



AutoCAD Level 1

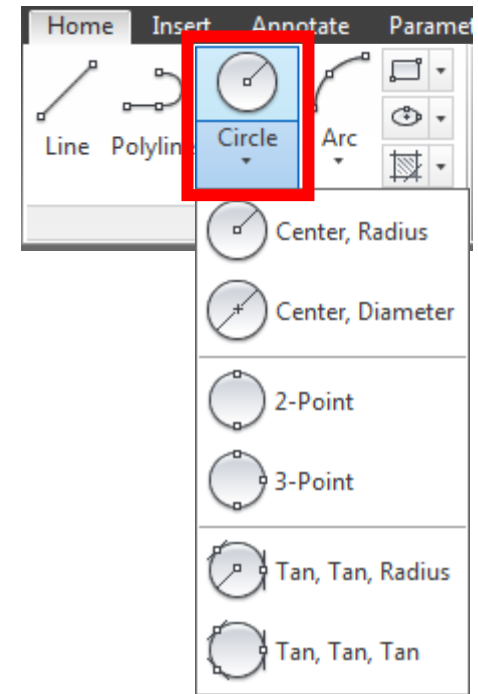
Session 3

AGENDA

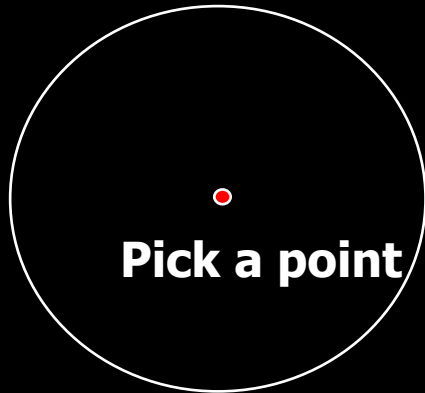
- **Drawing Circles**
- **Drawing Polylines**
- **Drawing Rectangles**
- **Drawing Polygons**
- **Drawing Arcs**
- **Drawing Ellipses**
- **Object Snap tool**

Drawing other shapes- Circle

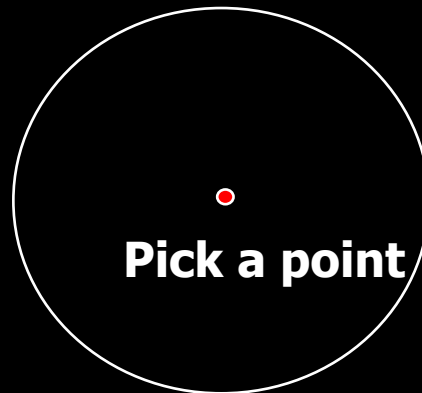
The Circle Command include: [Center, Radius], [Center, Diameter], [2 points], [3 points], [Tan, Tan, Radius], and [Tan, Tan, Tan]. The circle command is also located on the same toolbar as the Line command, on the left.



Radius



Diameter

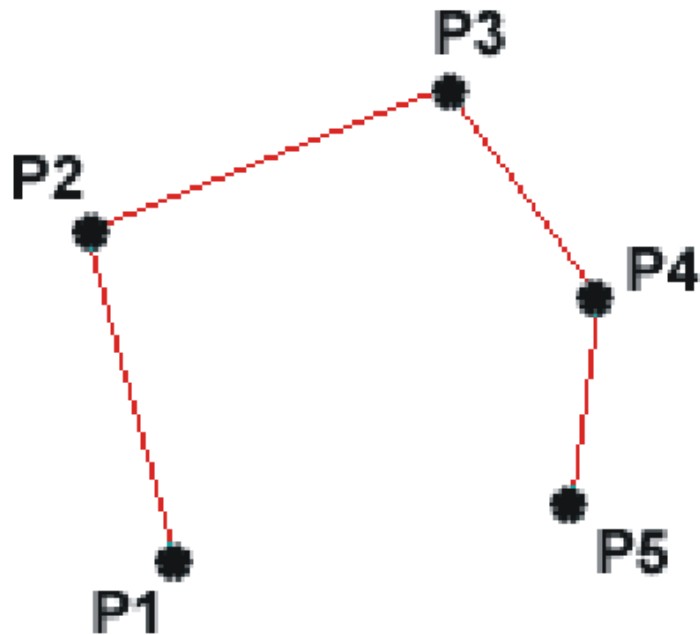
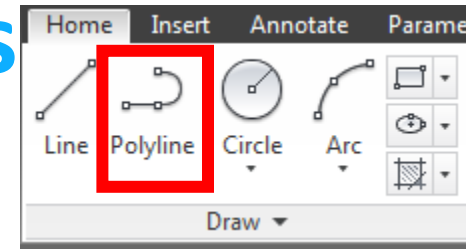


