INTRODUCTION to COMPUTER AIDED DRAFTING

What is CAD ?

⇒ Computer Aided Drafting (CAD) is a process of preparing a drawing of an object on the computer screen by using softwares and hardwares of computer system. It can be defined as "The use of computer system to assist in the creation, modification and analysis or optimization of a design.

The generation of design drawings interactively by the user suppling information to the computer via commands or with a menu driven series of instructions is called computer Aided Drafting.

These drawing can easily be edited, manipulated and stored for future usage.

The use of CAD process provides enhanced graphics capabilities which allows any designer to :—

1. conceptualize his ideas.
2. Modify the design very easily.
3. Perform animations.
4. Make design calculations.
5. Use colours, fonts and other features.

What are the limitations of manual drawing?

⇒ Manual drawing of engineering parts has following limitations:-

1. Manual drawing of engineering parts and their assemblies is time consuming and requires special skills.

2. changes in manual drawing are not easy to made as whole drawing is needed to be redraw.

What are the advantages (Benefits) of CAD?

⇒ Engineering drawing is an interactive process and when it is performed on drawing boards and includes their documentation, it is quite a time consuming process. Today in the Era of rapid technologies, product innovations, greater response to customer requirements, CAD gives an innovative and efficient system for design.

CAD has the following advantages over the traditional method of design:-

(i) CAD is faster and give more accurate results than conventional methods.

(ii) Developing the design of a component and assosiated drafting is a very easy task with CAD.

(iii) Kinematic feature of CAD pakages enables the designer to visualize the operational performance of the component.

(IV) Two or more designs can be compared analytically

(V) Improved productivity in drafting.

(VI) Shorter preparation time for drawing

(VII) Reduce manpower requirements.

(VIII) Modifications according to customers in drawing are easier.

(IX) Improved accuracy of drawing.

(X) Revisions are possible.

(XI) Colour can be used to customize the product.

(XII) Hatching of all sections with different filling patterns.

(XIII) preparation for assembly or sub-assembly drawings.

(XIV) preparation of part list are easy.

(XV) printing can be done to any scale.

(XVI) Better quality designs.

(XVII) Improved productivity.

(XVIII) Easy modifications.

What are the limitations (Disadvantages) of CAD?

⇒ The following limitations are experienced in the use of CAD System:-

1. 32-bit word computer is nessesary because of large amount of computer memory and time.
2. The size of the software pakage is large.
3. Skill and judgment are required to prepare the drawing.
4. It requires large investment.
5. Every new release of CAD software, operator has to update their skills.

What is CAD Softwares?

⇒ CAD System creates an environment to prepare drawing interactively. The software is an interpreter or translator which allows the user to perform specific type of application or job related to CAD. The user may utilise the software for drawing.

Major function to be performed by a computer aided drafting system:-

1. Basic setup of drawing.
2. Drawing the objects
3. Changing the objects properties
4. Translating the objects
5. Scaling the objects.

6. clipping the objects to fit image to the screen
7. creating symbol libraries for frequently used objects.
8. Text insertion.
9. Dimensioning.

The various types of CAD softwares are :-

1. AutoCAD
2. CATIA
3. Pro-E
4. Unigraphics
5. Corel CAD
6. 3D Max
7. 3D-Home
8. Micro station
9. Solid works
10. Corel Draw.

What is AutoCAD?

⇒ AutoCAD is one of the leading drafting softwares available. It was developed by "Autodesk" company By AutoCAD, drawing can be created, modified and erased with ease.

The first release of AutoCAD Version 1.0 was in 1982 Many updates and improvements were done constantly. 14th version of AutoCAD was introduced in 1998 and thereafter AutoCAD-2000.

The Auto-CAD-2004 allows the designer to use of internet and web enabled design and documentation software.

CAD uses 4-Basic commands (elements) for preparation of any drawing :-

1. Line.
2. Curves.
3. Text.
4. Filling Text.

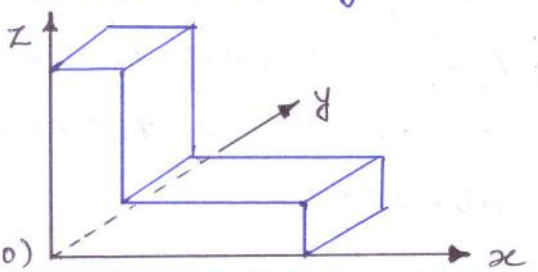
What is co-ordinate system?

⇒ A co-ordinate system uses one or more numbers of co-ordinates to uniquely define the position of a point or other geometric element in the working space.

The co-ordinate system can be modified in the AutoCAD. There are following types of co-ordinate system used in AutoCAD :-

1. Absolute co-ordinate system :-

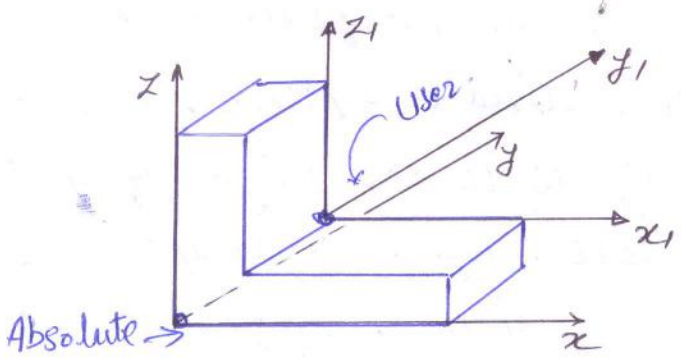
Absolute co-ordinate system are also called world co-ordinate system. In the absolute co ordinate system the points are located with respect to the origin (0,0) the syntax used is x, y,



2. User co-ordinate System :-

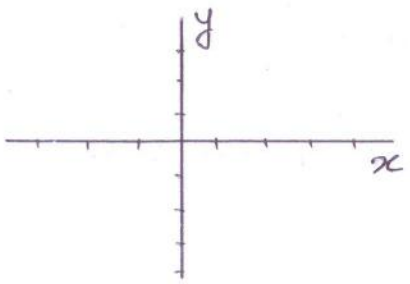
Sometime it becomes difficult to define certain geometries in absolute coordinate system. in such cases alternate co-ordinate systems can be defined relative to the absolute coordinate system. These co-ordinate systems are called user co-ordinate system or relative coordinate system or working coordinate system.

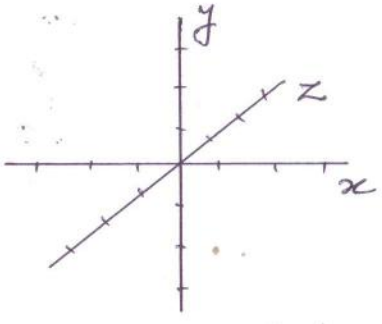
In the relative co-ordinate method, the displacement along x and y axis are measured with reference to the previous point. The syntax used is @ x, y.



3. Cartesian co-ordinate system :-

A Cartesian co-ordinate system or rectangular co-ordinate system provides a method of plotting graphs and indicating the position of points on a two dimensional surface or in a three dimensional space. The cartesian co-ordinate system is used to define positions on computer displays.



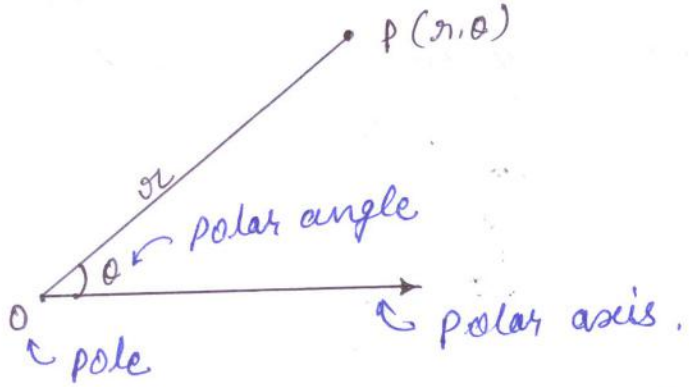


Each of this reference line is called a co-ordinate axis and the point where they intersect each other is called origin (0,0). The horizontal axis is called x-axis and vertical axis is called y-axis, and the plane containing those axis is called - x-y plane.

4. polar co-ordinate system :-

The polar co-ordinate system is a two-dimensional co-ordinate system in which each point on a plane is determined by a distance from a reference point and an angle from a reference direction.

In polar co-ordinate system, the reference point which is analogous to the origin of a cartesian system is called the pole. The ray that emantes from the pole in the direction of reference is called reference axis or polar axis. The position of a point is specified by its distance from the pole, called the radial co-ordinate ~~system~~ or radius, and the angle relative to the reference axis called as angular co-ordinate or polar angle.

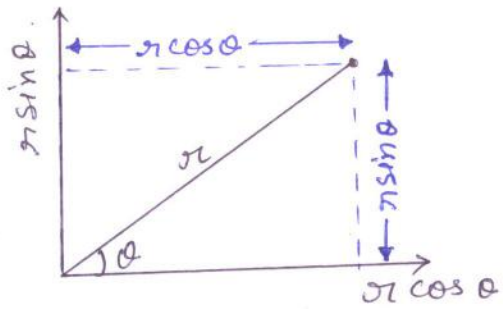


How can you convert polar coordinate system to Cartesian co-ordinate system :?

