



1. Interface

The term CAD (Computer Aided Design) applies to a wide range of programs that allow the user to create drawings, plans, and designs electronically. AutoCAD is one such program and its main claim to fame is that it is relatively easy to use, it is very comprehensive in its ability to create 2D and some 3D drawings, and it is very popular. Seventy percent of the CAD users in the world use AutoCAD.

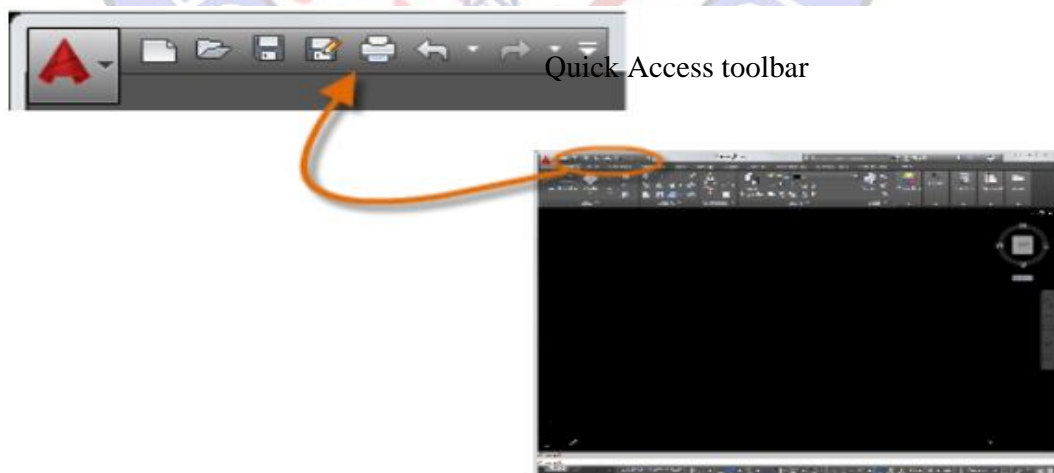
1.1. Starting AutoCAD

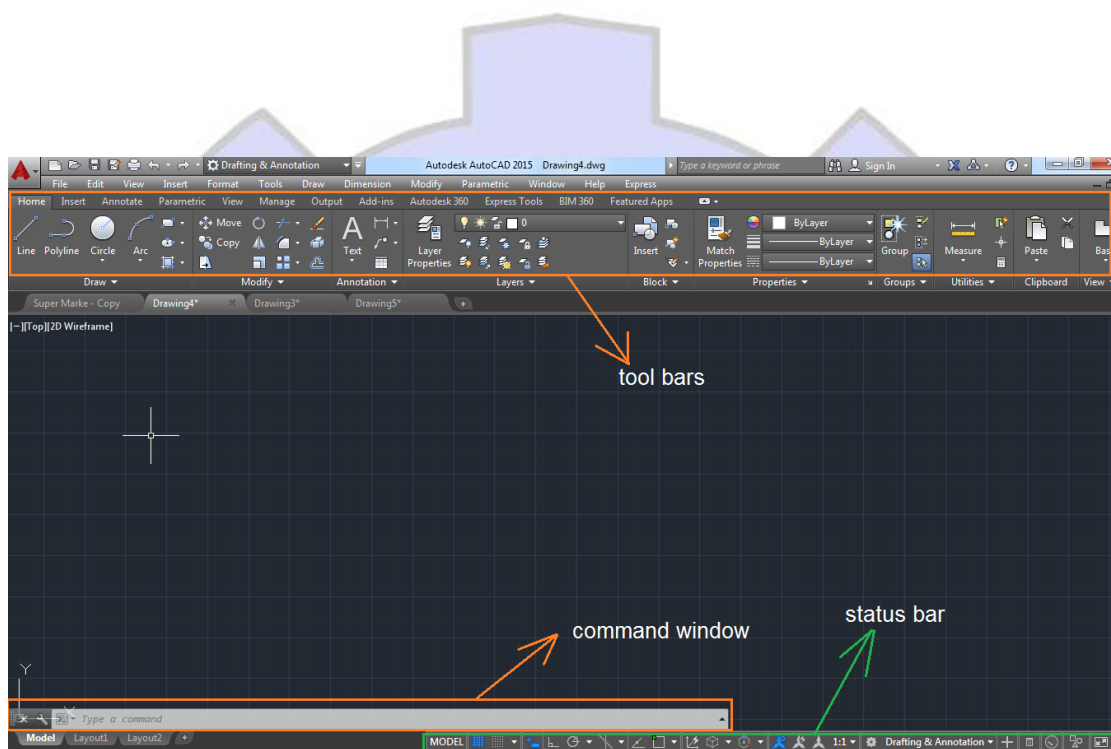
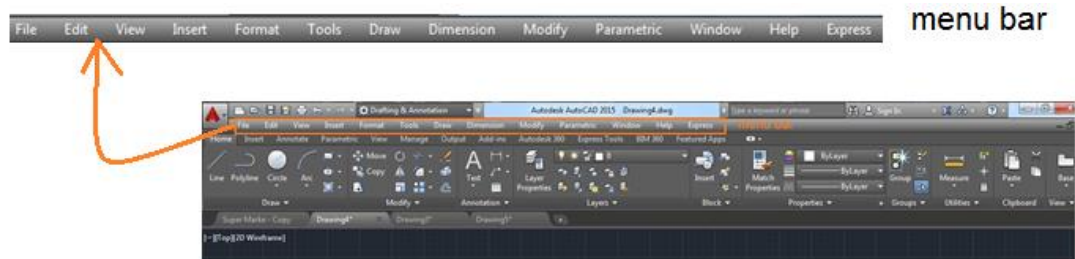
You can start AutoCAD by either double clicking on the program Icon on the desktop or by clicking on the program name in the Start menu. The dialog box in the middle will aid you in getting started at either creating a new drawing or continuing your work on a drawing that is not finished the program will start and should display a screen similar to the one shown below which is called (drawing area).

If you are continuing work on a drawing, click on the “A” icon in the extreme upper left corner of the window and **Open->Drawing**. A “Select File” dialog box will open allowing you to select the drawing file you want to open.

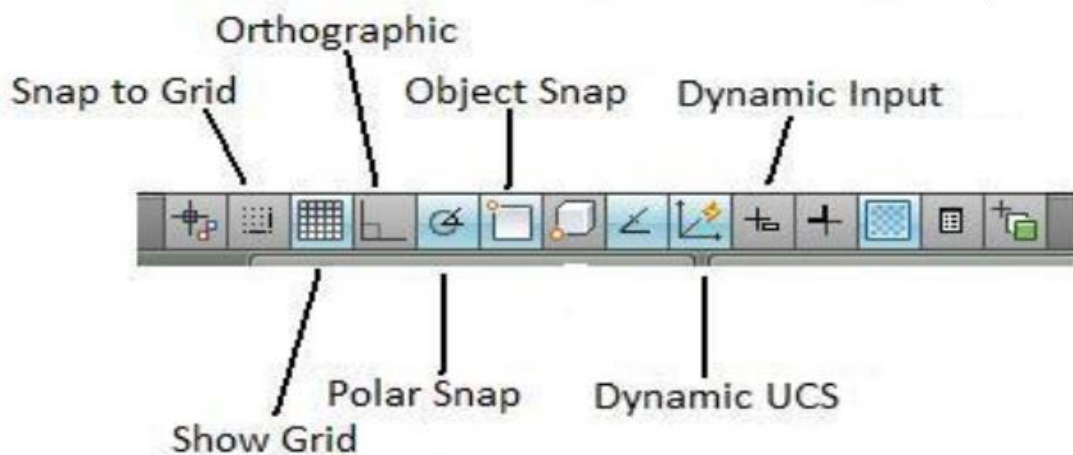
1.2. The AutoCAD screen areas

- Quick Access toolbar
- Menu bar
- Tool bars
- Document window or drawing area
- Command window
- Status bar






Status Bar

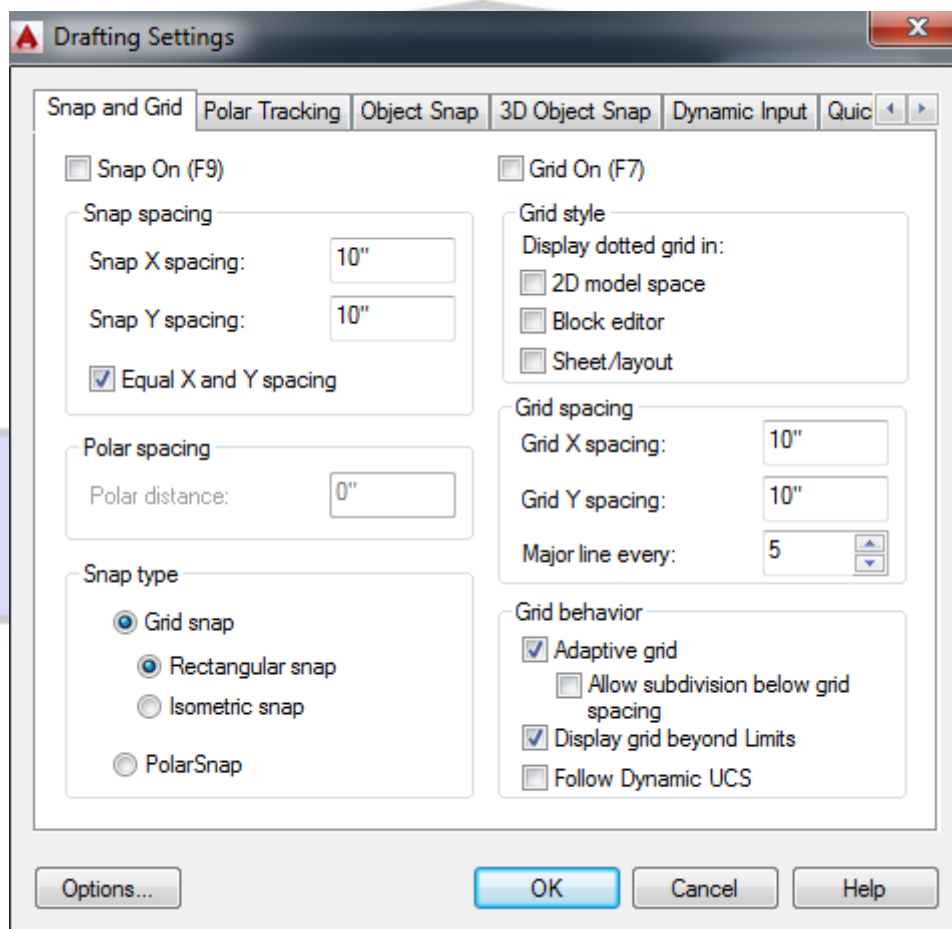




a. Alignment Grid

The grid command allows you to set an alignment grid of dots of any desired spacing, making it easier to visualize distances and drawing size. you can turn the grid on and off by pick the grid button located in the status bar or (by pressing  Ctrl + g or the F7 function key). The drafting setting dialog box permits you to review and make changes to the grid settings.

Menu bar \Rightarrow Tools \Rightarrow drafting setting \Rightarrow snap and grid



b. Snap grid

The snap grid is similar to the visual grid, but it is an invisible one. You cannot see the snap feature, but you can see the effects of it as you move the crosshairs Pick the snap button in the status bar to turn on the snap grid, (or pressing Ctrl + b or the F9 function key)

c. The Ortho Mode

Ortho short for orthogonal, allows you to draw horizontal or vertical lines quickly and easily. Ortho is on when the Ortho button on the status bar is depressed. You can toggle Ortho on and off by clicking the Ortho button. (or press F8 function key).

