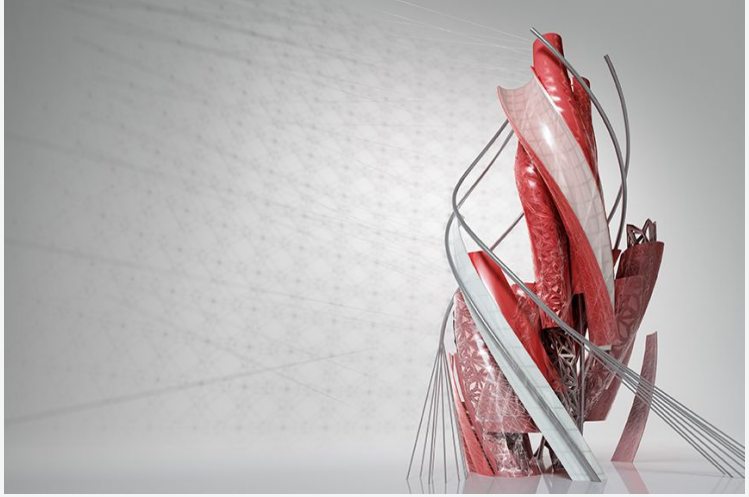
AUTOCAD



AutoCAD Command

LIMIT

Limits

use to specify drawing area.

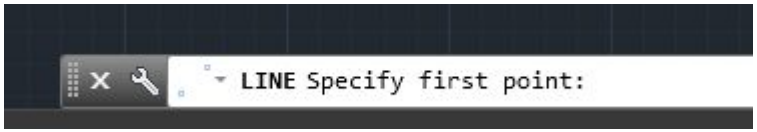
- Limits □
  - First corner point : 0,0 □ (X,Y)
  - Second corner point : 100’,100’ □

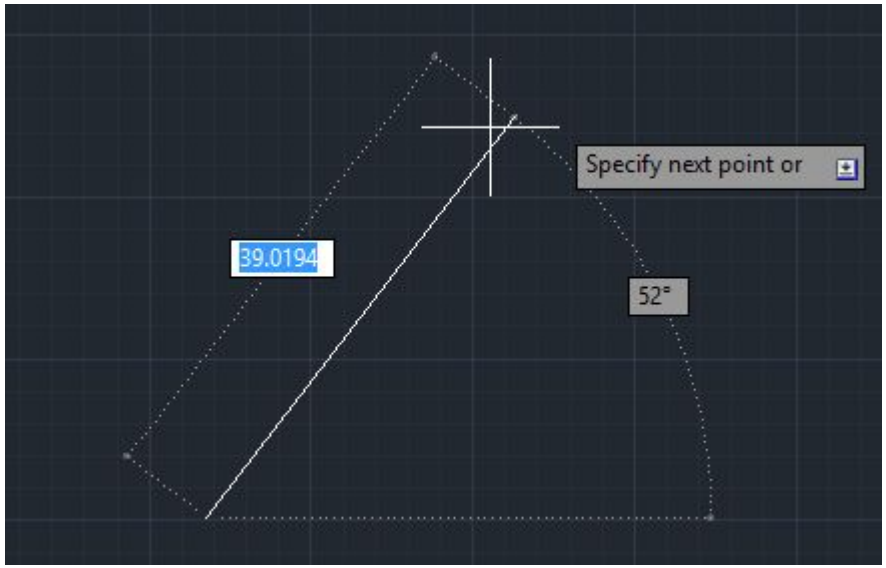
LINE

Line

use to make line

- Command: L
  - Specify First Point
  - give distance value
  - Or (Click on next Point)





DIMLINER

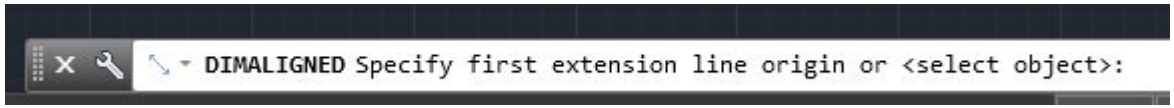
Use to measure horizontal line

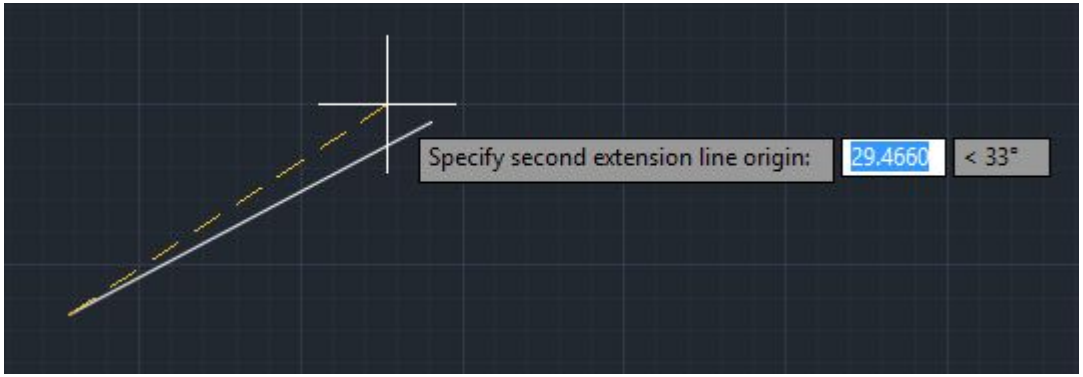
- Command: DIMLIN □
- Select first extension line origin
- Select second extension line origin

DIMALIGNED

Use to measure vertical or aligned line

- Command: DIMALIGNED □
- Select first extension line origin
- Select second extension line origin

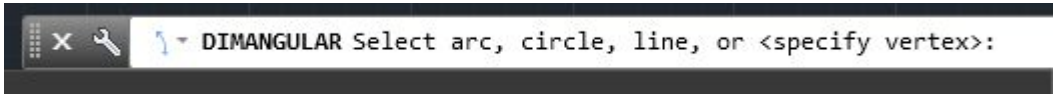


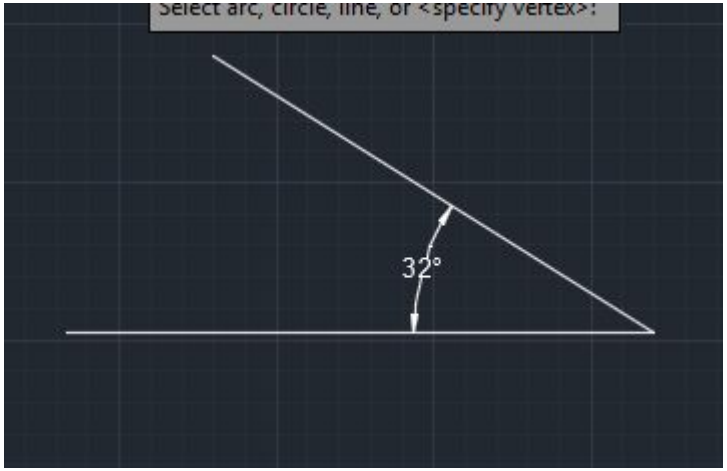


DIMANGULAR

Use to measure angle between any two line

- Command: DIMANG □
- Select first line
- Select second line.

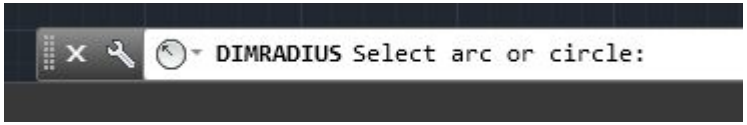


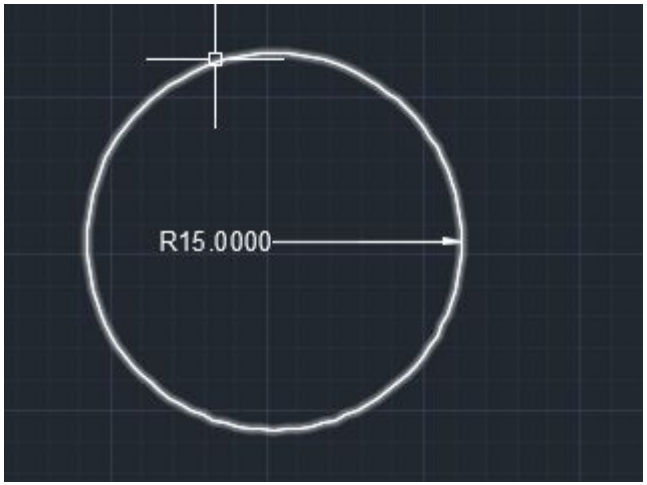


DIMRADIUS

Use to measure radius of circle.

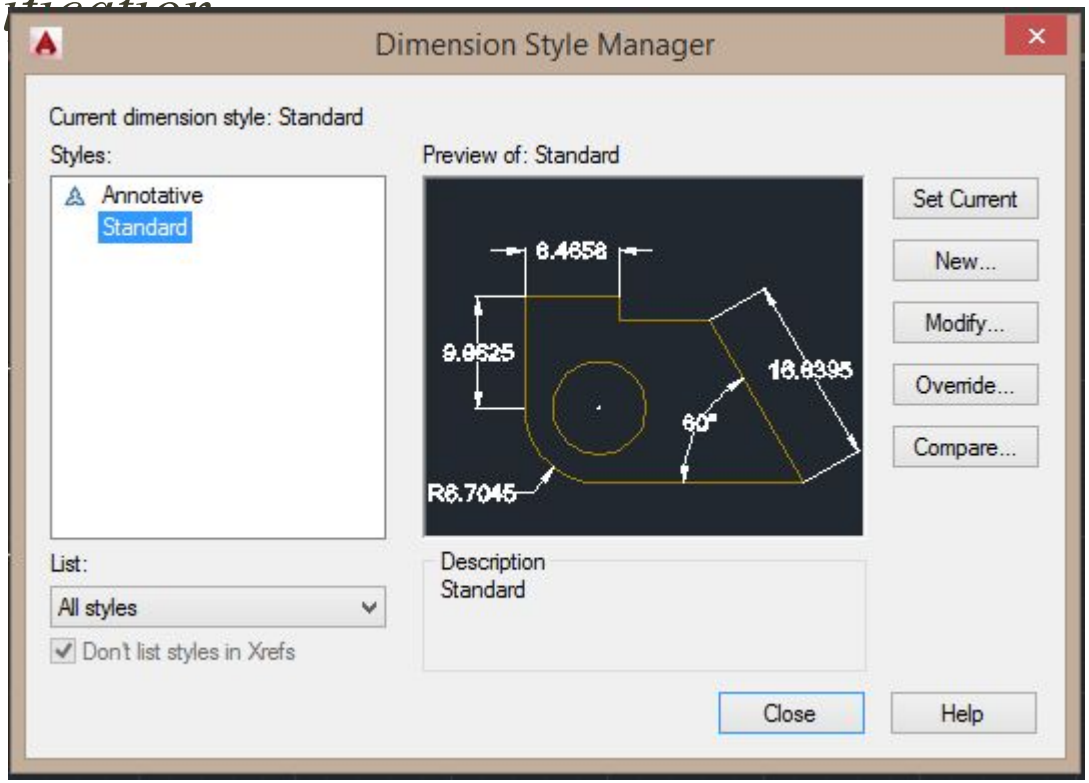
- Command: DIMRAD
- Select arc or radius.





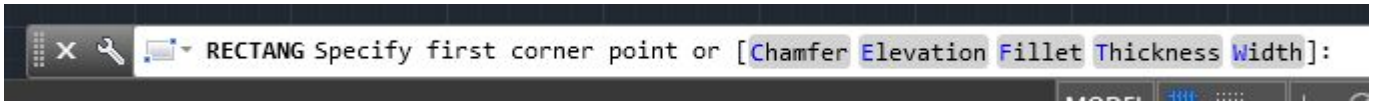
DIMSTYLE

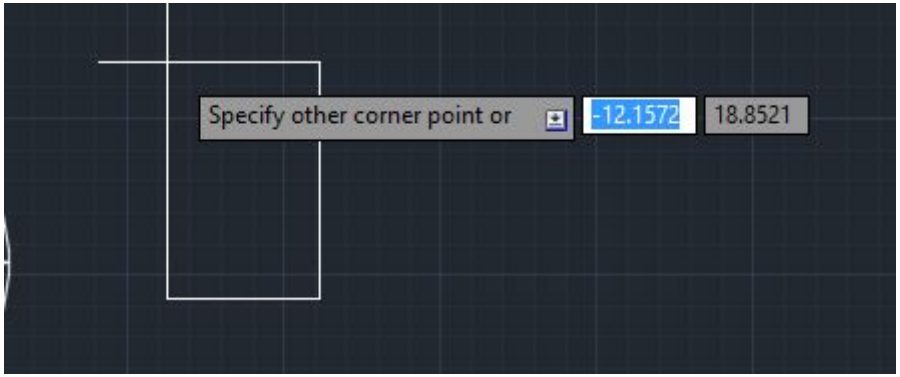
- **To Modify the Dimension Style**
- *Command: D*
- *Select the Dimension Style*
- *Click Modify*
  - *change the style modification*
  - *Ok*
  - *Set current*
  - *Close*

