

# LESSON 5

## LEARNING OBJECTIVES

*After completing this lesson, you will be able to:*

1. Draw an Inscribed or Circumscribed Polygon.
2. Create an Ellipse using two different methods.
3. Define an Elliptical Arc.
4. Create Donuts.
5. Define a Location with a Point.
6. Select various Point Styles.
7. Use 3 new Object Snap modes.

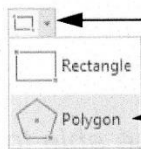
# Polygon

A polygon is an object with multiple edges (flat sides) of equal length. You may specify from 3 to 1024 sides. A polygon appears to be multiple lines, but in fact it is one object. You can specify the **center** and a **radius** or the **edge length**. The **radius** size can be specified **Inscribed** or **Circumscribed**.

## Center, Radius method

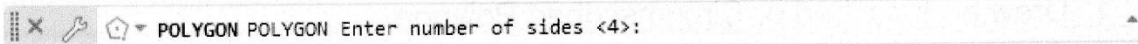
1. Select the **Polygon** command using one of the following:

**Ribbon = Home Tab / Draw Panel /**  
or  
**Keyboard = POL <Enter>**



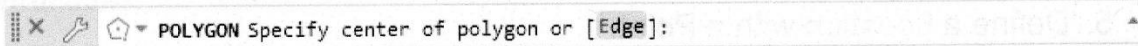
Select the down arrow and then select Polygon

The following prompt will appear on the Command Line:



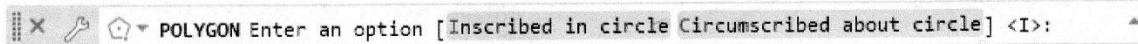
2. Type the number of sides and then press **<Enter>**.

The following prompt will appear on the Command Line:



3. Specify the center location (**P1**).

The following prompt will appear on the Command Line:



4. Type **I** or **C** and then press **<Enter>**.

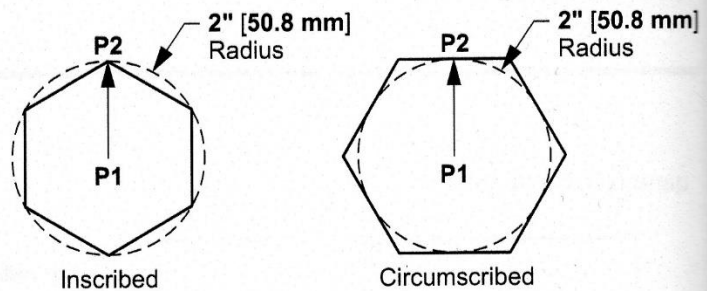
The following prompt will appear on the Command Line:



5. Type the radius or locate with the cursor (**P2**).

### Note:

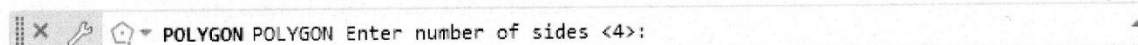
The dashed circle is shown only as a reference to help you visualize the difference between Inscribed and Circumscribed. Notice that the radius is the same (2") [50.8 mm], but the polygons are different sizes. Selecting **Inscribed** or **Circumscribed** is important.



## Edge method

1. Select the **Polygon** command using one of the options shown above:

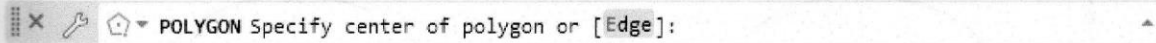
The following prompt will appear on the Command Line:



2. Type the number of sides and then press **<Enter>**.

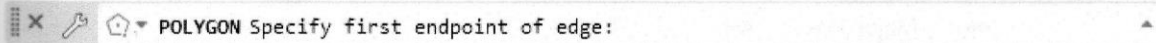
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The following prompt will appear on the Command Line:



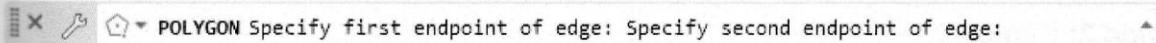
- Type **E** and then press **<Enter>**.