Command: **c**
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter]: **5**

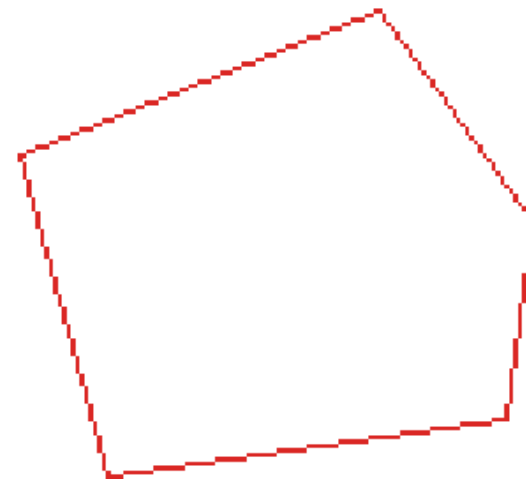
Command: **c**
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter]: **d**
Specify diameter of circle <10.0000>: **10**

Drawing other shapes- Polylines

- The Polyline or Pline command is similar to the line command except that the resulting object may be composed of a number of segments which form a single object.



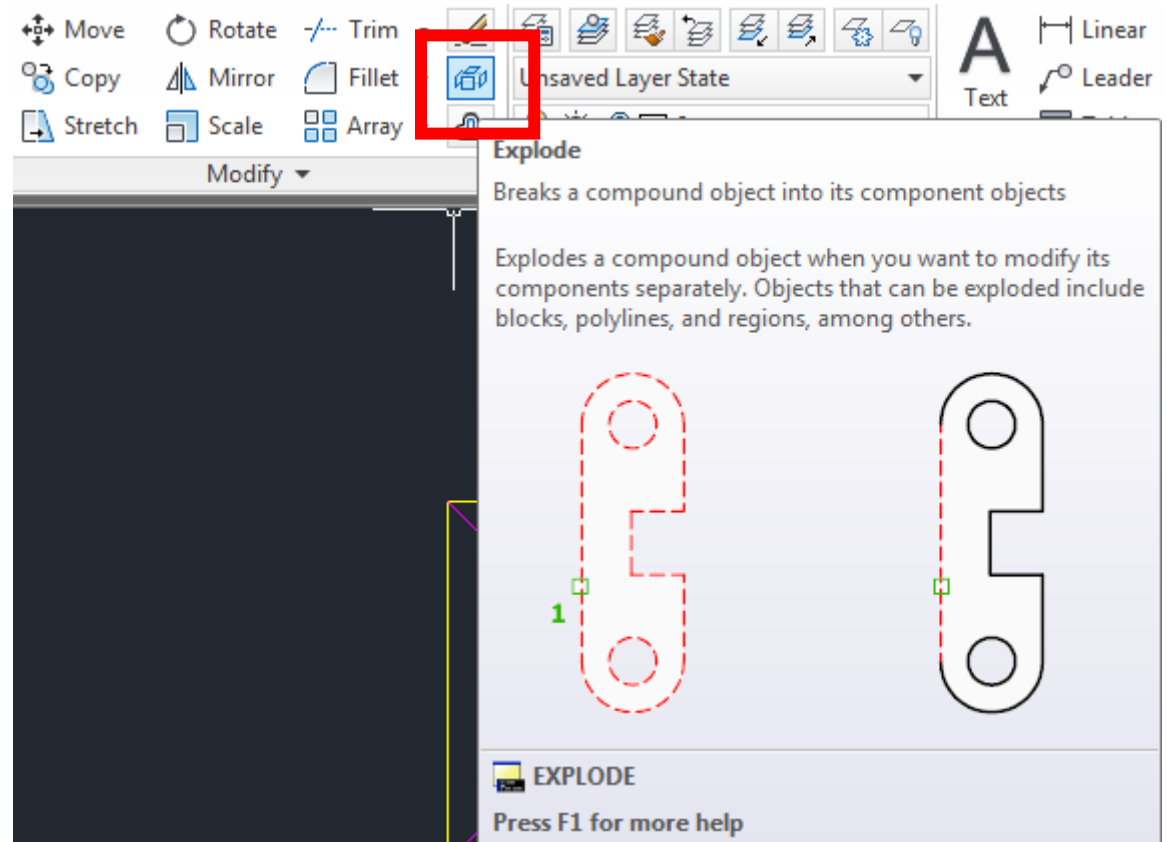
Open Polyline



Closed Polyline

Explode command.

