SOURCECAD

Learn AutoCAD basics in 10 days

Learn to make your first AutoCAD drawing with this guide

OVERVIEW

For a beginner in AutoCAD, this software might seem very intimidating with lots of tools, commands, and features to learn and thousands of books, video courses and lectures to choose from for learning the software. To ease this confusion of a beginner AutoCAD user I have prepared this concise beginner guide which will help you in focusing your learning on only the essential tools and commands that will make you up and running with AutoCAD in no time.

The complete learning guide is divided into 10 days with each day containing one or more of the AutoCAD topics and commands. I recommend you go through the learning guide in the order without skipping any day.

By the end of this 10 days guide you will be able to make your own AutoCAD drawings and modify the existing drawings too. The last day of this guide is also dedicated to practicing a sample drawin and we will use the tools and commands discussed in this guide to make the sample drawing.

I hope you will find this guide helpful in your journey to learning AutoCAD. For any questions reach me on admin@thesourcecad.com

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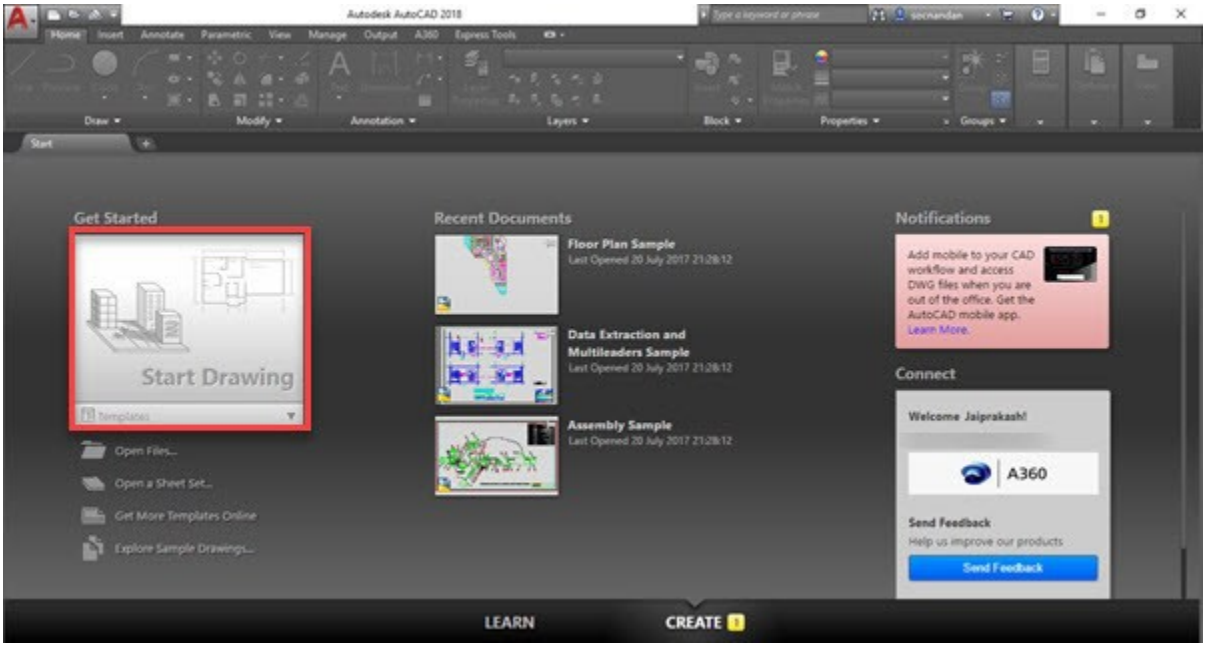
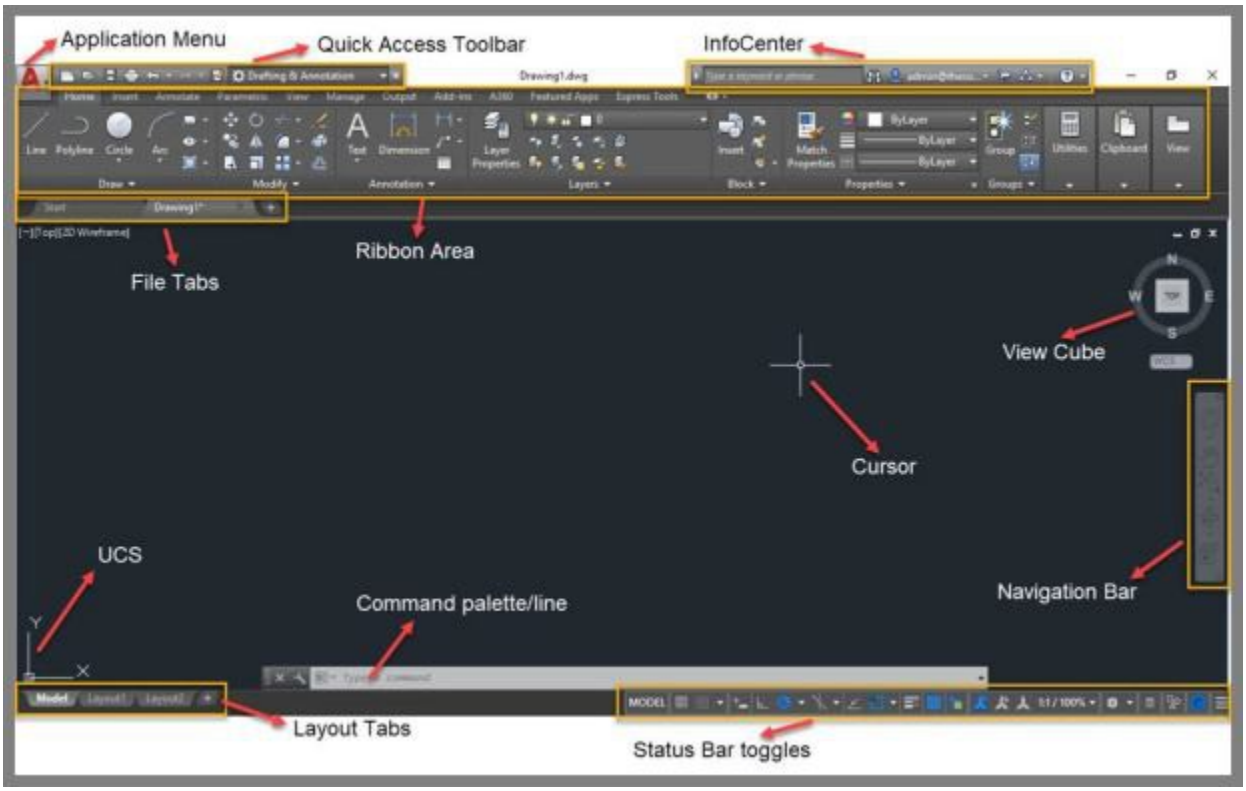
Day 1: Familiarizing with the user interface

When AutoCAD starts for the first time it will show you the start screen as shown in the image on the left. This screen is also called the start screen and it has the template box on the left shown in red box as well as a list of recently opened drawings on the next columns. To launch the drawing area simply click on the big start box marked in the red box.



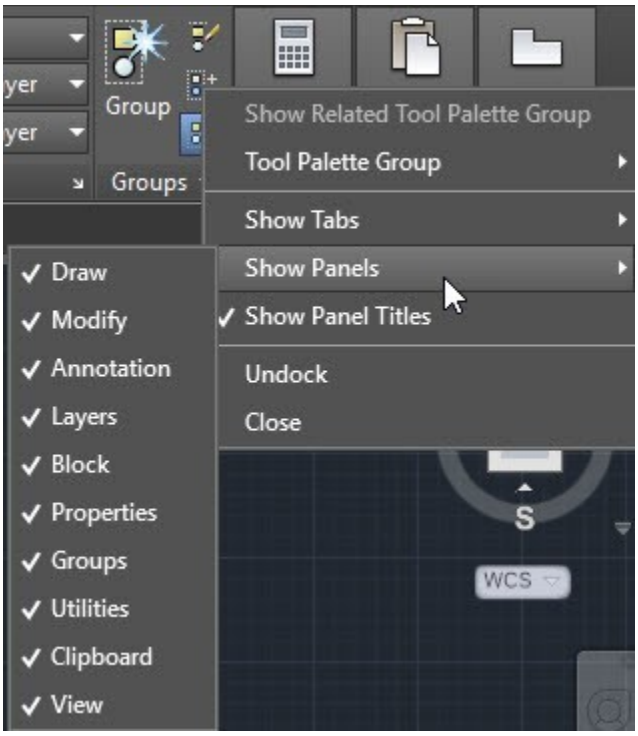
The AutoCAD interface will open. On the top left of this interface, you will see AutoCAD 2018 Logo which is also a drop-down menu called application menu. This menu contains many frequently used tools like open, save and print. On the right of that application button, you will see quick access toolbar which again has some frequently used tools.

Below the quick access toolbar, you will see AutoCAD ribbon with many tabs which are grouped in different categories like Home, Insert and Annotate. These tabs are further grouped into panels which contain commands of a similar category. For example, Home tab has all commands related to drawing features grouped into Draw panel, Modification of drawing related features are grouped into Modify panel and so on.



You can remove these tabs or panels and you can also bring more tabs and panel in the interface which are hidden by default. To add/remove tabs right-click in the empty area of any tab and select Show Tabs then the name of the tab from the right contextual menu.

You can activate any tab by clicking on its name and then all the panels of that particular will show upon the top. Here also you can right-click in the empty area of the tab and then select Show Panels and name of the panel to activate and deactivate it as shown in the image below.

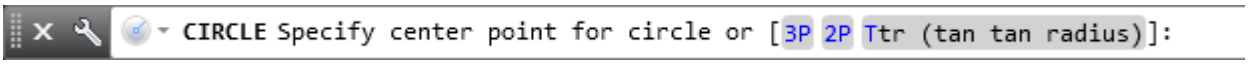


Below this ribbon area, you will see file tabs with the name of open drawing. By default, you will see the start and drawing 1 tab as shown in the image below.

The large canvas area with a grid overlay is the most obvious feature of AutoCAD interface and it is the place where all the magic happens. This drawing area will be used to create your drawings and make modifications to it.

The drawing area has view cube on the top right side and a navigation bar with Navigation-related tools under View Cube.

On the bottom of this drawing area, you will see the command line or the command bar which will show currently active command, its subcommands and history of all the commands entered so far.



On the bottom left of drawing area you will see UCS icon with X and Y axis visible in 2D space and if you are in 3D space X, Y and Z axis will be visible. Below drawing area, you will see the status bar which has layout tabs and status bar toggles which are very important tools and they will help you in making drawings precisely.

Understanding Mouse Operations

To use AutoCAD effectively you need to use a 3-button mouse with left, right button, and a middle mouse wheel with the button. Although you can use other navigation devices also and that depends completely on your preference.

The standard windows mouse operations are used in AutoCAD interface also. You can left click to select any object or tool and right click to open a context-sensitive menu also called contextual menu. The contents of this menu will depend on the place where you click on the AutoCAD interface.

If you press and hold middle mouse wheel and then move your mouse you will be able to pan your entire drawing in the AutoCAD drawing area. By rotating the mouse wheel, you can zoom in or zoom out different parts of the drawing. The place where you place your cursor will become center of zoom for the drawing.