The following prompt will appear on the Command Line:

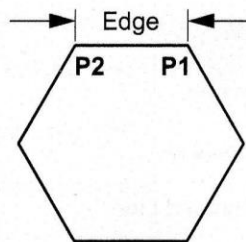


- Place the first endpoint of edge (**P1**).

The following prompt will appear on the Command Line:



- Place the second endpoint of edge (**P2**).



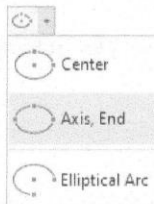
## Ellipse

There are three methods to draw an ellipse. You may (1) specify 3 points of the axes, (2) define the center point and the axis points or (3) define an elliptical arc. The following 3 examples illustrate each of the methods.

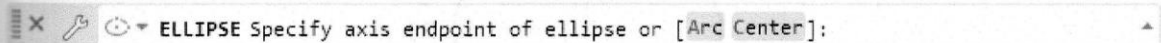
### Method 1: Axis, End

- Select the **Ellipse** command using one of the following:

Ribbon = Home Tab / Draw Panel /  
or  
Keyboard = **EL <Enter>**

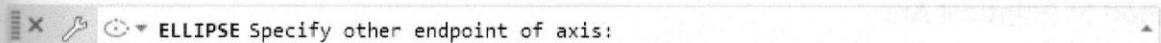


The following prompt will appear on the Command Line:



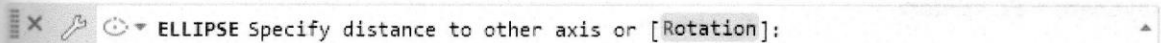
- Place the first endpoint of either the **major** or **minor** axis (**P1**).

The following prompt will appear on the Command Line:



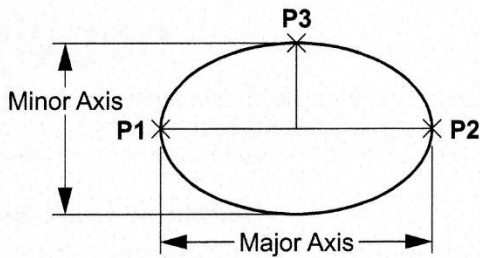
- Place the other endpoint of the first axis (**P2**).

The following prompt will appear on the Command Line:

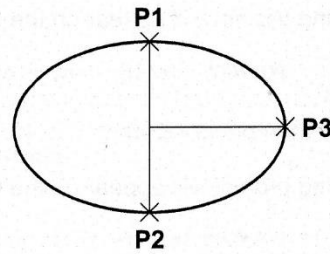


- Place the point perpendicular to the first axis (**P3**).

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Specifying the Major Axis first (P1/P2), and then the Minor Axis (P3).

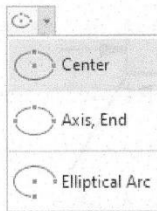


Specifying the Minor Axis first (P1/P2), and then the Major Axis (P3).

**Method 2: Center**

1. Select the **Ellipse** command using one of the following:

Ribbon = Home Tab / Draw Panel /  
or  
Keyboard = **EL** <Enter>



The following prompt will appear on the Command Line:



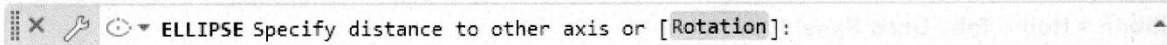
2. Place the center of the ellipse (P1).

The following prompt will appear on the Command Line:

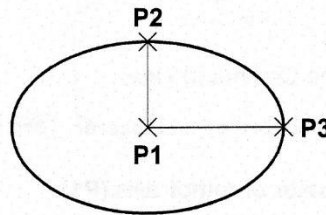


3. Place the first axis endpoint (either axis) (P2).

The following prompt will appear on the Command Line:



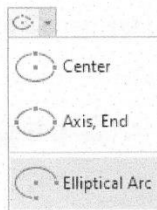
4. Place the point perpendicular to the first axis (P3).