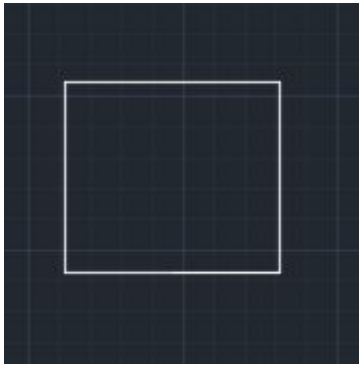


RECTANGLE

- Rectangle
- Use to make rectangle
  - Rec
    - Specify 1st Corner for Rectangle
    - D (For Dimension)
    - Define the length value
    - Define the width value
    - Click on screen place the Rectangle



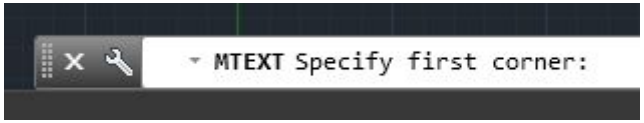


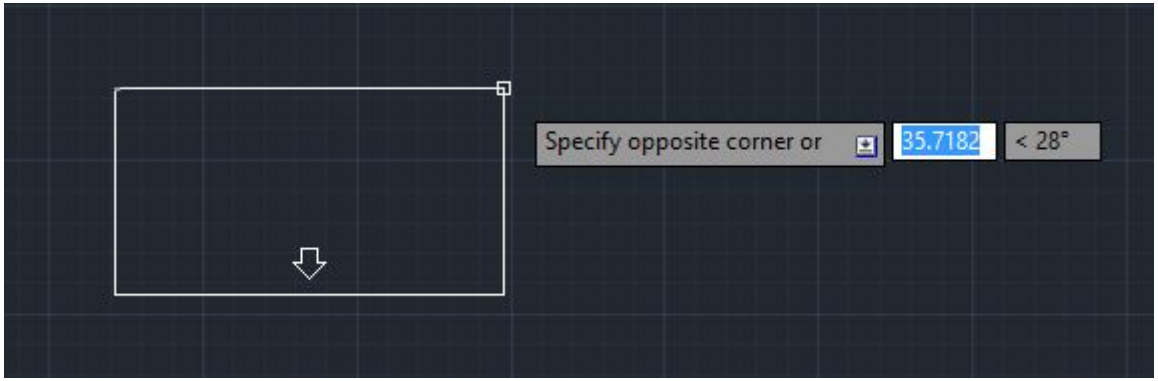


MULTILINE TEXT

- For Multiline Text
  - Mt↵
  - Specify first corner for text window
  - H↵ (Height of text)
  - W↵ (width of the writing space)
  - Write the required text
  - Press “Close Text Editor” to terminate from command.

Note : We can apply all text editing tools as we do in Word files

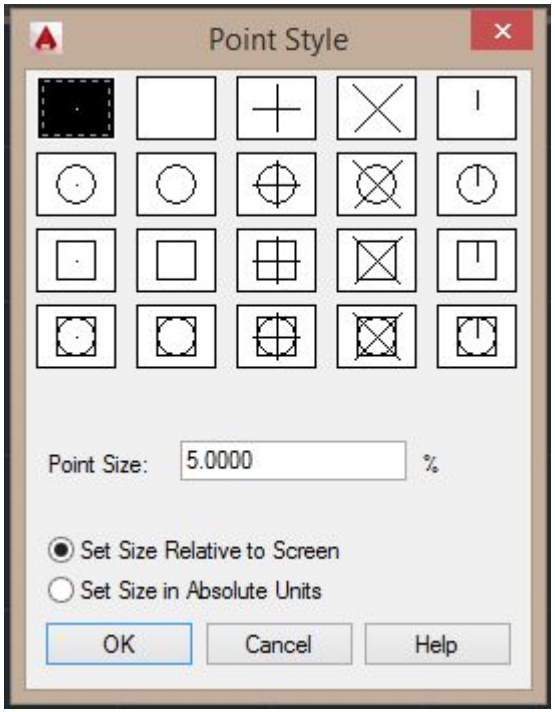


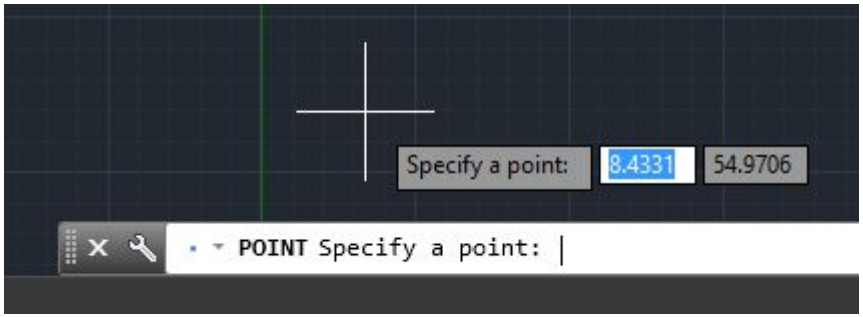


POINT

POINT & POINT STYLE

- FOR POINT
  - Command: Po ↵
  - Specify a point
  - Draw menu →Format →Point style

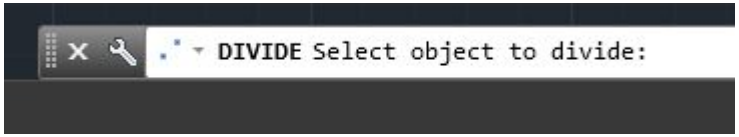


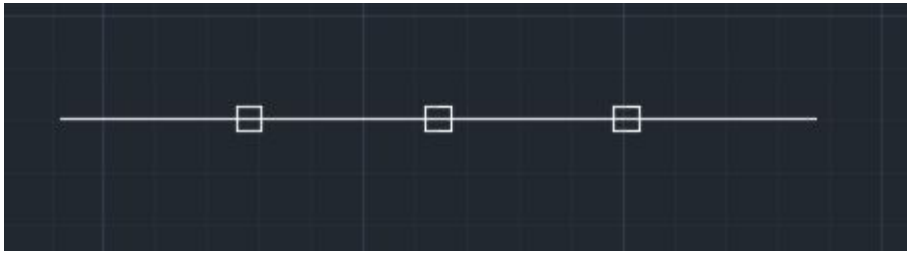


DIVIDE

Use to divide any object to number of segment

- Command: Div ↵
- Select object
- Give no of segments



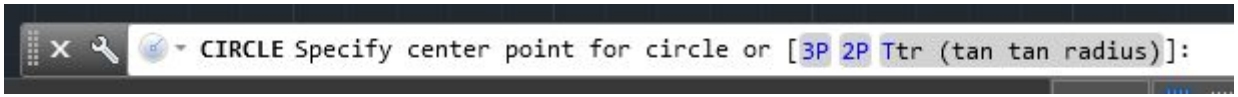


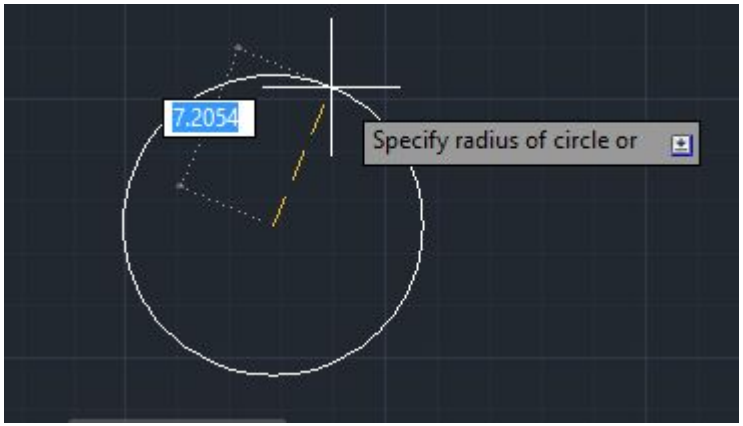
CIRCLE

❖Circle

Use to make circle

- Command: C □
  - Click Center Point
  - Specify Radius value □ (Dimension -D □)

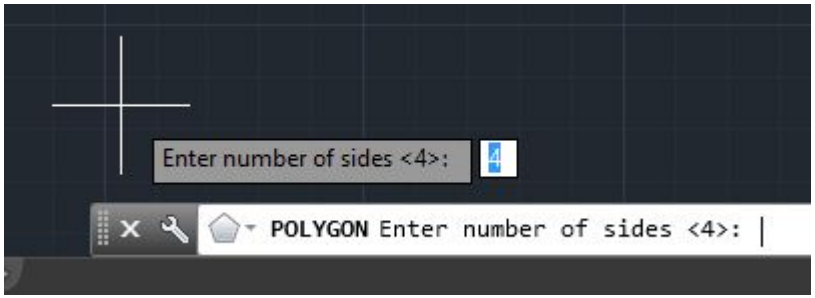


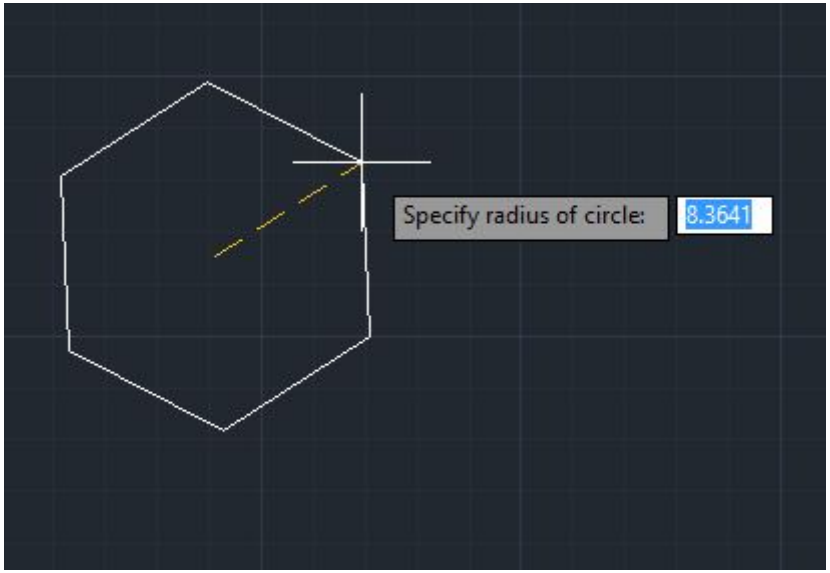


POLYGON

- Use to create polygon of any side

- Command: Pol □
- Give number of sides.
- Specify center of polygon.
- Choose option inscribed in circle or circumscribed about circle

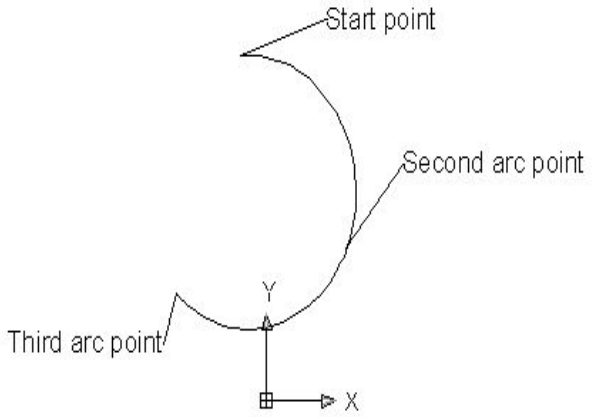


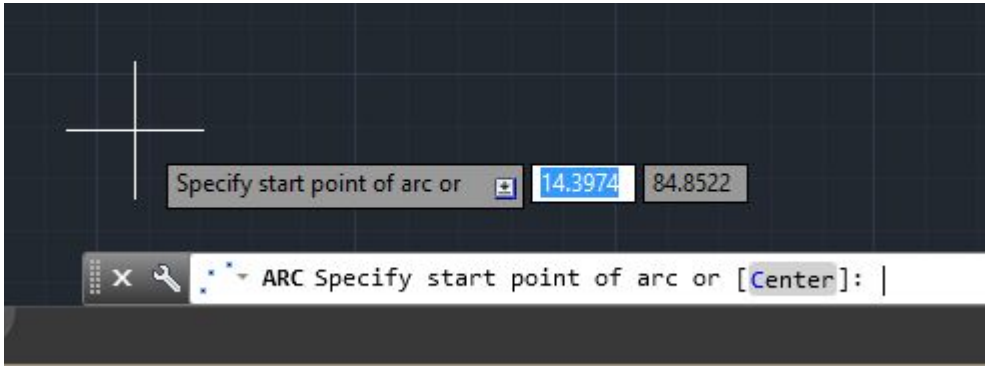


ARC

Use to make arc

- Command: A ↵
- Specify start point
- Specify second point of arc
- Specify end point of arc

