

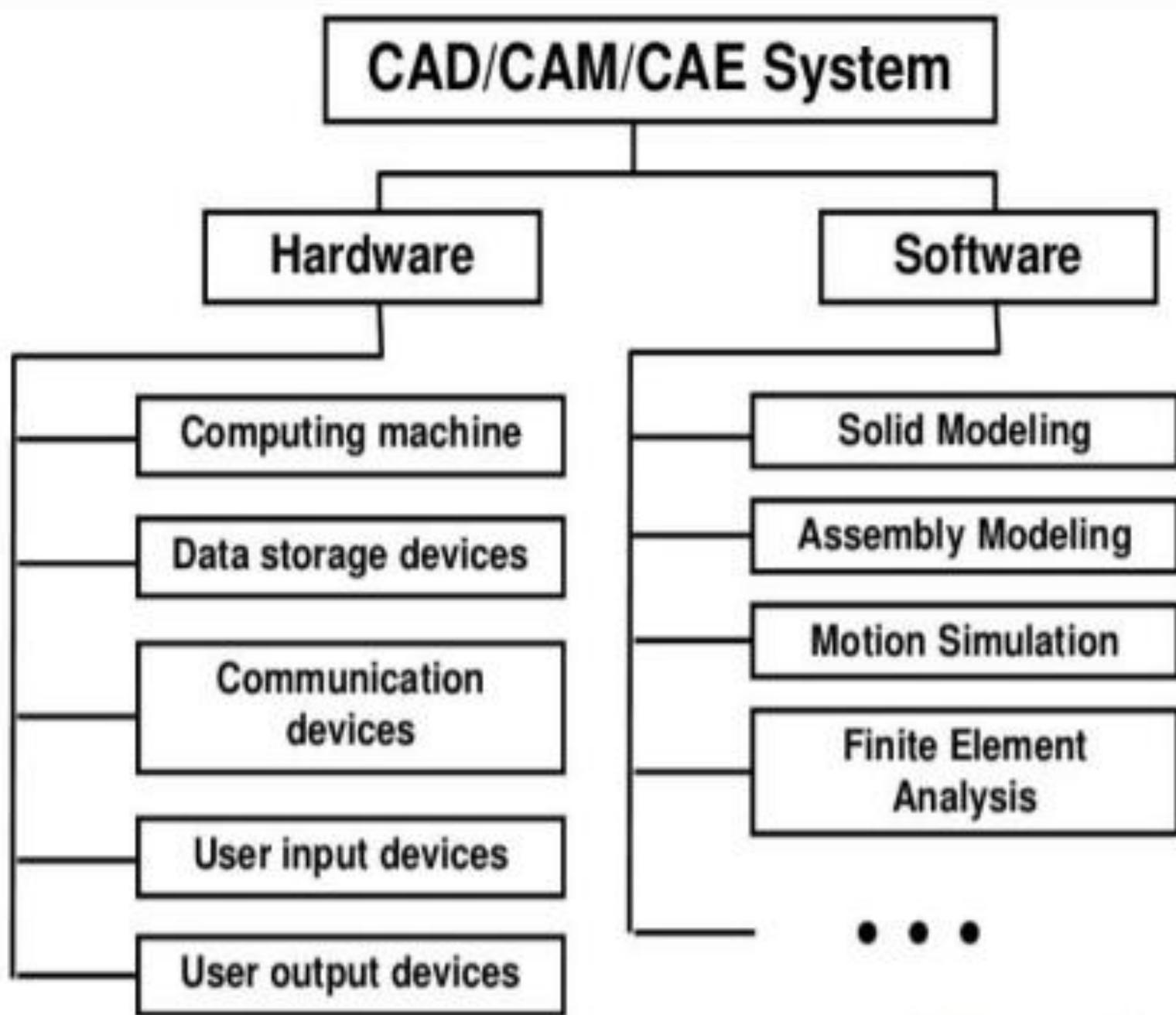
Introduction to CAD

- CAD (Computer Aided Drafting) is the process of making a drawing of an object on the display unit(i.e., screen) of a computer with the help of various tools or commands available in the CAD software.
- Due to its versatile capability and high benefits, modern designers and draftsman have adopted CAD in almost all disciplines of engineering. Mechanical engineering use CAD for design and drawing of a machine components, civil engineers use it for preparing plans and layouts of buildings, roads, dams, canals etc. It can be used to generate accurate drawings much faster as compared to conventional manual drafting.

Use of CAD

- CAD is used to accomplish design and layouts, design details and calculations, creating 3-D models, creating and releasing drawings, as well as interfacing with analysis, marketing, manufacturing, and end-user personnel.
- The modern CAD system make use of interactive computer graphics (ICG) in which computers are employed to create, transform and display data in the form of pictures and symbols. The basic components of ICG include a hardware, a software and a human designer.
- The use of CAD software tools allow the object to be viewed from any angle, even from the inside looking out. • One of the main advantages of a CAD drawing is that the editing is a fast process as compared to manual method

Components of CAD Systems



CAD SOFTWARE PACKAGES

Auto CAD

CATIA

Pro-E

SOLID WORKS

Corel CAD

Solid edge

Uni graphics

think three

Advantages of CAD

- Drawing with higher accuracy can be prepared faster than conventional manual drafting.
- Revisions and modifications in drawing can be done at any stage.
- It improves drafting productivity.
- Manpower requirement is reduced.
- Use of different color to customize product help to understand the design
- Drawing can be stored in electronic form and can be preserved for longer duration of time.
- Printing of the drawing can be done to any scale.
- Transfer of drawing is faster and cheaper.

Limitations of CAD

- It requires high storage capacity computer which is costly
- It requires CAD software packages also which involves some additional cost.
- Skilled operator is required to work on the CAD system.
- A huge initial investment is required in CAD system.

Introduction to Auto CAD

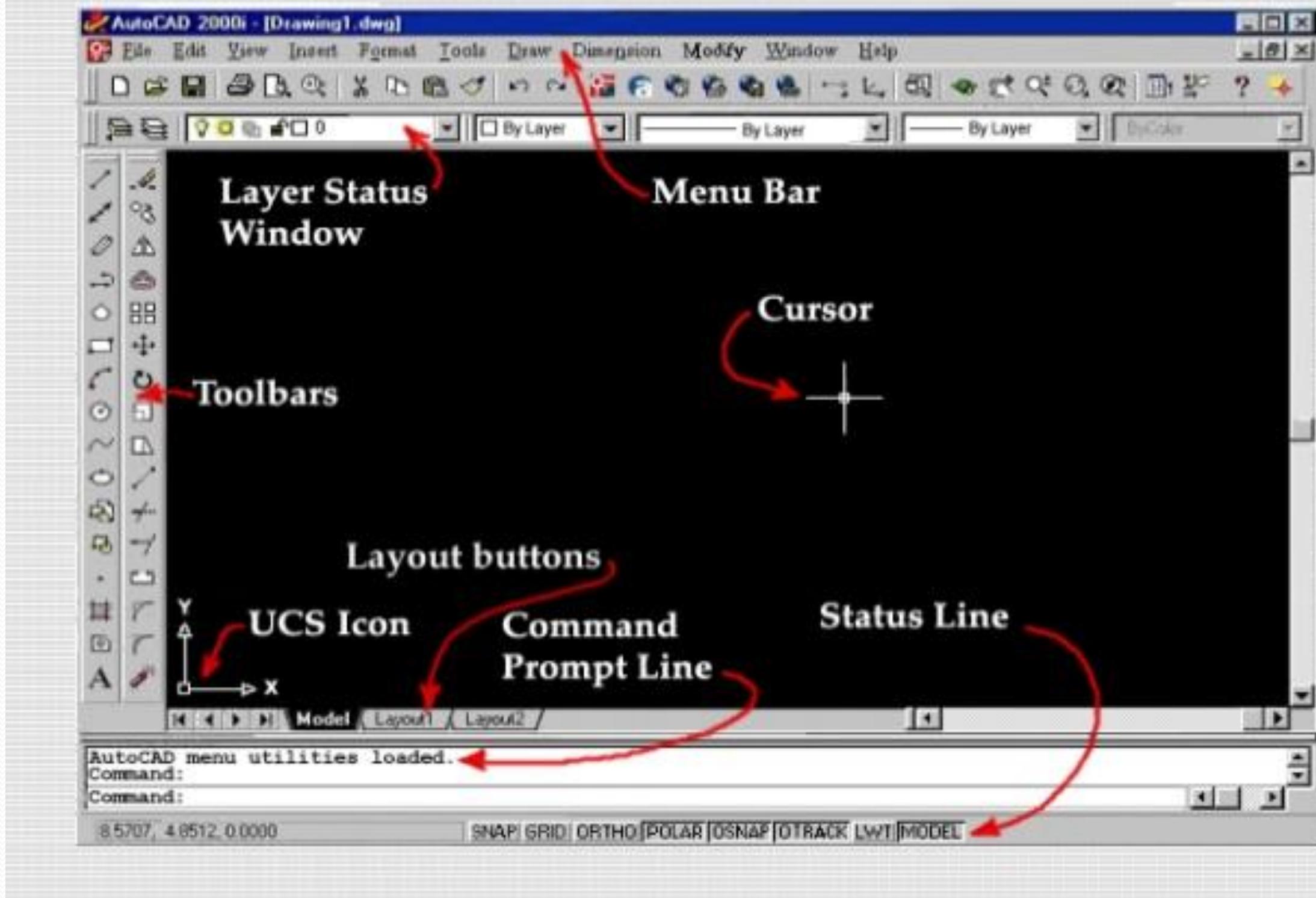
INTRODUCTION

- The Word AutoCAD is made up of two words "Auto (logo of company)" and CAD "(computer aided design)".
 - AutoCAD is 2D and 3D modeling software.
 - It is developed by Autodesk company.
 - Autodesk is an U.S.A based company.
 - It is widely used in industry for 2D drawing and 3D modeling.
 - In another way we can say that AutoCAD is a designing course , which is performed by the help of computer.
-

Version of AutoCAD

- AutoCAD software was firstly launched by Autodesk company in Dec. 1982.
 - It comes in India in 1988.
 - The first version of AutoCAD was R1 after that R2,R3,R4..... and so on.
 - In 2000,Autodesk launched a version of AutoCAD 2000 after that 2001,2002..... so on.
 - This time, we have the latest version of AutoCAD is 2014,which is launched on 27th march 2013.
 - Latest version is easy to use and over come the difficulties of old version.
-

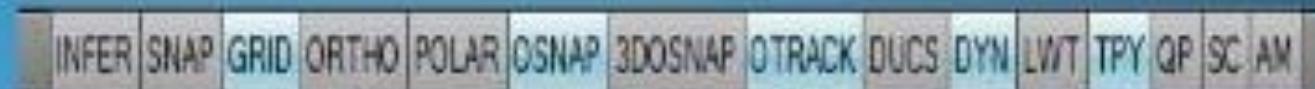
AutoCAD Screen



STATUS TOOL BAR

➤ GRID

➤ ORTHO



➤ POLAR

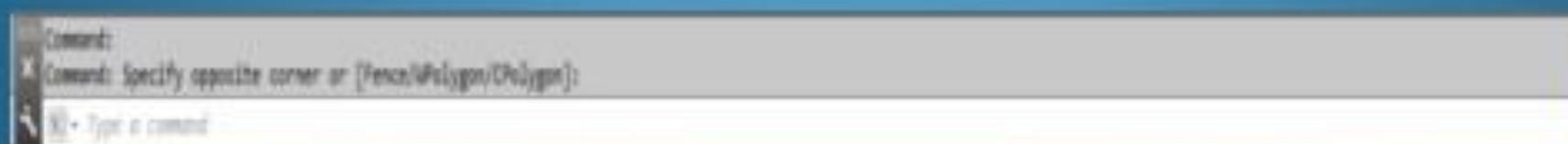
➤ OSNAP

**CADD
CENTRE**

COMMAND TOOL BAR

- Command tool bar is used for activating all the commands which are used in Auto Cad by typing.

- It is also the easiest way to give the command to the command bar and also to activate it.



**CADD
CENTRE**

APPLICATIONS FOR 2D DESINING

- Mechanical
- Electrical
- Civil
- Architecture

**CADD
CENTRE**

APPLICATIONS

- Automobile industries
- Power plant industries
- Piping industries
- Mass production industries
- Aerospace industries
- Now a days Auto Cad is almost used in all the industries and in every fields like (Civil, Electrical etc...)

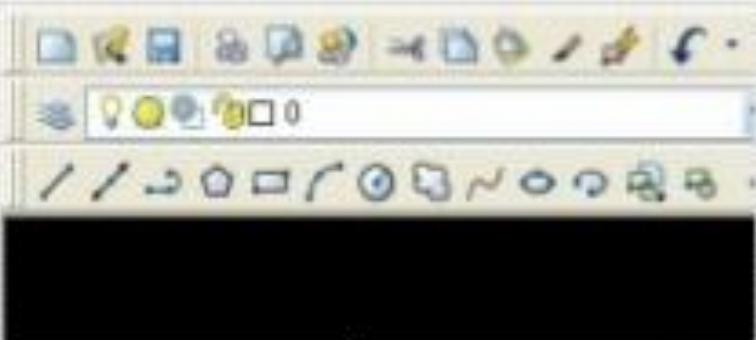
**CADD
CENTRE**

Way to Provide command

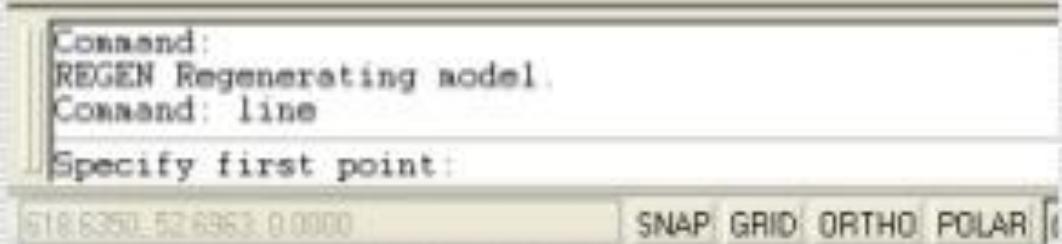
1.



2.



3.

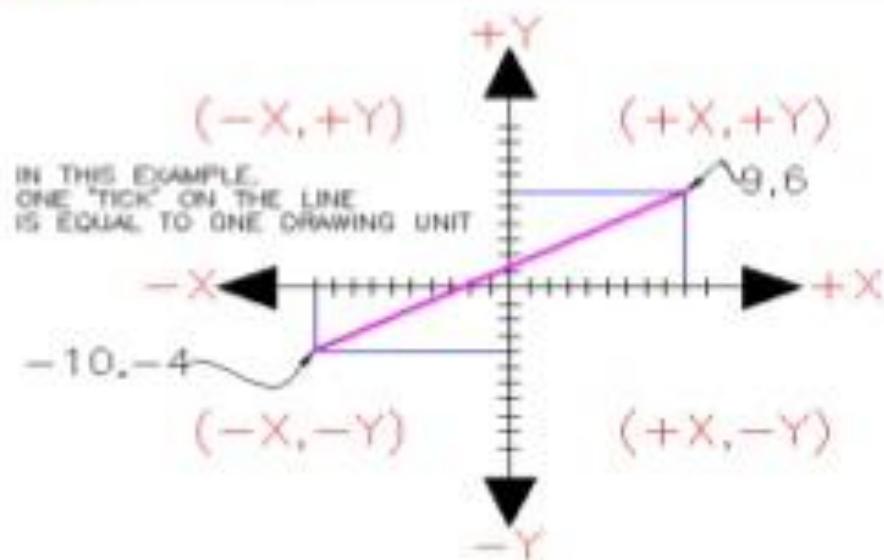


HOW AutoCAD WORKS

- ❖ There is a co-ordinate system used in AutoCAD.
 - ❖ Every drawing shows its co-ordinate.
 - ❖ In above next slide the line shows its co-ordinate that is (9,6) and (-10,-4).
 - ❖ There is so many commands like copy, move ,rotate ,mirror in 2D, path array , rectangular array, polar array & more.
 - ❖ Different types of drawing can be made in the same time by using a command that is LAYER.
 - ❖ Using line , arc , circle , rectangle , ellipse & polygon , so many drawing of different type can be made.
-

CO-Ordinate System

- Every thing that we draw in AutoCAD is exact.
- All object drawn on screen is based on simple X-Y co-ordinate system.
- In AutoCAD it is known as world co-ordinate system (WCS).
- We are drawing a line, so we have two points A(-10,-4) and B(9,6). As shown in figure.



