AUTO CAD PRACTICAL FILE

AUTODESK
AUTOCAD
2014

BABA FARID GROUP OF ENGINEERING AND TECHNOLOGY

ER. ASHISH KUMAR YADAV

MECHANICAL ENGINEER

ashishcadbfgi@gmail.com

Contact- 9501115414, 9780027927

(AUTO CAD 2014 FOR CIVIL, ELECTRONICS AND MECHANCIAL ENGINEERING)

INTRODUCTION OF THE AUTO CAD SOFTWARE AND ITS UTILITIES IN THE ENGINEERING SOFTWARE.

AUTODESK
AUTOCAD
2014

AutoCAD is a commercial software application for 2D and isometric and 3D computer-aided design (CAD) and Drafting software available since 1982 as a desktop application and since2010 as a mobile web- cloud based app market as AutoCAD 360. This software is developed and marketed by the Autodesk Company. AutoCAD is used across a wide range of industries, by architects, project managers, engineers, designers and other professionals.

1. You can install the software package of AutoCAD through DVD and also AutoCAD LT version is also available for the beginners.
2. Choose start> All Programs> Autodesk>AutoCAD 2014
   You can also double click AutoCAD 2014 icon on your Windows Desktop.

RIBBON

Drawing area

UCS (USER CO-ORDINTATE SYSTEM

Status bar

Command window
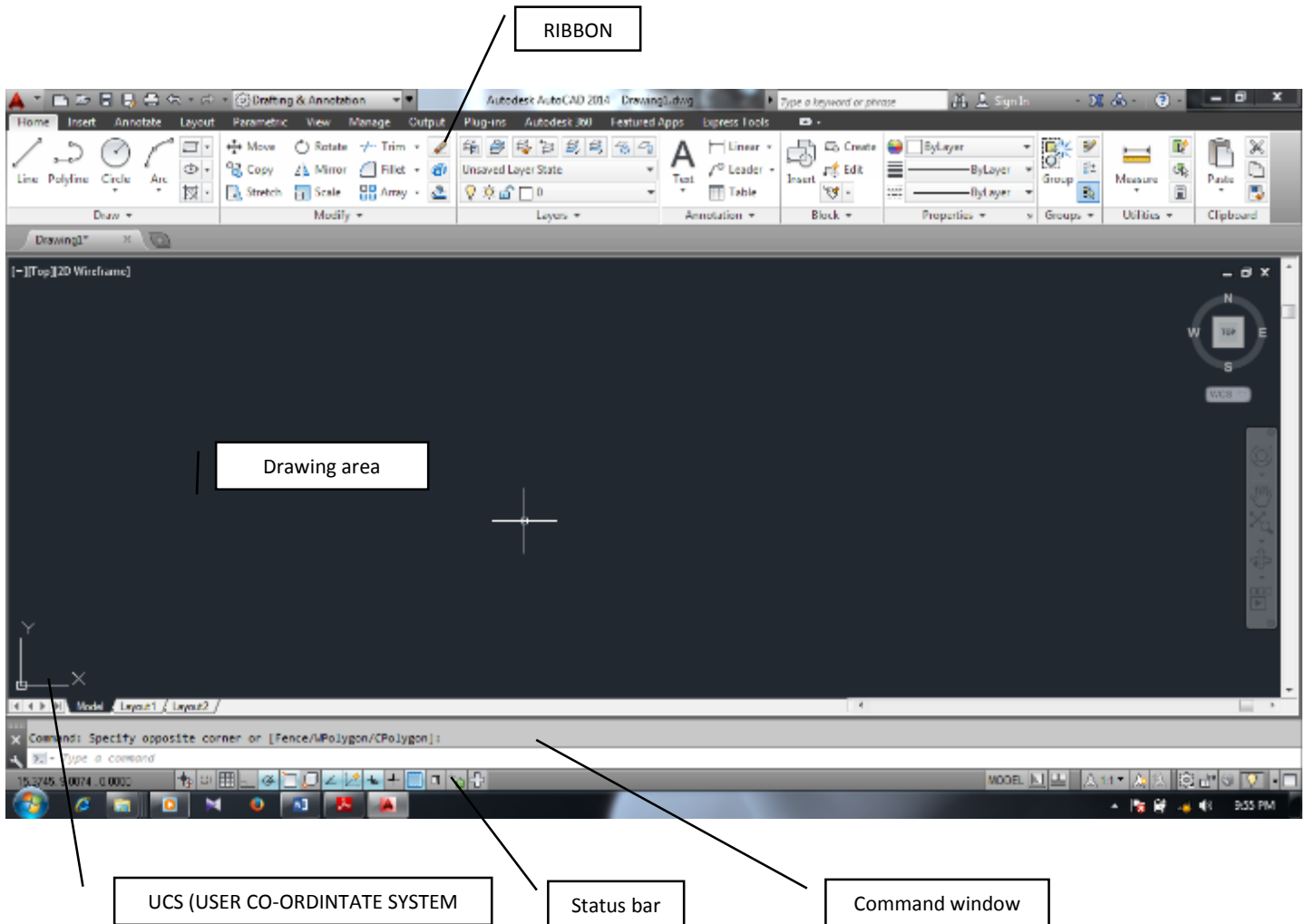
ADVANTAGES OF THE AUTO CAD

1. It has ability to producing very accurate designs. And drawing can be created in 2D or 3D .
2. **Easy to modify-** modifying the CAD geometry is easy; you will always have "copy" "cut" paste and delete and move or some similar editing option available with each of the package.
3. **Easy to Reproduce-** Draftsmen used to take days to complete a drawing by manual drafting, and reproducing the drawing means recreating the drawing from the beginning. But, in case of the CAD, you can reproduce the drawing in no time and makes as many copies as you can.
4. **Computer aided manufacturing-** The 3D CAD geometry is used as input for the CAM packages for generates NC codes. The manual drawings cannot be used for CAM packages.
5. **Environmental Friendly –** Manual drawings are necessarily created on the paper, but the CAD drawings can be stored and used electronically without using paper.
6. **Access Control –** some of the drawings and design documents are very crucial for a company business, and such drawings should not be accessible to all. Providing are easy for the CAD drawings. Strict access control and maintaining confidentiality is difficult in manual drawing.

STUDY THE BASIC INITIAL SETTING AND VIEWING OF THE DRAFTING SOFTWARE INTERFACE.

HOW TO SET UNITS

1. In autoCAD every object is measured in units. You determine the value of unit before you draw. For example, in one drawing, a unit might equal one millimeter of the real world object. In another drawing, a unit might equal to inch. You can set the unit type and number of the decimal place for the object length and angles.

