



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

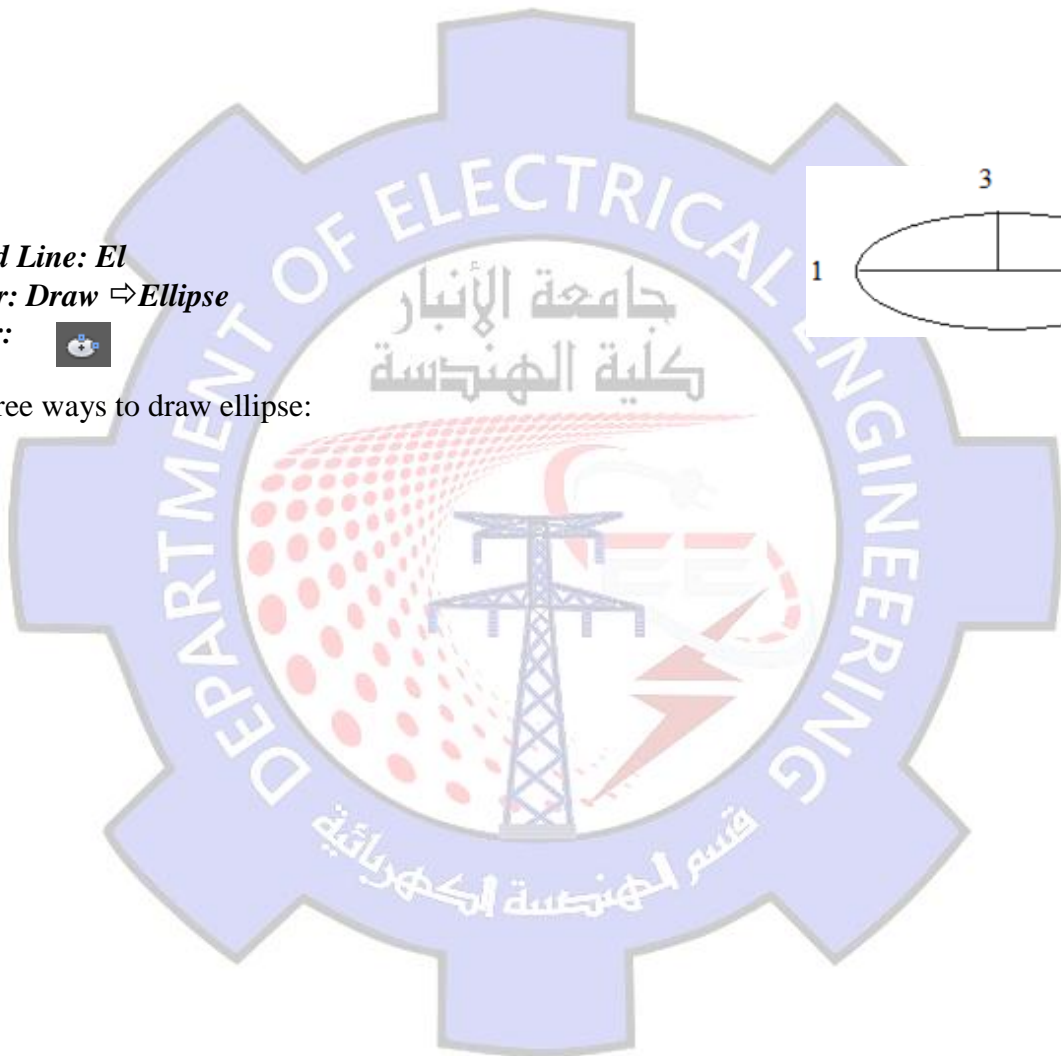
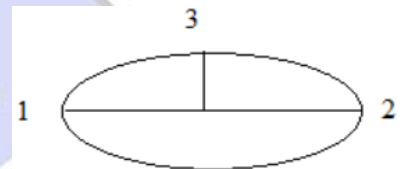
1. **Command Line: El**