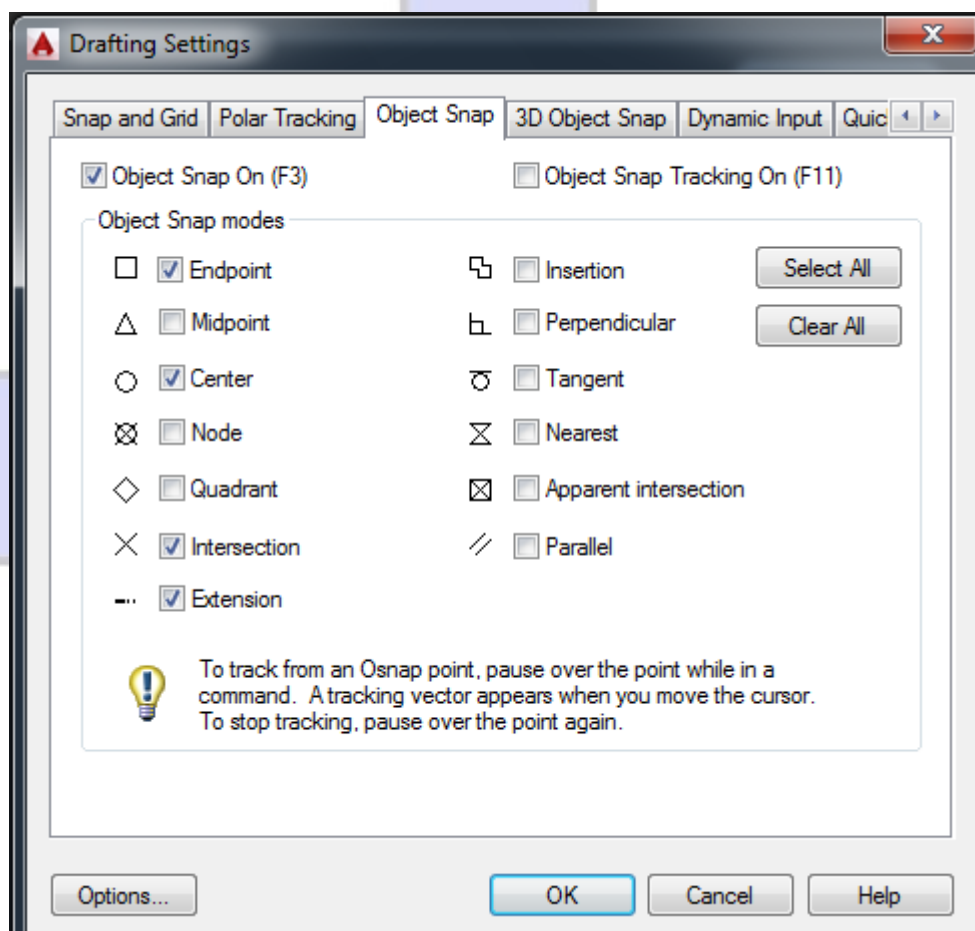


d. Polar

Polar makes it easy to draw lines at regular angular increments, such as 30, 45, or 90 degree. Using the F10 key or polar button toggles polar tracking on or off.

e. Object Snap (Osnap)

AutoCAD provides a capability called "object snap", or OSNAP for short, that enables you to "snap" to existing object end points, midpoints, centers, intersections, etc. available object snap mode are illustrated below:



f. Object Snap Tracking (Otrack)

Object snap tracking is active when the alignment paths appear from one or more acquired object snaps. This feature is a part of AutoCAD's Auto tracking. You can toggle Auto Tracking on and off with the Otrack button on the status bar or by toggling F11.



g. DYN

A feature that enables you to enter data or commands near the mouse pointer instead of entering in "the" command line

"To control the dynamic input properties right click on button and select setting

h. LWT

LWT short of line weight produces the line weight settings dialog box, this dialog box assigns line weights to objects.

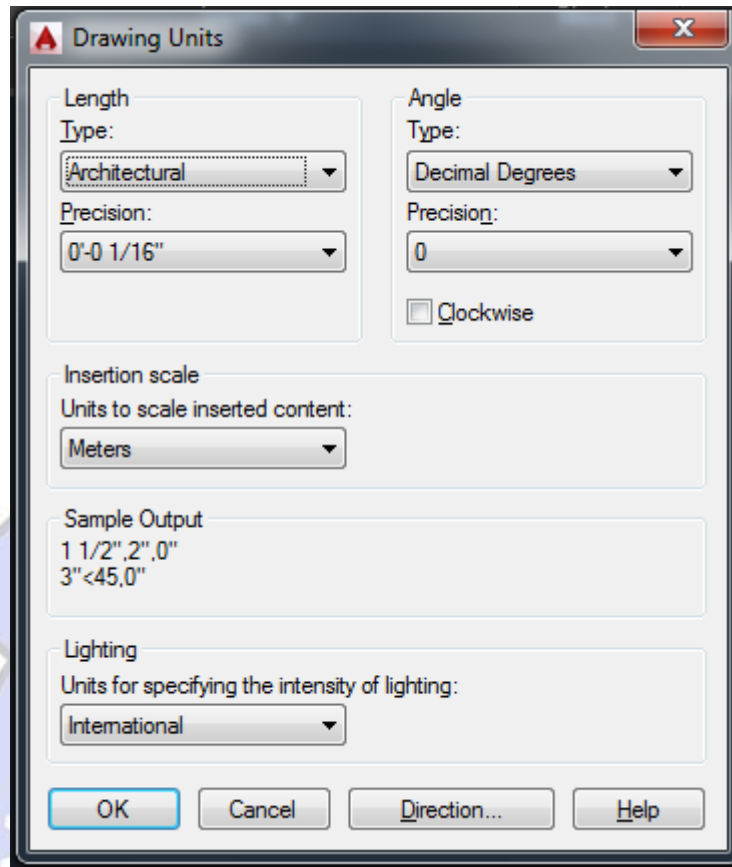
i. Model

Model Space used to construct engineering model where paper space used to print or draw engineering model.

j. Units

The units command open dialog box to determine measurement units for drawing angles and direction and precision. Access the command by:

- **Menu bar** ⇒ **format** ⇒ **units**
- **Length type.** Determine length types (scientific, decimal, engineering, architectural, and fractional).
- **Angle type.** Determine angle type. (The default decimal degree).
- **Precision.** Determine accuracy of angles and lengths. Precision is the number of places to the right of the decimal to display.
- **Clockwise.** Control angle direction.



k. Limits

Limits used to determine the limits of board. To access the command by:

Menu bar ⇒ **format** ⇒ **Drawing limits**

Specify lower left corner or [on / off] <0.0000, 0.0000>:

Specify upper right corner <420,297>:

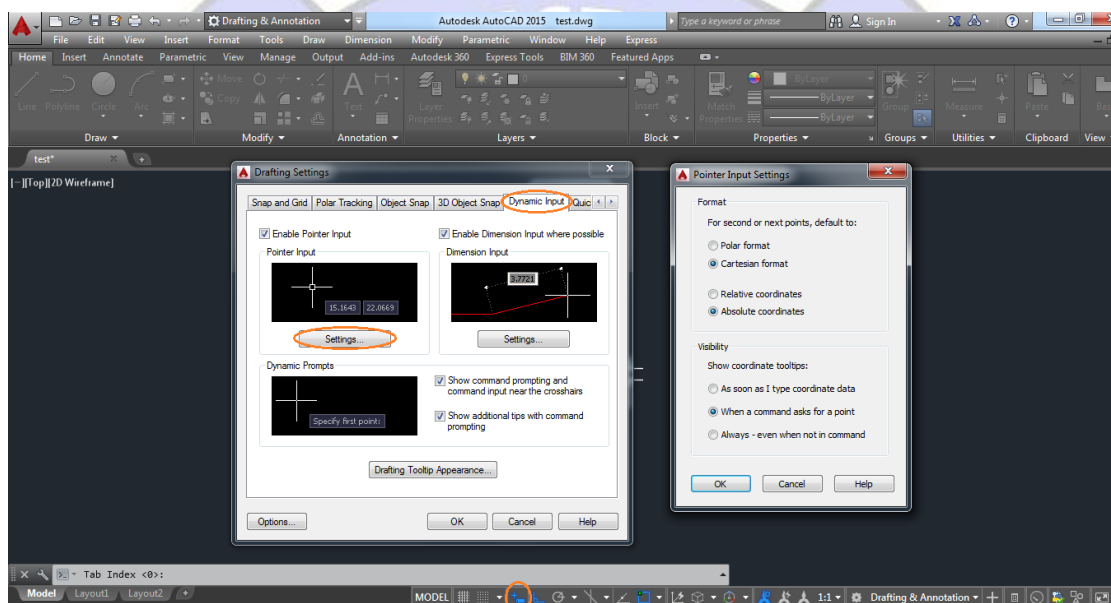
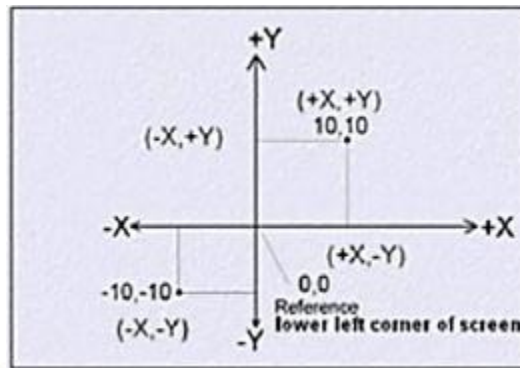
1.3.Cartesian and polar Coordinate System.

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

Each of these axis is further divided into segments. Each segment is given a value. The X axis segments increase in value to the right. The positive X values are to the right of the intersection of the two axis. The negative X values are to the left. The positive Y values are above the intersection and increase up. The negative Y values are below. Polar coordinates use a distance and an angle to locate a point. With both

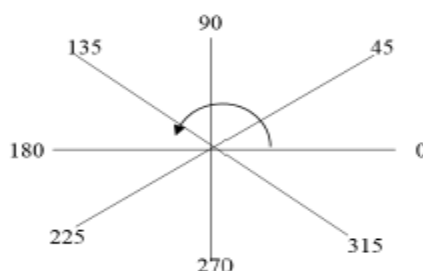


Cartesian and polar coordinates, can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.



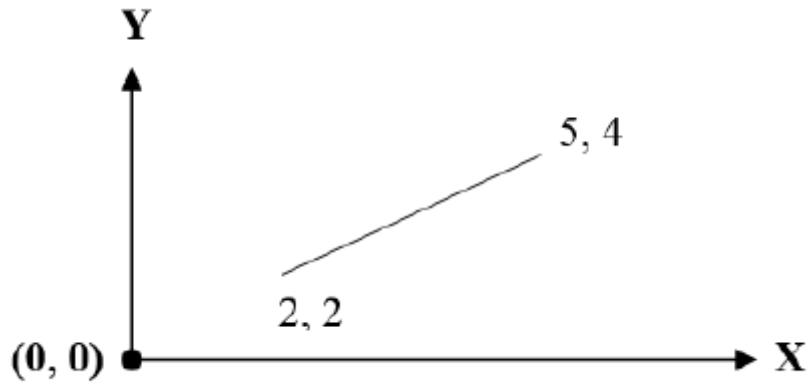
1.4. Angles in AutoCAD Programs

Angles measure sets the direction for angle 0. east (x positive) is the AutoCAD default. (Select clock wise from units dialog box if you want to change the AutoCAD default setting for measuring angles).





Example.



1. Draw Line using Absolute Cartesian Coordinate System.

Command line: Line

Specify start Point: 2, 2

Specify next Point: 5, 4

The length of line calculated from original point (0, 0).