The UCS and WCS

- The AutoCAD world is 3 dimensional. However, if we want to draw a 2d object, such as a plan or a section, we will use only 2 dimensions (x and y).
 - WCS (world coordinate system) is the imaginary plane that is parallel to the ground. It is the default coordinate system.
 - Modifications made to the World Coordinate System (WCS) result in a User Coordinate System (UCS). It is the plane that you work on. It enables the user to draw 3 dimensional objects.
 - To create a new UCS, type `ucs` on the command window, then say New and specify 3 points on your new UCS plane.
-

Polyline
Line

Polygon

Arc

Spline

Ellipse arc



Construction line

Rectangle

Circle

Ellipse

DRAW TOOLBAR

mechcadcam.com

MODIFY TOOL BAR

- ERASE
- COPY
- OFFSET
- MOVE



**CADD
CENTRE**

Some 2D Commands

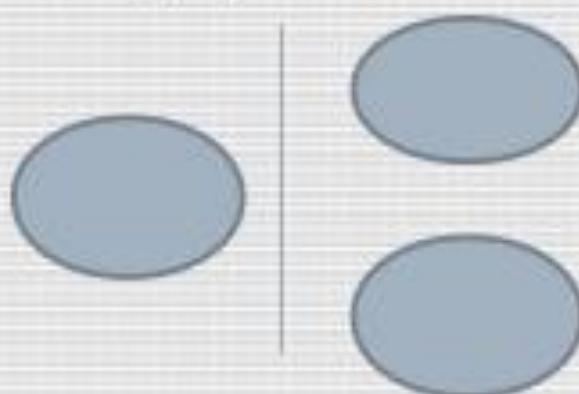
Chamfer



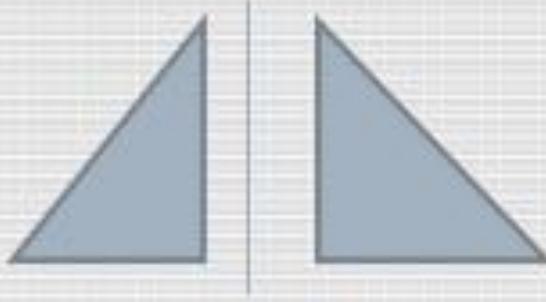
2. Fillet



Copy

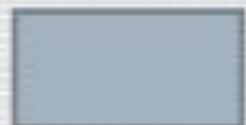


4. Mirror

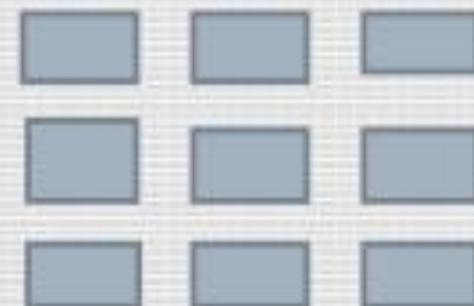


2D Commands

Scale



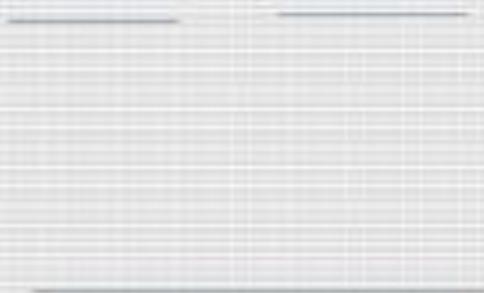
2. Array



Extend



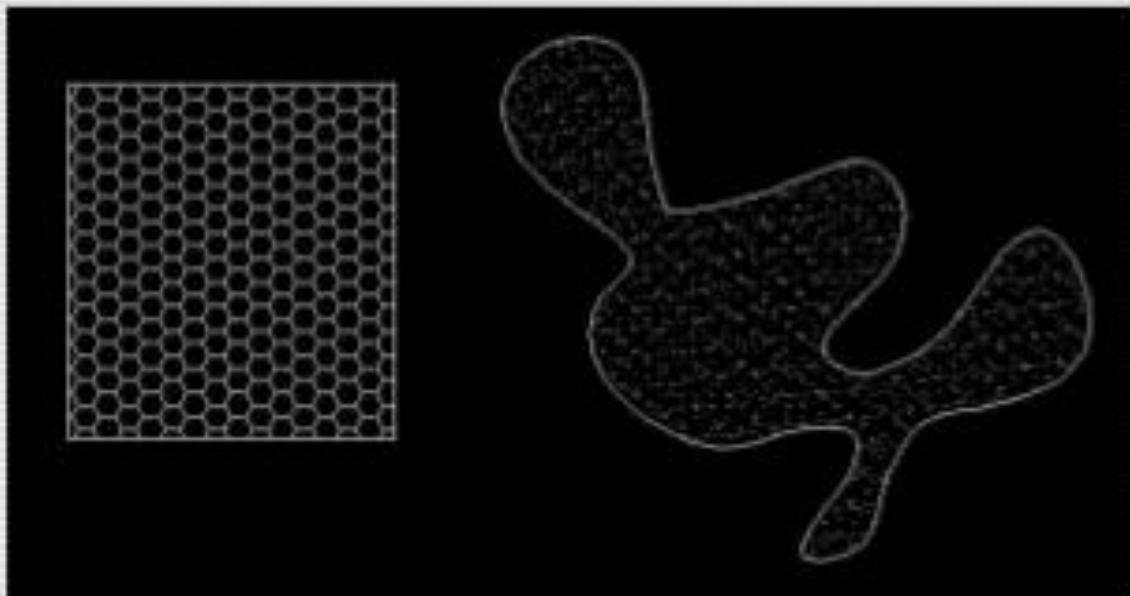
4. Join



Hatching

- ❑ Hatching is used to add shaded patterns to objects and shapes within an Autocad drawing. Hatch patterns can be used to indicate a material to be used, such as a concrete hatch.

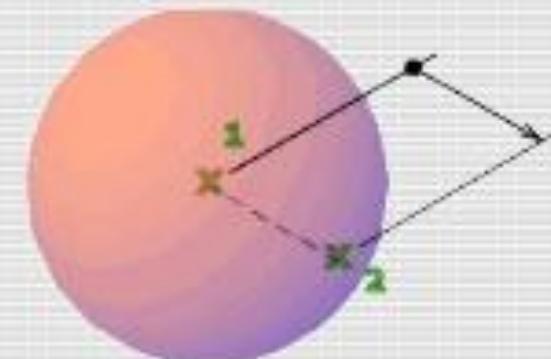
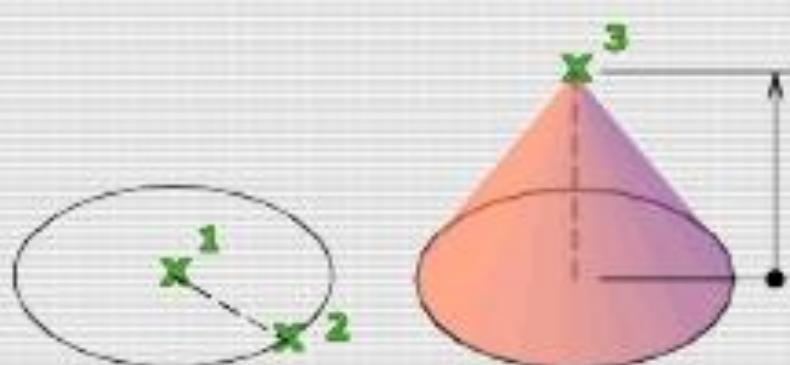
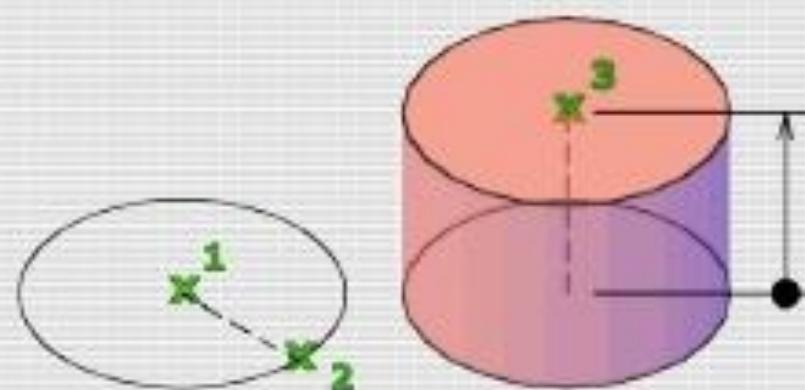
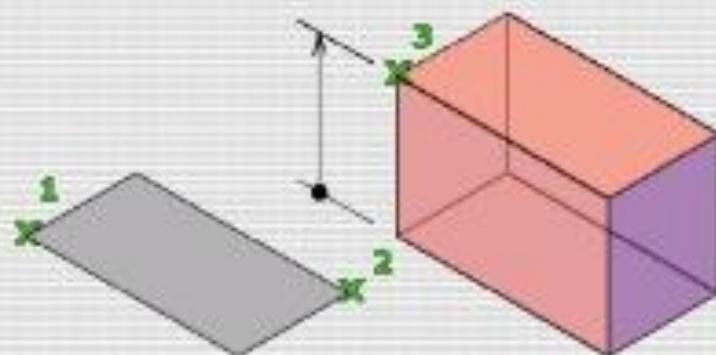
- ❑ You will pick:
 - Pattern
 - Scale
 - Angle
 - points



3D Modeling

- Solids contain the “mass properties” of 3D objects.
- You can use the Solids toolbar for readily accessible objects
 - Box
 - Cylinder
 - Wedge
 - Torus
 - Cone
 - Sphere
- You can use the Boolean operations of more complicated shapes.
 - Union (join two solids)
 - Subtract (carve out the second solid from the first)
 - Intersection (only the common area)

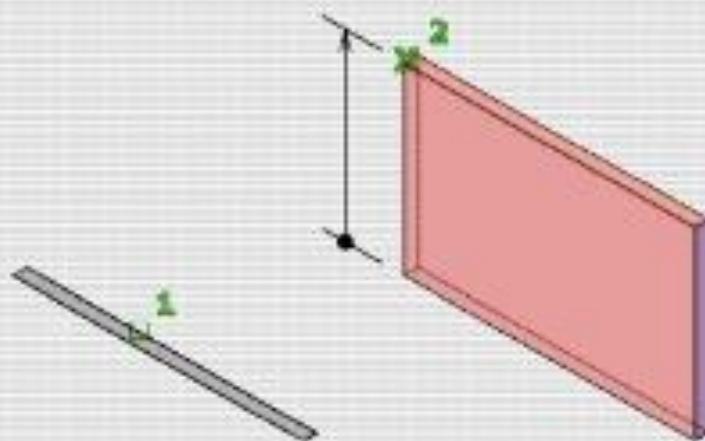
Some 3D Object



3D Commands

EXTRUDE

Extrusions can extend in the Z direction or be set to taper or follow a path. You can extrude an open or closed object to create a 3D surface or solid.



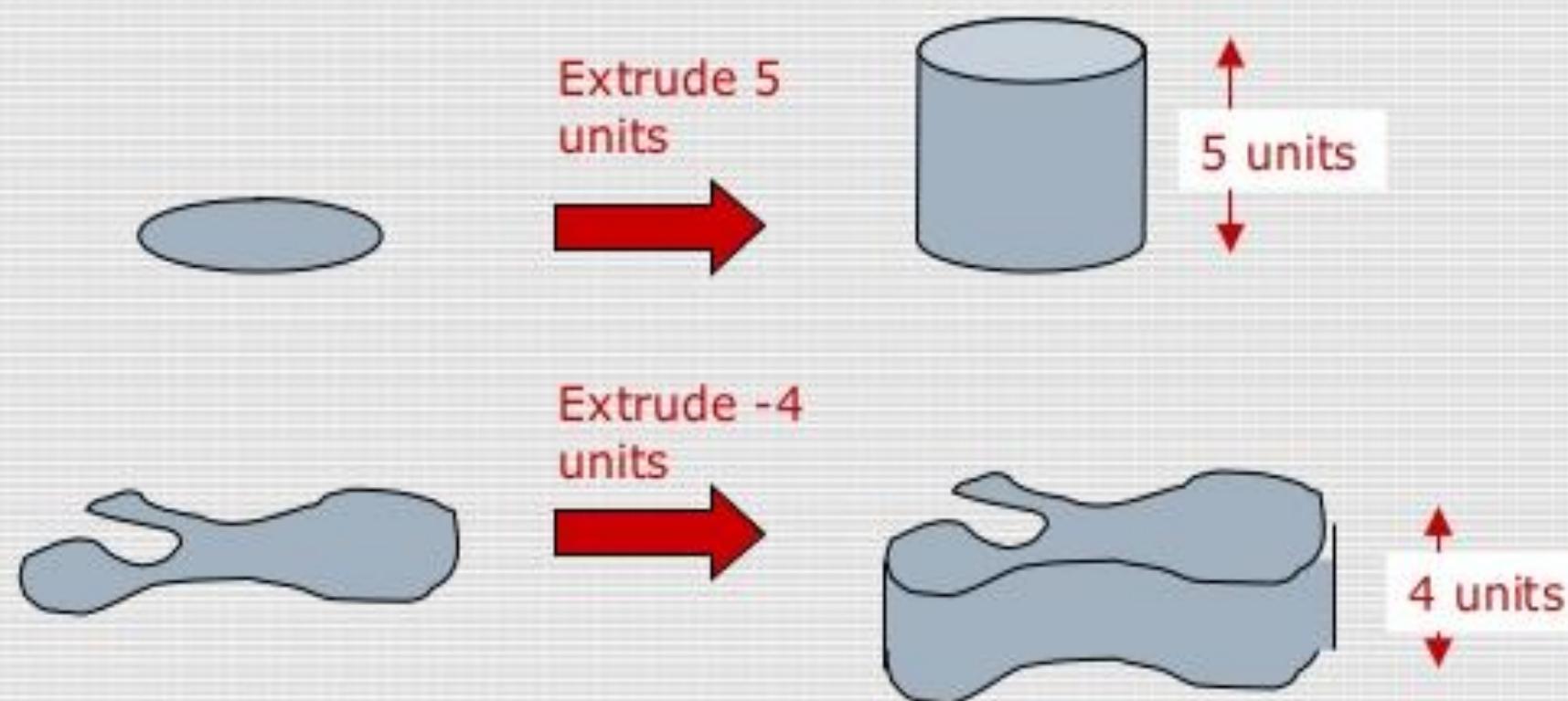
Vscurrent

It is used for setting of view style of diagram, which may be.....

