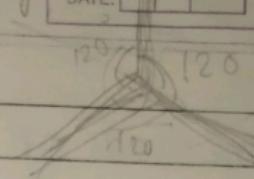


the view drawn with the true scale is called isometric drawing. while that drawn with the use of isometric scale is called Isometric projection.

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CAD (Computer Aided Drafting)

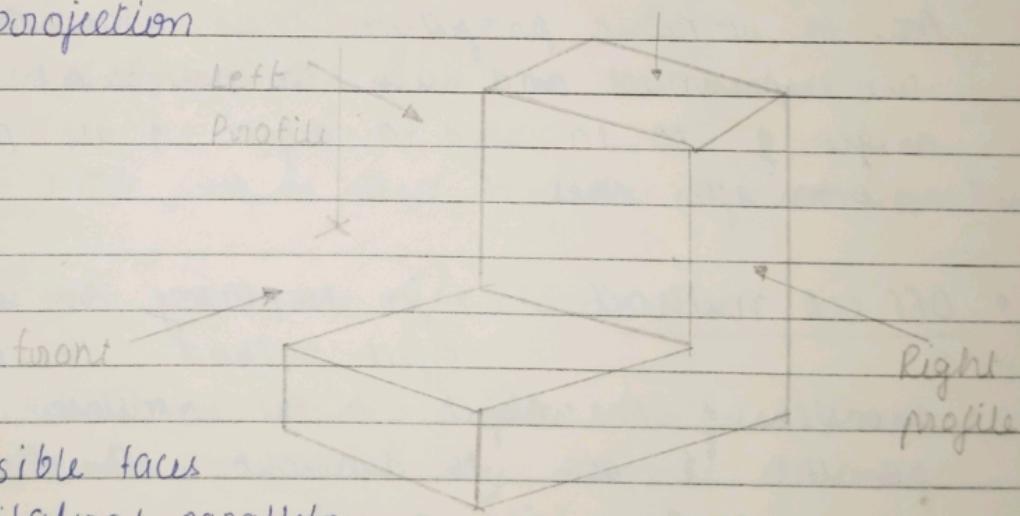


Q1 - what is Isometric projection? name various methods of isometric projection.

Isometric projection in technical and engineering drawing is a method of visually representing three-dimensional objects in two dimensions.

The isometric projection is one class of Orthographic projection and if the plane is oriented an isometric projection results so that it make equal angles.

- Isometric projection



The three visible faces appear as equilateral parallelogram in isometric drawing of a cube.

The plane is placed in such a way that all three visible sides of the object make the same angle with one another at an angle of 120° .

o methods of Drawing - ISOMETRIC Projection

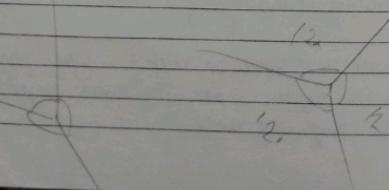
- i) Box method
- ii) Offset method

o Box method : The box method is easy and intelligent and for the drawing it takes much time.

The maximum length, height and breadth are noted in this method and by these dimensions, a box is constructed.

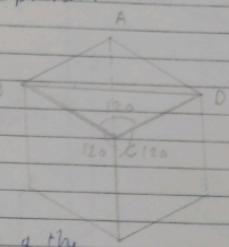
Acc. to isometric projection, these dimensions of the box are represented and with the horizontal line they make angle of 30, 30 and 90 degrees & all parts of object are shown after this.

o Offset method : for preparing an isometric view in offset method the side, length and breadth of the object to the isometric axis are drawn parallel & acc. to dimension. The other isometric & non-isometric line are drawn.



- Isometric axes, line and planes:

- o The three line CB, CD, CA meeting at the point C and making 120° angle with each other are termed isometric axes. The lines parallel to these axes are called isometric lines.



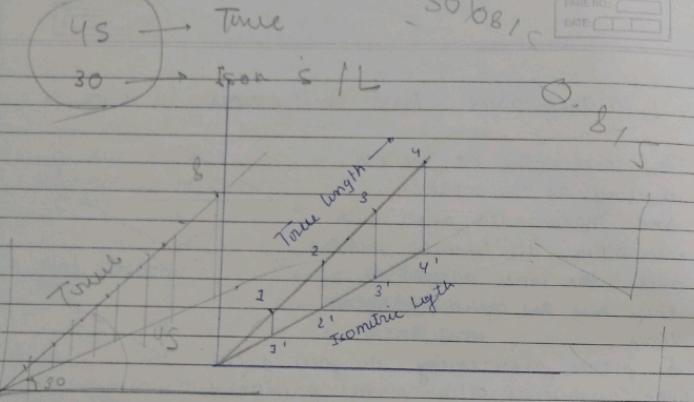
The planes representing the faces of the cube as well as other planes parallel to these planes are called isometric planes.