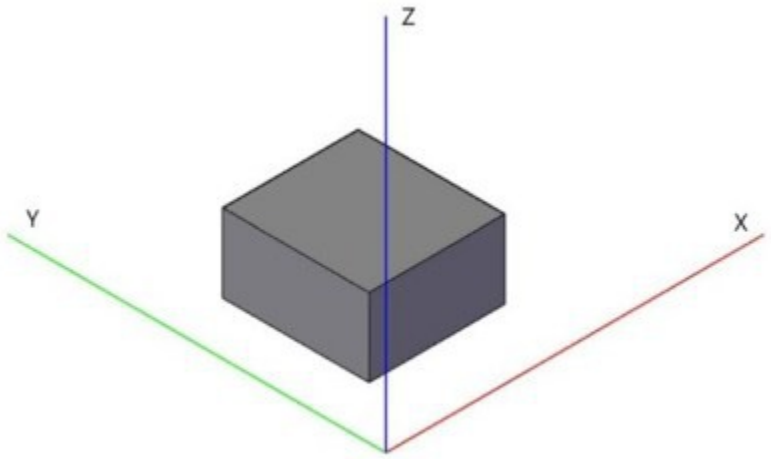
Day 2: Understanding coordinate system

Understanding coordinate system is essential to understanding the way AutoCAD works. In AutoCAD, you can assign length, angles as well as coordinate values to make the required geometries. Primarily there are two types of the coordinate system which we will use to make geometries in AutoCAD and they are Cartesian and Polar.

Cartesian coordinates

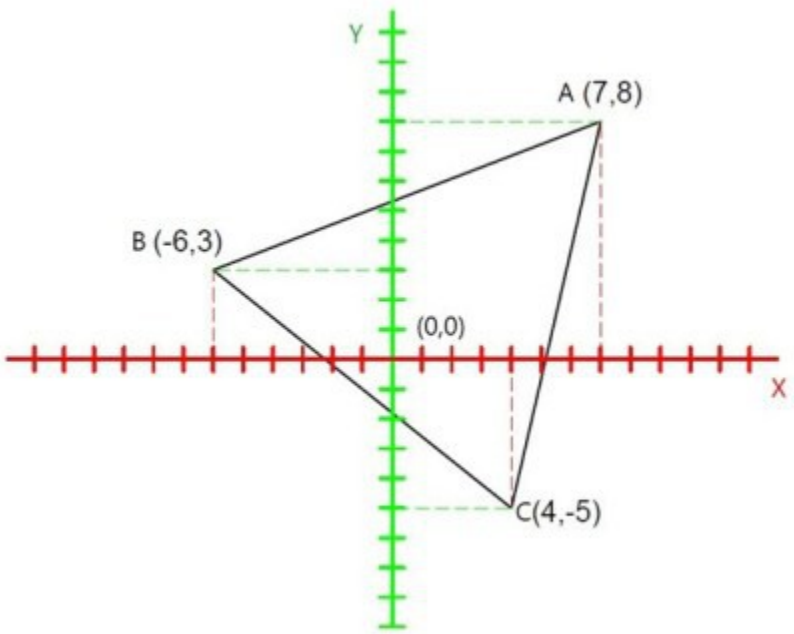
AutoCAD follows simple Cartesian coordinate system which is a graphical method of assigning coordinates to a point in space. The simple 3D space has three coordinates namely X, Y and Z

which are mutually perpendicular to each other as shown in the image below. In these cases, the point of intersection of the three mutually perpendicular axes is origin represented as (0,0,0).



The position of any point in 3D space can be specified using these three axes which are represented with Red, Green, and Blue axes here. But for 2D space, we only need to use X and Y axes to define the position of any point.

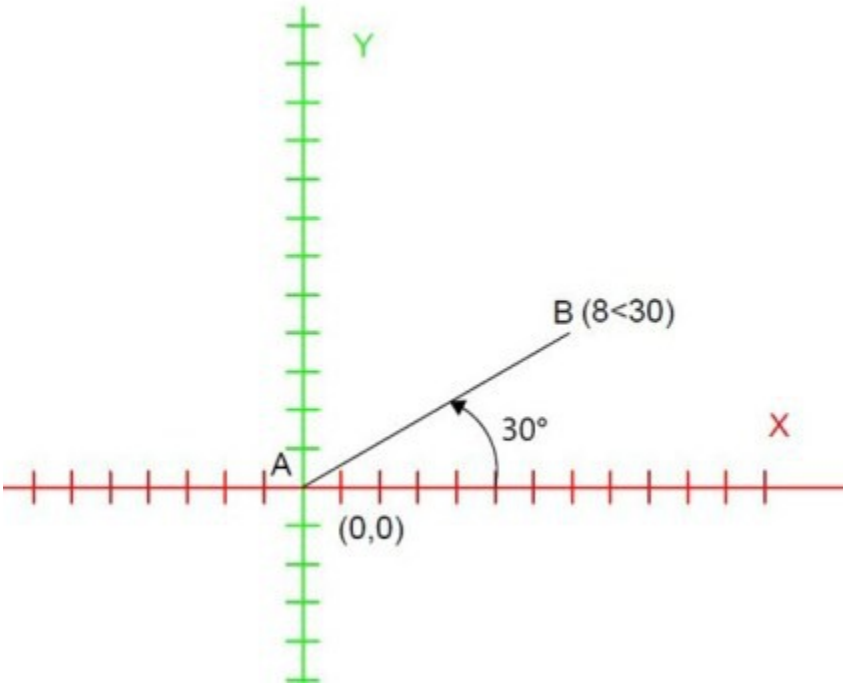
In 2D space the simple X,Y coordinate system is used and any point in 2D space can be defined using these two coordinates only. Take the example of the image shown below. Here the origin is mentioned as 0,0 which is also the point of intersection of the X and Y Axes represented by red and green axes respectively.



The point A (7,8) is at 7 units from the origin along the X-axis and at 8 units along the Y-axis. Similarly, the point B (-6,3) is at 6 units along the negative side of X-axis and at 3 unit along the positive side of Y-axis. In case of point C (4,-5) the distance from the positive side of X-axis is 4 units and its distance along the negative side of Y-axis is 5 units.

Polar coordinates

Using polar coordinates, we can also represent points in the 2D space but in this case, one polar distance and an angle with respect to X-axis are required instead of X and Y coordinate value. To explain it clearly, I will use the example shown below.

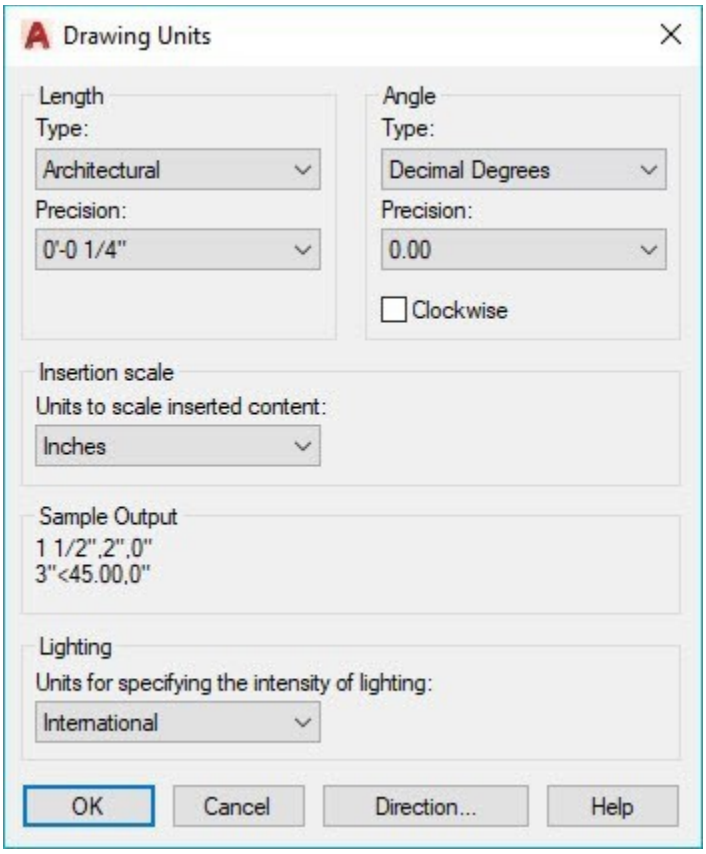


In this case, point B is represented with (8<30) where 8 is the distance between point A and point B where A is the origin and 30 is the angle between line AB and the positive X-axis along the anticlockwise direction.

This type of coordinate representation can also be used to specify the position of a point in 2D space and it's generally referred as the polar coordinate system.

Day 3: Setting drawing units

AutoCAD starts drawing with the unit set in the default template file but you can change this unit to your preferred settings using Units window. To open the Unit window type UN on the command line and press enter. The Unit window with all the default values will show up just like the image shown below.



In this window, let's start with the first Length panel. Change the length type to decimal if you want to use metric systems and change it to Architectural if you want to use Imperial system of feet and inches. In the next drop-down menu set the precision value of deciaml places.

In the Insertion Scale set the unit to millimeter or meter if you are using deciaml and Inches if you are using the imperial system. In the angle type, you can set your prefered value like decimal degrees or degree minute second from the drop down menu.

Once all of these changes are made, click on OK to accept the chanes and exit the Units window. You now have your preoper units set. You can also refer to this related article to learn more about changing units in AutoCAD.

AutoCAD cursor modes

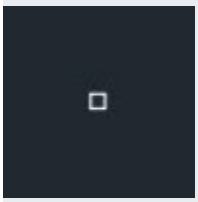
AutoCAD cursor behaves in different ways when the different type of commands are active. These cursor modes help you in making the selection easily and it also helps you in identifying whether any command is active or not.

Normal Cursor

In this case, the cursor will look like two perpendicular lines with a square box at the point of intersection. This is visible when no command is active.

Point selection cursor

In this case, the cursor will look like two perpendicular intersecting lines. It is visible for commands that require selection of point.



Pickbox type cursor

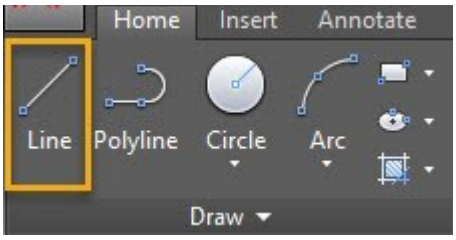
In this case, the cursor will look like a small square box. It is visible for commands that require selection of objects.

Day 4: Making simple lines

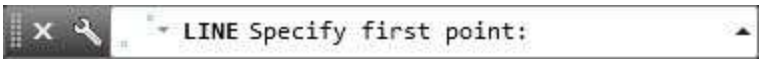
Using Line command

In AutoCAD, you can make drawings by direct distance entry or by entering coordinate values. To start making the drawing click on any draw tool icon from ribbon panel or you can use its command and press enter to start the command.

To start the Line command simply click on the Line tool from Draw panel of Home tab or use its command L.



You will notice that cursor will change into a point selection cursor with two perpendicular lines and command line will show the name of command along with the prompt.



Where is the command line?

In case you are not able to see the command line/bar at the bottom of the drawing area, don't panic simply press CTRL + 9 key to make it visible or to hide it.