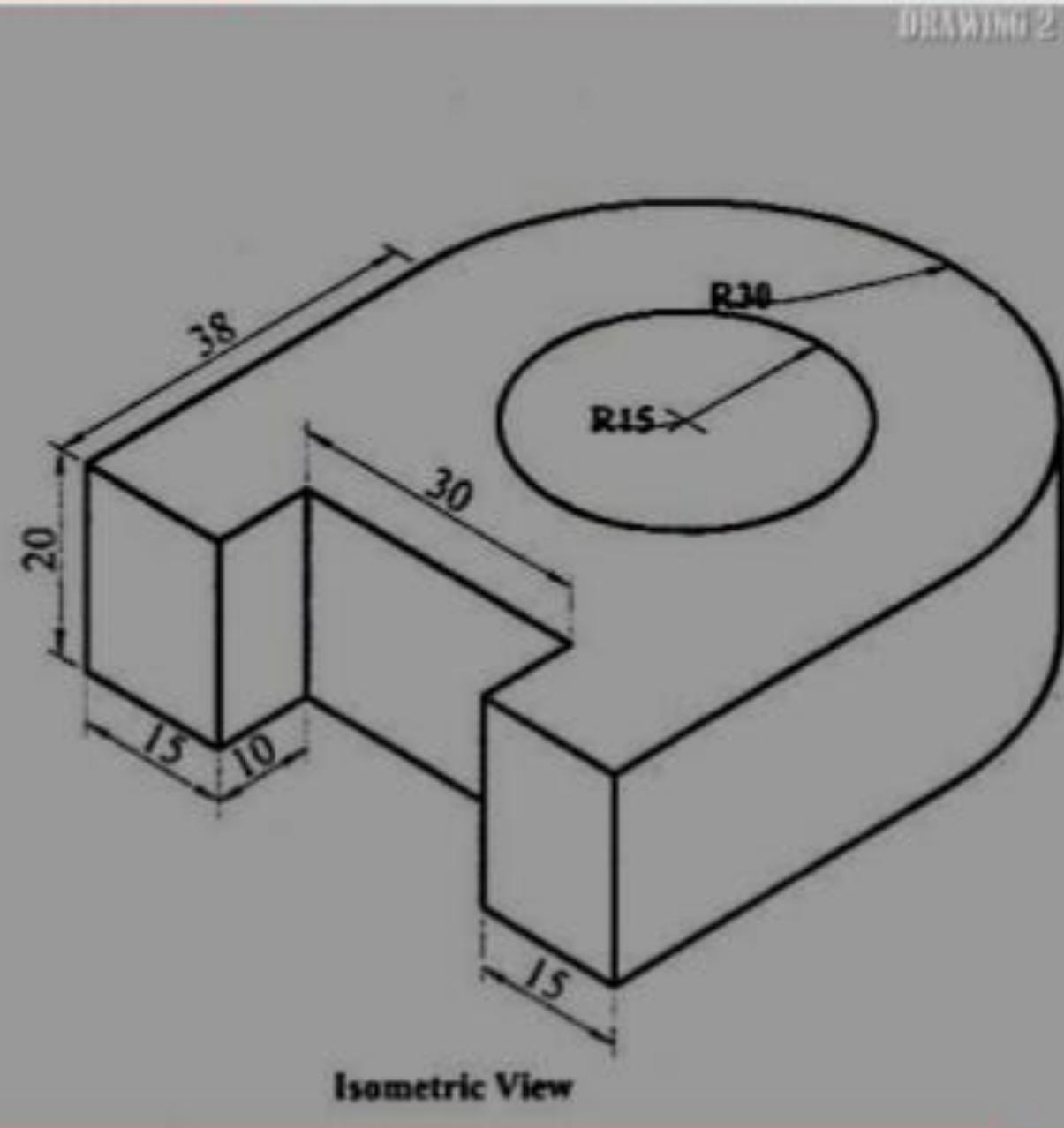
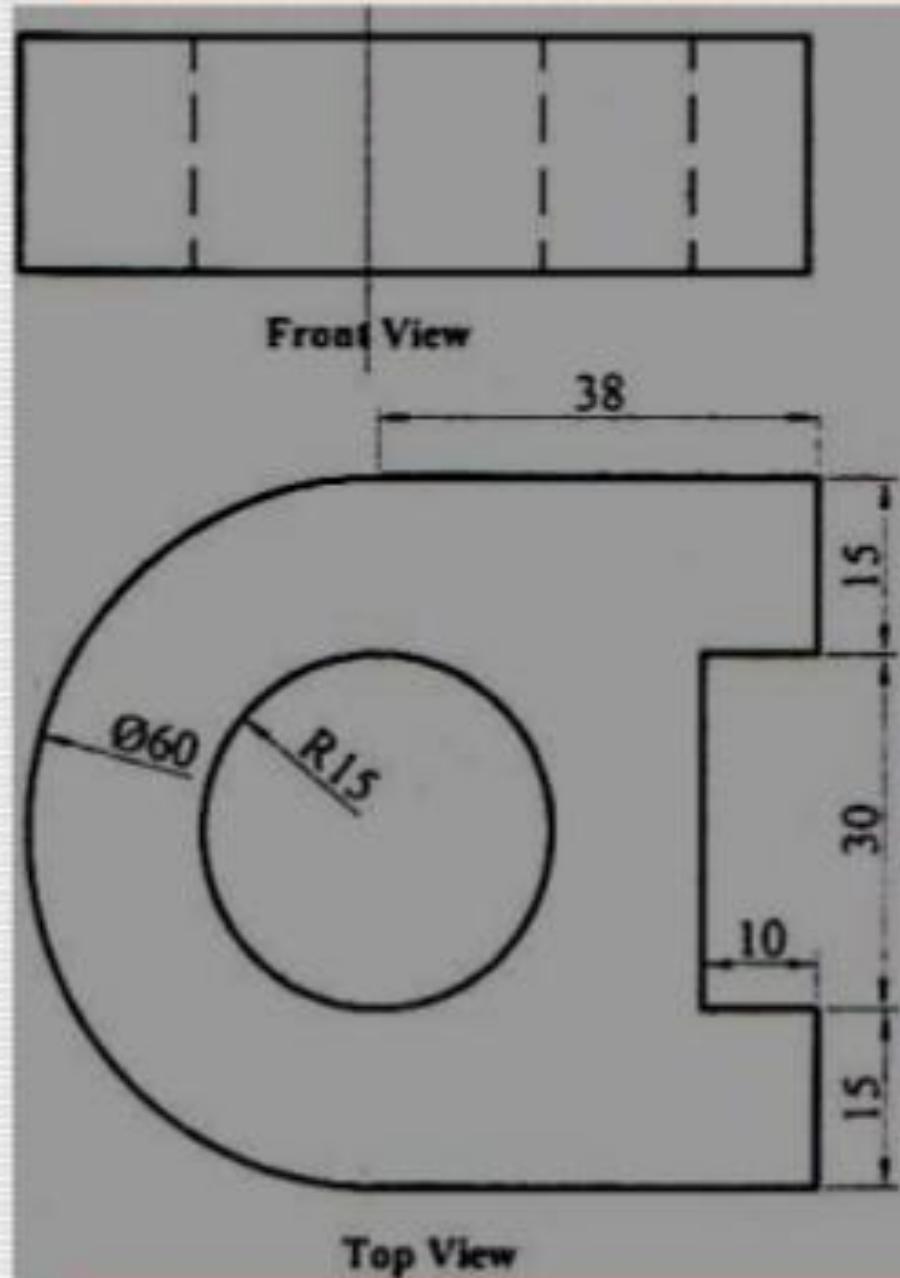
- 1.2D wireframe.
- 2.Realistic.
- 3.3D Hidden.

Use of Extrude Command

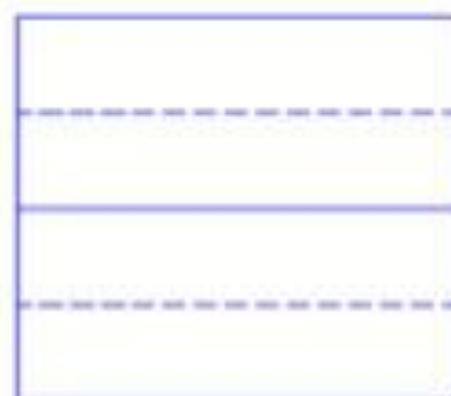
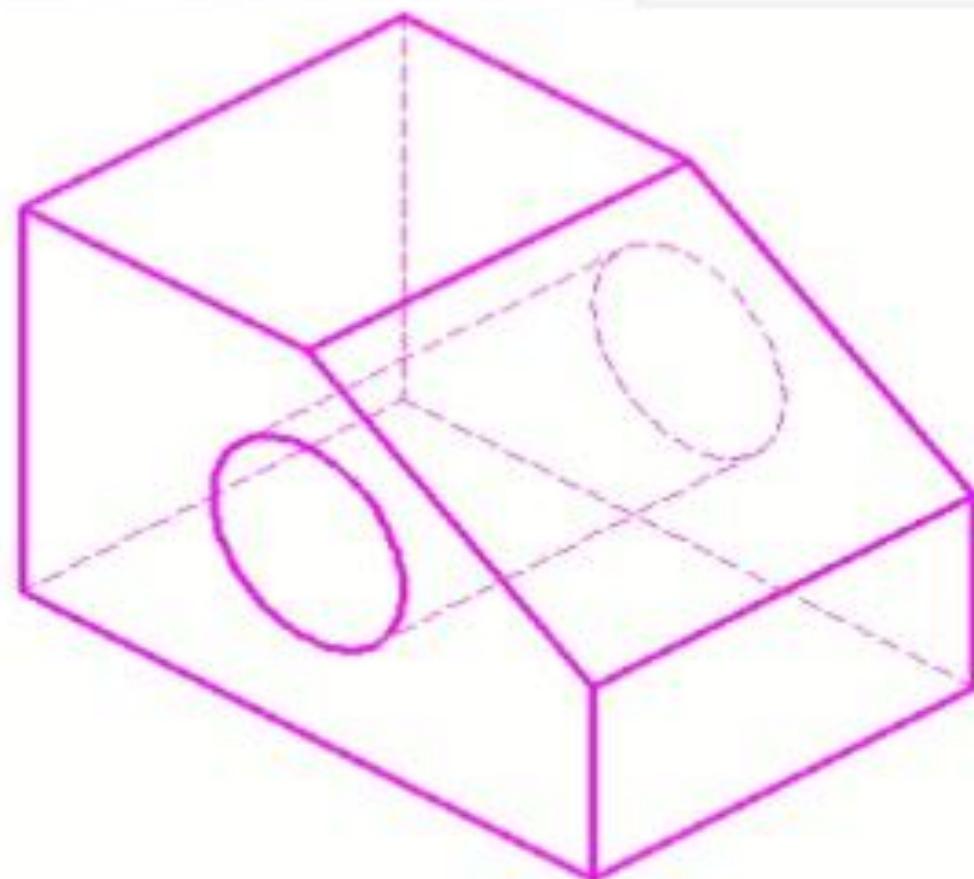
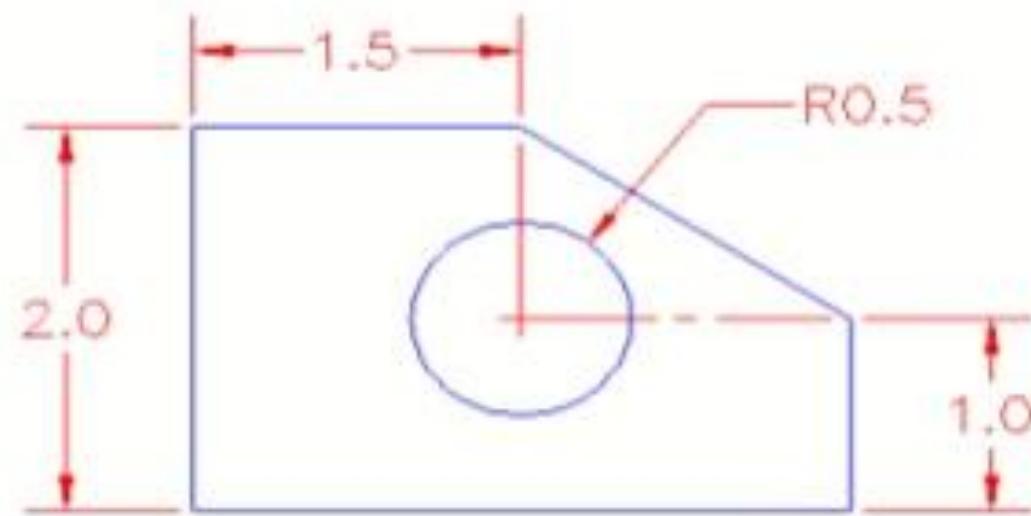
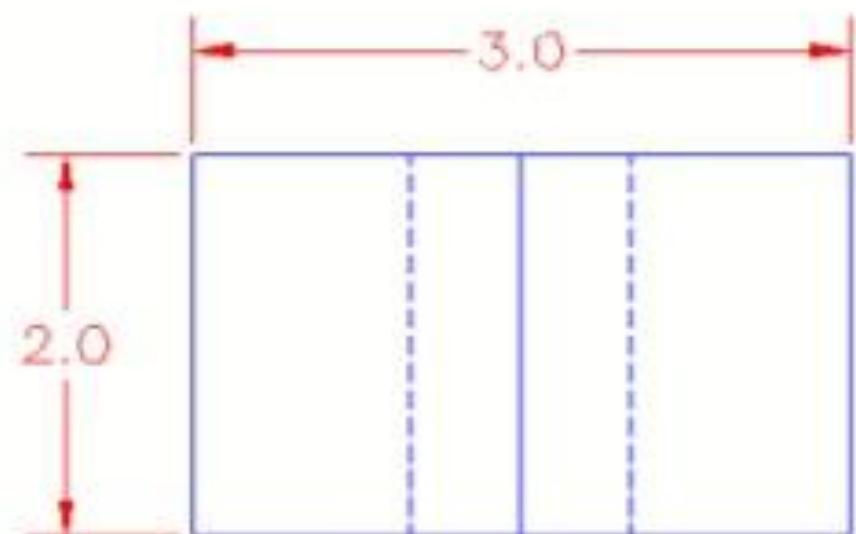
- If you “**Extrude**” a surface into the third dimension, you simply add a thickness in section. This basically is same as creating a “solid” object .



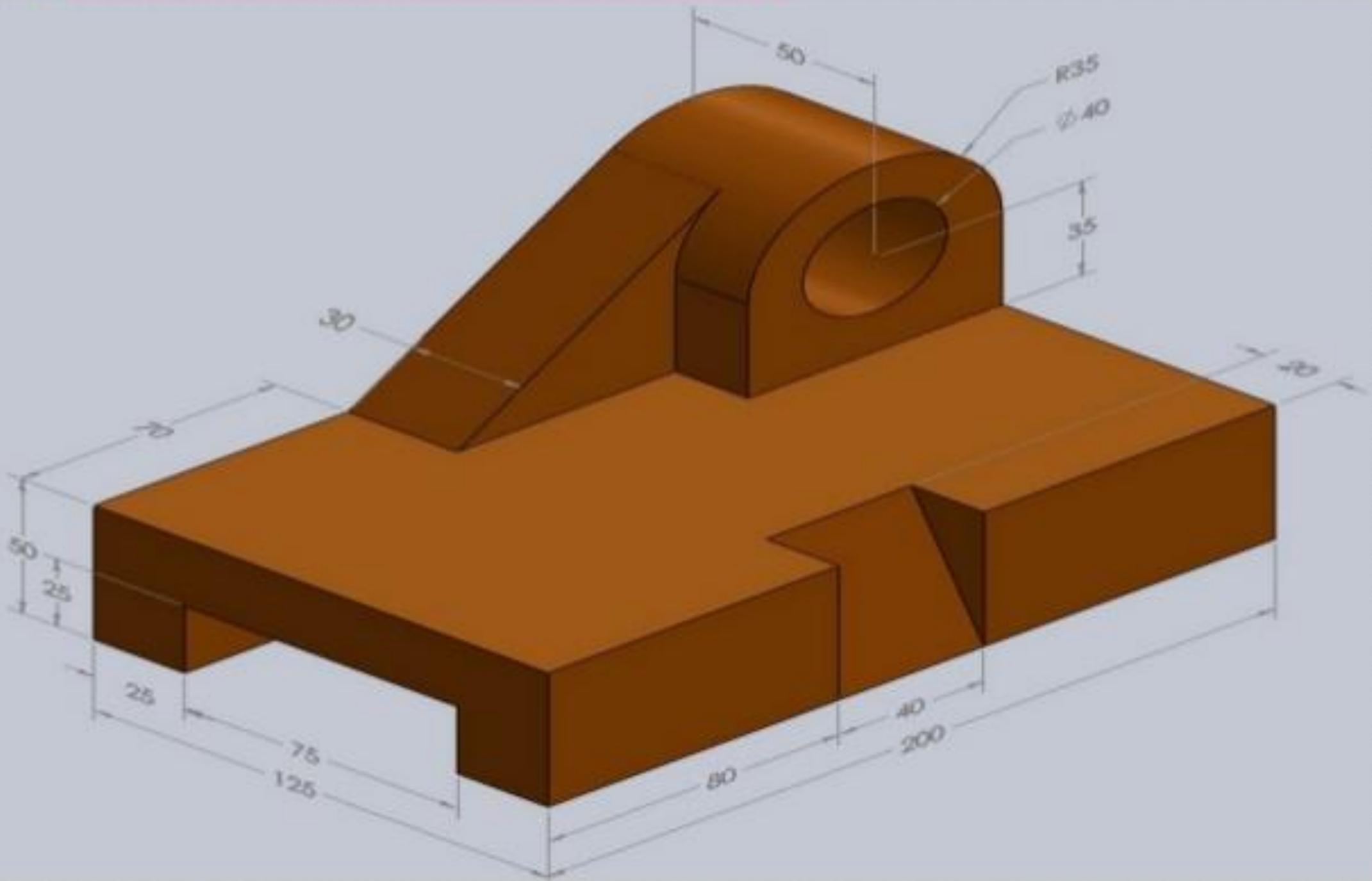
2D Project (1)



2D Project Work (2)



3D Project

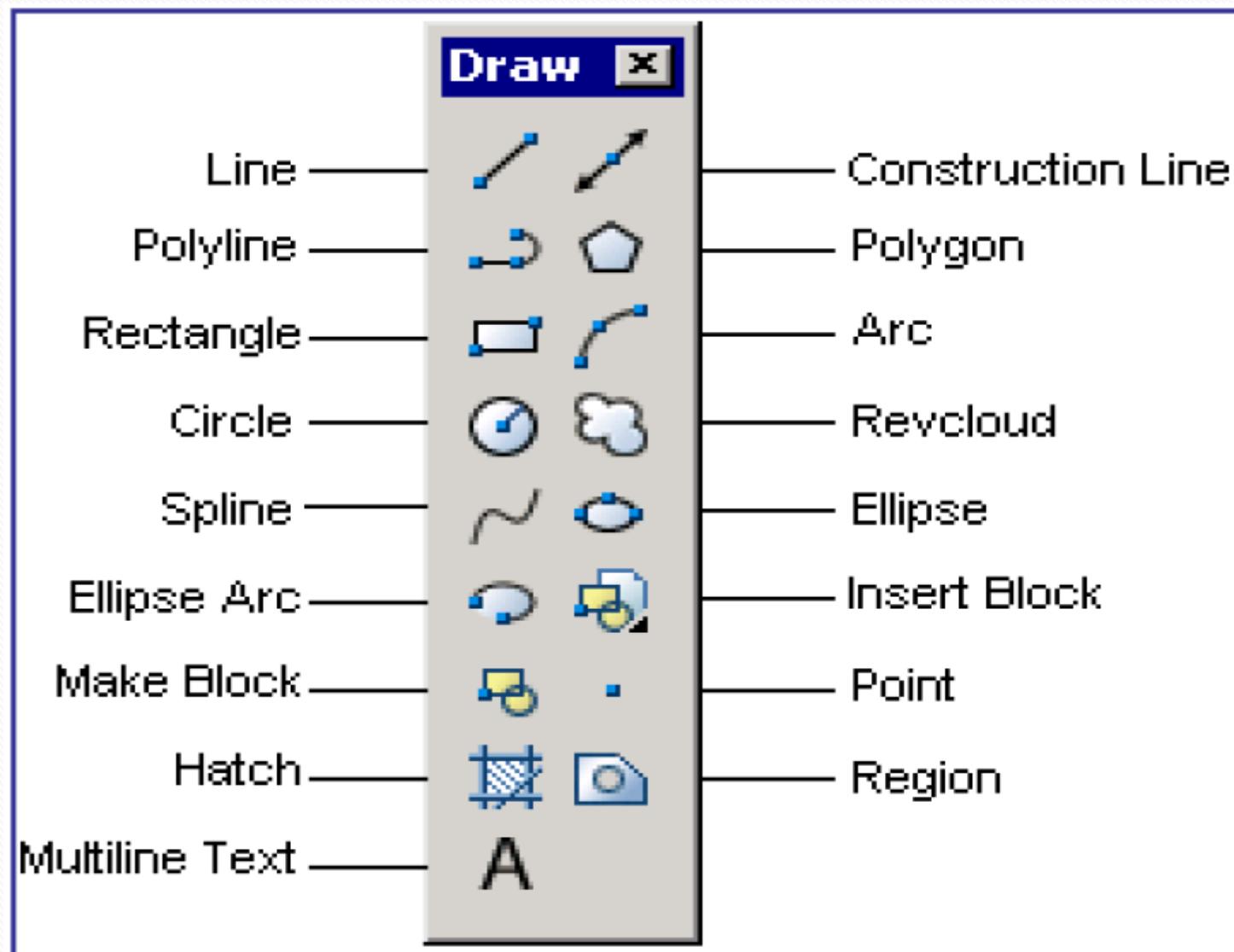


Benefits/Use of AutoCAD

- Quickly create designs.
 - Improved quality over hand drafting.
 - Easily modify.
 - More Accuracy.
 - Easy to transfer.
 - Long time save.
-

VARIOUS COMMANDS USED IN AUTO CAD

1. BASIC DRAWING COMMANDS OR DRAW ENTITIES



Line Command

The LINE command is used to draw a line, the line drawn may be continued with the same command and it is terminated by pressing ENTER key or right mouse button. Then select ENTER. The execution of LINE command is given as follows.