Q - What is Isometric Scale?

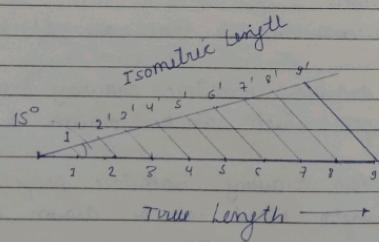
An isometric scale can be used to draw correct isometric projections.

All distance in this scale are $\frac{2}{3} \times$ true size.

- o for drawing isometric drawing / views we use true length which can be drawn by normal scale.
- o but for drawing isometric projection we use reduced length which can be drawn by Isometric scale.



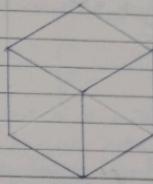
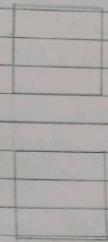
$$50 \times 0.815$$



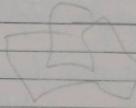
If you have to draw isometric projection multiply true length with 0.815

Q. Draw an isometric view of cube of side 25 mm.

FV



TV



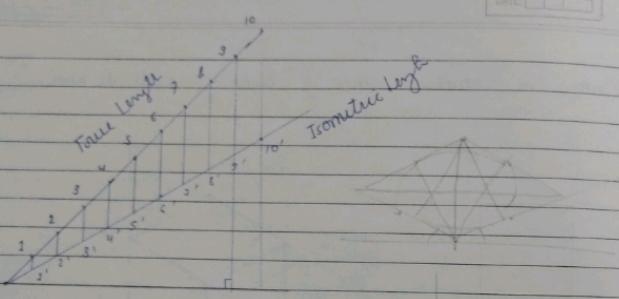
Q. Draw the isometric projection of the frustum of a cone of 50 mm base diameter, 25 mm top diameter & 50 mm height.

→ first draw isometric scale as this projection.

→ Draw the orthographic views

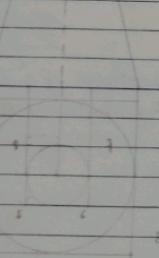
→ then for isometric view

→ measure 30° angle on horizontal axis

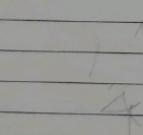


orthogonal view

60



orthographic view



Q - what is CAD (computer Aided Drafting)

Computer aided drafting/design is define as the process of digitally creating design simulations of real world goods and produce in 2D or 3D, complete with scale precision and physical properties to optimize & perfect the design.

→ Who uses CAD :

computer aided design is used in a wide variety of professions. CAD software is used heavily within various architecture, arts and engineering projects.

→ Professions that use CAD tools include, but are not limited to :

- Architect
- Engineers
- Graphic designers
- Drafters
- Interior designer
- Animation illustrator

◦ The major functions to be performed by CAD system :

- i) Basic set up of drawing
- ii) Drawing the objects
- iii) Dimensioning
- iv) Text insertion
- v) Translating the object

standard

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Q- Advantages of CAD ?

- Compared to traditional technical sketching and manual drafting, the use of CAD design tools can have significant benefits :

I. Low Maintenance cost . /

II. Efficient workflows (quicker project completion)

III. modelling and drafting (Provides 2D + 3D modularity)

IV. flexibility : (hundreds of computers, display devices)

V. Ease of use (User finds CAD system very easy to learn)

VI. modularity (std. input & output devices are fitted to standard connectors)

VII. high quality (high quality designs with documentation)

VIII. Decrease error percentage

IX. easy to share (easier to save files)

X. easy modification

VI) CAD software requires fairly expensive startup cost.

Q- Disadvantages of CAD ?

i) Work can be lost because of sudden breakdown of computer

ii) Work is prone to viruses

iii) Work could be easily hacked.

iv) Time taking process to know how to operate / run

v) need of regular updating of software .