⇒ A polar coordinate system can be converted to the cartesian coordinate system by using the formulas :-

$x = r\cos\theta$

$y = r\sin\theta$

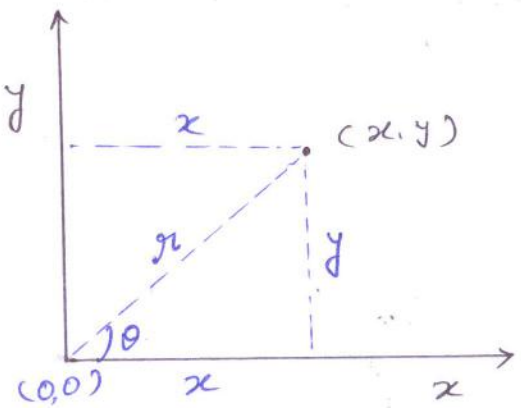


How can you convert cartesian coordinate system to polar coordinate system.

⇒ A cartesian coordinate system is converted to polar coordinate system by using the formulas :-

$$r = \sqrt{x^2+y^2}$$

$$\theta = \tan^{-1}(y/x)$$



What is drawing Entities?

⇒ An entity is a drawing element namely point, line, circle, arc, etc. These entities are used to create drawings of various complicated parts. Every engineering product is a combination of those entities.

Explain various methods to locate a point in CAD.

⇒ (I) pointing to the location on the screen by means of cursor control (from mouse or digitizer)

(II) Entering the co-ordinates via alpha-numeric keyboard. The co-ordinates of points can be defined either in rectangular or in polar co-ordinate system.

(III) Entering the offset (distance from x, y, z) from a previously defined point.

(IV) By defining the end point of an existing entity

(V) By defining the center point of an existing entity

(VI) By defining the intersection of two given entities.

State any three method by which lines can be drawn by using AutoCAD?

⇒ Line can be drawn by using the following three methods :-

1. using Absolute coordinate system :-

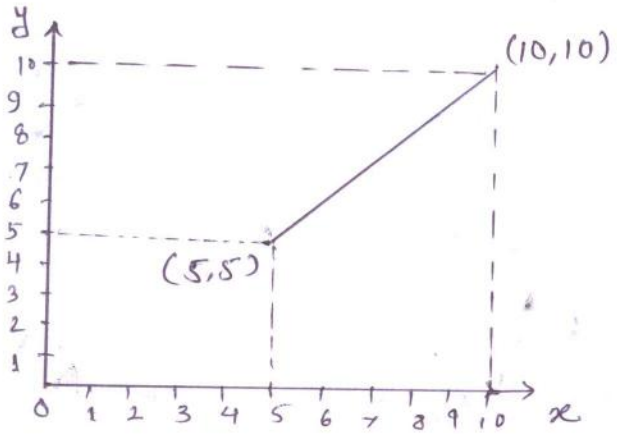
→ Draw a line from (5,5) to point (10,10)

command : LINE

from point : 5,5 (select the point by mouse or enter the coordinates by keyboard.)

To point : 10,10

To point : (press enter) ↵



2. using Relative coordinate System :-

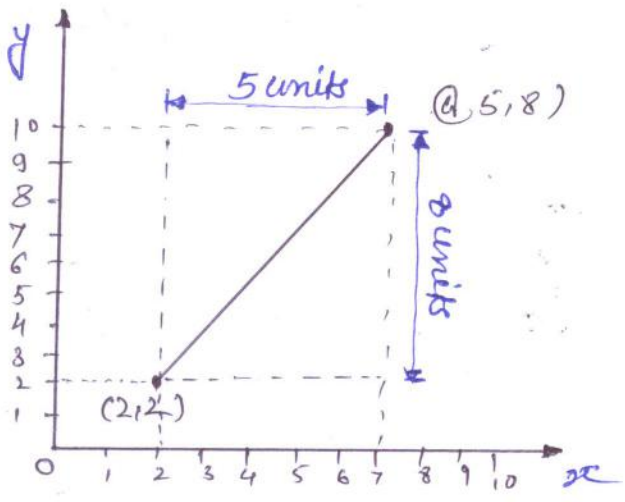
→ Draw a line from point (2,2) to point 5 units in x-axis and 8 units in y-axis relative to first coordinate

command : LINE

from point : 2,2

To point : @ 5,8

To point : (press enter) ↵



3. using polar coordinates :-

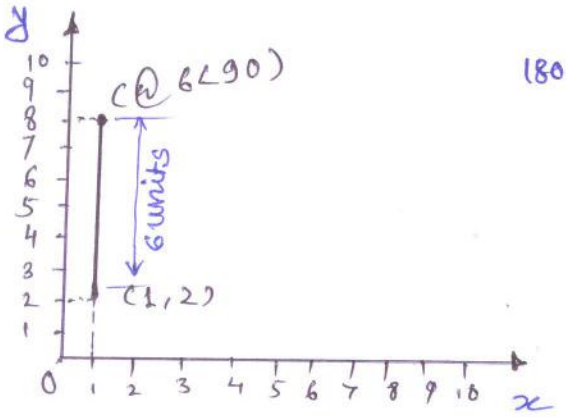
→ Draw a line from point (1,2) to a length of 6 units of 90°

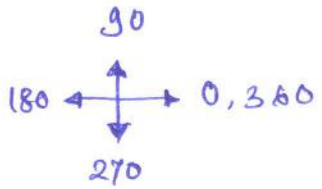
command : LINE

from point : 1,2

To point : @ 6<90

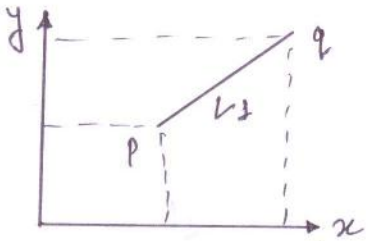
To point : (press enter) ↵



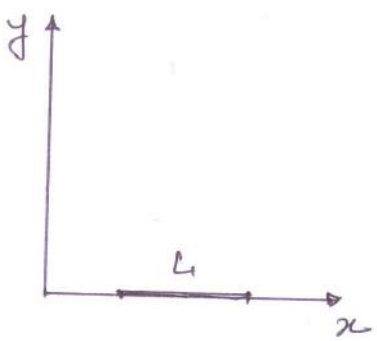


Explain methods of defining the lines is interactive computer graphics.

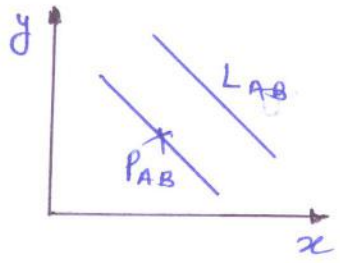
→ (i) By using two previously defined points :-

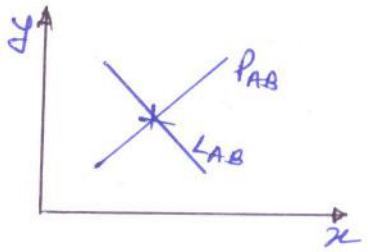


(ii) As one of the coordinates axis :-

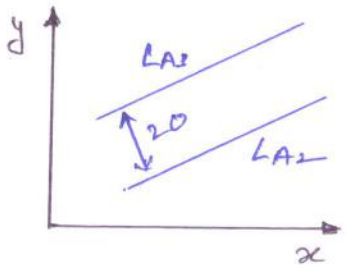


(iii) By making a line either parallel to or perpendicular to a given line and passing through a given point.

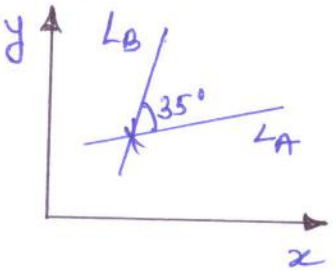




(iv) By making a parallel line at a given distance :-



(v) At an angle with a given line from a given point :-



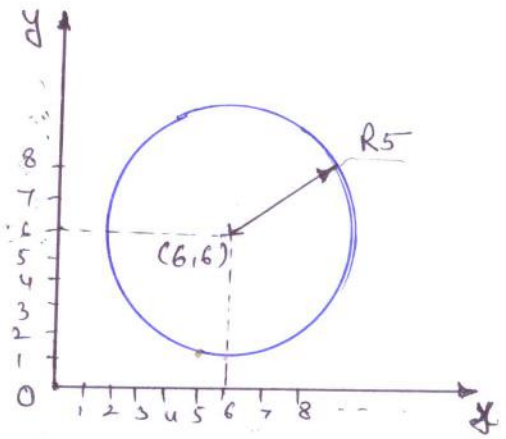
What are the methods to draw a circle in AutoCAD. Explain with example :-

⇒ Circle can be drawn by following five methods :-

1. Using Centre and Radius :-

⇒ Draw a circle with centre (6,6) and radius 5 units :-

Command : CIRCLE
3P/2P/TTR/<center point> : 6,6
Diameter / <Radius> : R
Radius : 5



2. using center and Diameter :-

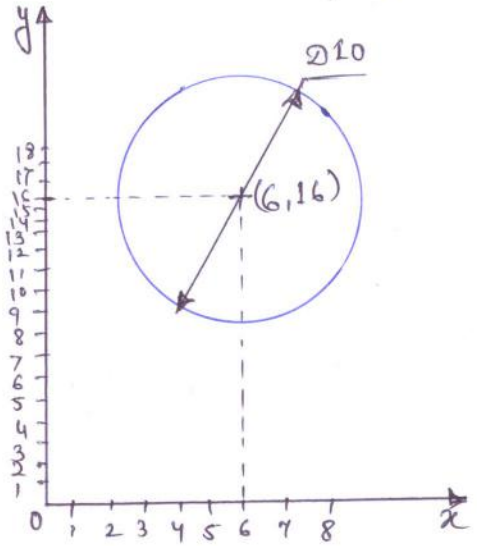
⇒ Draw a circle with center (6,16) and diameter 10 units

Command : CIRCLE

3P/2P/TTR/<center point> : 6,16

Diameter/<Radius> : D

Diameter : 10



3. using three points given (3P) :-

⇒ Draw a circle using the given 3 points : (5,30), (4,26), (10,25) by entering 3 given points to be on the circumference of the circle.