Command

Specify first point

Specify next point or [undo]

Specify next point or [undo]

Specify next point or [close/undo]

- : LINE or L.
- : Select any point in the drawing area on the screen using left mouse button.
- : Select second point using mouse.
- : Select third point using mouse.
- : C to close the lines.

Coordinate Systems in AutoCAD

There are four coordinate systems used in AutoCAD to locate a point on the screen. In AutoCAD, 2-D drafting is done by considering the screen as the XY plane i.e. X values are considered horizontally and Y values are taken vertically. By default, the left lower corner on the screen is considered as the origin $(0, 0)$. Auto CAD uses the following coordinate systems:

- (i) Absolute coordinates
- (ii) Relative coordinates
- (iii) Polar coordinates
- (iv) Direct distance entry.

Absolute Coordinates

In this method, the points are located to draw a line with respect to the origin (0, 0). To mark a point, value is given in pairs for X-coordinate value followed by Y-coordinate. Consider the following example to generate the given drawing.

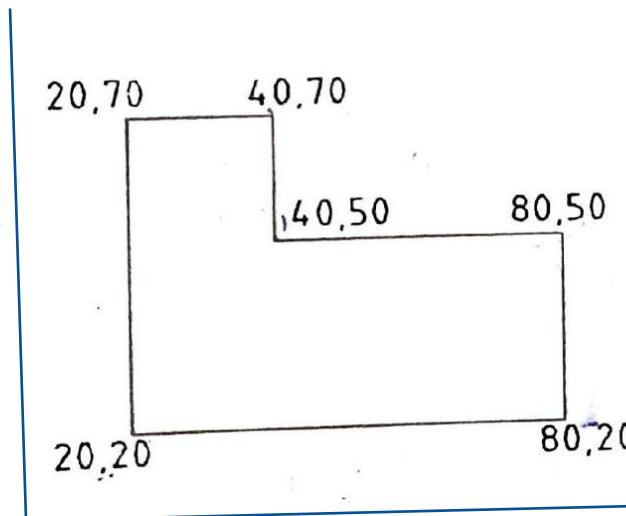


Figure 15.4

Command

Specify first point
Specify next point or [undo]
Specify next point or [undo]
Specify next point or [close/undo]
Specify next point or [close/undo]

: LINE Press ENTER key or mouse left button.
: 20, 20 Press ENTER
: 80, 20 Press ENTER
: 80, 50 Press ENTER
: 40, 50 Press ENTER
: 40, 70 Press ENTER
: 20, 70 Press ENTER
: 20, 20 Press ENTER
 Press ENTER to complete the drawing.

Relative Coordinates

In this method, the points are located to draw a line with reference to the previous point. Consider the following example to generate the given drawing.

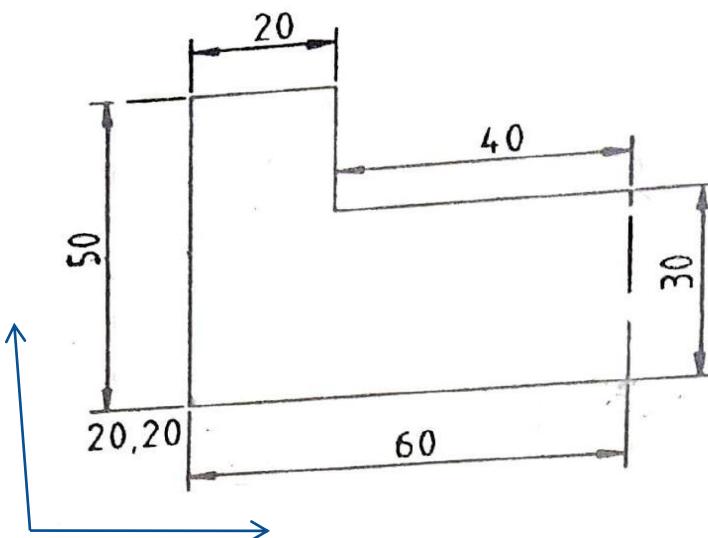


Figure 15.5

Command

Specify the first point

: LINE

: 20, 20

Specify the next point or [undo]

: @ 60, 0

Specify the next point or [undo]

: @ 0, 30

Specify the next point or [close/undo]

: @ -40, 0

Specify the next point or [close/undo]

: @ 0, 20

Specify the next point or [close/undo]

: @ -20, 0

Specify the next point or [close/undo]

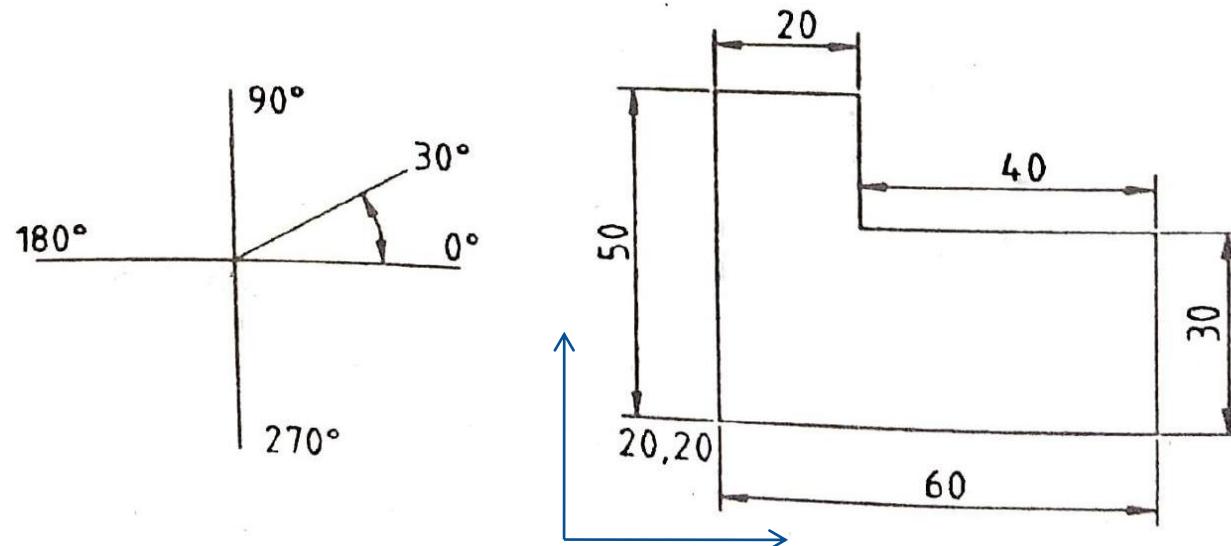
: @ 0, -50

Specify the next point or [close/undo] : Press ENTER to complete the drawing.

NOTE: press ENTER after giving the values.

Polar Coordinates

In this method, the points are located to draw a line by defining the distance of the point from the current position and the angle made to that line. The angle referred in AutoCAD is given in Fig. 15.6(i). Consider the following example to generate the drawing given in Fig. 15.6(ii).



Command

Specify the first point
Specify next point or [undo]
Specify next point or [undo]
Specify next point or [close/undo]
Specify next point or [close/undo]

LINE
20, 20.
@ 60<0
@ 30<90
@ 40<180
@ 20<90
@ 20<180
@ 50<270
Press ENTER to complete the drawing.

Direct Distance Entry

In this method, the points are located to draw a line using the distance entry in the direction of the cursor. Consider the following example to generate the drawing given in Fig. 15.7.

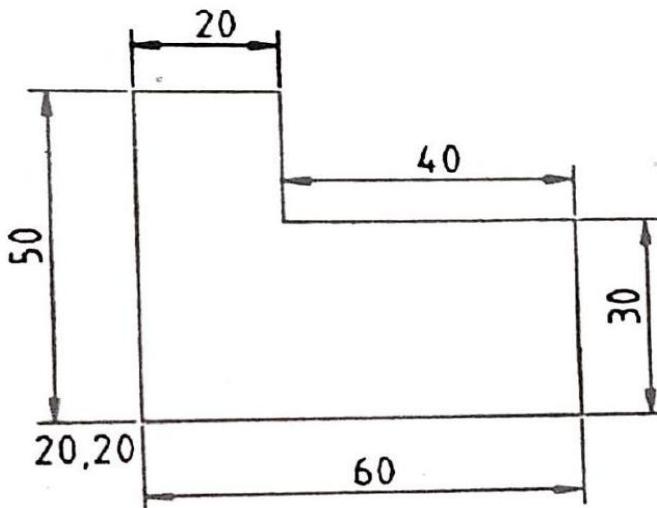


Figure 15.7

Command

Specify the first point

: LINE [ortho on]

Specify the next point or [undo]

: 20, 20 *move mouse horizontally right*

Specify the next point or [undo]

: 60 *move mouse vertically up*

Specify the next point or [undo]

: 30 *move mouse horizontally left*

Specify the next point or [undo/close]

: 40 *move mouse vertically up*

Specify the next point or [undo/close]

: 20 *move mouse horizontally left*

Specify the next point or [undo/close]

: 20 *move mouse vertically down*

Specify the next point or [undo/close]

: 50

Press ENTER to complete the drawing.

NOTE: To draw this drawing, the Ortho mode is on. Use F8 key to activate ortho on/off.

Example 9.5. State the series of auto CAD command steps to reproduce the object shown in Fig. 9.5 using line command.

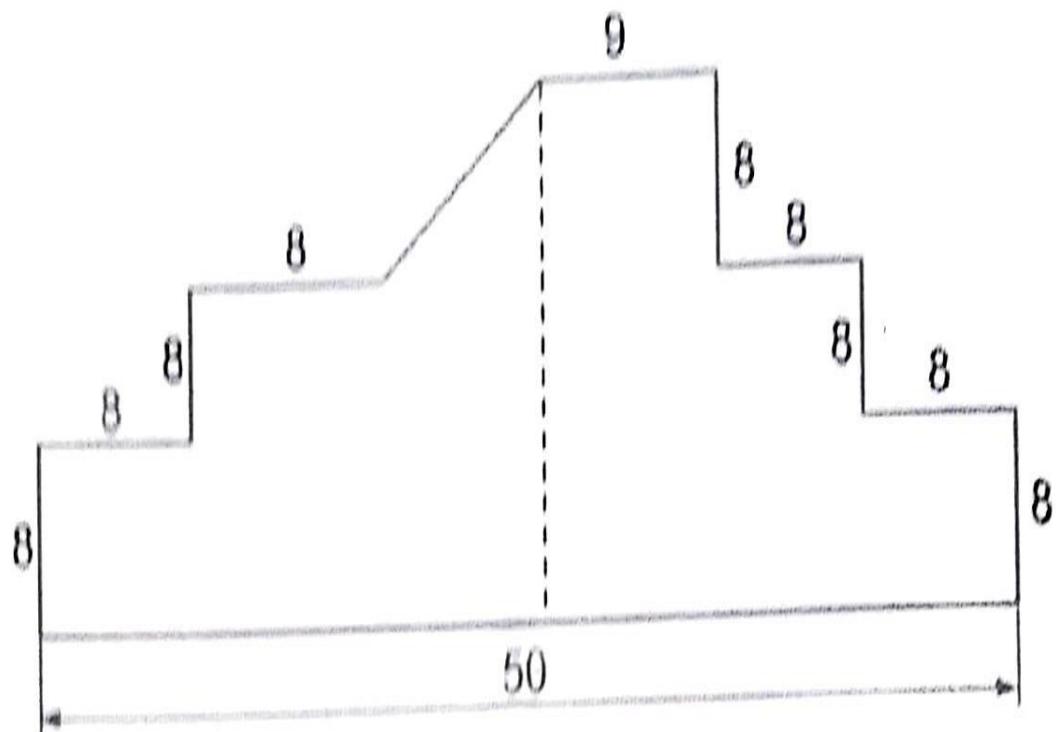


Fig. 9.5.

Solution : (Using rectangular coordinate system) (Refer Fig. 9.6).

Steps : Command : Line ↴

Specify first point : (Select any suitable point using mouse) ↴

Specify next point or [Undo] : @ 50, 0 ↴

Specify next point or [Undo] : @ 0, 8 ↴

Specify next point or [close/undo] : @ -8, 0 ↴

Specify next point or [Undo] : @ 0, 8 ↴

Specify next point or [close/undo] : @ -8, 0 ↴

Specify next point or [Undo] : @ 0, 8 ↴

Specify next point or [close/undo] : @ -9, 0 ↴

Specify next point or [Undo] : @ -9, -8 ↴

Specify next point or [close/undo] : @ -8, 0 ↴

Specify next point or [Undo] : @ 0, -8 ↴

Specify next point or [close/undo] : @ -, 8, 0 ↴

Specify next point or [Undo] : @ C ↴

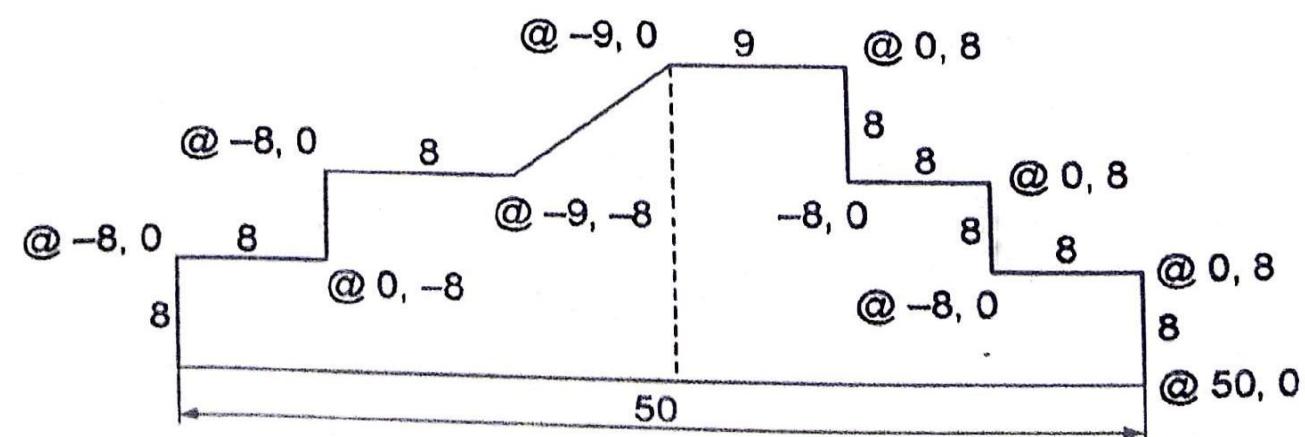


Fig. 9.6

■ 9.12 SETTING OF DRAWING SPACE

To start a new drawing on computer screen, it is important to set-up size and units as per following steps :

1. **Unit Command :** Click on Auto CAD main menu → Format → Units. After that, a dialog box will be displayed on screen where the following parameters are to be entered.

<p>(i) Length</p>	<p>(ii) Angle</p>
<p>(iii) Insertion scale</p>	<p>(iv) Direction</p>
<p>(v) Sample output</p>	
2. **Limit Command :** An invisible artificial boundaries on screen are to be created by using this command before starting a drawing. The default value of the lower left corner and the upper right corners are (0,0) and (17,11) respectively. To set the drawing limits, type limit ↴ on the command line.
3. **Scale Command :** Scale is usually chosen at the time of printing of drawing to arrange drawing within printing area. The required scale can be set through the plot scale section in the page set-up dialog box.