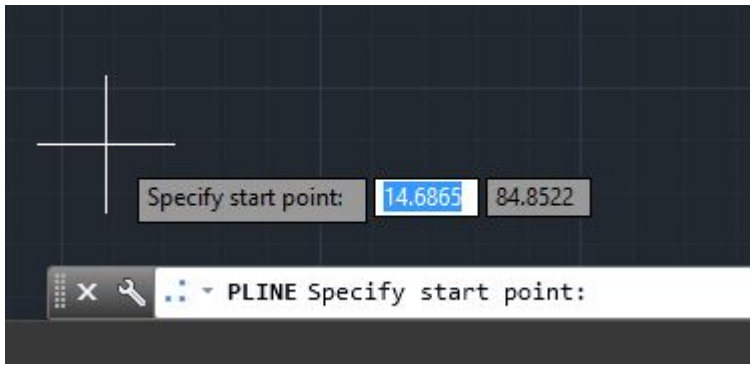


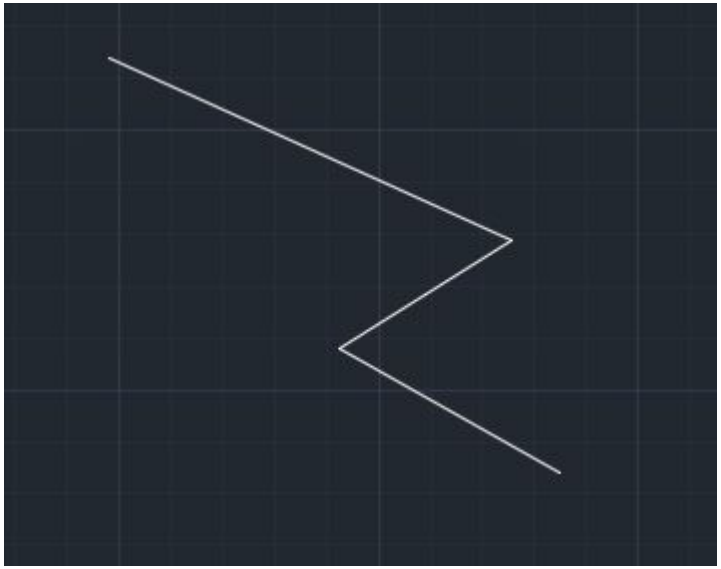


POLYLINE

Use to make different lines with single entity

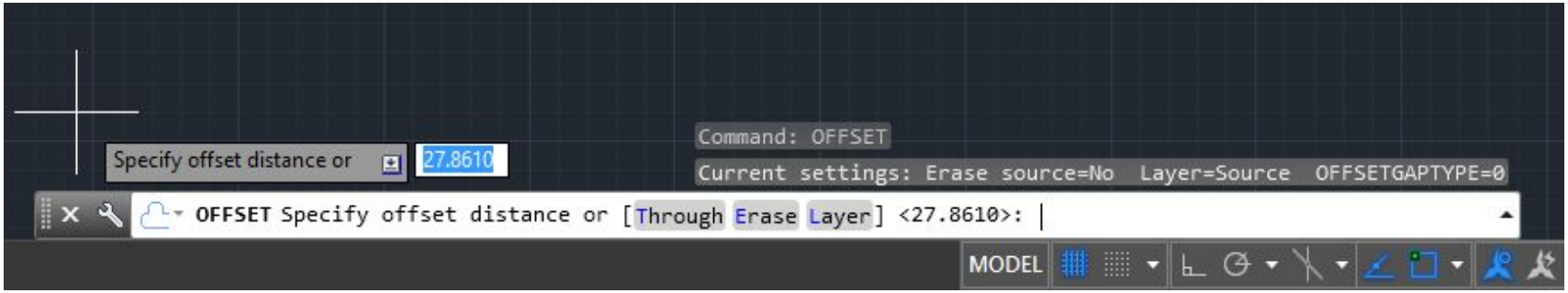
- Command: Pline
- Select first point.
- Select next point.
- Select next point till wee want to make line
-

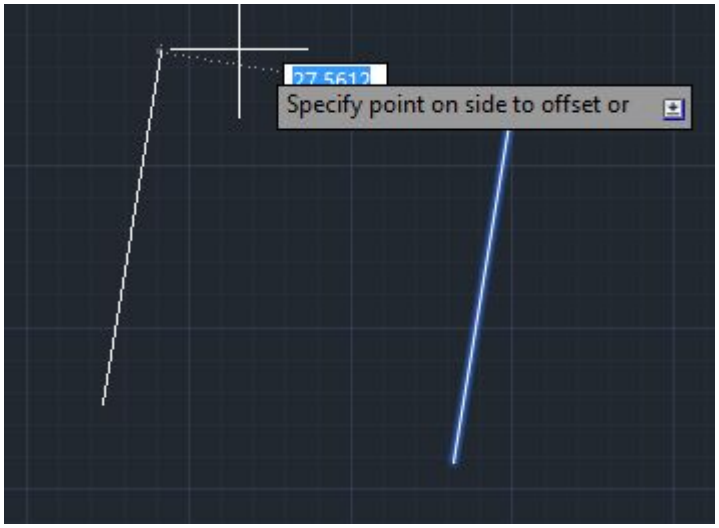




OFFSET

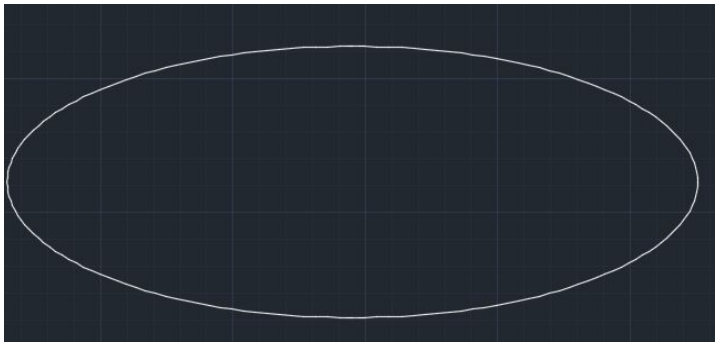
- Use to make parallel object at the given distance
  - Command: O
  - Specify Offset Distance
  - Select object
  - specify distension / direction



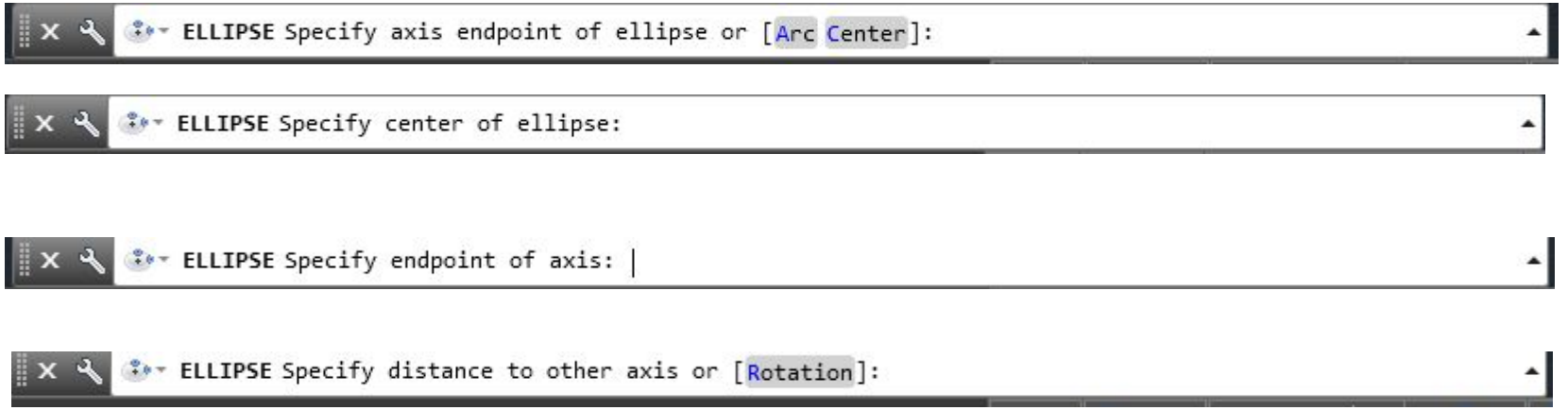


ELLIPSE

- Command: EL↲→C↲
- specify the center point
- define the 1st radius value ↲
- define the 2nd radius value ↲



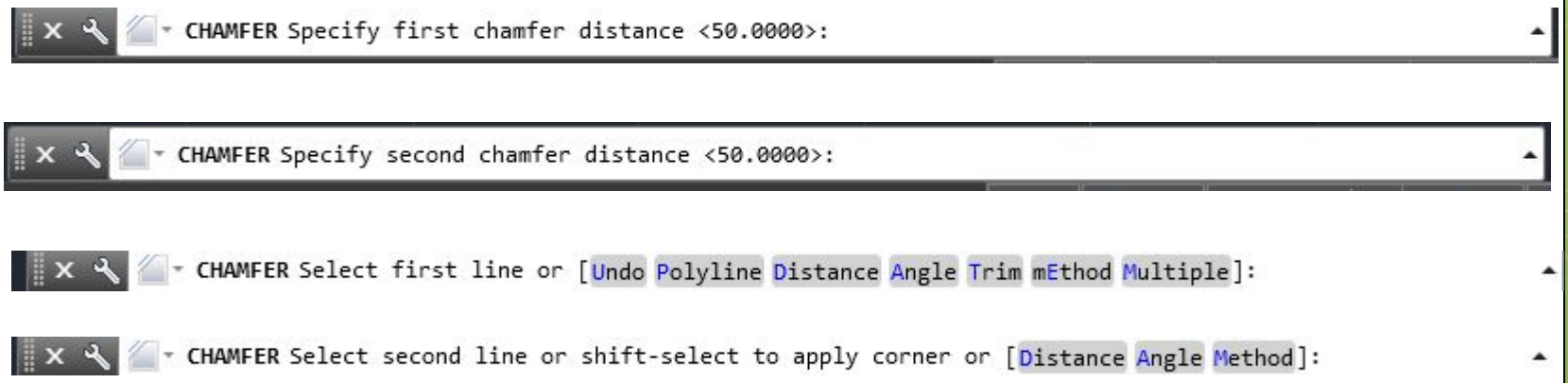
- Use – to make ellipse.



CHAMFER

- Command: Cha ↵
- D ↵ (For Chamfer Distance)
- Specify first chamfer distance ↵
- Specify second Chamfer distance ↵
- Select first line to chamfer
- Select second line to Chamfer

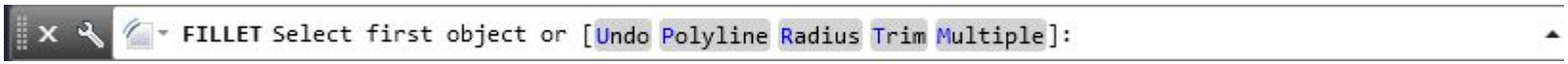
- Use – For chamfering the sharp corner of the objects

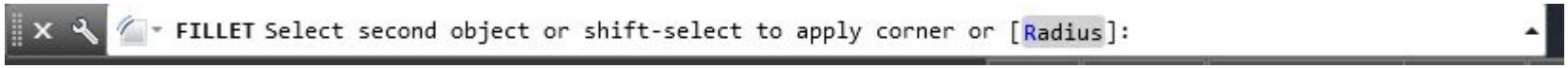


FILLET

- F ↲
- R ↲ (For radius of fillet)
- Specify fillet radius value ↲
- Select first object
- Select second object

- Use – To for making the corner round



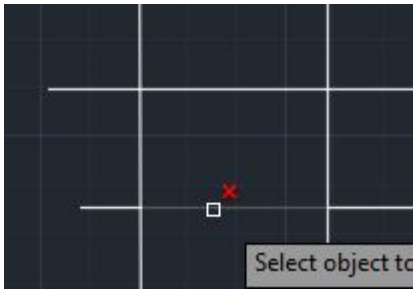


TRIM

- Command: Tr □ □

- Select the un necessary part.
- Note : profiles must be intersected

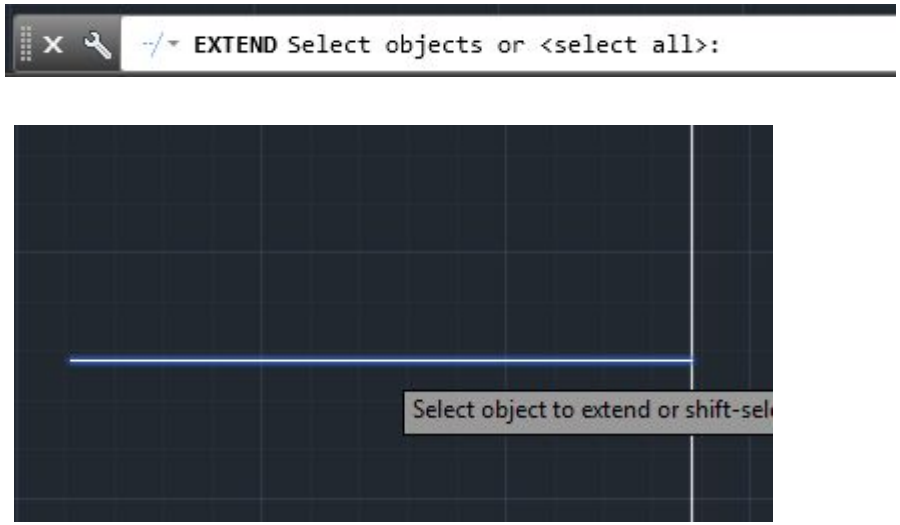
- Use- To cut unnecessary part



EXTEND

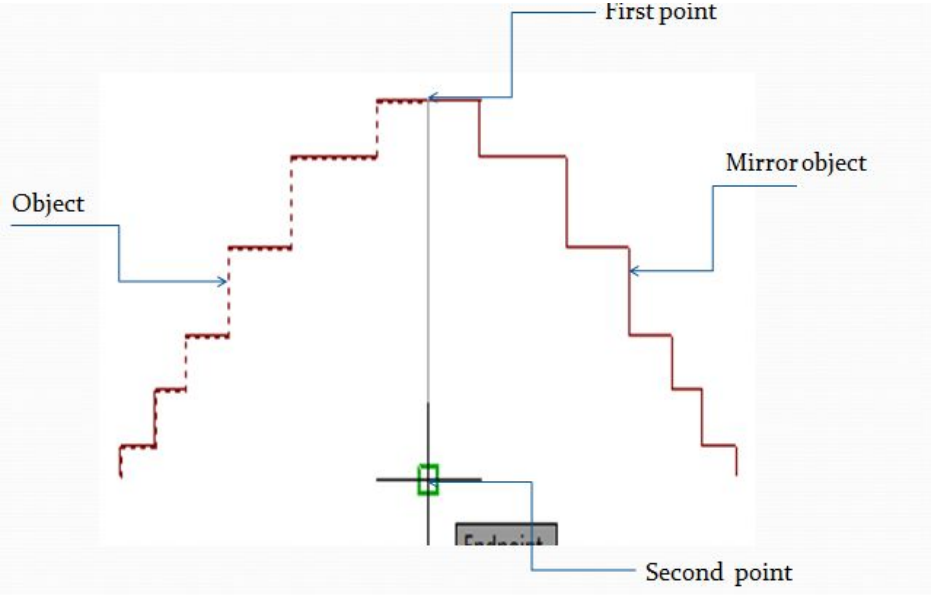
- Command: Ex □ □
-
- Select the line to extend
- Note : There must be a boundary.

