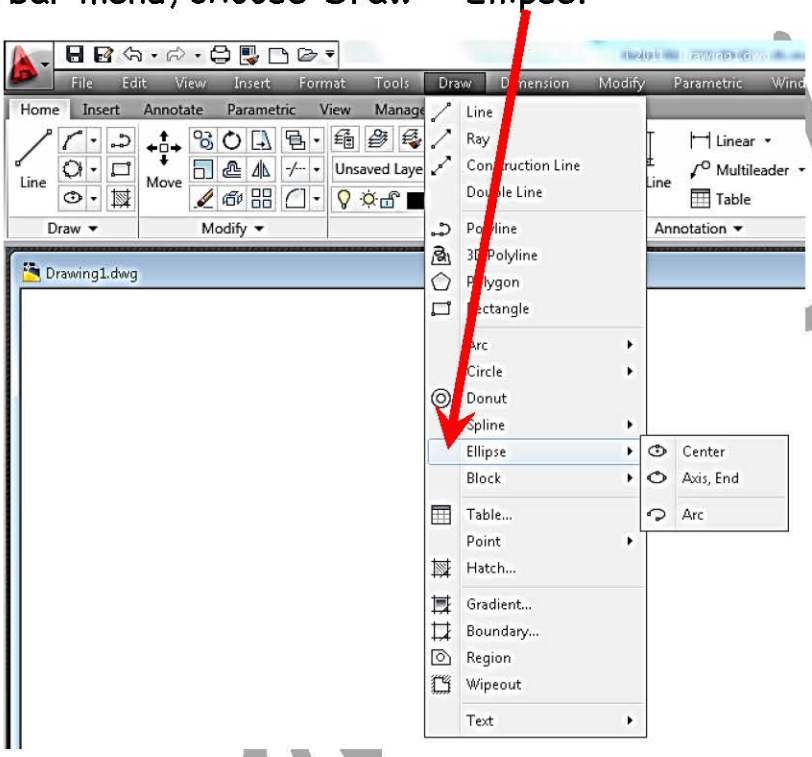
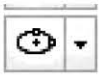
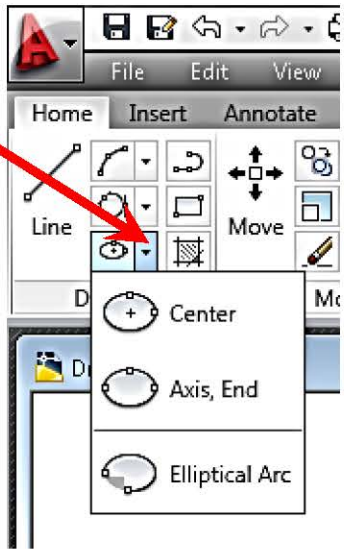
## 2-6 ELLIPSE

This command is used to draw ellipse and the abbreviation of this command is **EL**. There are three ways to activate this command:

1. From tools bar menu, choose Draw >> Ellipse.




2. Click the black triangle near ELLIPSE icon from the ribbon  to open the hidden ellipse drawing options menu then click the required icon.



3. Type ELLIPSE in the command bar or EL, as shown below:

Command: Ellipse or el, and press ENTER

Then the following message appears:

```
Command: ellipse
Specify axis endpoint of ellipse or [Arc/Center]:
```

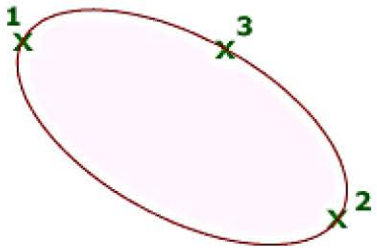
Which requests the coordinates of the first point of the diameter, let be 10,20, and then press Enter and the following message appears:

```
Specify axis endpoint of ellipse or [Arc/Center]: 10,20
Specify other endpoint of axis:
```

Which requests the coordinates of the other point of the diameter, let be 100,20, and then press Enter. After selecting it the following message appear:

```
Specify other endpoint of axis: 100,20
Specify distance to other axis or [Rotation]:
```

Which requires the length of the other radius, let be 25 and then press enter and the ellipse will be drawn.




