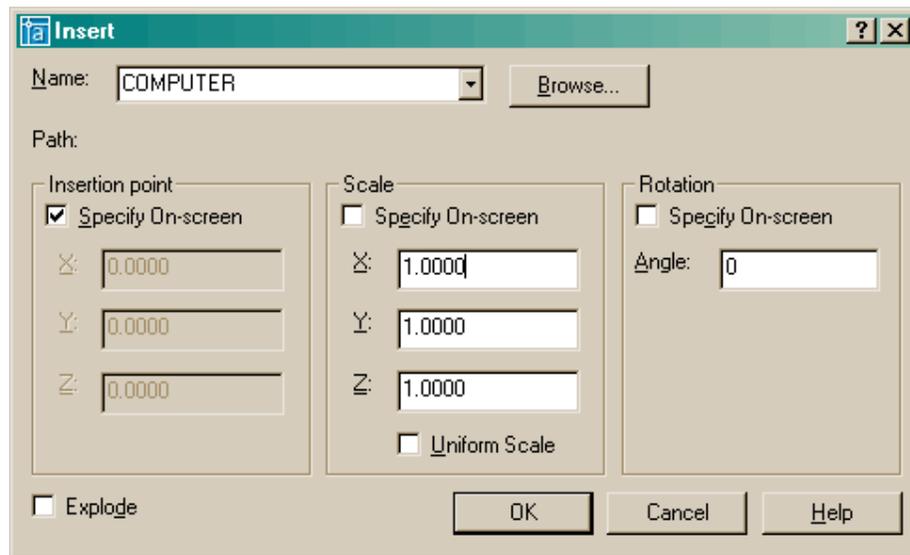
3 : Next you want to select the objects for your block. Pick the **Select Objects** button and then select all the parts of your computer and press <ENTER>.

4 : Now select the drawing units you used to create the original objects in.

5 : This is optional, but you can add a description here. This is good if you are creating specific parts, like maybe a motor and want to add some quick specifications.

6 : Pick the OK Button and the dialog box closes. It will look like nothing happened, but the drawing file now has a "Block Definition" for a Computer in it. Congratulations, you have created your first block.

Now that you have created a block, it's time to insert it. Start the Insert command by typing **I** <ENTER>. You will see this dialog box on the screen:



By default, all the options you need are pre-selected. Since you only have one block in your drawing, its name is displayed.

Make sure that the **Insertion Point - Specify On-screen** box is checked, and the **Explode** button is **not** checked. The Scale - **Specify On-screen** should not be checked. Then press the OK button. Pick anywhere on your screen and you will see the block appear.

Now insert a computer on every desk in your drawing. You can also copy the block instead of re-inserting each time, but make sure you know how to insert.

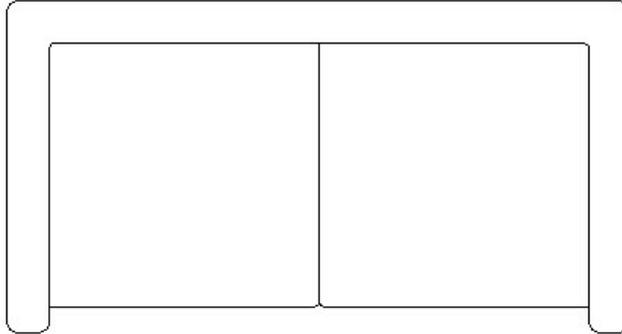
ATTRIBUTES

So far in using AutoCAD you have created geometry: lines, circles, etc. You have also added things such as **text and dimensions**. All of these things could also be done by hand, so what else does CAD have to offer?