- Use – To lengthen object to fix boundary



MIRROR

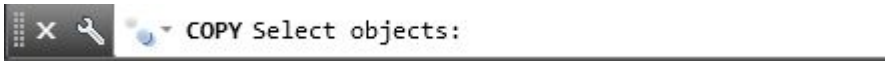
- Mi ↲
- Select object ↲
- Select first point of mirror line
- Select second point on mirror line
- Delete the source object (Y/N) ↲

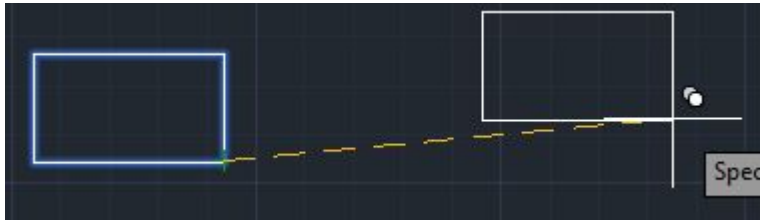


COPY

- Command: Co □
- Select Object □
- Select Base Point
- Specify the destination point

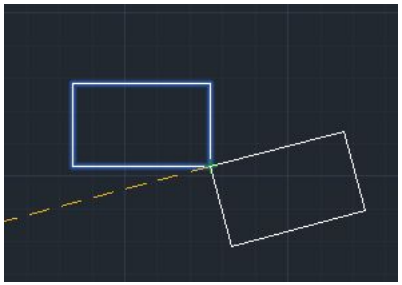
- Use – To copy the object .





ROTATE

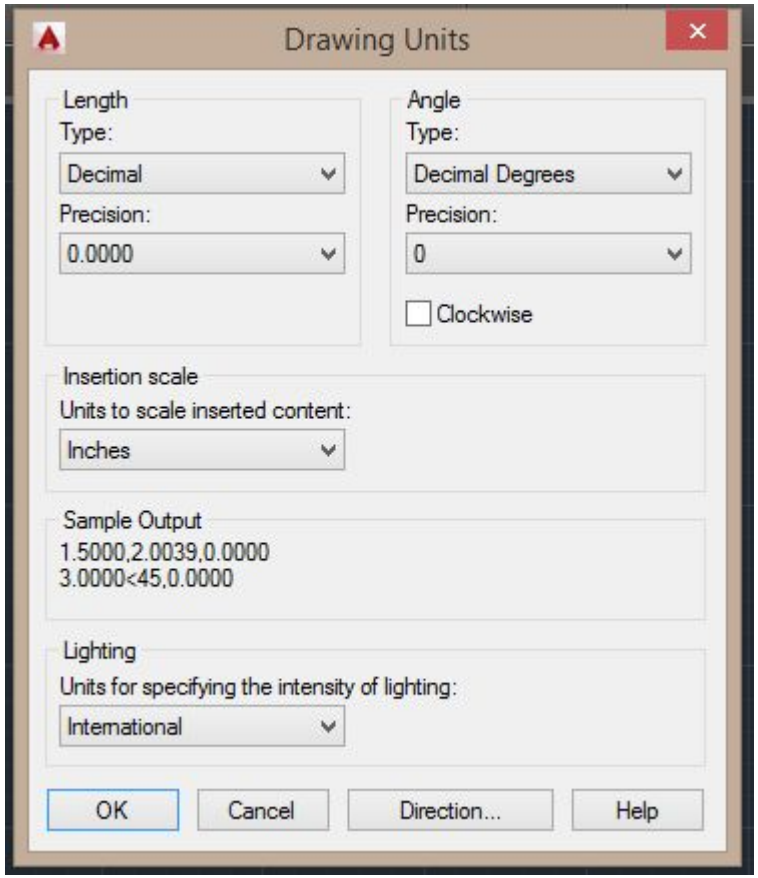
- Command: Ro □
- Select object □
- Select Base Point
- Specify Rotation Angle □
- Use –To rotate object about a fix point.



UNITS

- Command: Un □

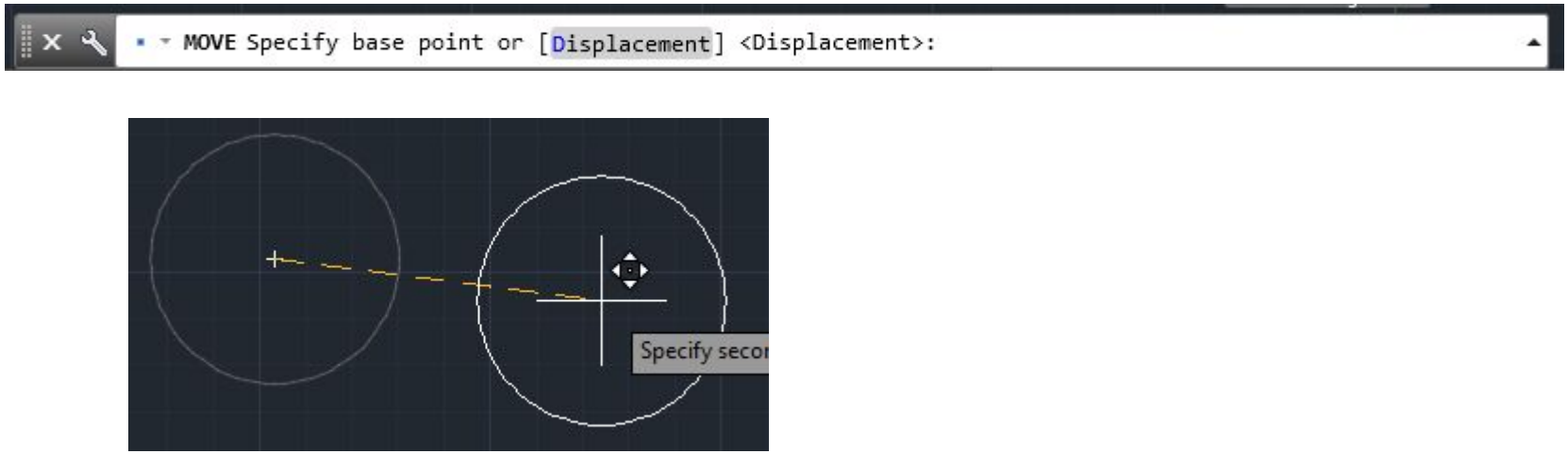
- Type : engineering
- Precision : 0’.0”
- Scale : inches
- OK



MOVE

- Move
- Command: M □
- Select object □
- Select base point
- Specify the destination point

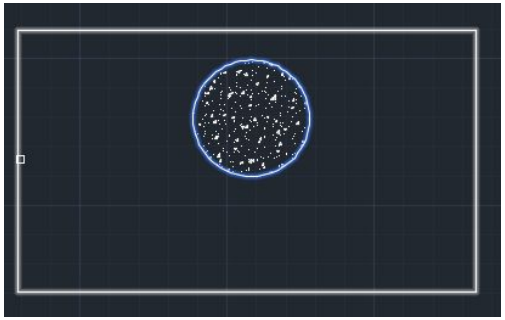
Use - To move objects from one place to another place.

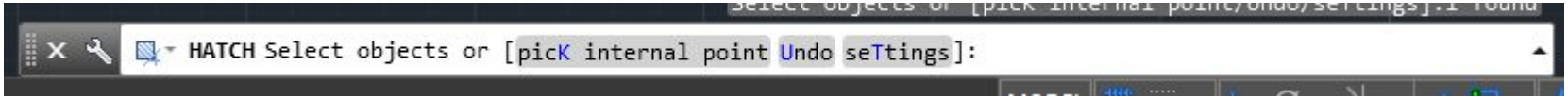


HATCH

- H↵
- Select Add pick point
- Click on object
- Select required pattern
- Change the hatch color, background color,
- change on angle,
- Change scale value according to requirement
- Close

- Use – To separate one object from other object by hatching



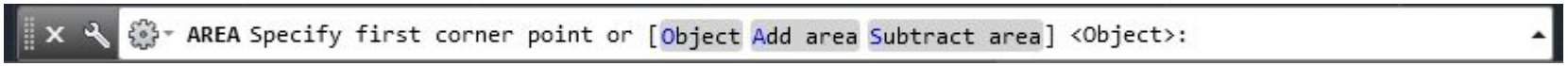


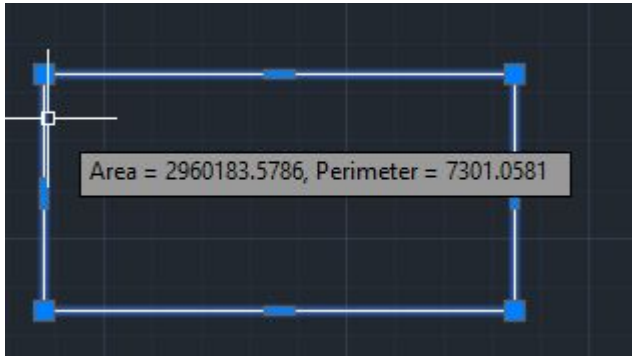
ARE

A

- Area
- Command: AA
- O
- Select the object

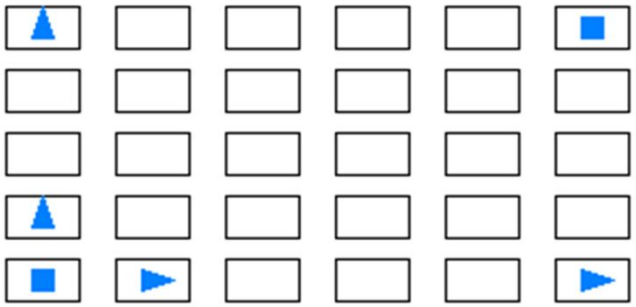
- Use – To calculate area and perimeter of any closed figure.



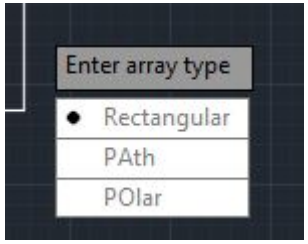


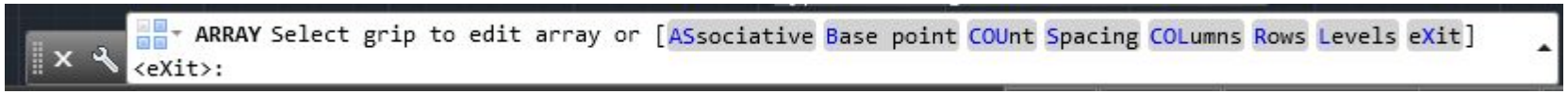
RECTANGULAR ARRAY

- Command: Ar ↵
- Select the object ↵
- Select the type of array r ↵
- Col ↵
- Sepecify no.of col ↵
- Specific distance between col
- Specify no.of row R ↵
- Specific distance between row



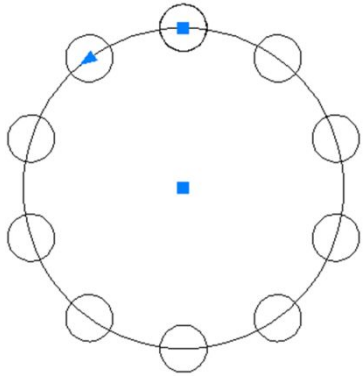
- use –To creat similar object in rectangular form.