RECTANGLE Command

The RECTANGLE command is used to draw a rectangle as a single entity by selecting two diametrically opposite corners.

Command

: RECTANGLE

Specify first corner point or

[chamfer/Elevation/Fillet/Thickness/width] : *Select first corner using mouse.*

Specify other corner point

: *Select second corner using mouse.*

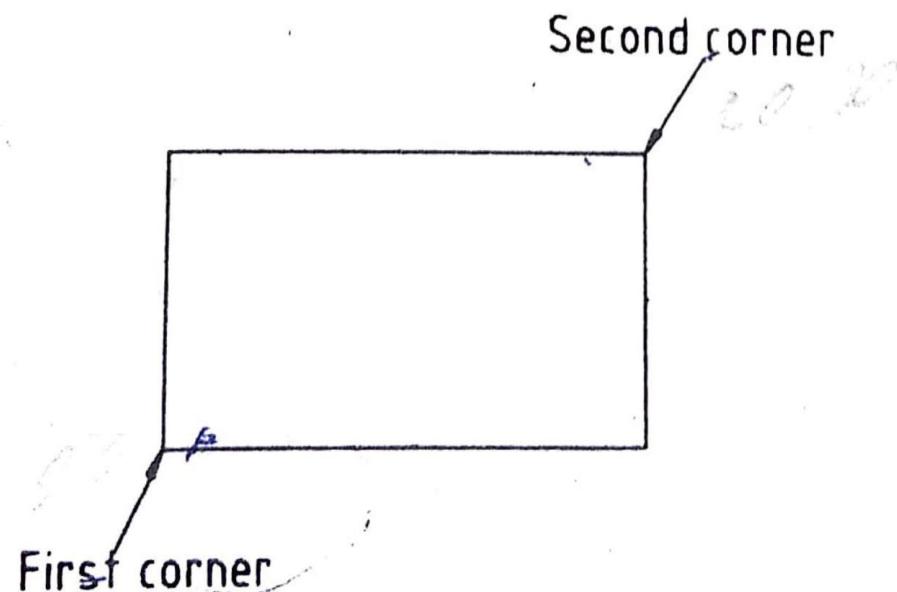


Figure 15.12 Rectangle drawn with opposite corners

- CIRCLE COMMAND : The circle command is used to draw circles of any given centre and diameter. There are five methods to draw a circle using Auto CAD.
 - (i) Centre-radius/diameter method.
 - (ii) Three points method (3P on circumference)
 - (iii) Two points method (2P specifying the end points of diameter of a circle)
 - (iv) Tangent-tangent-radius (TTR) method.
 - (v) Tangent-tangent-tangent (TTT) method.

(i) **Centre–Radius/Diameter Method**: The default setting to draw a circle is centre-radius method in Auto-CAD. Either on typing command CIRCLE or selecting it from a menu bar with the help of mouse, all circle drawing options are displayed.

The series of steps to draw a circle using centre-radius/ Diameter option, can be understood by following examples.

Example 9.9. Draw a circle with centre (80, 80) and diameter 50 mm using Auto CAD.

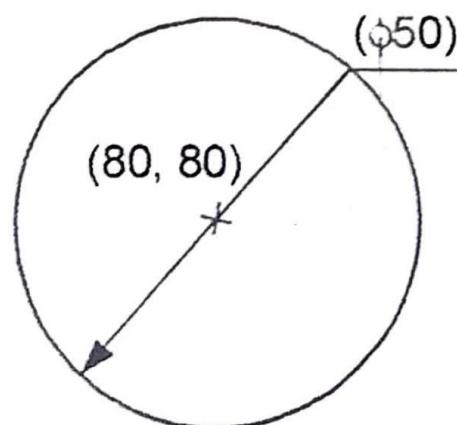
Solution : Steps to Draw a Circle : [Refer Fig. 9.10]

Command : Circle ↴ (Enter)

Specify centre point for circle or [3P/2P/Ttr (tan, tan, radius)] : 80, 80 ↴

Specify radius of circle or [Diameter] : D ↴

Specify Diameter of circle : 50 ↴



Three points option/method (3P method)

This specifies 3 points on the circumference of a circle. There is a unique circle passing through three given non-collinear points. The command of steps can be better understood with the help of suitable example given below :

Example 9.10. Draw a circle passing through three points whose coordinates are (100, 100) (200, 205) and (300,100) using auto CAD.

Solution : (3D method will be most suitable in this case)

Command : Circle ↴

Specify centre point for circle or [3P/2P/Ttr (tan, tan, radius)] : 3P ↴

Specify first point on circle : 100, 100 ↴

Specify second point on circle : 200, 205 ↴

Specify third point on circle : 300, 100 ↴

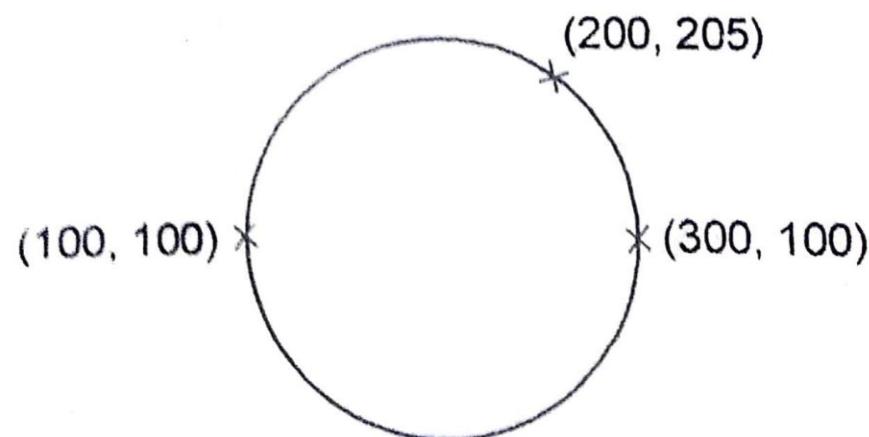


Fig. 9.11. Circle (Using 3P method)

(iii) **Two Points Option/method (2PU Method)** : This specifies the end points of diameter of a circle. The series of command can be better understood by the example given below :

Example 9.11. Draw a circle passing through two points (ends of a diameter) whose coordinates are (190, 150), (190, 110) using Auto CAD.

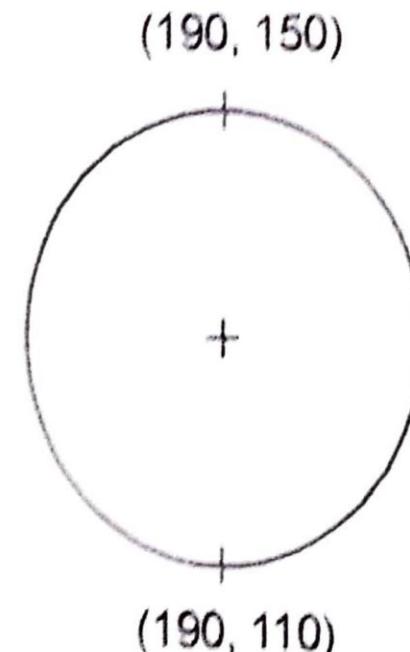
Solution : Command : Circle ↵

Specify centre point for circle or [3P/2P/Ttr (tan tan rad)] : 2P ↵

Specify first end point of circle's diameter : 190, 150 ↵

Specify second end point of circle's diameter : 190, 110 ↵

The circle will be drawn on screen as shown in Fig. 9.12.



(iv) **Tangent–Tangent–Radius (TTR) method** : This command draws a circle of specified radius that is tangent to two lines, circles or arcs.

The method can be better understood with the help of example given below :

Example 9.12. Draw a circle of 60 mm diameter touching two lines as tangents.

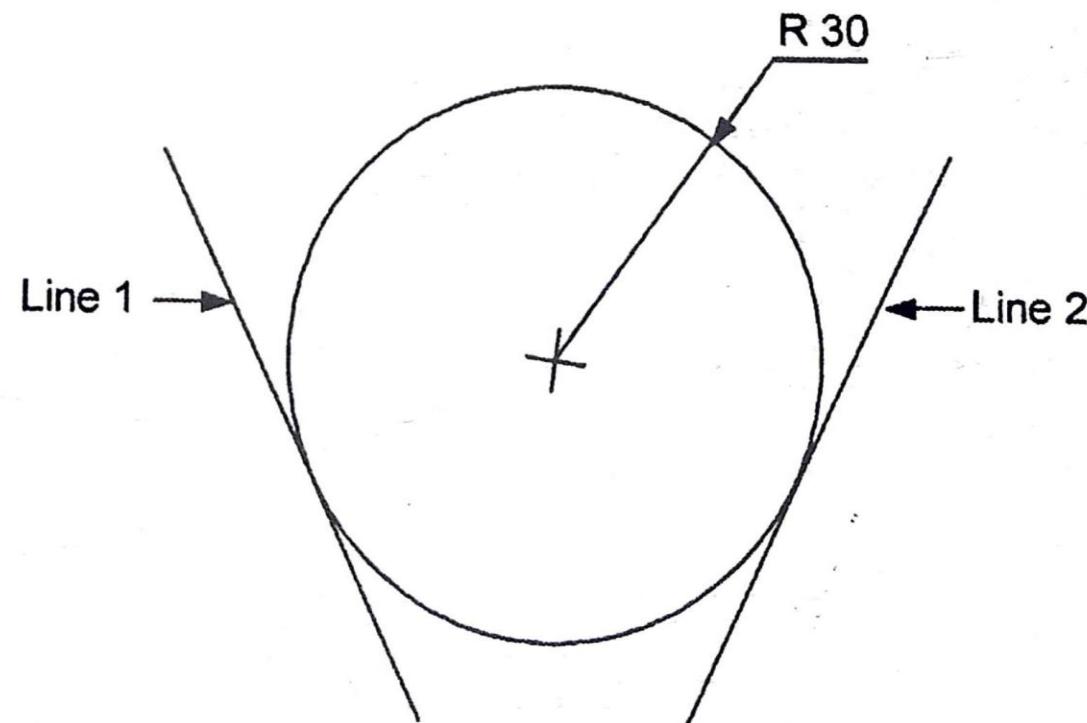
Solution : Circle ↵

Specify centre point for circle or [3P/2P/Ttr (tan tan radius) : Ttr ↵

Specify point on object for first tangent of circle : (Pick a point on line 1) ↵

Specify point on object for second tangent of circle : (Pick a point on line 2) ↵

Specify radius of circle : 30 ↵



Arc Command : This command is used to draw an arc. An arc is also considered as partial circle or a curve specified by centre and radius. This command can be invoked by any one of the following methods :

- (a) Draw tool bar
- (b) Command line : *ARC* or *A*
- (c) Menu bar : DRAW \Rightarrow ARC
- (d) Tool palettes

The ARC command offers following options to draw an arc :

1. Three points option/method
2. Start point–centre point–end point option.
3. Start point–centre point–length of chord option.
4. Start point–end point–angle option
5. Start point–end point–radius option
6. Start point–centre point–angle option
7. Start point–end point–direction

1. **Three Point Method** : In this method, three points, P_1 , P_2 , P_3 are given through which arc is passing. Steps of commands are explained with the help of example given below.

Example 9.13. Draw an arc passing through points A, B, C using Auto CAD.

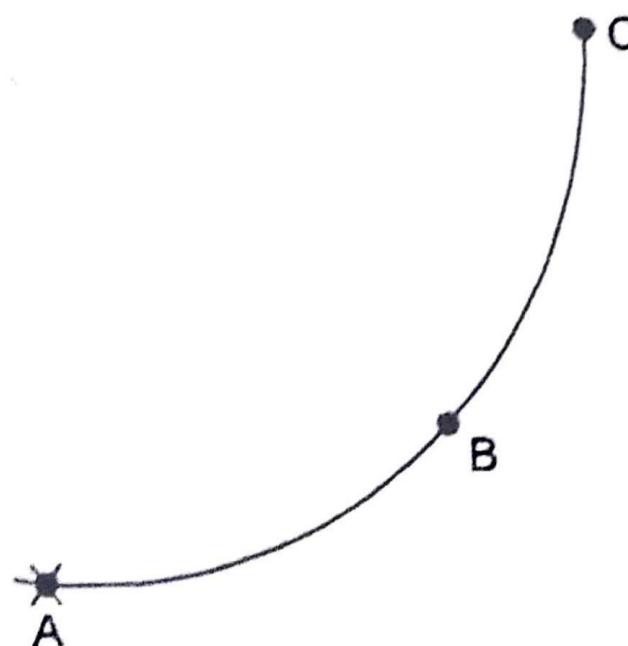
Solution : Command : Arc ↵

Specify start point of arc or [centre] : (Pick A) ↵

Specify second point of arc or [centre/End] : (Pick B) ↵

Specify end point of arc : (Pick C) ↵

(Refer Fig. 9.14)



2. Start Point-Center Point-end Point Method : Steps of command used in this method is given in example below :

Example 9.14. Draw an arc using start-point. Centre point-end point option of ARC command.

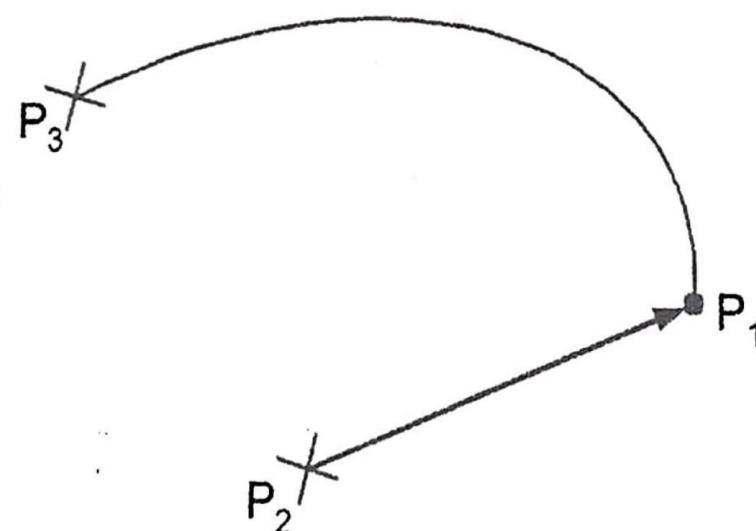
Solution : Command : Arc ↴

Specify start point of arc or [Centre] : (Pick start point) ↴

Specify second point of arc or [Centre/End] : C ↴

Specify centre point of arc : (Pick centre) ↴

Specify end point of arc or [Angle/Chord length] : (Pick end point) ↴



3. Start Point-Centre Point-Length of Chord Method : Series of command used in this method is given in example below :

Example 9.15. Draw an arc using start point-centre point-length of chord option of ARC method having arc length 50 mm.

Solution : Command : Arc ↴

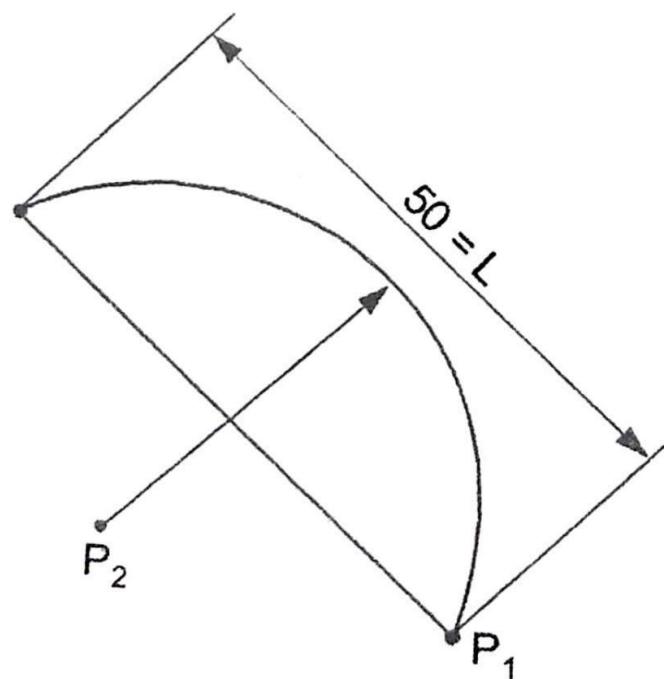
Specify start point of arc or [centre] : (Pick start point) ↴

Specify second point of arc or [Centre/End] : C ↴

Specify centre point of arc : (Pick centre point) ↴

Specify end point of arc or [Angle/Chord Length] : L ↴

Specify Length of chord : 50 ↴



4. Start Point-End Point-Angle Option/Method : The series of commands used in this method of arc is given in example below :

Example 9.16. Draw an arc using start point-end point-angle option of ARC command having included angle 105° .