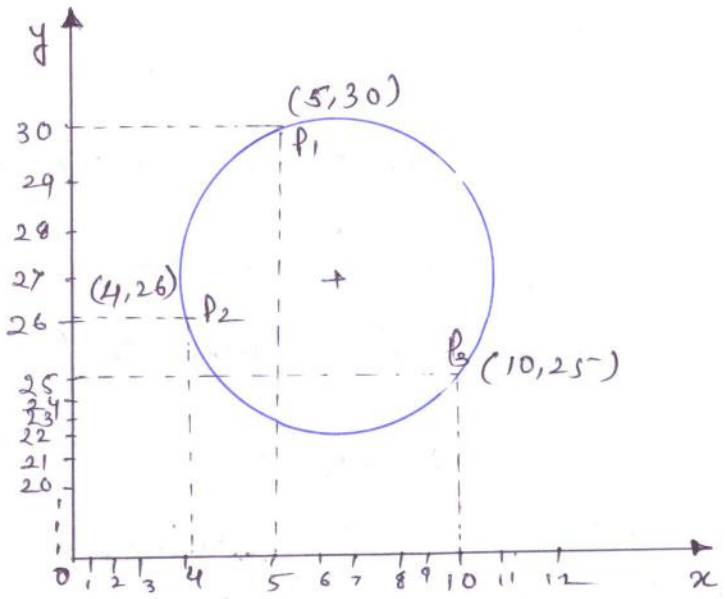
Command : CIRCLE

3P/2P/TTR/<center point> : 3P

first point : 5,30

Second point : 4,26

Third point : 10,25



4. Using Two points given (2P):

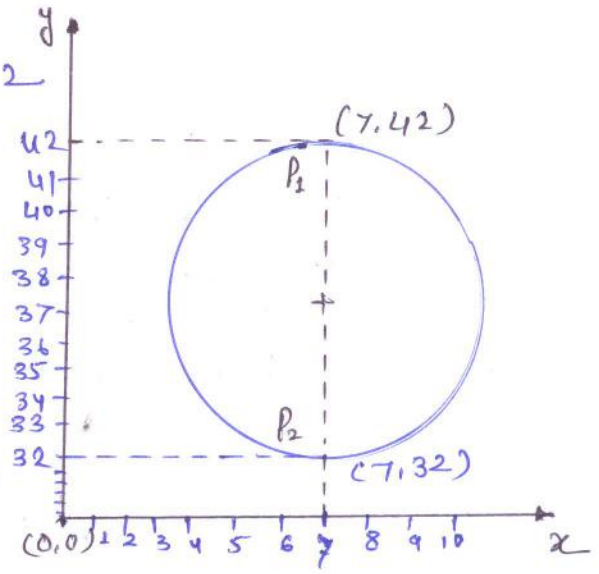
⇒ Draw a circle using the given 2 points : (7,32), (7,42) by entering two endpoints of the circle diameter

Command : CIRCLE

3P/2P/TTR / <center point> : 2P

first point on diameter : 7,32

Second point on diameter : 7,42



5. Using Tangent, Tangent and Radius (TTR):

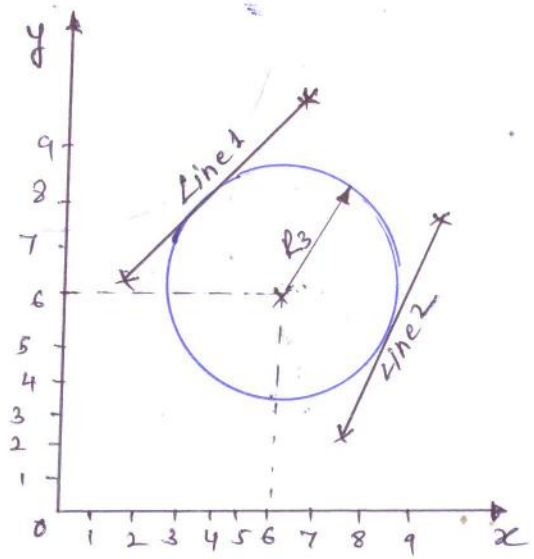
⇒ Command : CIRCLE

3P/2P/TTR/<center point> : TTR

Enter Tangent specification : Line 1 (pickup using mouse)

Enter second tangent specification : Line 2 (pickup using mouse)

Radius : 3



What are the methods to draw a polygon in AutoCAD? explain with example.

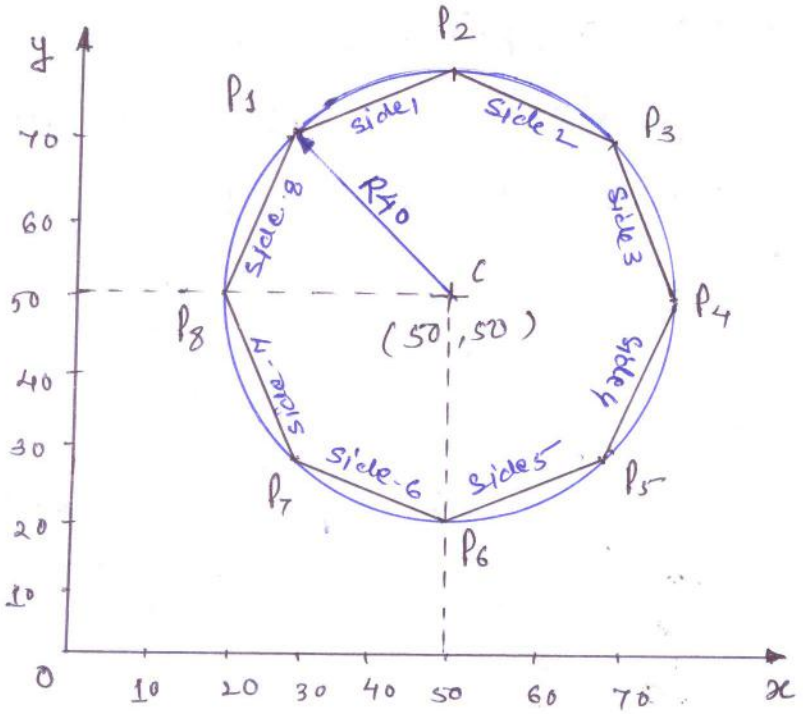
⇒ The polygon command draws regular 2D polygons with 3 to 1024 sides.

Any polygon can be drawn by the following three methods:-

1. Using radius of a given circle in which polygon is inscribed:

Ex:- Draw a polygon of eight sides with center (50,50) inscribed in a circle of radius 40 units.

Command : POLYGON
Number of Sides : 8
Edge / < Center of polygon> : 50,50
Inscribed in circle / circumscribed about Circle (I/C) : I
Radius of Circle : 40



2. using radius of a given circle on which polygon is circumscribed :-

Ex:- Draw a polygon of four sides with center (100,50) circumscribed on a circle of radius 40 units.

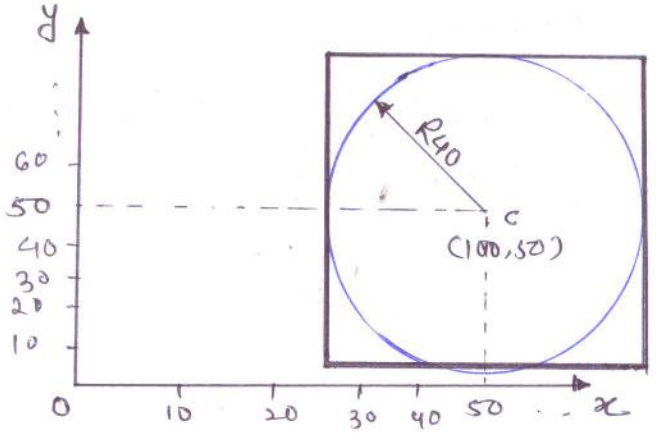
command : POLYGON

Number of sides : 4

Edge /< center of polygon> : 100,50

Inscribed in circle / circumscribed about circle (I/C) : C

Radius of Circle : 40



3. Using edge method :-

Ex:- Draw a polygon on 6 sides using "Edge method". The first endpoint of the edge is (30,50) and second is (60,50)

