Diyala University

College of Engineering

Computer Engineering Department

Drawing by using Computer

AutoCAD

First Year

Lecture 3

By

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Lecture 3

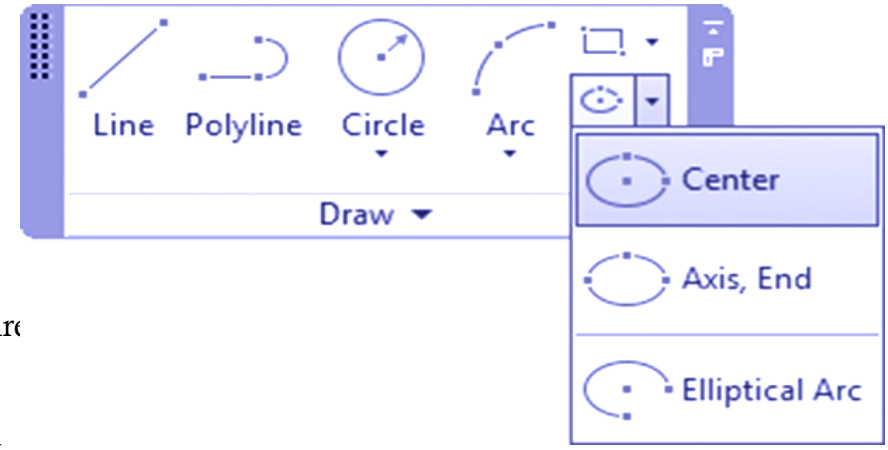
In This lecture

- What are the drafting priorities?
- How to draw lines, circles, and arcs using precise methods
- How to draw polylines using precise methods
- How to convert lines and arcs to polylines and vice versa
- What are Object Snap and Object Track?

3.1 DRAWING ELLIPSE

3.1.1 Using The Ellipse Command

This command will draw an elliptical shape or elliptical arc. To issue this command go to the Home tab, locate the Draw panel, then select the Ellipse button and select one of the methods:



There are thre

- Center
- Axis, End
- Elliptical Arc

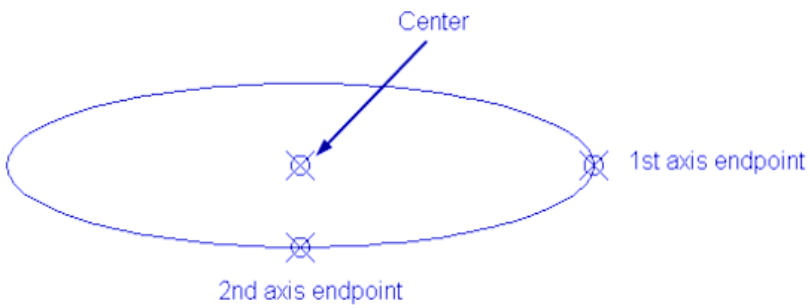
Where the first two options will draw an ellipse, the third option will draw an elliptical arc. Here is a discussion for each one of them:

3.1.2 Drawing An Ellipse Using The Center Option

Using this method, the user should specify three points, which are:

- Center point of the ellipse
- Endpoint of one of the two axes
- Endpoint of the other axis

The following picture will illustrate the concept:



You will see the following prompts:

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

Specify center of ellipse: Specify endpoint of axis:

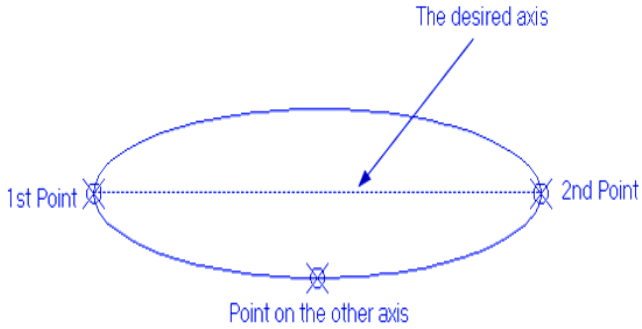
Specify distance to other axis or [Rotation]:

3.1.3 Drawing An Ellipse Using Axis Points

The user should specify three points, which are:

- Point on one end of one of the axis
- Point on the other end of the same axis
- Point on the other axis

The following picture will illustrate the concept:



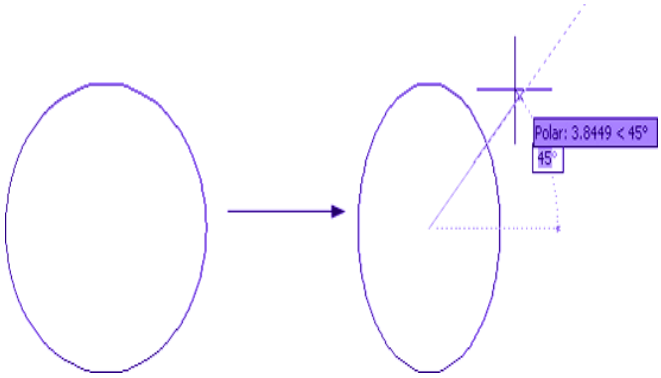
You will see the following prompts:

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

Specify other endpoint of axis:

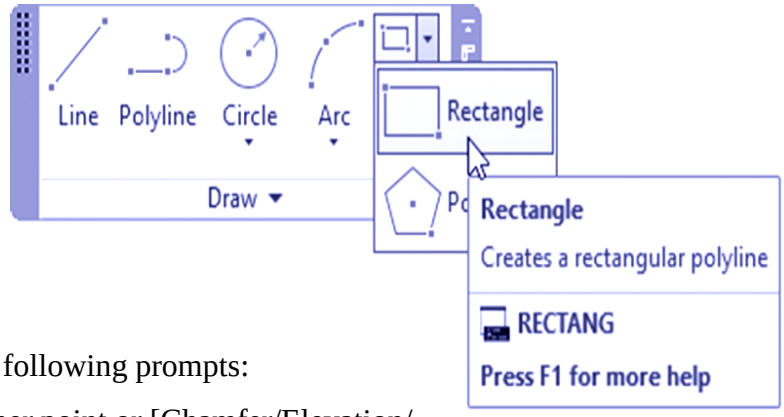
Specify distance to other axis or [Rotation]:

Using either method, the last step will include an option called Rotation. So what is rotation? After you define two points, you will draw a circle. Imagine this circle is in a plane and the plane is rotating; you will get an ellipse. Check the following illustration:



3.2 DRAWING USING THE RECTANGLE COMMAND

This command will draw a rectangle or square shape. The Rectangle command will use a polyline as an object. To issue this command go to the Home tab, locate the Draw panel, then select the Rectangle button:



You will see the following prompts:

Specify first corner point or [Chamfer/Elevation/ Fillet /Thickness/Width]:

Specify other corner point or [Area/Dimensions/ Rotation]:

By default, you can draw a rectangle by specifying two opposite corners. The other options are:

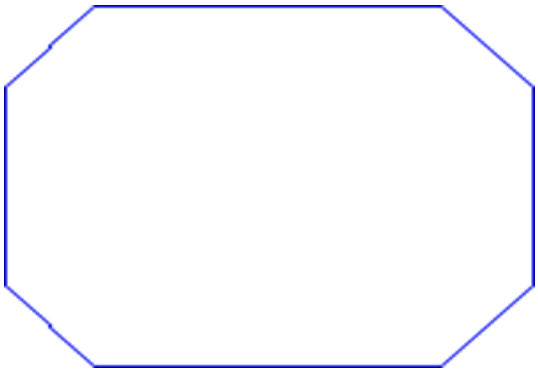
3.2.1 Chamfer Option

This option will draw a rectangle with chamfered edges. The user will see the following prompts:

Specify first chamfer distance for rectangles <0.00>:

Specify second chamfer distance for rectangles <0.2>:

Specify the first and second distance, then the Rectangle command will continue with normal prompts. Check the following example:



3.2.2 Elevation Option

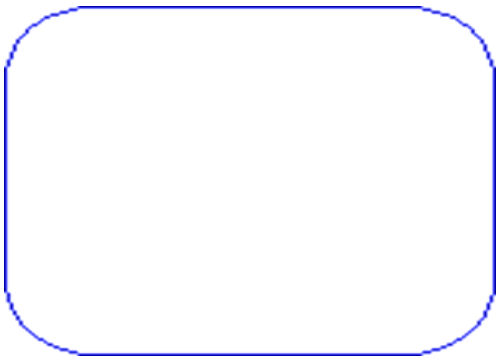
This option is used for 3D only.

3.2.3 Fillet Option

This option is identical to the Chamfer option, except the user has to in- put the Radius instead of Distances. The user will see the following prompt:

Specify fillet radius for rectangles <0.0000>:

Specify the fillet radius, then the Rectangle command will continue with normal prompts. Check the following example:



3.2.4 Thickness Option

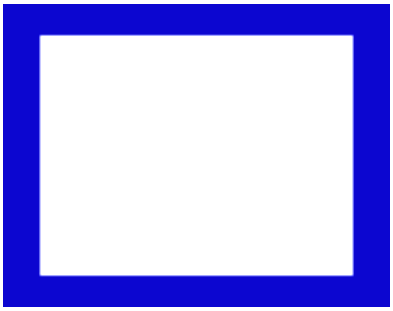
This option is available for 3D only.