- **The EXPLODE command allows you to change a grouped item into its individual elements so it can be edited.** For example: converting the previous polyline into 2 regular lines and 2 arcs.

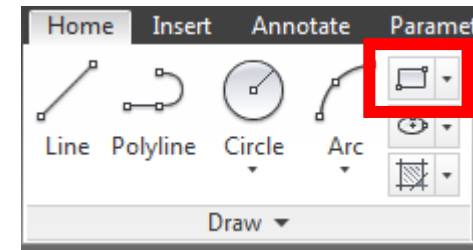
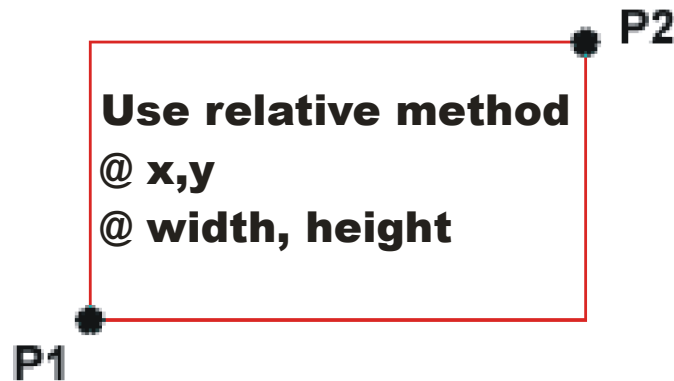


Joining objects into a polyline- Join Command

- Command: **JOIN**
- Select source object or multiple objects to join at once: : 6 found
- Select objects to join: enter
- 6 objects converted to 1 polyline

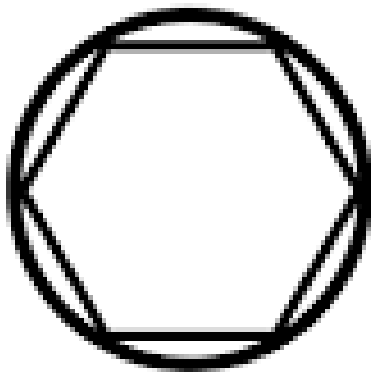
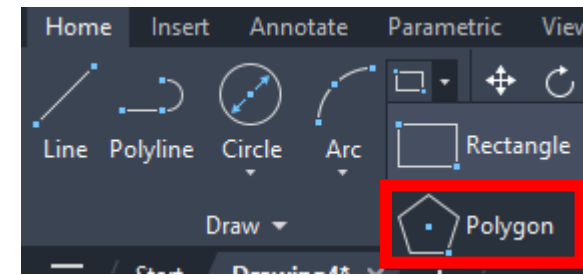
Drawing other shapes- Rectangle

- The Rectangle command is used to draw a rectangle whose sides are vertical and horizontal. The position and size of the rectangle are defined by picking two diagonal corners. The rectangle isn't really an AutoCAD object at all. It is, in fact, just a closed polyline which is automatically drawn for you.
- Method 1- Command: RECTANG
 - Specify first corner point or
[Chamfer/Elevation/Fillet/Thickness/Width]: (pick P1)
 - Specify other corner point or [Dimensions]: (pick P2)

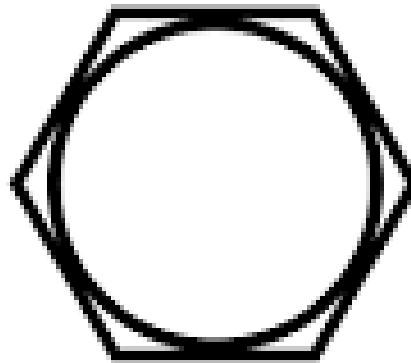


Drawing other shapes- Polygon

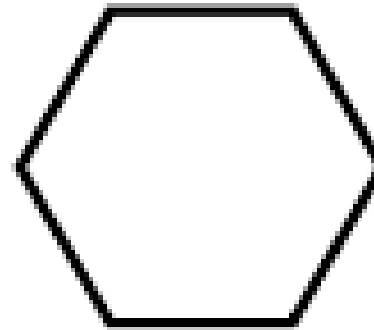
- A polygon is a shape with four or more sides.