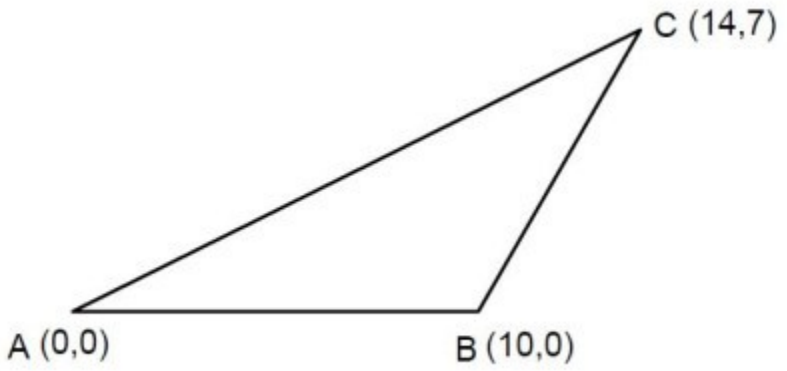
Now AutoCAD is ready for your input to make the line. Click at any point in the drawing area and the line will start from that point. Move your cursor and you will notice that the line will follow the movement of cursor and it will stretch with the cursor too. This line is also called rubber bending line which follows your cursor.

Click at the second point in the drawing area and the fixed segment of the line will be made and the rubber bending line will again follow from the last point where you clicked. Repeat the process to make additional lines and when you are done making the geometry press enter or esc key to exit the command.

In the example above you simply clicked at different points in the drawing area to make the line but instead of clicking at different points you can also enter the coordinate values of the point which we will see in the next example.

Using absolute coordinates

Let's take the example of this triangle shown below. In this case, we have all the three coordinates of this triangle mentioned as point A, B, and C. You can make this geometry using the Line command.

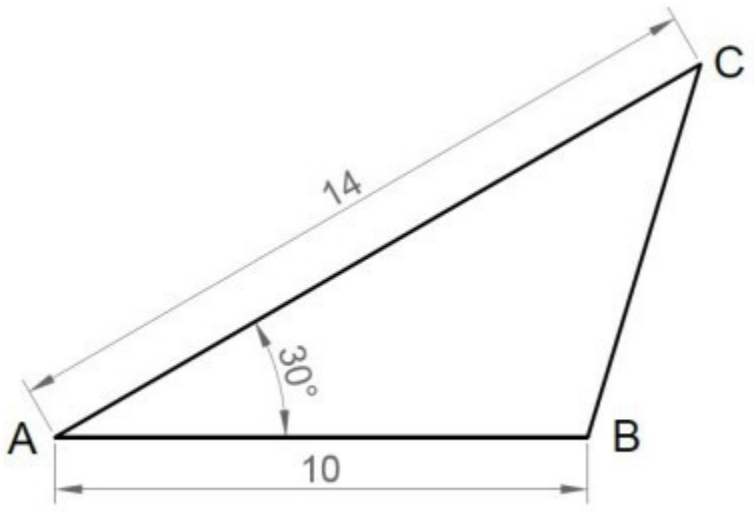


1. Open the AutoCAD window and select the Line tool from the Draw panel or use its command L to start the command.
2. The command line will prompt you to specify the first point for the line. Type 0,0 for the first coordinate point which is also the point A and press enter.
3. Now we need to specify the coordinates of the second point or point B, type 10,0 and press enter again.
4. Once again, we need to specify the coordinates of the next point C, so type 14,7 and press enter.
5. Now our cursor is at point C and you can simply type 0,0 and press enter to return to the first point which is the origin. To exit the command press, enter or esc key once.

As you can see that we only required the coordinates for making this triangle but the coordinates are rarely used for making real-life drawings and in most of the cases length and angle values are predominantly used. In this next example, we will learn to make the similar type of rectangle but instead of coordinates, we will use direct distance entry method.

Using direct distance entry

To explain this feature, I will use the triangle shown in the image below.

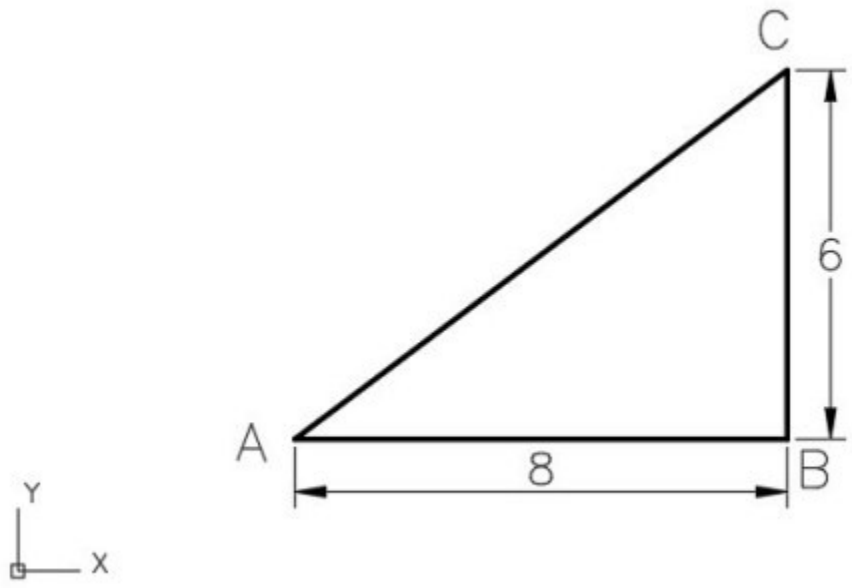


1. Open a blank drawing and start the line command by clicking on the Line tool on Draw panel or by using the L command.
2. Click at a point in the drawing area to start the rubber bending line and move your cursor towards right side type 10 the command line and press enter.
3. Press enter again to exit the command.
4. Click again at the starting point of the line (point A) and type <30 and press enter, notice the < angle sign before 30. In this case, entering the angle sign before 30 will tell AutoCAD to take the numeric value as angle and not distance. Once you press enter you will notice that the line will be locked at an angle of 30 degrees with respect to the positive side of X-axis.
5. Move your cursor in the direction of the 30-degree line and type 14 and press enter again. This will make a line at an angle of 30 degrees with length 14 units.
6. Click at the point B as shown in the image above and press enter again to exit the command.

In this case, you saw that a geometry can also be made by entering values of distance and angle directly on the command line. This method is relatively easy when compared with the coordinate entry method, this is also the most common way of making drawings in AutoCAD.

Using Relative coordinates

In this next example I will explain the method of relative coordinates and using relative coordinates we will make the triangle shown below.

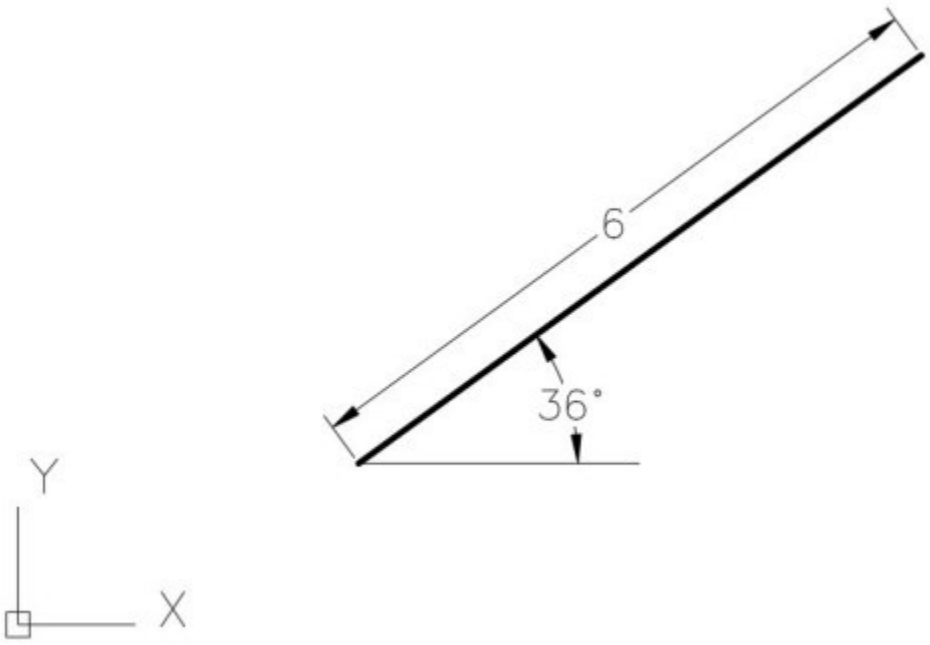


In this case, you can start the drawing at point A and then progressively make your drawing by entering distances. But you can also use relative coordinates as explained in the steps below.

1. Let's start by selecting the line command and then click at a point in the drawing area, that will be the point A of the drawing. In this case point, A is not the origin and origin is somewhere else in the drawing area so we don't know the exact coordinates of the point B. But you can use relative coordinates and assign coordinate values of point B with respect to coordinate values of point A.
2. In this case, if point A would be origin then point B should be 8,0. To make the horizontal line AB again select the line command and click at any point that will be point A and then type @8,0 for point B. Notice the @ sign before the coordinate value, this @ is added to indicate that the next coordinates are relative with respect to the point where we have previously clicked i.e. point A and it will assume point A as origin instead of absolute origin.
3. Once you have reached point B don't exit the line command and type @0,6 and press enter. You will notice that AutoCAD will reach point C and in this case also the point B will be treated as the origin and the coordinate values of point C with respect to point B is 0,6 which is shown with the @ sign.
4. You can complete the triangle by clicking again at the point A and then press enter to exit the line command.

Using Polar coordinates

Using polar coordinates, you can add distance as well as angle values directly to the command line and they need not be entered separately. For explaining this, I will use this drawing.



In this case, we need to draw the line which is inclined at an angle of 36 degrees with respect to the positive side of X-axis and it has a length of 6 units.

1. Type L↵ and press enter to start the line command.
2. Click at any point in the drawing area to specify the first point of the line.
3. Type @6<36 on the command line and press enter, the line of required value will be made.

In this case we are entering length of line after @ sign and the angle with which it is inclined is mentioned after < sign, hence @6<36 indicates a line of length 6 unit from the point of click and it is inclined at an angle of 36 degrees with respect to the positive side of X-axis.

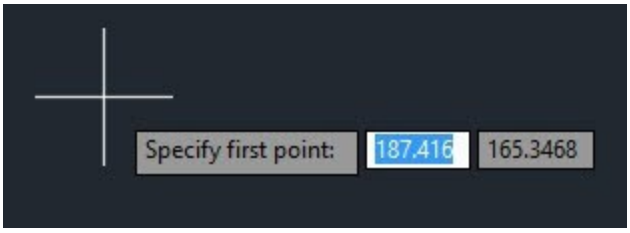
Day 5: Status bar modes

There is another way of making geometries in AutoCAD by simply bypassing all the considerations of absolute or relative coordinate system and it's called Dynamic Input.

To explain the dynamic input tool, I will use image shown in the previous section where we made a line of length 6 unit inclined at an angle of 36 degrees with respect to the positive side of the X-axis.