COMMAND- **UNITS** ↵ or **un** ↵

UNIT FORMAT	EXAMPLES
SCIENTIFIC	1.55E+01
DECIMAL	15.50
ENGINEERING	1'-3.50"
ARCHITECTURAL	1'-3 ½"
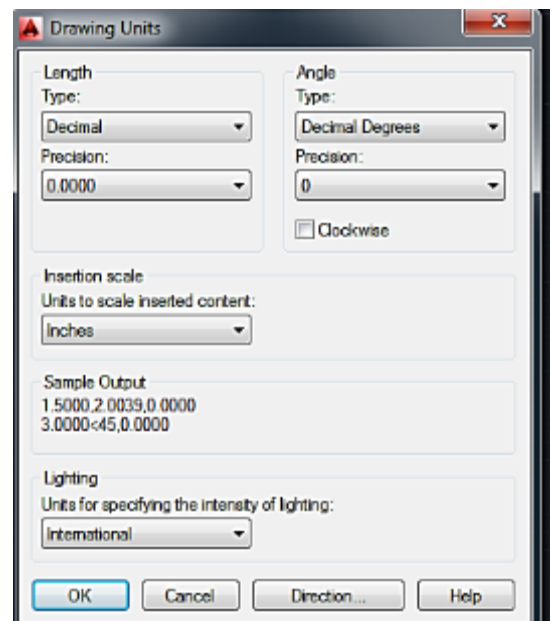
UNITS

- In units settings go to "Type" choose the desired unit standard e.g. 0.000 for the decimal, 1E3 (1000) for the Engineering etc.
- In "Precision" choose the desired preciseness you need in the drawing e.g. 1.001 for (0.000) and for 0.1 choose 0.0 and for 1 choose "0"
- Choose the insertion scale of your set perimeter or dimension e.g. millimeters, centimeters, inches, feet etc.
- Choose the angle type same as your desired units choose before e.g. decimal degrees, engineering, architectural etc.
- Click on "OK" to set the units and hit the Esc key to get out of the command.

How to set the limits in Auto cad

LIMITS in the rectangle boundary area in the AutoCAD where we draw our working drawings. Setting of the limits is the most important task because of in very large scale of drawing we need the actual limits of the drawings so that they can easily printable and zoom and easy to the find the object we create in specified limit.

- type – **LIMITS** ↵

1. Specify the lower left corner of the screen type (0,0) and hit the ENTER key
2. Specify the Upper right corner of the screen type (500,500) and hit the Enter key. 500 is the specified units distance e.g. if you choose millimeters then 500mm × 500mm is considered as a base × height.
3. Click the ZOOM tool in the status bar and type A ↵ OR

Type Z ↵ A ↵ or DOUBLE CLICK on the mid mouse button.
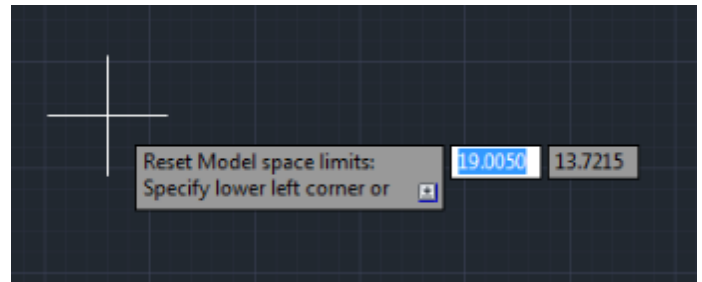
UPPER RIGHT CORNER

LOWER LEFT CORNER

HOW TO SET THE DRAFTING SETTINGS IN THE AUTOCAD

DSETTINGS – TOOLS>DRAFTING SETTINGS…

Drafting settings this dialogue box contain so many tabs like Snap and Grid, Polar tracking, Object snap, Dynamic Input etc. let discuss each tab one by one

Right click on the status bar and choose properties the drafting setting box will appears to you in the AutoCAD screen.

Command – DSETTINGS ↵

1. **Object snap (F3)** - it's used to turns the running snap on and off when the object snap is on it will shows the main snap points of the object which is relative to the other object e.g. end point, midpoint, center point, tangent point, perpendicular etc.
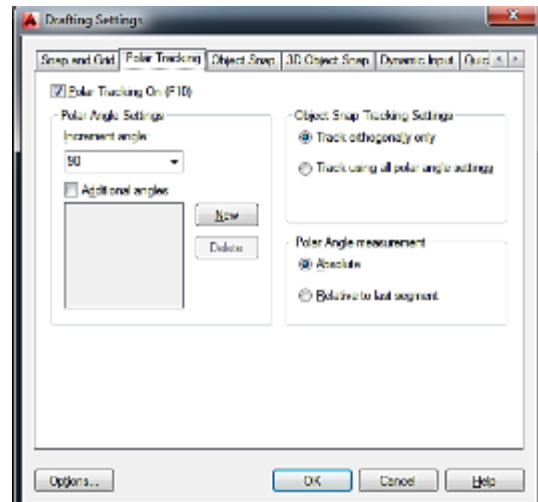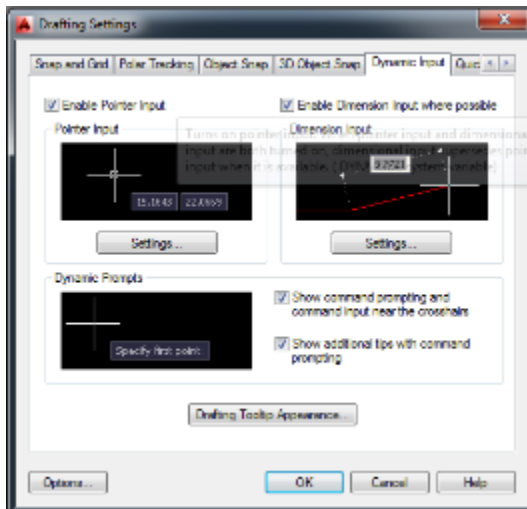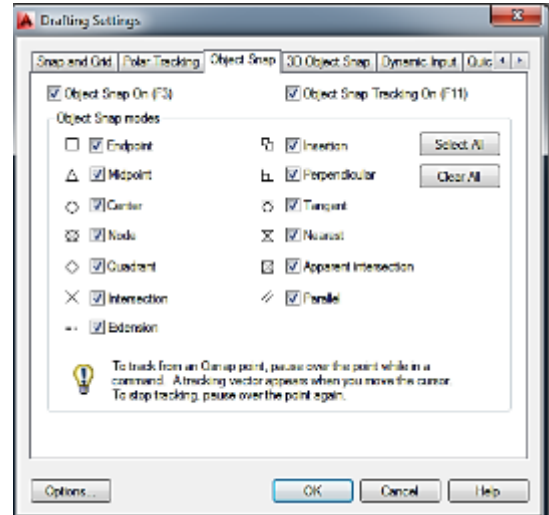
2. **Object Snap tracking (F11)-** object snap tracking gives the tracking point of the object near to the relative point of the another object which is more helpful and no need to draw any extra helping line to follow the object corner, center, quadrant tracking.

