Department of Computer Technology Engineering

Engineering Drawing

Prepared by

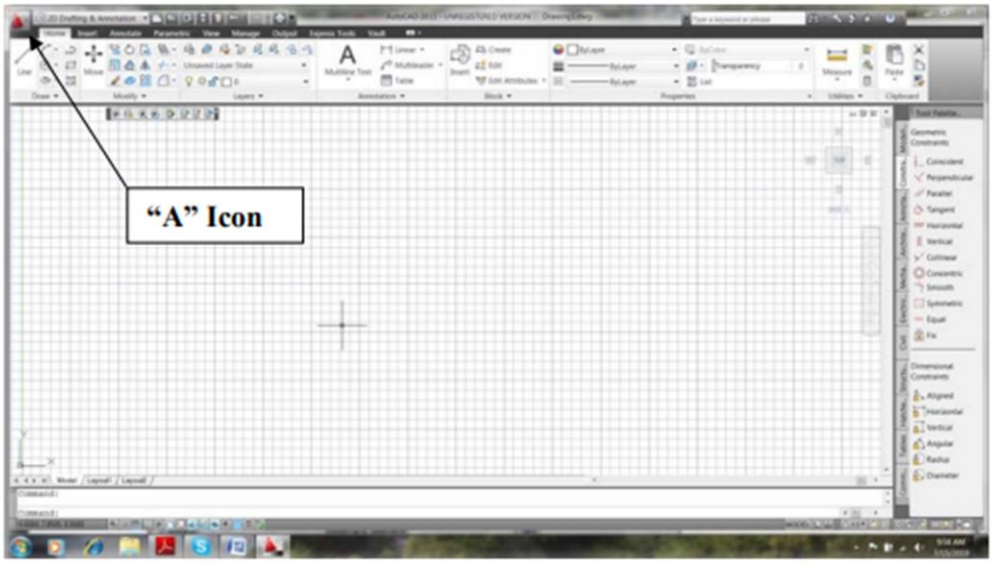
Huda Majeed

Introduction to AutoCAD

The term CAD (Computer Aided Design) applies to a wide range of programs that allow th user to created drawings, plans, and designs electronically. AutoCAD is one such program and it main claim to fame is that it is relatively easy to use, it is very comprehensive in its ability to create 2D and some 3D drawings, and it is very popular. Seventy percent of the CAD users in the world use AutoCAD.

Starting AutoCAD

You can start AutoCAD by either double clicking on the program Icon on the desktop or by clicking on the program name in the Start menu. The program will start and after a minute or so should display a screen similar to the one shown below. The dialog box in the middle will aid you in getting started at either creating a new drawing or continuing your work on a drawing that is not finished.:



Creating a New Drawing

To creates a new drawing file.

1. Choose File, New. or

2. Press CTRL + N or

3. Click the New icon. or

4. Type NEW at the Command prompt.

5. Choose One of the options for creating a new drawing.

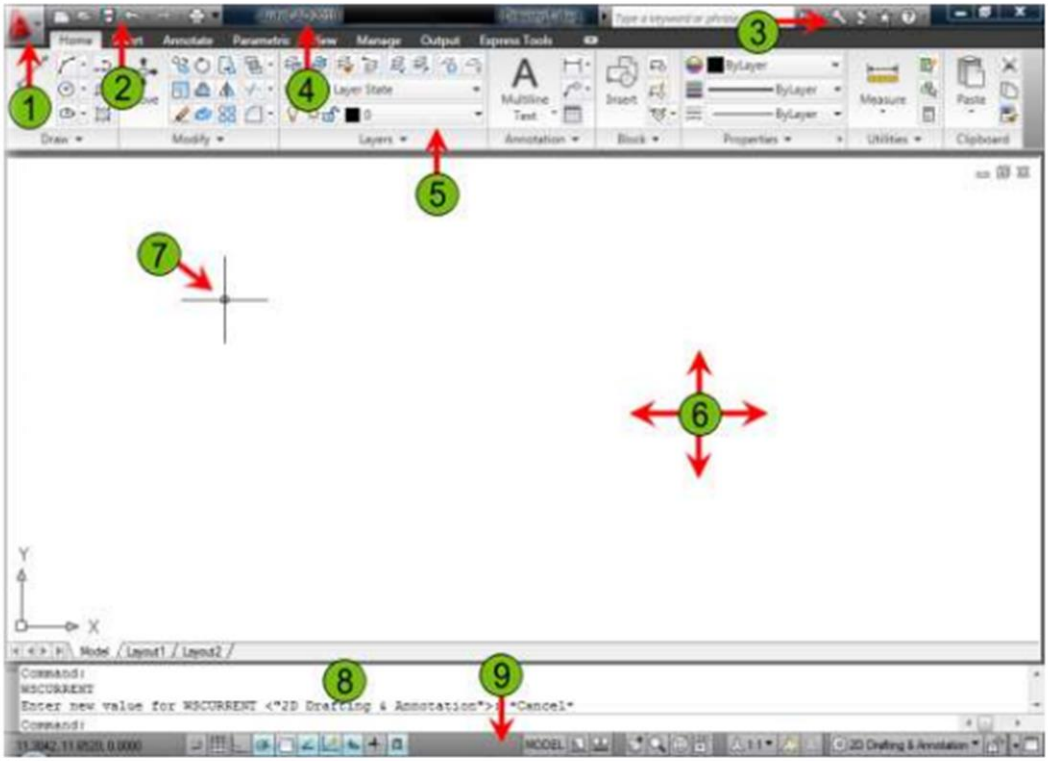
6. Click The OK button.

7. Save the drawing as another name.

If you are continuing work on a drawing, click on the “A” icon in the extreme upper left corner of the window and Open. A “Select File” dialog box will open allowing you to select the drawing file you want to open.

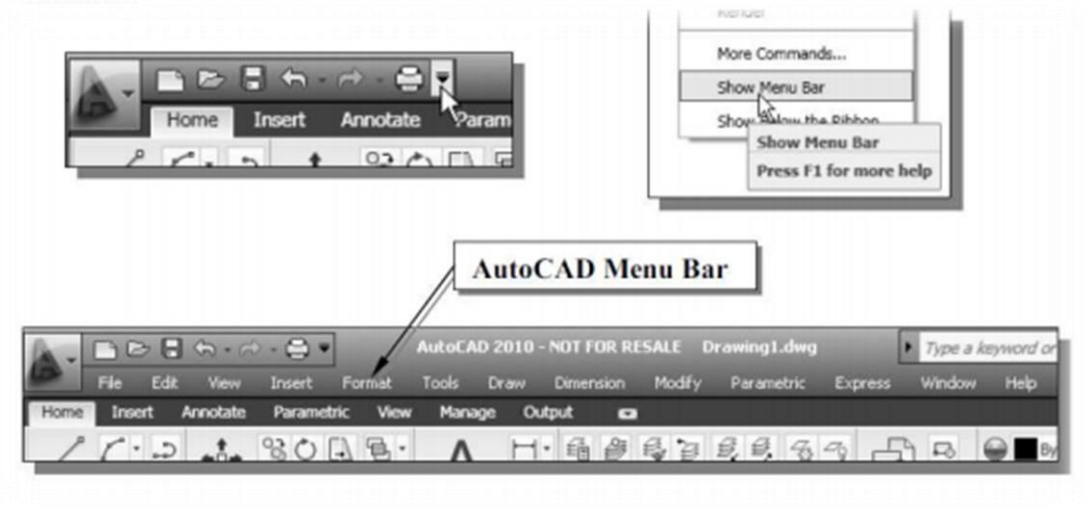
Navigating the Working Environment

The description of the working environment and the types of interface elements that you must become familiar with if you are to become proficient in the software. The following image identifies key interface elements:


1
2
3
4
5
6
7
8
9
Application Menu
Quick Access Toolbar
Info Center
Title Bar
Drawing Area
Crosshairs
Command Window
Status bar

Add or Remove Menu Bar

Click on the down-arrow in the Quick Access bar and select Show Menu to display the AutoCAD Menu Bar. The Menu Bar provides access to all AutoCAD commands.



Command Window and Status Bar

The following image shows the command window. Some what unique to a graphical windows application, the command window provides another method for the user to interact with the application. If we type the letter l then a line will be drawn and type e to erase.

To the right of the Coordinates Display, there are buttons that activate features to facilitate drawing construction. Collectively, these features are termed drafting settings.

Drafting Settings

[GRID] (F7)

The most basic drawing aid, grid, makes the canvas in AutoCAD look like graph paper.

Snap

snap constrain your ability to draw object so that they automatically start and end precisely at grid intersection grid and snap are most helpful when used together so that you can draw objects that snap to the grid .

[ORTHO] (F8)

When Ortho is ON, cursor movement is restricted to horizontal or vertical. When Ortho is OFF, the cursor moves freely.

[POLAR] (F10)

POLAR TRACKING creates “Alignment Paths” at specified angles.

[OSNAP] (F3)

The Object Snaps (Osnaps for short) are drawing aids which are used in conjunction with other commands to help you draw accurately. Osnaps allow you to snap onto a specific object location when you are picking a point. For example, using Osnaps you can accurately pick the end point of a line or the center of a circle. Osnaps in AutoCAD are so important that you cannot draw accurately without them. For this reason, you must develop a good understanding of what the Osnaps are and how they work.

Command Interface

Line Command

Creates single straight line segments

1. Choose Draw, Line. Or

2. Click the Line icon. Or

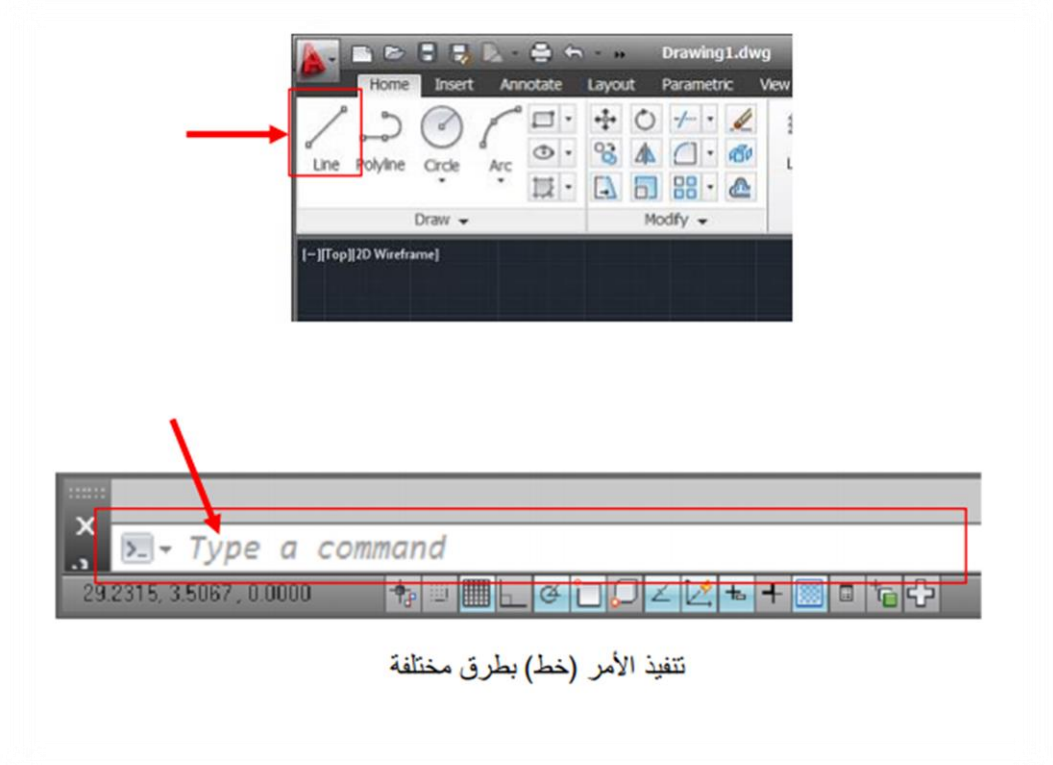
3. Type LINE from the command prompt Command: LINE or L

4. Press ENTER

5. Pick From point: (point)

6. Pick Specify next point or [Close/Undo]:(point)

7-press ENTER to end line sequence



تنفيذ الأمر (خط) بطرق مختلفة

Polyline Command

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.

1. Choose Draw, Polyline.

Or 2. Pick the Polyline icon.