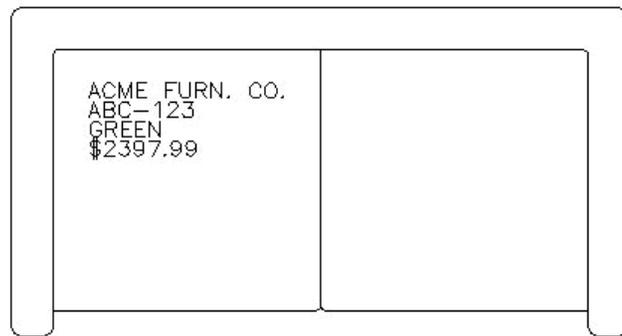
AutoCAD is also a database of information. Most of that database contains the information for reproducing what you have drawn, but you can also add information that is *non-graphical*. One of the easiest ways of adding non-graphical information is to use attributes. An attribute is text that can be attached to a block to convey more information than just the geometry on its own could convey.

Look at the two drawings below for an example:

The first example shows some lines and arcs that could be anything.



The second example shows the same geometry with the attributes visible so that you see what the shapes represent.



The pictures above show a couch. The attributes describe what the model number, color and cost are as well as the manufacturer. From this simple example, you can see that AutoCAD has a useful tool for showing more than just geometry.

This example shows information about furnishings in an office. Once you have drawn the floor plan, you could insert blocks of furniture that have information about the manufacturer, price, weight, and any other information you may need. This information can then be extracted out of AutoCAD and then used in a spreadsheet or other program which could generate a Bill of Materials. You can also use custom programs to use them in the CAD drawing.

In this lesson you will be creating attributes and attaching them to the computer block that you made previously. Like everything you do in AutoCAD, there are particular steps involved when you work with attributes.

1. First you have to define (or create) the attribute.
2. Next you will create the block with attributes.
3. Finally, when you insert the block you will give it the specific information.

Many of the commands that you will be using are tough to remember at first. You may find it easier to use icons (but then the icons look very similar). Pull down menus are another option, so you can see exactly what the command is that you are starting.

Command	Keystroke	Icon	Menu	Result
Define an attribute	DDATTDEF / ATT		Draw > Block > Define Attribute	Creates an attribute definition
Edit attributes	DDATTE / ATE		Modify> Object> Attrb.> Single	Edits the contents of an existing attribute
Block	Block / Bmake		Draw > Block > Make	Creates a block from separate entities and attributes.
Display Atts.	ATTDISP	None	None	Hides or shows attributes
Extract Attributes	EATTEXT		Tools > Attribute Extraction...	Extracts attributes using the wizard

Open up the floor plans you drew earlier with the Computer blocks.

Insert a computer block and then explode it (type **x** <ENTER> and then select the block - press <ENTER>.)

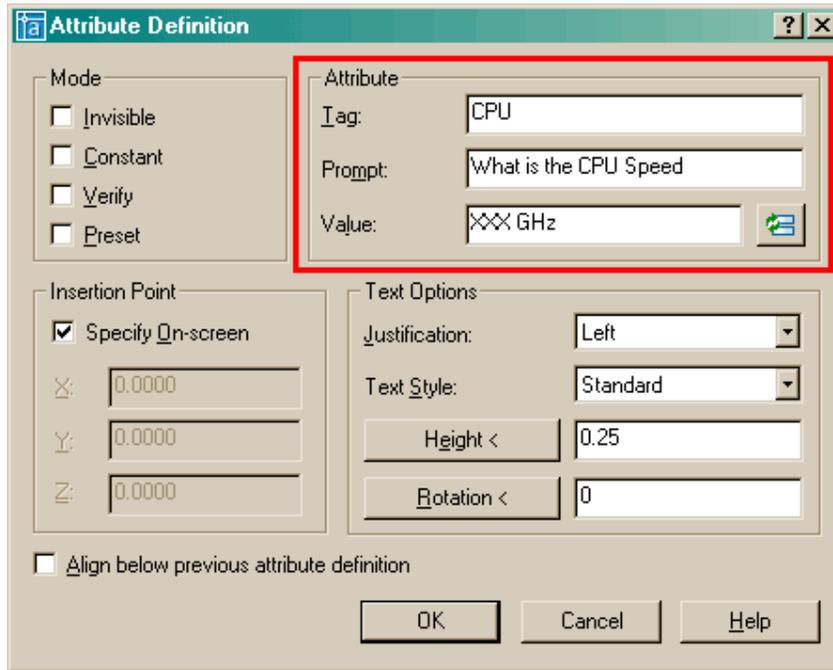
Begin the Attribute definition command: **ATT** (or) **ATTDEF** (or) **DDATTDEF** (AutoCAD has kept the commands from previous versions active in the new versions.)