2. Draw Line using Relative Cartesian Coordinate System.

Command line: Line

Specify start Point: 2, 2

Specify next Point: @3, 2

The length of line calculated from point (2, 2)

3. Draw Line using Absolute Polar Coordinate System

Command line: Line

Specify start Point: 2, 2

Specify next Point: 6.40<38.65

The length of line calculated from original point (0,0) using fethagorian triangle as following:

$$L = \sqrt{x^2 + y^2} = \sqrt{5^2 + 4^2} = 6.40$$

$$\frac{4}{5} = 0.8, \tan^{-1} 0.8 = 36.65^\circ$$

4. Draw Line using Relative Polar Coordinate System

Command line: Line

Specify start Point: 2, 2

Specify next Point: @3.60<33.69

The length of line calculated from point (2, 2) using fethagorian triangle as following:

$$L = \sqrt{x^2 + y^2} = \sqrt{3^2 + 2^2} = 3.60$$

The angle is calculated as following:

$$\frac{3}{2} = 0.66, \tan^{-1} 0.66 = 33.69^\circ$$



1.5. Methods of Zooming

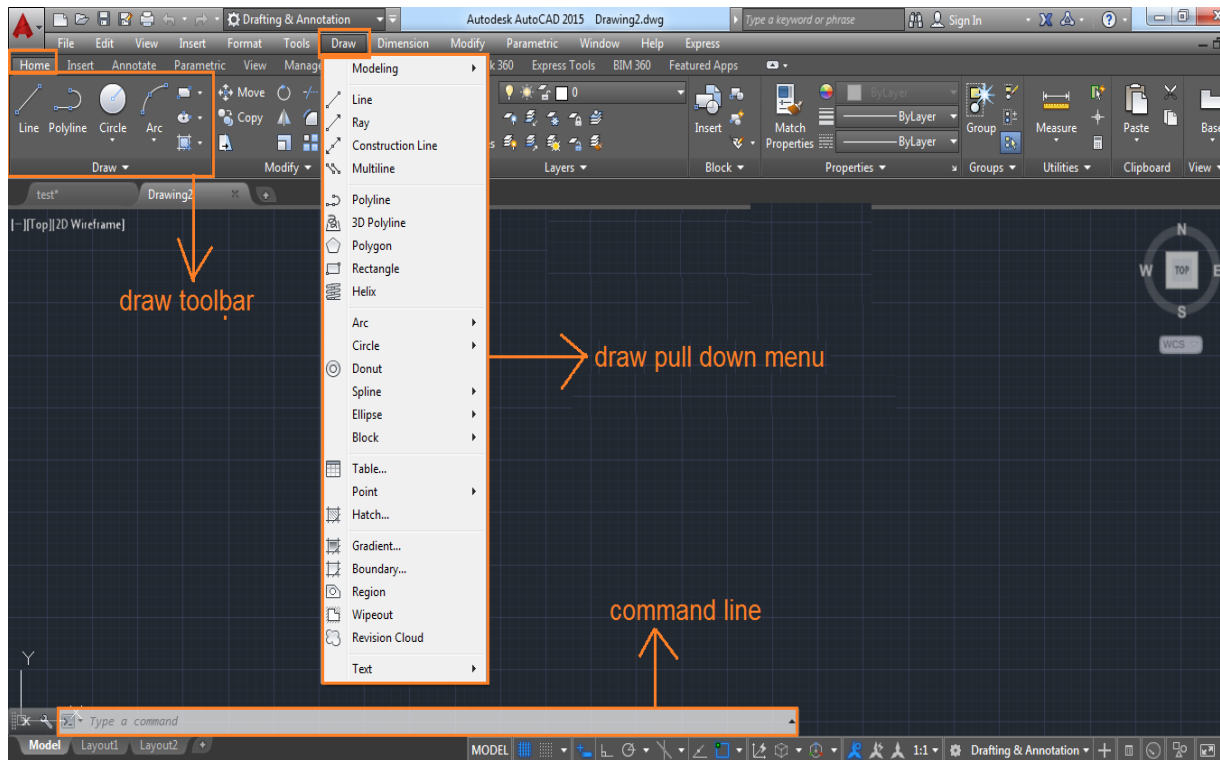
Zoom: zooming does not change the size of the drawing objects, zooming change only the display of objects.

- **Real time:** Real time is the default option of zoom. If you type zoom, just press enter to activate the real time option. You can type Rt zoom to invoke this option directly.
- **Window:** to zoom with a window is to draw a rectangular window around the desired viewing area. The windowed area is magnified to fill the screen.
- **All:** or enter Z and A at the key board. AutoCAD zooms the drawing to its original size.
- **Extents,** or enter Z and E at the keyboard. AutoCAD zooms the drawing as large as possible while still showing the entire drawing on the screen.
- **Scale:** this option allows to enter a scale factor for the desired display.
- **In:** zoom in magnifies the current display by a factor of 2X.
- **Out:** zoom out makes the current display smaller by a factor of 0.5X.
- **Center:** first specify a location as the center of the zoomed area. Then either specify a magnification factor, a height value for the resulting display, or pick two points forming a vertical to indicate the height for the resulting display.
- **Previous:** selecting this option automatically changes to the previous display.

Pan: Pan means to move the display area slightly without changing the size of current view window. Using the pan function, you drag the drawing across the screen to display an area outside of the current view in the drawing editor.

2. Drawing Commands

Drawing commands can be entered from the keyboard (command line), the draw toolbar, or the draw pull down menu.



2.1.Line

Draw Line simple lines or series of lines segments .access to command line by one way of the following:

1. **Command Line:** *Line* or *L*
2. **Menu Bar:** *Draw* → *Line*
3. **Draw Bar**



When execute command line the program requires specify first point, display in command line:

Specify first Point:

Specify next point or [Undo]:

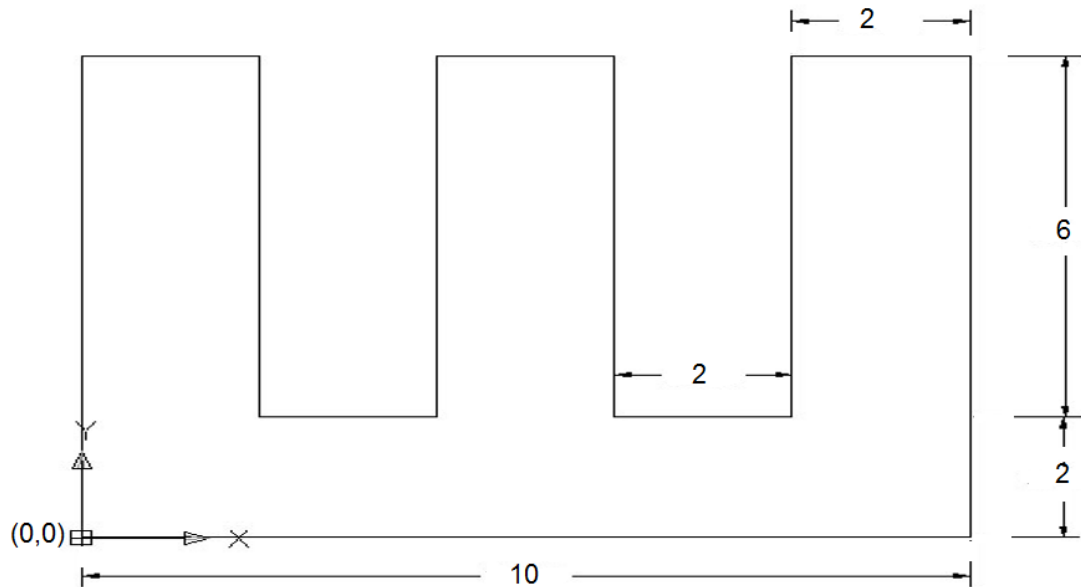
Specify next point or [Close/Undo]:

Options:

- **C:** this option close series of lines, connect first point with last point by line.
- **U:** enter U for undo. AutoCAD backs up one segment, undoing it so that you can recreate it.



Example: Draw the figure below, start from the original point (0, 0)



Absolute Coordinate System

Command: Line

Specify first point: 0, 0
Specify next point: 10, 0
Specify next point: 10, 8
Specify next point: 8, 8
Specify next point: 8, 2
Specify next point: 6, 2
Specify next point: 6, 8
Specify next point: 4, 8
Specify next point: 4, 2
Specify next point: 2, 2
Specify next point: 2, 8
Specify next point: 0, 8
Specify next point: 0, 0 or C

Relative Coordinate System

Command: Line

Specify first point: 0,0
Specify next point: @10,0
Specify next point: @0,8
Specify next point: @-2,0
Specify next point: @0,-6
Specify next point: @-2,0
Specify next point: @0,6
Specify next point: @-2,0
Specify next point: @0,-6



Specify next point:@-2,0
Specify next point:@0,6
Specify next point:@-2,0
Specify next point:@0,-8 or C

Relative Polar Coordinate System

Command: Line

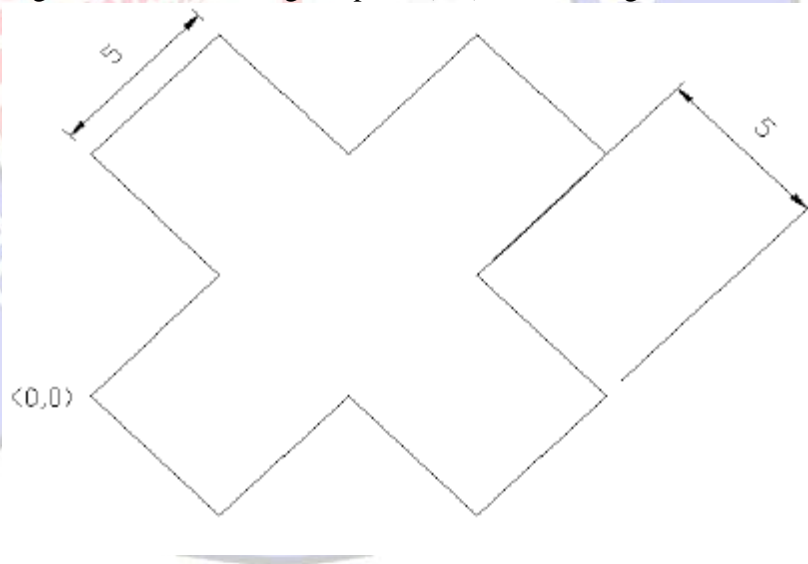
Specify first point:0,0
Specify next point:@10<0
Specify next point:@8<90
Specify next point:@2<180
Specify next point:@6<270
Specify next point:@2<180
Specify next point:@6<90
Specify next point:@2<180
Specify next point:@6<270
Specify next point:@2<180
Specify next point:@6<90
Specify next point:@2<180
Specify next point:@8<270 or C

Example: Draw the figure note the figure is start from original point (0,0) .and the angle between sides is 45

Absolute Coordinate System

Command: Line

Specify first point: 0, 0
Specify next point: 3.5,-3.5
Specify next point: 7, 0
Specify next point: 10.5,-3.5
Specify next point: 14, 0
Specify next point: 10.5, 3.5
Specify next point: 14, 7
Specify next point:10.5,10.5
Specify next point:7,7
Specify next point:3.5,10.5
Specify next point:0,7
Specify next point:3.5,3.5
Specify next point:0,0 or C



Relative Coordinate System

Command: Line

Specify first point:0,0
Specify next point:@3.5,-3.5
Specify next point:@3.5,3.5
Specify next point:@3.5,-3.5
Specify next point:@3.5,3.5
Specify next point:@-3.5,3.5
Specify next point:@3.5,3.5



Specify next point:@-3.5,3.5
Specify next point:@-3.5,-3.5
Specify next point:@-3.5,3.5
Specify next point:@-3.5,-3.5
Specify next point:@3.5,-3.5
Specify next point:@-3.5,-3.5 or C

Relative Polar Coordinate System

Command: Line

Specify first point:0,0
Specify next point:@5<315
Specify next point:@5<45
Specify next point:@5<315
Specify next point:@5<45
Specify next point:@5<135
Specify next point:@5<45
Specify next point:@5<135
Specify next point:@5<225
Specify next point:@5<135
Specify next point:@5<225
Specify next point:@5<315
Specify next point:@5<225 or C

2.2. Rectangle

1. **Command Line:** *Rectangle, Rec.*

2. **Menu** *Bar: Draw* ⇒ *Rectangle.*

3. **Draw Bar.**

Options:

RECTANG Specify first corner point or [Chamfer Elevation Fillet Thickness Width]:

C:

The chamfer command enables to place a chamfer at the rectangle corner. You can set the chamfer distance for the two lines independently.

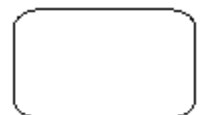
Specify first chamfer distance for rectangles<0.000>:

Specify second chamfer distance for rectangles<0.000>:



F: the fillet command creates both fillets and rounds on any combination of two lines, arcs, or circles.