Solution : Command : Arc ↴ (Enter)

Specify start point of arc or [Centre] : (Pick start point) ↴

Specify second point of arc or [Centre/End] : E ↴

Specify end point of arc : (Pick end point) ↴

Specify centre point of arc or (Angle/Direction/Radius) : A ↴

Specify included angle : 105° ↴ [Refer Fig. 9.17]

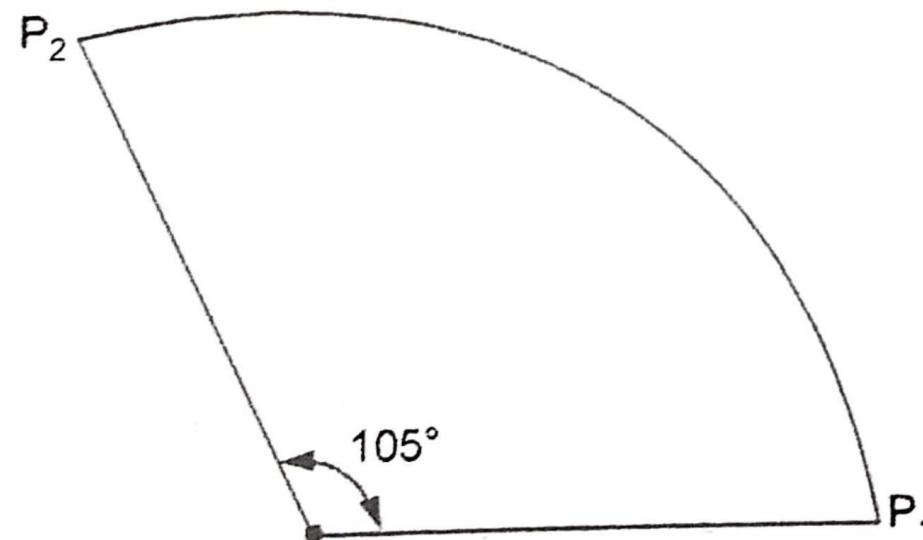


Fig. 9.17. ARC $P_1 P_2$

5. **Start Point-End Point-Direction Method** : The series of command used in this method is given in example below :

Example 9.17. Draw an arc using start point-end point-direction option of ARC command having tangent angle 75°.

Solution : Command : Arc ↴

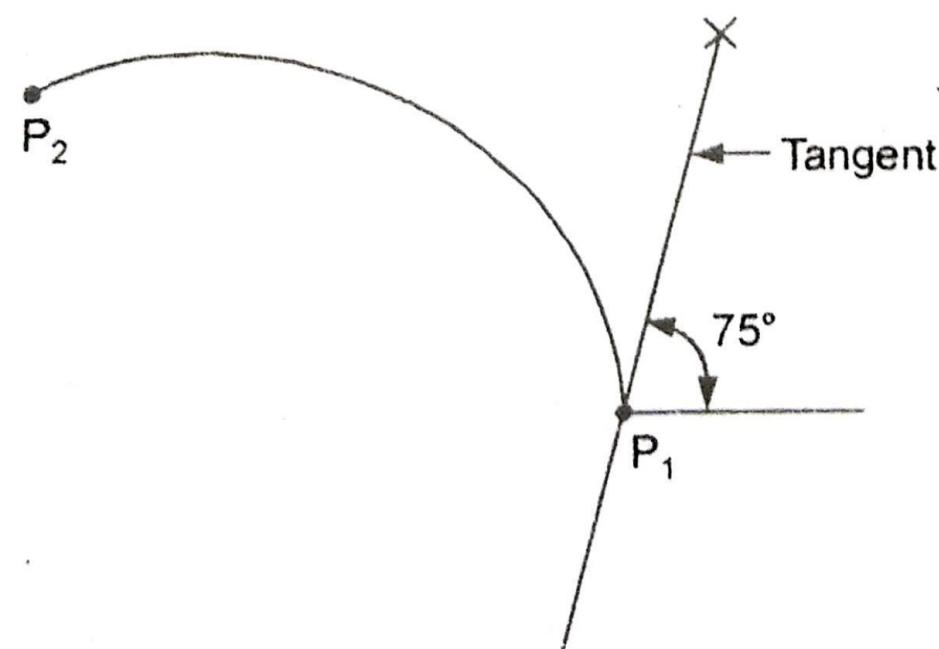
Specify start point of arc or [Centre] : (Specify first point) ↴

Specify second point of arc or (centre/End) : E ↴

Specify end point of arc : (Specify end point) ↴

Specify centre point of arc or [Angle/Direction/Radius] : D ↴

Specify tangent direction for the start point of arc : (Specify direction of tangent) ↴ [Refer Fig. 9.18]



6. Start Point-end Point-Radius Option/Method : The series of commands used in this method is given in example below :

Example 9.18. Draw an arc using start point-end point-radius option for Arc command having radius of arc 30 mm.

Solution : Command : Arc ↴

Specify start point of arc or [centre] : (Specify first point) ↴

Specify second point of arc or [Centre/End] : E ↴

Specify end point of arc : (Specify end point) ↴

Specify centre point of arc or [Angle/Direction/Radius] : R ↴

Specify Radius of arc : 30 ↴

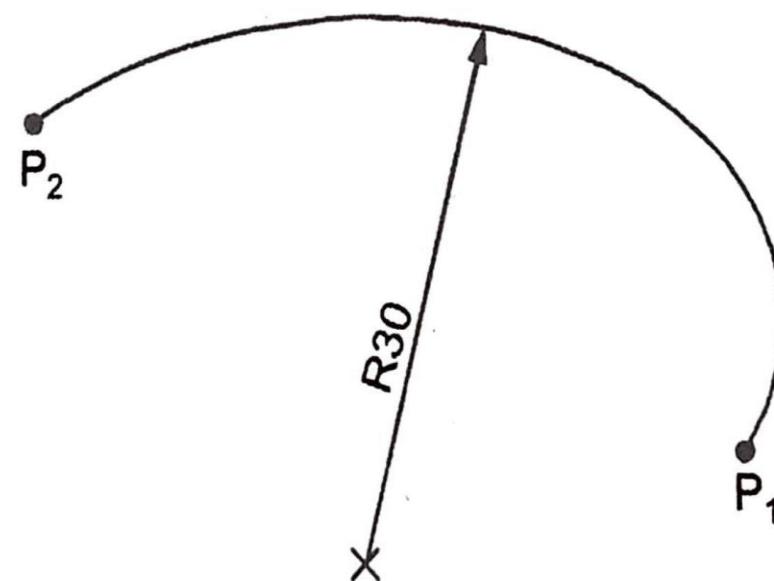


Fig. 9.19. Arc 'P₁ P₂' (centre)

7. **Ellipse Command** : This command is used to draw an ellipse using following methods/options.

- (i) Axis end points method/option.
- (ii) Centre-axis end point method/option.
- (iii) Arc option/method.

(i) **Axis End Points Option** : In this, end points of one of its axis (either major axis or minor axis) and the radial length of the other axis (or one end point of other axis) are given.

The series of commands used in this method can be better understood with the help of example given below :

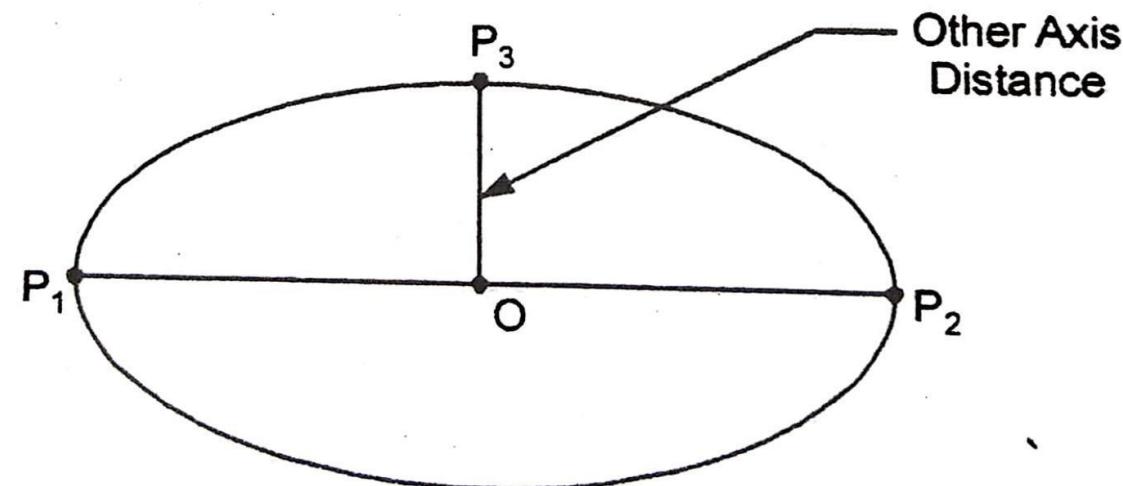
Example 9.19. Draw an ellipse using major axis end points P_1 and P_2 and minor axis end point P_3 using Auto CAD ELLIPSE command.

Solution : Command : Ellipse ↴

Specify axis end point of ellipse or [Arc/Centre] : (Specify point P_1) ↴

Specify other end point of axis : (Specify point P_2) ↴

Specify distance to other axis or [Rotation] : (Specify point P_3) ↴



(ii) **Centre-Axis end Point Method/Option** : The series of commands used in this method is given in example below :

Example 9.20. Draw an ellipse using centre point major axis end point and minor axis end point method.

Solution : Command : Ellipse ↴

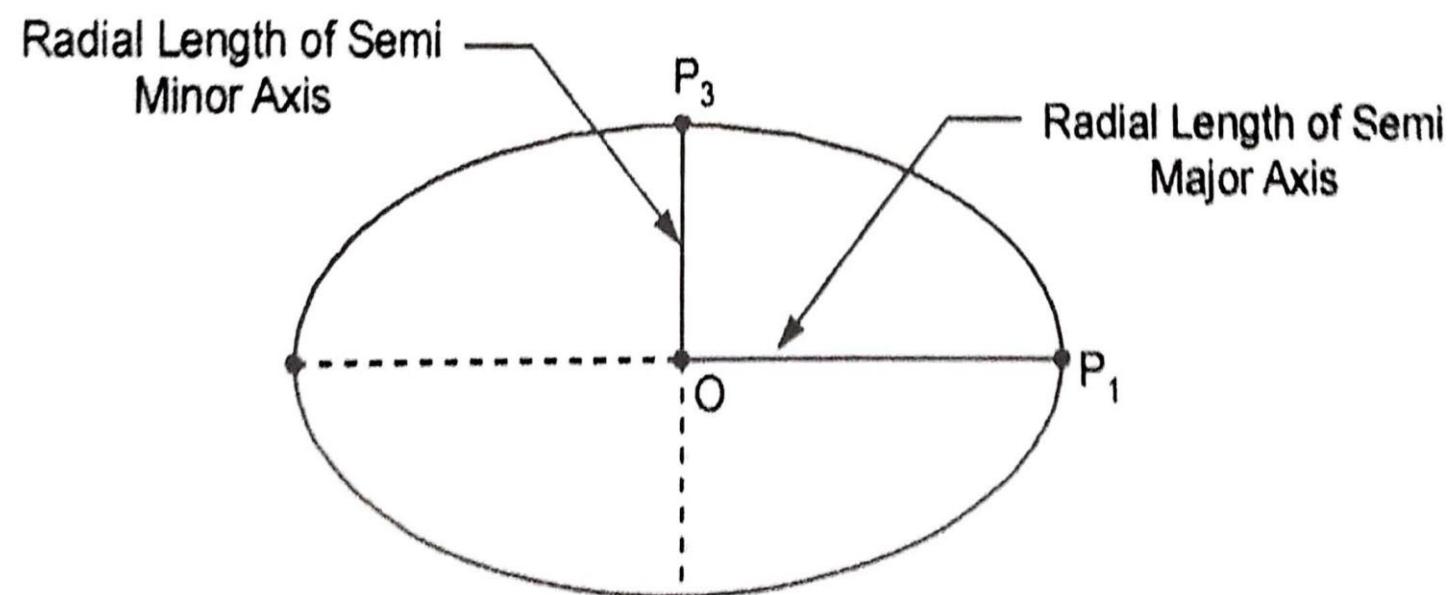
Specify axis end point of ellipse or [Arc/Centre] : C ↴

Specify centre of ellipse : (Specify centre point) ↴

Specify end point of axis : (Specify point P_2) ↴

Specify distance to other axis or [Rotation] : Specify point P_3) ↴

[Refer Fig. 9.21]



7. **Polygon Command** : POLYGON command is used to draw a regular polygon of sides like triangle, square, pentagon, hexagon etc. (range of side 3 to 1024). There are three options/methods to draw a polygon using this command :

- (i) Inscribed in circle method/option.
- (ii) Circumscribed about the circle method/option.
- (iii) Edge option/method

(i) **Inscribed in circle option** : If we use this option to draw polygon, then a polygon will be drawn such that its vertices lie on the circumference of an imaginary circle.

Series of commands of this method is given below in example :

Example 9.21. Draw a regular pentagon inscribed in a circle of 40 mm radius using auto CAD.

Solution : Command : Polygon ↵

Enter number of sides <4> : 5

Specify centre of polygon or [Edge] : (Specify centre) ↵

Enter an option [Inscribed in circle/Circumscribed about circle] <I> : I ↵

Specify radius of circle : 40 ↵

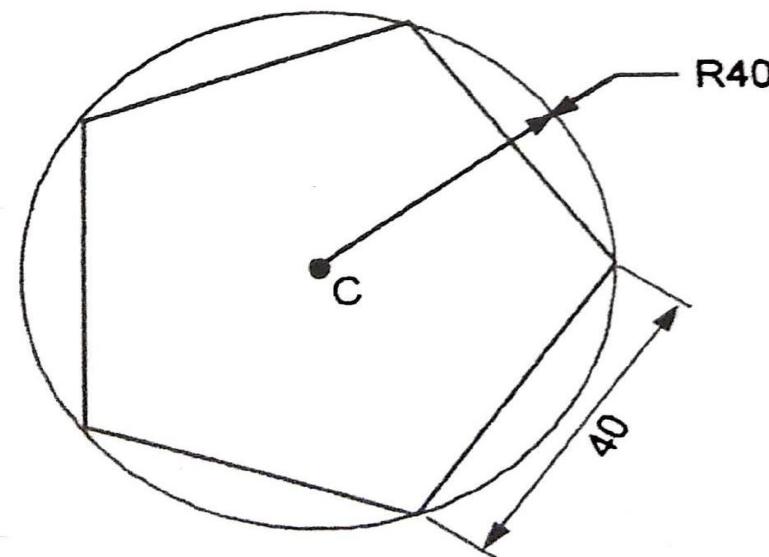


Fig. 9.22. Pentagon Inscribed in a Circle

(ii) **Circumscribed about the circle option** : Using this option, a polygon can be created such that mid-points of all the sides touch the circumference of an imaginary circle.

The series of commands used in this option is given in example below :

Example 9.22. Draw a regular hexagon circumscribed about a circle of 30 mm radius.