Look at the dialog box below. Fill in the edit boxes just like the example. Make sure to adjust the text height if necessary.

What you did is give the attribute its definition - a name (tag), a prompt to help the user inserting it, and a (default) value .

Once everything is entered, select the OK button.

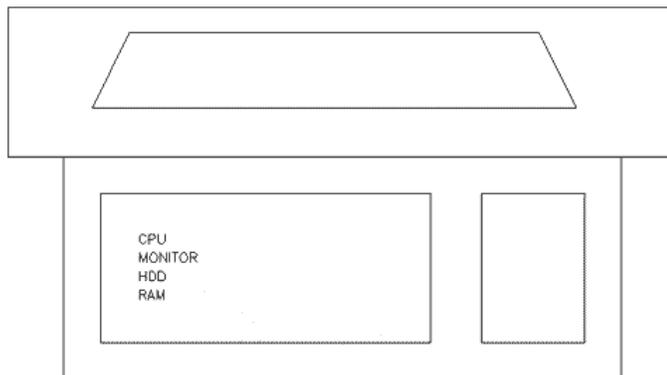
When you are back on the drawing screen, pick a point near the middle of the computer block. When you are done, the dialog box comes back so press OK.



Add the following attributes the same way, except instead of picking a point, check off the **Align below previous attribute** checkbox.

TAG	PROMPT	VALUE
MONITOR	What is the MONITOR SIZE?	XX"
HDD	What capacity is the Hard Drive?	X.XXGB
RAM	How much RAM is installed?	XXXMB
STATION	What station is this	XXX

Once all the attributes are created you should have something like this:



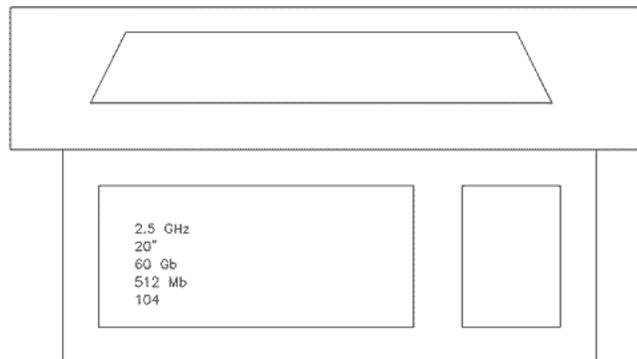
Now create a block that includes your four attributes.

Start up the **BLOCK** command. Create it as shown in the previous lesson. When you are asked for the block name, give it the name **COMP-AT**.

Insert the block and you will get a dialog box where you can enter the values for the tags. You will be prompted to provide answers to the prompts that you defined in the attributes.

```
Command: I ↵ INSERT ↵  
Specify insertion point or  
[Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]:  
Enter attribute values  
What is the CPU Speed <XXX GHz>: 2.6 GHz ↵  
What is the monitor size <XX">: 21" ↵  
What is the capacity of the Hard Drive <XXX Gb>: 200Gb ↵  
How much ram is installed <XXX Mb>: 1024Mb ↵
```

After inserting the block and answering all the prompts, your block should look like this:



Now by looking at the drawing, you can see exactly what type of computer it is and what station (location) it is. How much handier is *THAT*?