Specify fillet radius for rectangles<0.000>:



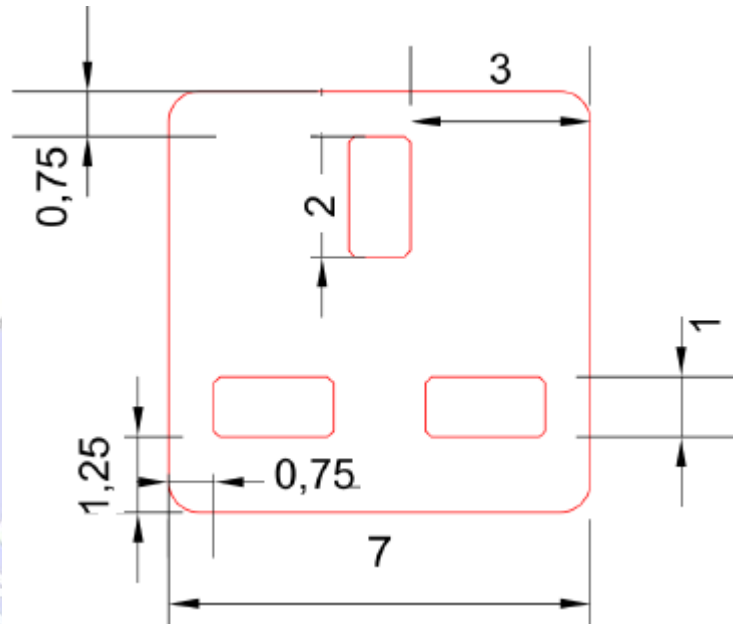
W: rectangle can have a width (the default width is 0).





Specify line width for rectangles<0.000>:

Example . Draw the figure below (13 Ampere socket outlet), note the figure is start from the point (0,0)



Absolute Coordinate System

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: f

Specify fillet radius for rectangles: 0.5

Specify first corner point: 0, 0

Specify other corner Point: 7, 7

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C Specify first chamfer distance for rectangles: 0.1

Specify second chamfer distance for rectangles: 0.1

Specify first corner point: 0.75, 1.25

Specify other corner Point: 2.75, 2.25

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.1

Specify second chamfer distance for rectangles: 0.1

Specify first corner point: 4.25, 1.25

Specify other corner Point: 6.25, 2.25

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.1

Specify second chamfer distance for rectangles: 0.1

Specify first corner point: 3, 4.25

Specify other corner Point: 4, 6.25



Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: f

Specify fillet radius for rectangles: 0.5

Specify first corner point: 0, 0

Specify other corner Point: @7, 7

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.1

Specify second chamfer distance for rectangles: 0.1

Specify first corner point: 0.75, 1.25

Specify other corner Point: @2, 1

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.1

Specify second chamfer distance for rectangles: 0.1

Specify first corner point: 4.25, 1.25

Specify other corner Point: @2, 1

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.1

Specify second chamfer distance for rectangles: 0.1

Specify first corner point: 3, 4.25

Specify other corner Point: @1, 2

Relative Polar Coordinate System

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: f

Specify fillet radius for rectangles: 0.5

Specify first corner point: 0, 0

Specify other corner Point: @9.89<45

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.5

Specify second chamfer distance for rectangles: 0.5

Specify first corner point: 0.75, 1.25

Specify other corner Point: @2.23<26.56

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.5

Specify second chamfer distance for rectangles: 0.5

Specify first corner point: 4.25, 1.25

Specify other corner Point: @2.23<26.56

Command: Rec

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]: C

Specify first chamfer distance for rectangles: 0.5

Specify second chamfer distance for rectangles: 0.5



Specify next point:-2,13
Specify next point:-6,16
Specify next point:-8,16
Specify next point:-8,15
Specify next point:-6,15
Specify next point:-2,12
Specify next point:0,12
Specify next point:0,0 or C

Command: Rec

Specify first corner point:7,16
Specify other corner point:9,17

Command: Rec

Specify first corner point:0,16
Specify other corner point:1,17

Command: Rec

Specify first corner point:-8,16
Specify other corner point:-6,17

Command: C

Specify center point for circle:8,19
Specify radius of circle:2

Command: C

Specify center point for circle:0.5,19
Specify radius of circle:2

Command: C

Specify center point for circle:-7,19
Specify radius of circle:2

Relative Coordinate System

Command: L

Specify first point:0,0
Specify next point:@1,0
Specify next point:@0,12
Specify next point:@2,0
Specify next point:@4,3
Specify next point:@2,0
Specify next point:@0,1
Specify next point:@-2,0
Specify next point:@-4,-3
Specify next point:@-2,0
Specify next point:@0,3
Specify next point:@-1,0
Specify next point:@0,-3
Specify next point:@-2,0
Specify next point:@-4,3
Specify next point:@-2,0
Specify next point:@0,-1
Specify next point:@2,0
Specify next point:@4,-3



Specify next point:@2,0

Specify next point:@0,-12 or C

Command: Rec

Specify first corner point:7,16

Specify other corner point:@2,1

Command: Rec

Specify first corner point:0,16

Specify other corner point:@1,1

Command: Rec

Specify first corner point:-8,16

Specify other corner point:@2,1

Command:C

Specify center point for circle:8,19

Specify radius of circle:2

Command:C

Specify center point for circle:0.5,19

Specify radius of circle:2

Command:C

Specify center point for circle:-7,19

Specify radius of circle:2

Relative Polar Coordinate System

Command:L

Specify first point:0,0

Specify next point:@1<0

Specify next point:@12<90

Specify next point:@2<0

Specify next point:@5<36.86

Specify next point:@2<0

Specify next point:@1<90

Specify next point:@2<180

Specify next point:@5<216.86

Specify next point:@2<180

Specify next point:@3<90

Specify next point:@1<180

Specify next point:@3<270

Specify next point:@2<180

Specify next point:@5<143.14

Specify next point:@2<180

Specify next point:@1<270

Specify next point:@2<0

Specify next point:@5<323.14

Specify next point:@2<0

Specify next point:@12<270 or C

Command: Rec

Specify first corner point:7,16

Specify other corner point:@2.23<26.56

Command: Rec



Specify first corner point:0,16
Specify other corner point:@1.41<45

Command: Rec

Specify first corner point:-8,16
Specify other corner point:@2.23<26.56

Command: C

Specify center point for circle:8,19
Specify radius of circle:2

Command: C

Specify center point for circle:0.5,19
Specify radius of circle:2

Command: C

Specify center point for circle:-7,19
Specify radius of circle:2

2.4. Arc

Arc command is often used to draw arcs.

1. **Command Line:** Arc or a

2. **Menu Bar:** Draw ⇒ Arc

3. **Draw Bar**

The following message appear:

ARC Specify start point of arc or [Center]:

Specify start point of arc or [C Enter]:

Specify second point of arc or [C Enter/ End]:

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

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

Options:

Angle. Draw arc using angle.

Chord Length. Draw arc using chord length.

Example: Draw the figure below. Start from the point (1, 1)

Absolute Coordinate System

Command: L

Specify first point:1,1

Specify next point:8,1

Specify next point:8,2

Specify next point:4,2

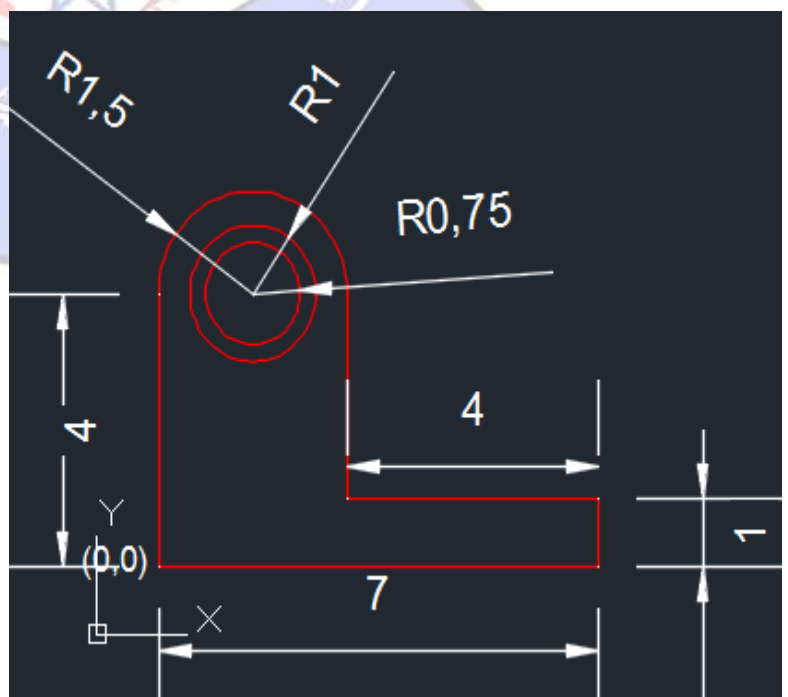
Specify next point:4,5

Command: L

Specify first point:1,1

Specify next point:1,5

Command: a





Specify first point of arc:4,5

Specify center point of arc:2.5,5

Specify end point:1,5

Command: C

Specify center point for circle:2.5,5

Specify radius of circle:0.750

Command: C

Specify center point for circle:2.5,5

Specify radius of circle:1

2.5. Ellipse

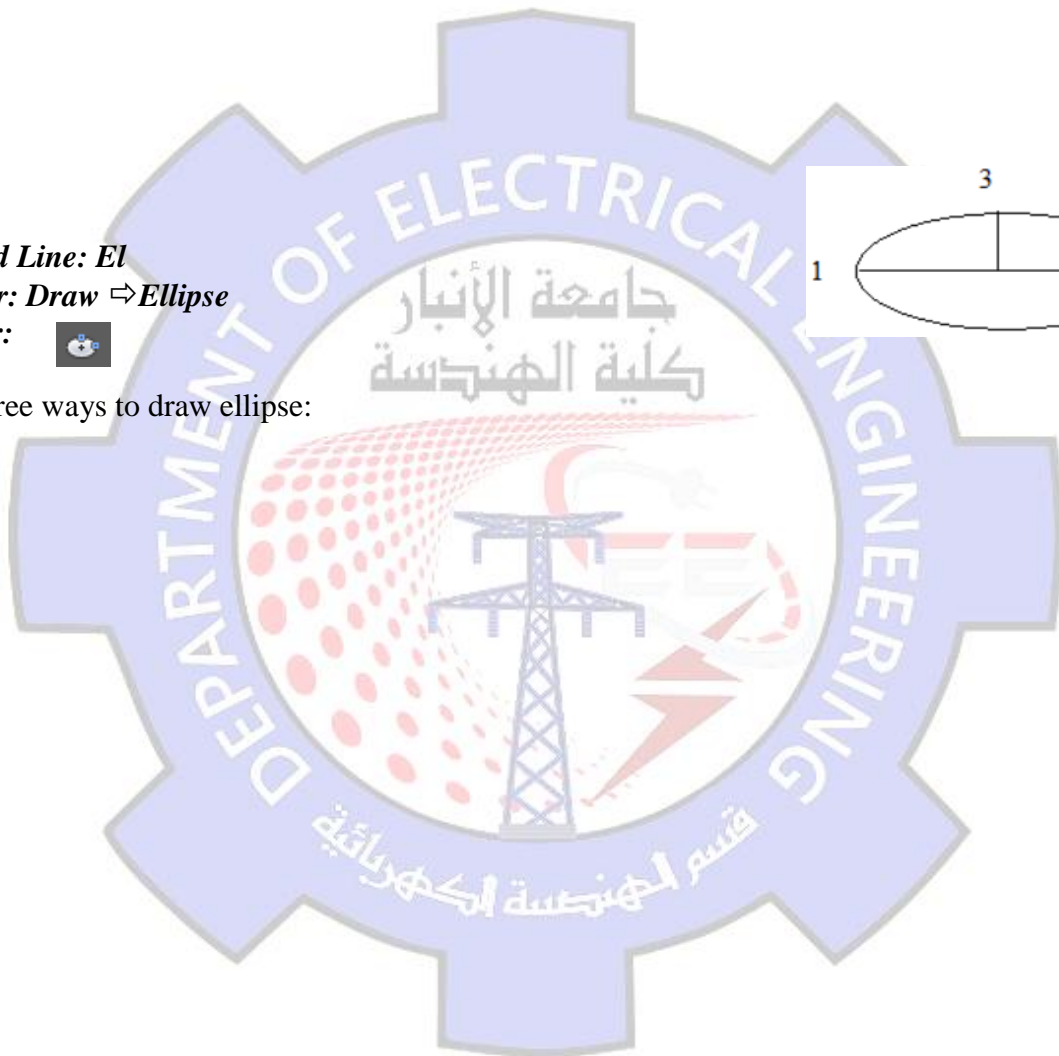
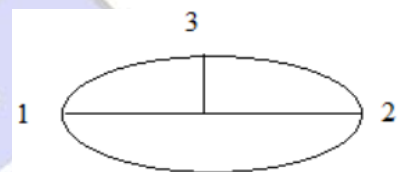
1. **Command Line: El**