INSCRIBED
IN
CIRCLE

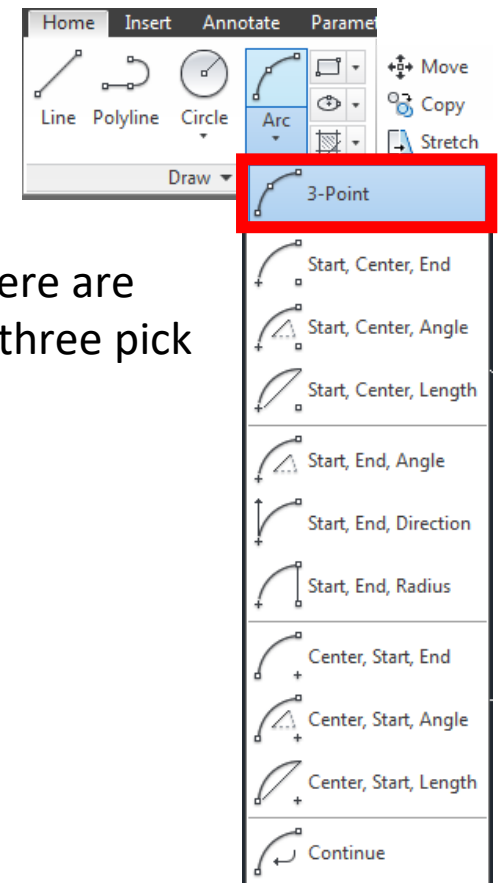


CIRCUMSCRIBED
ABOUT
CIRCLE



EDGE
POLYGON

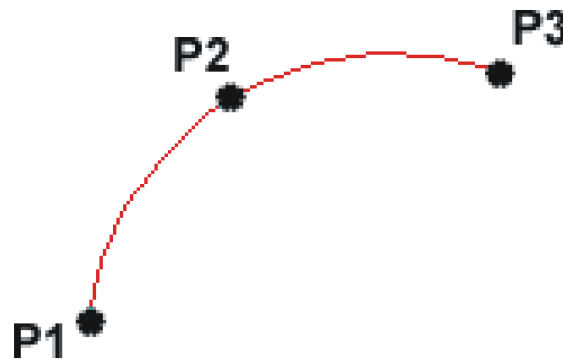
Drawing other shapes- Arc



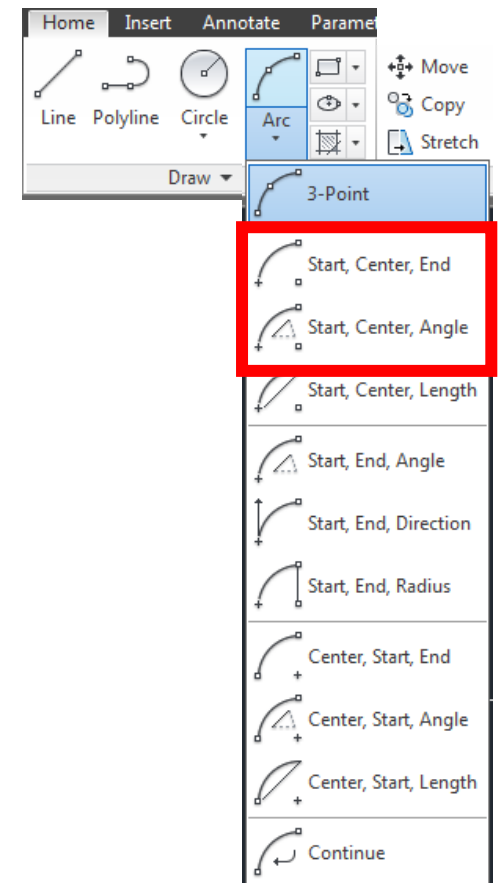
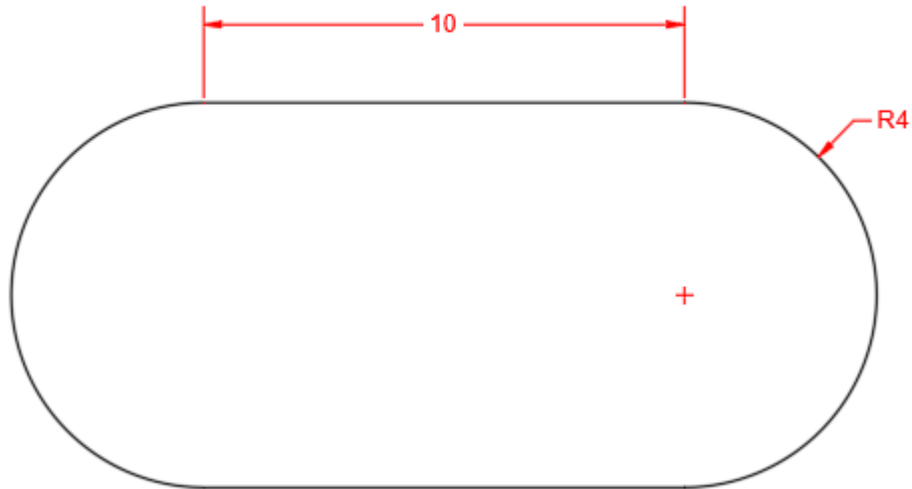
- The Arc command allows you to draw an arc of a circle. There are numerous ways to define an arc, the default method uses three pick points, a start point, a second point and an end point.

Command: ARC

- Specify start point of arc or [Center]: (pick P1)
- Specify second point of arc or [Center/End]: (pick P2)