Autocad Summary

- CTRL+0:** Toggles Clean Screen
- CTRL+1:** Toggles the Properties palette
- CTRL+2:** Toggles the DesignCenter palette
- CTRL+3:** Toggles the Tool Palettes Window
- CTRL+4:** Toggles the Sheet Set Manager
- CTRL+6:** Toggles the dbConnect Manager
- CTRL+7:** Toggles the Markup Set Manager
- CTRL+8:** Toggles the QuickCalc calculator palette
- CTRL+9:** Toggles the command window
- CTRL+A:** Selects all objects in the drawing
- CTRL+SHIFT+A:** Toggles Groups
- CTRL+B:** Toggles Snap mode
- CTRL+C:** Copies objects to Clipboard
- CTRL+SHIFT+C:** Copies objects to Clipboard with Base Point
- CTRL+D:** Toggles Dynamic UCS
- CTRL+E:** Cycles through isometric planes
- CTRL+F:** Toggles running object snaps
- CTRL+G:** Toggles the grid
- CTRL+H:** Toggles PICKSTYLE
- CTRL+I:** Toggles COORDS
- CTRL+J; CTRL+M:** Repeats the last command (pressing Enter does the same)
- CTRL+L:** Toggles Ortho mode
- CTRL+N:** Creates a new drawing
- CTRL+O:** Opens an existing drawing
- CTRL+P:** Prints/plots the current drawing
- CTRL+R:** Cycles through layout viewports
- CTRL+S:** Saves the current drawing
- CTRL+SHIFT+S:** Opens the Save As dialog box
- CTRL+T:** Toggles TABLET mode
- CTRL+V:** Pastes data from the Clipboard
- CTRL+SHIFT+V:** Pastes data from the Clipboard as a Block
- CTRL+X:** Cuts objects to Clipboard
- CTRL+Y:** Cancels the preceding Undo action
- CTRL+Z:** Reverses the last action
- CTRL+; CTRL+\:** Cancels the current command (pressing Esc does the same)
- CTRL+PAGE UP:** Moves to the next layout tab to the left of the current tab
- CTRL+PAGE DOWN:** Moves to the next layout tab to the right of the current tab
- ALT+F11:** Displays the Visual Basic Editor
- ALT+F8:** Displays the Macros dialog box

Shortcut Keys for AutoCAD

A	Arc	ICL	IMAGECLIP
AR	Array	IM	IMAGE
AT	Attributes	-IM	IMAGE
BR	Break	IMP	IMPORT
C	Circle	IN	INTERSECT
CH	Properties	INF	INTERFERE
D	Dimension Style	IO	INSERTOBJ
DI	Distance	L	LINE
DO	Donut	LA	LAYER
DT	Dtext	-LA	LAYER
E	Erase	LE	LEADER
ED	Ddedit	LEN	LENGTHEN
EL	Ellipse	LI	LIST
EX	Extend	LS	LIST
F	Fillet	LT	LINETYPE
F1	Help	-LT	LINETYPE
F2	Text Window	LTS	LTSCALE
F3	Object Snap Menu	M	MOVE
F5	Isoplane Toggle	MA	MATCHPROP
F6	Coordinates	ME	MEASURE
F7	Grid Toggle	MI	MIRROR
F8	Ortho Toggle	ML	MLINE
F9	Snap Toggle	MO	DDMODIFY
F1	FILTER	MS	MSPACE
G	GROUP	MT	MTEXT
-G	GROUP	MV	MVIEW
GR	DDGRIPS	O	OFFSET
H	BHATCH	OS	DDOSNAP
-H	HATCH	-OS	-OSNAP
HE	HATCHEDIT	P	PAN
HI	HIDE	-P	-PAN
I	DDINSERT	ICL	IMAGECLIP
-I	INSERT	IM	IMAGE
IAD	IMAGEADJUST	-IM	IMAGE
IAT	IMAGEATTACH	IMP	IMPORT