3. Type polyline at the command prompt

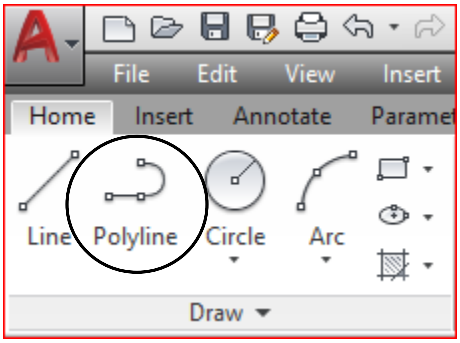
Command: PLINE or PL

4. Pick A point on the drawing to start the polyline

From point:(select)

5. Type One of the following options

6. Pick A point to continue drawing



or

تنفيذ امر ال(polyline) باكثر من طريقة

Department of Computer Technology Engineering

Engineering Drawing

Prepared by

Huda Majeed

Rectangle

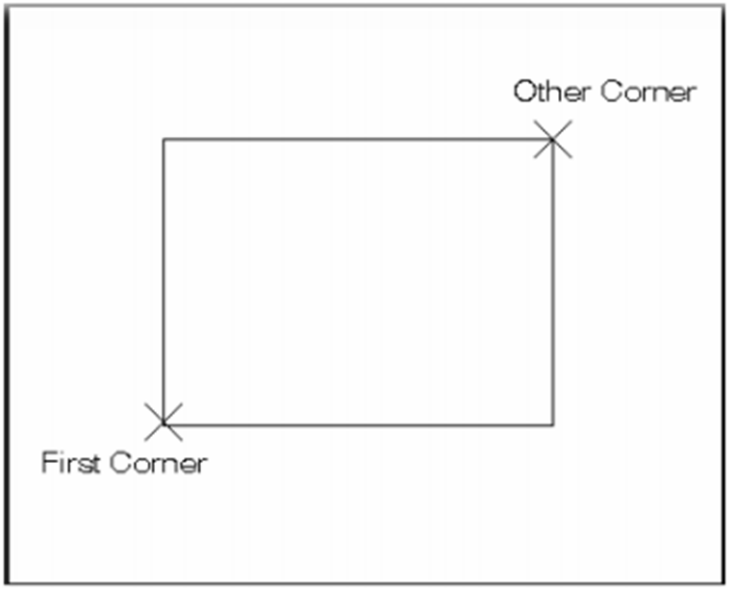
1-choose Draw, Rectangle or

2-click the rectangle icon in home

3-Type rectangle at the command prompt command .

4-pick first corner

5-pick the other corner



Circle

Circle command

1-choose Draw ,circle or

2-click the circle icon

3-or type circle on command prompt

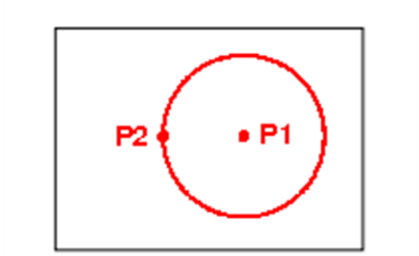
Pick a center of circle

Type of Circle

Center, Radius: (Default option)

1. Specify the center (P1) location.

2. Specify the Radius (P2).



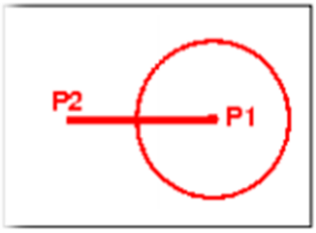
Center, Diameter:

1. Specify the center (P1) location.

2. Select the Diameter option using the shortcut menu

or type “D” <enter>.

3. Specify the Diameter (P2).



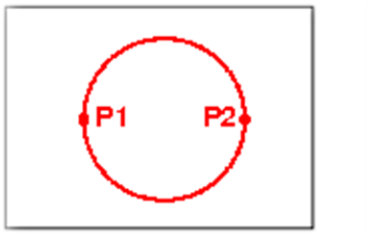
2 Points:

1. Select the 2 point option using the short cut menu or type

2P <enter>.

2. Specify the 2 points (P1 and P2) that will determine the

Diameter .



3 Points:

1. Select the 3 Point option using the short cut menu or type

3P <enter>.

2. Specify the 3 points (P1, P2 and P3) on the circumference.

The Circle will pass through all three points.

Tangent, Tangent, Radius:

1. Select the Tangent, Tangent, Radius option using the

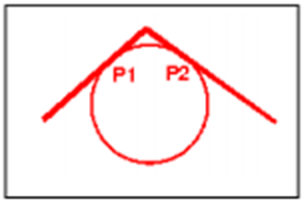
short cut menu or type T <enter>.

2. Select two objects (P1 and P2) for the Circle to be

tangent to by placing the cursor on the object and

pressing the left mouse button.

3. Specify the radius.



Tangent, Tangent, Tangen

1. Select the Tangent, Tangent, Tangent option using the

pull down menu. This option is not available in the

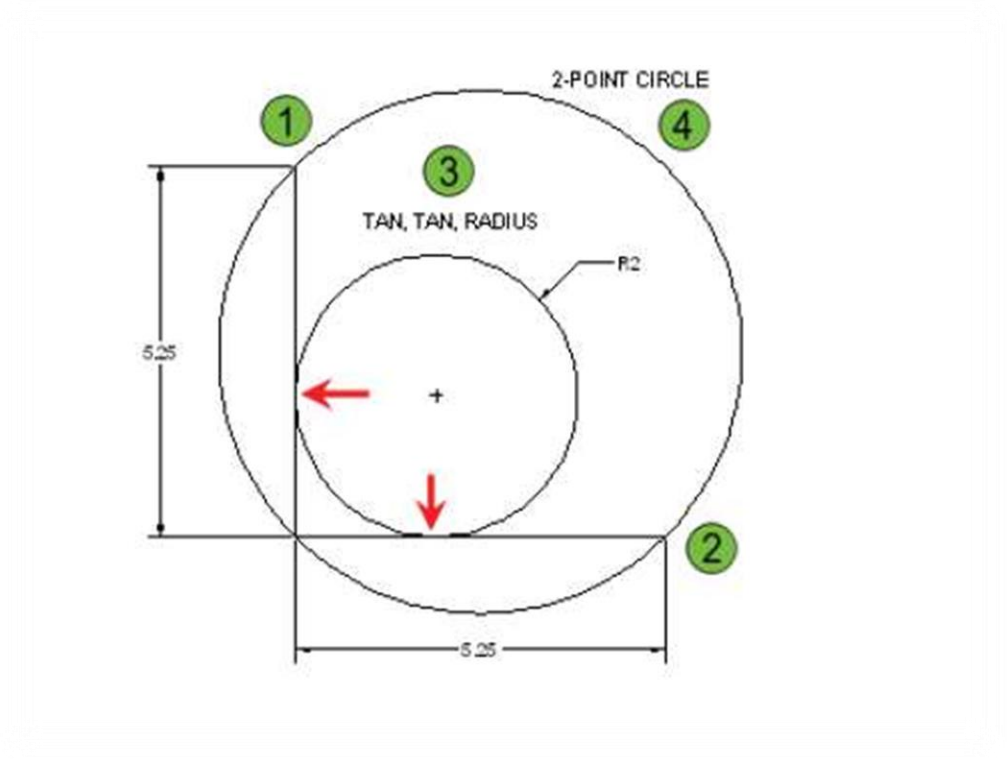
short cut menu or the command line.

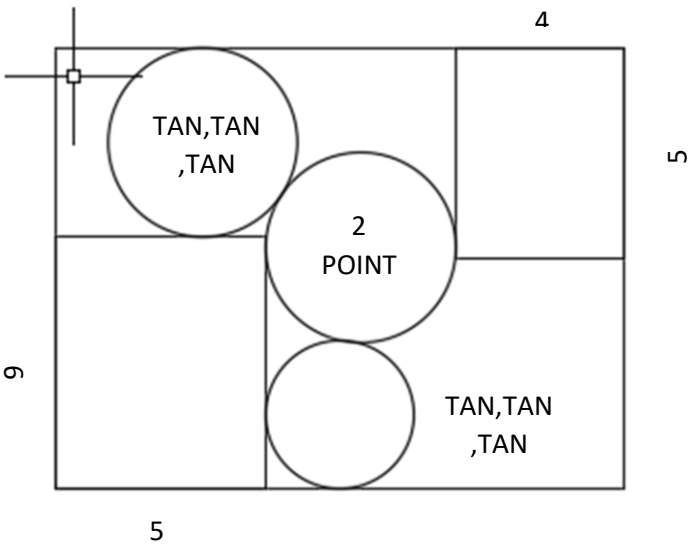
2. Specify three objects (P1, P2 and P3) for the Circle to

be tangent to by placing the cursor on the object and

pressing the left mouse button. (The diameter will be calculated by the computer.)

Exercise : Draw the following figure.





Department of Computer Technology Engineering

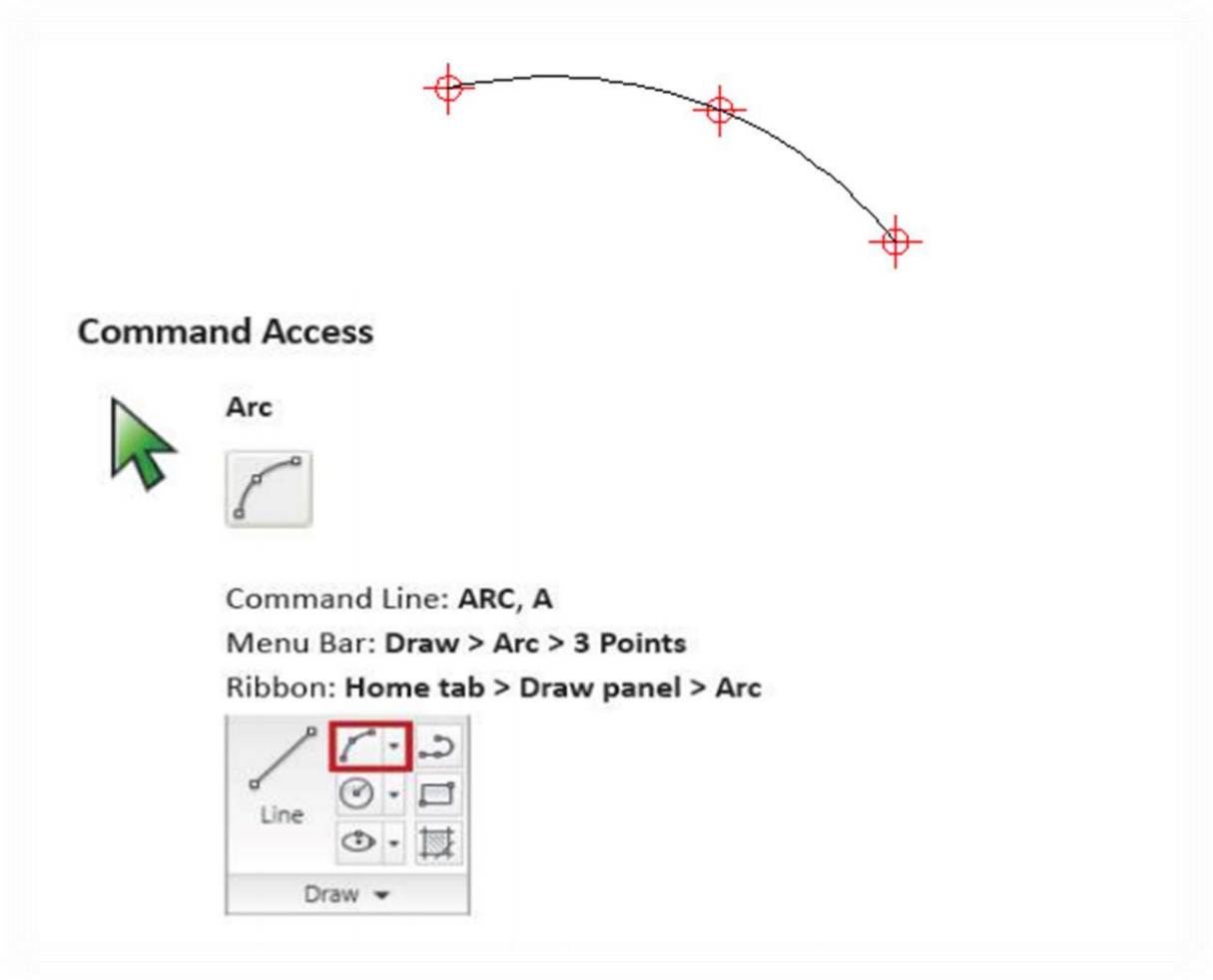
Engineering Drawing

Prepared by

Huda Majeed

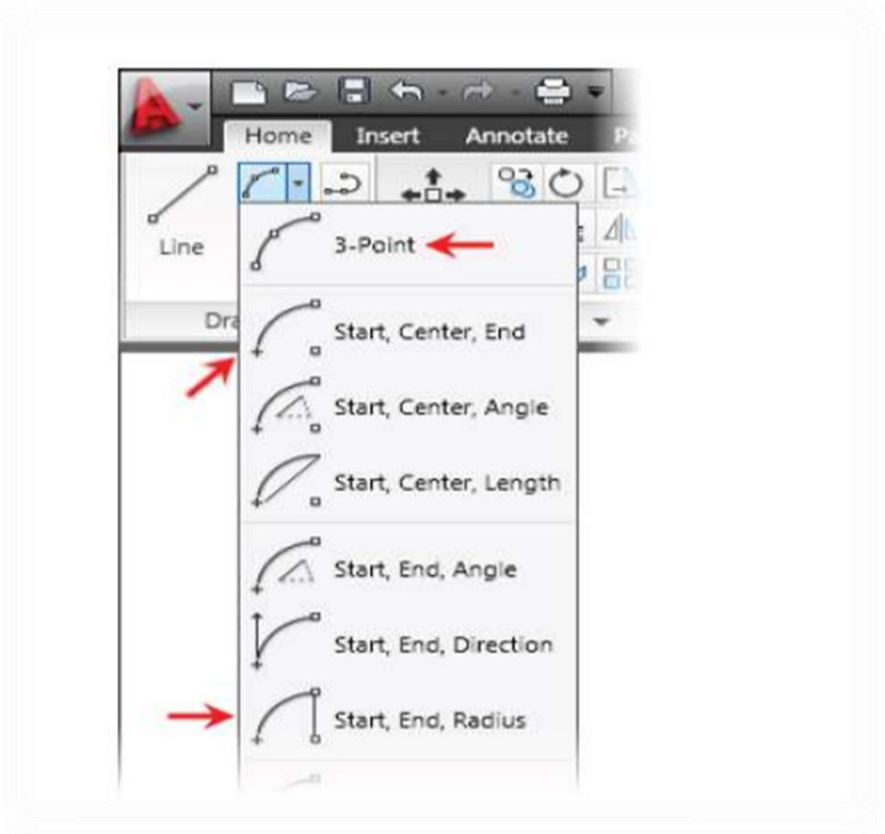
Arc Command

The arc is defined as the smooth curve formed by joining two or more points.The arc of a circle is defined as the portion of the circumference of a circle. By default, the arc is drawn counter-clockwise. We are required to press the Ctrl key to change the direction of the arc in the clockwise. The arc can be drawn by specifying the starting point, center, and the endpoint. We can also draw an arc by determining the angle, radius, and direction. The icon of Arc on the ribbon panel looks like the below image .