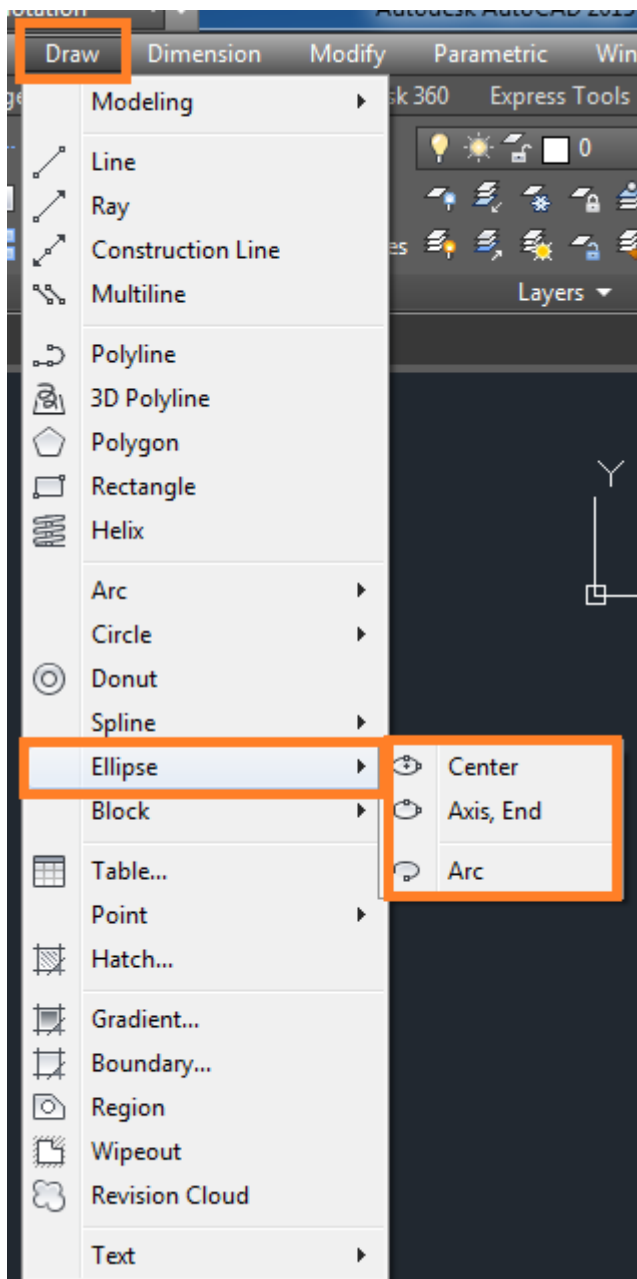
2. **Menu Bar: Draw** ⇒ **Ellipse**

3. **Draw Bar:**



There are three ways to draw ellipse:





1. Center (Specify the center and the end of each axis)

Specify axis end point of ellipse or [Arc / Center]:C

Specify center of ellipse:

Specify end point of axis:

Specify distance to other axis or [rotation]:

2.Axis end (Specify one axis and the end of second)

Command : ellipse

Specify axis end point of ellipse or [Arc / Center]:



Specify other end point of axis:

Specify distance to other axis or [Rotation]:

3. Arc (create an ellipse arc)

Specify axis end point of ellipse or [Arc / Center]:A

Specify axis endpoint of elliptical arc or [center]:

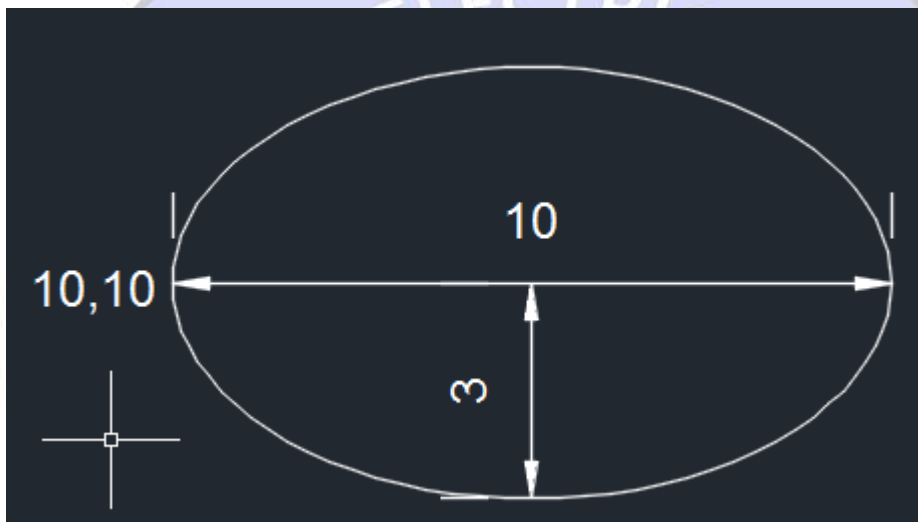
Specify other endpoint of axis:

Specify distance to other axis or [rotation]:

Specify start angle or [parameter]:

Specify end angle or [parameter / included angle]:

Example 1. Draw the following figure



Command: ellipse

Axis end

Specify axis end point of ellipse or [Arc / Center]:10,10

Specify other end point of axis:20,10

Specify distance to other axis or [Rotation]:3

Center

Specify axis end point of ellipse or [Arc / Center]:C

Specify center of ellips:15,10

Specify end point of axis:20,10

Specify distance to other axis or [rotation]:15,13

Arc

Specify axis end point of ellipse or [Arc / Center]:A

Specify axis endpoint of elliptical arc or [center]:20,10

Specify other endpoint of axis:10,10



Specify distance to other axis or [rotation]: r

Specify start angle or [parameter]:0

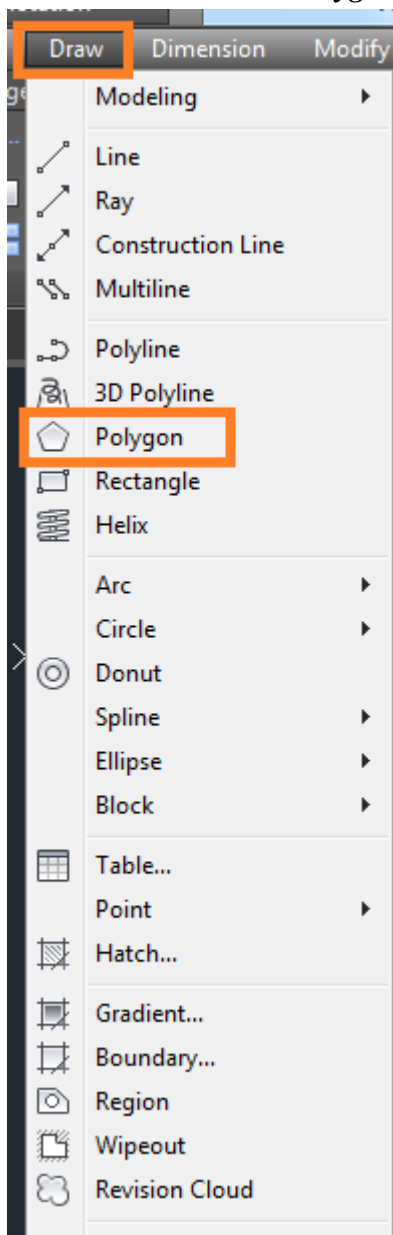
Specify end angle or [parameter / included angle]:45

2.6. Polygon

The Polygon command enables to create regular polygon with 3 to 1024 sides. A polygon is one with sides of equal length.

1. **Command Line:Pol**

2. **Menu Bar:Draw⇒ Polygon**



3. **Draw Bar:**





When enter polygon command, the following message appear:

Enter number of sides< >:

Specify center point or [edge]:

If our chose the Edge option, the program require the following:

Specify first endpoint of edge:

Specify second endpoint of edge:

If our option specifies Center point, the program requires the radius of circle:

Enter an option [Inscribed in circle / Circumference about circle] < >:

Options:

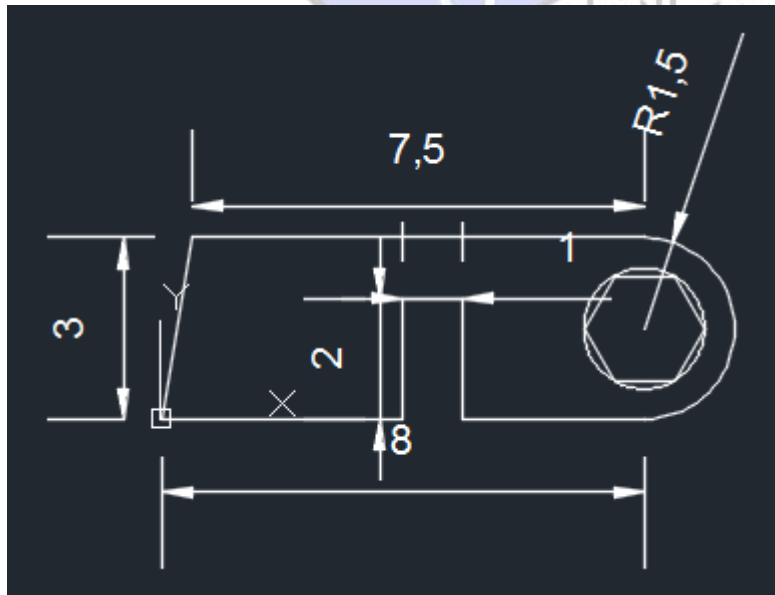
I. Inscribed option allows control of one half of the distance across the corner.

C. Circumscribed option allows control of one-half of the distance across the flats.

When determine one of the options the program display the message:

Specify radius of circle:

Example 1. Draw the following figure. Note the figure start from original point (0,0).



Absolute Coordinate System

Command: line

Specify first point:0,0

Specify next point:4,0

Specify next point:4,2

Specify next point:5,2

Specify next point:5,0

Specify next point:8,0

Esc from keyboard

Command: line

Specify first point:0,0

Specify next point:0.5,3



Specify next point:8,3

Command: a

Specify first point:8,0

Specify center point:8,1.5

Specify endpoint:8,3

Command: Pol

Enter number of sides:6

Specify center point or [edge]:8,1.5

Specify radius of circle:1

2.7. Donut

The donut command allows to create thick – walled or solid circles, known in AutoCAD as donuts. Access the command by:

1. *of donut Command Line: do*
2. *Menu Bar: Draw⇒Donut*

Specify inside diameter:

Specify outside diameter of donut:

Example: Draw the following figures:



2.8. Hatch

A hatch or crosshatch is a repetitive pattern of lines or symbols that shows a related area of a drawing.

1. *Command Line: Hatch or H*

2. *Menu Bar :* Draw ⇒ Hatch

3. *Draw Bar :*

Specify pattern name or [?/Solid/User defined]<current pattern>:

Specify a scale for the pattern <current default>:



2.9. Construction Line

1. Command Line: *XL*

2. Menu Bar: *Draw* ⇨ *Construction Line*

3. Draw Bar :



An Xline is a line with infinite length, therefore having no ends, but does have a root , which is the theoretical midpoint.

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

The Xline command provides several options that allow creating specific types of construction lines.

Options:

Hor: the horizontal option forces all the lines to be exactly horizontal.

Ver: the vertical option forces all the lines to be exactly vertical.

Ang: the angle option use to create construction lines at angles other than 0 or 90 degrees.