3. **Dynamic input (F12) -** The dynamic input enables the graphics cursor with the power of the command line. Thus you enter dimensional and command option information as you work, you get immediate feedback when your dynamic input is ON.

4. **Snap (F7) and Grid (F9)-** Snap and Grid mode is used for the showing the graphical representation and when we move the mouse cursor it will not move freely it will move point to point as we set the graph snap point. **e.**g. if we set snap 0.5 to 0.5 the mouse corsur move to the distance .5 on the computer screen.

5. **ORTHO MODE (F8) –** Ortho mode is used to draw a straight line if we need to draw a perfect straight line you need to on the ortho mode by pressing F8 button on your keyboard.

6. **POLAR TRACKING (F10)-** Polar tracking is used when we work on the specified angel it will automatically tracks the angel we choose and indicate us with green hidden line indication. You can also choose your costum angle by click on the additional angle>New and put the specified angle you need.

All the drawings are suprimposed on an invisible grid, or co-ordinate system with Horizontal axis and Vertical axis which is commonly known as X axis and Y axis.
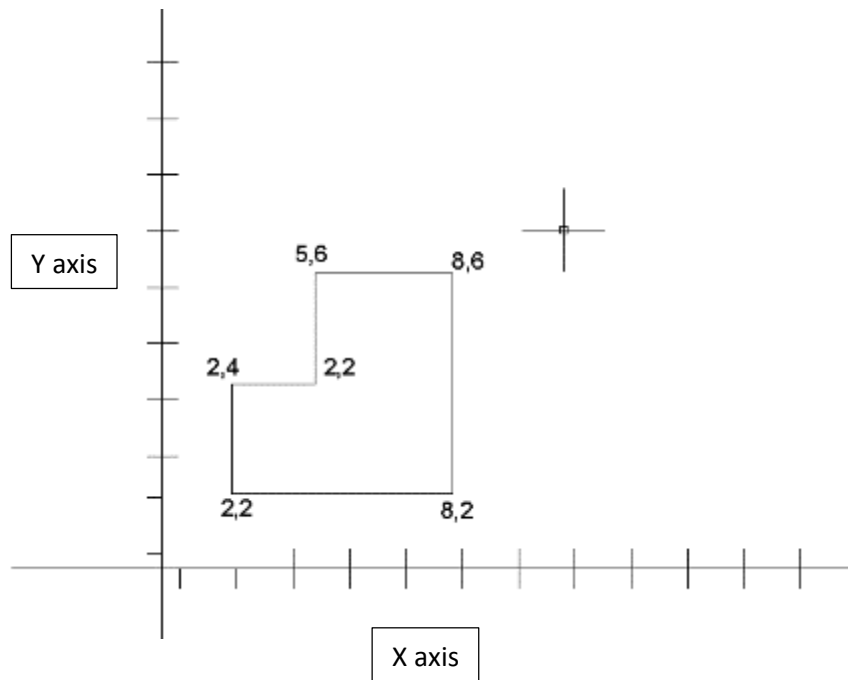
In AutoCAD there are main three types of CO-ORDINATE system used. Each of the co-ordinate are explained follows.
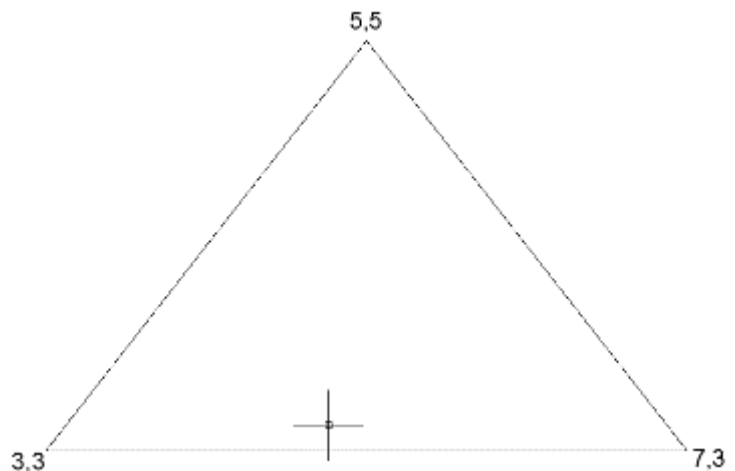
Absolute Co-ordinate System (x, y)

In absolute coordinates works when you know the precise X and Y values of the point from the origin. The following sequence of the coordinate system shown.

Command - _line Specify first point : 3,3 ↵

Specify next point or [Undo]: 7,3 ↵

Specify next point or [Undo]: 5,5 ↵

Specify next point or [Undo]: c ↵

RELETIVE RECTANGULAR CO-ORDINATE SYSTEM

Relative rectangular coordinates system is used when we know the position of a point with respect to the previous point.

@ x-displacement, y-displacement

Command - _line Specify first point : @2,2 ↵
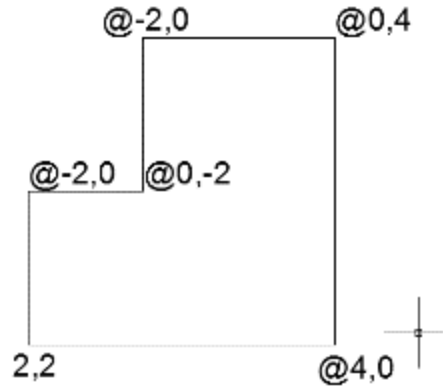Specify next point or [Undo]: @ 4,0 ↵
Specify next point or [Undo]: @ 0,4 ↵
Specify next point or [Undo]: @-2,0 ↵
Specify next point or [Undo]: @0,-2 ↵
Specify next point or [Undo]: @-2,0 ↵
Specify next point or [Undo]: c ↵

POLAR CO-ORDINATE SYSTEM

(@ DISTANCE< ANGLE)

Polar co-ordinate system uses when a distance and an angle with reference to the previous point to locate a point. Angle are measured in Anti-cock direction, taking 0^0 towards right.