Command Options

Arc options can be accessed from the drop-down menu next to the Arc button .The most common Arc options are indicated by red arrows in the following illustration

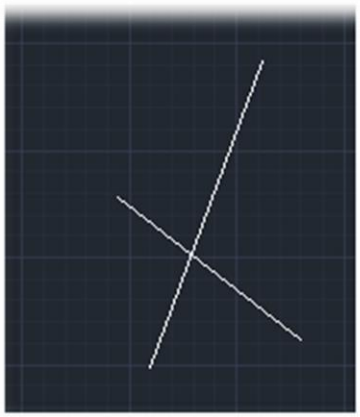


- **3 Point**

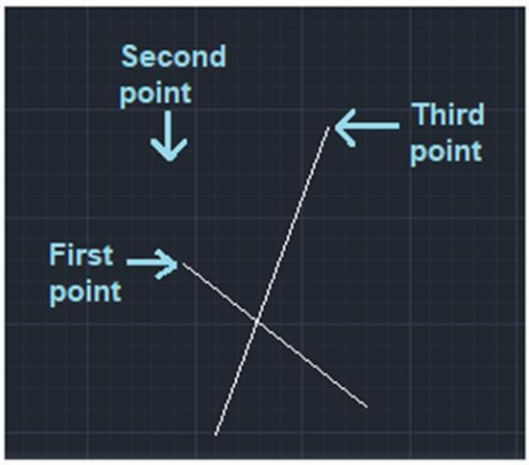
Example: To create an arc on two intersecting lines. Here, we need to specify **three** points to create an arc.

The steps to create a 3-point arc are listed below:

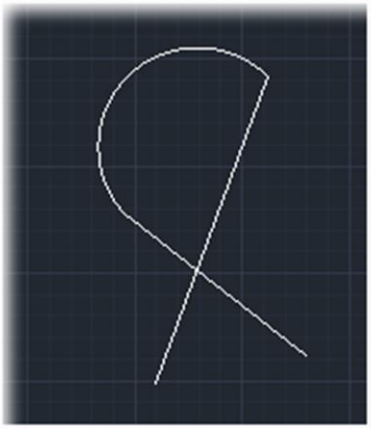
1. Create two intersecting lines, as shown in the below image



1. Select the **Arc** icon from the ribbon panel.
   Or
   Type **A** on the command line and press **Enter**.
2. Specify **first**, **second**, and the **third** point over the top of two lines, as shown in the below image:
3. We can change the position of points accordingly.



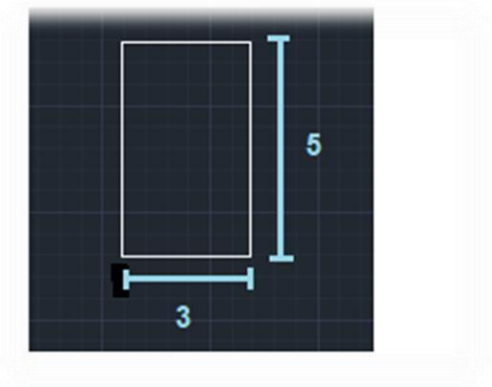
4. The arc is shown in the below image



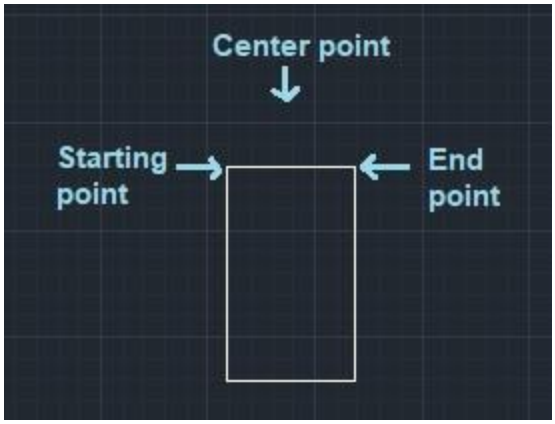
Start, Center, End

Example: Arc on a rectangle.The steps to create an arc are listed below: Create a rectangle. Here, we have created a rectangle with length 3 and width 5, as shown in the below image .

5. Select the **Arc** icon from the ribbon panel.
   Or
   Type **A** on the command line and press **Enter**.



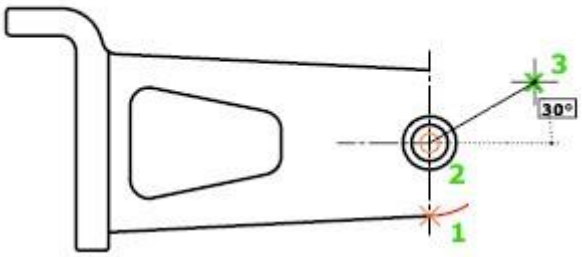
6-Specify the **starting, center,** and **end** point of the arc on the rectangle, as shown in the below image:



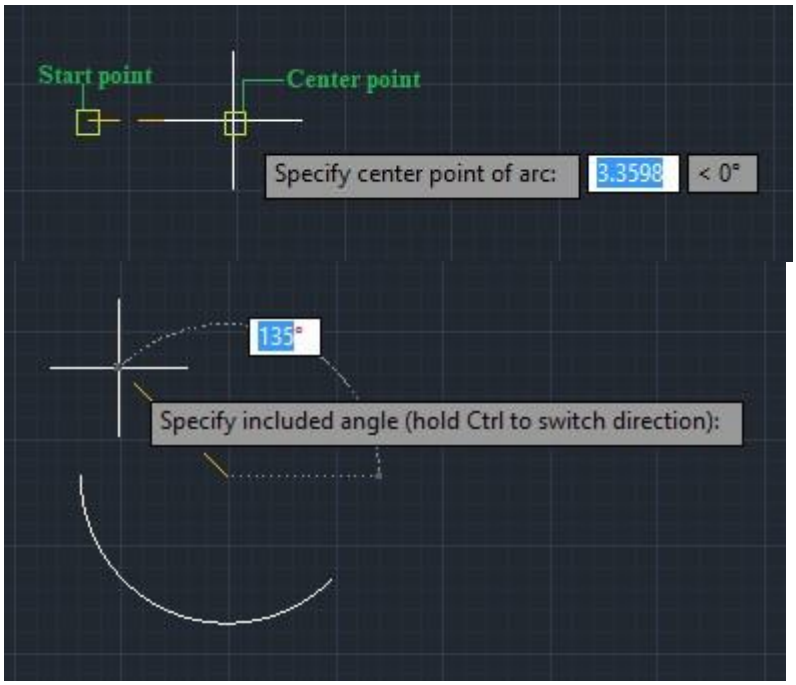
6. The arc on the rectangle is shown in the below image:

- **Start, Center, Angle**

It creates an arc using the start point, center point, and an included angle. Consider the below image:



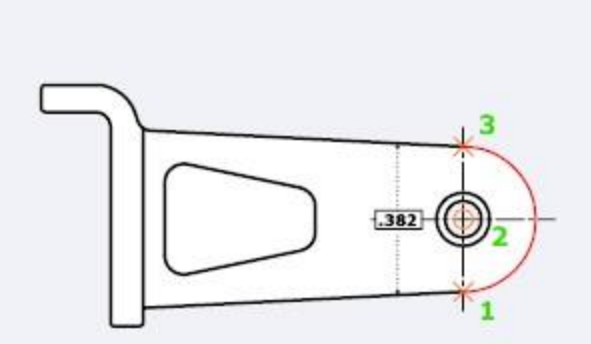
Here, 1 and 2 are the **starting** and **center** point, and 3 is the specified **included** angle.The radius of arc is defined as the distance between the center point and the start point.Let's understand by another example.The process is shown below:



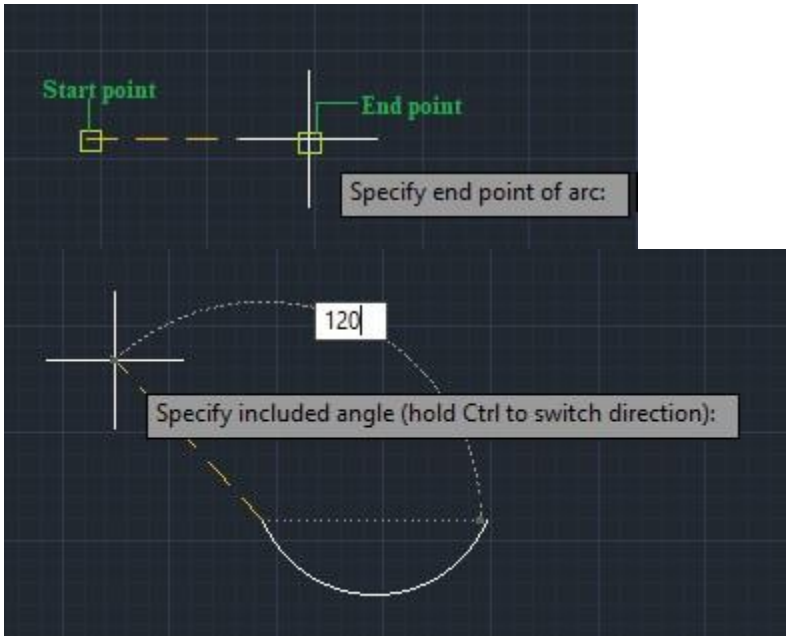
The arc will be created counter-clockwise from the start point. We need to hold the **Ctrl** key to switch the direction of the arc.