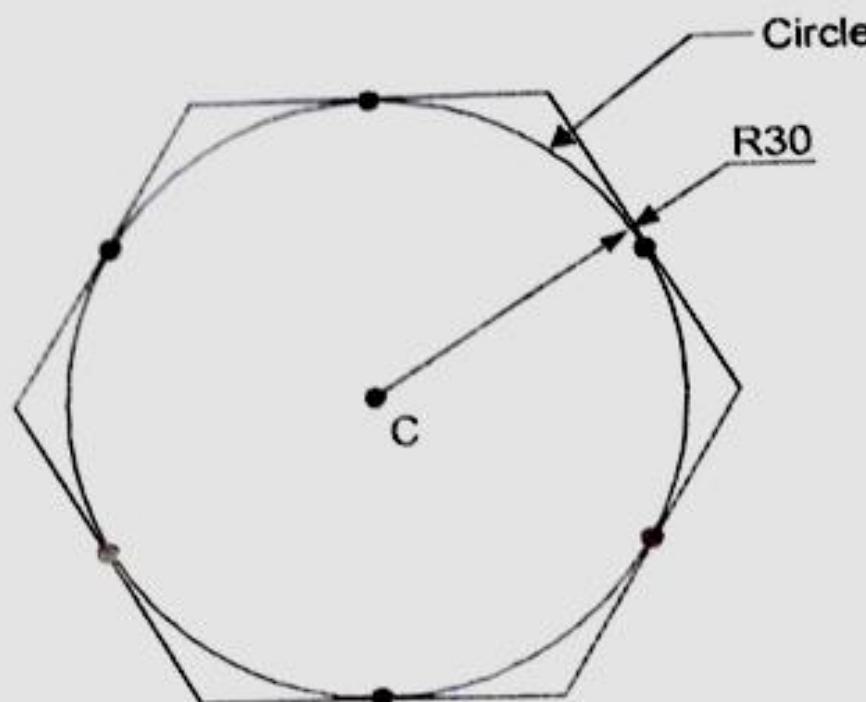
Solution : Command : Polygon ↴

Enter number of sides <5> : 6

Specify centre of polygon or [Edge] : (Specify a point C) ↴

Enter an option [Inscribed in circle/circumscribed about circle] <I> : C ↴

Specify radius of circle : 30 ↴



Viva Questions

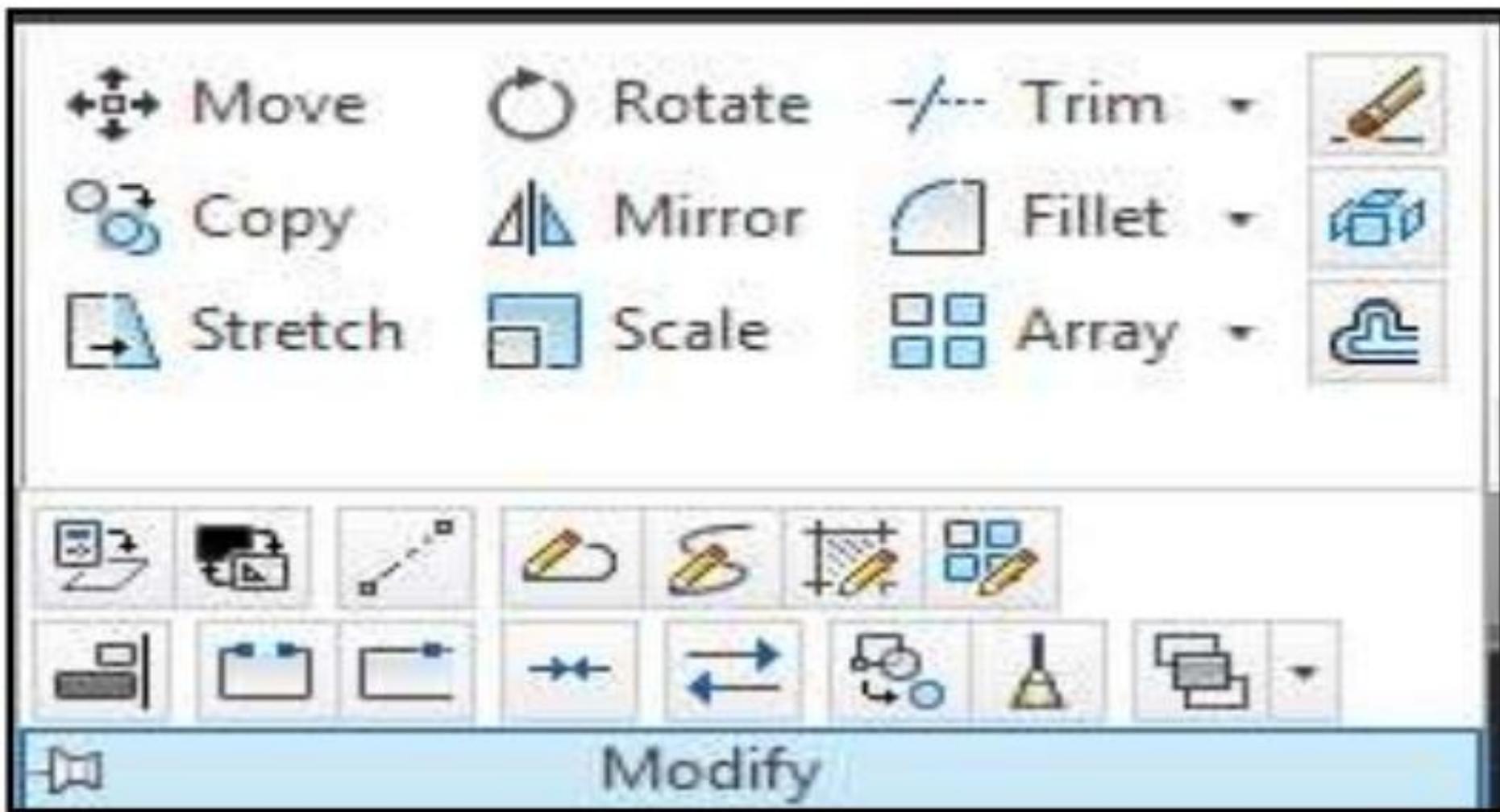
- Define CAD
- Name five software of CAD
- Write five advantages of CAD over manual drawing.
- Limitations of CAD over manual drawing.
- Name five draw commands
- Name different coordinate system used in autoCAD.
- Explain absolute coordinate system.
- Explain polar coordinate system.

Editing Commands in AutoCAD



Modifying COMMANDS

BIMSOLUTIONS



EDITING COMMANDS IN AUTOCAD

Commands which are used to modify the existing drawings to the required size and shape are referred to as editing commands. Following are the list of editing commands commonly used in engineering drawing preparation.

OFFSET Command

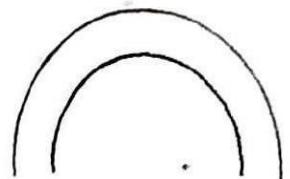
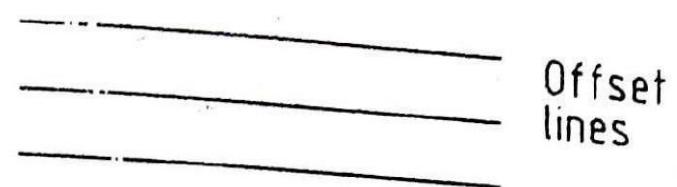
The OFFSET Command is used to draw parallel lines, arcs, concentric circles etc to the specified distance.

Command

Specify offset distance or [Through] <Through>
Select object to offset or <exit>
Specify point on side to offset
Select object to offset or <exit>

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- : **OFFSET**
- : *Enter the distance using keyboard*
- : *Select object to offset using mouse.*
- : *Specify the side for offsetting using mouse.*
- : *Press ENTER to Complete the command.*



Offset curves

UNDO Command or U Command

The UNDO command is used to undo the operations which are previously executed.

Command : UNDO or U

REDO Command

The REDO command is used to REDO the operation immediately following the UNDO command, if necessary.

Command : REDO

ZOOM Command or Z command

The ZOOM command is used to enlarge or reduce the size of the object on the screen. Note that the actual size of the object, remains unchanged.

Command : ZOOM or Z

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time> :

Specify the window to zoom by picking the lower left corner and upper right corner using mouse.

MOVE Command

The MOVE command is used to move the object from the present position to a new location.

Command

Select objects

Select objects

Select base point or displacement

Select any point as a base point to move the object.

Specify second point of displacement: Select the new location for the base point.

Consider the following illustration to move the object. The object in the original position will be removed.

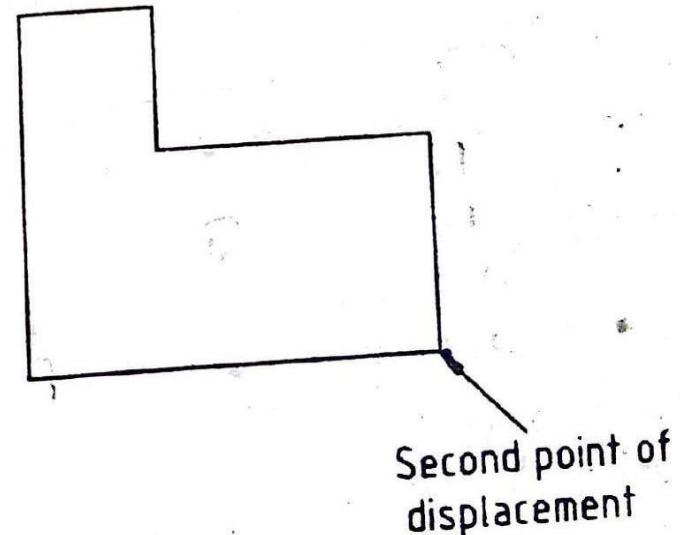
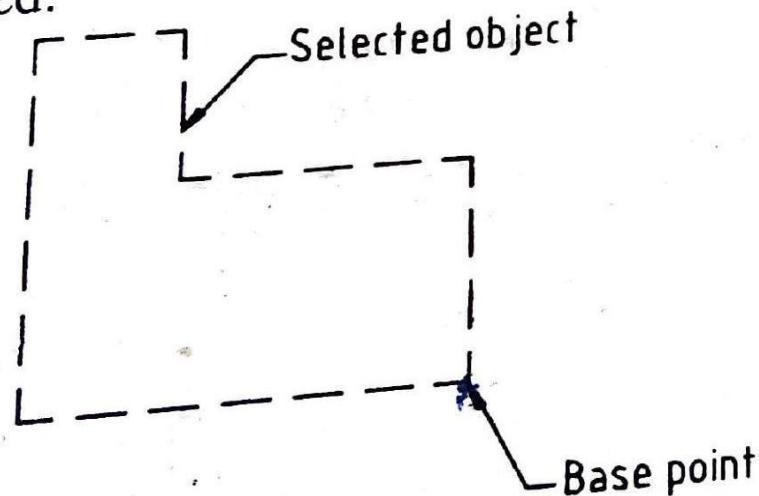


Figure 15.8 Moving an object using MOVE command

COPY Command

The COPY command is used to copy the existing objects, to a new location. Multiple copies can also be made by selecting multiple option M in this command.

Command : **COPY**

Select objects : *Selects objects to copy individualy or in a window using the mouse.*

Select objects : *Press ENTER to complete the selection.*

Select base point or displacement, or [Multiple] : *Select any point as base point.*

Select the second point or displacement : *Select the new location for base point.*

Consider the following illustration to copy the object. The object in the original position is retained.

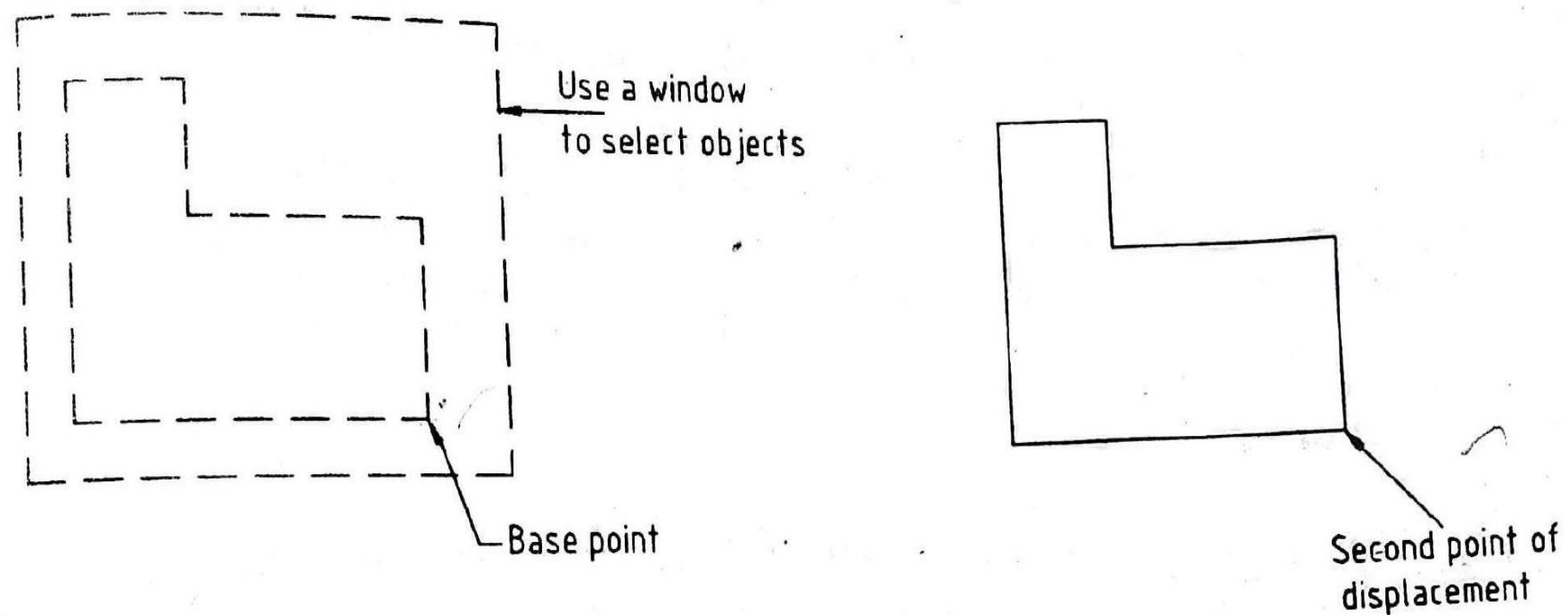


Figure 15.9 Copying an object using COPY command

MIRROR Command

The MIRROR Command is used to get a mirror copy of a symmetrical object.

Command

Select objects

Select objects

Specify first point of mirror line

Specify second point of mirror line

Delete source objects? [Yes/No] <N>

MIRROR

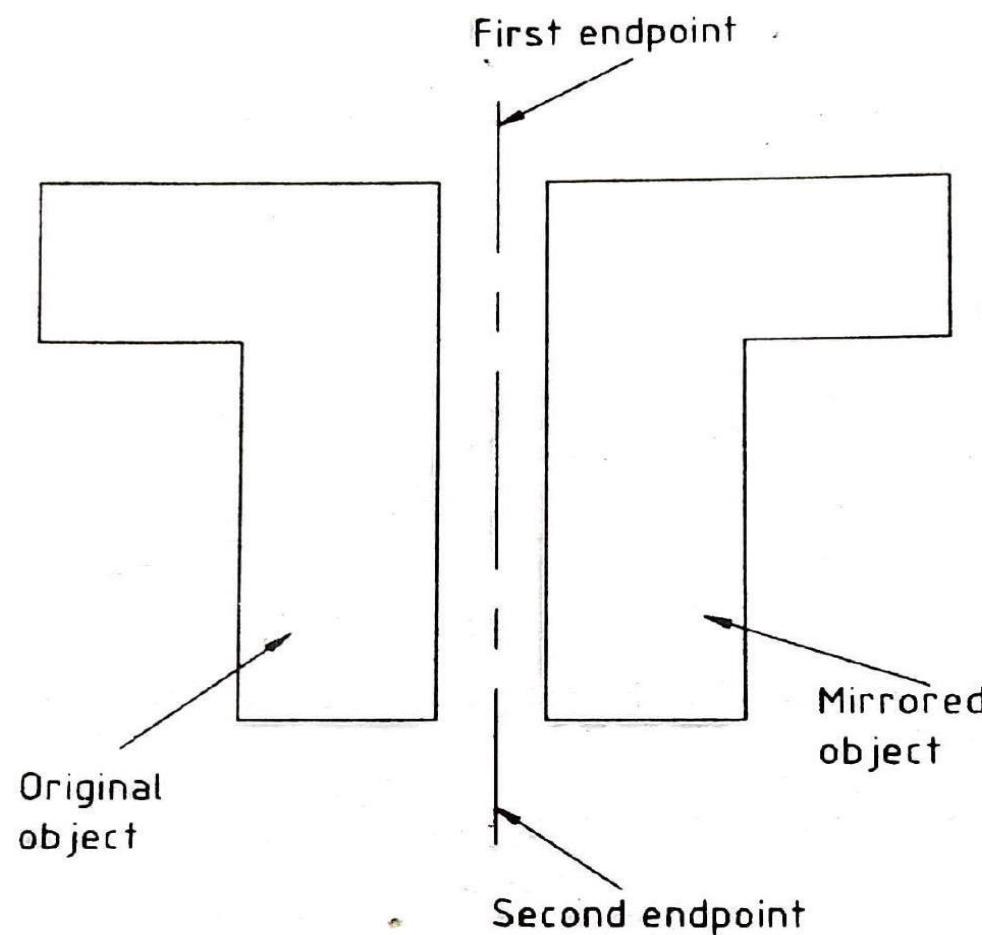
Select the objects to be mirrored using mouse.

Press ENTER.

Specify first end point

Specify second end point.

Press ENTER.



ARRAY Command

The Array Command is used to make multiple copies of an object in a rectangular or polar fashion.