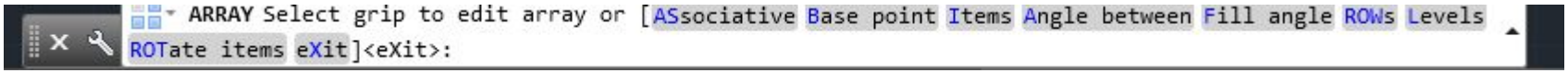


POLAR ARRAY

- Command: Ar ↵
- Select the object ↵
- Select the type of array Po ↵
- Sepecify centre point of array
- I(items) ↵
- Enter no. of items ↵



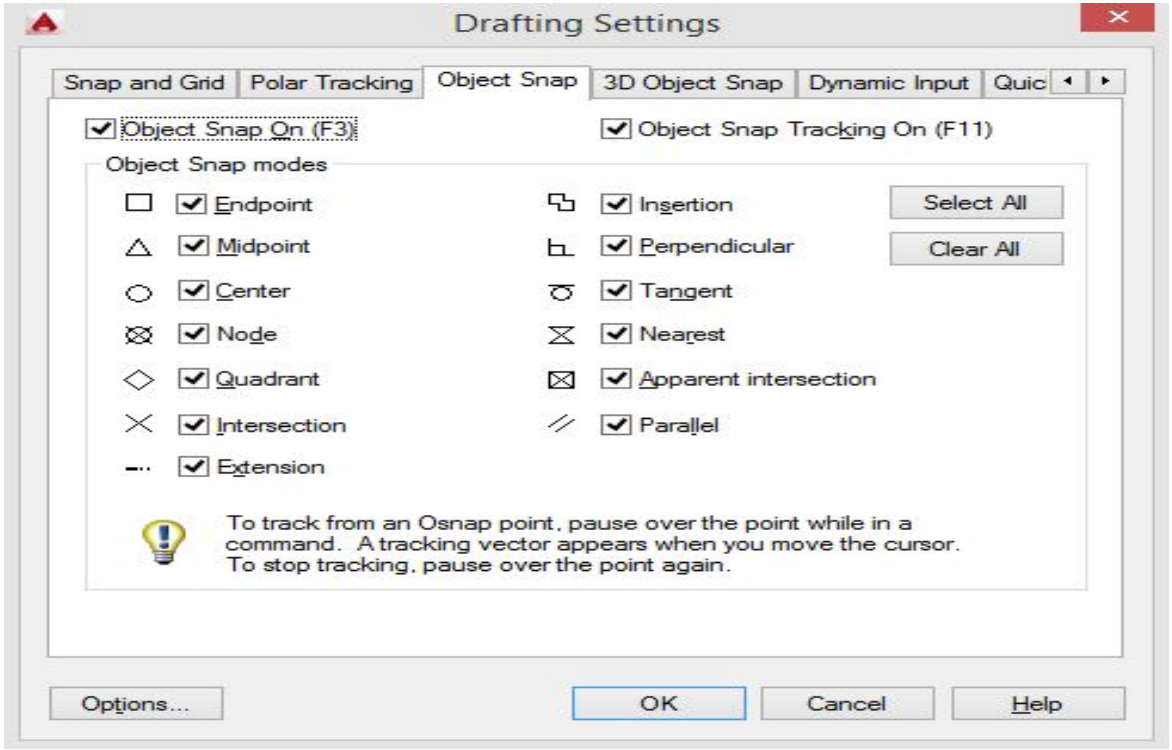
- use –To creat similar object in around a point.



DRAFTING SETTING

USE TO CHANGE THE SETTING OF OBJECT SNAP

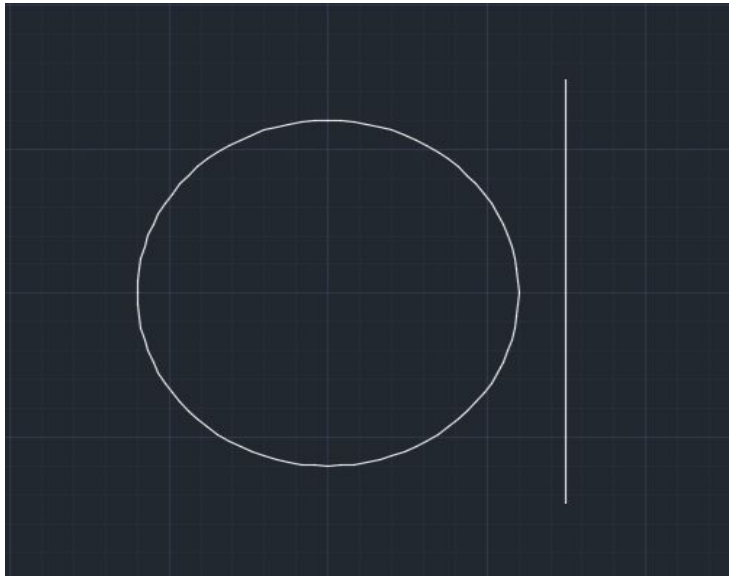
- Command -: DS
- Press DS and Enter
- Click on Object Snap
- Press on Select All option

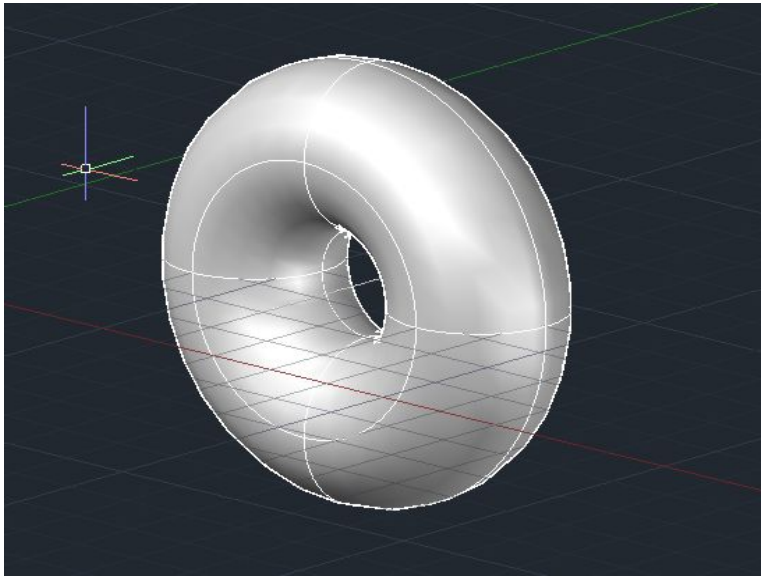


REVOLVE

USE TO REVOLVE AN OBJECT

- Command-: REV
- Select The Object
- Select a reference point
- Press enter

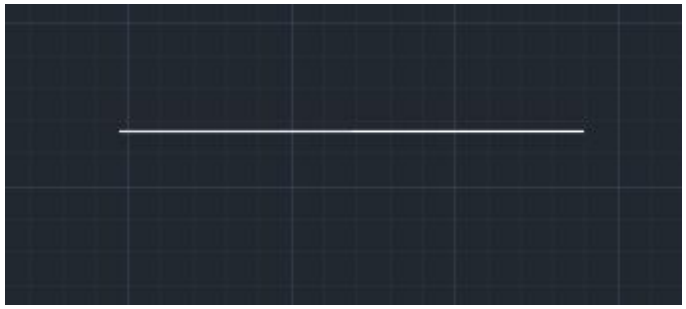




BREAK

USE TO BREAK THE LINE

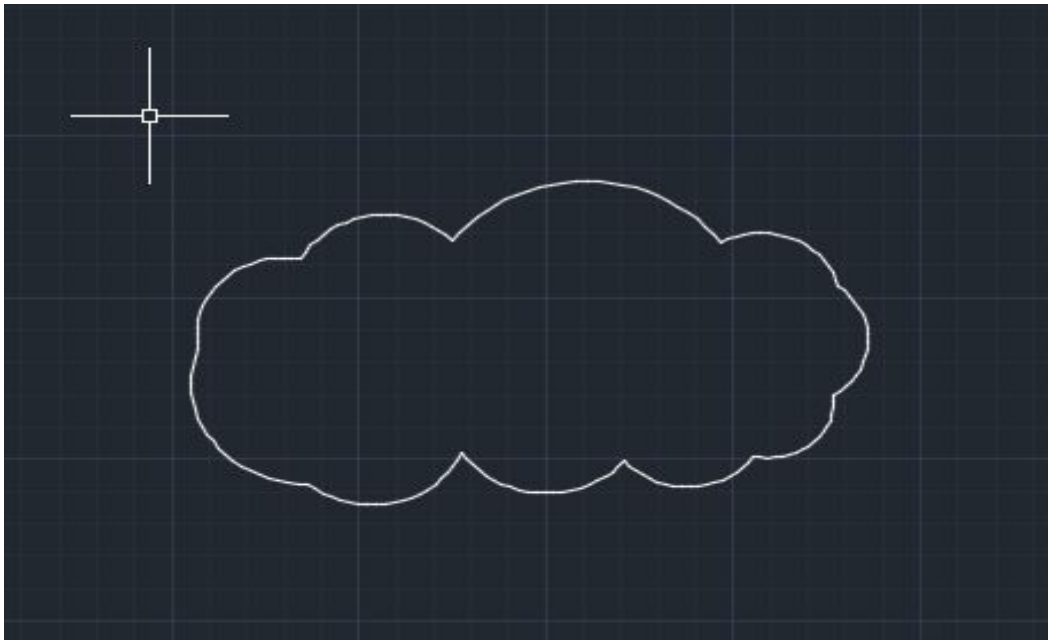
- Command-: BR
- Select The object to Break
- Select The Starting Point
- Select The End Point
- Press Enter



REVCLOUD

USE TO MAKE CURVE FIGURES

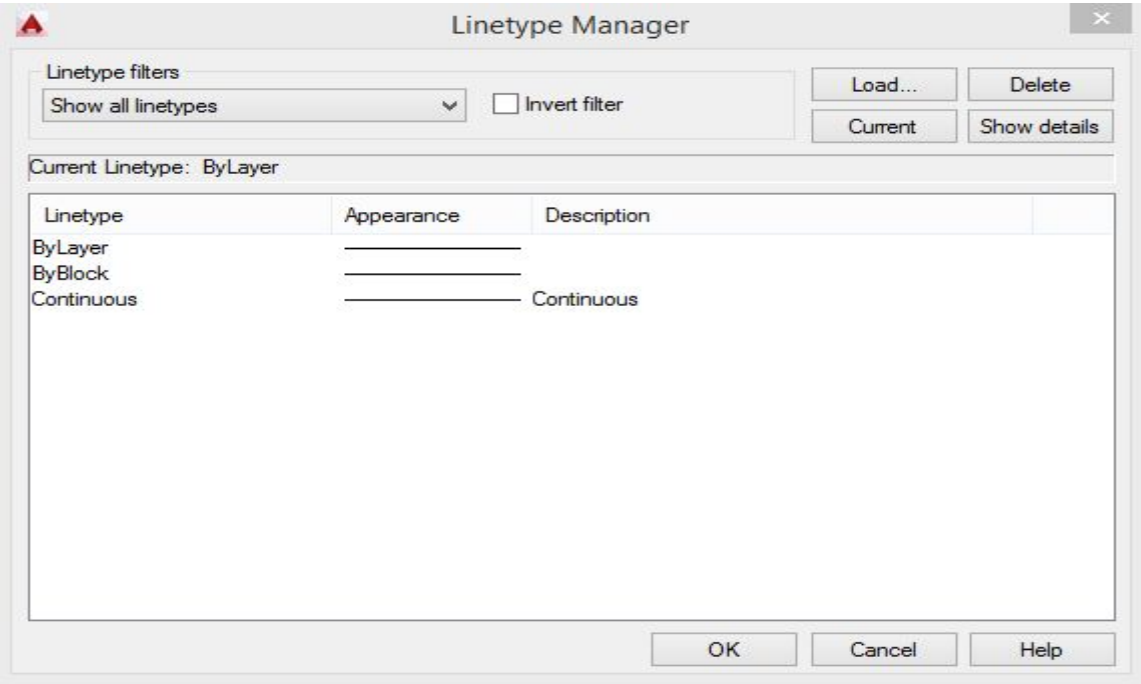
- Command-: REVCLOUD
- Select The Starting Point
- Select The Guide Crosshairs Space
- Coincide The End Point to Starting point



LINE TYPE

USE TO MAKE DIFFERENT TYPES OF LINES

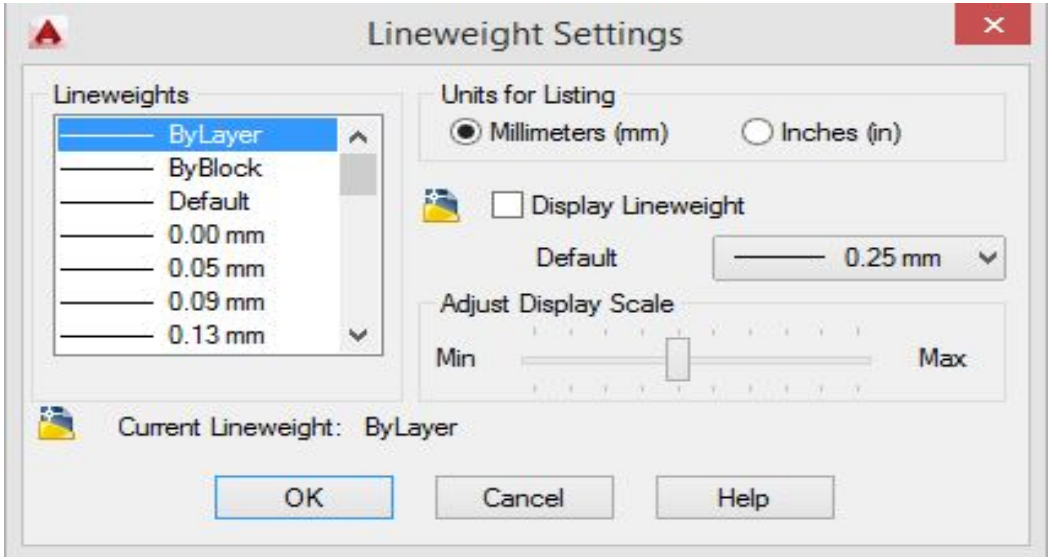
- Command-: LINETYPE
- Go to Line type command
- Select any type of line
- Set Current and Press Enter