Command - _line Specify first point : 2,2 ↵
Specify next point or [Undo]: @4<0 ↵
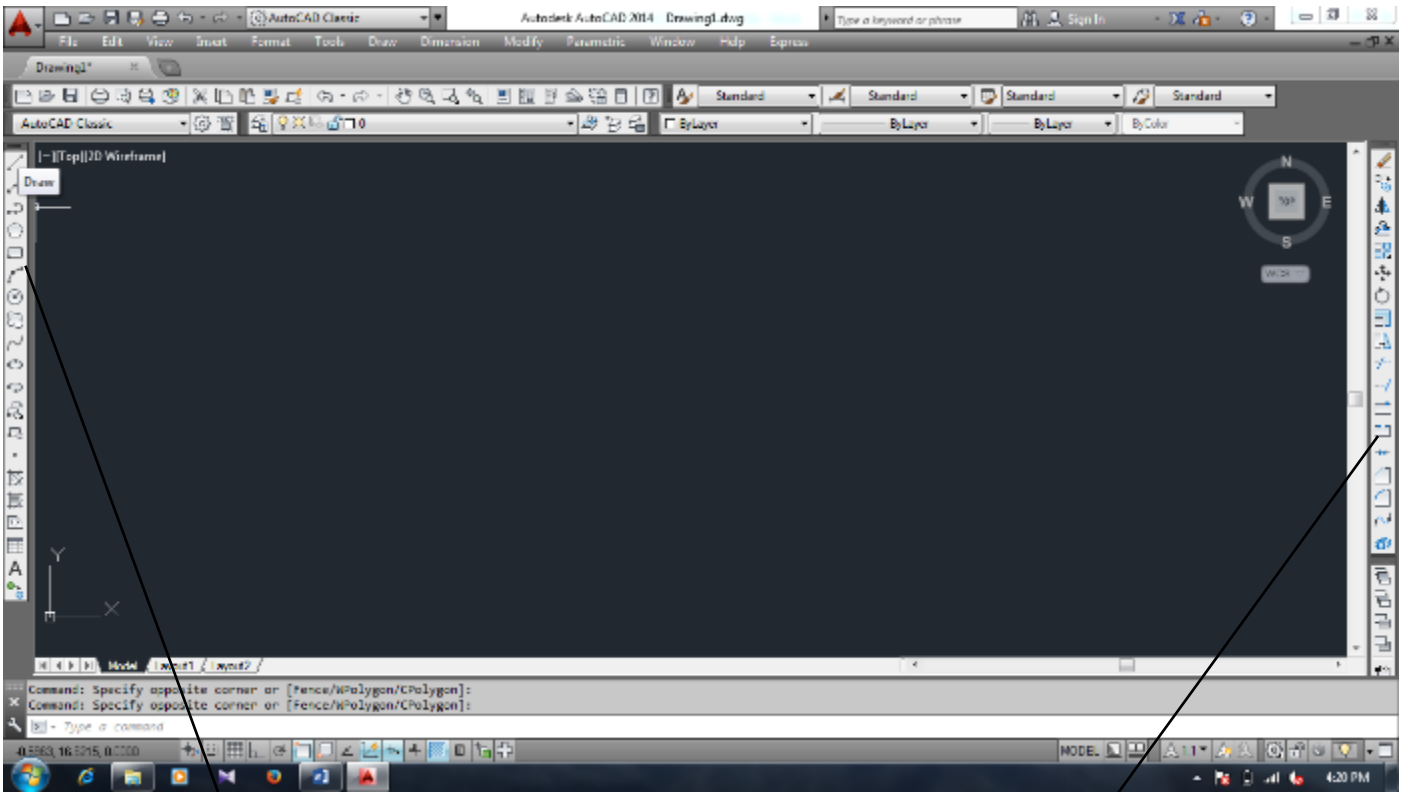Specify next point or [Undo]: @4<90 ↵
Specify next point or [Undo]: @4<180 ↵
Specify next point or [Undo]: c ↵
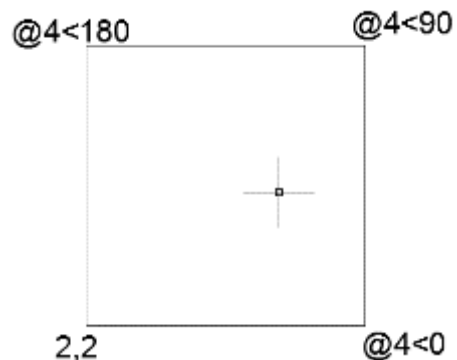
Use of Drawing tools.
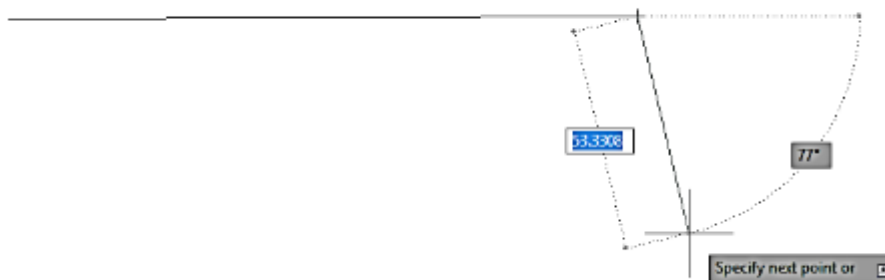

DRAWING TOOLBAR   MODIFY TOOLBAR

There are various commands tools available in drawing toolbars let's discuss each tool one by one.

LINE COMMAND-

Draw>line

Line command allows you to create a line where the end point are specified by two-dimensional of three dimensional co-ordinates.

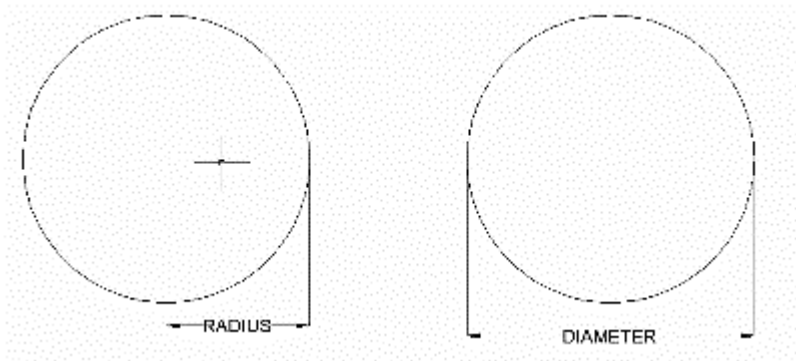**CIRCLE- c ↵**

DRAW>CIRCLE>{options}

In circle you create circle in several ways either by click on the command or pressing command through keyboard. And there are so many options to draw the circle.