Q- What is CAD software ? Name three popular CAD softwares

→ CAD software, is a powerful tool for engineering design. It allows you to create, modify, analyse and optimise digital models of physical objects & systems.

most CAD system creates an environment to prepare drawings interactively. most CAD system available commercially are menu driven. Commands can either be typed directly with the help of keyboard or can be picked up from the screen menu

CAD system, it also allows experimentation with different ideas and solutions .

- Popular CAD software are :

I. AutoCAD

one of the oldest and most used CAD software for 2D + 3D drafting + design. It has ability to make blue-print + model documentation.

- used by
- i) Electrical engineers
 - ii) interior engineers
 - iii) industrial engineers.

II. Civil 3D

An autocad product originally created as an add on for AutoCAD. Civil 3D is a software used by civil engineers + other professionals to plan design.

- Features →
- i) surface creation tools
 - ii) support for building information

III. Inventor

A 3D modelling software and the main competitor of SolidWorks used in mechanical design, + product development.

- Features →
- Product data management
 - visualization
 - 3D solid modeling

Q - What is AutoCAD?

AutoCAD is a low cost yet very efficient computer aided design and drafting software. AutoCAD is acceptable as the industry standard and it is preferred by a large community of CAD users in world.

It allows you to draw and edit digital 2D + 3D design more quickly, easily saved and stored in cloud.

- Benefits of AutoCAD:
- i) Easy edit

ii) faster production

iii) Better Accuracy.

• Drawing entities in AutoCAD:

1. Line : A line is specified by giving its two endpoints.
2. Pline : Polyline are interesting drawing entities consisting of straight line segments.
3. Polygon : A polygon is also a polyline with equal length of sides.
4. Arc : This command is used to draw an arc accurately usually there are three parameters required for drawing an arc.
5. Circle : There are many ways of drawing a circle the default being the center, points + circle + radius.

Q. Explain any two methods of drawing arc in AutoCAD.

In AutoCAD, the Arc command is often used to produce arc:

- Command Line: Arc or a
- menu bar: Draw Arc
- Draw bar: Arc symbol is there.

The following message appears:

- specify start point of arc or [center]
- specify the second point of arc or [center/end]
- specify endpoint of arc or [Angle/chord/Length]
- specify center point of arc or [Angle/Div"/Radius]

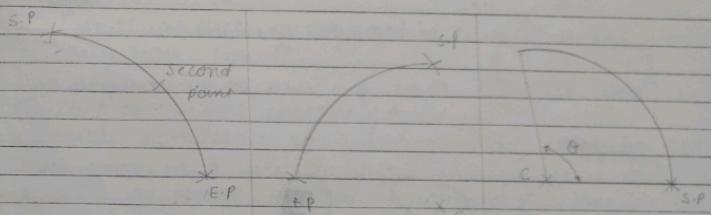
Ans:- There are various methods to draw an Arc.

1. 3 point Arc
2. start, center
3. start end.

I. 3 point Arc: The arc is drawn by specifying three points on the chord of arc. The first and third point define the start and end point of arc respectively.

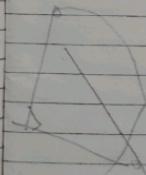
II. Start, center: This option needs start point/circle point of an arc. The third parameter may either be an end point, included angle, or length of chord.

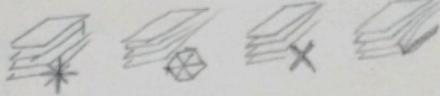
III. Start, End: This option asks the user to enter the start and end points of an arc. The arc is completed by either specifying radius or included angle or center point.



1. 3-point Arc

2. start, center 3. start end





In a floor plan of a building, the walls, ceiling, plumbing fixtures can be put on separated layers.

In this way we can display or plot them individually or combine them in different ways.

Any no. of layers can be made visible at any time.

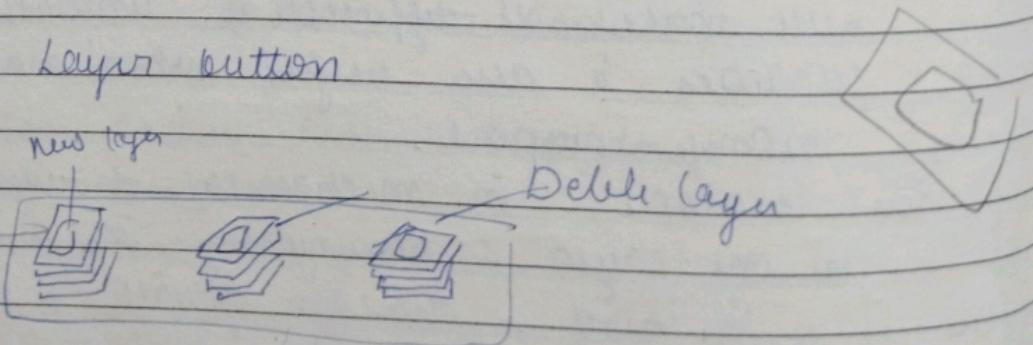
A carefully planned layering scheme helps to produce a drawing that contains the types of information needed in each case.