LINEWEIGHT

USE TO BROADEN THE LINE

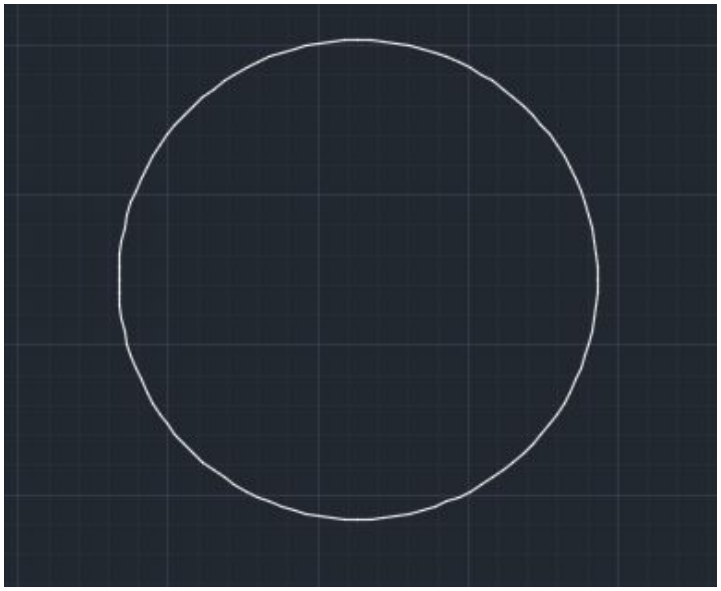
- Command-: LWEIGHT
- GO To LINEWEIGHT Command
- Select any Type of Line and Thickness
- Press ok

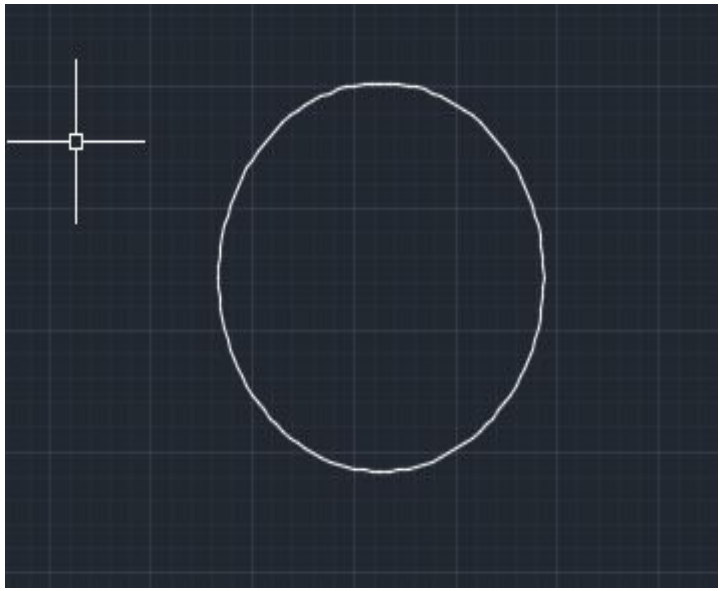


SCALE

USE TO EXTEND THE SHAPE BY FIXING A POINT

- Command-: SC
- Select The Object
- Select The Fixed Point
- Extend as Per Require





POLYLINE EDIT

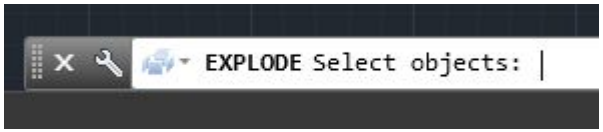
USE TO CONVERT LINE TO POLYLINE

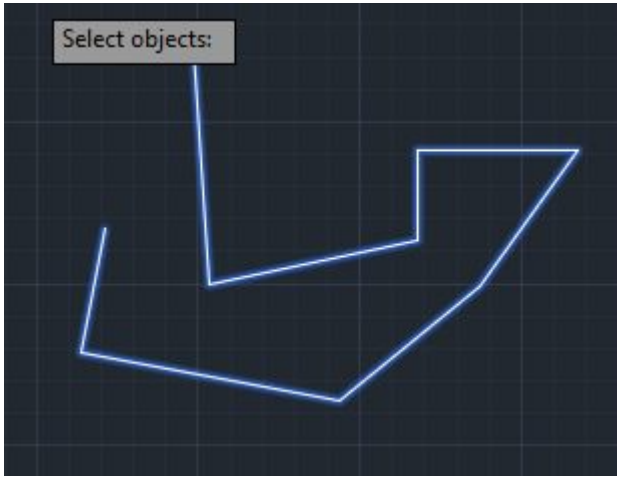
- Select all Different Line
- Press enter
- Select on any option given

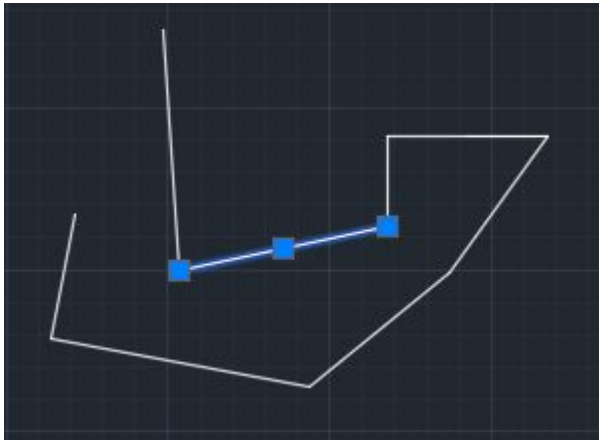
EXPLOD E

USE TO CONVERT POLYLINE TO LINE

- Command-: EXPLODE
- Select a Polyline
- Press the Command
- Press Enter



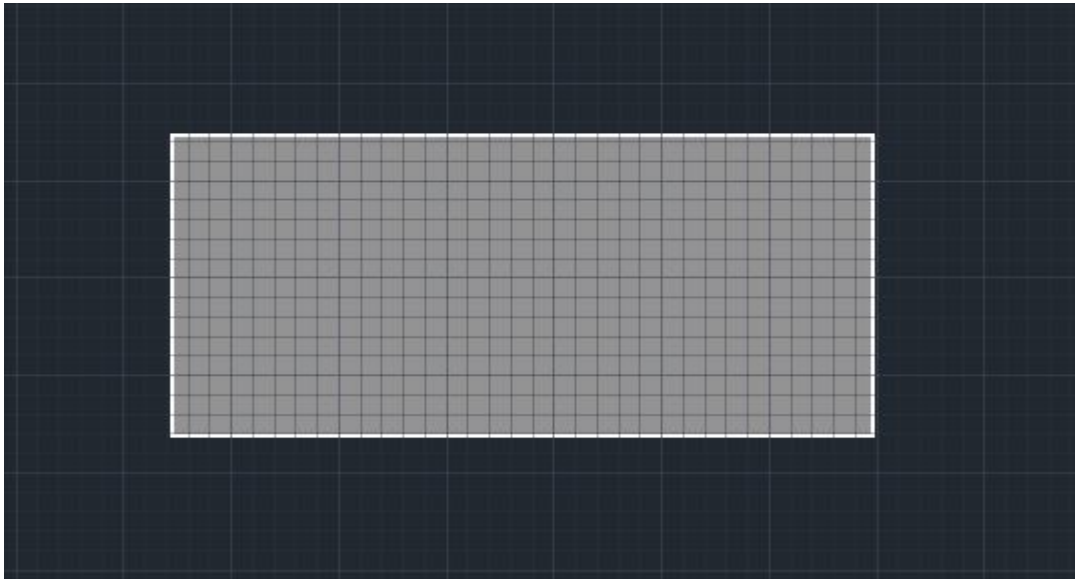




3D FACE

USE TO MAKE 3 POINT LINE

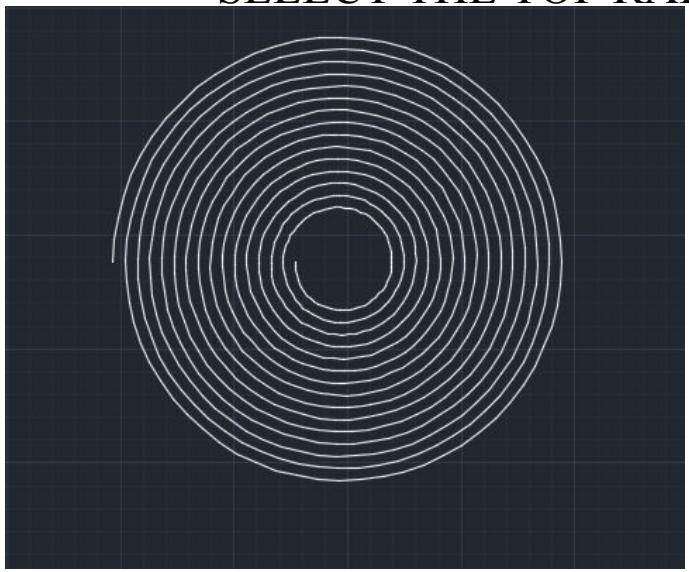
- Command-: 3DFACE
- Select the first point
- Select the second point
- Select the third point
- Press enter