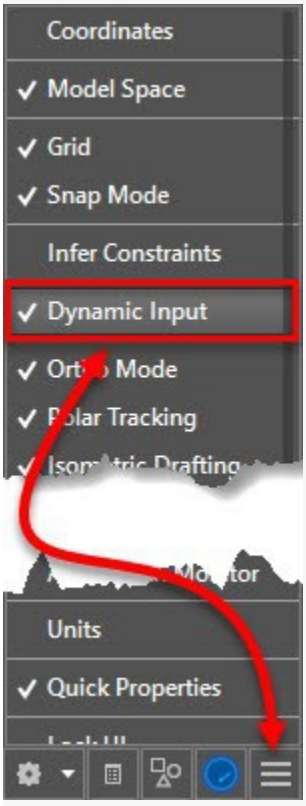
Click on the dynamic input icon of the status bar and it will turn blue when active. You can also type DYNMODE↵ then type 3 and press enter again to activate the dynamic input mode, the default value of DYNMODE system variable is -3.

Now select the line command and now you will notice a tooltip on the cursor with the value of X and Y coordinates of the point as shown in the image below.

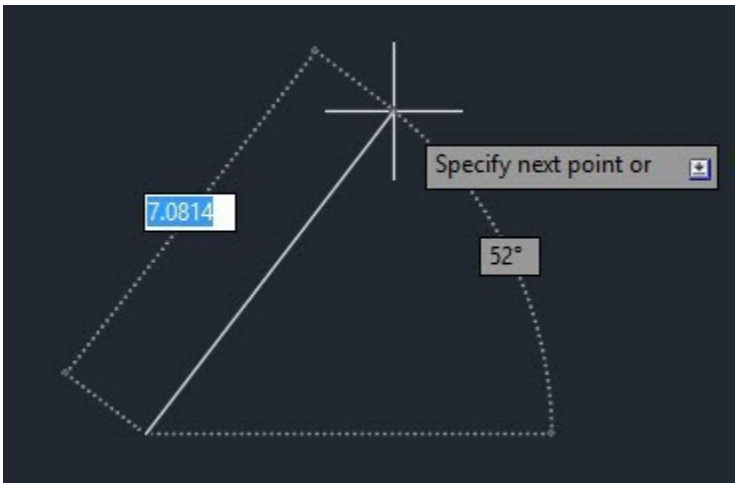


Can’t see the dynamic input icon?

If dynamic input icon is not available on your status bar then click on the customization icon shown as three dashes on the far-right side of the status bar and activate dynamic input from the menu as shown in the image here.



Now click at a point in drawing area and your tooltip on the cursor will change to represent a length and an angle field as shown in the image below.



Here the length field is active and it is ready for your input, type the length of the line in this field and press tab key on the keyboard. You will find that the line will be locked with a length of 6 unit and a lock sign will also appear next to the length field of the tooltip. Now enter the angle without any angle sign which in this case is 36 and press enter.

And our required line is made. So, this is another quick and effective way of making geometries in the drawing. If for any reason you don't want to use this Dynamic Input tool then click on the dynamic input icon again on the status bar to deactivate it.

Using Ortho Mode

So far, we have used line command to make geometries in arbitrary directions but if you want to restrict the direction of your lines in horizontal and vertical directions then you can use Ortho mode. To activate the Ortho mode click on the Orthomode icon the status bar as shown in the image below or press F8 function key on the keyboard.

Select the line tool from the Draw panel or use its command L and click at a point in the drawing area. Now move your cursor around and you will see that the line will remain restricted to horizontal or vertical direction depending upon the movement of your cursor.

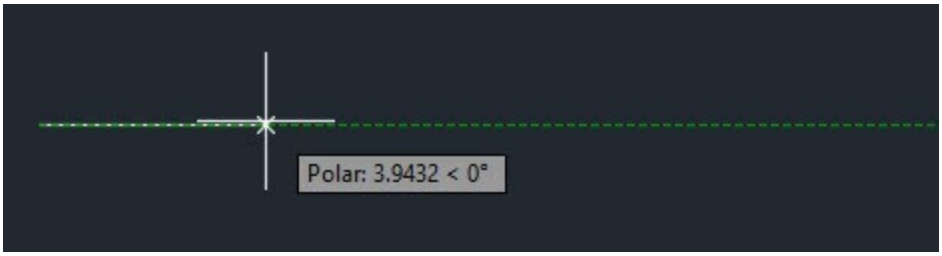
You can click at different points to make the geometry but the lines will always remain horizontal or vertical. To deactivate the Ortho mode simply click on its icon on the status bar again or press F8 function key on the keyboard.

Using Polar Tracking

Polar tracking allows you to make geometries at any angle you want. To active Polar, Tracking click on its icon on the status bar as shown in the image below. You can also activate Polar Tracking by pressing the F10 function key on the keyboard.

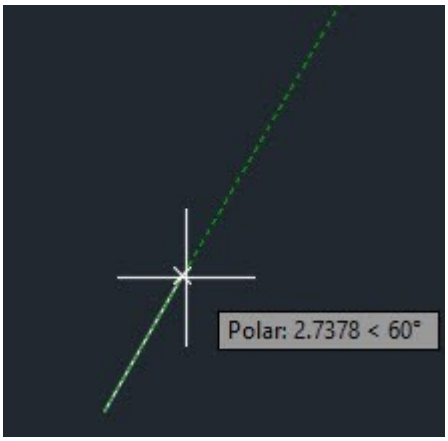
When Polar tracking is active it will automatically deactivate the Ortho Mode.

Start the Line command again by selecting its command L. Click at a point in the drawing area and move your cursor, you will notice a green tracking vector when the cursor is horizontal or vertical and this will help you in restricting the line to a horizontal and vertical direction.



But this is not all, in this case, you can assign a different angle to the polar tracking mode and it will start restricting your lines to those angles. To change the angle of Polar Tracking click on the small arrow right beside the Polar tracking icon and select the angle which you want to choose. By default, 90 and its multiples will be selected but we will change it to 30 and its multiples as shown in the image below.

You can select any other angle value as well if you want from the list. After making the selection move your cursor again in the drawing area and now you will find green tracking vector after an interval of every 30 degrees or at every angle which is a multiple of 30 degrees.

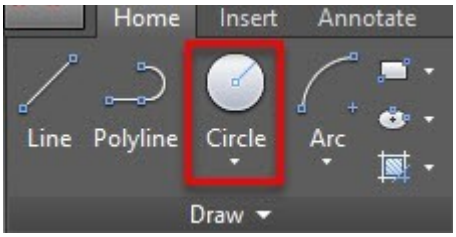


Drawings in AutoCAD are made with geometries like Line, Circle, Arc, and Spline. In this section, we will learn about making these simple geometries. At the beginning of the chapter we have already learned about the line command, here we will start with the circle command.

Day 6: Circle and Arc

Making Circles

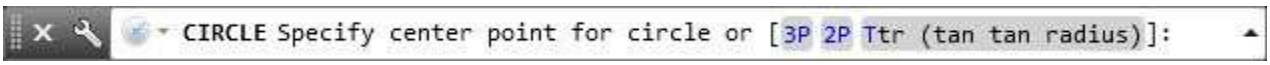
You will find circle command on the Draw panel of Home tab as shown in the image below.



If you click on the arrow right below the circle icon you will see a list of six different options which can be used to make the circle. I will start the first one which is Center, Radius.

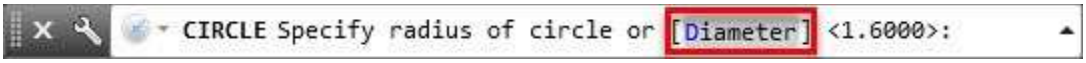
Click on the Center, Radius icon to the command or you can also use its command C.

Now the command line will show a prompt



Click in the drawing area to specify the center point or you can also enter the coordinates of the point. For this example, I will click at a point in the drawing area.

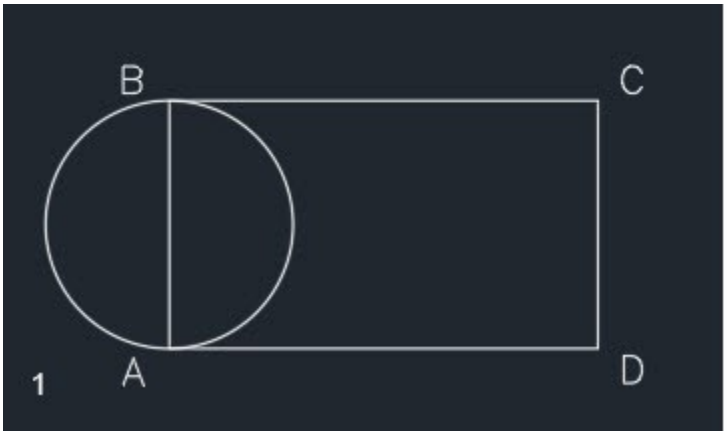
Now command line will prompt you to specify the radius of the circle. Type the value of radius which you want to use for this circle and press enter. If you want to specify the diameter value instead of the radius then click on the diameter text on the command line or type D and press enter.

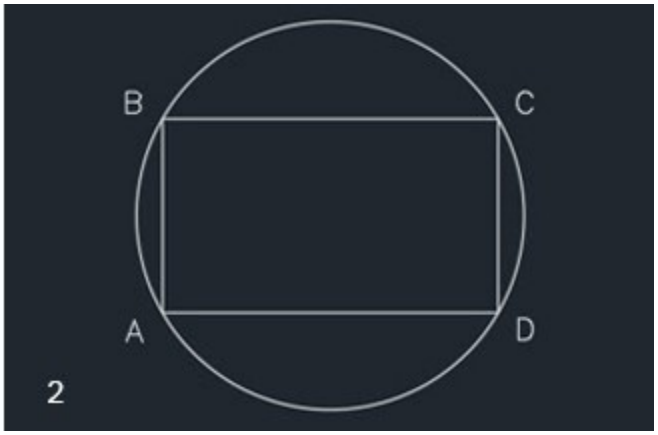


Command line will now prompt you for the diameter value instead of the radius. Enter the value and press enter to make the circle.

There are also other options like Two point and Three-point circles which work in a different way.

Here I have used the two-point circle to make it on the side AB of the rectangle which has length 20 and width 10 units. To make the circle with the two-point option I have first clicked on the point A and then I clicked on the point B of image 1 shown below.

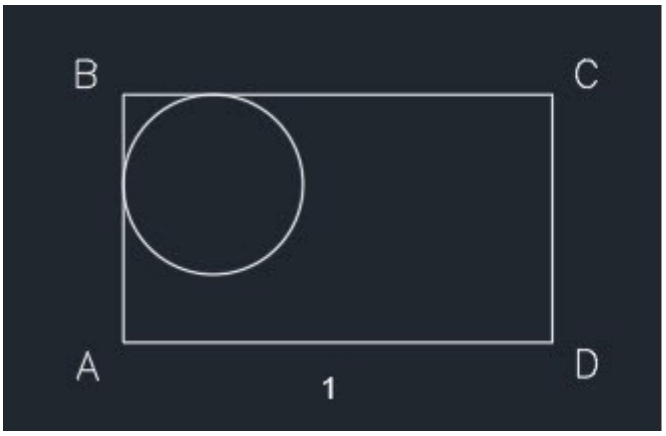


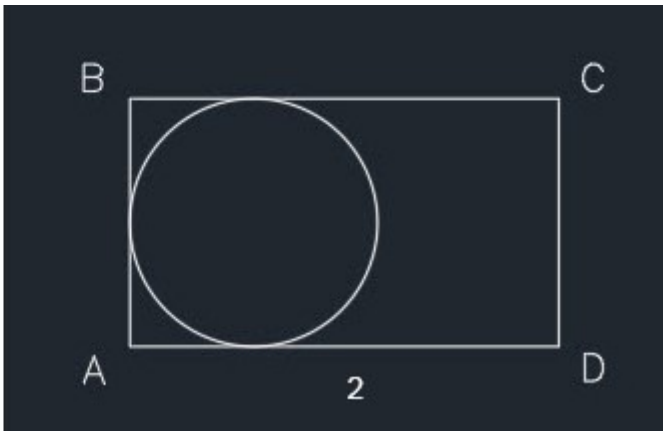


In this case, AB is the diameter of the circle. Similarly, you can make a circle with the three-point option. In this case, the circle will pass through all those three selected points. The circle in the second case is made with the three-point option, here I clicked on point A, B, and C to make the circle.