When you select the ROTATION command by typing abbreviation letter R, the following message appear:

```
Specify distance to other axis or [Rotation]: r
Specify rotation around major axis:
```

Which asks to determine the angle of rotation of the plane containing the ellipse. Specify the angle, let be 30°, and press Enter then the ellipse rotated as shown in the figure below:

The sub option Center, its abbreviation is the letter **C**, is used to specify the position of the ellipse center.

Type letter C and press Enter then this message appears:

```
Specify axis endpoint of ellipse or [Arc/Center]: _c
Specify center of ellipse:
```

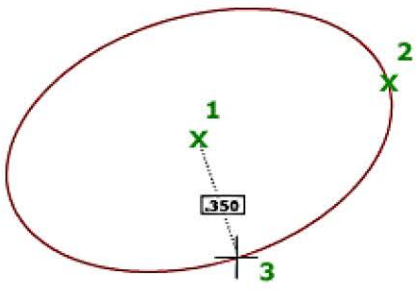
Which requests the coordinates for the position of the ellipse center point, let be 50,100, and then press Enter and the following message appears:

```
Specify center of ellipse: 50,100
Specify endpoint of axis:
```

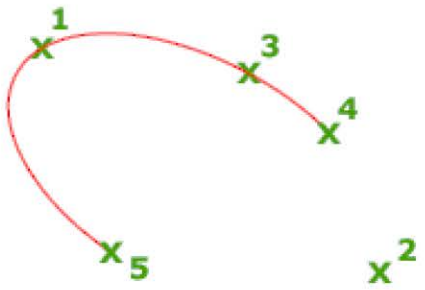
Which requests the coordinates of the end point of the radius, let be 100,100, and then press Enter and the following message appear:

```
Specify endpoint of axis: 100,100
Specify distance to other axis or [Rotation]:
```

Which requires the length of the other radius, let be 20 and then press enter and the ellipse will be drawn.



Arc command and its abbreviation is **A** character. This command is used to draw an arc of an ellipse depending on the selection of one of the two diameters and length of the other radius.




After you choose Ellipse command and then choose Arc command and press Enter, the following message appears:

```
Specify axis endpoint of ellipse or [Arc/Center]: a
Specify axis endpoint of elliptical arc or [Center]:
```

Which asks to specify the coordinates of the first point of the diameter to be 20,40, and then press Enter, the following message appears:

```
Specify axis endpoint of elliptical arc or [Center]: 20,40
Specify other endpoint of axis:
```

Which requests the coordinates of the other point of the diameter to be 200,40, and press Enter, the following message appears:

```
Specify other endpoint of axis: 200,40
Specify distance to other axis or [Rotation]:
```

Which requires the length of the other radius to be 50 and then press Enter, the following message appears:

```
Specify distance to other axis or [Rotation]: 50
Specify start angle or [Parameter]:
```

Which asks for the angle of the starting point of the arc to be 30, then press Enter and the following message appears:

```
Specify start angle or [Parameter]: 30
Specify end angle or [Parameter/Included angle]:
```

Which asks for the end point angle of the arc to be 135, and then press Enter. Then the ellipse Arc had drawn, Figure (2-5).

```
Specify end angle or [Parameter/Included angle]: 135
Command:
```