Offset: offsetting a line means creating the line at a specific distance from another line.

Bisect: this option draws the xline at an angle between two selected points.

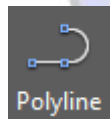
2.10. Polylines

A polyline is a connected sequence of line or arc segments that is created as a single object.

1. Command Line: *pline* or *PL*

2. Menu Bar: *Draw* ⇨ *PolyLine*

3. Draw Bar :



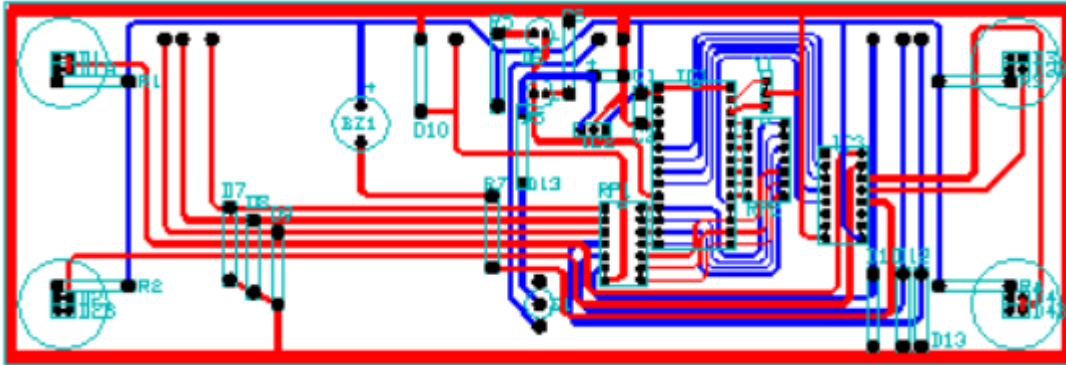
Quick: **From point:** To start the pline, **click** on the screen with the Left mouse button, type in a **coordinate**, or use an **Osnap**.

Quick: **Arc / Close / Halfwidth / Lenth / Undo / Width / <Endpoint of Line>:** select an **option** or select the next end of the **Pline** segment.

Use the **PLINE** command to create open or closed polylines for:

- Geometry that requires fixed-width segments
- Continuous paths for which you need to know the total length
- Contour lines for topographic maps and isobaric data
- Wiring diagrams and traces on printed circuit boards
- Process and piping diagrams

Here is an example of a printed circuit board in which the traces were created with wide polylines. The landing pads were created with the **DONUT** command.



Polylines can have different starting and ending widths for each segment as shown here:



2.11. Text

there are two different types of text objects:

- **Dtext** command: inserts one or more lines of text as single line objects. The Dtext command enables you to display Auto CAD text dynamically in the drawing as you type it. it also allows you to justify or align, the text in several ways, including left, right, and center justification.
- **Mtext** Command: treats one or more paragraphs of text as single objects. The Mtext command creates a multiple line object called Mtext. AutoCAD uses a text editor to create Mtext objects.

1. Command Line :Mtext

2. Menu Bar : Draw ⇒ text

3. Draw Bar:



Command: text

Specify start point of text or [justify / style]:

Specify height <default height>:

Specify rotation angle of text < 0 >:

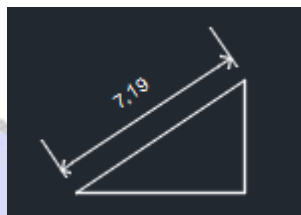
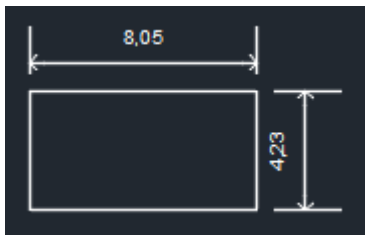
Enter text:



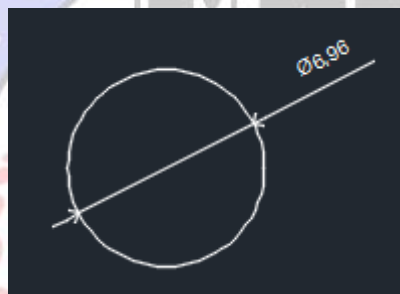
2.12. Dimensions

AutoCAD only requires that you Pick an object or specify the extension line origins, and pick the location of the dimension line. AutoCAD then measures the feature and draws the extension line, arrow heads and dimension text.

Linear dimensions:  



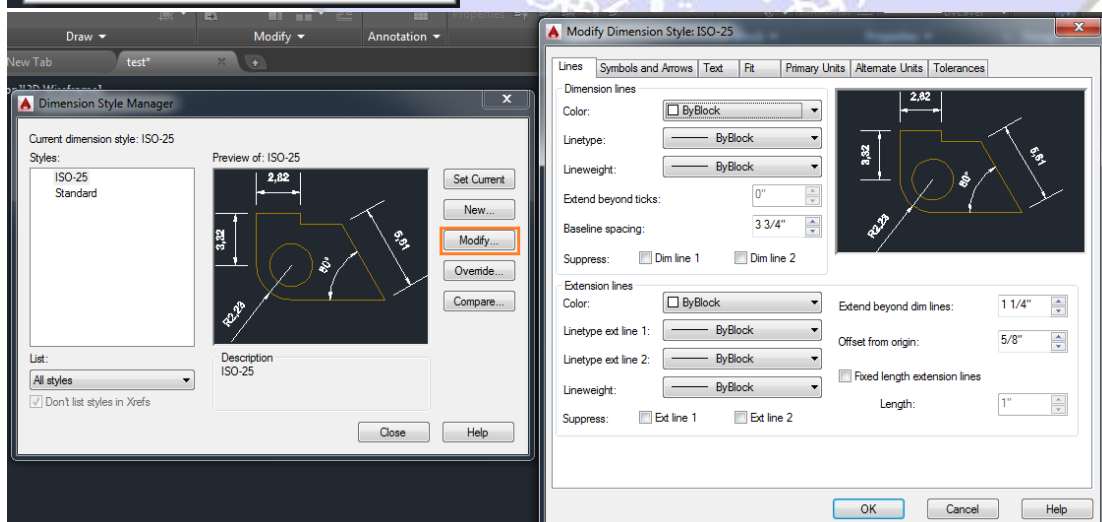
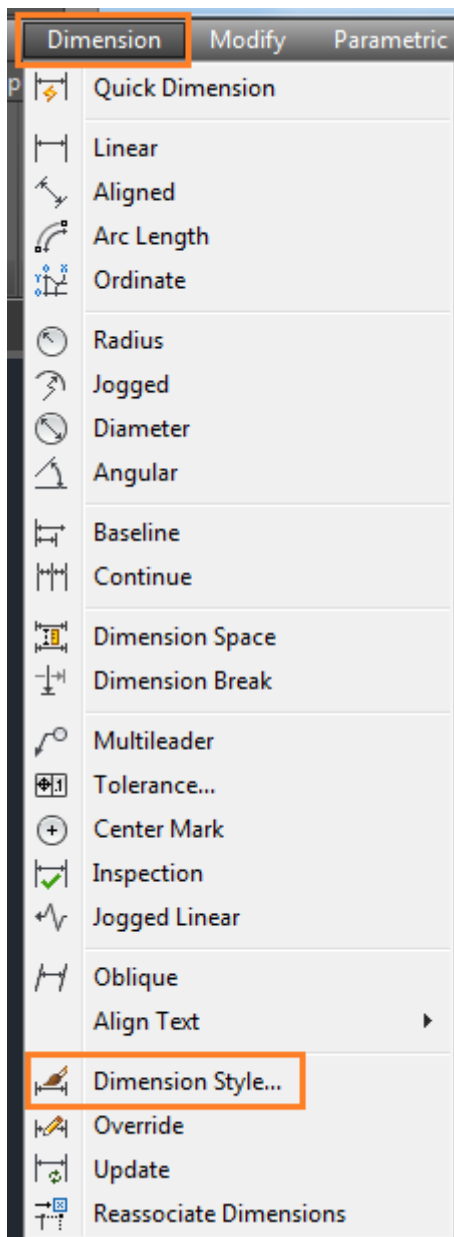
Dimensioning Round Features:  



Dimensioning Angles 



To make dimension setting in preparation for dimensioning a drawing click on the style, option in the dimension pull-down menu at the top of the screen, then click on the modify button to open diminution style dialog box.

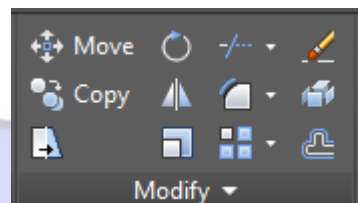




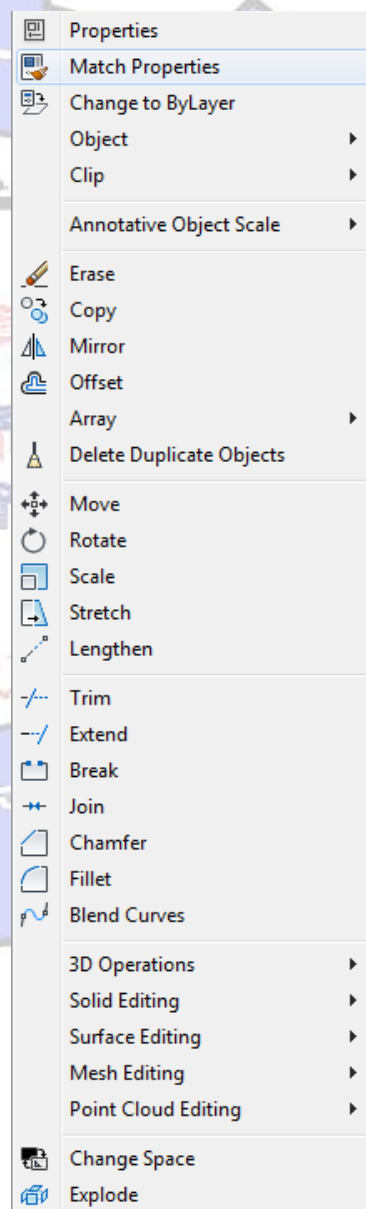
3. Modify Commands

After you have created some objects such as lines or circle they may needed to be rotated, moved, copied, scaled larger or smaller, etc., Such operations and others are performed by what is known as modify commands. There three ways to access to this which are:

1- Modify toolbar, also use modify pull down.



2- Menu bar




3- type the command in command line.



3.1. Erase

This command use to delete objects. Access the command from

1. **Menu bar:** *Modify* ⇒ *Erase*
2. **Command line:** *Erase* or *E*
3. **Draw bar:** 

Command: *Erase*

Select Objects:

The indicator will change to small square. Selected objects by:

1. Pick the objects. This default option is used for selecting one object at a time.
 2. Window. Only objects completely within the window are selected. not objects that cross through or outside the window.(ragging to the right creates a window instead).
- notice: *Oops command lets to undo the last erase you are doing.*

3.2. Move

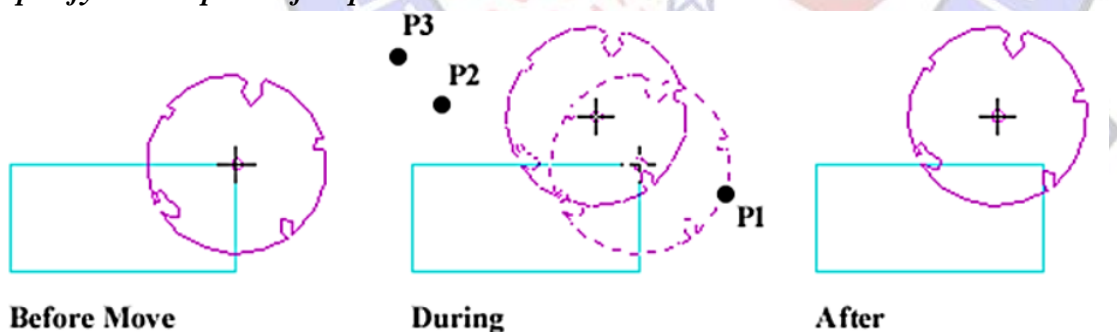
- 1- **Menu bar:** *Modify* ⇒ *Move*
- 2- **Command line:** *Move* or *M*
- 3- **Draw bar:** 

Command: *Move*


Select Objects:

Specify base point or displacement:

Specify second point of displacement:



3.3. Copy

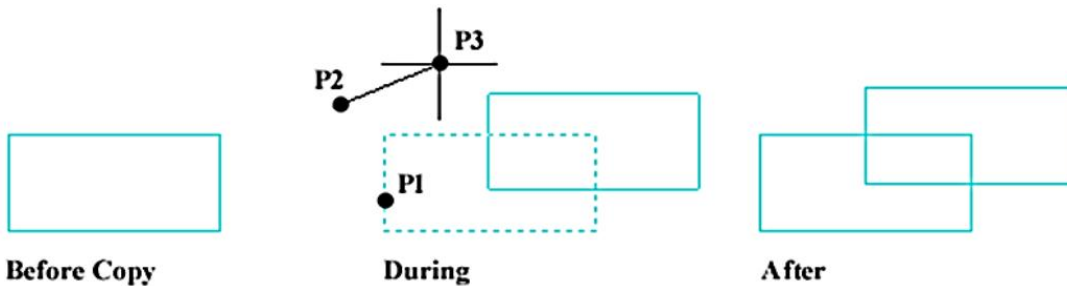
- 1- **Menu bar:** *Modify* ⇒ *copy*
- 2- **Command line:** *copy* or *co* or *cp*
- 3- **Draw Bar:** 

Command: *Copy*

Select Objects:

Specify base point or displacement or [Multiple]:

Specify second point of displacement:



3.4. Offset

This command use to draw object parallel to selected object by determine distance.