You can also make a circle which is Tangent to two geometries and has a specific value of radius. Select the Tangent, Tangent, Radius option from the circle flyout. Click anywhere on the line AB then click on line BC. Command line will prompt you to specify a radius value, type 4 on the command line and press enter. The circle as shown in the image 1 below will be made.

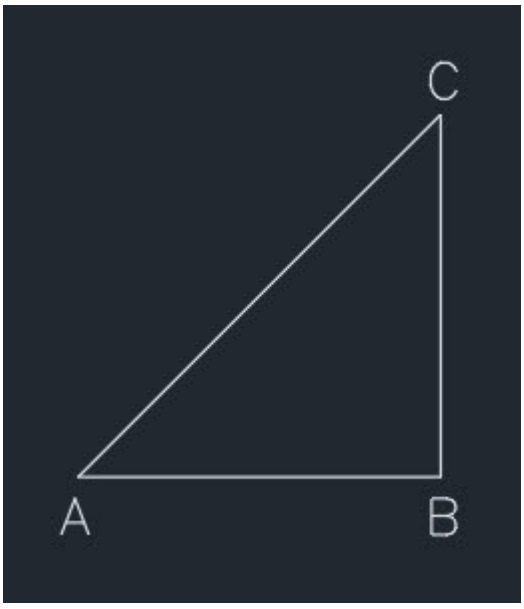
To make the circle which is tangent on three different geometries simply use the Tangent, Tangent, Tangent option of the circle flyout and click on any three geometries. In this case, you don’t need to specify the radius of the circle. In the image 2 shown below the circle has been made with Tangent, Tangent, Tangent option and it is tangent on line AB, BC and AD.



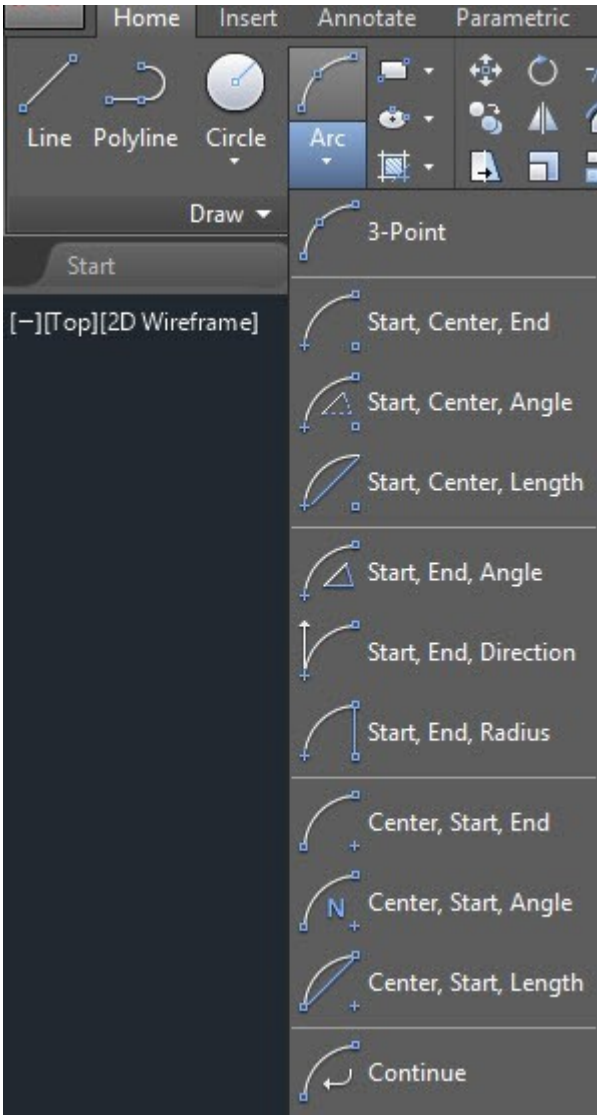


Making Arc

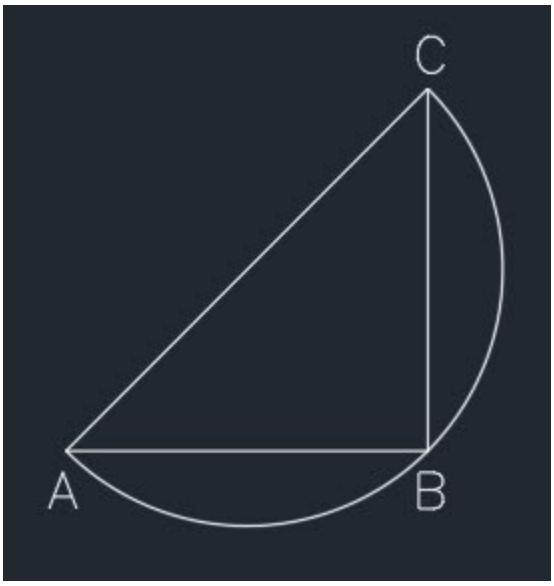
An Arc is a segment of a circle and there are lots of ways with which you can make it in AutoCAD. The method which you use to make the arc depends on the type of geometry which you want to make. In this case, I will explain some of the most frequently used methods of making the arc and I will use this right-angle triangle to explain it.



Select the Arc tool from the Draw panel of Home tab, you can also use its command ARC.



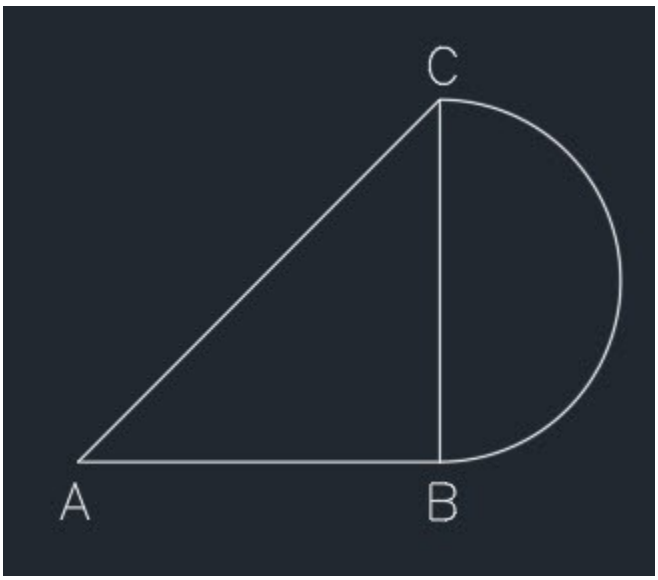
The command line will now prompt you to select the first point of the arc, click on point A of the triangle, then specify the second point as point B and third point as point C. An arc connecting points A, B and C will be formed as shown in the image below.



This was the most basic and obvious way of making an arc in AutoCAD but there are lots of other ways of making the arc as well. Let's select Start, Center, End option from the Arc dropdown.

As the name of command suggests you need to specify the Start point first then center point and lastly Endpoint. Click on point B to specify the start point then click on the midpoint of line BC that will be taken as the center point of the arc and lastly click on point C.

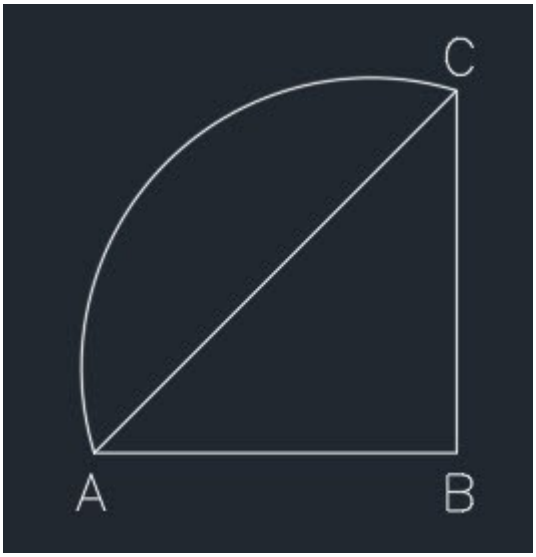
An arc which starts from point B with the center on the midpoint of line BC and ends on point C will be formed as shown in the image below.



In this case, the arc is formed outside the triangle because the arc will be formed in the anticlockwise direction with respect to the starting point which is point B. If you select point C as the starting point and point B as the end you will get your arc on opposite side. Alternatively, you can also press and hold the CTRL key while making the arc to change the direction of arc irrespective of the starting and end point.

The next Arc tool which I will tell you about is Start, End, Radius. For this arc tool, you need to specify the start point, end point, and the radius value. The direction of the arc will be determined by the order in which you select the start and end points.

To make this arc I will select the option from the Arc drop-down menu of the Draw panel and click on the point B then on point A. Now the command line will prompt to specify the radius of the arc. Enter the radius value on the command line and press enter. The final arc will look like the image shown below.

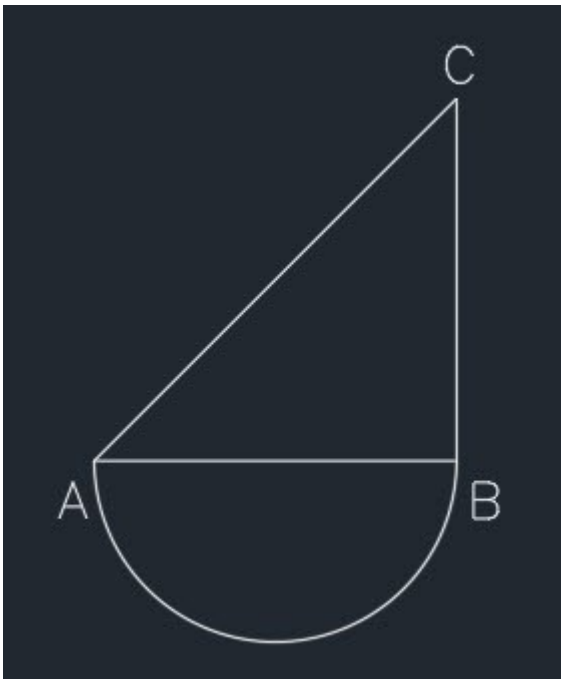


The diameter of the arc, in this case, should always be greater than the length of line AC else the arc will not be possible to construct. For our example, I have used a diameter of 16 units (or a radius of 8 units) and the length of line AC is 14.

In this case, too you can change the order of starting and end points to reverse the direction of the arc or you can press and hold CTRL key while making the arc to reverse the direction of the arc.

The last arc in this series of the arc option which I will explain here is the Center, Start, End and by now you must have understood the workflow of making the arc. In this case, you need to click on the Center point then the start point of the arc and lastly the endpoint.