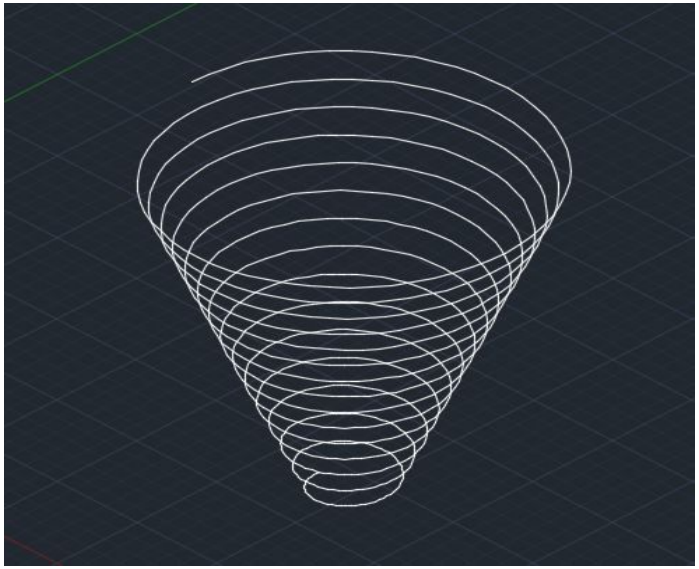


HELIX

USE TO MAKE HELICAL STRUCTURE

- Command-: HELIX
- SELECT THE CENTER POINT
  - SELCT THE BOTTOM RADIUS
- SELECT THE TOP RADIUS

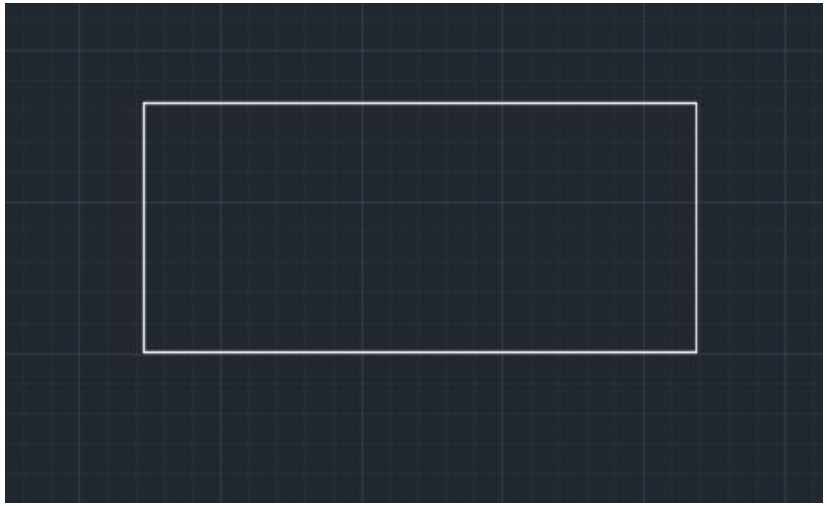


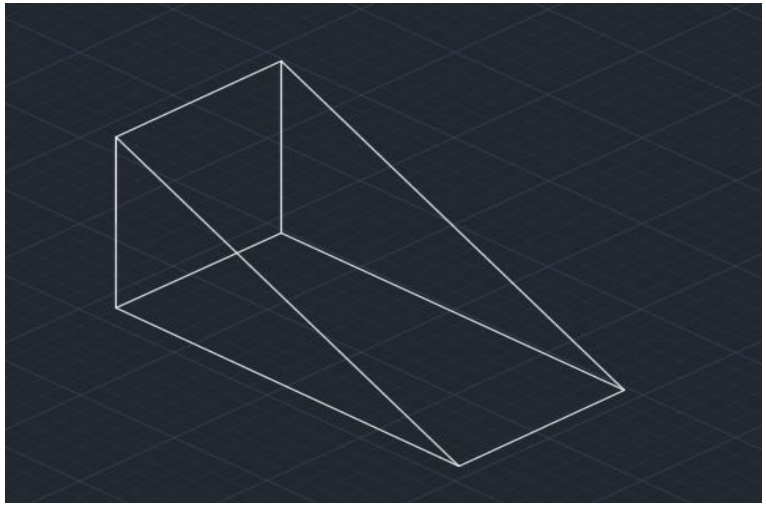


WEDGE

USE TO MAKE WEDGE

- Command-: WE
- SELECT FIRST CORNER
- SELECT ANOTHER CORNER
- SPECIFY HEIGHT
- ENTER

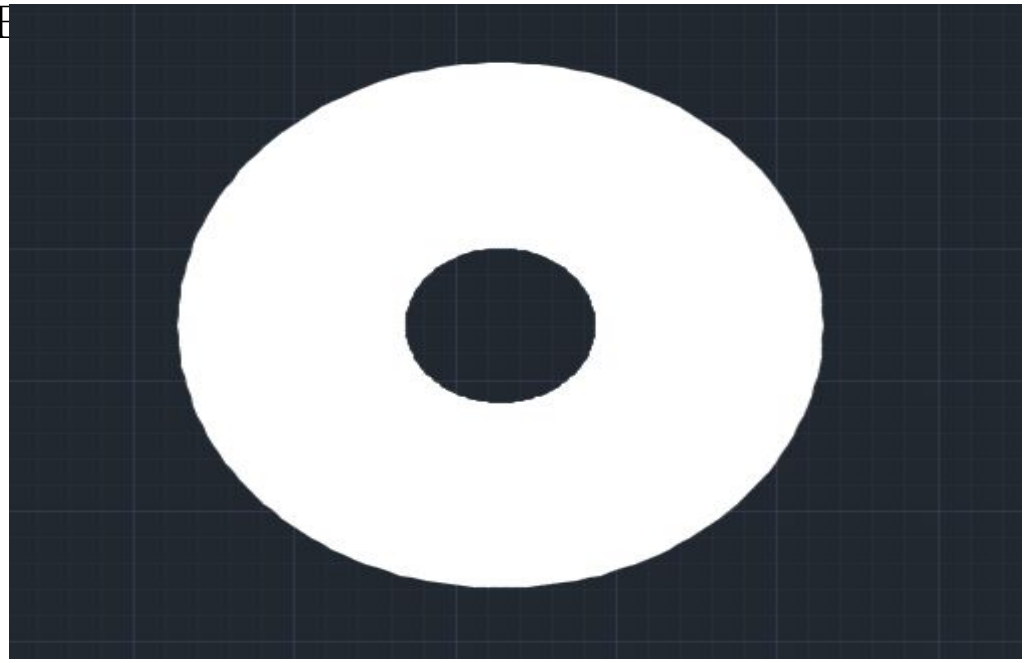




DONUT

USE TO MAKE CONCENTRIC CIRCLE

- Command-: DO
- SELECT THE INNER DIAMETER
- SELECT THE SECOND POINT
- SELECT THE OUTSIDE DIAMETER
- PRE



TORUS

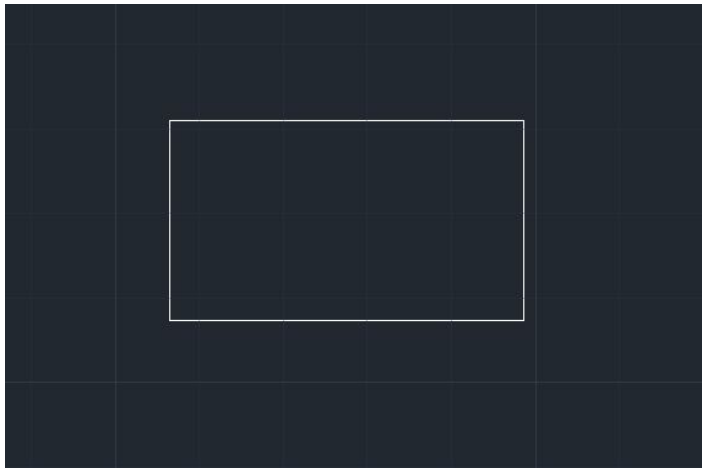
USE TO MAKE HOLLOW CIRCLE

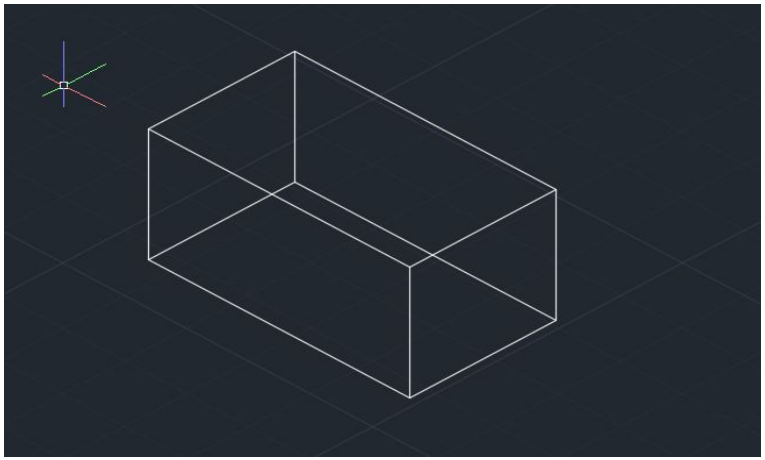
- Command-: TOR
- SELECT THE CENTER
- SELECT THE RADIUS
- SELECT THE TUBE RADIUS
- 

EXTRUDE

USE TO CREATE A 3D SOLID SURFACE

- Command-: EXT
  - SELECT OBJECT TO EXTRUDE
- PRESS ENTER

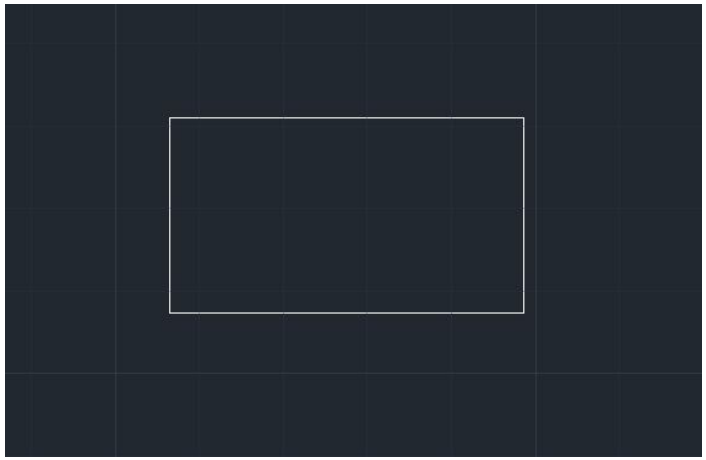


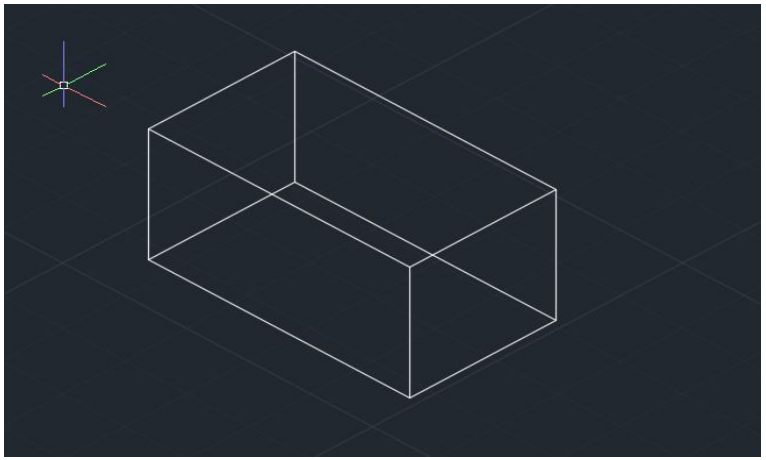


PRESSPULL

USE TO EXTEND ANY FIGURE

- Command-: PRESSPULL
- SELECT OBJECT OR BOUNDED AREA
- PRESS ENTER

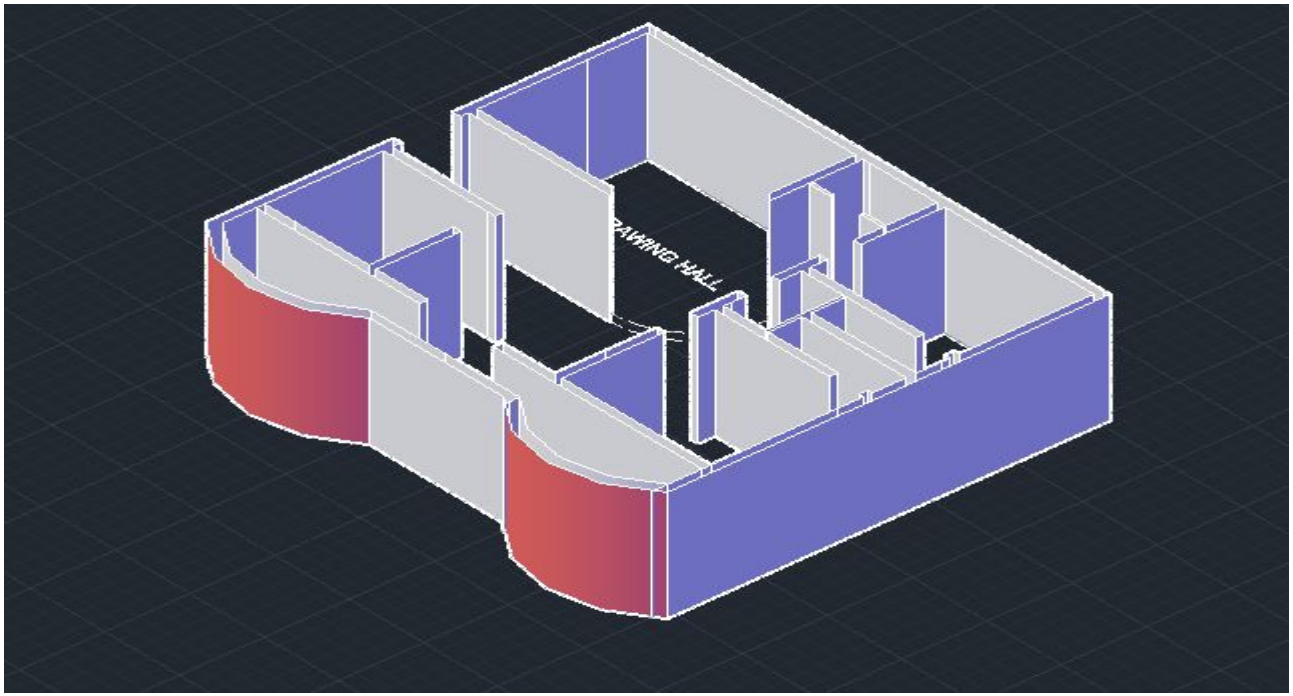




3-D ORBIT

USE TO CHANGE THE VIEW IN 3-D

- Command-: 3DO
- SELECT ANY POINT
- DRAG THE MOUSE TO SEE THE VIEW

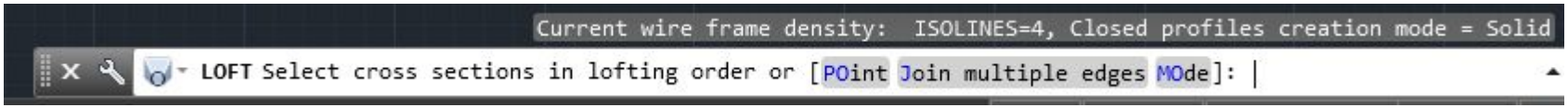


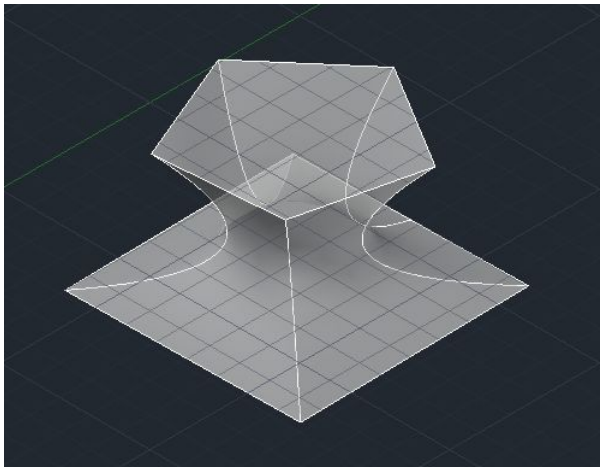
LOFT

It is used to create 3D solid between several cross-section.

- Command : **loft**↲
- Draw menu →modelling →loft →select the cross-sections in a sequence →↲↲

Note : profiles must be co-planner and having some vertical gaps between each profiles.