Start, Center, Length

It creates an arc using the start point, center point, and the length of the chord. Consider the below image:



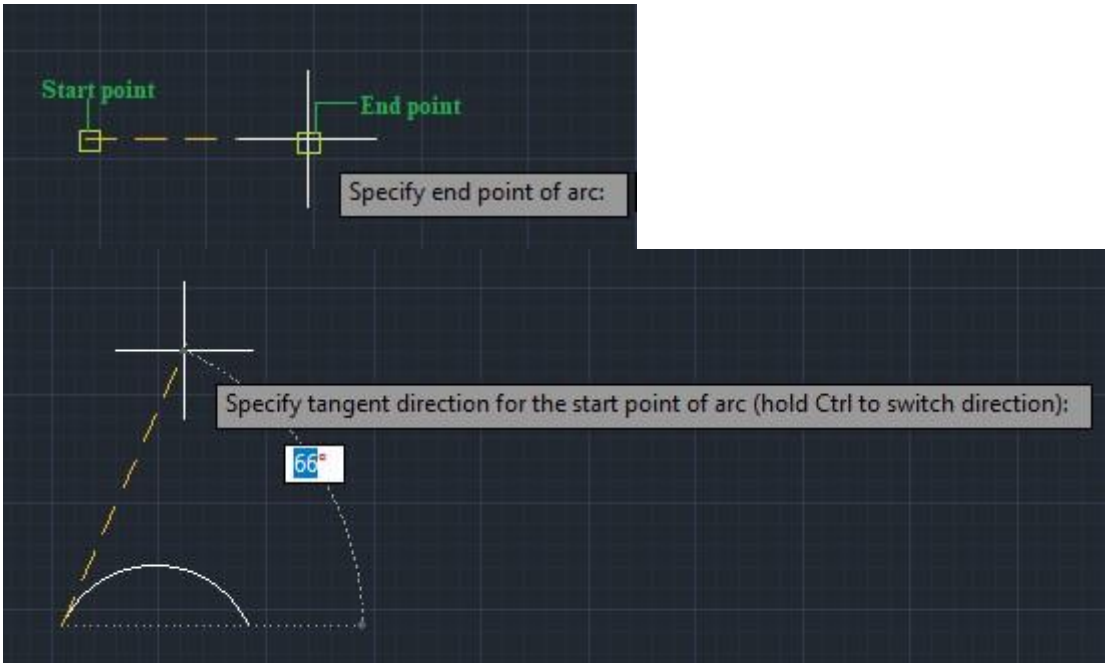
Here, 1 and 2 are the **starting** and **center** point. The distance between points 1 and 3 determines the **length** of the chord.The distance between the center point and the start point determines the radius.Let's understand by another example.The process is shown below: The process is shown below:



Here, we have specified the included angle of **120**.The created arc is shown below:

Start, End, Direction

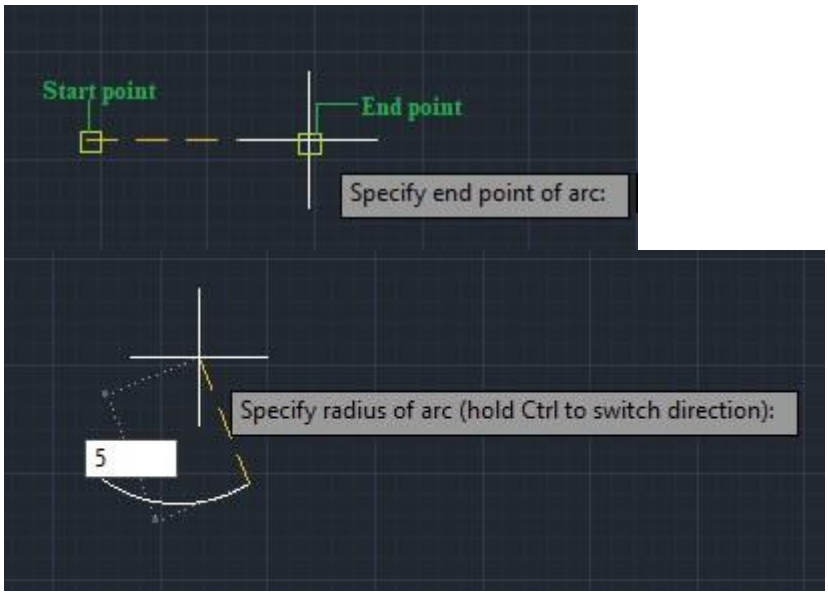
It creates an arc using the start point, end point, and a tangent direction at the start point.Let's understand by example.The process is shown below:



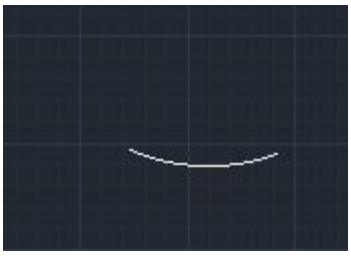
We can specify the direction of the tangent either by a point on the desired tangent line or by specifying the angle value. We can also change the order of the ndpoints.

Start, End, Radius

It creates an arc using the start point, end point, and a radius.Let's understand by example.The process is shown below:



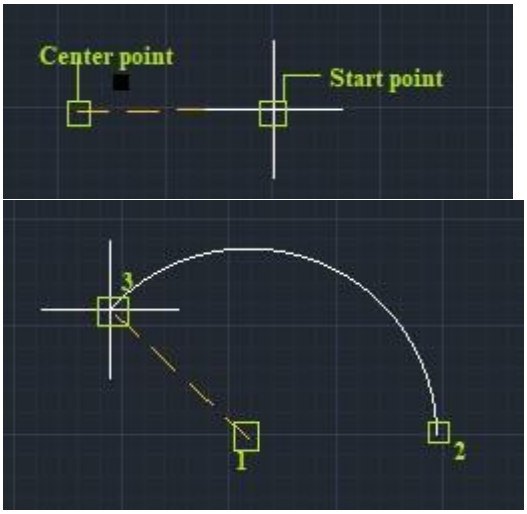
Here, we have specified the radius value 5.The created arc is shown below:



The direction is determined by order of the endpoints.We can specify the radius by entering the radius value. We can also determine it by specifying the point at the desired distance.

Center, Start, End

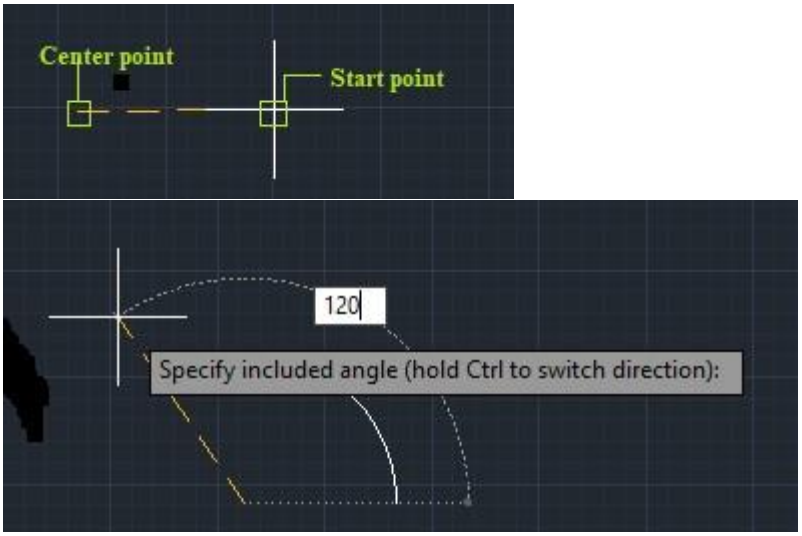
It creates an arc using **center** point, **start** point, and the **endpoint**. The endpoint can also be determined as the third point.Let's understand with an example.The process is shown below:



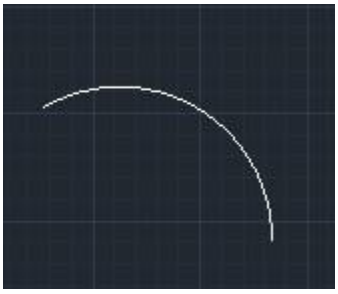
Here, 1 and 2 are the **center** point and the**start** point. 3 is the third point or **end**point.The radius of arc is defined as the distance between the center point and the start point.The arc is created in an anti-clockwise direction.To switch the direction, we need to press the **Ctrl** button.

Center, Start, Angle

It creates an arc using **center** point, **start** point, and the included **angle**.Let's understand with an example. The process is shown below:



Here, we have specified the included angle= 120.The created arc is shown below:

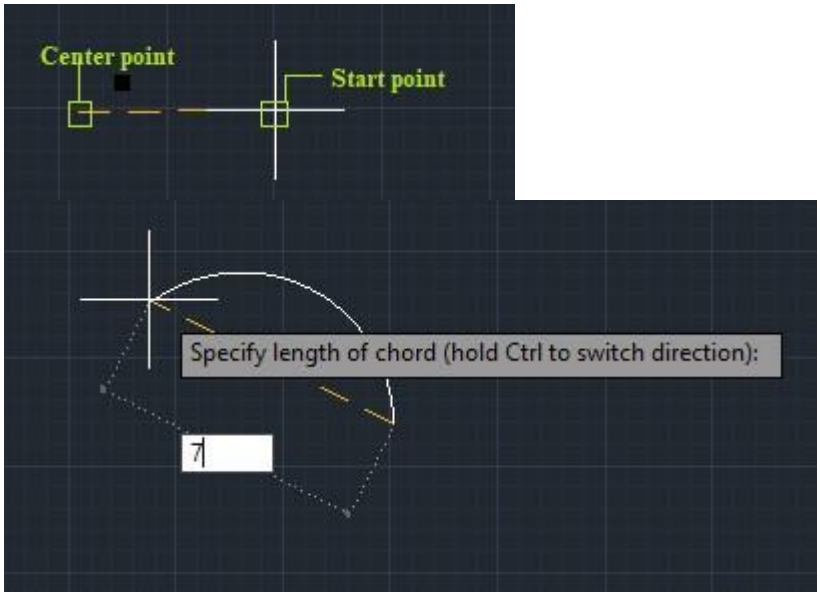


The radius of arc is defined as the distance between the center point and the start point.The arc is created in an anti-clockwise direction.To switch the direction, we need to press the **Ctrl** button.

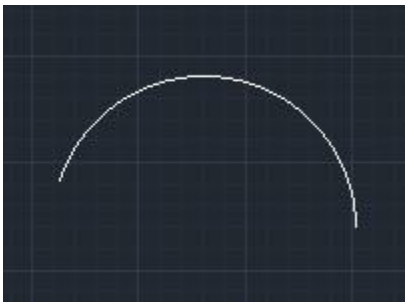
Center, Start, Length

It creates an arc using **center** point, **start** point, and length of the chord.

Let's understand with an example.The process is shown below:



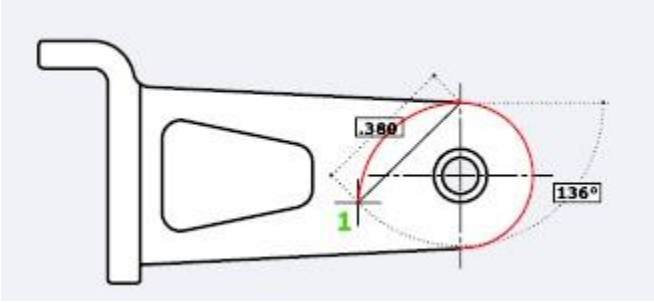
Here, we have specified the length of the chord= **7**.The created arc is shown below:



The radius of arc is defined as the distance between the center point and the start point. The arc is created in anti-clockwise direction.To switch the direction, we need to press the **Ctrl** button.

Continue

It creates an arc from the last line or arc drawn. Consider the below image:



After the drawn arc or line, we can immediately create an arc that is tangent at an endpoint.

We need to only specify the endpoint of the arc

Department of Computer Technology Engineering

Engineering Drawing

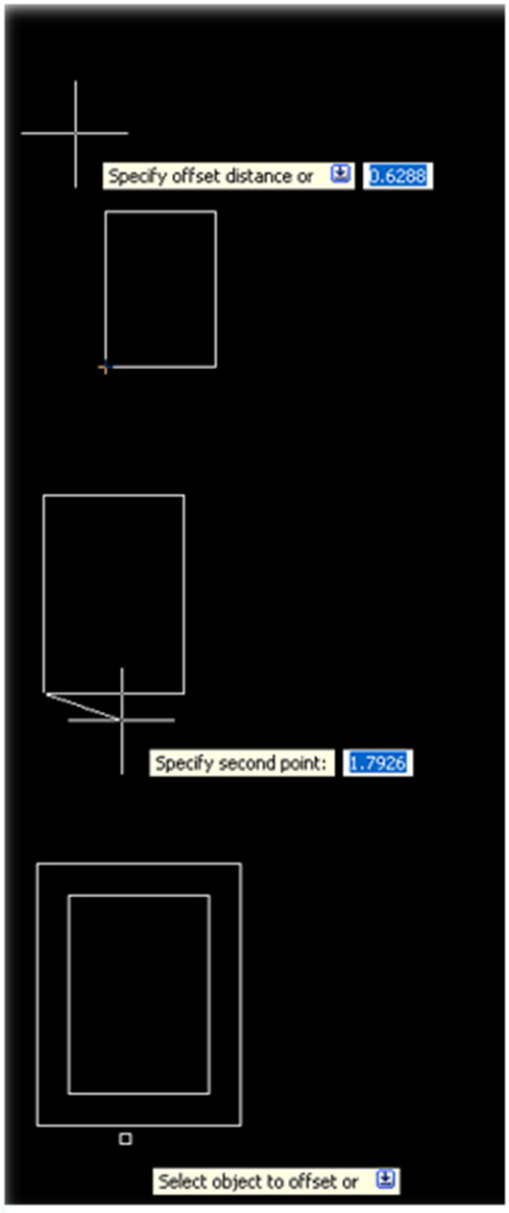
Prepared by

Huda Majeed

Offset (o)