3.2.5 Width Option

The user will be able to draw a rectangle with width by using this option. The user will see the following prompt:

Specify line width for rectangles <0.0000>:

Specify the width value, and then the rectangle command will continue normally. Check the following example:



3.2.6 Area Option

This option will specify the total area of the rectangle prior to specifying the second corner. The user will see the following prompts:

Enter area of rectangle in current units <25.0000>:

Calculate rectangle dimensions based on [Length/Width]

<Length>:

Enter rectangle length <10.0000>:

AutoCAD asks the user to input the total area, and then asks the user to input either the length (in X-axis) or width (in Y-axis). AutoCAD will then draw a rectangle above and to the right of the first corner.

3.2.7 Dimensions Option

This option will draw a rectangle by specifying length (in X-axis) and width (in Y-axis). The user will see the following prompts:

Specify length for rectangles <10.0000>:

Specify width for rectangles <10.0000>:

Specify other corner point or [Area/Dimensions/

Rotation]:

AutoCAD asks you to input the length and the width; in the final prompt, the user is invited to input the position of the second point.

3.2.8 Rotation Option

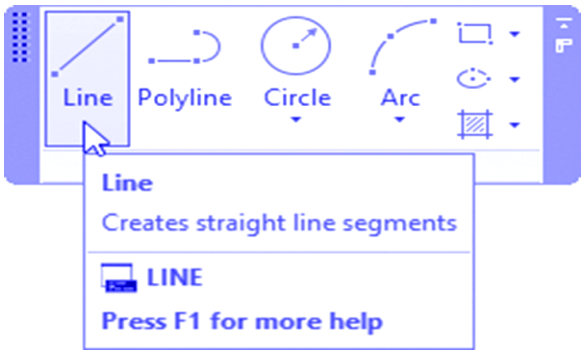
This option will draw a rectangle with a rotation angle. The user will see the following prompt:

Specify rotation angle or [Pick points] <0>:

Specify the rotation angle either by typing the value or by specifying points.

3.3 DRAWING LINES USING LINE COMMAND

This command will enable you to draw straight lines; each line segment presents a single object. To issue this command go to the Home tab, locate the Draw panel, and then select the Line button:



You will see the following AutoCAD prompts:

Specify first point:

Specify next point or [Undo]: Specify next point or [Undo]: Specify next point or [Close/Undo]:

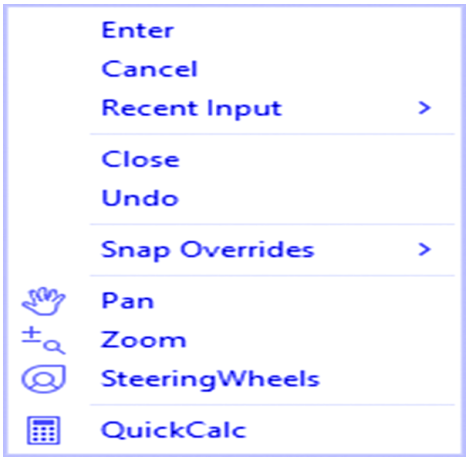
Using the first prompt, specify the coordinates of your first point. Keep specifying points, and when you are done, do one of the following:

■ If you want to stop without closing the shape, simply press [Enter] ([Esc] will do the job as well, but don't make it a habit, as [Esc] generally means abort)

■ If you want to close the shape and finish the command, press C on the keyboard or right-click and select the Close option

■ If you made any mistakes, you can undo the last point by typing U on the keyboard or right-clicking and selecting the Undo option

This is what the right-click menu looks like:



3.3.1 what Is Dynamic Input In AutoCAD?

Dynamic Input has multiple functions:

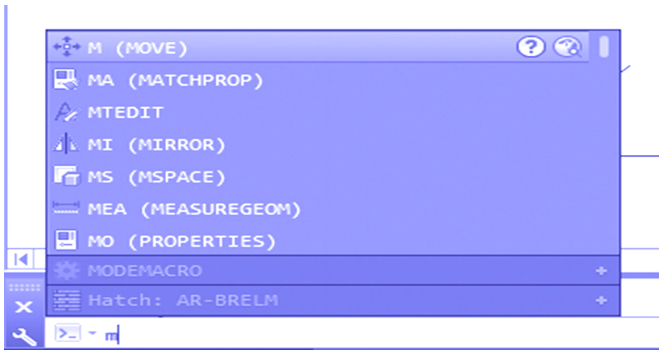
■ It will show prompts at the command window in the graphical area

■ It will show the lengths and angles of the lines before drafting, which will specify them accurately

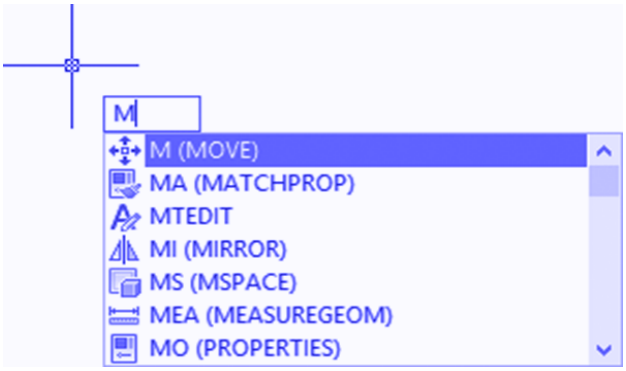
In order to turn on/off the **Dynamic Input**, click the following button on the Status bar:

3.3.2 Example for Showing Prompts

By default, if you type any command using the command window, AutoCAD will help you by showing all the commands starting with the same letters, as in the following example:



We typed the letter m, and accordingly AutoCAD gave us all the commands starting with this letter. While Dynamic Input is on, this is applicable to the crosshairs as well, as in the following:

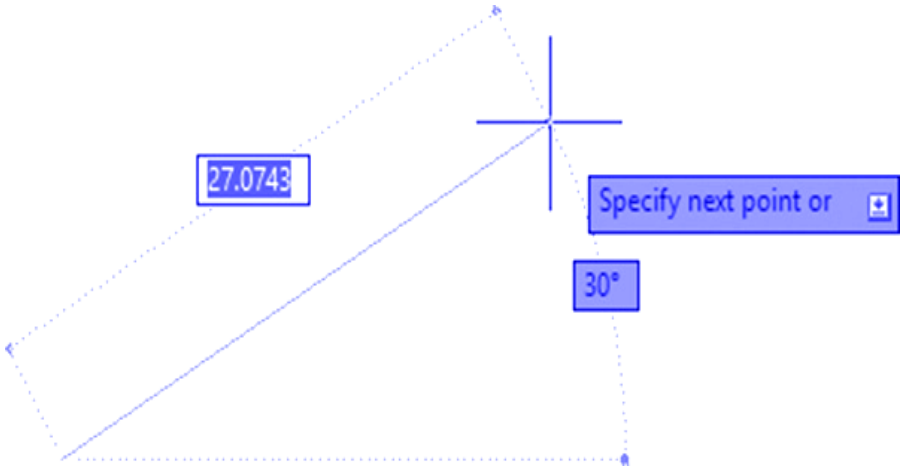


Let's assume we started the command window, using either way. Once you press [Enter], the following prompt will appear:

Type in the x and y coordinates, using the [Tab] key to move between the two fields:

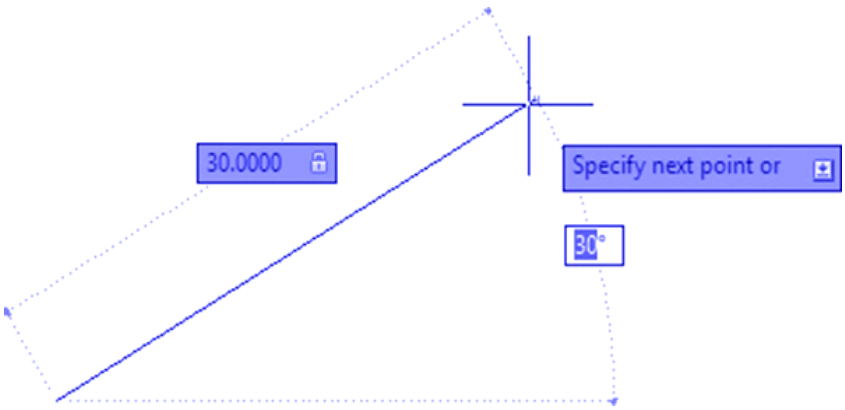
3.3.3 Example for Specifying Lengths and Angles

Once you specify the starting point, AutoCAD will use Dynamic Input to show the length and the angle of the line using rubber band mode:



NOTE- Angles are measured CCW starting from the east, but only for 180°, unlike the angle system in AutoCAD, which will be for the whole 360°.