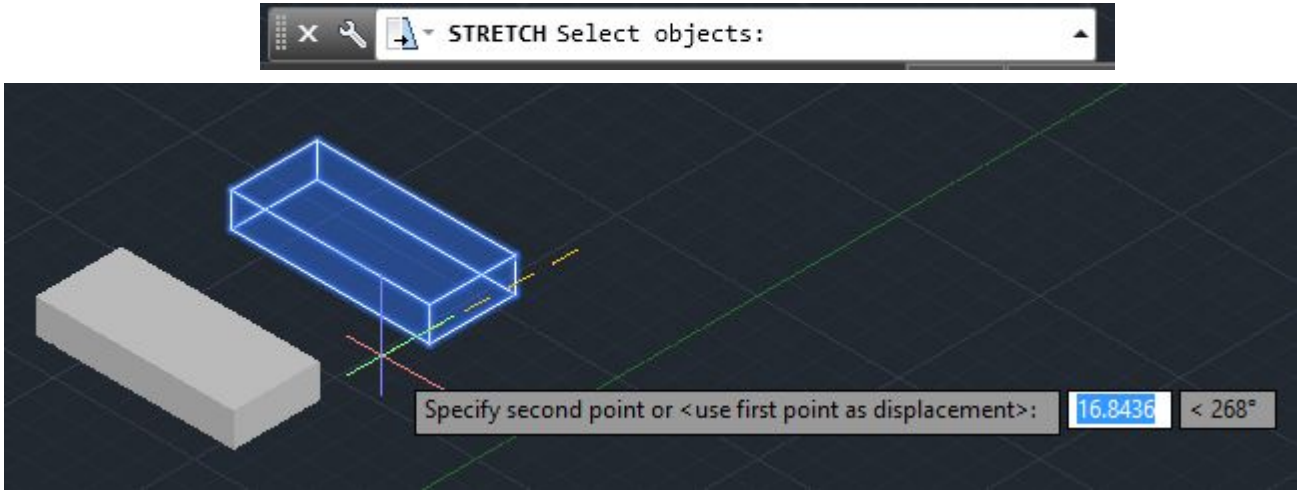




STRETCH

Used to stretch object from its position

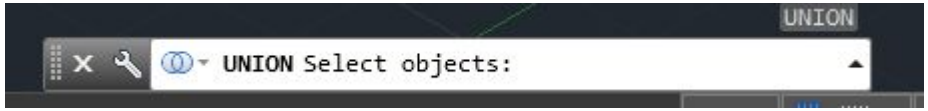
- Command : S ↵
- Click & Drag to draw half crossing selection ↵
- Specify base point for stretch & drag at desire point/destination

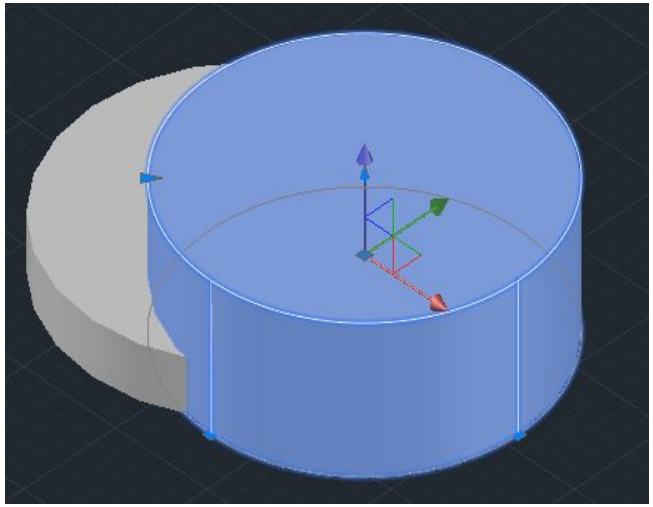


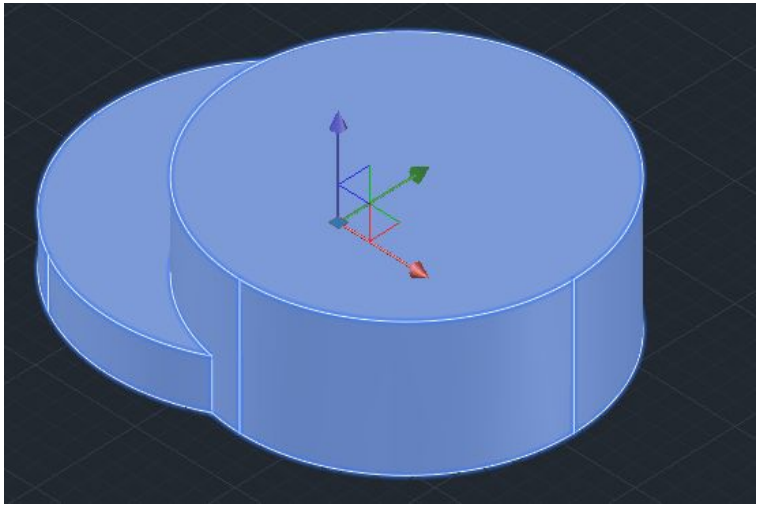
UNION

It is used to combine 3D solids

- Command : uni↲
- Modify menu → solid editing →union →select the 3D objects →





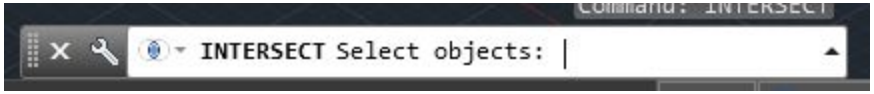


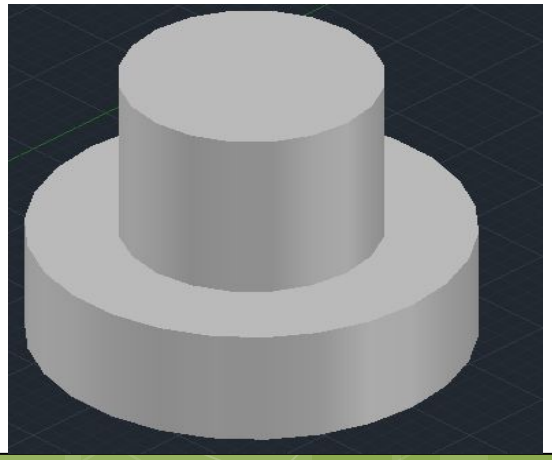
INTERSECT

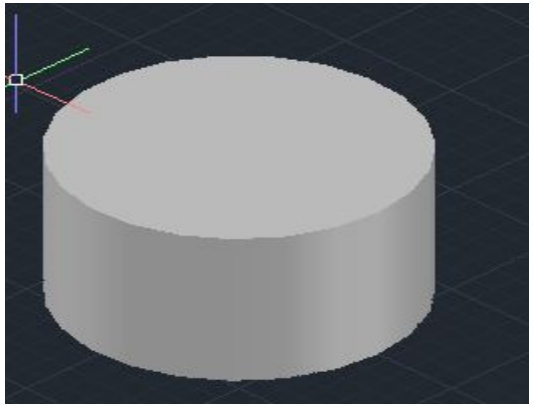
It is used to keep the common parts of the overlapping solids.

. Command: IN □

- First select two concentric object.
- SE Isometric view→ extrude to different height→intersect →select the overlapping solids → ↲

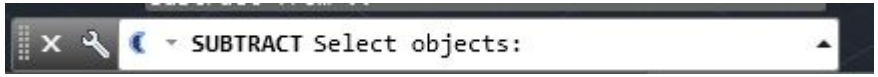




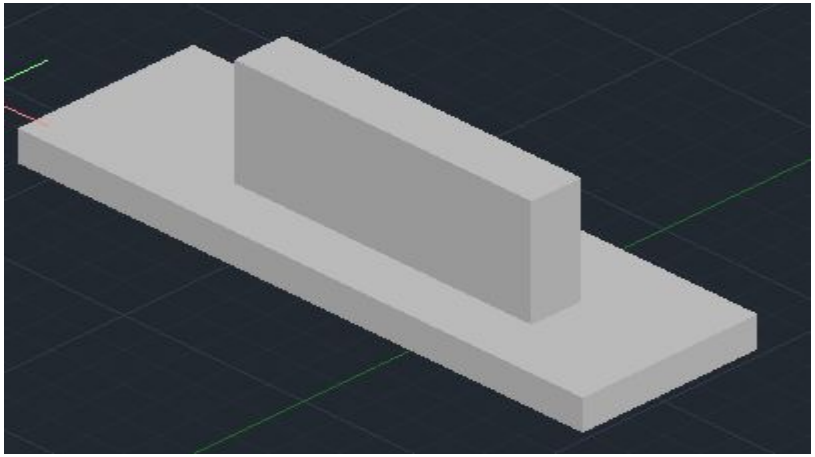


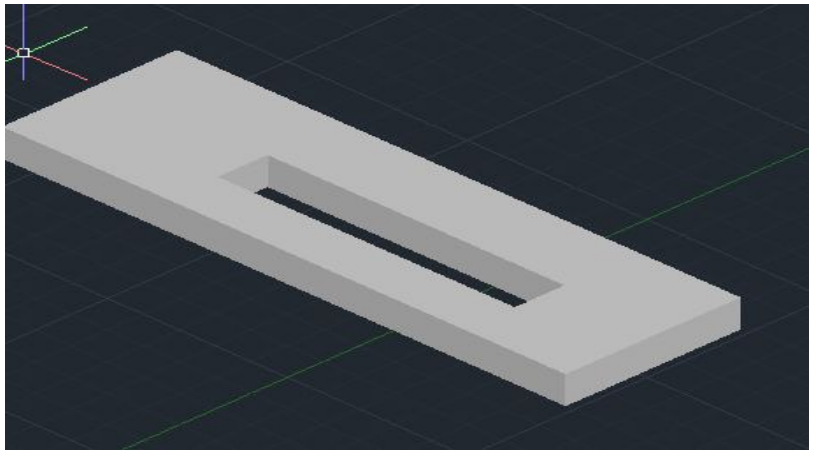
SUBTRACT

It is used to combine the 3D solids and remove solid from a overlapping solid.



- Command : su↲
- Modify menu → solid editing →subtract →select the object to retain →right click →select the object to remove → ↲

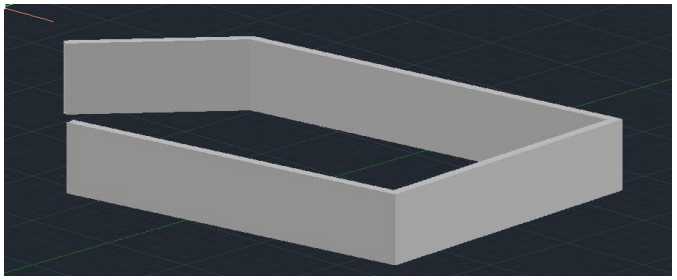


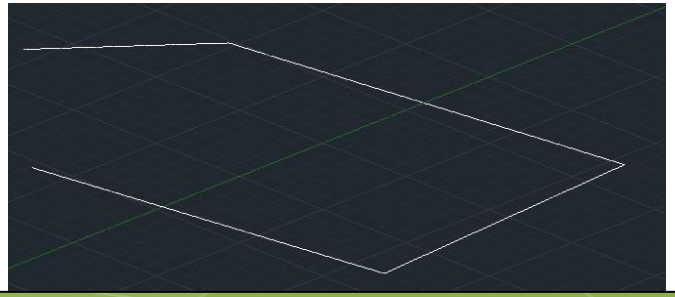


POLYSOLID

It is used to create walls with straight and curved segment with specified height and width.

- Command : **psolid** ↲
- Draw menu →modelling →polysolid →**h**↲(height) →define the wall height → w ↲(width) → define the wall width →**j** ↲(justification) →L/R/C ↲ (left/right/centre) →specify the 1st point →specify the next point (OR) **o** ↲ (object) →select the profile



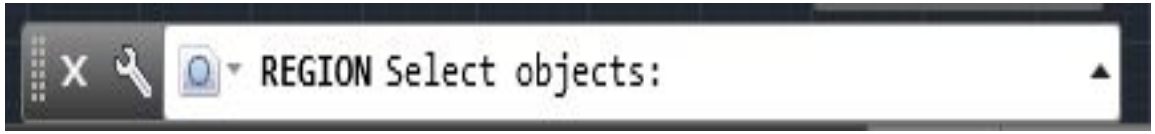


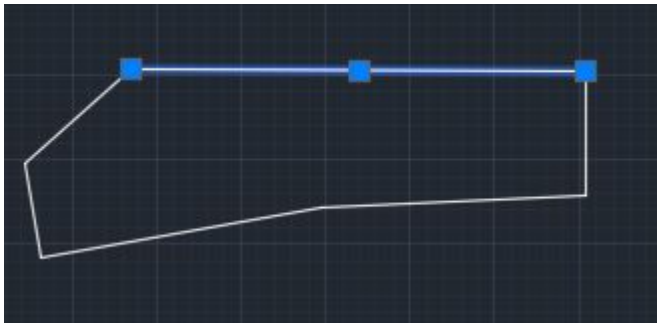
REGION

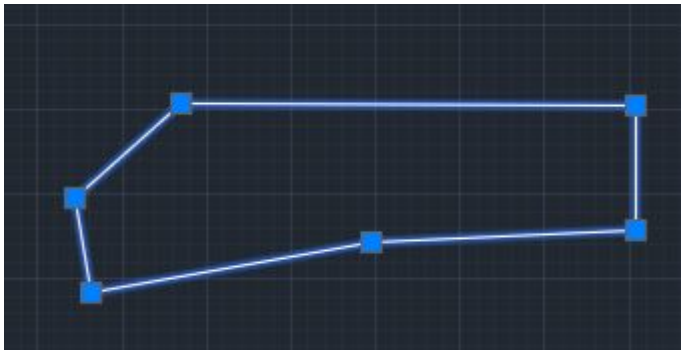
It is used to convert multiple entities to single object.

- Command : REG ↲
- Draw menu → region →select the multi entity profile by clicking over it → ↲

- Condition : profile must be closed and must not be intersected.







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