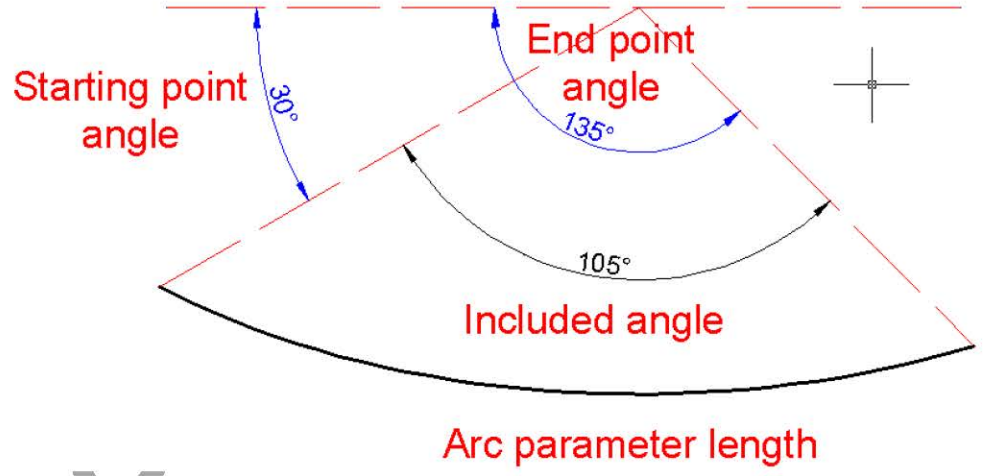


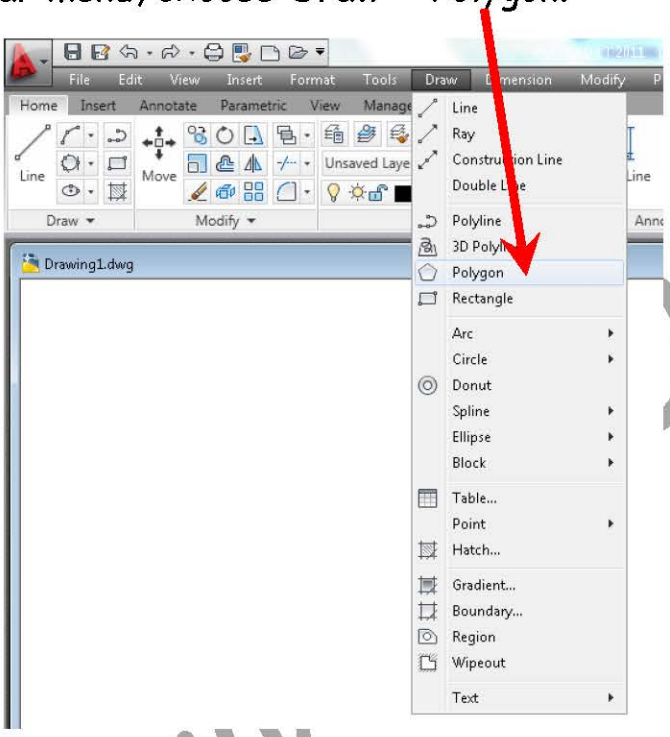

Figure (2-5): Ellipse Arc drawing.

- Parameter option using for specify the arc length.
- Included angle option used to specify arc include angle.

## 2-7 POLYGON

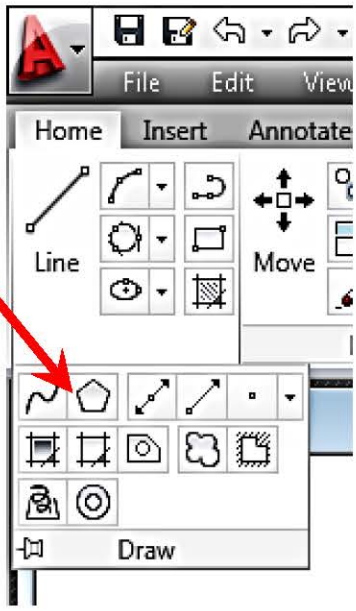
This command is used to draw a regular polygon and its abbreviation is **POL**. There are three ways to activate this command:

1. From tools bar menu, choose Draw >> Polygon.




2. Click the black triangle near draw panel from the ribbon 

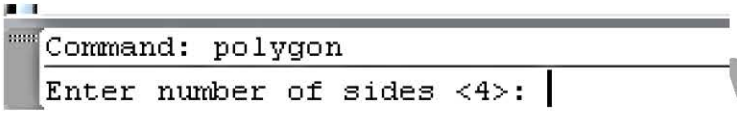
to open the hidden draw icons menu then click the Polygon icon.