- Specify end point of arc: (pick P3)

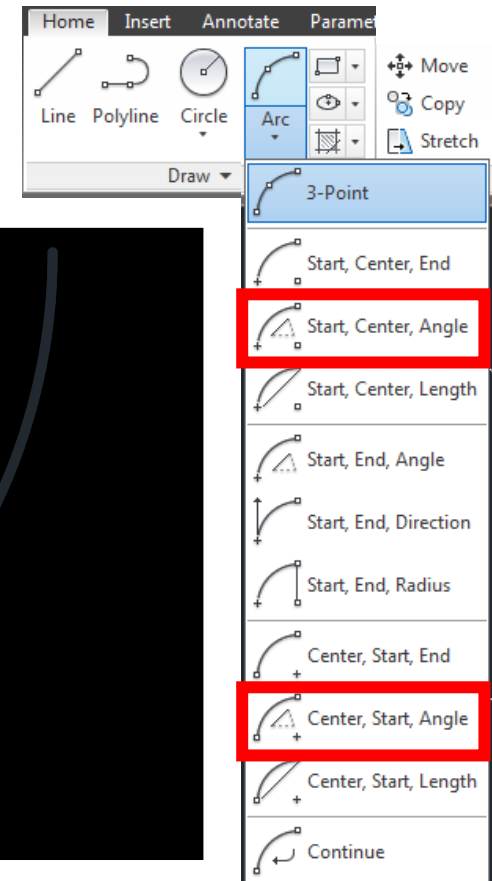


Drawing other shapes- Arc



Drawing other shapes- Arc for a door swing

Use the door line to pick the start point and centre point of the arc (door swing). Do not forget to use the object snap tool endpoint to pick the points accurately.



Command: _arc Specify start point of arc or [Center]: **end**
Of **pick start pt**

Specify second point of arc or [Center/End]: **_c** Specify center point of arc: **end**
Of **pick centre pt**

Specify end point of arc or [Angle/chord Length]: **_a** Specify included angle: **90**