Select the tool from the Arc drop-down menu on the Draw panel and click on the midpoint of line AB as the center point. Now click on point A and then click on point B. The arc as shown in the image below will be made.



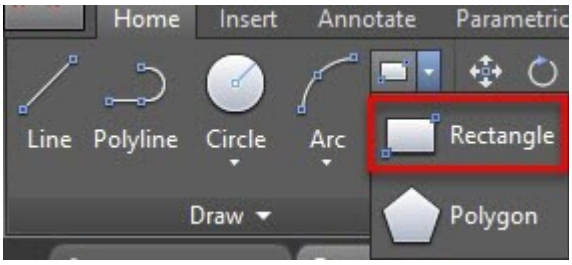
Here also we can reverse the order of selection of start and end points to reverse the direction of the arc. For example, you can select midpoint of the line AB as the center of the arc then point B as the starting point and then point A as the endpoint and the arc will be formed inside the triangle.

You can also press and hold the CTRL key to reverse the direction of the arc while making it. In this case line AB will be the diameter of the arc.

Day 7: Rectangle and Polygon

Making Rectangle

To make a rectangle use the command REC or use the RECTANGLE tool from Draw panel of Home tab. In the first example, I will make a rectangle which starts from origin and has a length of 8 units and width 3 units.



1. Click on rectangle tool of Draw panel.

2. The command line will prompt you to specify the first point, type 0,0 and press enter.
3. Then type 8,3 on next prompt and press enter again
4. The final rectangle will be made with the length of 8 units and width of 3 units.

In this case, the first point was the lower left point of the rectangle which is also the origin and the second point was the upper right vertex with coordinates 8,3

Here, the rectangle started from origin hence the coordinates of point 2 also represented length and width but if you don't want the rectangle to start from the origin instead you want it to start from any other point then you can use the following method.

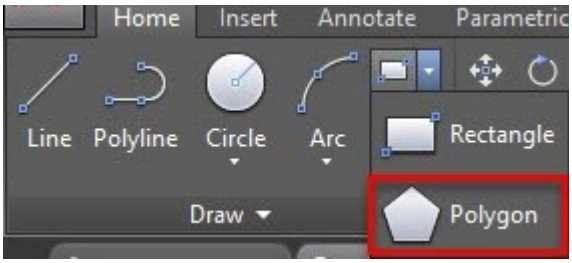
1. Start rectangle command from the Draw panel or use its command REC.
2. Click on any point in the drawing area to specify the first point of the rectangle.
3. Type @8,3 and press enter to specify the next point of the rectangle.
4. The rectangle will be made with the length of 8 units and width of 3 units.

In this case, we have used @ sign additionally to specify the coordinates of the second point because the first point was chosen randomly from the drawing area and it was not on the origin. So, adding the @ sign will signify that we are choosing the first point as origin temporarily for making this rectangle and the coordinates will be measured from the first point.

Here is a video that will make things more clear for you, rectangle command in AutoCAD.

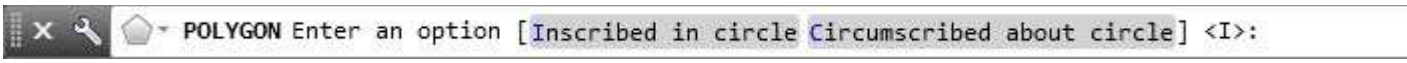
Making Polygon

Polygons are closed geometries made with 3 or more sides. The smallest polygon is a triangle made with three sides and the largest polygon is a circle which is made with an infinite number of sides. In AutoCAD, you can make a polygon with a minimum of three sides and a maximum of 1024 sides.



In this case, I will make a pentagon which is a polygon of 5 sides.

1. Select the Polygon tool from the Draw panel of Home tab or select its command POL.
2. Command line will now prompt you to specify the number of sides of the polygon. Type 5 and press enter.
3. Now command line will prompt you to specify the center of the circle. Click at a point in the drawing are to specify the center.
4. Now command line will prompt you to specify the type of polygon “inscribed in a circle” or “circumscribed about a circle”.



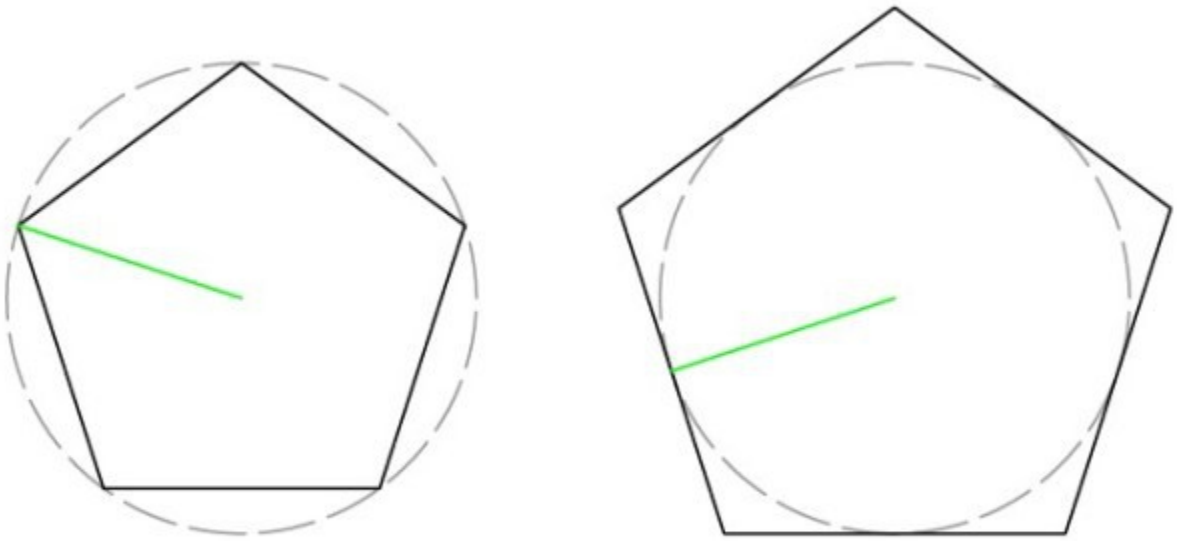
5. Select Inscribed in circle for this case.
6. Now command line will prompt you to specify the radius of the circle. Enter 5 as the radius and press enter key.

You will notice that the inscribed polygon with radius 5 unit will be made. In this case, the radius of the polygon is the radius of the inscribed circle within which this polygon is made.

You can use similar process to make the polygon which is circumscribed about the circle.

Inscribed and circumscribed polygons

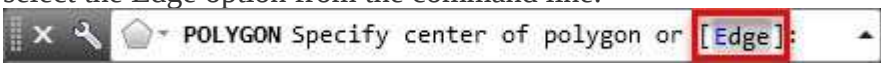
In the image shown here the first polygon is inscribed in a circle with radius equal to the length of the green line. In this case, the vertices of the polygon are touching the circumference of the circle. In second case the polygon is circumscribed about the circle which has a radius equal to the length of green line and in this case the midpoint of sides of polygon are touching the circumference of the polygon.



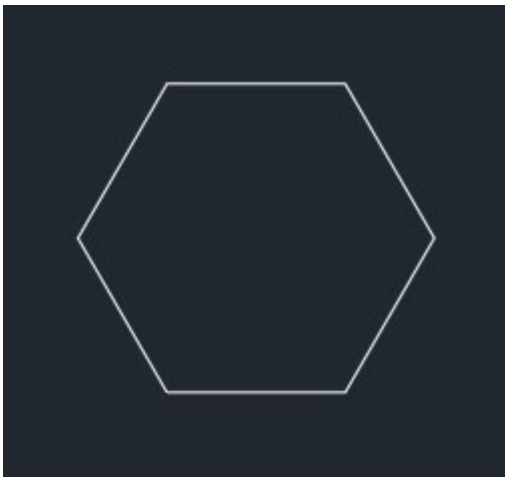
It is not always necessary to make the polygon using the reference of inscribed or circumscribed circle. You can make a polygon with a known edge length also.

In this example, I will make a hexagon with the length of its sides as six units.

1. Start the Polygon command from draw panel or use its command POL.
2. Now command line will prompt you to specify the number of the sides, type 6 and press enter.
3. Now command line will prompt you to select the center of the polygon but in this case, select the Edge option from the command line.



4. Now specify the first point of the edge by clicking anywhere in the drawing area.
5. You will notice that the cursor will now follow the edge of your polygon. Simply click at the second point to make the polygon of required length or specify length on the command line and press enter.



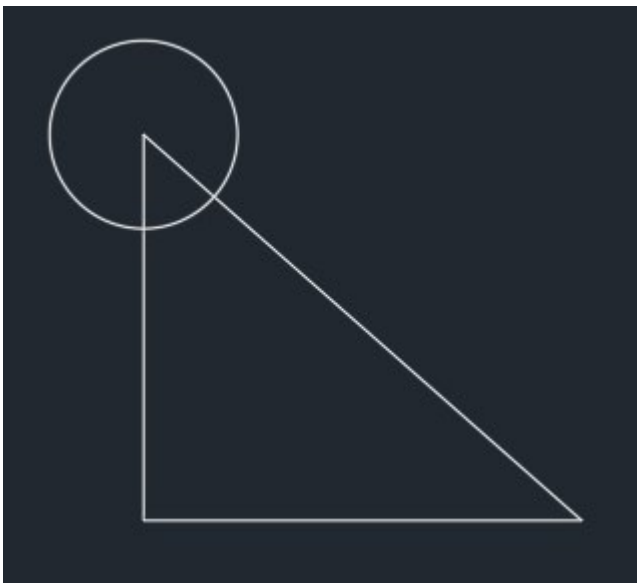
The polygon with required edge length will be made. In this case, you were not required to specify inscribed or circumscribed circle as the reference.

Day 8: Move, Copy and Rotate

Move and Copy Commands

To move a geometry in AutoCAD you can use Move command from Modify panel of the Home tab or you can use its command M.

In this case, we will use the Move tool to move the Pentagon from one of the vertices of the triangle shown in the image below to the other.

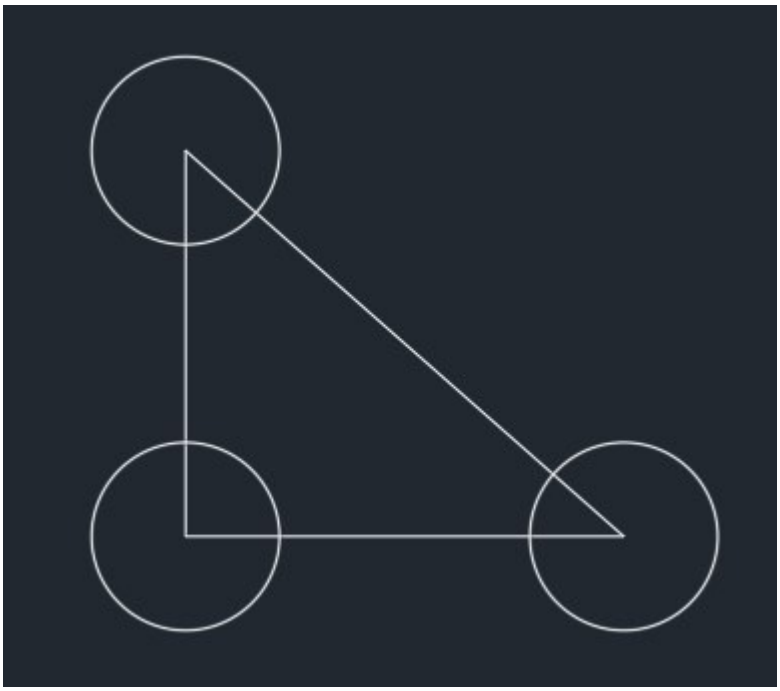


1. Select the Move tool from the Modify panel and then click on the circle from the drawing area and press enter.
2. Now click on the center of the circle, this center will become the base point from where the circle will be picked.
3. Now click on the lower right vertex of the triangle, the circle will be moved to its new location.