Type the length of the line, then press [Tab] to input the angle (it will give 1° increments); once you are done, press [Enter] to specify the first line. Continue doing the same for the other segments:



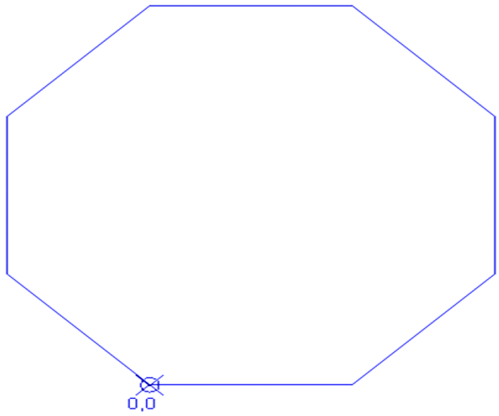
PRACTICE 3-1 DRAWING LINES USING DYNAMIC INPUT

1.Start AutoCAD 2019

2.Open file Practice 2-1.dwg

3.Using status bar, click off, Polar Tracking, Ortho, and Object Snap, and make sure Dynamic Input is on

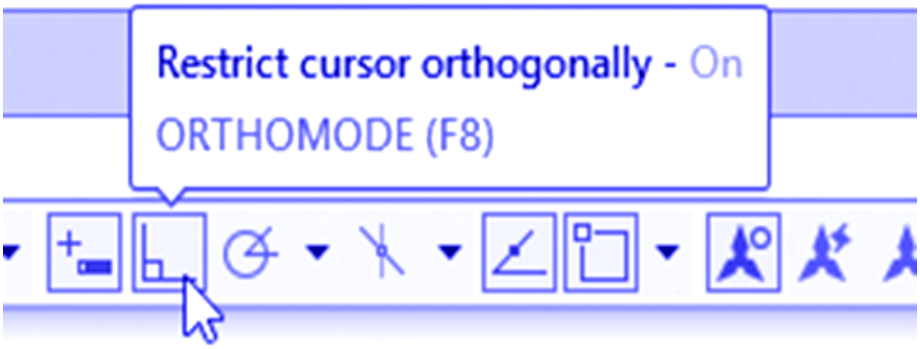
4.Draw the following shape, using 0,0 as your start point, bearing in mind all sides = 4, and all angles are multiples of 45°:



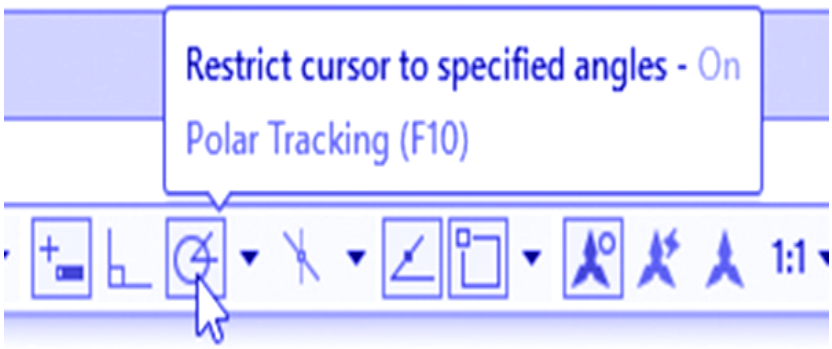
5.Save the file and close it.

3.3.4 Exact Angles (Ortho Vs. Polar Tracking)

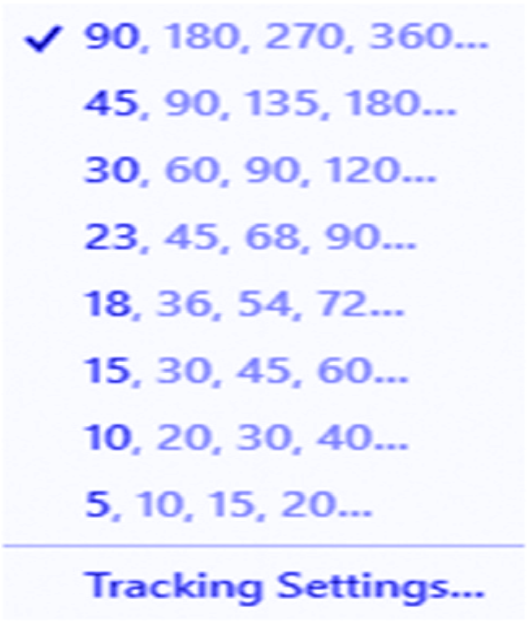
When using Dynamic Input angles are incremented by 1°, but still the user can't depend on it to specify angles precisely in AutoCAD. The **Ortho** function will force the lines to be at right angles (orthogonal) using the following angles: 0, 90, 180, and 270.In order to turn on/off the **Ortho**, use the following button in the status bar:



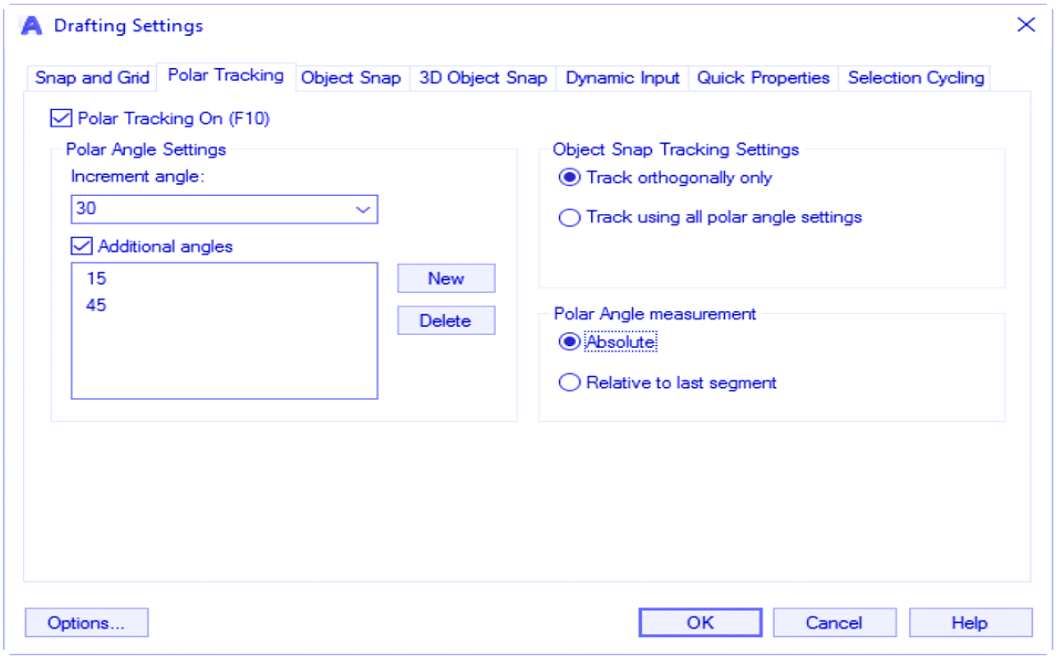
However, what if we want to use other angles such as 30, 45, 60, etc.? Ortho will not help in this case; for this reason, AutoCAD introduced an- other function called Polar Tracking, which will show in the graphical area rays starting from the current point heading toward angles like 30, 45, etc. Since Ortho and Polar Tracking contradict each other, when you switch one on, the other will be turned off automatically. In order to turn on/off **Polar Tracking**, use the following button in the status bar:



If you click the small triangle at the right of the button, you will see the following menu:



You can select the desired angle or select Tracking Settings to change some of the default settings of Polar Tracking. You will see the following dialog box:



3.3.5 Increment Angle

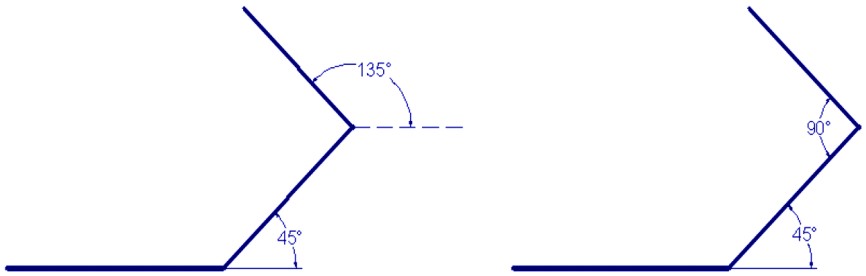
Increment Angle is the angle to be used along with its multiples. Select one from the list or type your own.

3.3.6Additional Angles

If you are using 30 as your increment angle, then 45 will not be among the angles that Polar Tracking will allow you to use. Hence, you will need to specify it as an additional angle. However, be aware you will not use its multiples.

3.3.7 Polar Angle Measurement

When you are using Polar Tracking, you have the ability to specify angles as an absolute angle (based on 0° at the east) or use the last line segment to be your 0 angle. Check the following illustration:

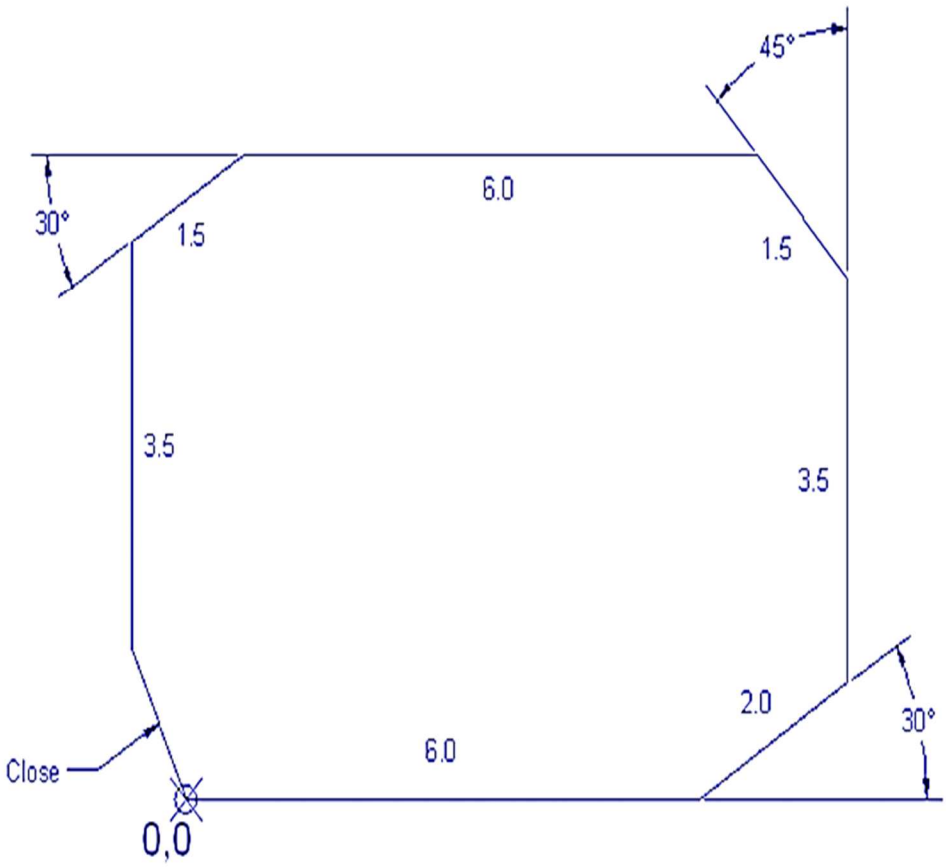


Measured using 0.0 angle Measured using the last segment

NOTE- While you control the angle using either Ortho or Polar Tracking, you can type in the distance desired and then press [Enter] to draw accurate distances. This method is called Direct Distance Entry.