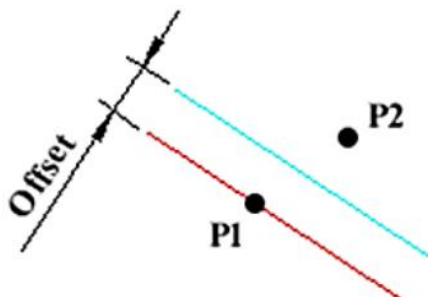
- 1- *Menu bar: Modify* ⇒ *offset*
- 2- *Command line: O*
- 3- *Draw Bar:* 

Command: Offset

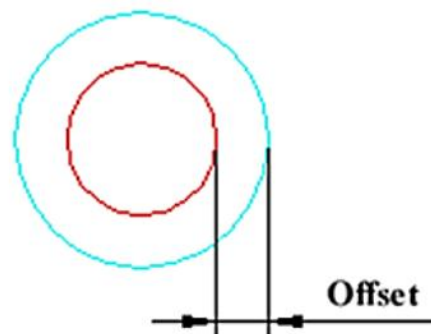
Specify offset distance or [through]:

Select object to offset or <exit>:

Specify point on side to offset:




Offsetting a Line



Offsetting a Circle

3.5. Mirror

- 1- *Menu bar: Modify* ⇒ *Mirror*
- 2- *Command line: Mi*
- 3- *Draw Bar:* 

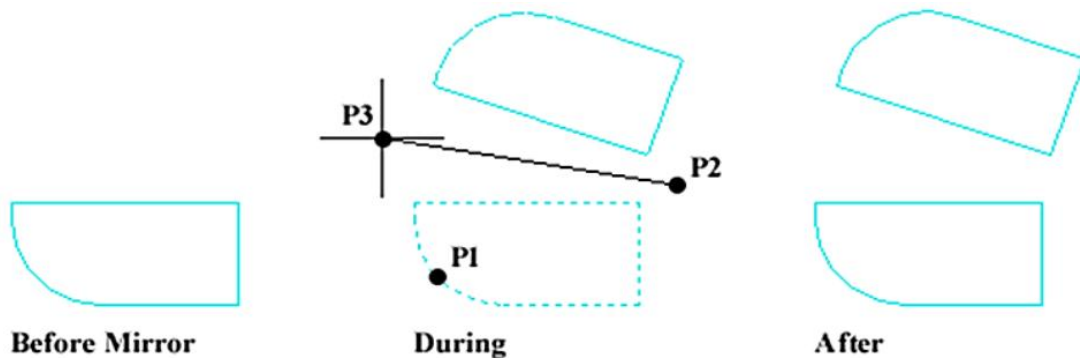
Command: Mirror

Select objects:

Specify first point of mirror line:

Specify second point of mirror line:

Delete source object?[yes/no]:




Example. Draw the left half of the figure then mirror the objects to create the right half.



3.6. Trim

This command cut required intersections

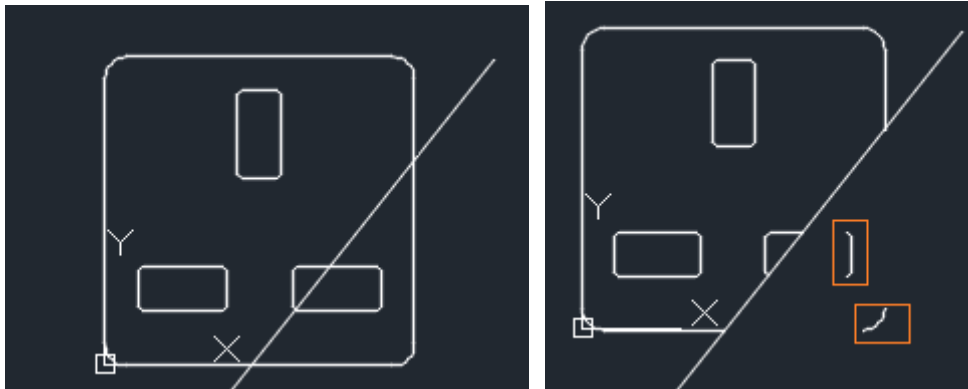
- 1- **Menu bar:** *Modify* ⇒ *trim*
- 2- **Command line:** *trim* or *tr*
- 3- **Draw bar** 

Command: *Trim*

Current settings: *Projection= UCS=None*

Select cutting edges:

Select object to trim or [Project / Edge / Undo]:



We note that there are parts that were not removed by the command **Trim** because this command only removes the intersections.

3.7. Extend

Extend command use to length object to meet another object.

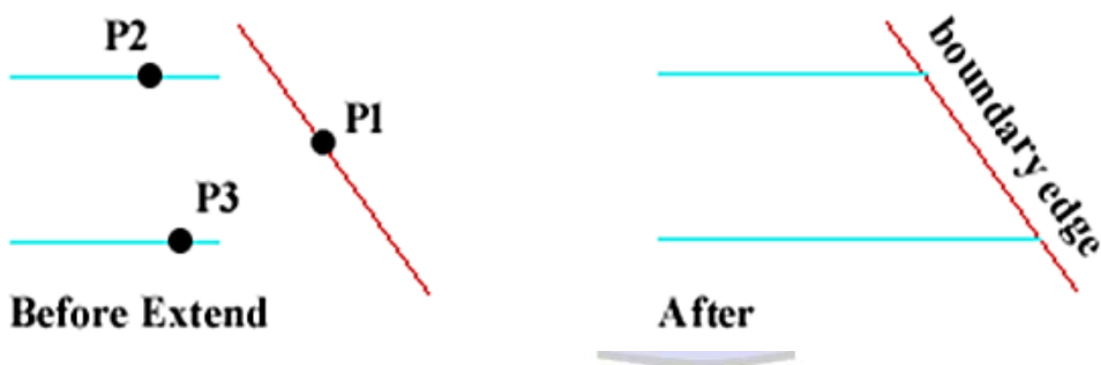
- 1- **Menu bar:** Modify ⇒ **Extend**
- 2- **Command line:** *Extend or Ex*
- 3- **Draw bar:** 

Current settings:


Projection= UCS=, Edge=None

Select boundary edges

Select object to extend or [Project / Edge / Undo]:



3.8. Break

- 1- **Menu bar:** Modify ⇒ **Break**
- 2- **Command line:** *Break or Br*
- 3- **Draw bar:** 



Select Object:

Specify second break point or [First Point]: f

Specify First break point:

Specify second break point:



3.9. Rotate

This command use to rotate sleeted objects around point by determining angle.

1- **Menu bar:** *Modify* ⇒ *Rotate*

2- **Command bar:** *Rotate* or *RO*

3- **Draw bar:**

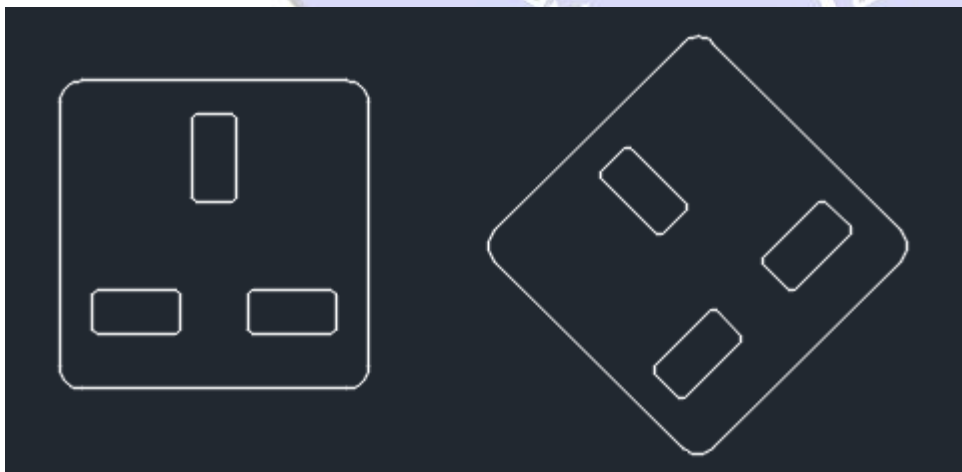
When use the command, the program require the following:

Select objects:

Specify base point:

Specify rotation angle or [Reference]:

Example: draw the figure below and rotate it by 45 degree





Command: **rotate**

Select object: **drag to window the drawing**

Specify base point:


Specify rotation angle or [Reference]: **45**

3.10. Scale

To enlarge or minimize the drawing according to specific scale

1- **Menu bar:Modify** ⇒ **Scale**

2- **Command line:Scale or Sc**

3- **Draw bar** 

Select objects:

Specify base point:

Specify scale factor or [Reference]:

Example: minimize the red shapes to the amount of half

Command: scale

Select object: drag the red shapes

Specify base point:

Specify scale factor or [Reference]: 0.5



3.11. Lengthen

By this command can check length of selected line, and control line length.

1- **Menu bar:Modify** ⇒ **Lengthen**

2- **Command line:Len**





3- Draw bar

The following message appear:

Select an object or [Delta / Percent / Total / Dynamic]:

Options:

Delta: in this option determine increment or decrement in length, then click the object, the object is lengthen in determined value.

Percent: in this option determine length percentage to original length .for example: 110 mean increment 10%, 90 mean decrement 10 %.

Total: determine all length value to object.

Dynamic: in this option-using mouse to increment or decrement object to new location.

3.12. Stretch

This command stretch selected object from selected side.

1- **Menu bar:**Modify ⇒ **Stretch**

2- **Command line:** S

3- **Draw bar**

The following message appear:

Select objects to stretch by crossing window or crossing polygon:.

Specify base point or displacement

Specify second point of displacement:

3.13. Fillet

Use this command to change the direct meeting of two objects to a meeting with a circular arc.

1- **Menu bar:**Modify ⇒ **Fillet**

2- **Command line:** Fillet

3- **Draw bar**

Options:

Select first object: this option require select first object.

Radius: this option used to determine radius of fillet.



Poly line: this option used when the base object is pl and wants circular all angels by this option.

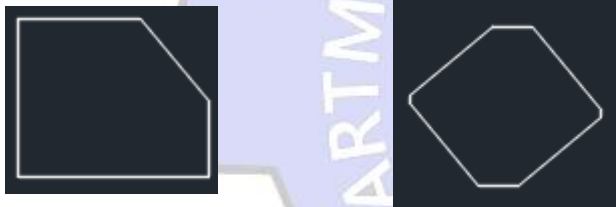
Trim: to distinguish between trim original lines or no trim.