- Using layers also allows easy classification of drawing
- AutoCAD allows unlimited no. of layers

Q → How can we create new layer?

By default all AutoCAD drawings have a single layer named 0. This is default layer which cannot be deleted.

- i) click the layer properties tool OR type LAYL at command line.
- ii) click new layer button



Q - Name various methods of defining a line?

- A line is specified by giving its two endpoints. The line command can be used to draw a single line or a series of lines with end point of one being the start point of the next.

methods of defining a line:

I. Absolute coordinate method: In absolute coordinate method the point is located with respect to origin (0,0)

The syntax is X, Y .

II. command: LIMITS (\downarrow)
rect mode space limits

3. command: ZOOM (\downarrow)
specify corner of window, enter a scale factor

3. command: LINE (\downarrow)
specify first point: (X, Y)

II. Relative coordinate method: In the relative coordinate method the displacement along the X and Y axis are measured with reference to previous point.
The syntax is $@X, Y$.

III. Relative polar coordinate method:

In relative polar coordinate method a point is located by defining the direct distance of the point from current point + angle that direct distance makes with the X-axis in counter clockwise direction

The syntax is $@ direct distance < angle$

first angle
Proj.
Third angle
Proj.

Q - Difference between 1st angle proj. & 3rd angle projection

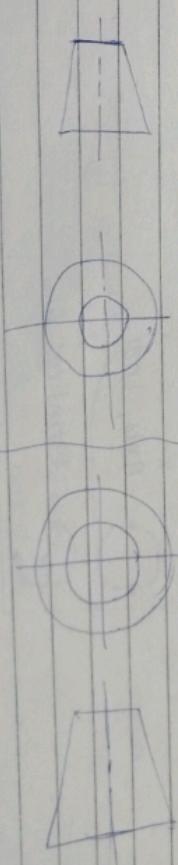
• The object is kept in 1st quadrant
- object

The object is assumed to be
kept in 3rd quadrant.

• Object lies between the other
object and plane of Proj.
The plane of projection lies
b/w the object + object.

• Proj. is assumed to be
non transparent.

The plane of projection is
assumed to be transparent.



Q - Purpose of ellipse using four different methods in AutoCAD.

The Ellipse is defined as a curve surrounding two focal point.

The Ellipse is general command to make ellipse in the AutoCAD display we can create the ellipse by specifying the two point on an axis and the endpoint on another axis.

The icon of ellipse will look like :



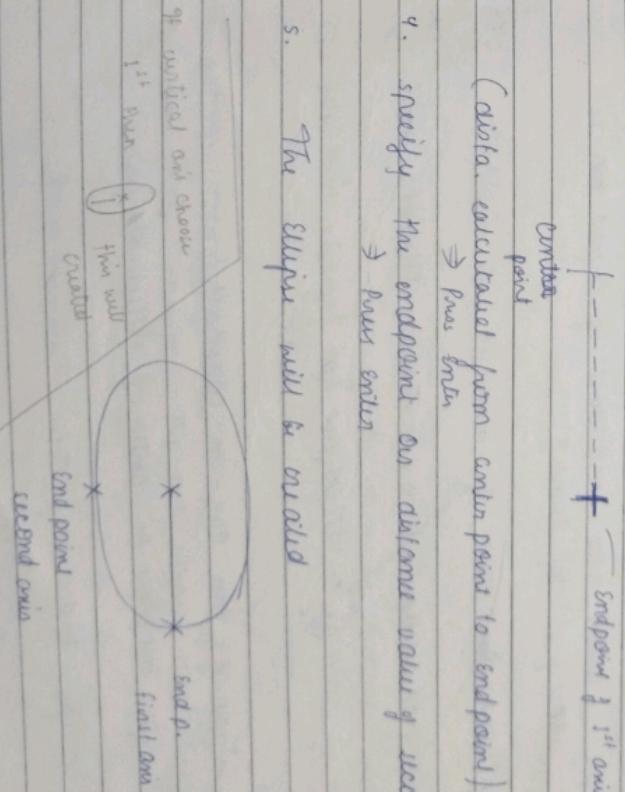
This icon consist of the drop-down list which includes other option such as

i) Center

- ii) Axis, End
 - iii) Elliptical Arc.
- (dia, calculated from center point to endpoint)
→ Press Enter

4. specify the endpoint or distance value of second axis
→ Press Enter

5. The ellipse will be created



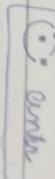
- o **Center:** In this method, the first axis is determined by the center point and endpoint while the second axis is determined by the length.