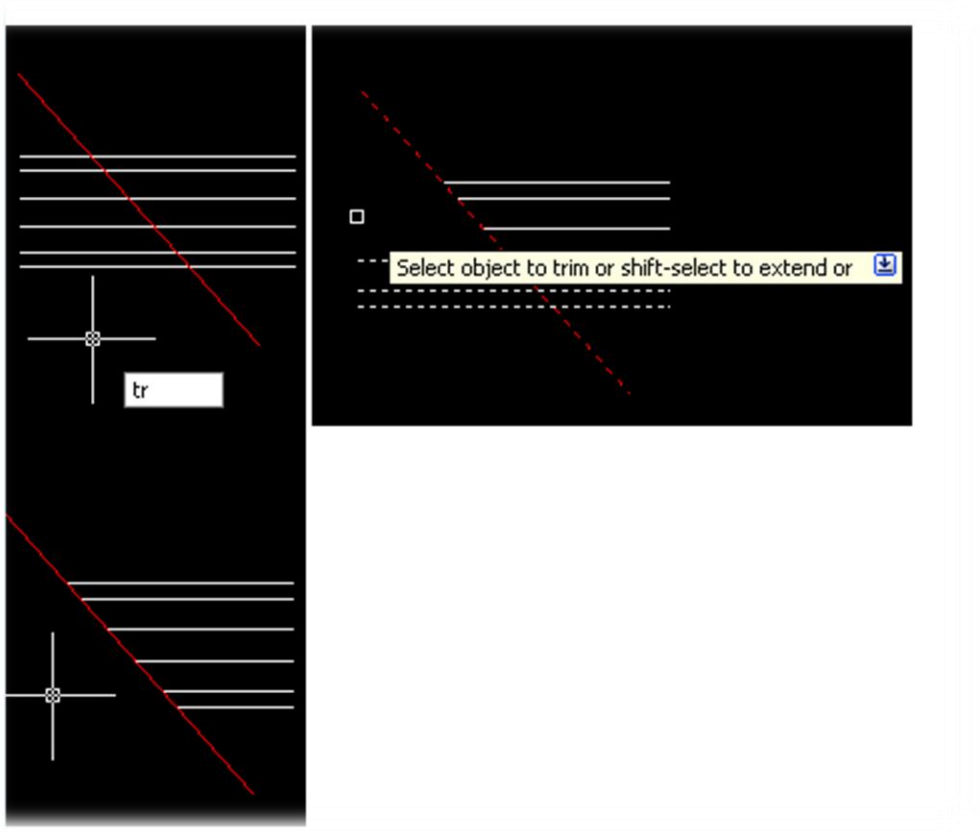
Create a duplicate object parallel with the original object. If this object is a Polyline or a Circle, the duplicate shape will be transformed inwards or outwards. This option can be useful to make closed steel profiles.

To offset: First select the offset distance; [enter]; select the original object; specify on which side you want to offset.



Trim (tr)

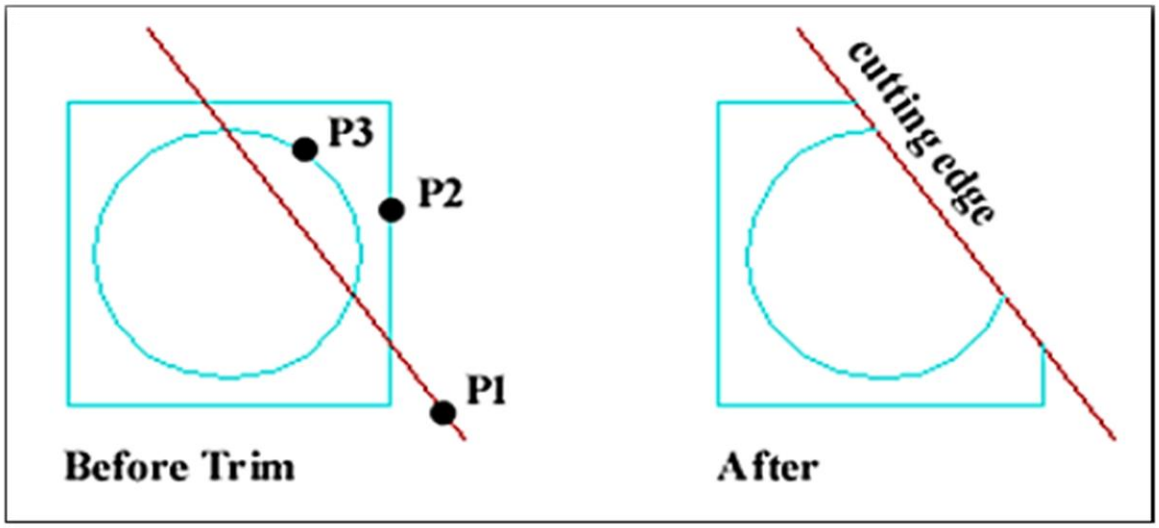
This command allows to trim (shorten) the end of an object back to the intersection of anther object. The middle section of an object can also be trimmed between two intersecting objects.Access the command by:



1. Menu bar: Modify → trim

2. Command line: trim or tr

3. Draw bar 



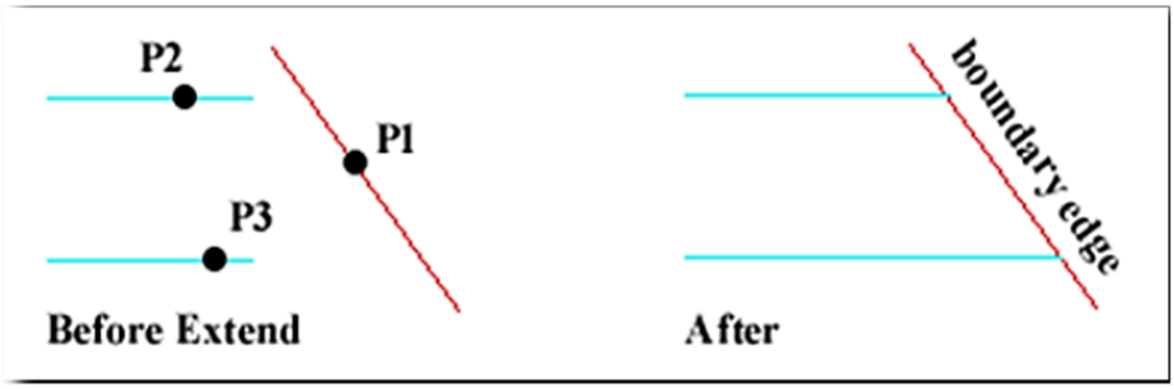
Extend (ex)

Extend command use to length object to meet another object. The objects can be extended include arc, ellipse, lines, Mlines, rays. Access the command by:

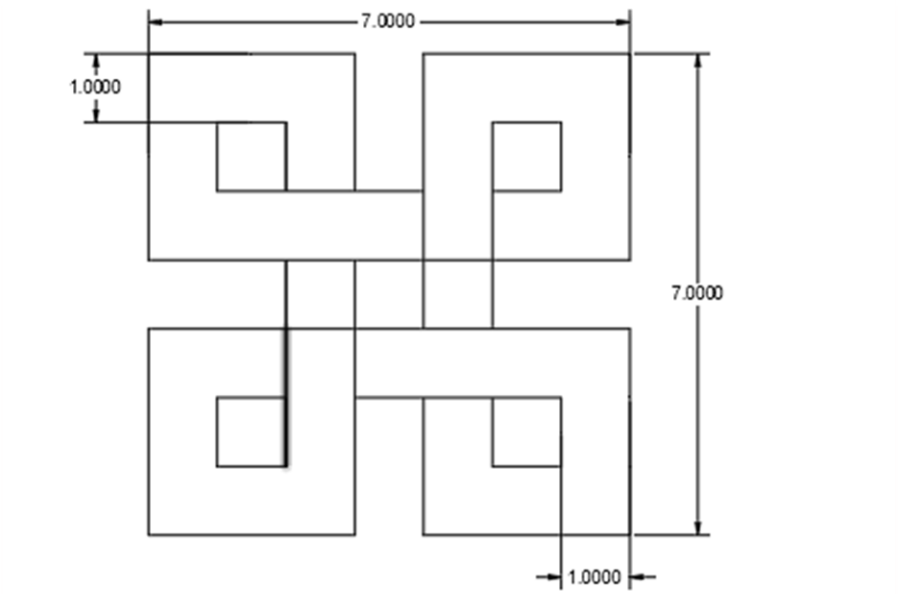
1. Menu bar:Modify ⟶ Extend

2. Command line: Extend or Ex

3. Draw bar: 



Example : Draw the following shape.



Department of Computer Technology Engineering

Engineering Drawing

Prepared by

Huda Majeed

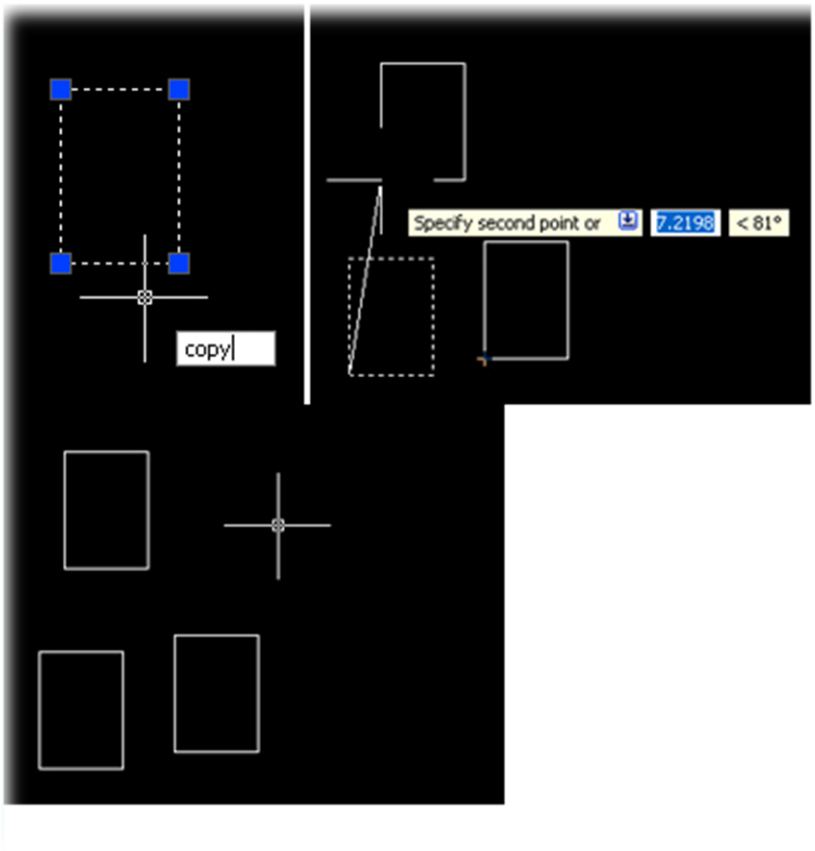
Copy (co)

Copy's one or more objects.

Method:

Select objects, press copy and specify a base point (in a similar fasion as with the move tool)

Now you can position the object in the same way as you would with a starting point of a line.



Array

The ARRAY command in AutoCAD is the magical command you will find yourself requesting a lot while working on AutoCAD. This command helps create a pattern of objects ranged in the following manner:

Polar arrangement

Rectangular arrangement

Path arrangement

These features are respectively represented by the following activation keywords:

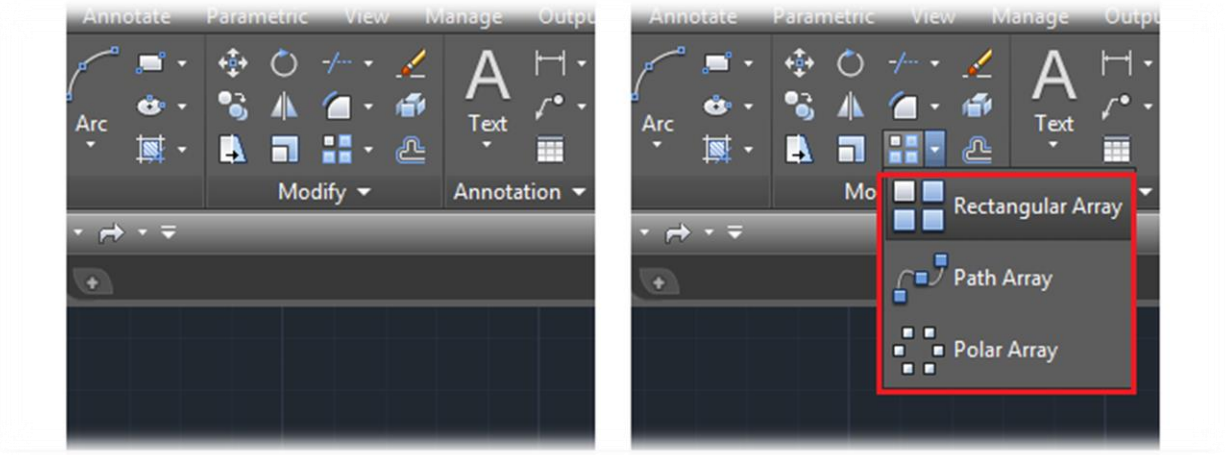
ARRAYRECT

ARRAYPOLAR

ARRAYPATH

Where the ARRAYRECT will help distribute identical objects in rows and columns, the ARRAYPOLAR will help distribute objects in a polar pattern around a center point and the ARRAYPATH will help distribute objects along a path.

You can find all these commands under the Modify section of the ribbon Panel.



