

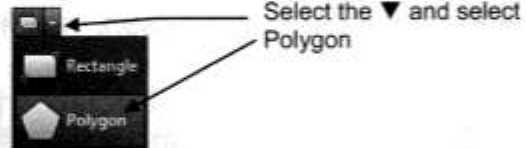
POLYGON

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>

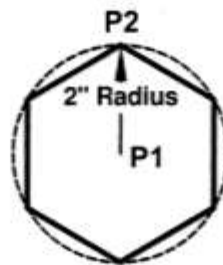


2. The following prompts will appear on the command line:

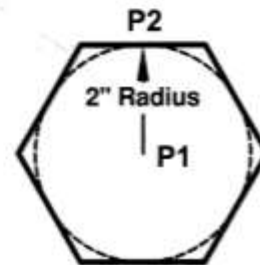
_polygon Enter number of sides <4>: **type number of sides <enter>**
 Specify center of polygon or [Edge]: **specify the center location (P1)**
 Enter an option [Inscribed in circle/Circumscribed about circle]<I>: **type I or C <enter>**
 Specify radius of circle: **type radius or locate with cursor. (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") but the Polygons are different sizes. Selecting Inscribed or Circumscribed is important.



INSCRIBED

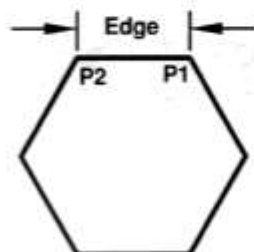


CIRCUMSCRIBED

EDGE METHOD

1. Select the **Polygon** command using one of the options shown above.
2. The following prompts will appear on the command line:

_polygon Enter number of sides <4>: **type number of sides <enter>**
 Specify center of polygon or [Edge]: **type E <enter>**
 Specify first endpoint of edge: **place first endpoint of edge (P1)**
 Specify second endpoint of edge: **place second endpoint of edge (P2)**



ELLIPSE

There are 3 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 pages illustrates each of the methods.

AXIS END METHOD

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

Ribbon = Home tab / Draw panel
or
Keyboard = EL <enter>



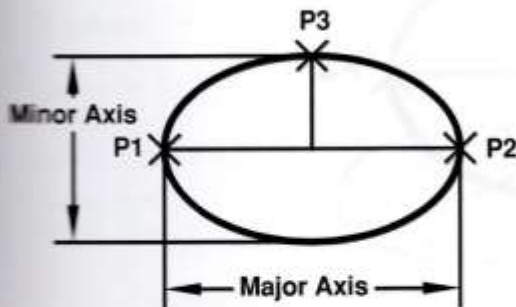
2. The following prompts will appear on the command line:

Command: _ellipse

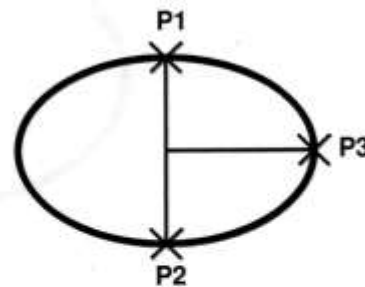
Specify axis endpoint of ellipse or [Arc/Center]: **place the first point of either the major or minor axis (P1).**

Specify other endpoint of axis: **place the other point of the first axis (P2)**

Specify distance to other axis or [Rotation]: **place the point perpendicular to the first axis (P3).**



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



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

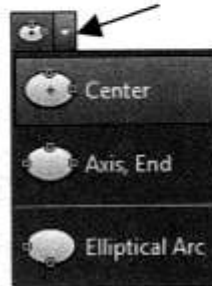
Continued on the next page...

ELLIPSE....continued

CENTER METHOD

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

Ribbon = Home tab / Draw panel
or
Keyboard = EL <enter> C <enter>



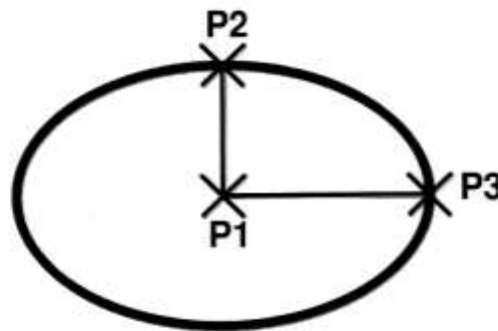
2. The following prompts will appear on the command line:

Command: `_ellipse`

Specify center of ellipse: *place center of ellipse (P1)*

Specify endpoint of axis: *place first axis endpoint (either axis) (P2)*

Specify distance to other axis or [Rotation]: *place the point perpendicular to the first axis (P3)*



Continued on the next page..