1. click on ellipse icon

or

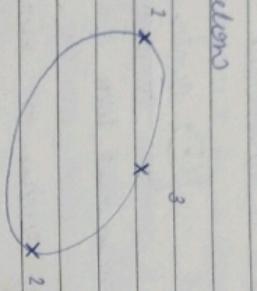
Type ellipse on command line

2. specify the center of ellipse on workspace.



- Axis, End : The ellipse is formed by defining the three points similar to above.

It is shown below

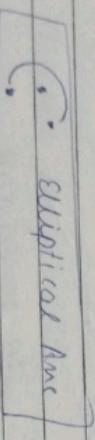


view

The location and length of first axis is specified by the points 1 and 2, while the distance b/w the center and end point of second axis is specified by point 3.

- Elliptical Arc :

1. click on ellipse icon & choose Elliptical arc



- Q - Explain the purpose of zoom command.
- zoom command ; increases or decreases the magnification of the view in current window you can change the magnification of view by zooming in and out.

or

Type of zoom as on command line

1. specify end point on center of elliptical arc.

when you zoom, the absolute size of things in design does not change . The only thing that changes is enlargement of the view.

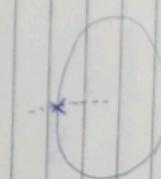
2. specify end point of axis
3. specify endpoint of axis

- a. specify endpoints of other axis

- b. specify start angle →

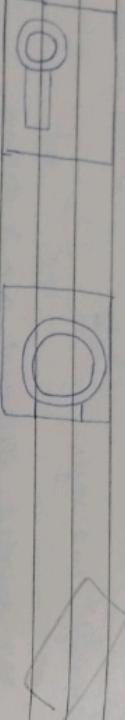


- c. specify end angle.

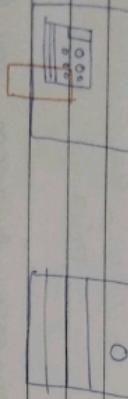


arc

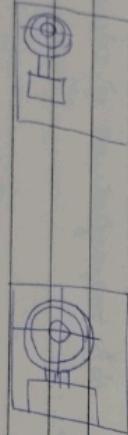
- **center**: zooms to show a view determined by a center point and a magnification or height setting



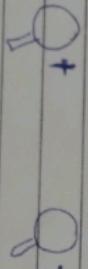
- **Dynamic**: A rectangular window box is used to pan and zoom. You click on the window box which you may expand.



- **Extant**: zooms in on all items to show their full extent.

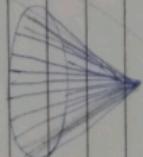


In real time: zoom in and out to change the magnification of image.



Box/ cone/ dish/ box/ Hull/ Pyramid/ Sphere/ Torus/ wedge:
Base center point : 3, 5, 0

Top



Diameter/ \langle radius \rangle of box : 3
Diameter/ \langle radius \rangle of top : 0
height : 6
No. of segments : 36 >

- P - explain the method of drawing wireframe mode of the following objects
- i) cone
 - ii) pyramid
 - iii) Prism

Line wireframe modelling is the process of visual presentation of 3-D on physical object used in 2-D computer graphics (a technique for representing 3D objects in which all surfaces are widely outlined in lines, including the opaque sides and all internal components that are normally hidden from view).

cone A

A surface wireframe model of cone can be drawn using 3D command. The prompt sequence will be:

ii) Pyramid : wireframe mode of pyramid can be drawn using 3D command.

The visual style drop down list located upo left side of drawing, and selecting wireframe option.