Polar Array

The Polar Array is the most used of these and the command corresponding to it is ARRAYPOLAR (If you type ARRAYPOLAR and hit ENTER it will activate this command)

to use the Polar Array command

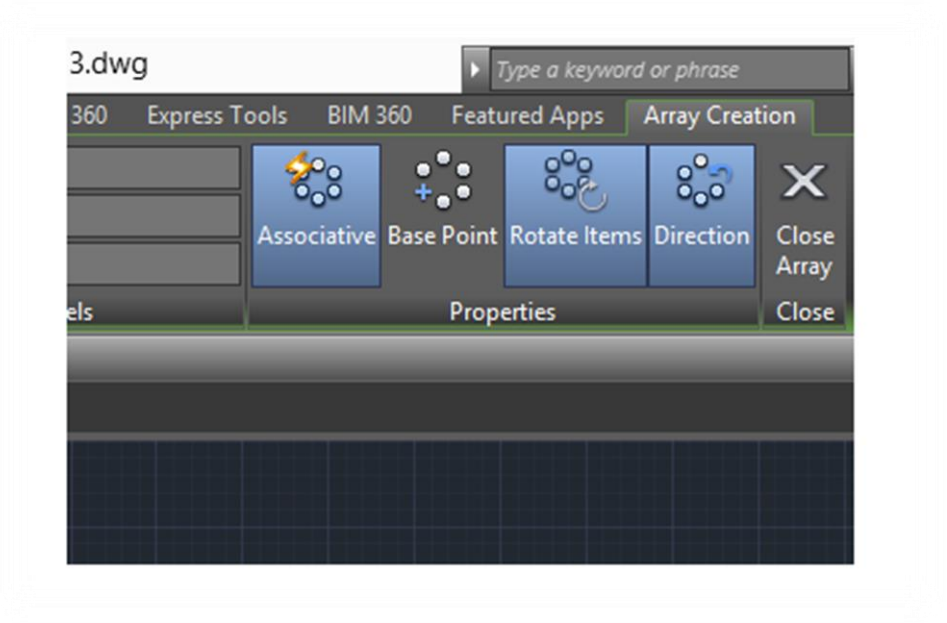
1. Click on its icon (*shown above*) or type **ARRAYPOLAR** and hit Enter
2. Select the reference object(s) and hit Enter
3. Specify the center point of the array

You will then have access to the following tools

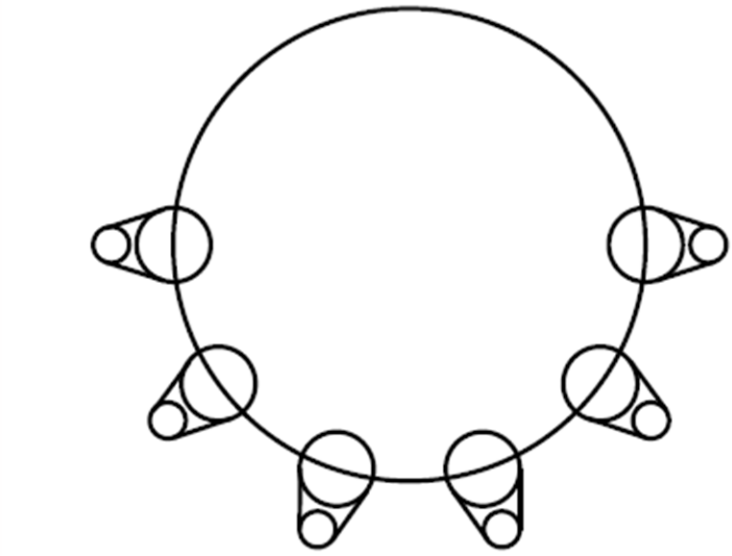
As you can see in the image of the polar array we created above, we have 6 elements. You can Enter the number of items you need in the provided box shown on the image below.

polar array in autocad

The maximum fill angle is 360. You can decide to rotate items on the circle or not to rotate them. (Fig 2 shows rotated items and Fig 3 shows item not rotated), The Direction features help you rotate items clockwise or counterclockwise, this makes more sense when the fill angle is less than 360 degrees. The Associative feature unites all items in one block.



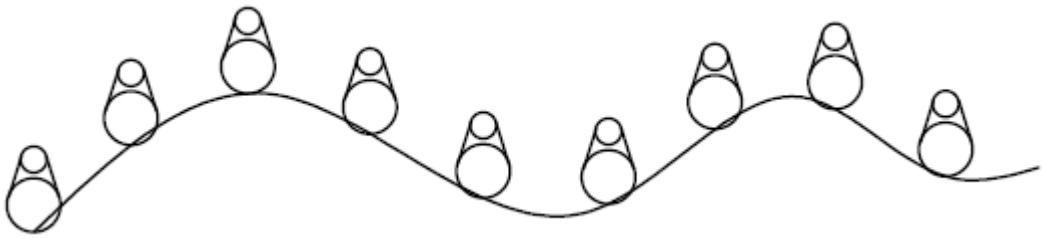
The following image shows a polar array with 6 items, fill on 180 degrees



Path Array

The Path Array corresponds to the command line ARRAYPATH.

It works just like the polar Array but will distribute objects along a path like on the image below



Rectangular Array

The rectangular array is defined as the arrangement of the object into rows, columns, and levels that form a rectangle.

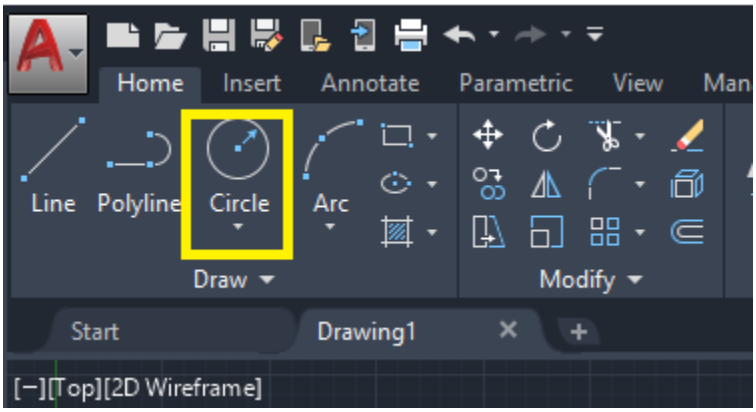
In AutoCAD, we can create multiple copies of an object in the form of a rectangle using the Rectangular Array.

Let's consider an example.

The steps to create a rectangular array are given below:

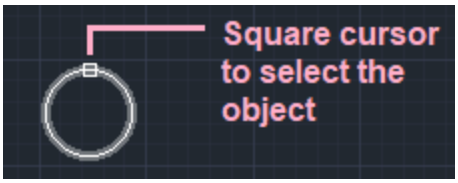
1. Open the AutoCAD software.

2. Click on the **Circle** icon on the Ribbon Panel, as shown in the below image:

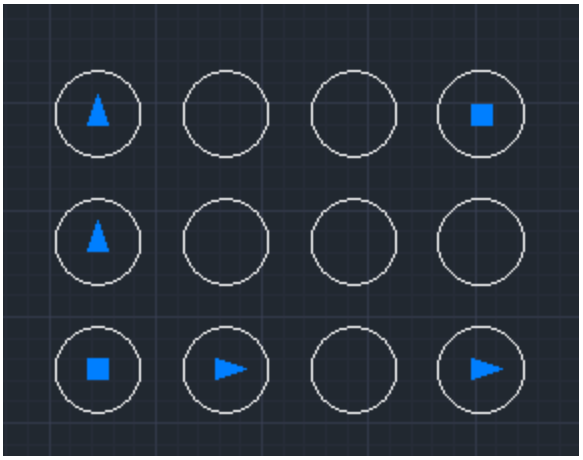


We can also write **Circle** or C on the command line.

3. Specify the center point on the workspace or drawing area to draw the circle.
4. Specify the diameter of the circle. Here, we have specified the value 1. We can provide any value of diameter according to the requirements.
5. Select the **Rectangular Array** from the ribbon panel.
6. Select the object with the small square cursor and Press **Enter**, as shown in the below image:

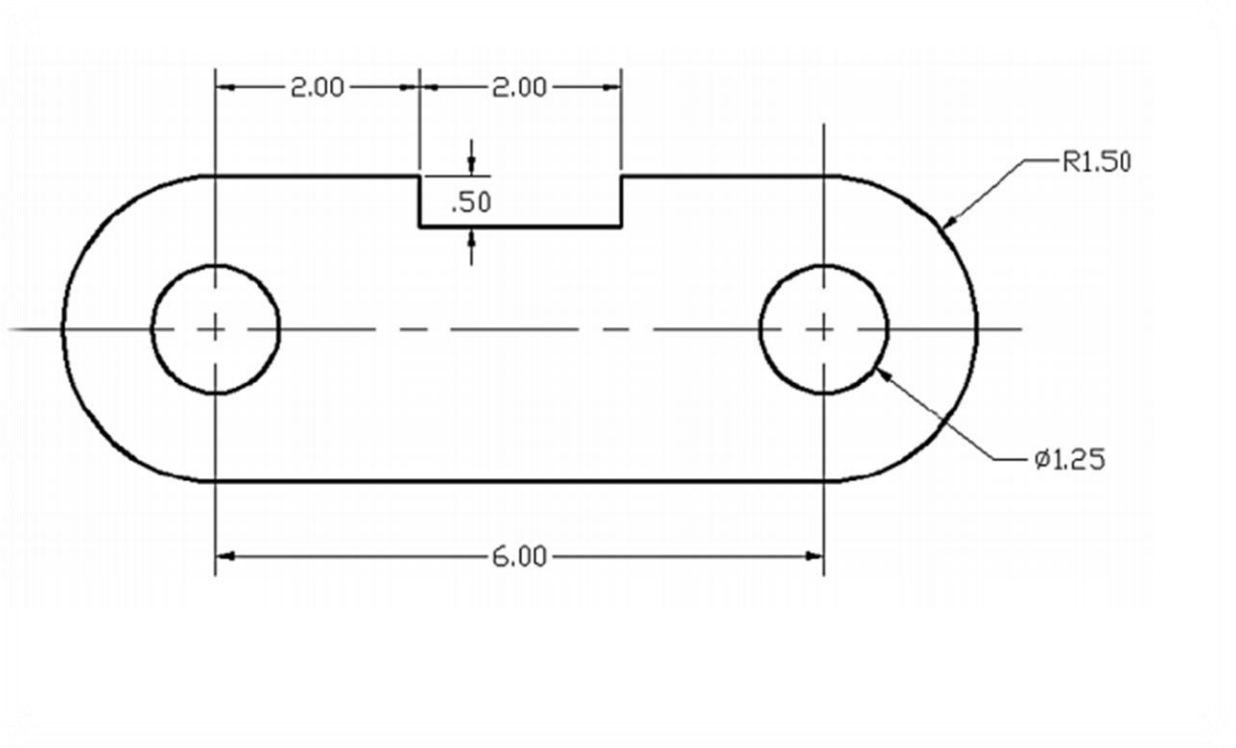


7. The default rectangular array will be created, as shown in the below image:



We can also type **AR** - select the object with the square cursor-
select **Rectangular** - and then Press **Enter** on the command line.

8. Press the **Esc** button on the top left of the keyboard to exit the array command. If you want to modify the rectangular array further, without exit, follow the steps given below.



Department of Computer Technology Engineering

Engineering Drawing

Prepared by

Huda Majeed

Move

Moves one or more objects:

Select the objects and specify the base point.

The base point is essentially the point where you 'grab' the objects. So if you input coordinates, the base point is the point where the coordinates relate to.

Moving an object can either be done with the aid of object snap, or by using relative coordinates.