Command: _circle

Specify center point for circle or [3P/2P/Ttr (tan tan radius)] : use one of point fixing method or enter an option

Center, Radius/Center, Diameter

Specify radius of circle or [Diameter] : Enter the value of radius or Press "D" and hit enter to put diameter.
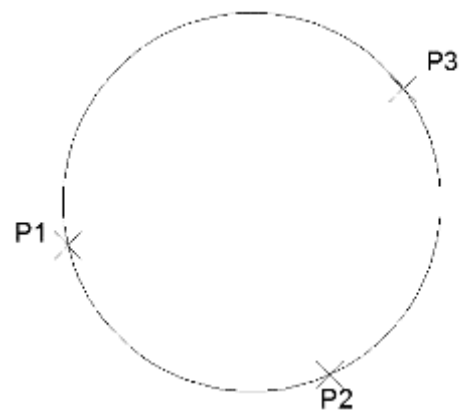

3P - you can also draw a circle with any 3 point on the screen this 3 point is working as a circumferential point of a circle.

2P – you can draw circle with also 2 point click on anywhere on screen it will create a circle as per 2 point you selected.
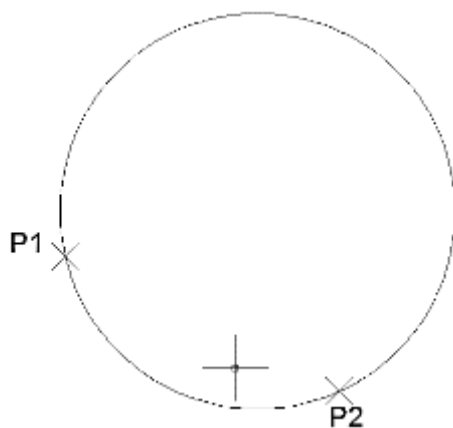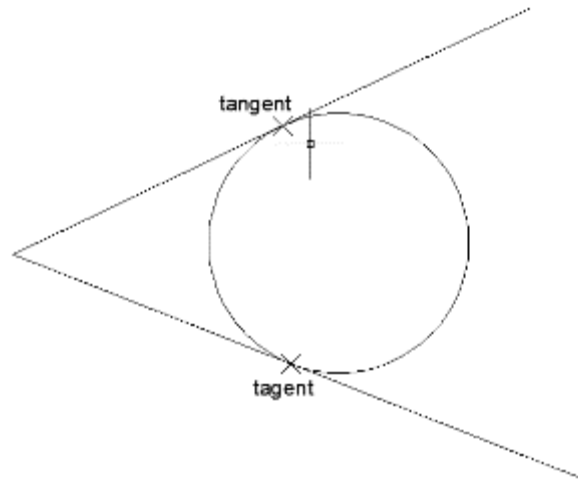

2 point   3 point

Ttr (tangent tangent radius) - circle will draw with both of object tangent to the circle or circle is tangent to the object with given radius.
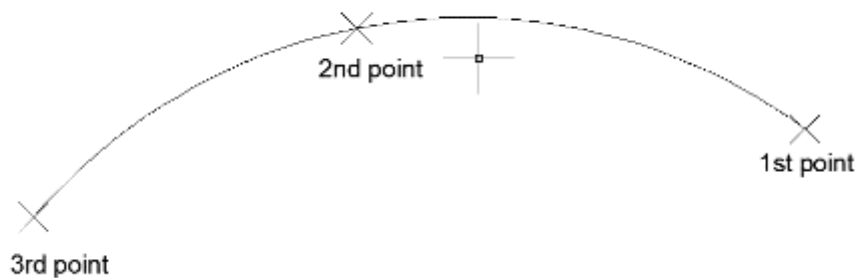

ARC –

DRAW>ARC> 3 POINT

You can also create various type of arc as per your need goto the "draw" choose ARC and choose the type of arc you need to use in arc options.

e.g. you can draw arc by choosing "start, center, end" or "start, center, angle" etc.

Command: _Arc

Specify start point of arc or [CEnter] : use one of point fixing method
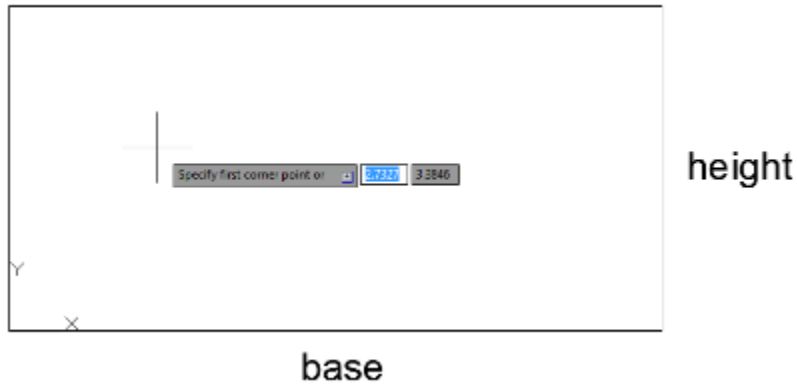

RECTANGLE- REC ↵

Rectangle command is used to create a rectangular box with the selection of the direction of the X axis and Y axis or say you have to choose first Base and then Height of the object.

Command: __Rectangle

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: enter the value of [base, height]
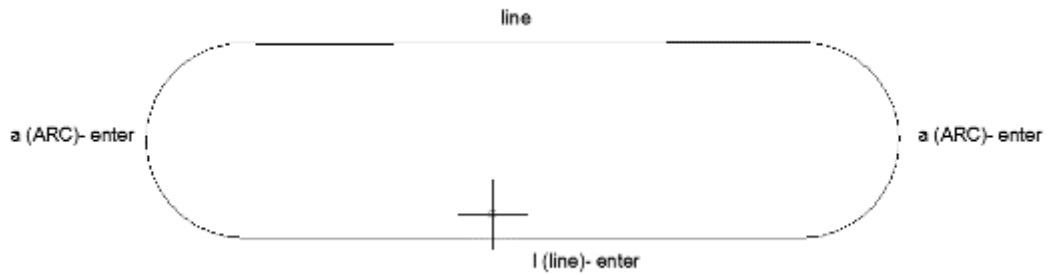

POLYLINE – PL ↵

Polyline is used to create a line and arc and in polyline you draw anything with polyline command it always considered as a single entities in the autocad.