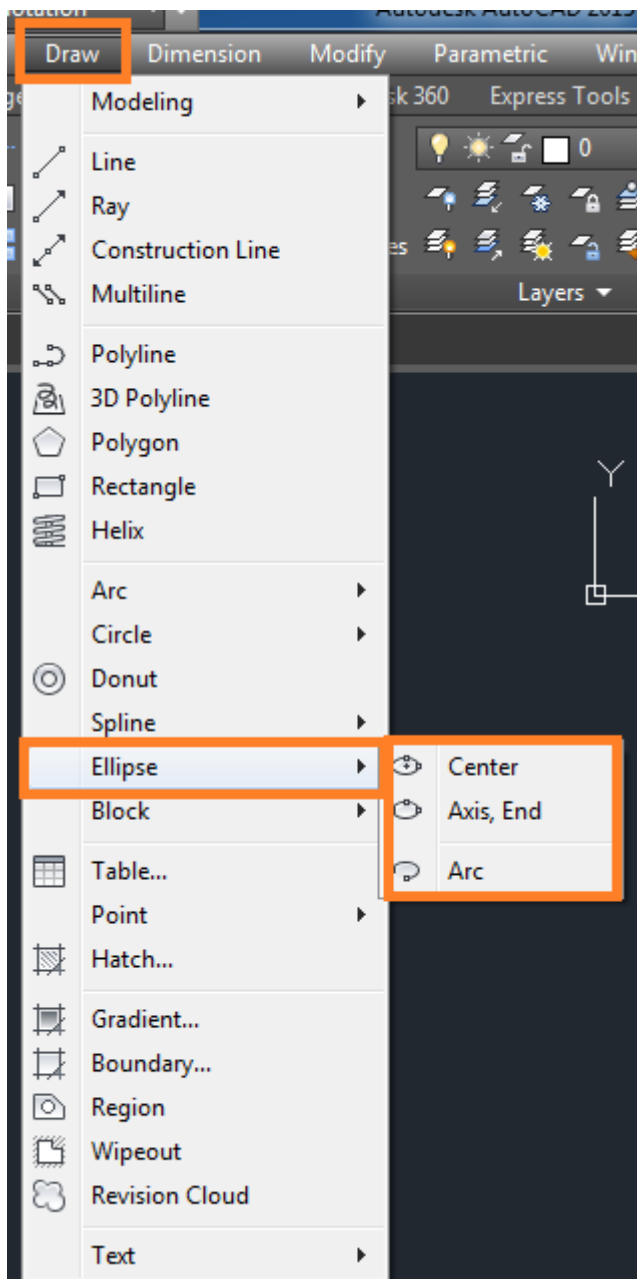
2. **Menu Bar: Draw** ⇒ **Ellipse**

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]:

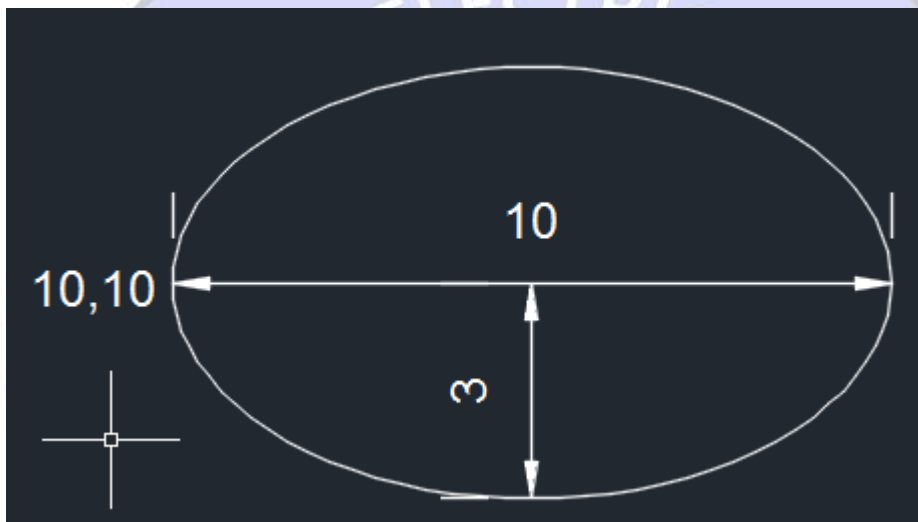
Specify other endpoint of axis:

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

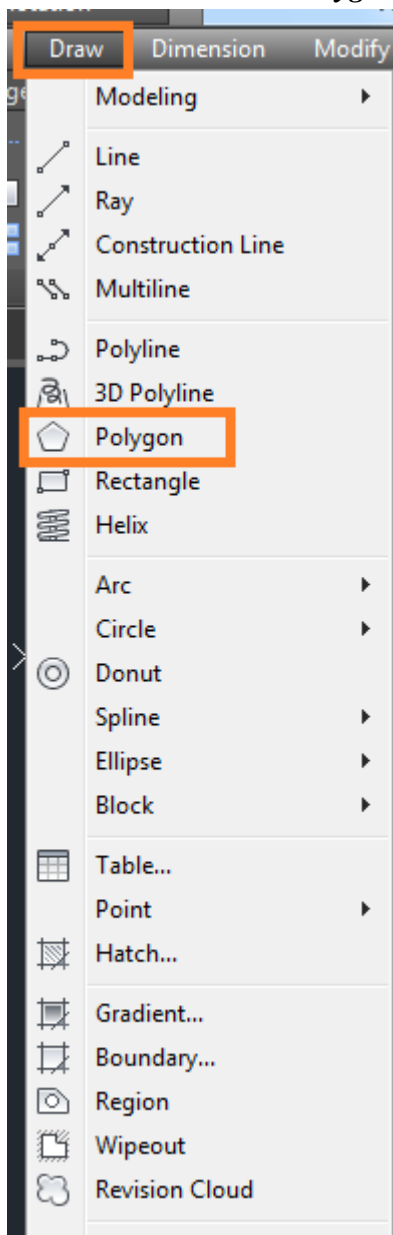
Specify end angle or [parameter / included angle]:45