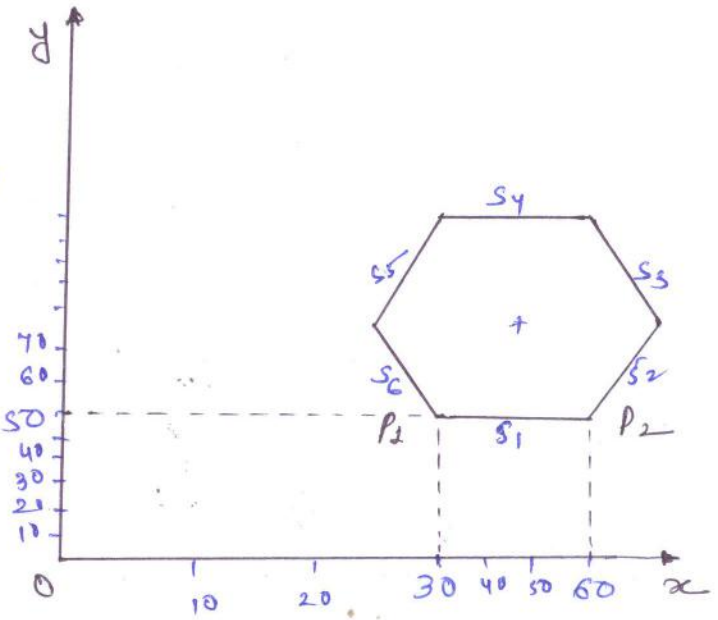
command : POLYGON

Number of sides : 6

Edge /< center of polygon> : E

first end point of edge : 30,50

second end point of edge : 60,50



What are the methods to draw an Ellipse in AutoCAD ? Explain with Example.

⇒ Ellipse can be drawn by any one of the following four methods :-

1. Using major axis and minor axis :-

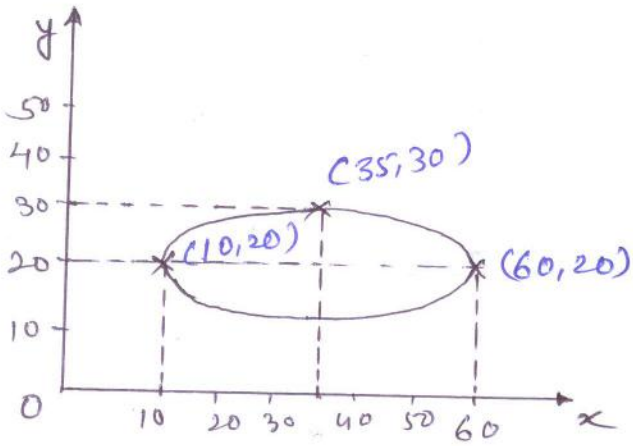
Ex :- Draw an ellipse using major axis endpoints, (10,20) and (60,20) and minor axis end point (35, 30).

Command : ELLIPSE

< Axis endpoint 1 > / ~~10, 20~~ center : 10,20

< Axis endpoint 2 > : 60,20

< other axis distance > / Rotation : 35, 30



2. Center of Ellipse, Axis Endpoint and other axis distance

ex :- Draw an ellipse with center (50,50) major axis endpoint (90,50) and minor axis endpoint (50,70)

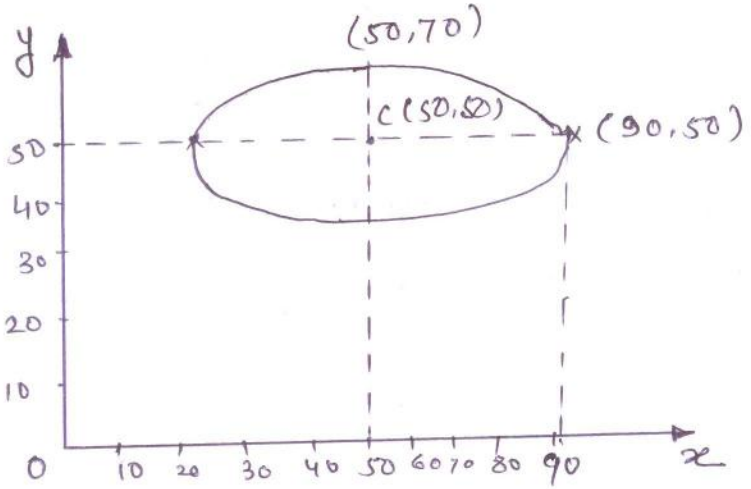
Command : ~~Ellipse~~ ELLIPSE

< Axis end point 1 > / center : c

Center of Ellipse : 50, 50

Axis end point 2 : 90, 50

< other axis distance > / Rotation : 50, 50



3. using first endpoints and rotation angle of circle around the axis :-

Ex:- Draw an ellipse using major axis endpoint (10, 40) and (110, 40) and 60° rotation around major axis.

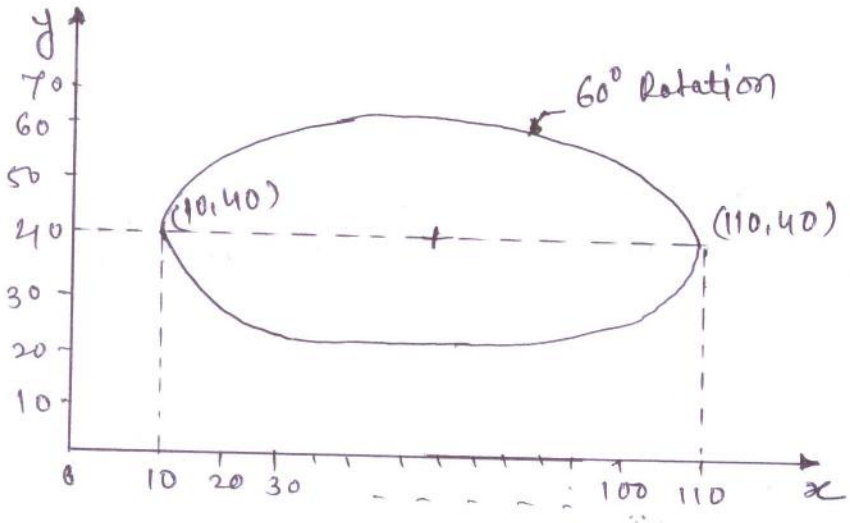
Command : ELLIPSE ↵

<Axis endpoint 1> / center : 10, 40

Axis end point 2 : 110, 40

<Other axis distance> / Rotation : R

Rotation around major axis : 60°



Note:- Angle of rotation should be always less than 90°

4. Using center, end point and rotation angle of circle around the axis :-

Ex:- Draw an ellipse with center (35,30) major axis end point (60,30) and 70° angle of rotation around the major axis.

Command : ELLIPSE

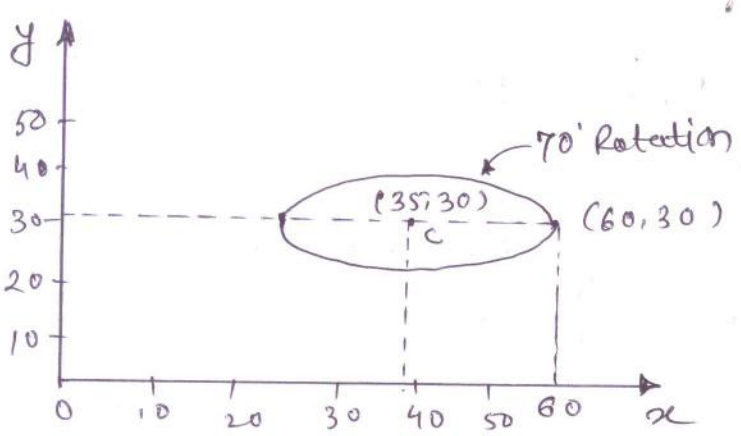
<Axis endpoint 1> / center : C

Center of Ellipse : 35, 30

Axis endpoint 2 : 60,30

<other axis distance> / Rotation : R↵

Rotation around the major axis : 70°



What are the methods to draw an arc in AutoCAD ? Explain with example.

⇒ Following methods are used to draw arc in AutoCAD :-

1. Using three given points :-

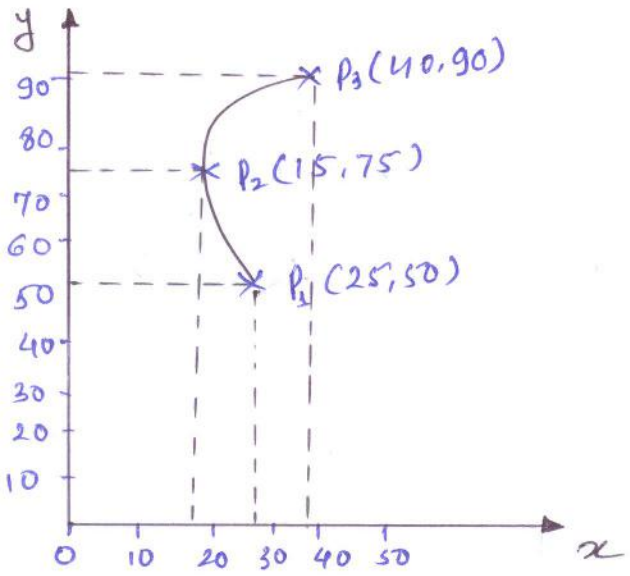
Ex:- Draw an arc using the three given points (25,50) (15,75) and (40,90).