Command: _pline
Specify start point:
Specify next point : 10
Specify next point: a ↵

POLYGON-

Polygon command create a regular polygon of no. of faces we can draw the minimum- 3 polygon and maximum- 1024 sides of the polygon inside the Autocad software.
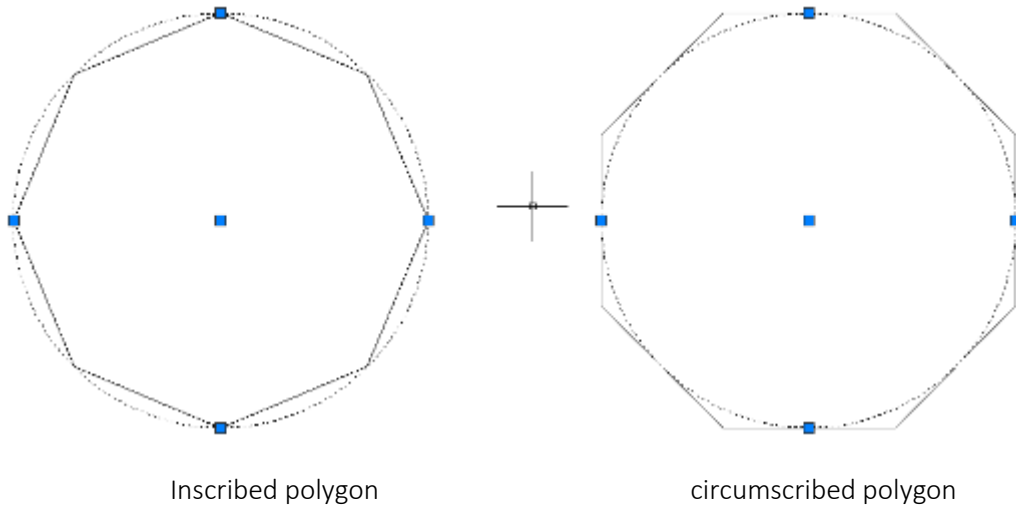
Command: __polyline

Enter the no. of sides = 8

Specify the center point of the polygon= choose by point fixing method.

Circumscribed or Inscribed

Choose circumscribed if you want to create polygon outside of a circle or choose inscribed if you want to create polygon inside the circle

Inscribed polygon   circumscribed polygon

MODIFY COMMANDS

Modify command is used for the modify the object we draw with the help of some modify commands like erase, copy, move, array etc. let's discuss each of important modify commands one by one

1. **COPY –**

   Copy command is used to copy the object to the another place without drawing it again and again just choose the copy command or type "copy" Enter. You will start copy command in autocad.

   Copy [Enter] – select object [enter]
         Specify the base point [select the center point of the circle]
         Move the object to the another place you want copy that object and click it once [repeat the same procedure when all the copy is done]
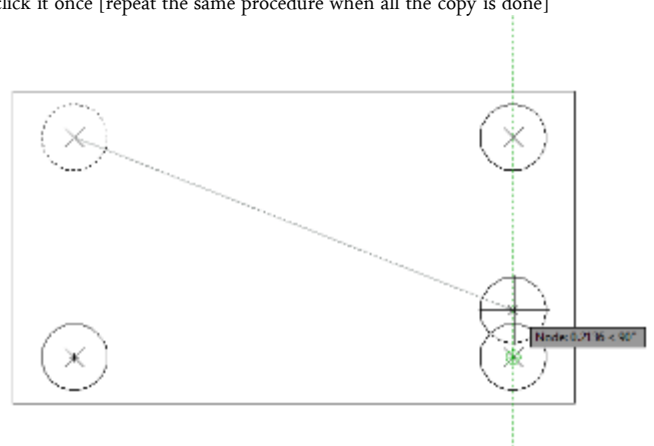
Move –

Move command is similar as copy command but in move command the object is moved instead of copy follow the above procedure same as for the move command.

Mirror – Mi ↵

Sometimes we need to create an object which is symmetrical about a particular axis. In such case the mirror command is most useful command which helps you to create the half of object mirrored about the reference axis without creating the full object which is also very time consuming.

Command- mirror [enter]

Select the object [select the object you want to mirror] – enter

Select first reference point – choose the top point of the reference line

Select the second point – choose the bottom point of the reference line

Delete the source object ? = No [enter]

MIRROR
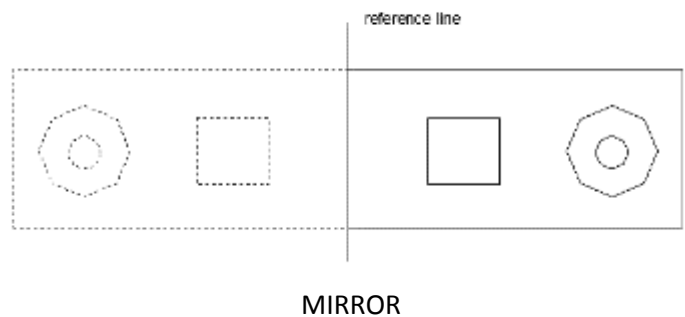
ARRAY= ar ↵

Array command is used to create the multiple copies of the same object in a regular way (in terms of rows and columns) when the use or row and columns to use the multiple copies this type of copy command is known as **Rectangular Array** When the multiple copies is in the way of circular pattern then the this type of array is known as **Polar Array.** Let's discuss each type of array one by one.
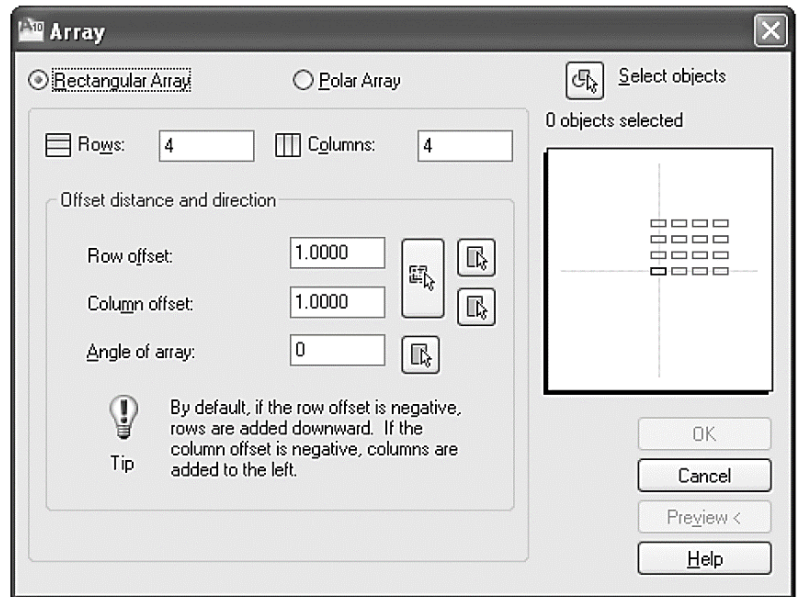