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:

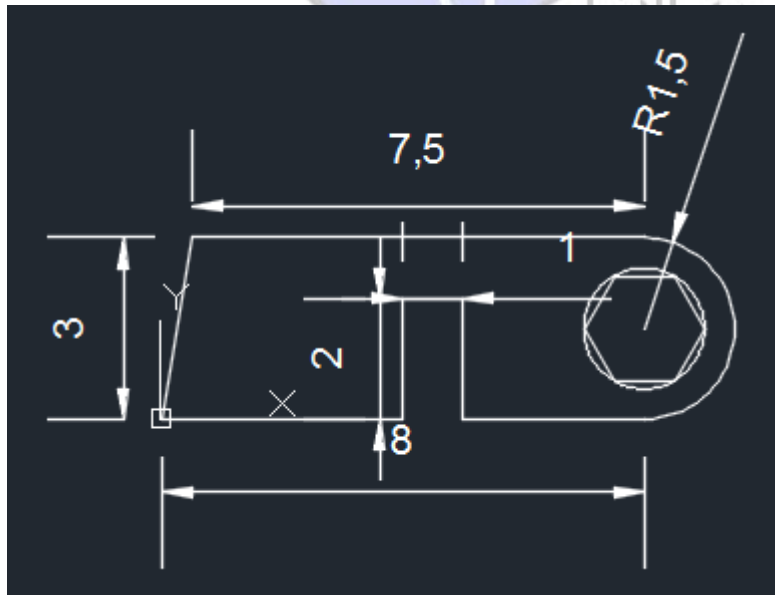
I. Inscribed option allows control of one half of the distance across the corner.

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.

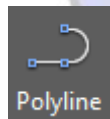
2.10. Polylines

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 :



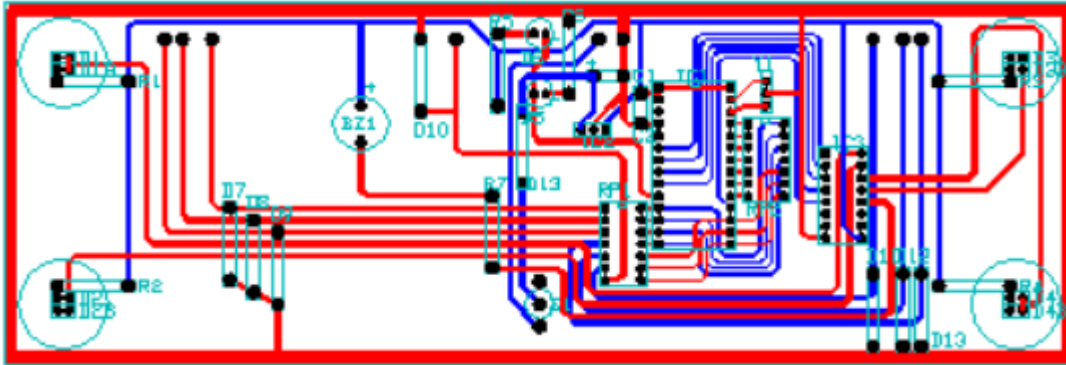
Quick: **From point:** To start the pline, **click** on the screen with the Left mouse button, type in a **coordinate**, or use an **Osnap**.

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 >:

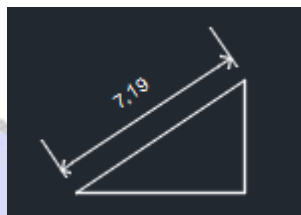
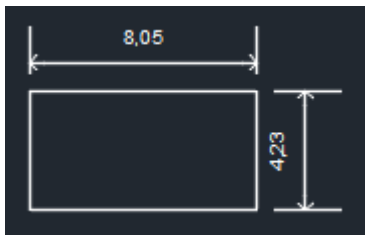
Enter text:



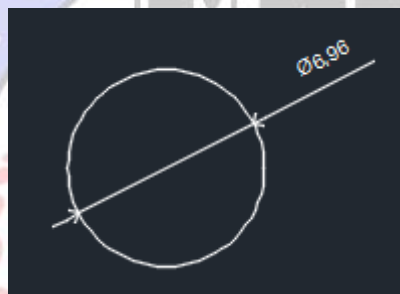
2.12. Dimensions

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.