Command

Select objects

Select objects

Enter type of array [Rectangular/Polar]<R>

Enter the number of rows (---) <1>

Enter the number of columns (|||) <1>

Enter the distance between rows or specify unit cell (---) : 30

Enter distance between columns (|||) : 40

: ARRAY

: Select the circle using the mouse

: Press ENTER

: Press ENTER

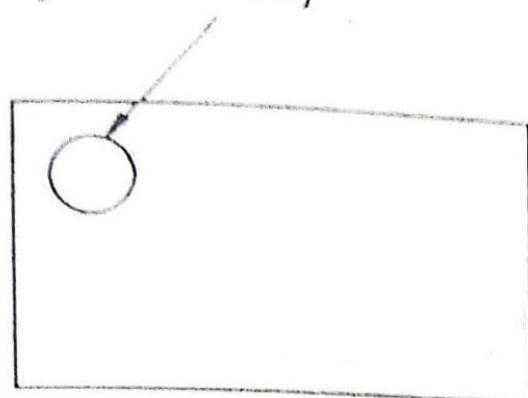
: 2

: 3

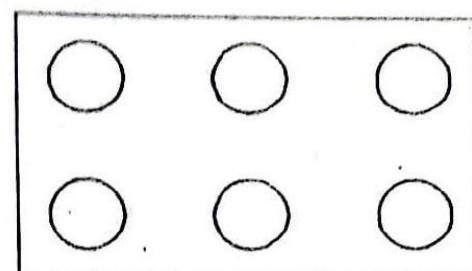
: 30

: 40

Object to array



Original Object



After ARRAY

The polar array arranges the objects around a point in a circular pattern. Consider the following example

Command

Select objects

Enter type of array [Rectangular or polar array] <R> :

Specify center point of array

the

Enter the number of terms in the array

Specify the angle to fill(+ = CCW, - = CW)<360

Rotate arrayed objects [Yes/No] <Y>

: **ARRAY**

: Press Enter

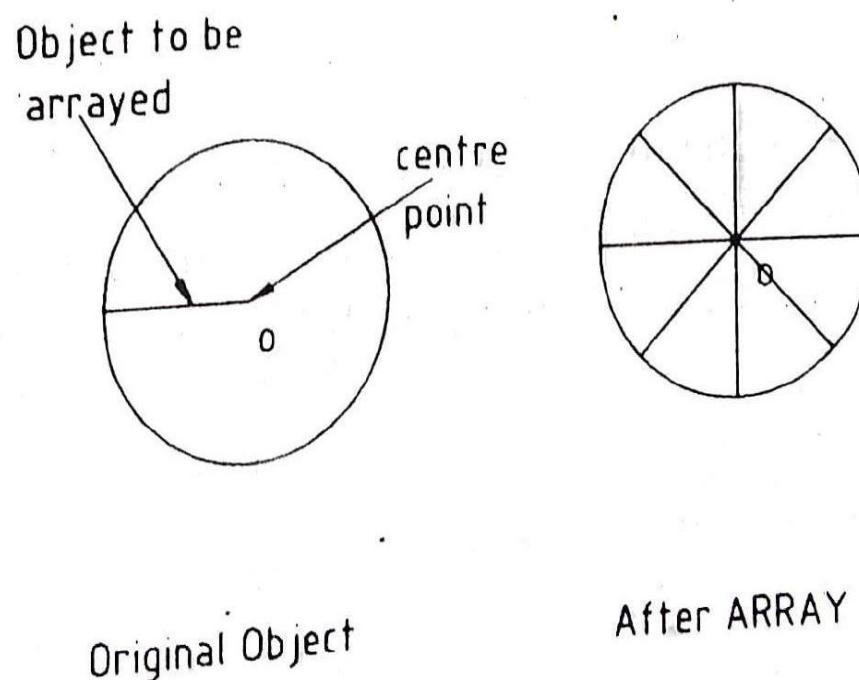
: P

: *Select the end point of the line at center of the circle.*

: 8

: *Press ENTER*

: *Press ENTER*



FILLET Command

The FILLET Command is used to draw chamfering arcs connecting two lines of specified radius.

Command

Current settings

Select first object or [polyline/Radius/Trim]

Select second object

: FILLET

: Mode = TRIM, Radius = 3.0000

: *Specify first object using mouse*

: *Specify second object using mouse*

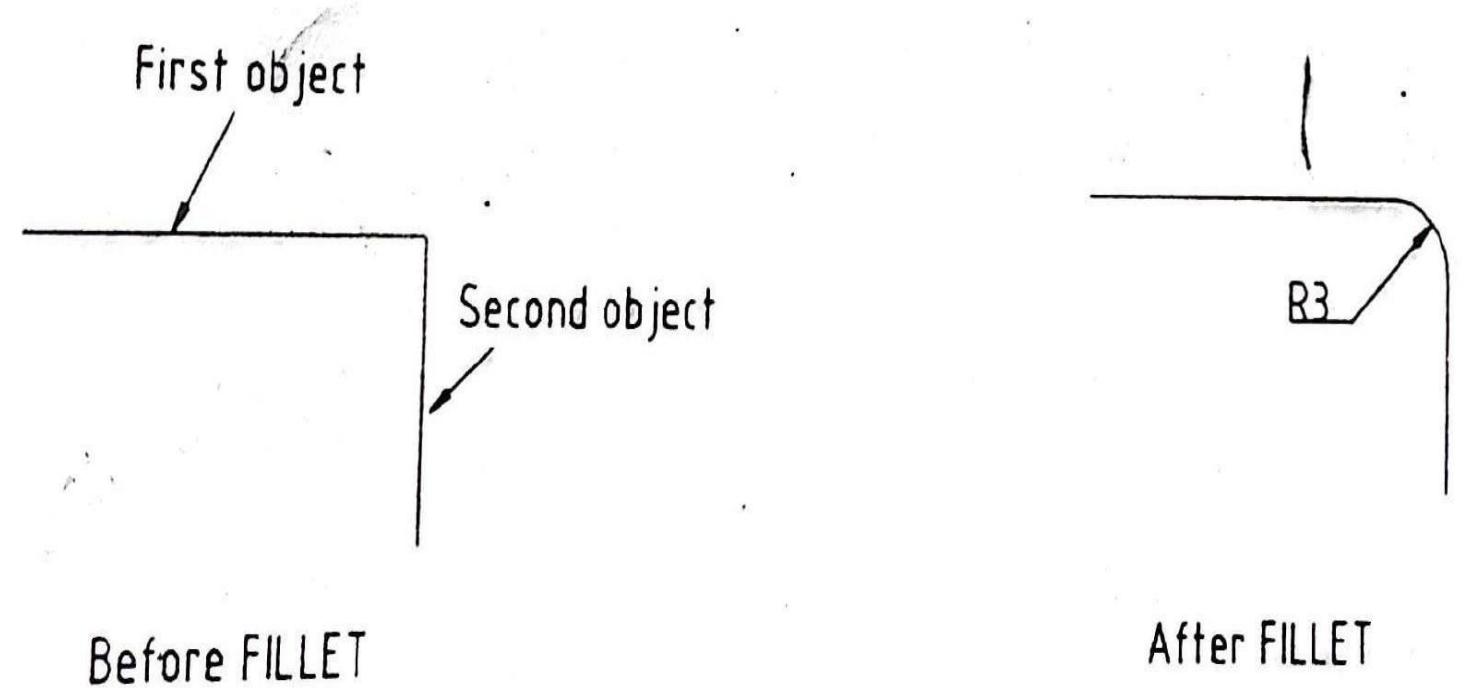


Figure 15.18

CHAMFER Command

The CHAMFER Command is used to draw beveled lines connecting two lines at specified distances from the corner of two lines.

Command

(TRIM mode) current chamfer Dist 1 = 3.000, Dist 2 = 3.000

[Polyline/Distances/Angle/Trim/Method]

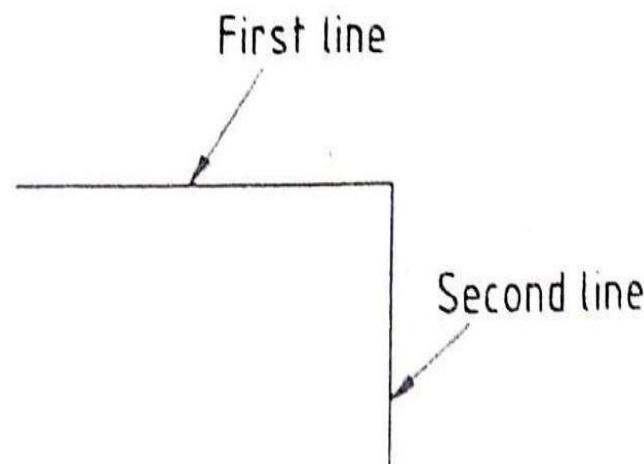
Select first line or

Select second line

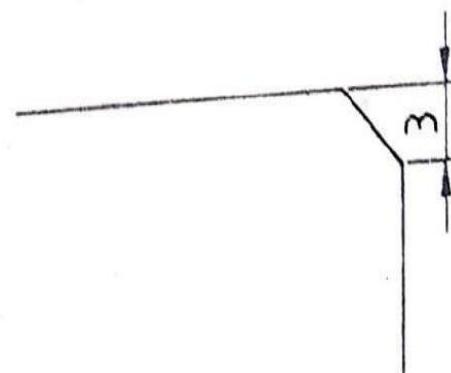
: **CHAMFER**

: Specify the first line using mouse.

: Specify the second line using mouse.



Before CHAMFER



After CHAMFER

TRIM Command

The TRIM Command is used to trim or cut the lines projecting beyond the specified boundary or cutting lines.

Command

: TRIM

Current settings : Projection = UCS Edge = None

:

Select cutting Edges....

: *Specify the cutting edges using mouse*

Select objects

: *Press ENTER*

Select objects

: *Select edges*

Select object to trim or [Project/Edge/Undo]

: *Press ENTER*

Select object to trim or [Project/Edge/Undo]

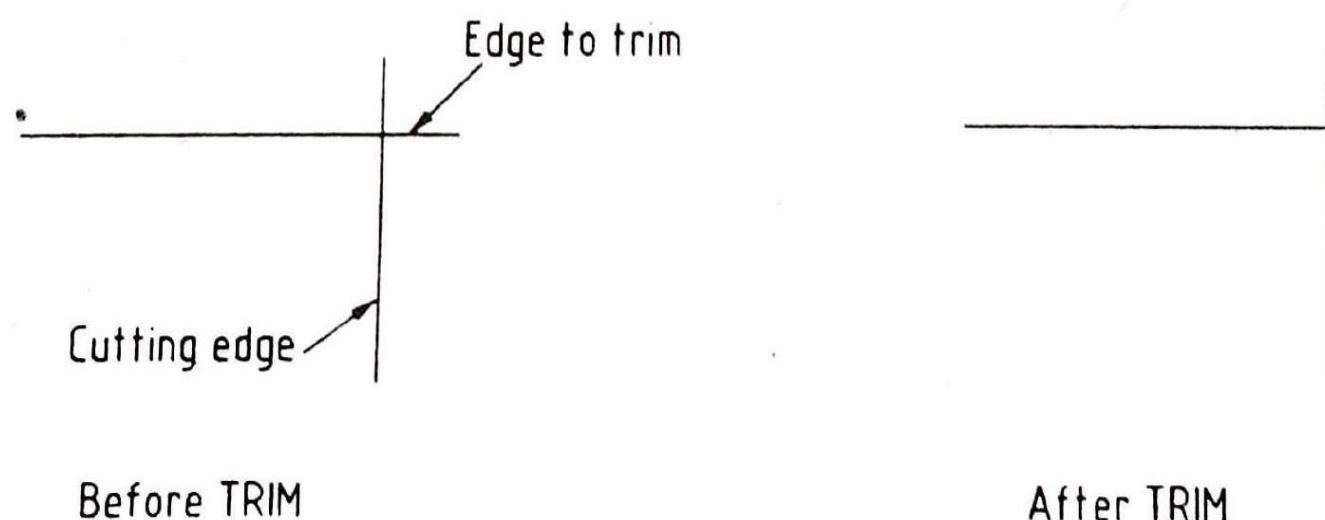


Figure 15.20

EXTEND Command

The EXTEND Command is used to extend or lengthen a line to meet other object which is selected as the boundary edge.

Command

Current settings : Projection = UCS Edge = None

Select boundary edges...

Select objects

Select objects

Select object to extend or [Project/Edge/Undo]

Select object to extend or [Project/Edge/Undo]

: EXTEND

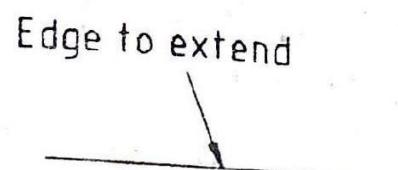
: *Select the boundary edge using mouse.*

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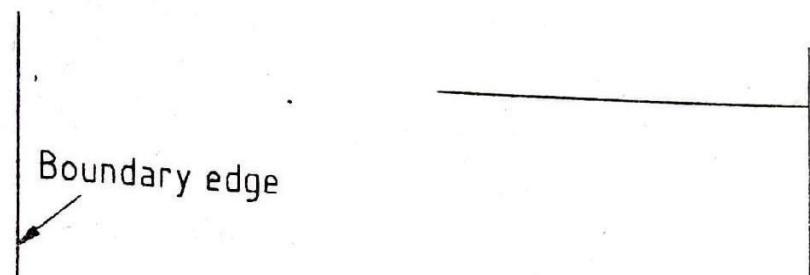
: Press ENTER

: *Select the lines to be extended using mouse.*

: Press ENTER



Before EXTEND



After EXTEND

Figure 15.21

BREAK Command

The BREAK Command is used to remove a part of the selected objects like line, arc, circle etc.

Command

: **BREAK**

Select object

: *Select the object using mouse*

Enter second break point or [First point] : *Specify the second point using mouse*

NOTE: Option F may be given to select the first point, then enter the second break point.

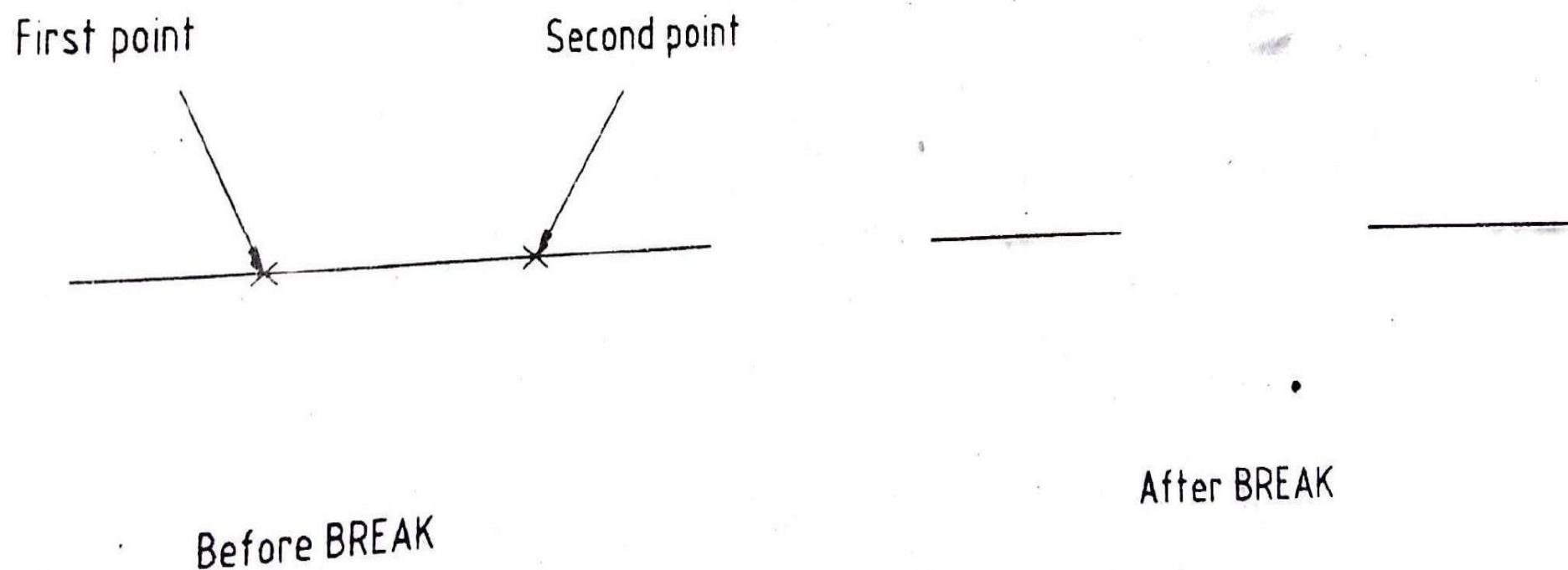


Figure 15.22

ROTATE Command

The ROTATE Command is used to rotate an object to a specified angle

Command

Current positive angle in UCS

Select objects

Select objects

Specify base point

Specify rotation angle or [Reference]

: **ROTATE**

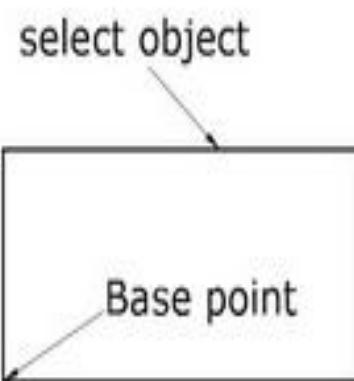
: ANGDIR = current ANGBASE = current

: Select the objects for rotation using mouse

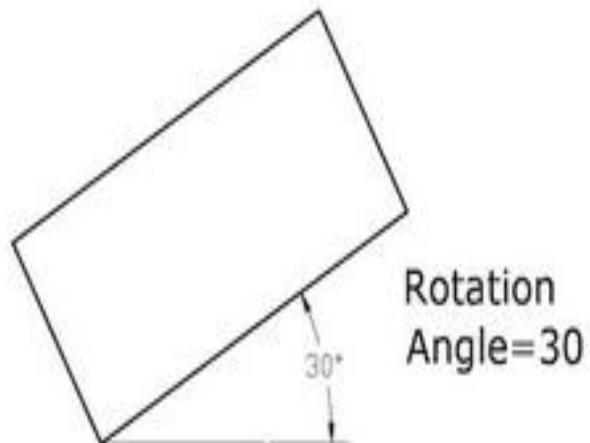
: Press ENTER

: Select a base point on or nearer to the object using mouse.

: Enter the angle of rotation using keyboard.



(Before Rotate)



(After Rotate)

ERASE Command

The ERASE command is used to erase objects which are drawn earlier. Select the objects to be erased one by one, or select them in a window by picking the lower left corner and upper right corner using the mouse.

■ 9.15 VARIOUS COMMANDS USED IN AUTO CAD

9.15.1 Utility Commands

[RGPV, June 2011]

These commands are used to control the basic functions of auto CAD. The following commands are mainly used for this purpose :

1. NEW : It is used to create a new drawing file.
2. OPEN : It is used to open an existing file.
3. CLOSE : It is used to close active drawing.
4. SAVE : It is used to save the drawing.
- EXIT : It is use to comes out of the auto CAD.