Command : ARC

Center / <start point> 25,50

Center / End / <second point> : 15,75

End point : 40,90



2. Using Start point, centre and Endpoint : (S. C. E)

Example:- Draw an arc using start point (30,20), centre point (40,60) and Endpoint (40,100)

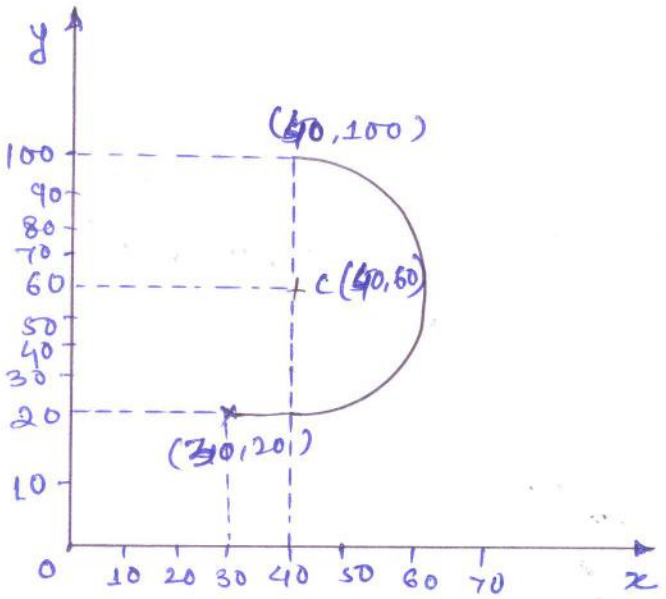
Command : ARC

Centre / <start point> : 30,20

Centre / End / <second point> : C

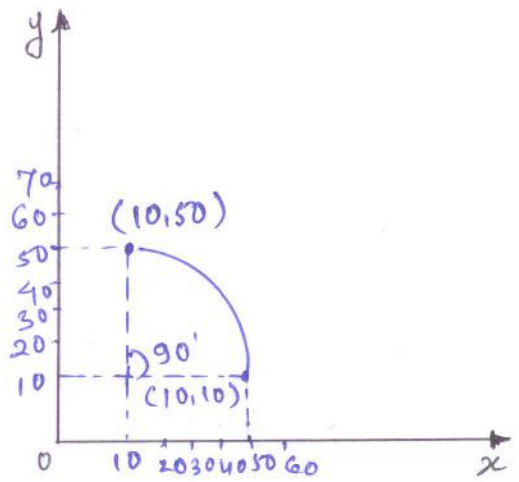
Centre point : 40,60

Angle / Length of chord / <End point> : 40,100



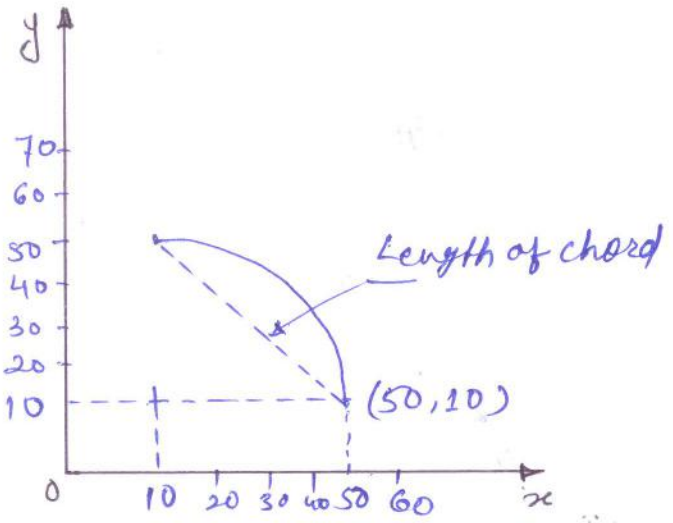
3. Using start point, centre and Included Angle: (S, C, A)

Example:- Draw an arc using start point (10,50) Centre point (10,10) and Included Angle 90°



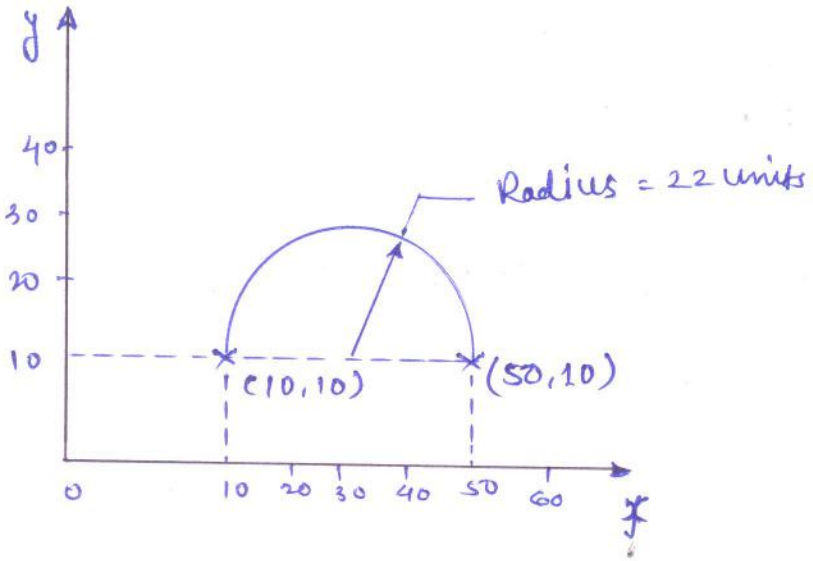
4. Using start point, centre and Length of chord: (S.C.L)

Example:- Draw an arc using start point (50,10). Centre point (10,10) and length of chord 45 units.



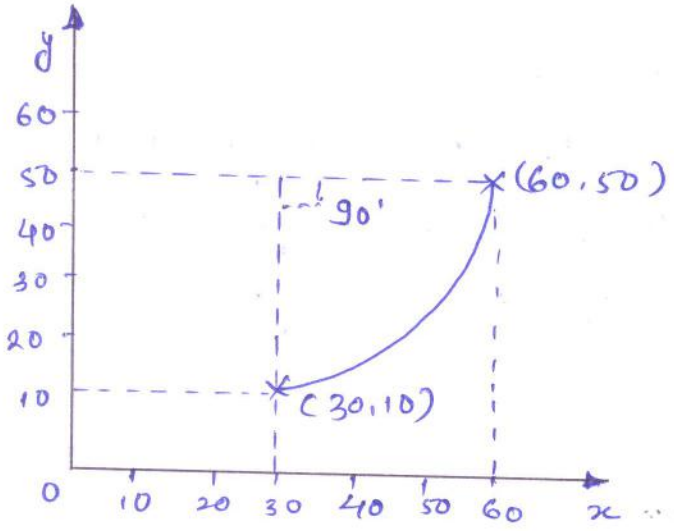
5. Using start point, Endpoint and Radius : (S.E.R)

Example:- Draw an arc using start point (50,10), Endpoint (10,10) and radius 22 units.



6. Using start point, Endpoint and Included Angle : (S,E.A)

Example:- Draw an arc using start point (30,10), End point (60,50) and Included angle 90°

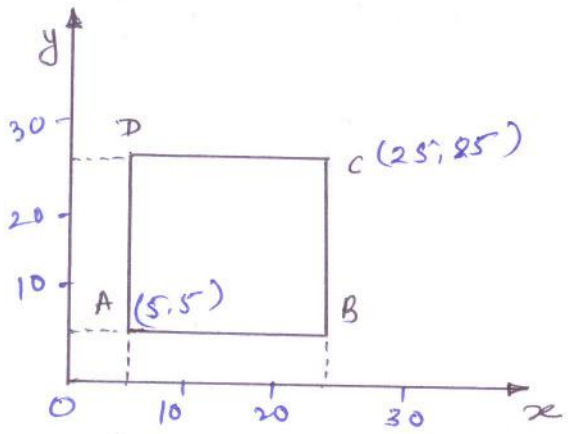


Explain the method to draw a rectangle in Auto CAD with example.

1. By specifying the coordinates of first corner point and other corner point :-

For example :-

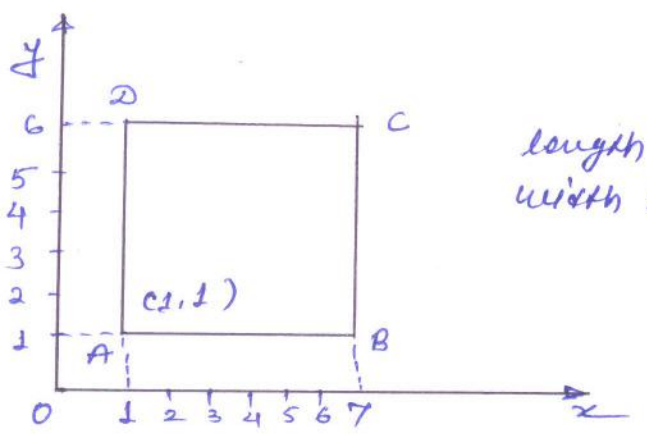
command : RECTANGLE
Specify lowermost left corner : 5, 5
Specify topmost right corner : 25, 25



2. By specifying length distance and width distance of rectangle

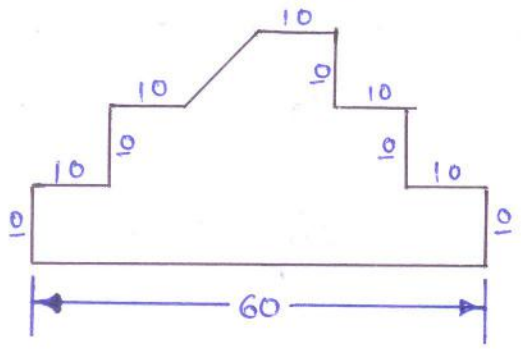
For example :-

command : RECTANGLE
Specify first corner of the rectangle : 1, 1
Specify the length of the rectangle : 6
Specify the width of the rectangle : 5



length = 6
width = 5

state the series of commands steps required to reproduce the object, using line commands.