3.14. Chamfer


Use this command to change the direct meeting of two objects to a meeting with a line.

- 1- **Menu bar:** Modify ⇒ **chamfer**
- 2- **Command line:** Chamfer
- 3- **Draw bar**  :



3.15. Array

This command used for create multiple copies of object in the pattern

- 1- **Menu bar:** Modify ⇒ **array**
- 2- **Command line:** array
- 3- **Draw bar :** 

3.16. Explode

This command used to destroy the connection between one object. After the command explode, done we can apply (copy, move, erase, mirror,).

- 1- **Menu bar:** Modify ⇒ **Explode**
- 2- **Command line:** Exp





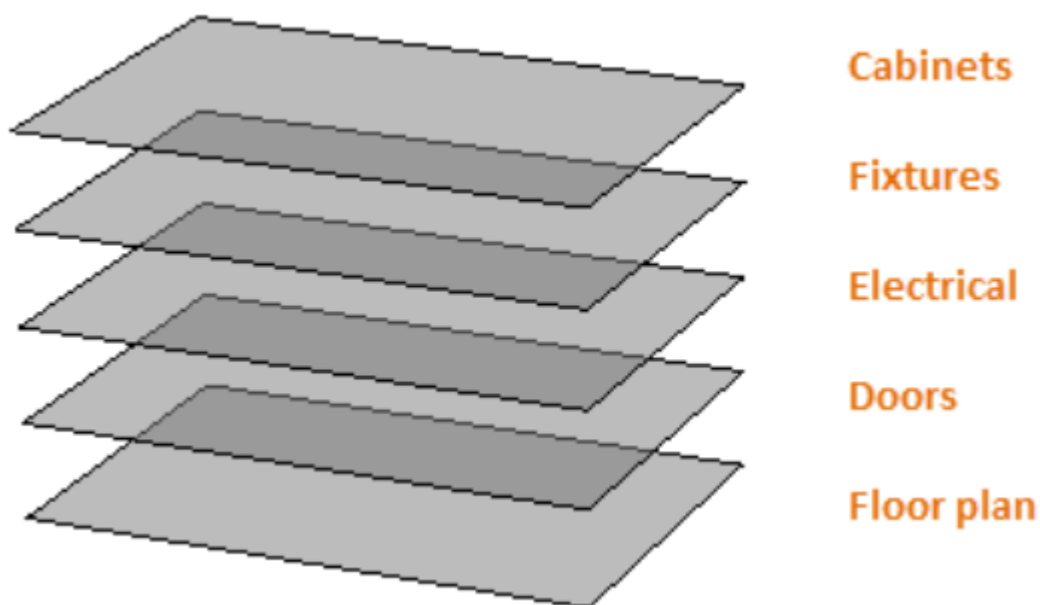
3- Draw bar :



4. The Layers

4.1. Introduction.

Organize the objects in the drawing with layers associated to a specific function or purpose for more control. Layers can be considered as transparent plastic sheets.



Layers advantages:

- Associate objects by their function or location
- Display or hide all objects related to a single operation
- Enforce line type, color, and other property standards for each layer

1- **Tool Bar: format** ⇨ **Layer**

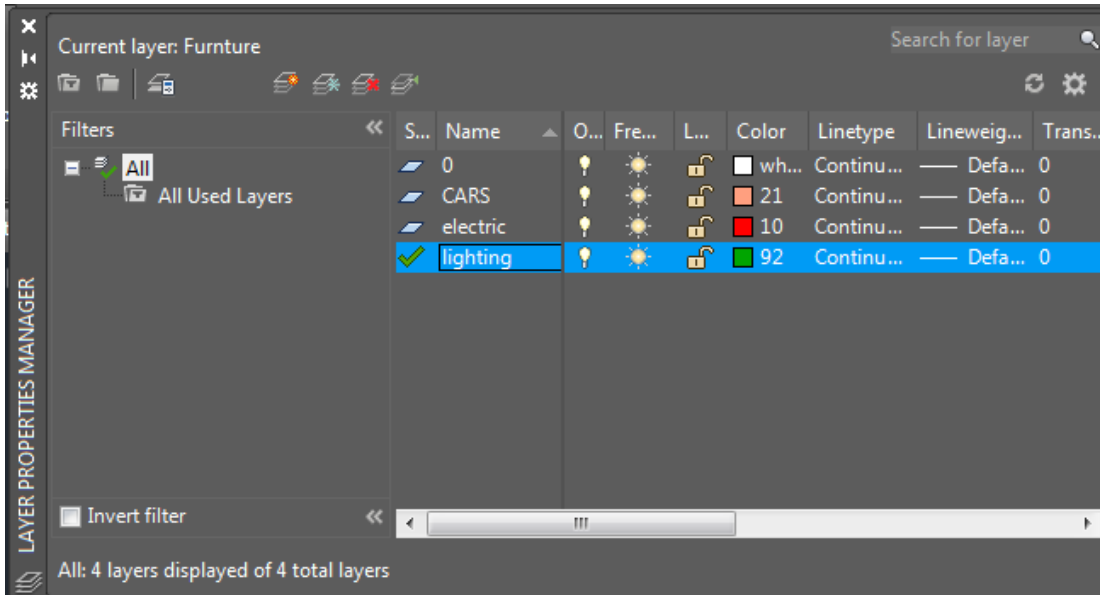
2- **Command line Window: Layer or LA**

3- **layer properties manager:**



4.2. Layer Properties manager Layer List


The right area of the Layer Properties manager is the layer list that displays the layer list in the drawing. In this section, you can control the current visibility settings and the properties specific to each layer. This menu displays all layers in the drawing. New layers can be created by selecting the New Layer button or New Layer option from the right-click menu. There may be only one layer in the drawing layer "0".



4.3. Steps of Create Layers

- 1- Click  on the left of properties bar.



- 2- Click , immediately a new row will adding to existing layers. The "0" layer is a default layer. Appear in new row name of new layer we can change it to convenient name to easily access to layer we need, such as wall, electric, lighting.....
- 3- To determine layer color click on the square under address **(color)** in new raw color window appear select from it appropriate color.
- 4- To determine line style click on **(continuous)** in new raw under address **(line type)**, window appear, select from it appropriate line type .click on **(load)** to appear more line types.
- 5- To determine line weight used for drawing objects click **(default)** under address **(line weight)** then select weight we need from the window that appear.

4.4. Layers Managements

Select drawing layer to be activate.



1. Click on manager Layer List.
2. Select layer from it.
3. Click on drawing board .the selected layer be the active layer .note that object color and line type and line weight all must be assistant to(**By layer**).


Move drawing object from layer to another.

1. Select drawing object.
2. Click on manager Layer List to open it.
3. Double click on select layer want to move the object to it or click on ESC twice to undo select object, we saw change the layer color and line to color of selected layer.


Hide layer.

1. Click on manager Layer List to open it.
2. Click on yellow light  for layer wants to hid it by put it to off then click on enter we obtain the drawing without the selected layer.

Lock objects belong to determine layer.

1. Click on manager Layer List to open it.
2. Click on lock button  for layer wants to lock it then click enter, the layer is locked. (The locked layer it is the layer that can't change or modified to it unless we open it by click the lock button again.

Change color of drawing layer.

1. Click on  properties bar, the dialog box appear for control layers and lines types.
2. Click on color, color control box appear.
3. Select appropriate color.
4. Double click on appropriate color saw all the objects belong to the layer their color are change to new color.

4.5. Object Properties

a- Assigning Colors

You can control the color of each object individually as follow.

1- *Command Line Window: Color or Col*

2- *Tool Bar: format*

⇒ *color*

3- *layer properties manager:*



b- Assigning Line Types

Various types of lines are used in the formulation to show different elements of the drawing.

1- *Command line Window: line type or l type or lf*

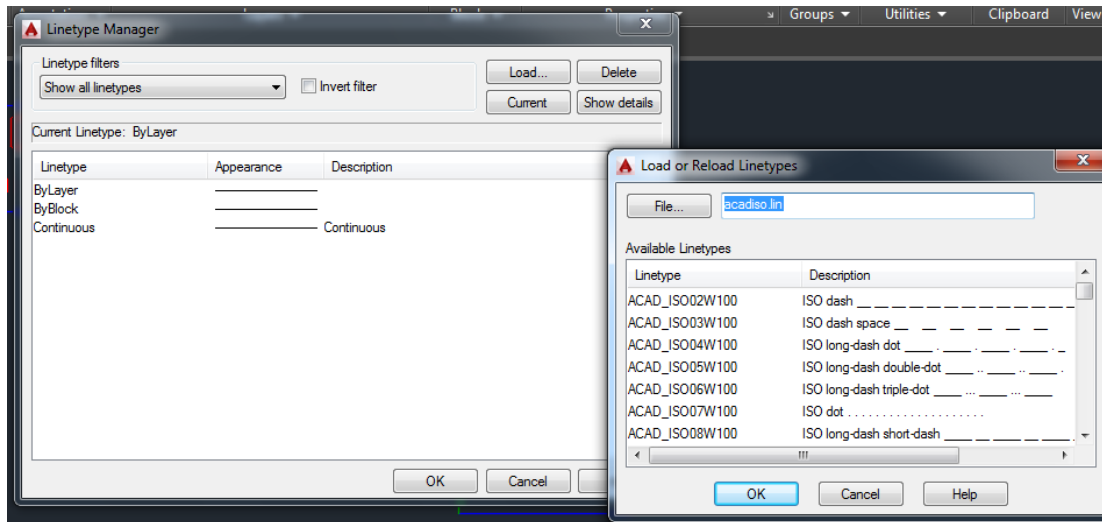


2- *tool bar: format* ⇨ *line*

3- *layer properties manager:*



type



c- Setting Line Weights

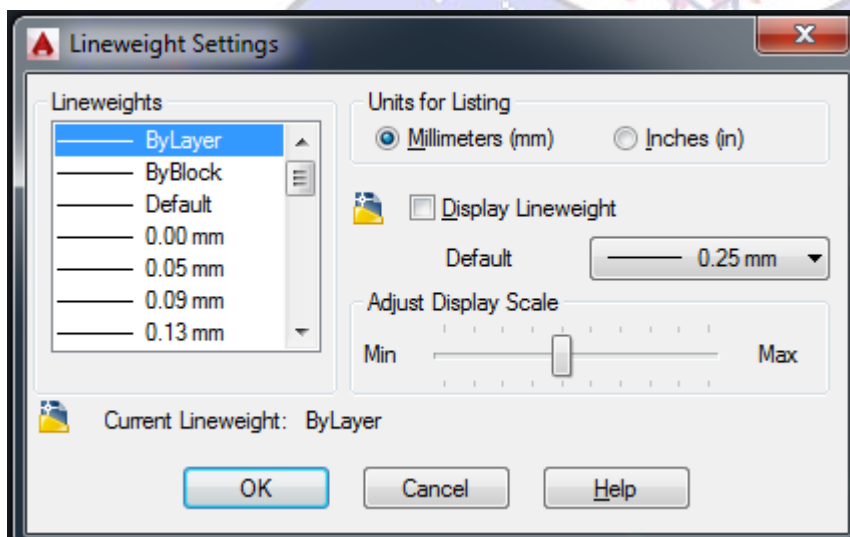
Line weights are important in technical drawing. For example, object lines should be thicker so they stand out more than dimensions. Normally, object lines are thick, hidden lines and text are of medium thickness, and center lines, dimensions, and hatch lines are thin.

1- *Command line Window: Lweight or lw*

2- *Tool Bar: format* ⇨ *line*

weight

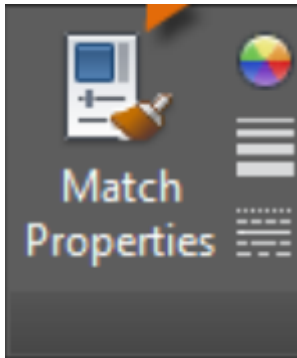
3- *layer properties manager*





d- Match the Properties of Objects

For a fast way to copy the properties of a selected object to other objects, use the Match Properties tool, or enter **MATCHPROP** or **MA** in the Command window.



The Mouse

Most people use a mouse as their pointing device, but other devices have equivalent controls.