PA	PASTESPEC	AT	DDATTDEF
PE	PEDIT	-AT	ATTDEF
PL	PLINE	ATE	DDATTE
PO	POINT	-ATE	ATTEDIT
POL	POLYGON	B	BMAKE
PR	PREFERENCES	-B	BLOCK
PRE	PREVIEW	BH	BHATCH
PRINT	PLOT	BO	BOUNDARY
PS	PSPACE	-BO	-BOUNDARY
PU	PURGE	BR	BREAK
R	REDRAW	C	CIRCLE
RA	REDRAWALL	CH	DDCHPROP
RE	REGEN	-CH	CHANGE
REA	REGENALL	CHA	CHAMFER
REC	RECTANGLE	COL	DDCOLOR
3A	3DARRAY	CO	COPY
3F	3DFACE	D	DDIM
3P	3DPOLY	DAL	DIMALIGNED
A	ARC	DAN	DIMANGULAR
AA	AREA	DBA	DIMBASELINE
AL	ALIGN	DCE	DIMCENTER
AP	APPLOAD	DCO	DIMCONTINUE
AR	ARRAY	DDI	DIMDIAMETER
AAD	ASEADMIN	DED	DIMEDIT
AEX	ASEEXPORT	DI	DIST
ALI	ASELINKS	DIV	DIVIDE
ASQ	ASESQLED	DLI	DIMLINEAR
ARO	ASEROWS	DO	DONUT
ASE	ASESELECT	DOR	DIMORDINATE
PA	PASTESPEC	DOV	DIMOVERRIDE
PE	PEDIT	DR	DRAWORDER
PL	PLINE	DRA	DIMRADIUS
PO	POINT	DST	DIMSTYLE
POL	POLYGON	DT	DTEXT
		DV	DVIEW

E	ERASE	MI	Mirror
ED	DDEDIT	ML	Multiline Text
EL	ELLIPSE	MT	Multiline Text
EX	EXTEND	O	Offset
EXIT	QUIT	OS	Object Snap Menu
EXP	EXPORT	P	Pan
EXT	EXTRUDE	P	Previous Selection
F	FILLET	PE	Polyline Edit
REC	RECTANGLE	PL	Polyline
REG	REGION	PR	Preferences
REN	DDRENAME	R	Redraw
-REN	RENAME	R	Remove from Selection
REV	REVOLVE	RE	Regenerate
RM	DDRMODES	REC	Rectangle
RO	ROTATE	RM	Drawing Aids
RPR	RPREF	RO	Rotate
RR	RENDER	S	Stretch
S	STRETC	SC	SCALE
GR	Grips	SCR	SCRIPT
H	Hatch	SE	DDSELECT
HI	Hide	SEC	SECTION
L	Line	SET	SETVAR
LA	Layers	SHA	SHADE
LE	Leader	SL	SLICE
LEN	Lengthen	SN	SNAP
LI	List	SO	SOLID
LT	Linetype	SP	SPELL
M	Move	SPL	SPLINE
MA	Match Property	SPE	SPLINEDIT
ME	Measure	ST	STYLE
E	ERASE	MI	Mirror
ED	DDEDIT	ML	Multiline Text
EL	ELLIPSE	MT	Multiline Text
EX	EXTEND	O	Offset
EXIT	QUIT	OS	Object Snap Menu

SU	SUBTRACT	T	Text
T	MTEXT	TI	Tilemode
-T	-MTEXT	TO	Toolbars
TA	TABLET	TR	Trim
TH	THICKNESS	U	Undo
TI	TILEMODE	X	Explode
TO	TOOLBAR	Z	Zoom
TOL	TOLERANCE		
TOR	TORUS		
TR	TRIM		
UC	DDUCS		
UCP	DDUCSP		
UN	DDUNITS		
-UN	UNITS		
UNI	UNION		
V	DDVIEW		
-V	VIEW		
VP	DDVPOINT		
-VP	VPOINT		
W	WBLOCK		
WE	WEDGE		
X	EXPLODE		
XA	XATTACH		
XB	XBIND		
-XB	-XBIND		
XC	XCLIP		
XL	XLINE		
XR	XREF		
-XR	-XREF		
Z	ZOOM		
SU	SUBTRACT		
T	MTEXT		
-T	-MTEXT		
TA	TABLET		
TH	THICKNESS		