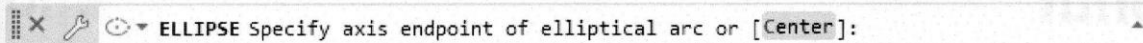
**Method 3: Elliptical Arc**

1. Select the **Ellipse** command using one of the following:

Ribbon = Home Tab / Draw Panel /  
or  
Keyboard = **EL** <Enter>

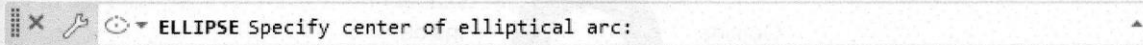


The following prompt will appear on the Command Line:



2. Type in **C** and then press **<Enter>**.

The following prompt will appear on the Command Line:



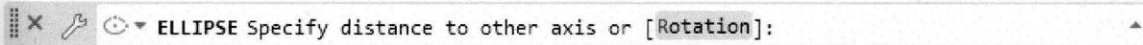
3. Place the center of the elliptical arc (**P1**).

The following prompt will appear on the Command Line:



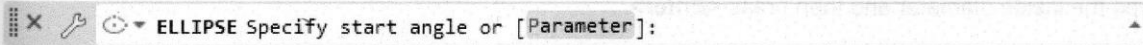
4. Place endpoint of the first axis (**P2**).

The following prompt will appear on the Command Line:

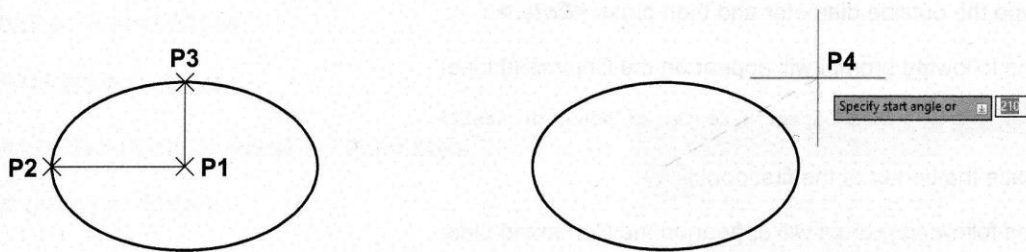


5. Place the endpoint perpendicular to the first axis (**P3**).

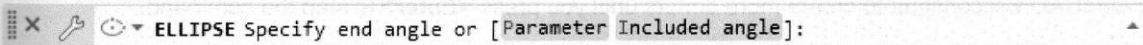
The following prompt will appear on the Command Line:



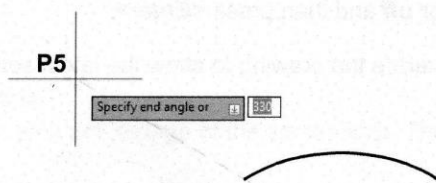
6. Place the **start angle** (**P4**).



The following prompt will appear on the Command Line:

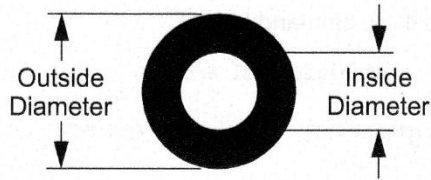


7. Place the **end angle** (**P5**).



# Donut

A Donut is a circle with **width**. You will define the **Inside** and **Outside** diameters.

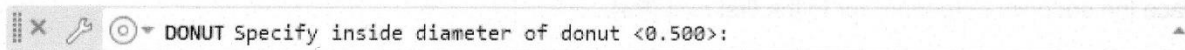


1. Select the **Donut** command using one of the following:

**Ribbon = Home Tab / Draw Panel /**  
or  
**Keyboard = DO <Enter>**

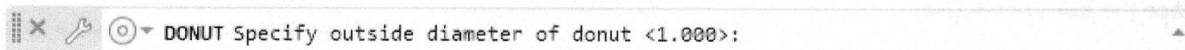


The following prompt will appear on the Command Line:



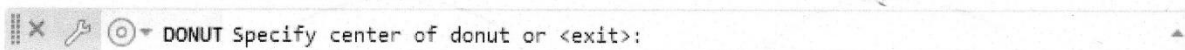
2. Type the inside diameter and then press **<Enter>**.

The following prompt will appear on the Command Line:



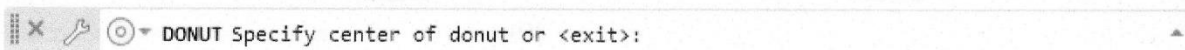
3. Type the outside diameter and then press **<Enter>**.