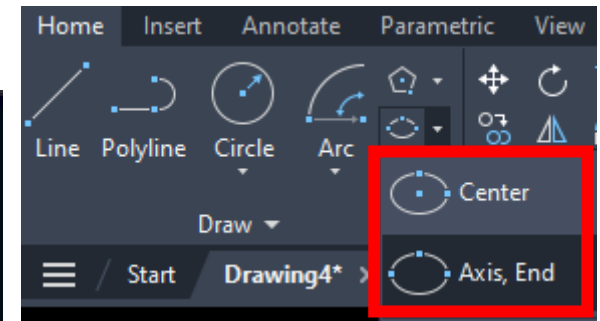
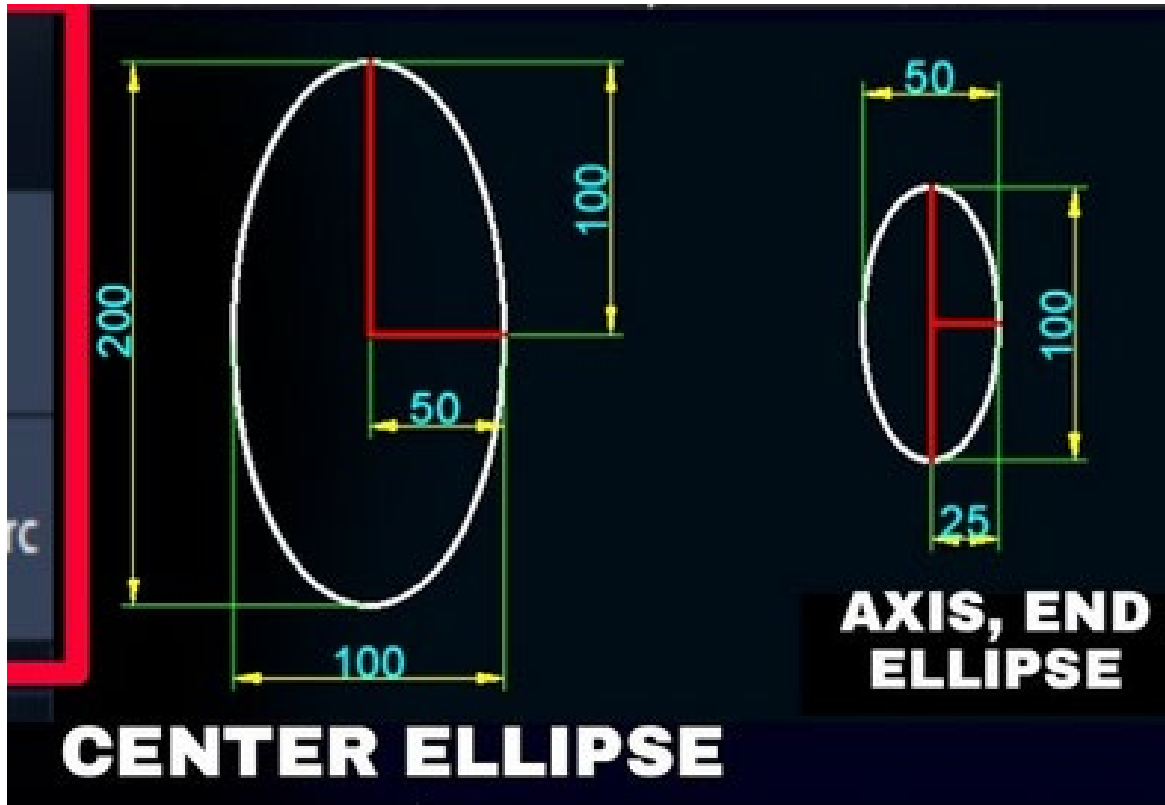
Start point