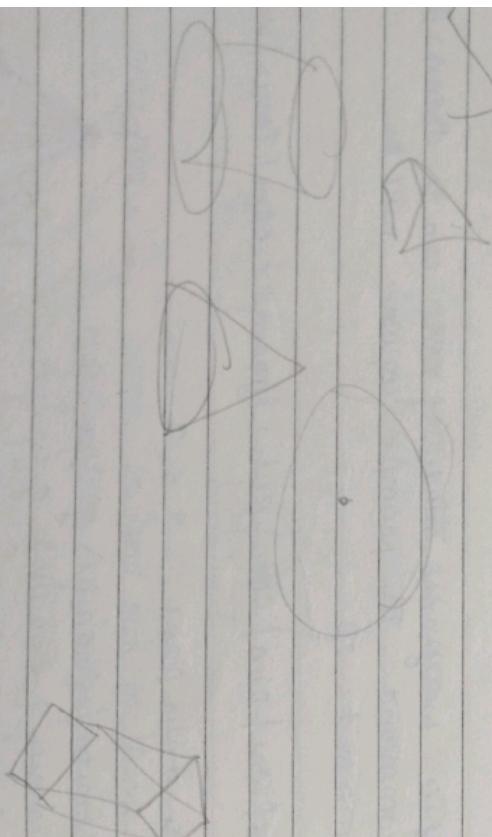
command : pyramid,

specify side circumference

specify base radius on [edge/size] : 3,3

specify base radius on [inscribed] <3.000>: 2

specify height : 4



Q. What are 5 editing commands in CAD?

i. The fundamental commands to edit a drawing are :

→ move

→ copy

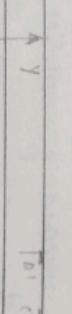
→ mirror

→ offset

→ array

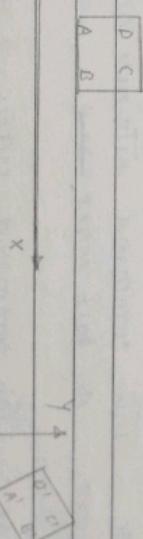
→ move

→ rotate



10' 0"

0' 0"



45°

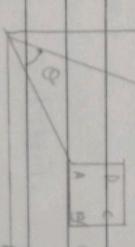
copy

original

A B

3. ROTATE : rotates selected object

through a specified angle
about a base point



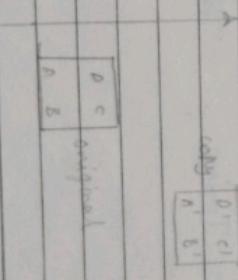
45°

copy

original

A B

3. COPY : creates one or more copies of selected object
at another location.



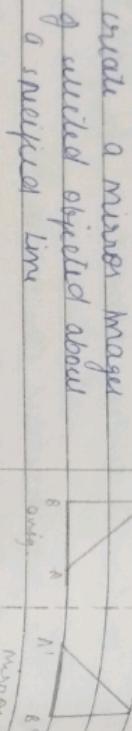
45°

copy

original

A B

4. Mirror : Create a mirror image of selected object about a specified line



- i) Tool bar
- ii) Edit commands
- iii) 2D and 3D modul.

Q - Define the following:

5. Array : This command calls multiple copies of selected object in rectangular form.

6. Erase : This command erases all the selected entities. A record of entities erased is always maintained.

(unseen)

7. OOPS : This command retrieves all objects erased by the last erase.

8. Break : This command erases a portion of line arc, circle or 2D blob into selected points.

9. Stretch : The stretch command can either lengthen or shorten them, thus alter their shape.

10. Trim : This command trims the object that extend beyond required point of interaction.

Line

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1. Extension line : An extension line is also a thin continuous line drawn in extension of an outline. It extends by about 3 mm beyond the dimension line.

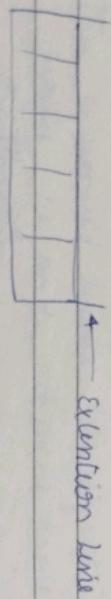
2. construction lines : These are continuous thin lines.

They are drawn for contributing figures. They are shown in geometrical drawing only.

3. leader lines : A leader on a pointer is a thin continuous line connecting a note or dimension figure with the feature to which it applies.

One end of leader terminates either in an arrowhead or a dot.

The leader line is never drawn vertical or horizontal or curved. It is drawn at a convenient angle of not less than 30° to the line to which it touches.



Q - Various components of AutoCAD program Windows

- i) Application menu.
- ii) Quick access Toolbar

iii) Infozoom

iv) Drawing Area

v) Drawing Tabs

vi) UCS icon

vii) View port controls

viii) Navigation bar

ix) Status bar

x) Command window