To copy the circle on all three vertices you can use the Copy tool from the Modify panel of the Home tab or you can use its command CO.

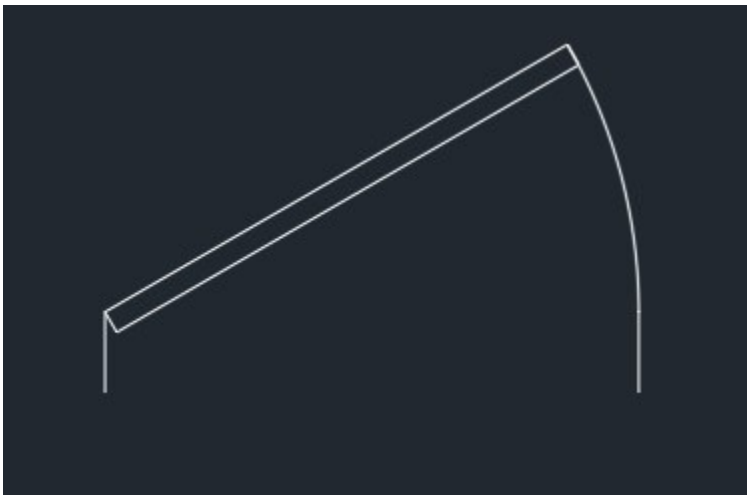
1. Click on copy command from Modify panel and then click on the circle from the drawing area which you want to copy and press enter.
2. Now once again click on the center of the circle and that will become the base point of your selection.
3. Click on the other two vertices of the triangle to paste the circle there.

To end the command press, enter again. The final drawing after copying circles on all three vertices will look like this.



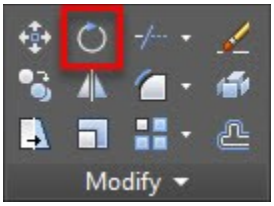
Rotate Command

As the name suggests this command can be used to rotate the object about a point. In this case, I will use this Door symbol to explain the tool.



Currently, the door is horizontal, and we can rotate it to change its rotation angle. To do this I will use Rotate command.

Select Rotate command from the modify panel or use its command ROTATE.



The command line will prompt you to select the objects, click on the door to select it and press enter. Now command line will prompt you to specify the base point, click on the lower left corner of the door as the base point. The base point will become the pivot point of rotation and it will remain fixed and the door will rotate about that point.

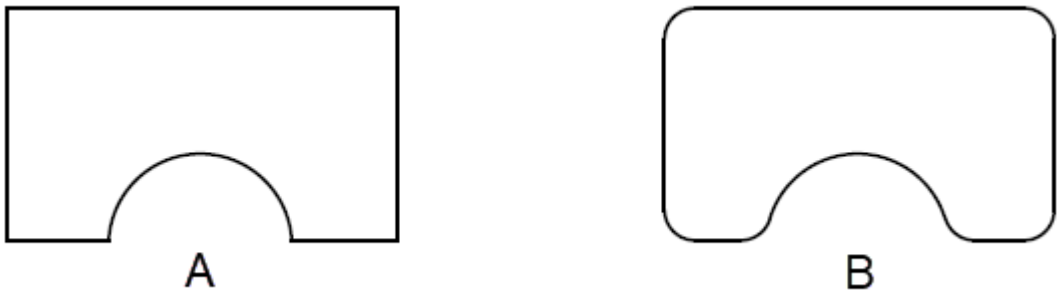
Now command line will prompt you to specify the angle of rotation, type 30 and press enter. Your door will rotate to an angle of 30 degrees with respect to the positive side of the X-axis and the final door should look like this image.

In a similar way, you can specify different base points and rotation angles to get different results. To know more about rotate tool watch this video, Rotate and Scale command in AutoCAD.

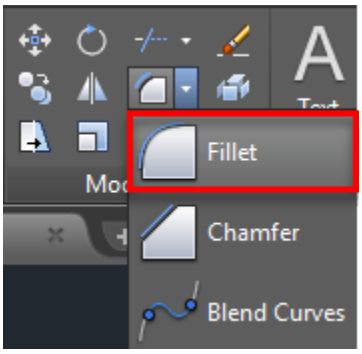
Day 9: Fillet and Trim Commands

Fillet Command

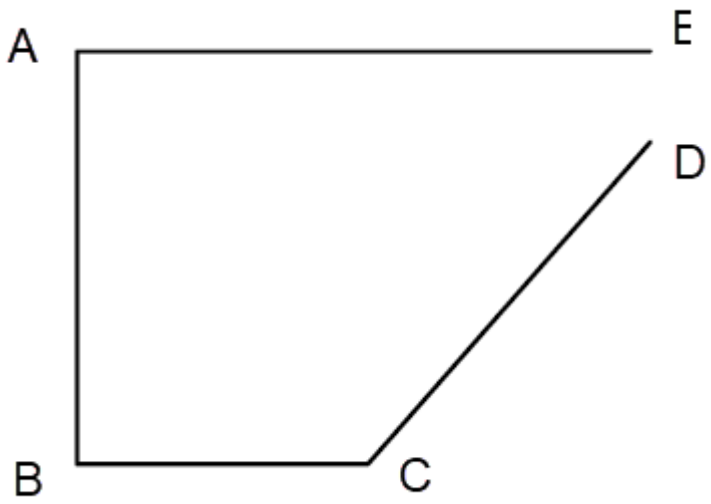
Fillet command can be used to add round corners on the sharp edges of the drawing. For example, in this case, the fillets are added on the vertices of this drawing A shown below to make it look rounded on the corners as shown in drawing B.



To use the fillet command select it from the modify panel of Home tab as shown in the image below or use its command F.

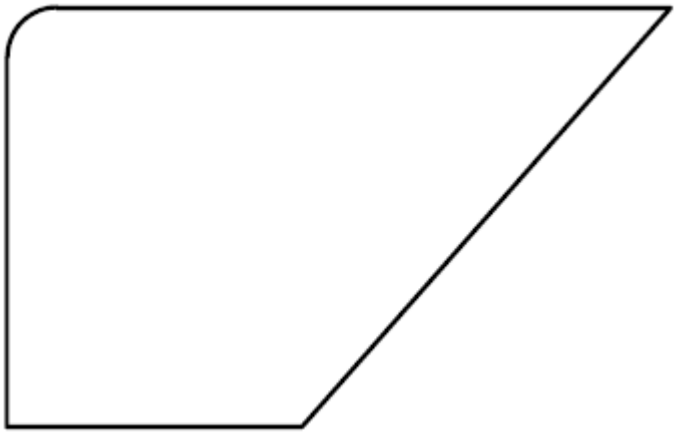


Let’s take the example of the drawing shown in the image below, this drawing has been made with a polyline.



To add the fillet on the vertices of this drawing select the command from the modify panel or use its command F. Now select the radius option from the command line and enter the value of radius that you want to apply on the vertices. Now command line will prompt you to select the objects to fillet.

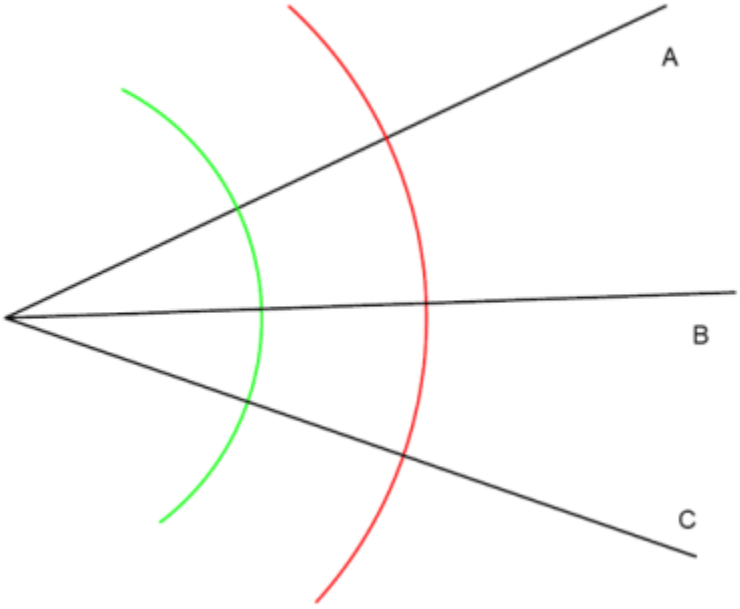
Click on the line AB, then on the line AE and the fillet will be applied on the vertex A. To apply the fillet on the open edge ED use the same method bust use line segments CD and AE as objects to fillet. If you want to merge the line CD and AE at their apparent point of intersection then use the fillet radius of 0 and apply the fillet of CD and AE edges.



The line segments will merge into a vertex. To know more about the fillet command watch this video, Fillet chamfer command of AutoCAD.

Trim Command

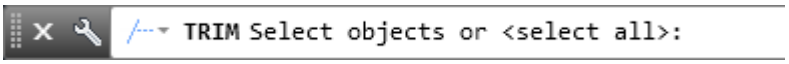
Using trim command you can remove parts of drawing up to next intersection point or vertex. To explain this command properly I will use the drawing shown in the image below.



Select Trim command from the modify panel or use its command TR. Now the command line will prompt you to select the object, click the green arc and press enter. Now you can select the objects to trim and they will be trimmed with respect to the green arc.

Simply click on the right side of the line A and you will notice that it will be trimmed up to the green arc. Similarly, click on other lines and they will also be trimmed up to the green arc.