PRACTICE 3-2 EXACT ANGLES

1. Start AutoCAD 2019
2. Open Practice 2-2.dwg
3. Draw the following shape (without dimension) using line command, starting from 0,0 as your starting point, bearing in mind you have to use Polar Tracking. Set the proper Increment angle and Additional angles, using the Direct Distance Entry method to input the exact distances:



4.Save and cl

3.4PRACTIS DRA FTING USINING OBJECT SNAP

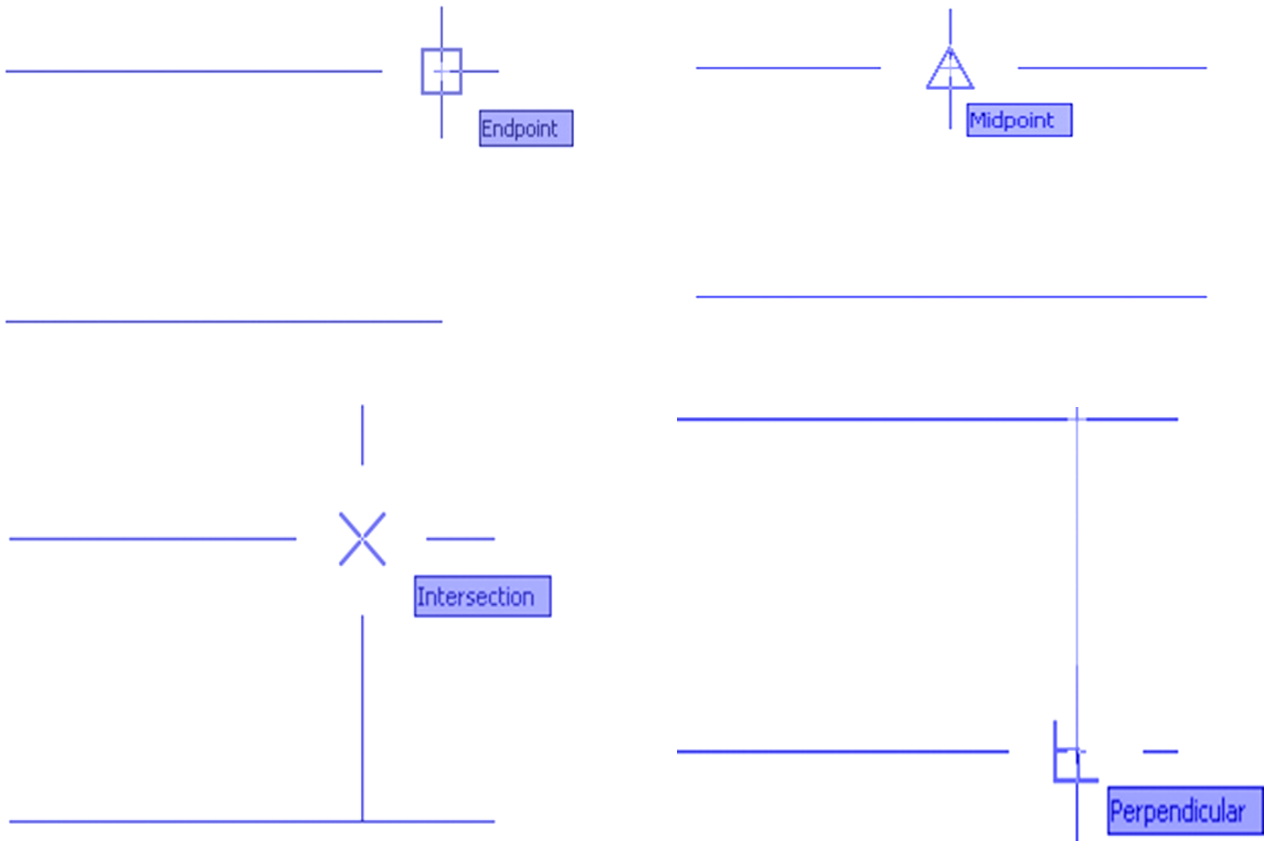
Object Snap, or OSNAP, is the most important accuracy tool to be used in AutoCAD for 2D and 3D as well. It is a way to specify points on objects precisely using the AutoCAD database stored in the drawing file.

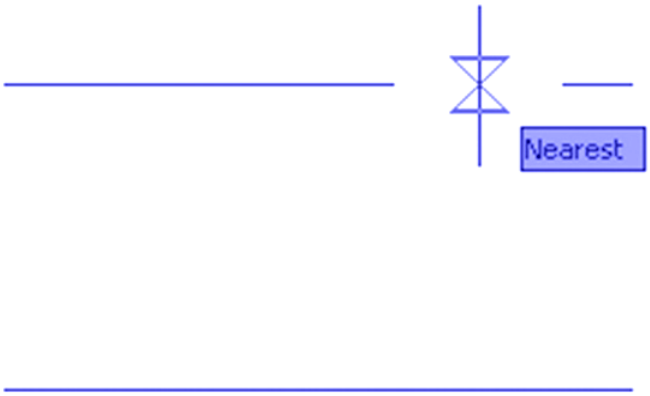
Some of the Object Snaps are:

- **Endpoint**: To catch the Endpoint of a line, or arc
- **Midpoint**: To catch the Midpoint of an a line, or arc
- **Intersection**: To catch the Intersection of two objects (any two objects)
- **Perpendicular**: To catch the Perpendicular point on an object
- **Nearest**: To catch a point on an object Nearest to your click point

We will discuss more object snaps when we discuss more drawing objects.

Here are graphical presentations of each one of these OSNAPs:



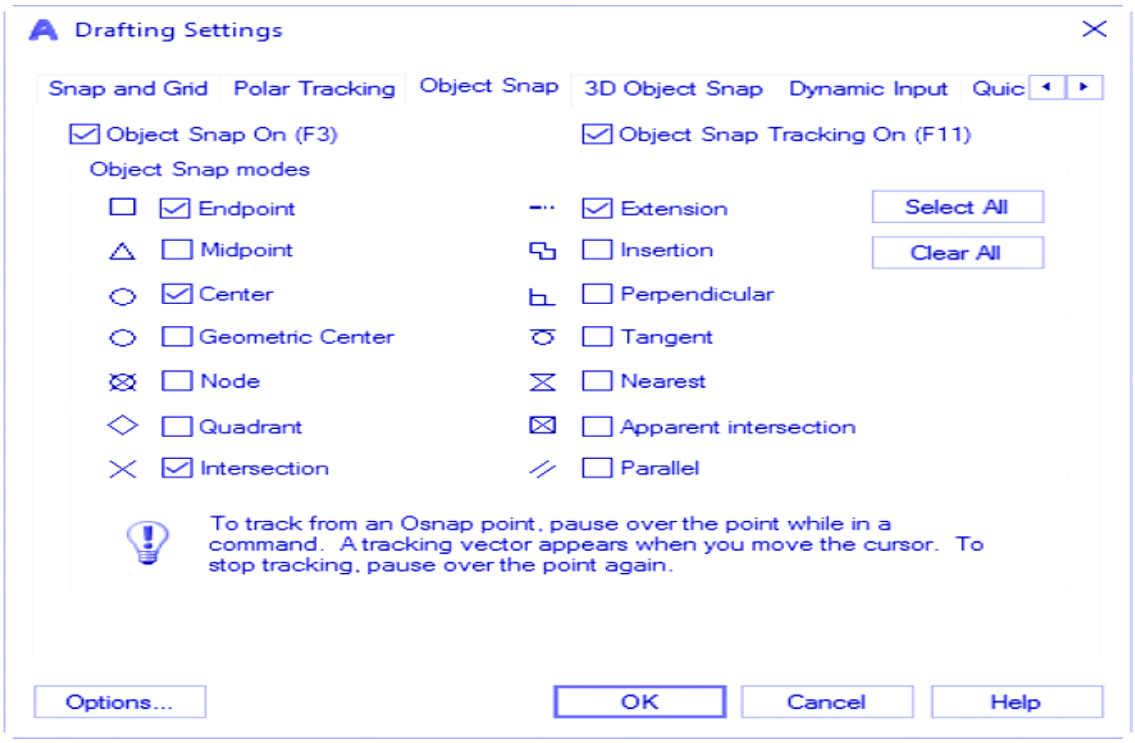


3.4.1 Activating Running OSNAPs

To activate running OSNAPs in the drawing, click on the Object Snap button in the status bar:

If you click the small triangle at the right of the button, you will see the following menu:

You can switch on the desired OSNAPs one by one. If you want to make a more convenient change, select the **Object Snap Settings** option, and the following dialog box will appear:



There are two buttons at the right: **Select All** and **Clear All**. We recommend using Clear All first and then selecting the desired OSNAPs. When done, click **OK**.