RECTANGULAR ARRAY
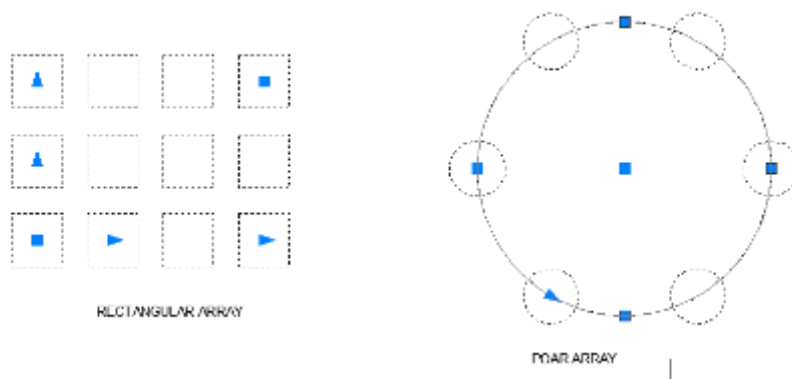
POAR ARRAY

Rectangular Array.

Command- array [enter]

- One box will appear in the autocad window screen (select the rectangular array shown in figure below)

- Click on the select object (now box will hide and choose the object you want to array) [enter]
- Choose the no. of columns and rows you need to copy
- Set the Row offset distance as per you need and columns distance also.
- Click on "Preview" if you want to check you array pattern or simply click ok.

POLAR ARRAY :-

Command- array [enter]

- One box will appear in the autocad window screen (select the Polar array shown in figure below)
- Click on the select object (now box will hide and choose the object you want to array) [enter]
- Choose the center point of the circle in which you want to be polar array work.
- Choose the no. of items
- Also choose the angle to fill (360^0 for the full circular array)
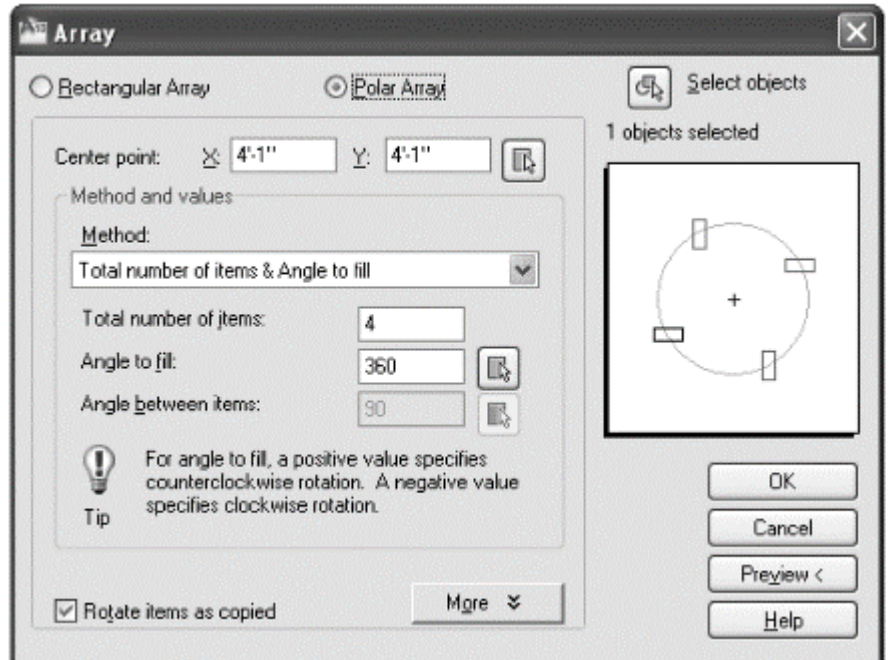
TRIM – tr ↵

trim command is used to remove the extra or unwanted object from the autocad this command works only when the interference between the two object is known in other words trim is trims off an object using cutting edges defined by other objects.
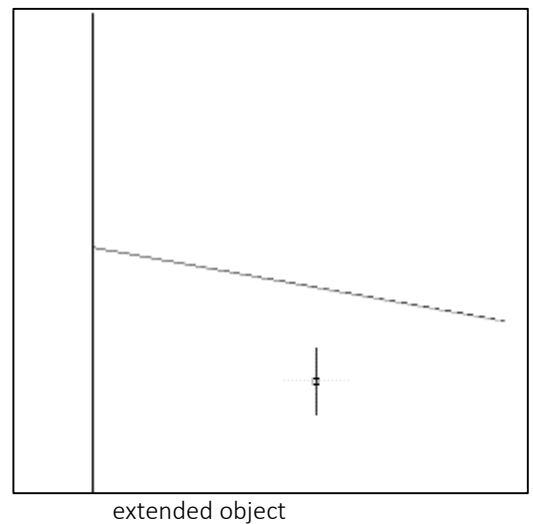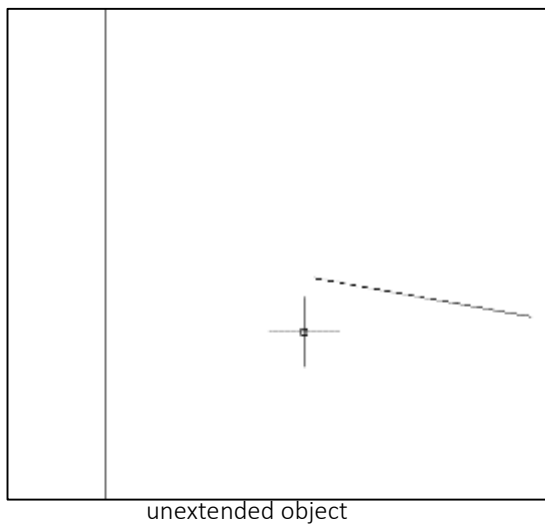
Command- trim [enter]