⇒ command : LINE

specify first point : (select any point by mouse.)
Specify next point or [undo] : @ 60,0
Specify next point or [undo] : @ 0,10
Specify next point or [close/undo] : @ -10,0
Specify next point or [close/undo] : @ 0,10
Specify next point or [close/undo] : @ -10,0
specify next point or [close/undo] : @ 0,10
Specify next point or [close/undo] : @ -10,0
specify next point or [close/undo] : @ -10,-10
Specify next point or [close/undo] : @ -10,0
Specify next point or [close/undo] : @ 0,-10
Specify next point or [close/undo] : @ -10,0
Specify next point or [close/undo] : C

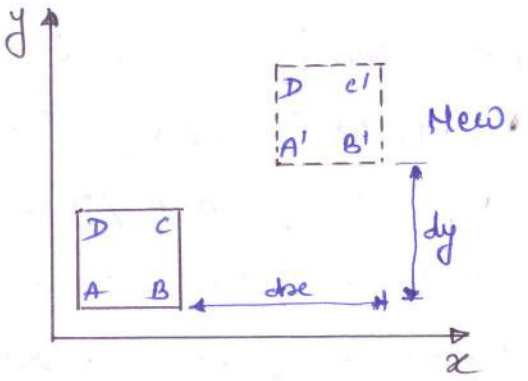
Explain the various basic Editing Commands used in AutoCAD :-

⇒ A drawing may be modified either during its preparation or during the revision of design, This is called editing of the drawing.

The Editing commands also called as modifying Commands :-

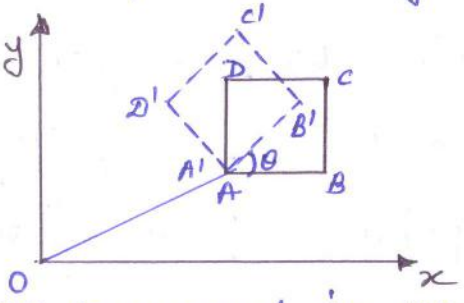
1. MOVE :- Move command is used to move a selected object to another location about a base point or fixed point.

⇒ Syntax : MOVE



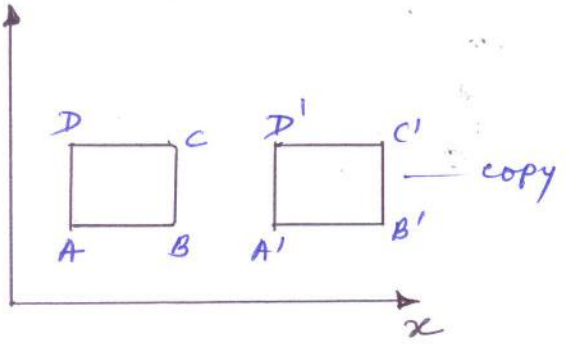
2. Rotate :- Rotate command is used to rotate a selected object through a specified angle about a base point.

⇒ Syntax : ROTATE



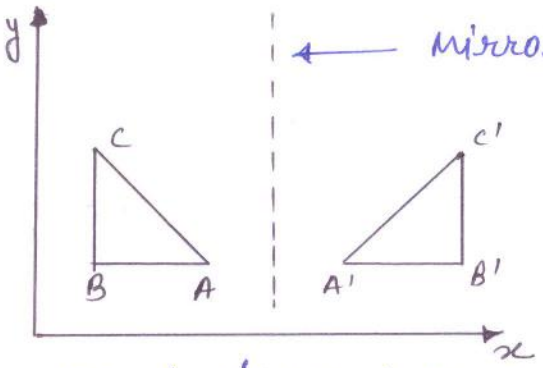
3. Copy :- Copy command is used to create one or more copies of selected object at another location.

⇒ Syntax : COPY



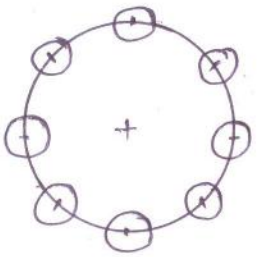
4. Mirror:- Mirror command is used to create a mirror image of the selected objects about a specified line.

⇒ Syntax: MIRROR

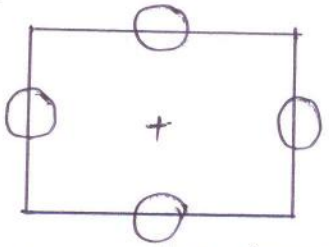


5. Array :- This command is used to create multiple copies of selected objects in rectangular or polar form This is a form of copy command

⇒ Syntax: ARRAY



(Polar Array)



(Rectangular Array)

6. Erase :- This command deletes the selected entities. A record of entities erased is always maintained. The most recent entity can be unerased by OOPS command.

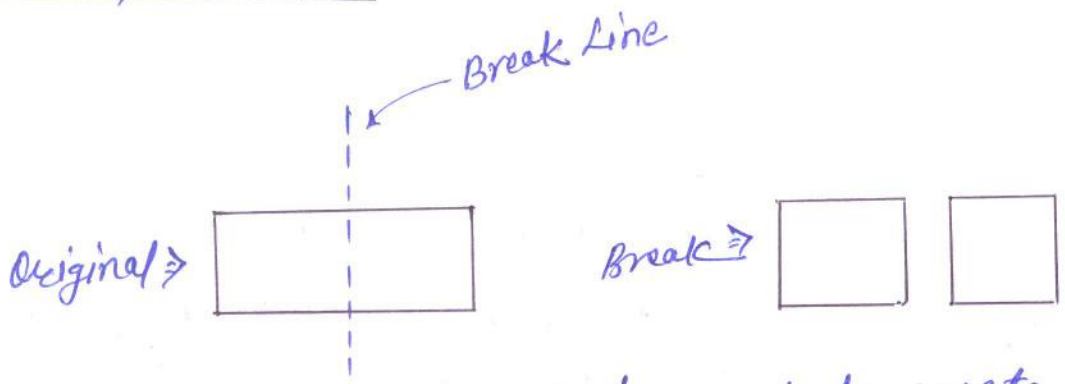
⇒ Syntax: ERASE

7. OOPS : This command retrieves all objects erased by the last erase and after executing block or wblock command

⇒ Syntax: OOPS

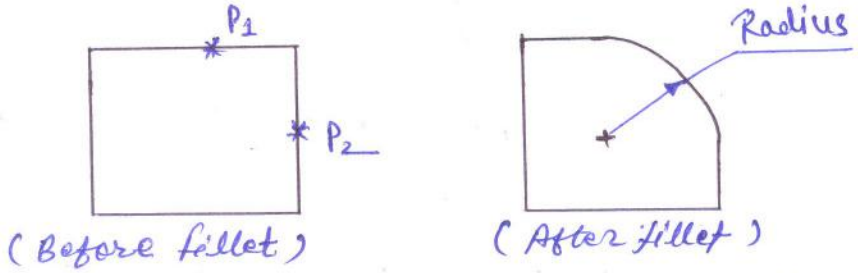
8. Break : This command erased a portion of line, arc, circle or a 2D polyline between two selected points.

⇒ Syntax : BREAK



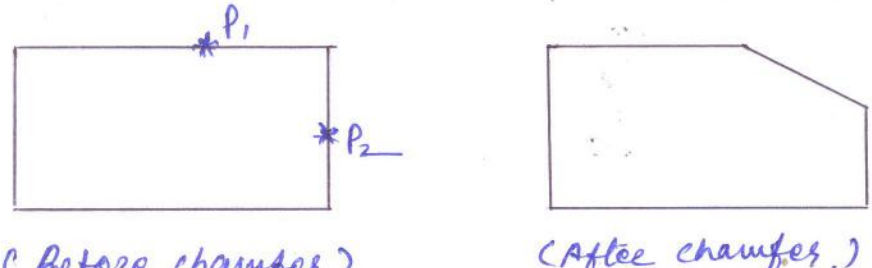
9. Fillet : This command is used to create a round corner between two lines. fillet works on any type of combination of two lines, arc, circle, non-parallel lines or a single polylines.

⇒ Syntax : FILLET



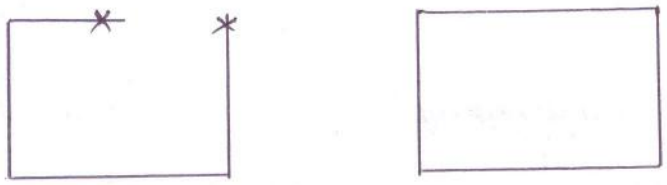
10. chamfer : This command works on two lines or a single polyline to create a bevelled edge.