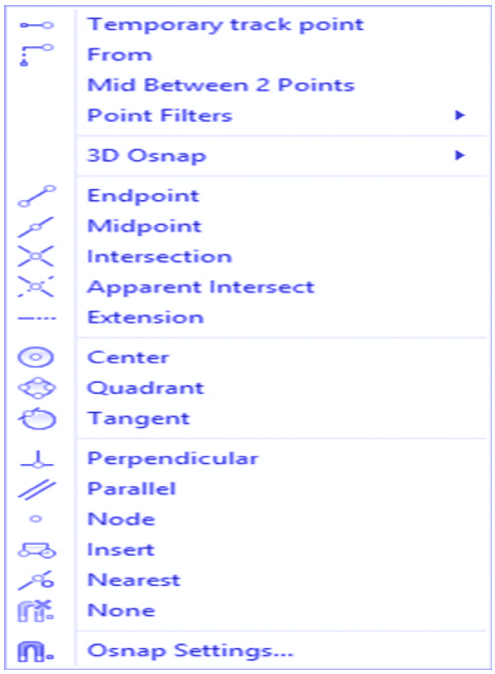
3.4.2 OSNAP Override

While the Object Snap button is on, several osnaps are working, and others are not! Assume you want to temporarily switch all of them off, use a single osnap, and then after finishing set everything back to normal. This is what we call OSNAP Override.

There are two ways to activate an override:

- Using the keyboard, type the first three letters of the desired OSNAP

Using the keyboard, hold the [Shift] key, and then right-click. You will see the following pop-up menu:



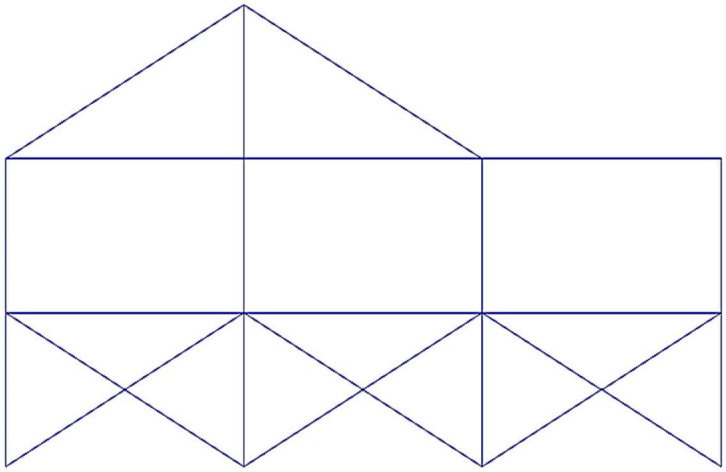
PRACTICE 3-3 OBJECT SNAP (OSNAP)

1.Start AutoCAD 2019

2.Open Practice 2-3.dwg

3.Check the Object Snap at the status bar and make sure that Endpoint, Midpoint, Intersection, and Perpendicular are the only OSNAPs switched on

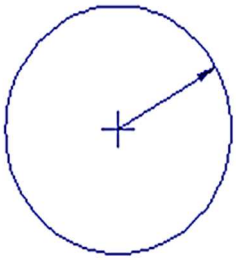
4.Using Object Snap and Line command, draw lines in the drawing to cre- ate the following:



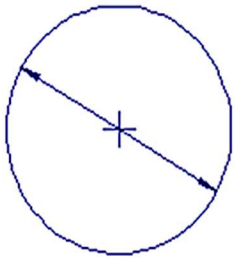
5.Save and close the file.

3.5 Drawing Circles Using Circle Command

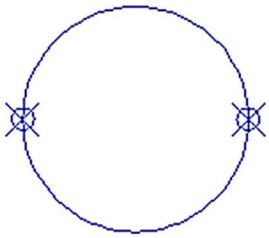
This command will draw a circle using different methods based on the available data. If you know the coordinates of the center, there are two possible methods. If the user knows the coordinates of points at the diameter of the circle, there are another two methods. Finally, if there are drawn objects like lines, arcs, or other circles which can be used as tangents for the to-be-created circles, there are two more methods. The following are the six methods to draw a circle in AutoCAD:



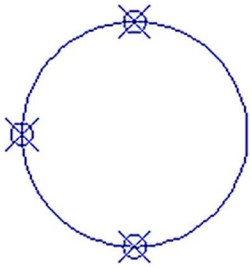
Input Center and Radius



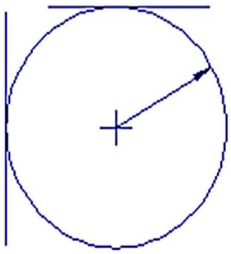
Input Center and Diameter



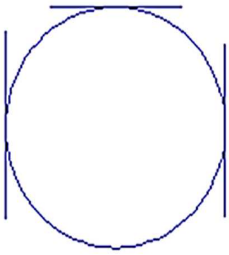
Input two points (diameter)



Input any three points

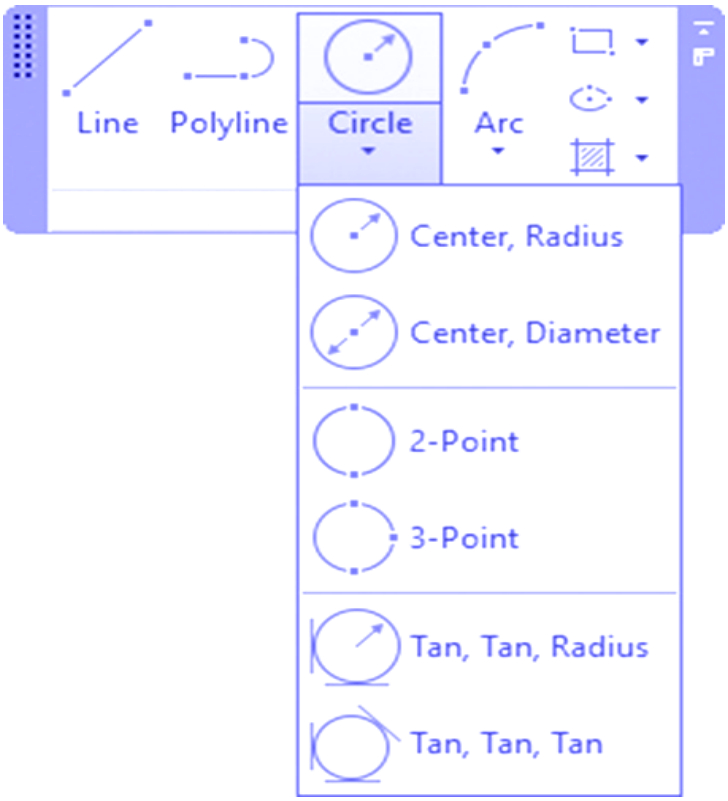


Tangent, Tangent, Radius



Tangent, Tangent, Tangent

To issue this command, go to the **Home** tab, locate the **Draw** panel, then select the arrow near the Circle button to see all the available methods



3.5.1 Drawing Circular Arcs Using Arc Command

This command will draw an arc part of a circle. To make our lives easier, AutoCAD uses eight pieces of information related to a circular arc. These are:

■ The starting point of the arc

■ Any point as a second point on the parameter of the arc

■ The ending point of the arc

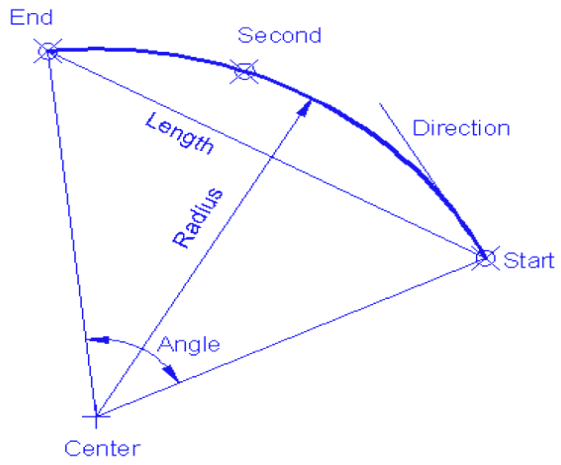
■ The Direction of the arc, which is the tangent that passes through the Start point. User should input the angle of the tangent

■ The distance between the starting point and the ending point which is called Length of Chord

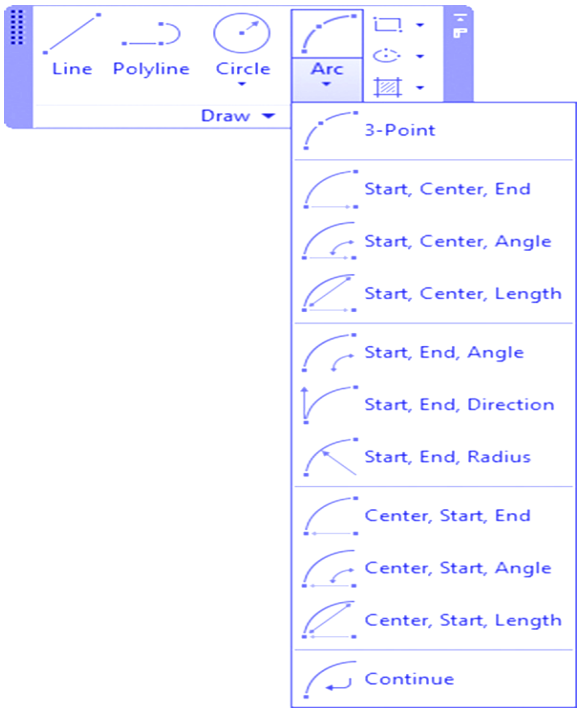
■ The Center point of the arc

■ The Radius

■The angle between Start-Center-End which is called Included Angle Check the following illustration:



If you provide three of these eight pieces of information, AutoCAD will be able to draw an arc, but not with any three. The combination of the information needed can be found in the **Home** tab, using the **Draw** panel, while clicking the arrow near the **Arc** button to see all the available methods:



As you can see, the Start point is always required information. Normally the user should think counterclockwise when specifying points, but if on the other hand you want to work clockwise, simply hold the [Ctrl] key and it will change.