- select the object first you need to keep

- then select the object you want to remove

- or TYPE "TR ↵ ↵ " (TR DOUBLE ENTER and choose directly the object you want to remove)
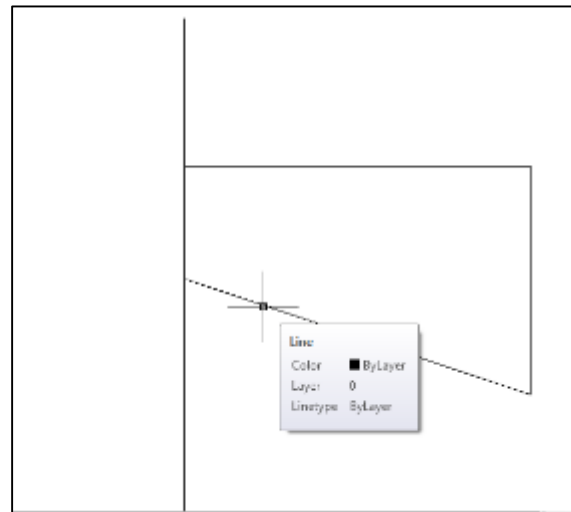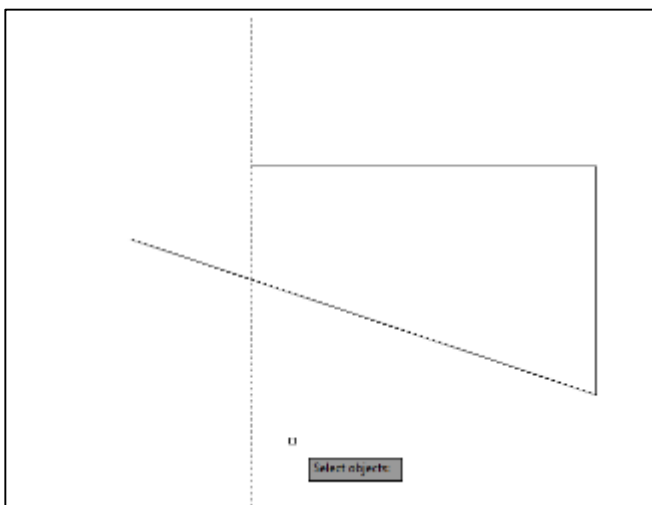
EXTEND:- EX ↵

Extend command is used to elongate the object this command gives you the facilities to extend an object to a boundary defined by the object.

unextended object   extended object

Command- trim [enter]

- select the object first you need to keep

- then select the object you want to remove

- or TYPE "**EX** ↵ ↵ " (EX DOUBLE ENTER and choose directly the object you want to EXTEND)

SCALE –

Scale command is used to scale the object as per the size you need the scale command is working in auto cad in the scene of double, triple, four times etc.

Command: Scale [Enter] – select object [enter]

- Specify the base point [select the center point of the circle or corner point of the rectangle or any other object]

- Scale the object put 2 for 2times of the present dimension, 3 for 3 times, 4 for the 4 times

- If you need to reduce the object put 0.5 for half, 0.8 for the 80 percent reduction to the object dimension
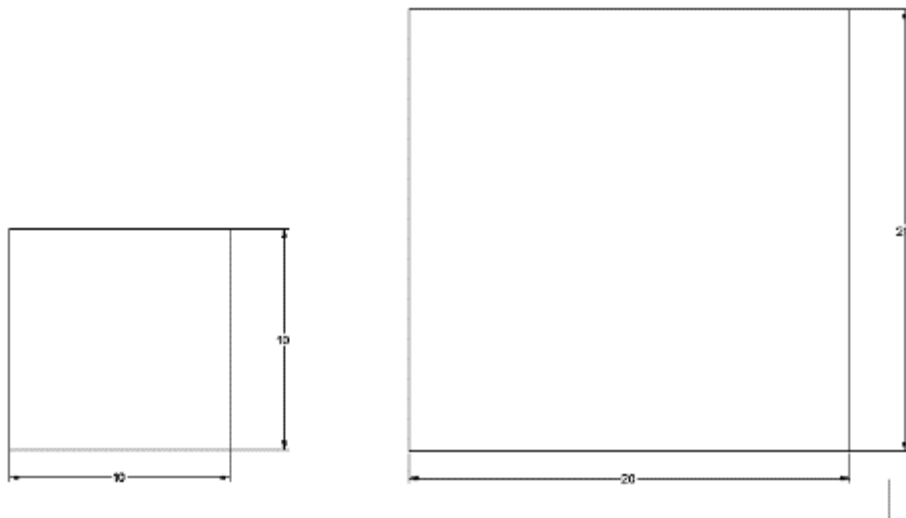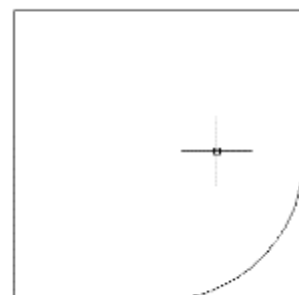
Scale object

FILLET –

Fillet command is used to round the sharp corner of the rectangular object.

Command- fillet [enter]

Type – R [enter] choose the radius you need

Select the first corner line
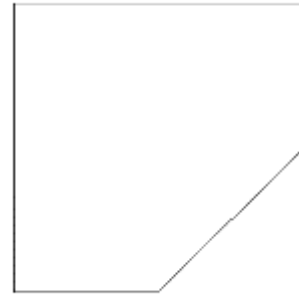
Select the second corner line .

CHAMFER:-

Chamfer is used to create a cut the corner of the rectangular object.

- Command- chamfer [enter]

Type – D [enter] choose the distance you need

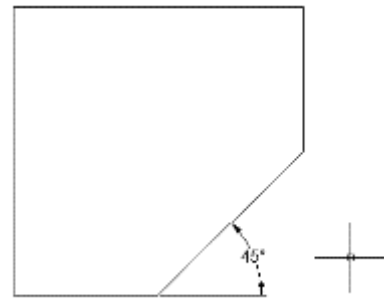Select the first corner line

Select the second corner line

- Command- chamfer [enter]

Type – A [enter] choose the desired angle you need to chamfer

Choose the distance you need

Select the first corner line

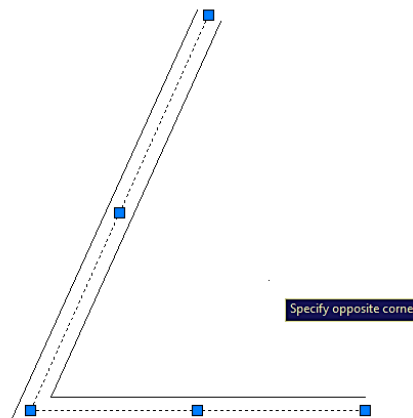Select the second corner line

OFFSET :-

Offset command is used to create an object parallel to the pervious drawn object . it is helpful command when we use to create multiple parallel object.

- Command- offset [enter]

Type – type offset distance say 5 for the 5mm [enter]

Choose the object which you want to create offset

Click on the left side if you want to create an object parallel to the left side or click right side if you want to create at right side.

DRAWING FOR PRACTICE

Dimensions in mm

R 50 R32 R50 R38 30° R15

40 R38 R38

R50

200

Dimensions in mm

Ø24 Ø60

10

Ø40 Ø36

100

24

A B

21 R6

60

33 R48

R18 R48

Dimensions in mm