Rotate

 **Rotate (ro)**

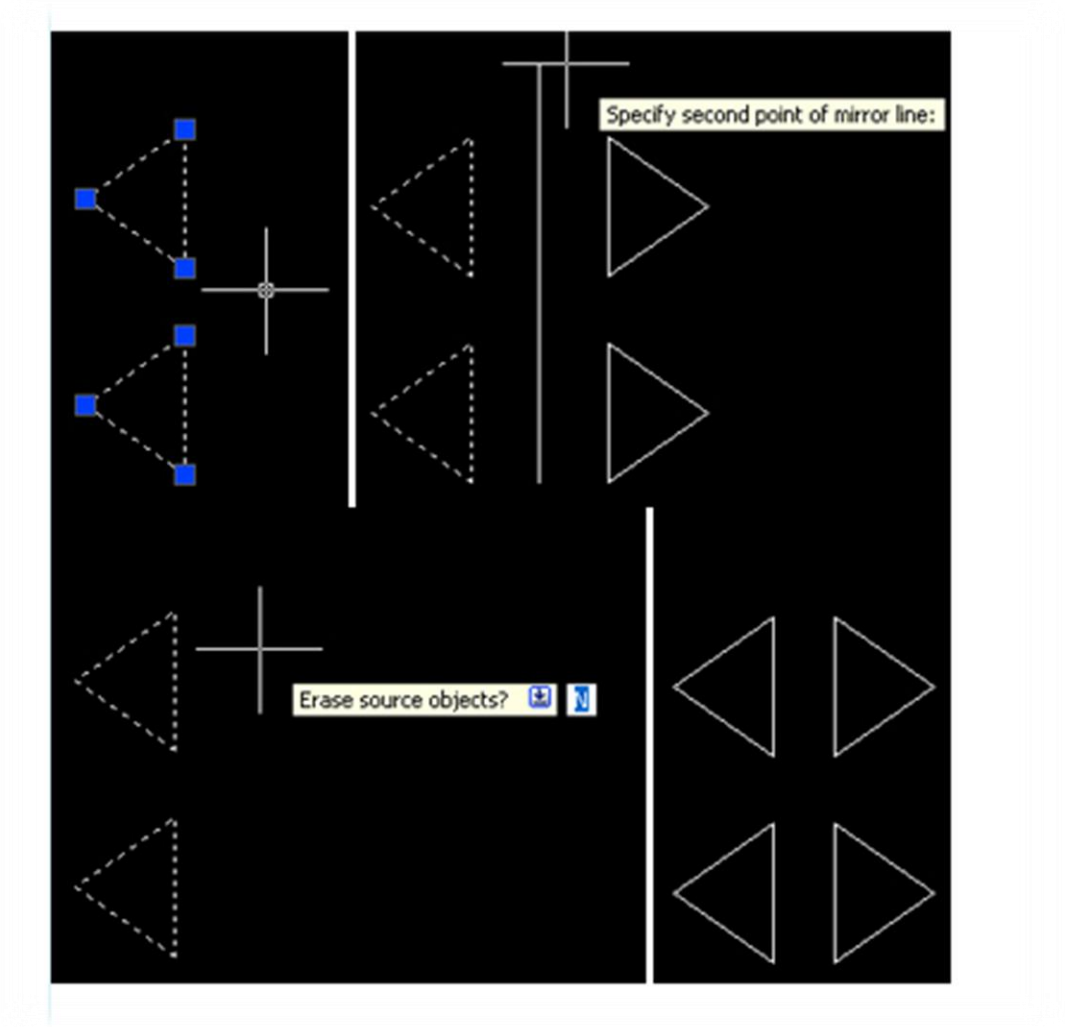
You can rotate objects with an absolute or relative angle. When using an absolute angle: Specify the base point and then specify the rotation angle.

Mirror

 **Mirror (mi)**

Create a mirror image of a object. It is useful for creating symmetrical objects because you can quickly draw half the object and then mirror it instead of drawing the entire object.

You flip the object about an axis called a mirror line to create a mirror image. First select the object. To specify the temporary mirror line, you enter two points. You can choose whether to delete [y] or retain the original [n] object.



Scale

Scale (sc)

To scale an object you can specify a base point and a length, which will give a scale factor. A scale factor greater than 1 enlarges the object. It is also possible to scale an object using a reference object. This method scales the object equally in all directions.

Scaling using a scale factor: Select the object; type sc in the command line; scale factor; [enter]

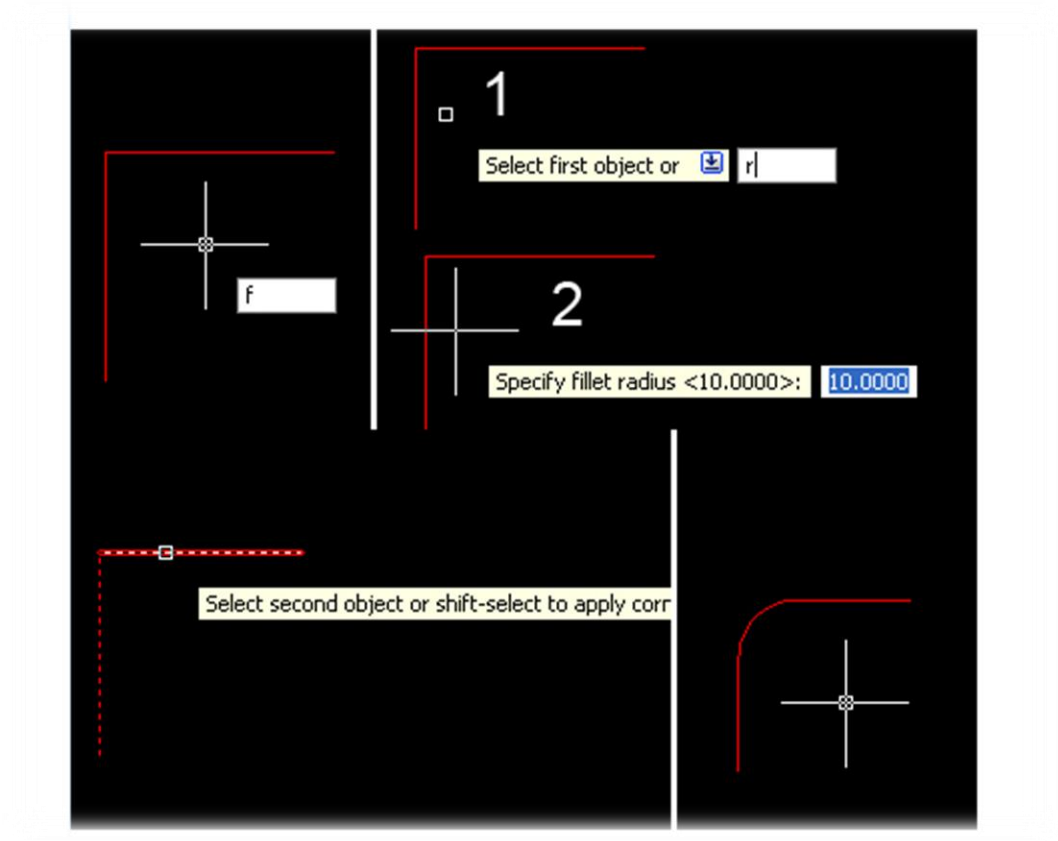
Scaling using a reference: Select the object; type sc in the command line; specify base point; choose r to use reference; specify the reference length of the original object; specify the new length of the original object.

Fillet

Fillet (f)

You can use the fillet tool to connect two objects with an arc with a specified radius. The inside corner is called a fillet and an outside corner is called a round.

To fillet: type f in the command line; type R for the radius (optional); specify the radius; [enter]; select the first line; select the second line.



Chamfer is almost identical, but it will will make a straight line instead of an arc.

Explode

 **Explode (ex)**

Polylines, hatches or blockes can be converted into individual elements with the explode option.

If you explode a polyline every segment will become a separate line.

To Explode a block: First select the block; type ex; [enter].

Department of Computer Technology Engineering

Engineering Drawing

Prepared by

Huda Majeed

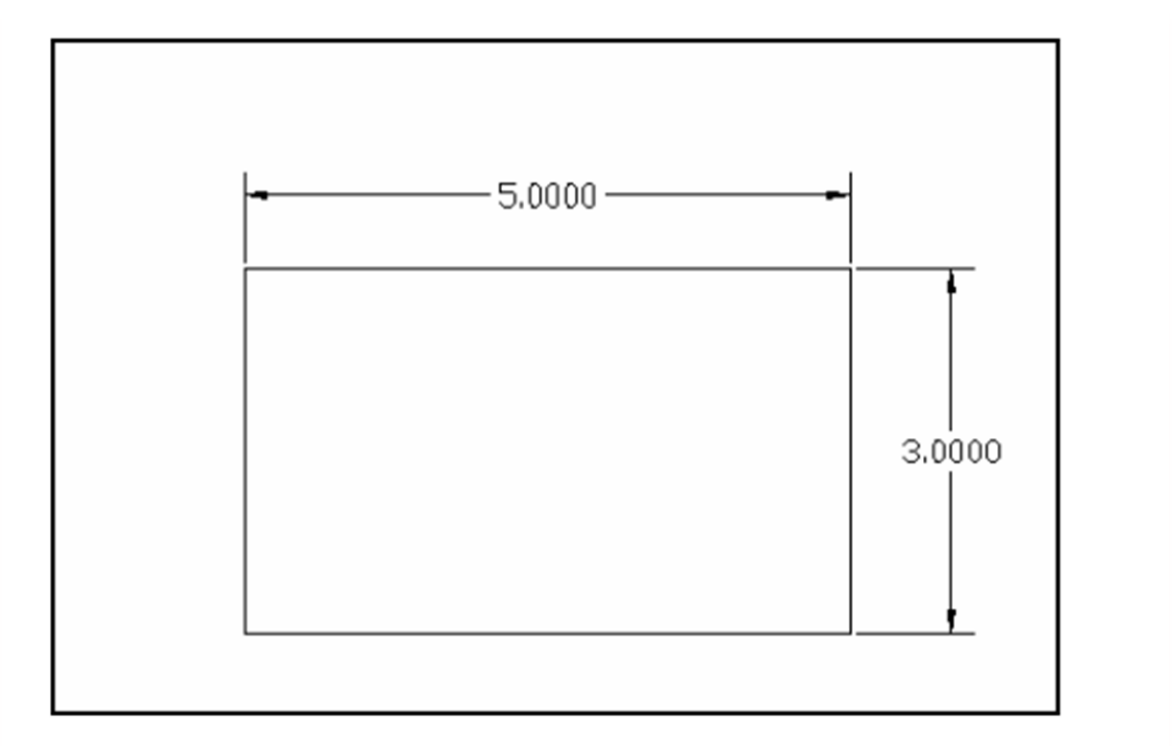
Linear Dimensions

Choose Dimension, Linear.

or 2. Click the Linear Dimension command from the toolbar.

or 3. Type DIM at the command prompt.

Command: DIM Dim: HOR or VER



Aligned Dimensions

1. Choose Dimension, Aligned.

or

2. Click the Aligned Dimensioncommand from the

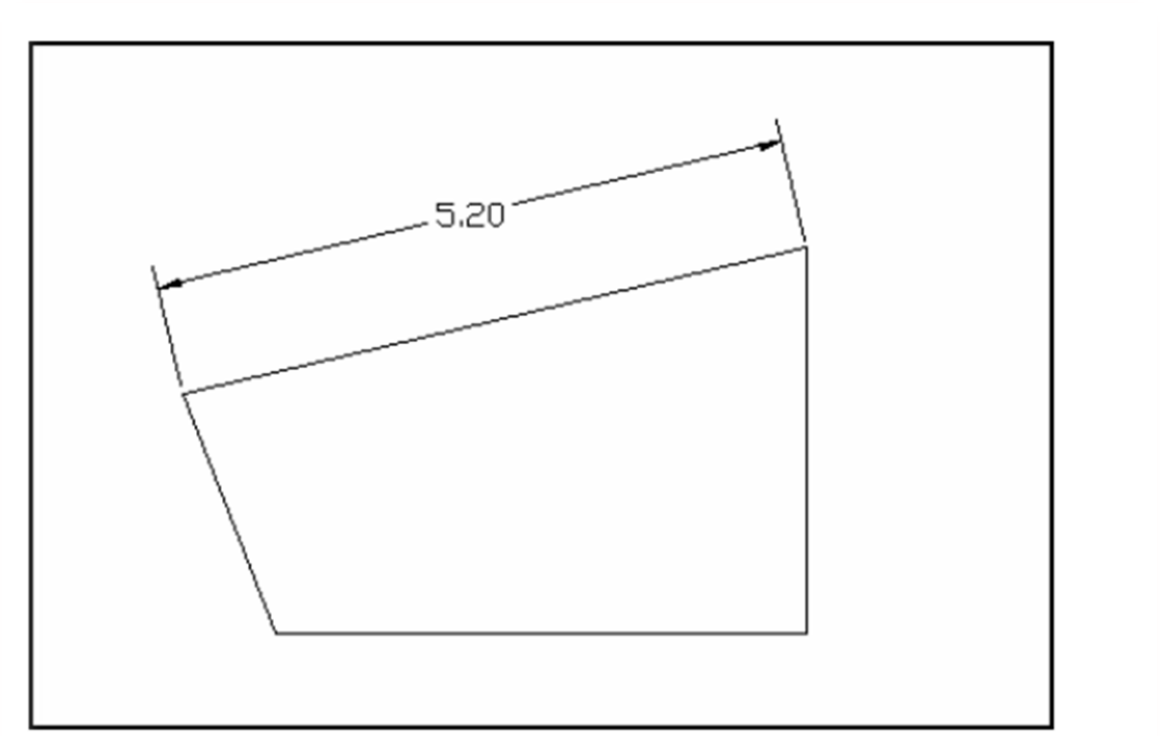
toolbar.