Centre point

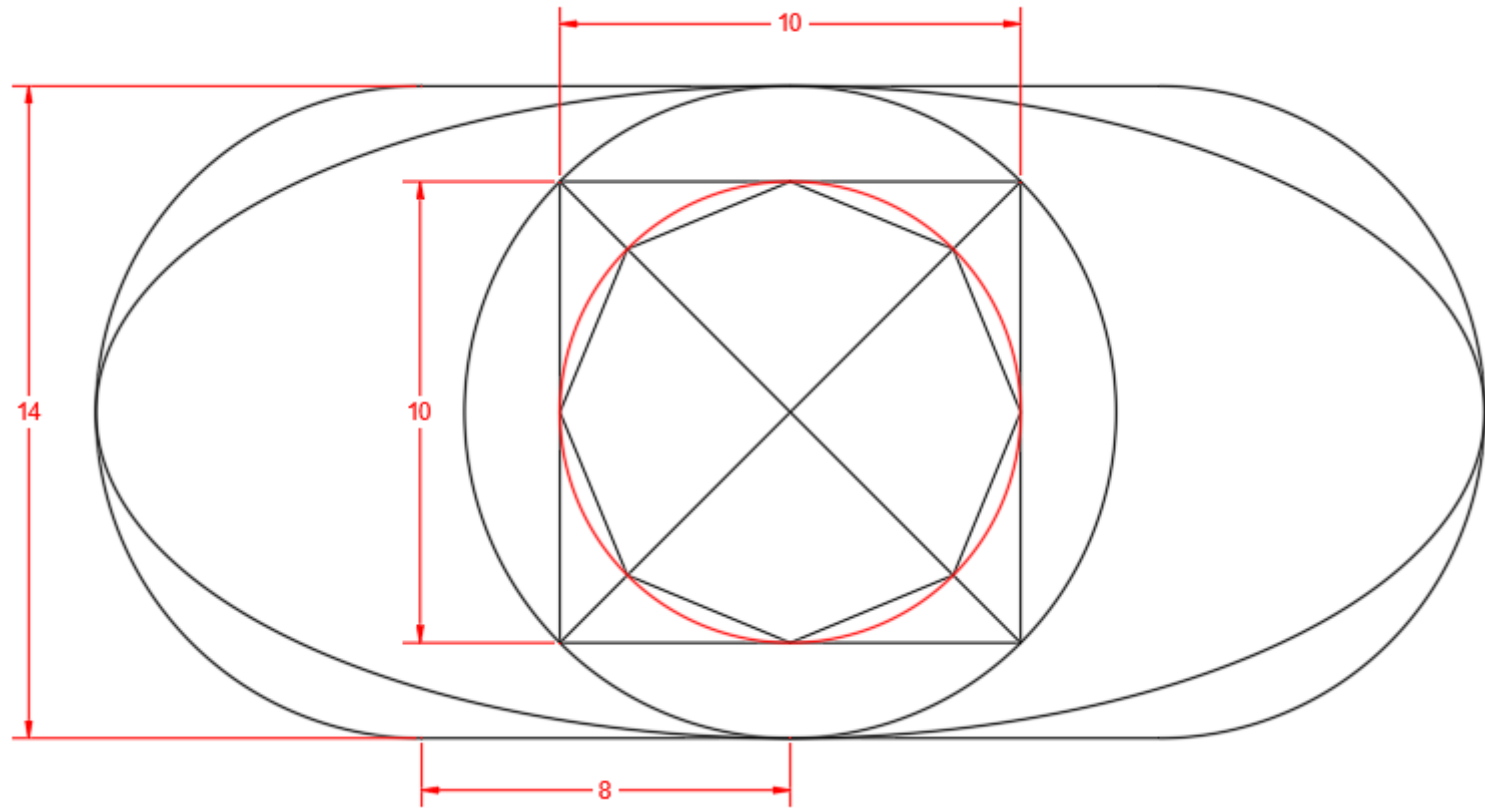
90 degrees

- You can start your arc by specifying the center of the arc or the start point. If you choose the Center option, AutoCAD prompts you for the center point first and the start point second. AutoCAD defines arcs counterclockwise, so pick a start point in a clockwise direction from the endpoint. After you specify the center and start point, AutoCAD presents several options you can choose, including the following:
- Angle: This option specifies the included angle that the arc sweeps out. A 180-degree angle, for example, is a semicircle.
- Length of chord: This option specifies the length of an imaginary straight line connecting the endpoints of the arc. Most people seldom or never use this option.
- Endpoint: This option specifies where the arc ends. It's the default option and is often the easiest to use.
- If you specify the start point as the first option, you can choose among the following three command line options as well:
- Center: This option prompts you for the arc's center point and then finishes with the three options listed previously.
- End: This option specifies the endpoint of the arc. You then need to define the angle the arc covers, its direction, its radius, or its center point.
- Second point: This is the default option. The second point you choose is not the endpoint; instead, it's a point on the arc that, along with the start and end points, defines how much the arc curves. After you enter the second point, you must enter an endpoint to complete the arc.