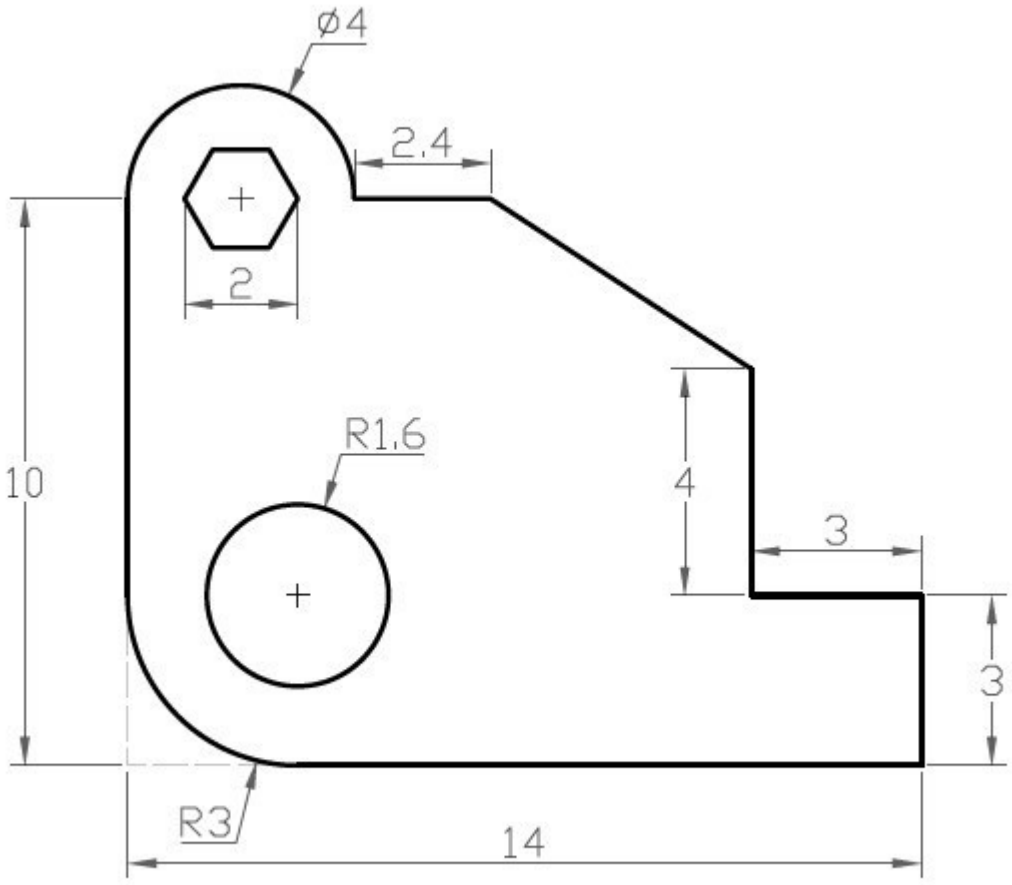
In this case, the trim command will ignore the red arc for trimming the line as it was not included as trimming boundary. To include everything as a trimming boundary select the trim command again and press enter when the command prompts to select the object as shown in the image below.



This will select everything in the drawing area as trimming boundary and now if you click on any object it will be trimmed up to the next boundary.

Day 10: Making first AutoCAD drawing

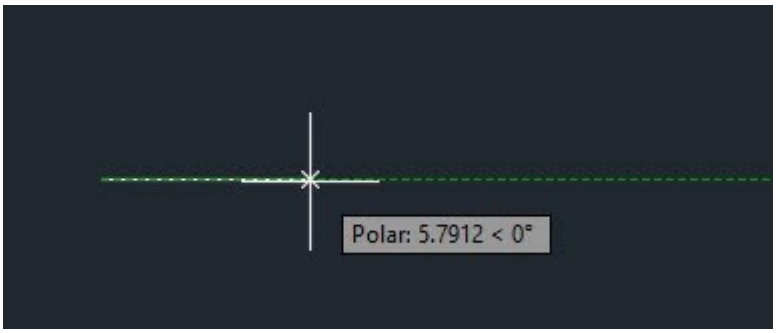
So now we are equipped with the basic drawing tools that are required for making simple geometries in AutoCAD. We will start using these tools and many other tools as well to make our first drawing as shown in the image below.



We will start making this drawing with the line command.

Type L and press enter then click at any point in the drawing area to start the line.

Move your cursor towards right direction so that green tracking vector of Polar tracking is visible and then type 14 and press enter again. This will make our base line of length 14 units.

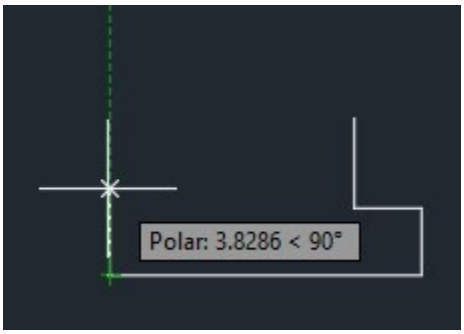


Move your cursor upwards so that the green tracking vector is again visible then type 3 and press enter again.

Now move towards left type 3 and press enter again.

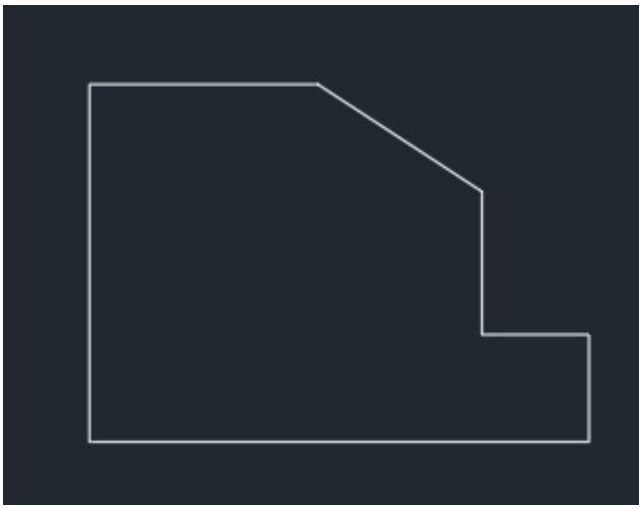
Move in upwards direction type 4 and press enter twice to make the line and exit the command.

Click again at the starting point of the first line then move your cursor in upwards direction then type 10 and press enter.



Move your cursor towards right side again and then type 6.4 and press enter.

Lastly, join the line with the open end to complete the drawing’s outer geometry.



Now the outer boundary of our drawing is prepared we will now add more details like polygon and circle. I will start with the circle command.

Move to the Draw panel and select Tan, Tan, Radius circle option from the flyout.

Now click on the horizontal line at the bottom and then click on the vertical line which is on the left.

Type 3 for the radius and press enter to make the required circle.

Now select the Center, Radius circle and click on the center point of the circle made in the previous step.

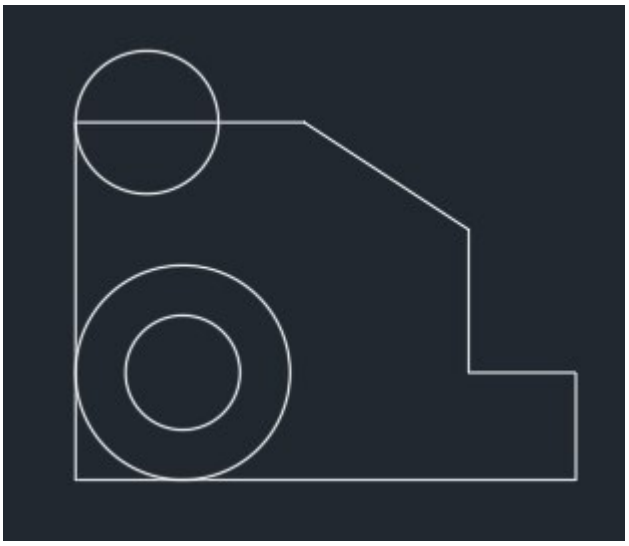
Type 1.6 for the radius of the circle and press enter again.

Once again go to the Draw panel and select the 2-Point circle from the circle flyout.

Now click on the top left point of the geometry and move your cursor towards right direction so that the green tracking becomes visible.

Enter 4 units on the command line and press enter key again.

A new circle with the diameter of 4 units will be made at the topmost point of the geometry. Your final drawing should look like the image shown here.

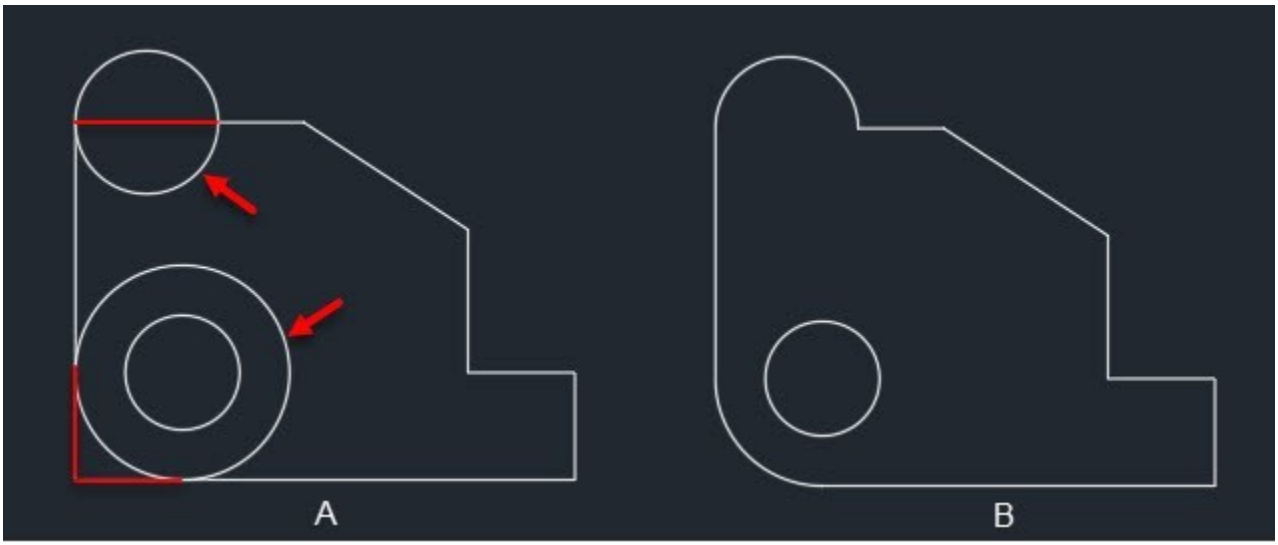


Now we need to clean all the extra geometry which is present in the drawing. And for that, we will use trim command.

Select the trim command from the modify panel of the home tab or use the command TR

When the command line prompts you to select objects simply press enter without selecting anything and that will select all the geometries in the drawing.

Carefully click on the segments that need to be removed as shown in the image A here. The final geometry after trimming extra geometries will look like image B.



Note: *In this case, we used Tangent, Tangent Radius circle to make the arc on the bottom left of the drawing but you can also use Fillet command to make this arc shape easily.*

Now we only need to add the hexagon and for that, we will use the Polygon tool.

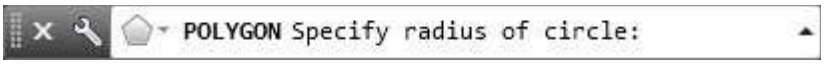
Select the Polygon tool from the Draw panel of Home tab or use its command POL.

Type 6 for the number of sides and press enter.

Now click on the center of the arc which is made on the top of the geometry. This center is also the center of the hexagon which we are making.

Select Inscribed in the circle from the options of the command line.

Type 1 on the radius prompt and press enter again.



The inscribed hexagon with radius 1 will be added to the geometry.

Jaiprakash Pandey

I am a mechanical engineer and I have worked with Design, Manufacturing and Training industries and currently, I work as CAD corporate trainer. I train corporate clients in AutoCAD, Fusion 360, and Geometric dimensioning and tolerancing and help them develop the skill set of their workforce.

I am Autodesk AutoCAD Certified Professional and an Autodesk expert elite member. I am a regular contributor to AUGI World magazine and apart from SourceCAD I also develop AutoCAD video courses for Pluralsight and other E-Learning businesses.

SourceCAD is a platform developed by me and it hosts video courses and tutorials on different CAD software and engineering drawing related topics. The courses and tutorials are free as well as paid.

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