or

3. Type DIM at the command prompt.

Command: DIM

Dim: ALIGNED

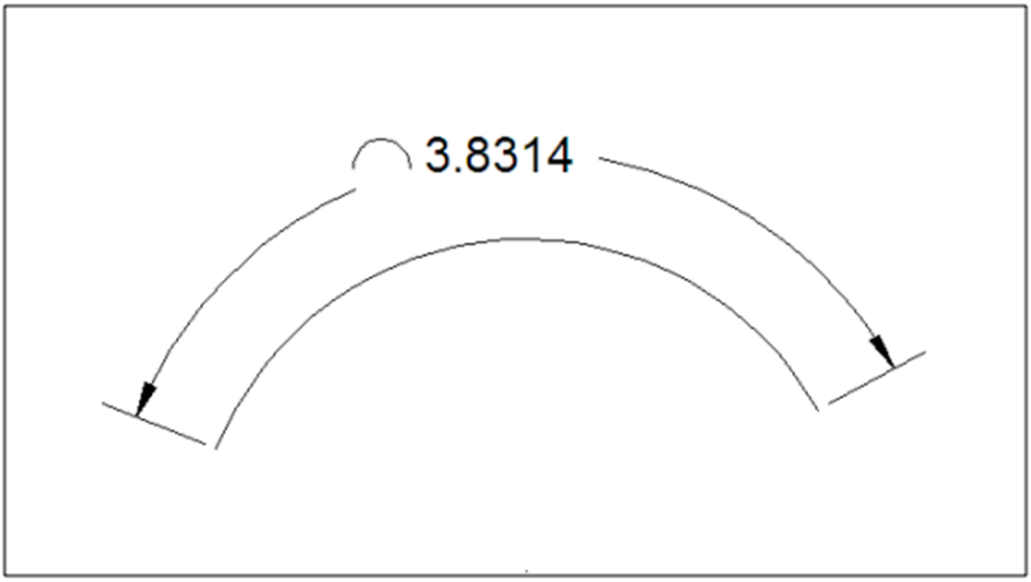


Arc length Dimensions

1. Choose Dimension, Arclength.

or

2. Click the ArcLength Dimension command from the toolbar.



Radial Dimensions

1. Choose Dimension, Radius or Diameter.

or

2. Click the Radial Dimensions command from the

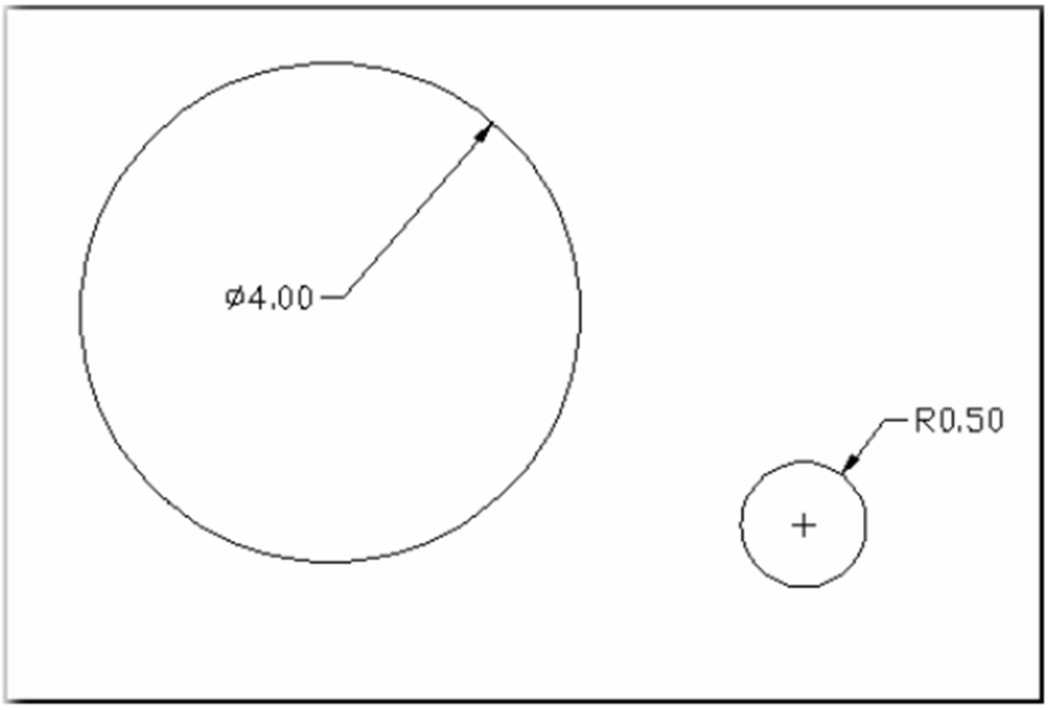
toolbar.

or

3. Type DIM at the command prompt.

Command: DIM

Dim: RADIUS or DIAMETER


ø4.00
R0.50

-

Department of Computer Technology Engineering

Engineering Drawing

Prepared by

Huda Majeed

The text in AutoCAD

To create a text

A- Create Multiline Text

For longer notes and labels with internal formatting, use multiline text.

Click Home tab ⟶ Annotation panel ⟶ Multiline Text.

1-Specify first corner of abounding box.

2-Specify opposite corners of a bounding box to define the width of the multiline

3-Enter the text.

4- To change individual characters, words, or paragraphs, highlight the text and specify the formatting changes.

5-To save your changes and exit the editor, use one of the following methods:

- Click OK on the Text Formatting toolbar.
- Click in the drawing outside the editor.
- Press Ctrl + Enter.

B-Create Single-line Text

For short, simple notes and labels, use single-line text.

1-Click Home tab ⟶ Annotation panel ⟶ Single Line Text.

2-Specify the insertion point.

If you press ENTER, the program inserts the new text immediately below the last text object you created, if any.

3-Enter a height or click to specify the height of the text.

4-Enter an angle value or click to specify the rotation angle.

5-Enter the text.

6-Press ENTER on a blank line to end the command.

Introduction to Layers and Layer Dialog Box

1. Choose Format, Layer.

or

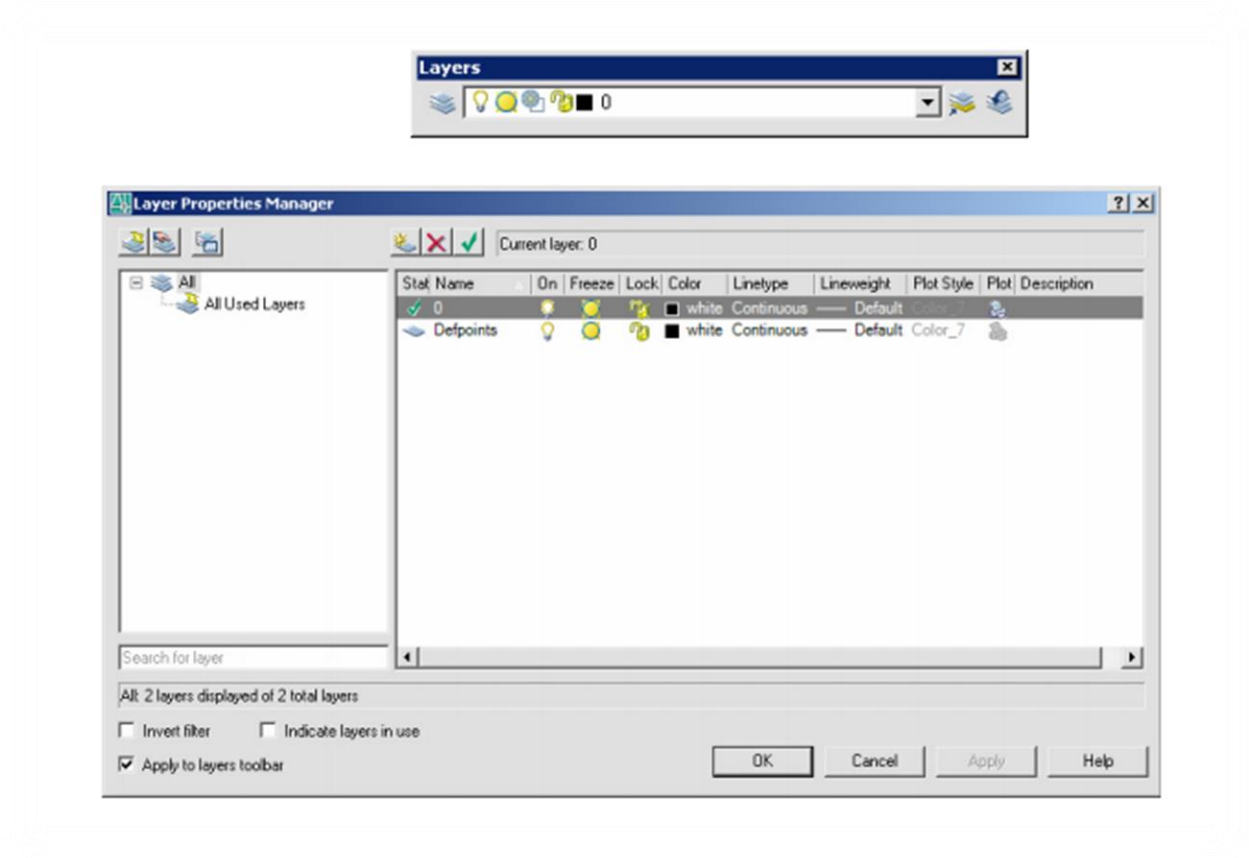
2. Type LAYER at the command prompt.

Command: LAYER (or LA)

or

3. Pick the layers icon from the Layer Control box

on the object properties toolbar



make: Creates a new layer and makes it current.

Set: Sets current layer.

New: Creates new layers.

ON: Turns on specified layers.

OFF: Turns off specified layers.

Color: Assigns color to specified layers.

Ltype: Assigns linetype to specified layers.

Freeze: Completely ignores layers during regeneration.

Thaw: Unfreezes specified layers Ltype.

Lock: Makes a layer read only preventing entities from being edited but available visual reference and osnap functions.

Unlock: Places a layer in read write mode and available for edits.

Plot: Turns a Layer On for Plotting

No Plot: Turns a Layer Off for Plotting

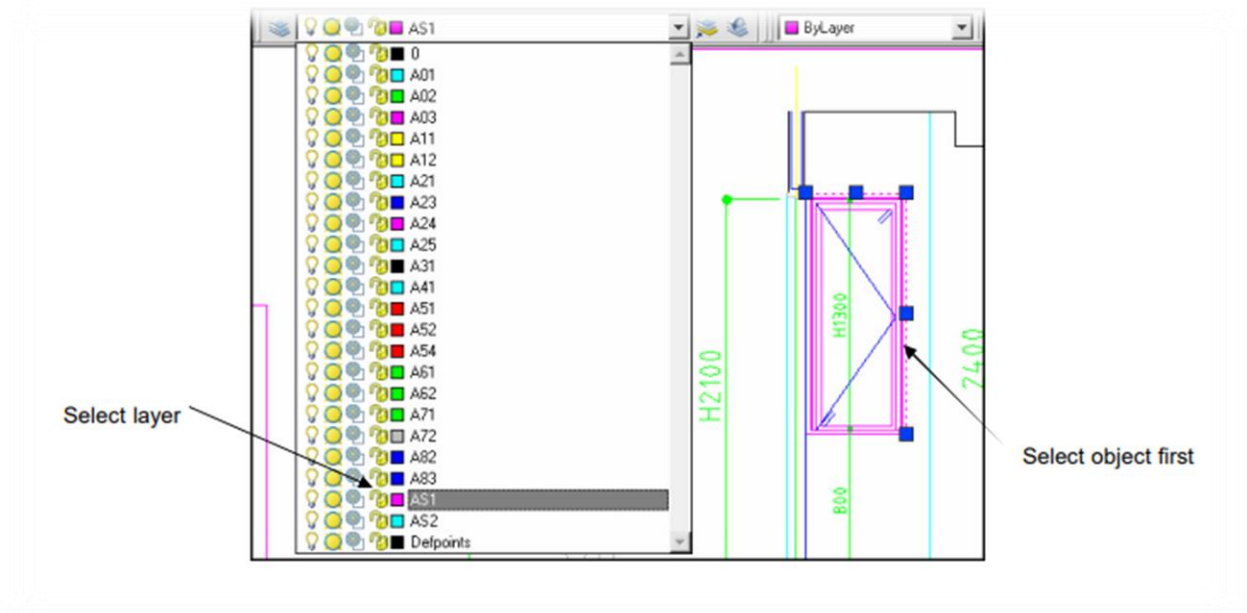
LWeight: Controls the line weight for each layer

Changing the Layer of an Object

1. Click Once on the object to change.

2. Select the desired layer from the Layer Control Box dropdown.

AutoCAD will move the object to the new layer.



Making a Layer Current

1. Click once on the Make Object's Layer Current icon.

2. Select object whose layer will become current:

Lineweights

Loading and Changing Lineweights

1. Choose Format, Lineweight...

or

2. Type LINEWEIGHT at the command prompt.

Command: LINEWEIGHT or LWEIGHT

or

3. Pick a lineweight to make current from the Object Properties menu.

Linetypes

Loading and Changing Linetypes

1. Choose Format, Linetype...

or

2. Type DDLTYPE at the command prompt.

3. Choose Load... to see a list of available linetypes.

4. Choose the desired line type to assign.

5. Click OK.