⇒ Syntax : CHAMFER



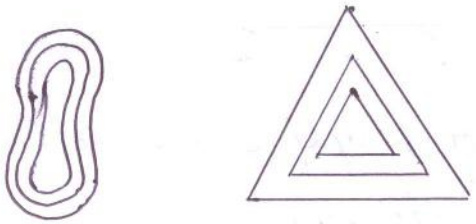
11. Extend : This command extends the lines, polylines and arc to a boundary edge which can be a line, polyline, arc or circle.

⇒ Syntax : EXTEND



12. Offset : This command creates a parallel single copy of line, arc, circle, rectangle, polygon or 2D polyline of a given offset distance. Each offset creates a new entity with the same line type, colour and layer setting.

⇒ Syntax : OFFSET



13. Stretch : The stretch command can either lengthen entities or shorten them and thus alter their shapes The centre points of arcs or polyline ares are - adjust accordingly.

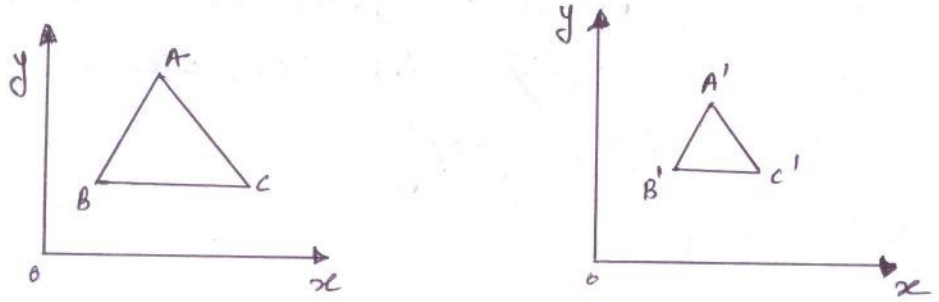
⇒ Syntax : STRETCH

14. Trim :- Trim command trims the object that extend beyond a requised point of intersection.

⇒ Syntax :- TRIM

15. Scale :- The scale command allows to shrinks or enlarge the already existing drawing objects about a base point by specifying scale factor.

⇒ Syntax :- SCALE



16. Pedit :- A polyline is a single entity which is made up of a continuous series of line and arc segments. The PEDIT command is exclusively used for editing of polyline properties.

⇒ Syntax : PEDIT

17. Explode :- This command breaks a polyline into its individual segments. These segments can then ~~end~~ individually be edited, and rejoined again to form an edited polyline.

⇒ Syntax : EXPLODE.

18. U :- U command reverse the effect of a series of previously used command and hence allows back stepping.

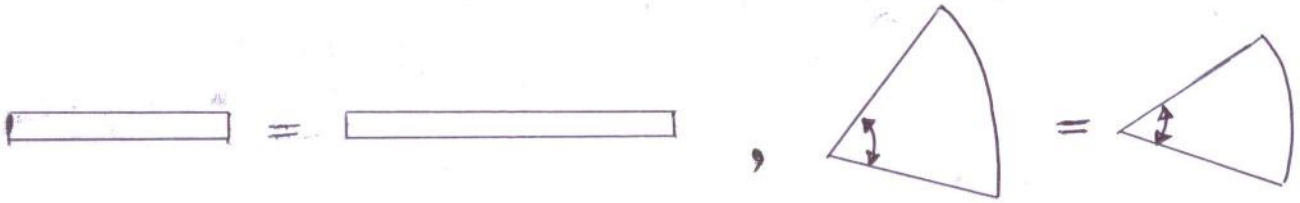
⇒ Syntax :- U or UNDO

19. Hatch :- Hatch command is used to fill an enclosed area or selected objects with a hatch pattern, solid fill or gradient fill.

⇒ Syntax :- HATCH

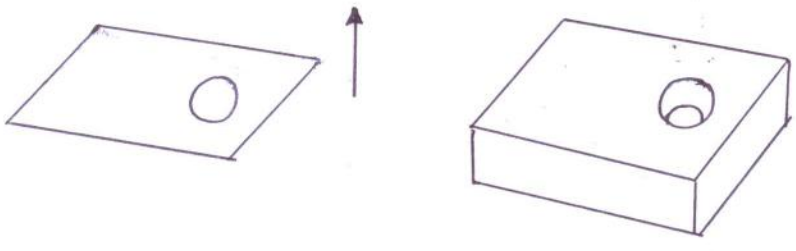
20. LENGTHEN :- This command changes the length of objects and the included angle of arcs.

⇒ Syntax : LENGTHEN



21. Extrude :- This command is used to convert 2D object to 3D object along a path.

⇒ Syntax :- EXTRUDE.

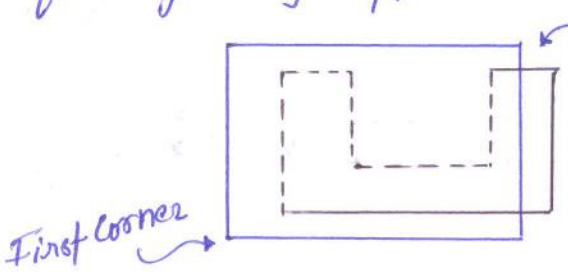


What do you understand by object selection method for editing ?

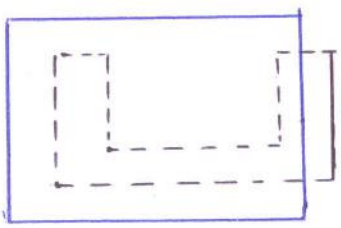
⇒ following are the methods for selection of object in AutoCAD

1. Object Picking :- In this option a single object is selected by pressing the left mouse button touching the entity. A small box is marked over the selected entity as shown in figure :-

2. Window option :- A single entity or a group of entities can be selected by bringing them fully inside a rectangular rubber band like window. window is created by picking the first corner by ~~pressing~~ pressing the left mouse button and then moving the mouse over the desired position of diagonally opposite corner.



3. cross window :- It works like window but selects all ~~selects~~ the entities with in window or touching the window boundaries :-



what are the display commands used in AutoCAD?

⇒ The display commands used in AutoCAD are as follows:-

1. Zoom :- The zoom command increases or decreases the apparent size of objects in the current viewport.

command: ZOOM

2. PAN :- The PAN command slides or shifts the current drawing around on the drawing area without changing the magnification of the view. command: PAN

3. REGEN :- The co-ordinates, angles, radii etc. are stored in a database as floating point values for greater precision. However, the drawing is placed on the screen using pixels which are integers, therefore every time data have to be transformed from the floating point values to the integer values, this process is called regeneration.

command :. REGEN or RE

4. REDRAW :- The AutoCAD saves the integer screen co-ordinates calculated during the regeneration process and can replay them very quickly by the .REDRAW command.

command : REDRAW or R

What are the commands used for drafting in AutoCAD?

⇒ Drafting commands used in AutoCAD are following :-

1. Limits :- This command allows to change the upper and lower limits of the drawing area while working on a drawing.

2. Layer :- Objects of a drawing can be placed on one or more transparent sheets called the layer. The LAYER command is used to create new layers.

3. Dimensioning :- The manufactured drawing must be dimensioned for size and tolerances so that the right information can be conveyed. The appearance and size of dimensions, arrow, size and style of dimension text with or without tolerance and the layer on which dimensions are placed can be controlled by setting dimension variables.

4. Object Snap : A very useful drawing command, the OSNAP identifies the points on drawing entities that are visible on screen.

command : OSNAP.

what are the basic utility command used in AutoCAD ?

⇒ The commands which control the basic functions of AutoCAD are called as utility commands.

1. LIMITS :- 'LIMITS' command allows you to change the upper and lower limits of the drawing area while working on a drawing.

2. SAVE :- 'SAVE' command saves the current drawing which is in the screen, and remains in the drawing editor for further editing.

3. QUIT :- 'QUIT' command returns to the main menu without updating the drawing file.

4. END : 'END' command returns to the main menu and updates or saves the drawing file.

5. HELP : 'HELP' command lists all the commands used in AutoCAD. The list displays the name of command with its basic function. This can be used in case you have forgotten any command.

Explain Dimensioning in AutoCAD.

⇒ The dimensioning are inserted in the drawing by the use of 'DIM' command. There are various types of dimensioning used in AutoCAD :-

P.T.O.

1. Linear Dimensioning :- → Horizontal.
→ Vertical.
→ Aligned.
→ Rotated.

2. Angular Dimensioning :- This allows angular Dimensioning of objects.

3. Radial Dimensioning :- This allows radial Dimensioning of circle or arc.

4. Diametral Dimensioning :- This allows diametral Dimensions of a circle.

SOLID MODELLING

What do you mean by solid modelling or Geometric Modelling ?

⇒ It is a process of generating 3D objects for analysis, design, drafting and manufacture. It creates a database which is used to display the object to prepare drawings of the object with different views to prepare data for analysis and design.

In CAD, Geometric modelling is concerned with the computer compatible mathematical description of the geometry of an object i.e. graphical representation of the shape and size of the physical objects.