The following prompt will appear on the Command Line:



4. Place the center of the first donut.

The following prompt will appear on the Command Line:

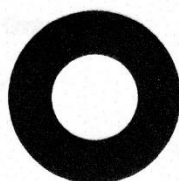


5. Place the center of the second donut or press **<Enter>** to stop.

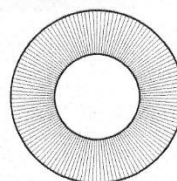
**Note:** AutoCAD will continue to create more donuts until you press **<Enter>** to stop the command.

## Controlling the “Fill Mode”

1. Command: Type **Fill** and then press **<Enter>**.
2. **FILL** Enter mode [ON OFF] <on>: Type **on** or **off** and then press **<Enter>**.
3. Type **Regen** and then press **<Enter>** to regenerate the drawing to show the latest setting of the **FILL** mode.



Fill = On



Fill = Off

# Point

Points are used to locate a point of reference or location. A point may be represented by one of many Point Styles shown below in the Point Style box.

The only Object Snap option that can be used with Point is Node.  
(Refer to the next page for more information on Node Object Snap.)

## How to Use the Point Command

1. Select the **Point** command using one of the following:

Ribbon = Home Tab / Draw Panel ▾ /

or

Keyboard = **PO** <Enter>



2. The following prompt will appear on the Command Line:



3. Place the point location.

The following prompt will appear on the Command Line:



4. Place another point or press the <Esc> key to stop placing points.

## How to Select a "Point Style"

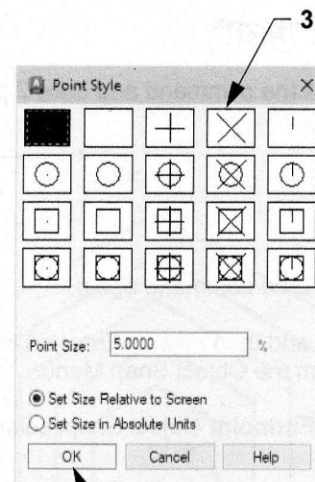
1. Open the **Point Style** dialog box:

Ribbon = Home Tab / Utilities Panel ▾ / Point Style...

or

Keyboard = **ddptype** <Enter>

2. The Point Style dialog box will appear.
3. Select a Point Style tile.
4. Select the **OK** button.



### Point Size:

#### Set Size Relative to Screen

Sets the point display size as a percentage of the screen size. The point display does not change when you zoom in or out.

#### Set Size in Absolute Units

Sets the point display size as the actual units you specify under Point Size. Points are displayed larger or smaller when you zoom in or out.

# More Object Snaps



**NODe**

This option snaps to the object "**Point**" described on the previous page. Select **Node** Object Snap and place the cursor on the **Point**. The cursor will snap to the **Point**.

**Note:** This is the only Object Snap that you can use with the object **Point**.



**NEArest**

Snaps to the nearest location on an object. For example, if you want to attach a line somewhere on a circle between quadrants:

Select the **Line** command and then select **Nearest** Object Snap. Place the cursor anywhere on the circumference of the circle and press the left mouse button. The line will now be accurately attached to the circle at the location you selected.



**M2P**

## Mid Between 2 Points

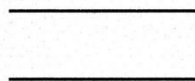
Locates a midpoint between two points.



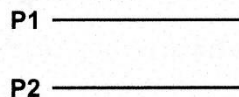
You may select the **M2P** option from the Object Snap Menu using **<Shift> + right click**, or you may type **M2P <Enter>** when prompted for an endpoint.

## How to use "M2P"

1. Select the **Line** command and draw 2 parallel lines.



2. Select the **Line** command again.
3. Type **M2P** and then press **<Enter>**. Or hold down the **<Shift>** key and right click, and select **Mid Between 2 Points** from the Object Snap Menu.
4. Using the **Endpoint** Object Snap, select each of the 2 endpoints (**P1**) and (**P2**).



5. The new line's first endpoint should start **exactly** midpoint between those 2 endpoints.