3.5.2 Object Snaps Related To Circles And Arcs

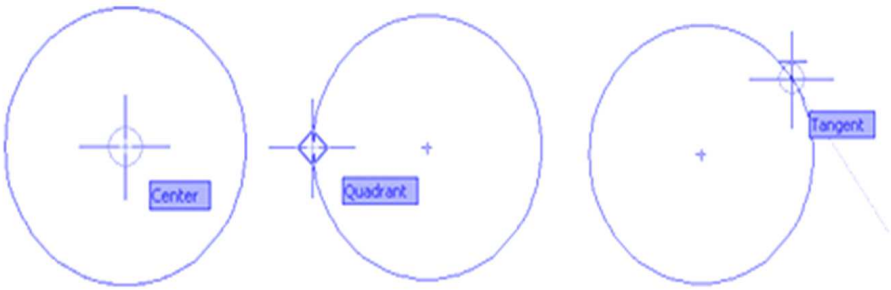
Some of the Object Snaps related to circles and arcs are:

■ **Center**: To catch the Center of an arc or circle

■ **Quadrant:** To catch the Quadrant of an arc or circle

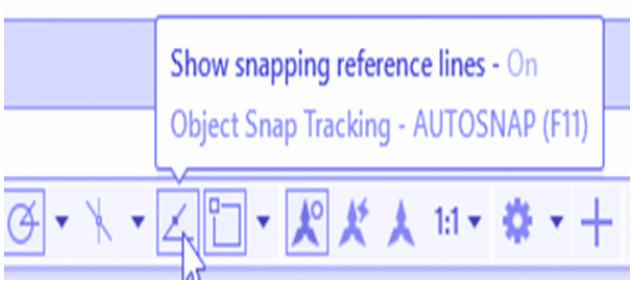
■ **Tangent**: To catch the Tangent of an arc or circle

Here are graphical representations of each one of these OSNAPs:



3.5.3 Using Object Snap Tracking with OSNAP

Sometimes OSNAP alone is not enough to specify desired points, especially if we need complex points. To solve this problem in the past we used to draw dummy objects to help us specify complex points, like in the case when you want to specify the center of the circle at the center of a rectangle. We used to draw a line from the midpoints of the two vertical lines, and the same for the horizontal lines. But since the introduction of Object Snap Tracking, or OTRACK, in Au -to CAD 2000, the drawing of dummy objects diminished. OTRACK depends on active OSNAP modes, which means if you want to use the midpoint, OTRACK requires that you switch the midpoint on first. To activate OTRACK go to the status bar and click the **Object Snap Track king** button on:

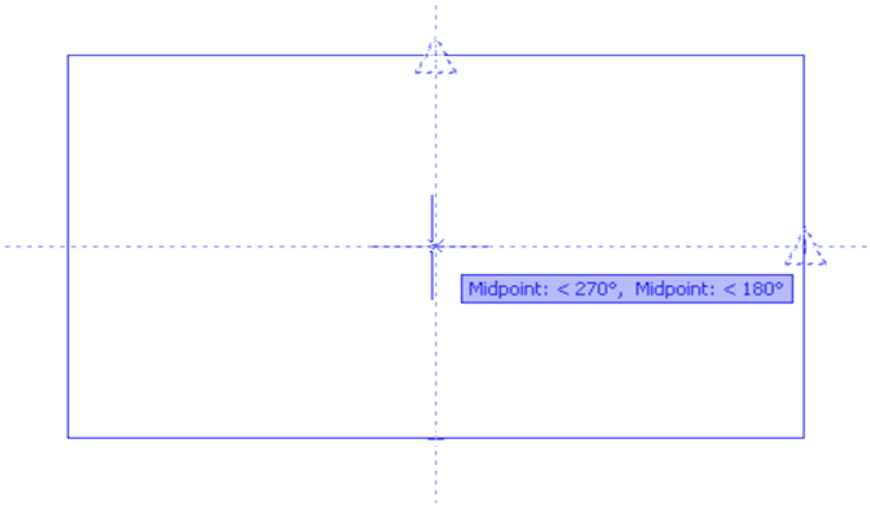


The procedure is very simple:

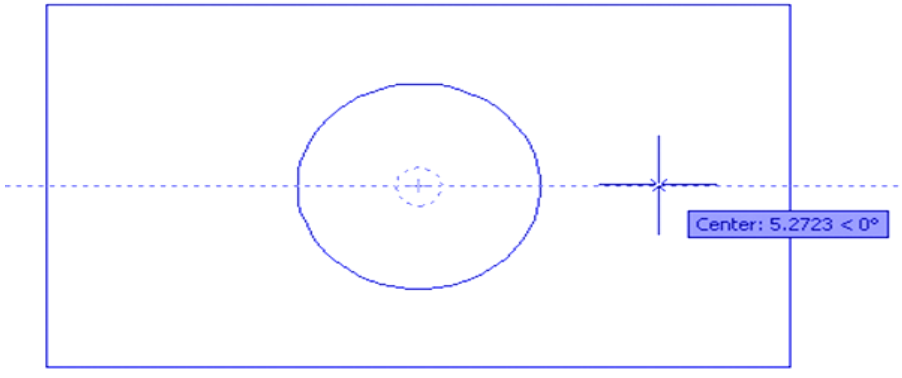
■ Using OSNAP go to the desired point and stand still for a couple of seconds (don't click), then move to the right or left (also up and down depending on the next point), and you will see an infinite line extending in both directions (this line will be horizontal or vertical depending on your movement).

■ If you want to use a single point to specify your desired point, move to the needed direction, type in the desired distance, and press [Enter].

■ If you need two points, go the next point, and stay for a couple of seconds, then make the movement toward the desired direction. Another infinite line will appear. Go to the intersection point of the two infinite lines, and that will be your point.

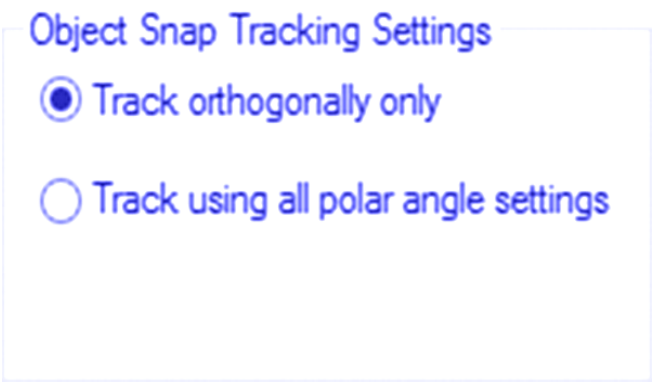


Two points Otrack



Single point Otrack

Polar command will have a major contribution here as well. Let's go back to the same dialog box:



Under **Object Snap Tracking Settings**, there are two choices:

■ Track orthogonally only (default option)

■ Track using all polar tracking settings

This means you can use the current Polar angles (increment and additional angles) to specify points using OTRACK.

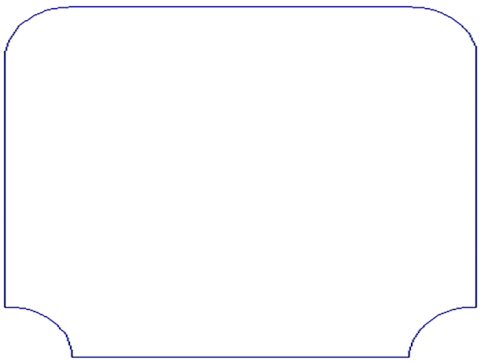
NOTE *- To deactivate an OTRACK point, stay at the same point again for a couple of seconds, and it will be deactivated.*

PRACTICE 3-4 DRAWING USING OSNAP AND OTRACK

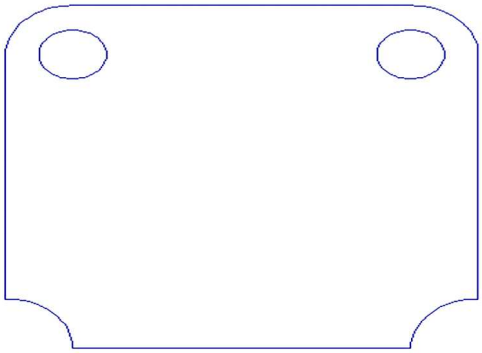
1. Start AutoCAD 2019

2.Open Practice 2-4a.dwg

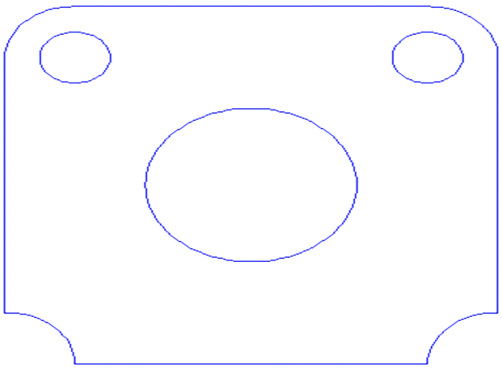
3.Using the proper OSNAP and OTRACK settings, create the four arcs as shown below:



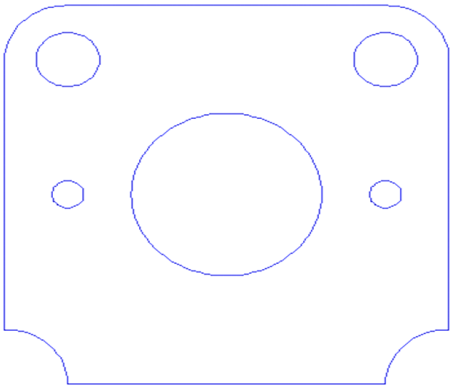
4.Create the two circles as shown below (Radius = 1)



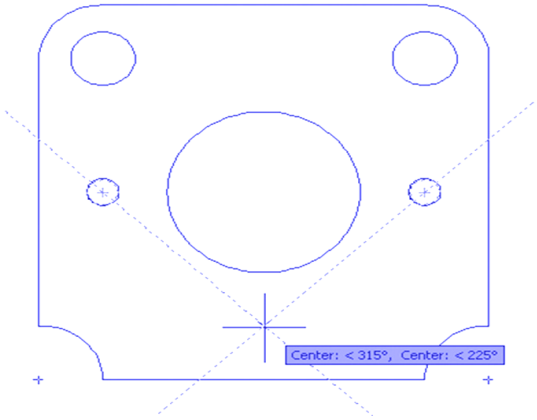
5.Using OSNAP and OTRACK (using two points), draw the circle at the center of the shape (Radius = 3.0)



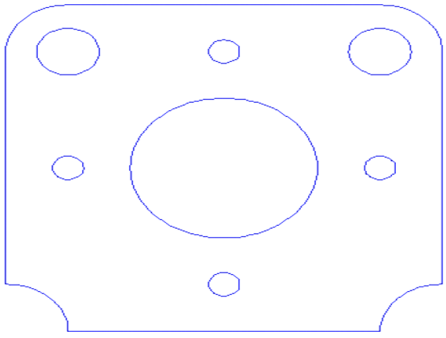
6.Using OSNAP and OTRACK (one point) draw the two circles at the right and left (distance center-to-center = 5.0, and Radius = 0.5)



7.Change the increment angle in the Polar Tracking dialog box to 45. Make sure that Track using all polar angle settings is on, draw a circle (Radius = 0.5), its center specified using OSNAP, OTRACK, and polar tracking as shown below:



8.Do the same procedure to draw a circle at the top, to end up with the final shape as follows:



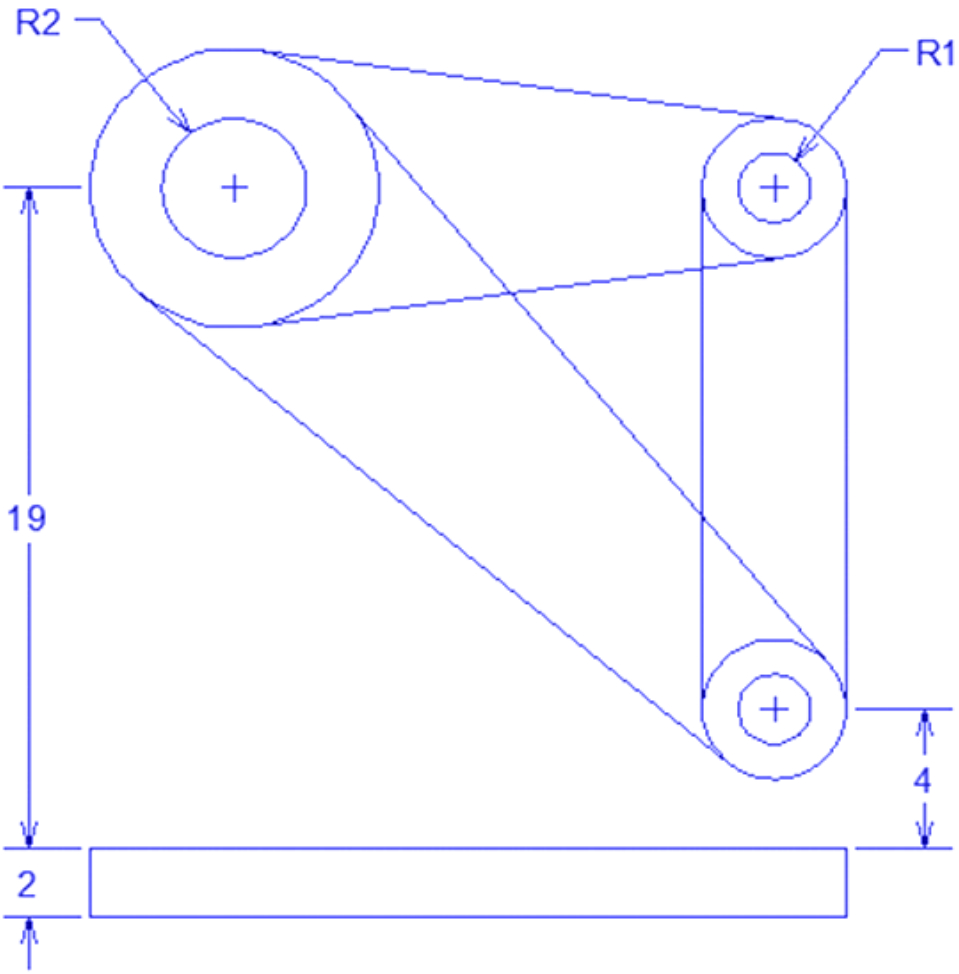
9.Save and close the file.

PRACTICE 3-4 DRAWING USING OSNAP AND OTRACK

1.Start AutoCAD 2019

2.Open Practice 2-4b.dwg

3.Using the proper OSNAP and OTRACK settings, add lines and circles to make the shape look like the following:



4.Save and close the file.

3.6 Drawing Lines and Arcs Using

POLYLINE COMMAND

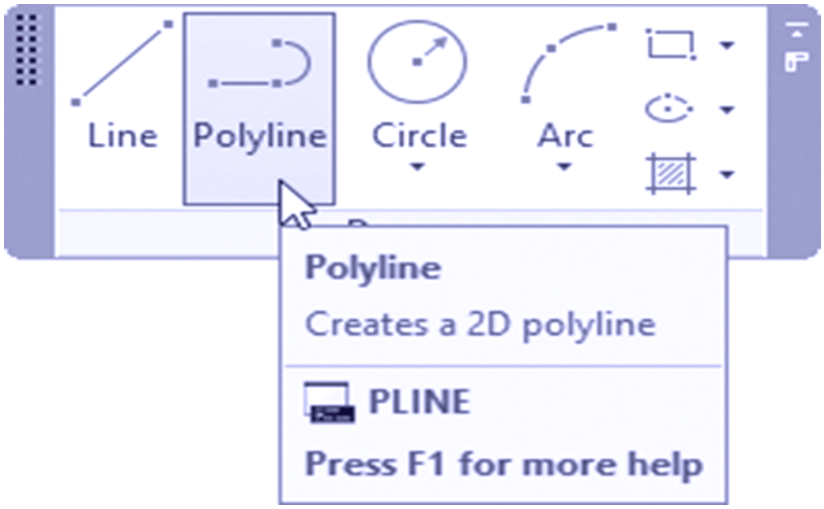
Polyline command will do all or any of the following:

■Draw both line segments and arc segments

■Draw a single object in the same command rather than drawing segments of lines and arcs, like in Line and Arc commands

■Draw lines and arcs with starting and ending widths

To use the command, go to the Home tab, locate the Draw panel, then select the Polyline button:



The following prompt will appear:

Specify start point: Current line-width is 1.0000

Specify next point or [Arc/Half width/Length/Undo/ Width]:

AutoCAD will ask you to specify the first point, and when you do, AutoCAD will report to you the current line-width; if you like it, continue specifying points using the same method we learned in the Line command. If not, change the width as a first step by typing the letter W, or right-clicking and selecting the Width option, which will bring up the following prompt:

Specify starting width <1.0000>:

Specify ending width <1.0000>:

Specify the starting width, press [Enter], and then specify the ending width. The next time you use the same file, AutoCAD will report these values for you when you issue the Polyline

command. The process for Half width is the same, but instead of specifying the full width, you specify half width.

The Undo and Close options are identical to the ones at the Line command.

Length will specify the length of the line using the angle of the last segment.

Arc will draw an arc attached to the line segment; you will see the following prompt:

Specify endpoint of arcor[Angle/CEnter/CLose/Direction/Half width/Line/Radius/Second pt/Undo/Width]:

The arc will be attached to the last segment of the line, or will be the first object in a Polyline command; using either method, the first point of the arc is already known, so we need two more pieces. AutoCAD will make an assumption (which you have the right to reject): AutoCAD will assume that the angle of the last line segment will be considered the direction (tangent) of the arc. If you accept this assumption, you should specify the endpoint. If not, choose from the following to specify the second piece of information:

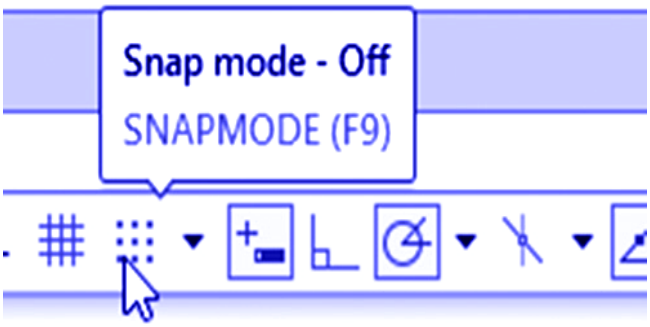
- The Angle of the arc
- The Center point of the arc
- Another Direction to the arc
- The Radius of the arc
- The Second point which can be any point on the parameter of the arc

Based on the information selected as the second point, AutoCAD will ask you to supply the third piece of information, but by all means it will not be out of the eight we discussed before.