Drawing other shapes- Ellipse
















Exercise 6.dwg

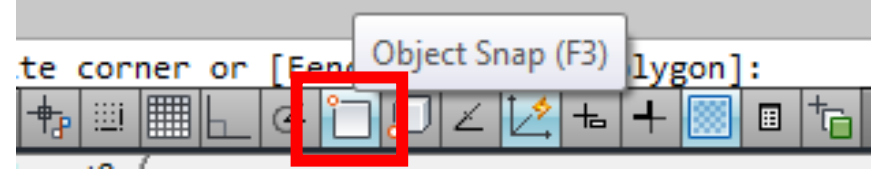


Object snap

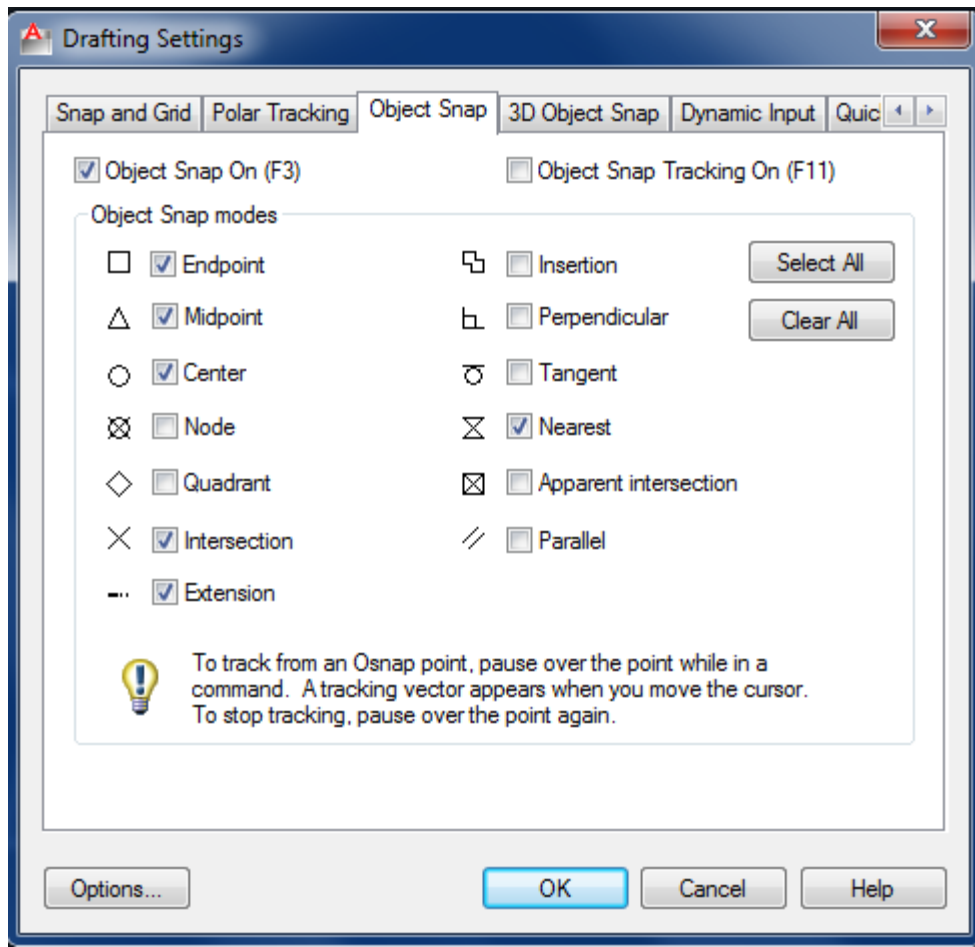
- Object snaps (Osnaps) are one of the handiest tools in AutoCAD. Without them, you would have a lot of trouble doing even the simplest dimension accurately. Before discussing how they are used, first think about what they are. Their name, "object snaps" means that they snap to objects, or more precisely, they snap to specific parts of an object. A line for example has 3 points that you can snap to: a midpoint and two endpoints. A circle has 5 points: a center and 4 quadrants. To use Osnaps effectively, you must know what points an object has that you are able to snap to.
- There are also some Osnaps that are not precise. These allow you to snap to (sometimes) arbitrary points along an object. These would include nearest, tangent, perpendicular and to some extent, intersection.
- There are two different ways of working with Osnaps; in running mode or invoking them individually as required.

	Endpoint
	Midpoint
	Center
	Node
	Quadrant
	Intersection
	Extension
	Insertion
	Perpendicular
	Tangent
	Nearest
	Apparent Intersection
	Parallel

Running Object snap (F3)



- Running Osnaps mean that certain ones are turned on and running in the background. This means that they are available when needed in the middle of a command.
- Right click on the object snap icon then choose settings.



Object snap