The geometric models can be brodly categorized into two types –

(i) Two-Dimensional.
(ii) Three-Dimensional.

⇒ Use of geometric modelling :- Geometric modelling could be used for following functions :-

1. Design Analysis.
2. Drafting.
3. Manufacturing.
4. production Engineering
5. Inspection and quality control.

What are the types of solid modelling or geometric modelling?

⇒ Basically there are three types of modelling system :-

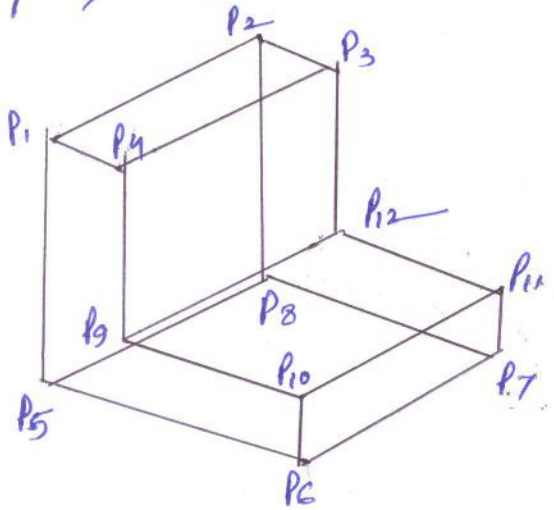
1. Wire frame modelling :- It is the simplest of the three models. These are generated by continuing line segments. The points and lines generate the object and then translation, rotation or scaling is carried-out. The wireframe models define the edges and surface of 3D objects. All the objects are formed of polygon meshes. These models carry 3D wirframe surface.

Wire frame modelling can be classified into following three types :-

(I.) 2-D = used for two dimensional representation of flat objects.

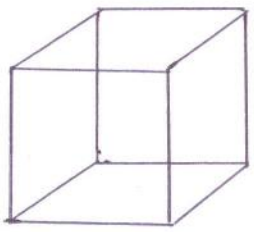
(II) $2\frac{1}{2}$ D = This can be used to represent three dimensional object as long as it has no side walls.

(III) 3D = This is used for full 3D modelling of a complex solid.



Explain the types of solid modelling :-

⇒ 1. Constructive Solid Geometry :- The constructive solid geometry (CSG) approach is a very powerful ~~technique~~ technique for representing fairly complex jobs with relative ease. This ~~technique~~ technique is based on the topological notation that an object can be divided into a set of primitives (basic elements or shapes), which can be combined again to form the object by following a set of Boolean operations. Some commonly used primitives in the solid modelling are shown as follows :-



Block

Pyramid

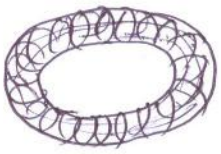
Cylinder

Cone

frustum of cone

Hemisphere

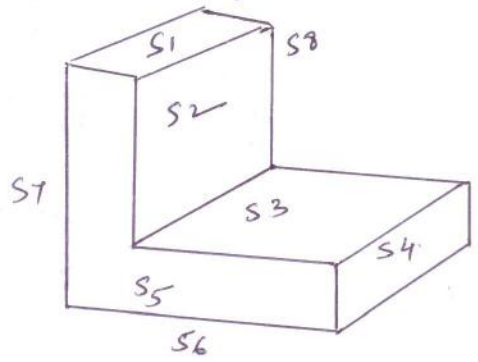
sphere



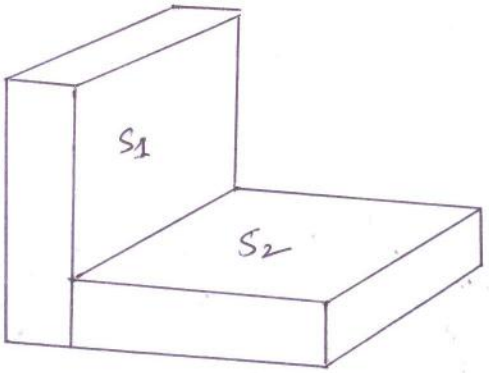
Torus

2. Boundary Representation (B-rep) :- Boundary - representation (b-rep) is one of the most popularly used technique of solid modelling. A b-rep model is represented as a volume contained in a set of faces together with topological information about the relationship between the faces.

2. Surface Modelling :- Surface model is constructed from surfaces such as planes, curved surfaces and even very complex surfaces, such as Bezier and B-splines. These models are capable of completely representing the solid from the manufacturing point of view. However a surface model does not give any information regarding the interior of the solid, required for generating the NC cutter data.



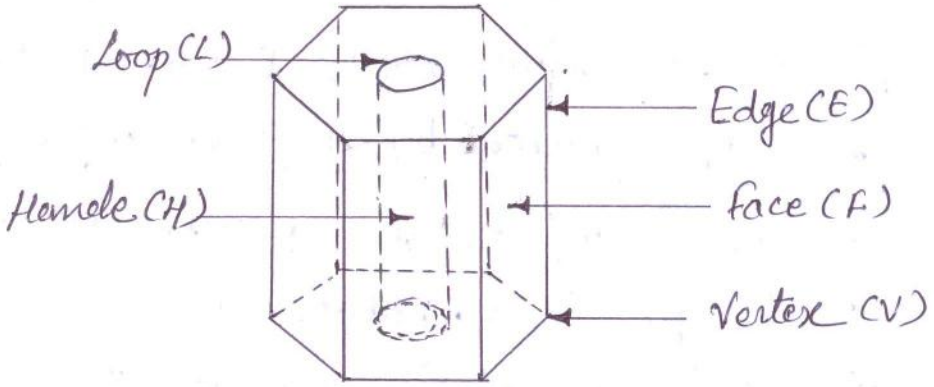
3. Solid modeling :- Solid modelling is the most powerful method for the three-dimensional solid representation. In this technique models are represented as solids, thus there is lesser risk of mis-interpretation. By adding colour to these drawings they just become realistic.



A b-rep solid is bounded by its surface, which separates points inside from points outside of the solid. Because of this a b-rep solid is represented as a closed space in 3-D space. The geometry of a b-rep solid is described by its vertices, edges and surfaces. An edge is bounded by vertices and face is bounded by edges. faces of solid can be formed by either straight lines or curved segments.

A B-Rep model of any object consists following elements:-

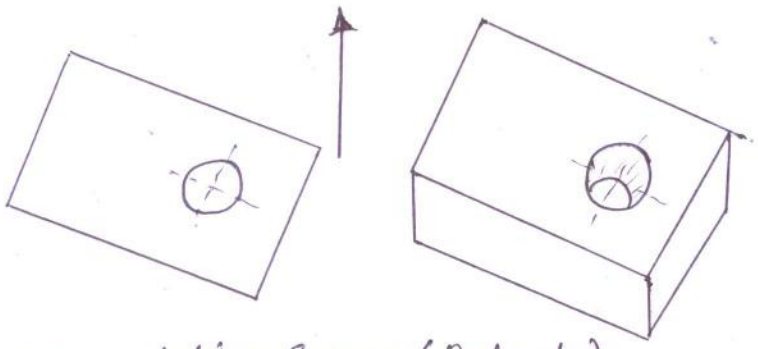
(I) Vertex
(II) Edge
(III) face
(IV) Loop
(V) Handle
(VI) Body



Name and discuss two methods of Geometric constructions:-

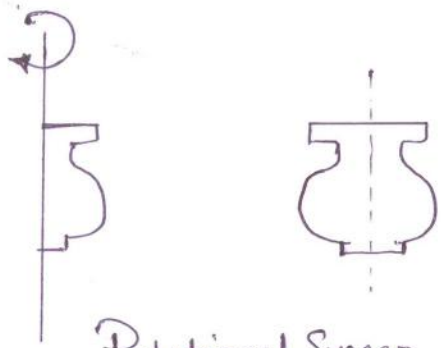
⇒ 1. Linear Extrusion:- (Translation Sweep):-

In linear extrusion or also known as translation sweep, a three dimensional solid as shown in fig. can be generated by linearly sweeping a two-dimensional surface along a straight line. The sweeping does not need to be a straight line, it can be any three-dimensional space curve for generating reasonably complex geometry.



Translation Sweep (Extrude)

2. Rotational Sweep :- In Rotational Sweep, the planer two dimensional point-set is rotated about an axis of rotation by a given angle. For sweeping helical or Spiral objects an additional axial or radial offset can be provided. Rotational Sweep can only be used for axi-symmetric components.



Rotational Sweep

Compare 2D and 3D wireframe modelling with respect to their utility for an engineering industry.

⇒

Sr. No.	2D - Models	3D - Models
1.	Used for drawing of simple 2D objects	Used for drawing of Complicated 3D objects.
2.	Requirs only one global coordinate system (WCS)	May require the use of several user defined coordinate system to create features on various faces of the part.
3.	It is difficult to reconstruct 3D Images	Along with 3D images, 2D views and even pictorial views can be generated easily.
4.	Ends of lines are represented by (x, y) coordinates	Ends of lines are represented by (x, y, z) coordinates.
5.	Curved edges are represented by circle, elipses, splines	Curved edges are represented by suitably spaced generators.
6.	To correctly represent a complex object, and additional view, sectional views are required.	Using hidden line/surface elimination complex components can be interpreted correctly.