NOTE- Normally the user should think counterclockwise when specifying points; on the other hand, if you want to work clockwise, simply hold the [Ctrl] key and it will change.

3.6.1using snap and grid to specify points accurately

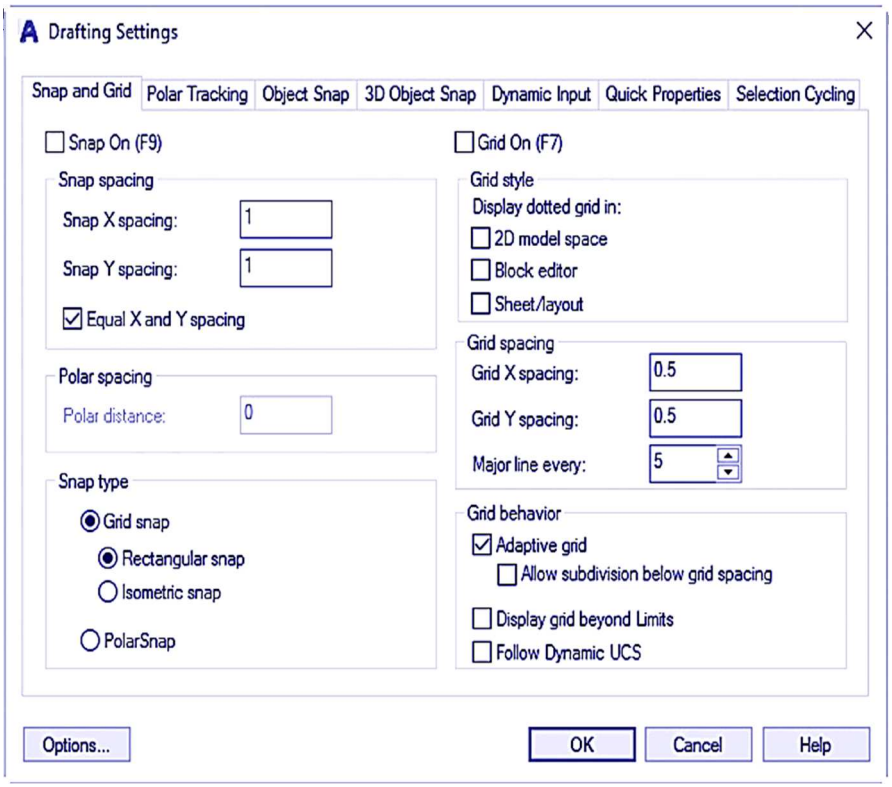
Snap and Grid will be the third method to help us specify points accurately in the XY plane. By default, the mouse is not accurate, so we can't depend on it to specify points. We need to control its movement, which is the sole function of Snap. Snap can control the mouse to jump in the X and Y plane with exact distances. The grid by itself is not an accurate tool, but it will complement the Snap function. It will show horizontal and vertical lines replicating the drawing sheets. In order to turn Snap on/off, use the following button in the status bar:



: In order to turn the Grid on/off, use the following button in the status bar

Most likely, switching both on will not help; you will need to modify the settings to set your own requirements. Using the small triangle at the right of the Snap button, you will see the following:

Select the Snap Settings option; the Drafting Settings dialog box will pop up:



Input the Snap X Spacing and Snap Y Spacing (by default they are equal). Switch off the check-box to make them unequal. Do the same for the Grid Spacing in X and Y. If you want the Grid to follow Snap, set the Grid spacing to zeros. In the Grid there are major and minor lines; set the major line frequency.

Set if you want to see the Grid in dots (as it was before AutoCAD 2011) and where (2D Model space, Block editor, or Sheet layout).

Grid behavior is for 3D only.

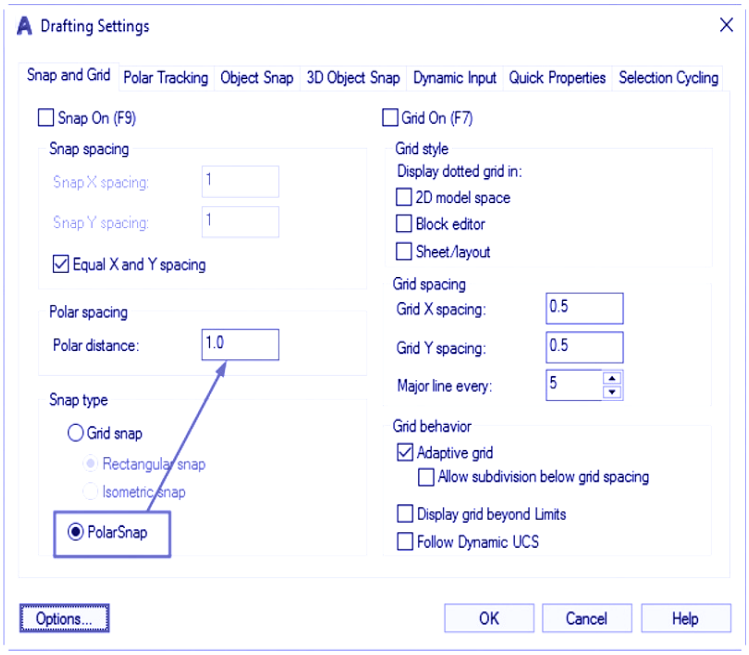
Specify the type of Snap: Grid Snap or Polar Snap.

The user can use function keys to turn on/off both Snap and Grid:

- F9 = Snap on/off
- F7 = Grid on/off

3.6.2using Polar Snap

While using the Snap command, the mouse will use the increment distance to specify the exact distance only in horizontal and vertical directions. While going diagonally, Snap will not help you. To solve this problem AutoCAD provided the concept of Polar Snap, which will specify exact increments using all the angles specified in the Polar Tracking dialog box. To activate the Polar Snap, AutoCAD will switch off the normal Snap (which is called Grid Snap). To do that go to the Snap button, click the triangle at the right of the button, and select the Snap Settings option, which will bring up the following dialog box:



Select the Polar Snap button, then specify the the Polar distance as shown in the above illustration.

3.6.3 SNAP AND GRID

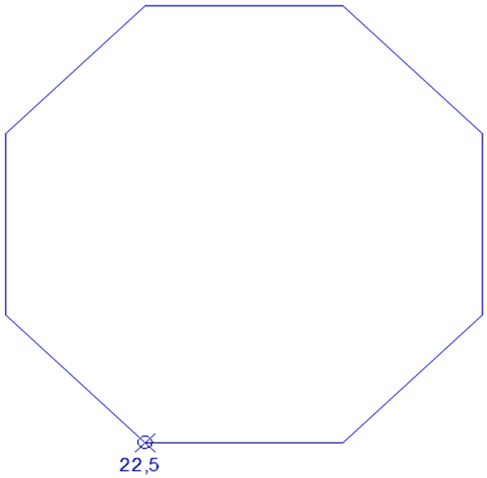
1.Start AutoCAD 2019

2.Open Practice 2-6.dwg

3.Change Polar Tracking to use angle = 45

4.Change the Snap to Polar Snap and set the distance to 0.5

5.Draw the following shape, bearing in mind to start from point 22,5, make all segment lengths 6.5, and use an angle of 45:



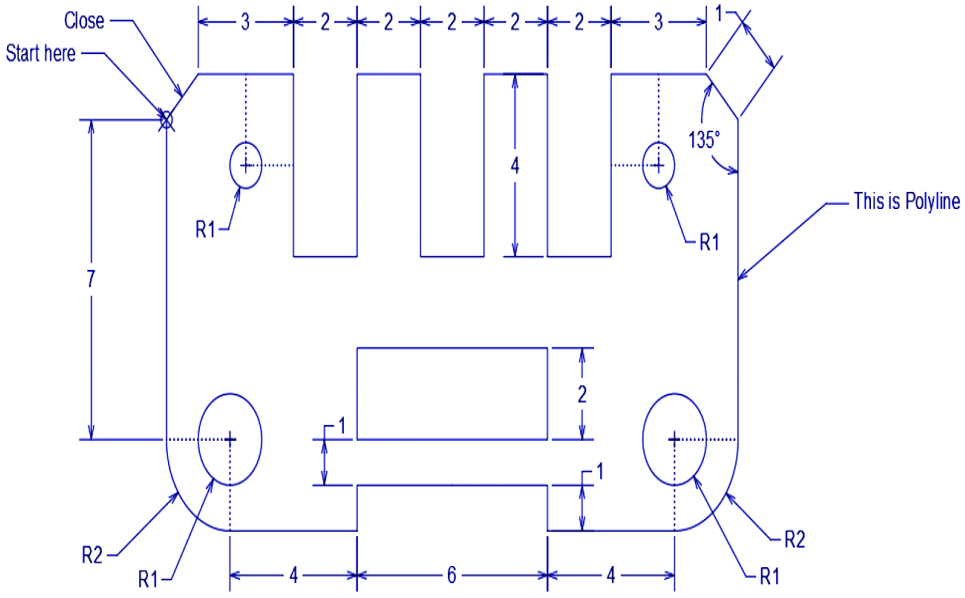
6.Save and close the file.

PRACTICE 3-5 SMALL PROJECT

1.Start AutoCAD 2019

2.Start a new drawing using acad.dwt

3.Using all the commands and techniques you learned in Chapter 2, draw the following drawing (without dimensions) starting from any point you wish:



4.Save and close the file.