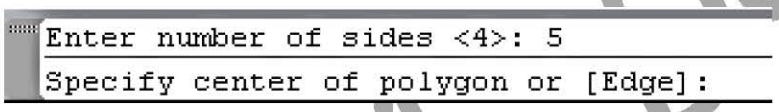
3. Type POLYGON in the command bar or POL, as shown below:

Command: Polygon or pol, and press ENTER

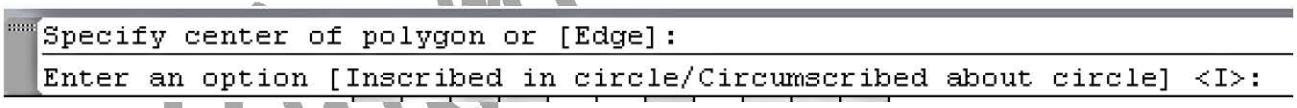
Then the following message appears:




Which asks to specify number of polygon sides (default number is four). Print 5 and press Enter, then the following message appears:




Which asks to specify the position of polygon center or Edge of its side and press Enter. If you specify the required position of the polygon center, (where we define any point by one of the ways to enter coordinates or by the mouse), then the following message appears:




Which requires the circle drawing method, the first option (default option) means the circle that passes through the outer headers of the polygon (circle surround polygon). The second option is the circle that touches the inner sides of the polygon (polygon surround circle), Figure (2-6).

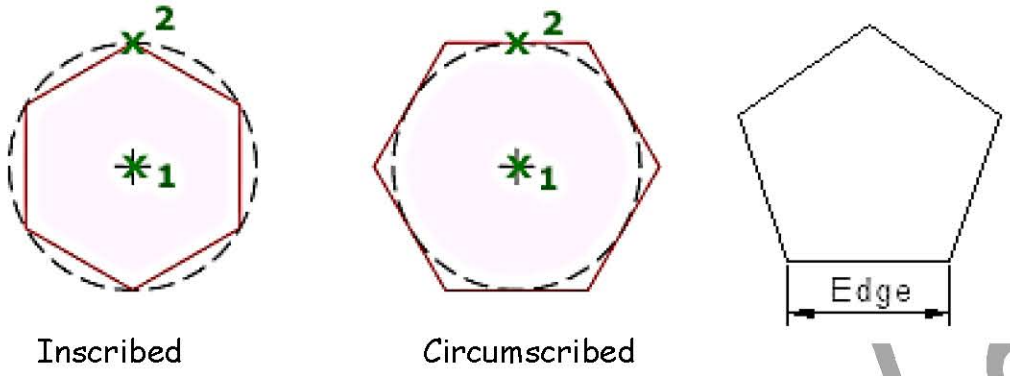



Figure (2-6): Polygon drawing options.

Execute the first command by typing a shortcut, which is the letter **I** (for the second command type letter **C**) and then press Enter. The following message appears:

```
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: i
Specify radius of circle:
```

Which require the determination of radius of the circle.

When you select the sub-option Edge (its abbreviation the character **E**) and pressing Enter the following message appears:

```
Command:  POLYGON Enter number of sides <5>:
Specify center of polygon or [Edge]: e Specify first endpoint of edge:
```

Which requires the insertion of the coordinates of the starting point position for the polygon side, by one of the coordinates input methods listed earlier, the following message appears:

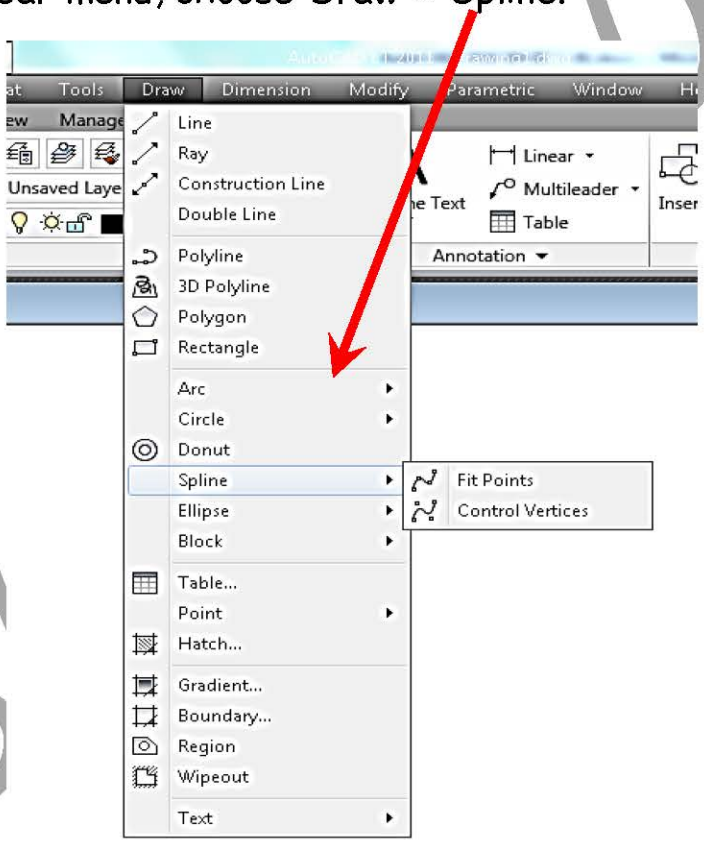
```
Specify center of polygon or [Edge]: e Specify first endpoint of edge: 20,25
Specify second endpoint of edge:
```

Which requires the insertion of the coordinates of the endpoint of the polygon side, or the length of the polygon side. After you entered the line end coordinate and pressing Enter the polygon drawn.

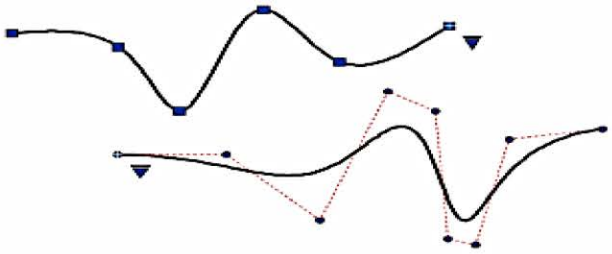
## 2-8 SPLINE

The SPLINE command creates a particular type of spline known as a Non uniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points. This command is used to draw a curved line in which a set of points is specified where the line passes, and then we define the tangent slope at the starting point and then at the end point. There are three ways to activate this command:

1. From tools bar menu, choose Draw >> Spline.

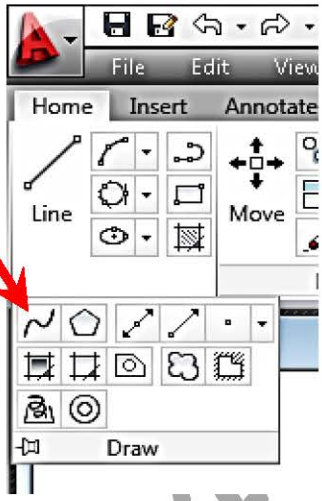



There are two methods for creating spline in AutoCAD: with fit points or with control vertices. Each method has different options.



2. Click the black triangle near draw panel from the ribbon 

to open the hidden draw icons menu then click the Spline icon.




3. Type SPLINE in the command bar or SPL, as shown below:

Command: Spline or spl, and press ENTER

Then the following message appears:

```
Current settings: Method=Fit    Knots=Chord
Specify first point or [Method/Knots/Object]:
```

Which asks the coordinates of the first point to be 45,50, then press Enter. The following message appears:

```
Specify first point or [Method/Knots/Object]: 45,50
Enter next point or [start Tangency/toLerance]:
```

Which requests the coordinates of the second point to be 60,60, then press Enter. The following message appears:

```
Enter next point or [start Tangency/toLerance]: 60,60
Enter next point or [end Tangency/toLerance/Undo/Close]:
```

Which require the coordinates of the next point to be 80,40, then 100,70, then the last point 140,35 (pick points until you are done drawing splines ), then press Enter or close to complete the spline.