ELLIPSE....continued

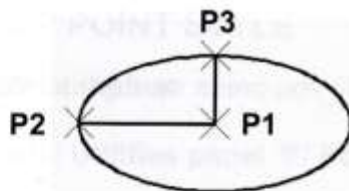
ELLIPTICAL ARC METHOD

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

Ribbon = Home tab / Draw panel
or
Keyboard = EL <enter> A <enter>



2. The following prompts will appear on the command line:
Command: _ellipse
Specify axis endpoint of elliptical arc or [center]: **type C <enter>**
Specify center of axis: **place the center of the elliptical arc (P1)**
Specify endpoint of axis: **place first axis point (P2)**
Specify distance to other axis or [Rotation]: **place the endpoint perpendicular to the first axis (P3)**
Specify start angle or [Parameter]: **place the start angle (P4)**

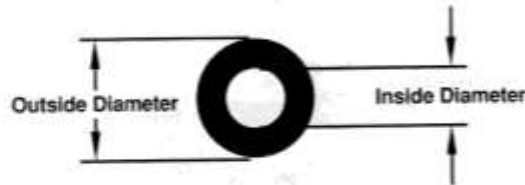


- Specify end angle or [Parameter/Included angle]: **place end angle (P5)**




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>

2. The following prompts will appear on the command line:

Command: `_donut`
 Specify inside diameter of donut: *type the inside diameter <enter>*
 Specify outside diameter of donut: *type the outside diameter <enter>*
 Specify center of donut or <exit>: *place the center of the first donut*
 Specify center of donut or <exit>: *place the center of the second donut or <enter> to stop*

Note:

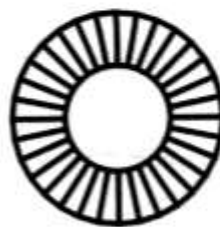
It will continue to create more donuts until you press <enter> to stop the command.

Controlling the “FILL MODE”

1. Command: *type FILL <enter>*
2. Enter mode [ON / OFF] <OFF>: *type ON or OFF <enter>*
3. Type **REGEN** <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>

Note: The Draw panel option creates multiple points until you press the **ESC** key.

The keyboard entry method creates single points.

2. The following prompts will appear on the command line:

Command: `_point`

Current point modes: `PDMODE=3 PDSIZE=0.000`

Specify a point: *place the point location*

Specify a point: *place another point or press the "ESC" key to stop*

HOW TO SELECT A "POINT STYLE"

1. Open the Point Style dialog box:

Ribbon = Home tab / Utilities panel ▼ / Point Style

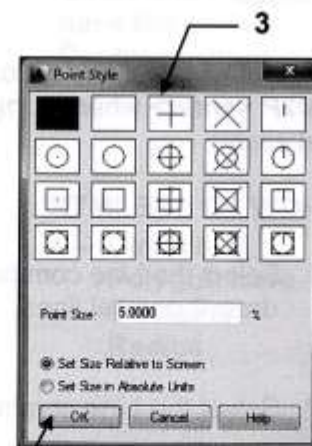
or

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

3 MORE OBJECT SNAP OPTIONS:



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 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

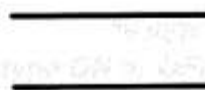
Mid Between 2 Points

Locates a midpoint between two points

*You may select this option from the object snap menu (shift+Rt Click) or you may type **M2P <enter>** when prompted for an endpoint. No tool or running object snap option is available.*

HOW TO USE "MTP":

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



2. Select the **Line** command again.

3. Type **M2P <enter>**

4. Using **Endpoint object snap** select each of the 2 endpoints (P1) and (P2)

P1 _____

P2 _____

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

