

January : 2016 (CBCS)

Note :

Max. marks : 60

- (i) Attempt any five questions.
- (ii) All questions carry equal marks.
- (iii) Assume suitable data or dimensions, if necessary, clearly mentioned it.

Q.1 (a) What is dimensioning? List the various dimensioning systems.

Ans. For a drawing component in addition to providing complete shape description, it must also furnish information regarding the size description.

It may provide through the distance between the,

- (i) Surface
- (ii) Location of holes.
- (iii) Nature of surface finish.
- (iv) Type of material etc.

1. General principle of dimensioning :

- (i) As far as possible, it should be placed outside the view.
- (ii) It should be taken from visible line rather than hidden lines.
- (iii) Dimensioning of a centre line should be avoided except when the centre line passes through the centre of a holes.
- (iv) Dimension should be placed on the view or section which is most clear to the corresponding features.
- (v) Each dimension should be dimensioned once on a drawing.
- (vi) Each drawing should have the same dimensional unit.
- (vii) More than one dimension should not be used for features of same parts.

2. Method of execution :

Element of dimensioning includes :

- (i) Projection line
- (ii) Leader line
- (iii) Dimension line termination
- (iv) Dimension line
- (v) Origin indication
- (vi) Dimension itself.

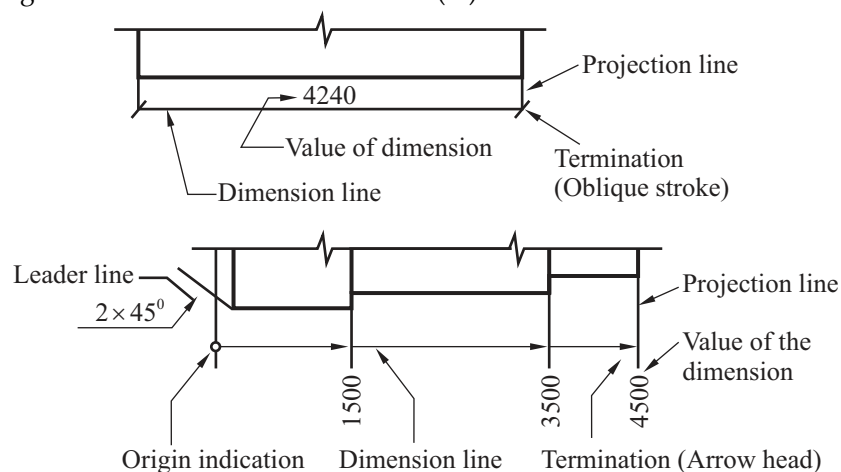


Fig. Elements of dimensioning.

Important instructions :

- (a) Projection and dimension lines should be drawn as thin continuous line.
- (b) Projection line should be extended slightly beyond the respective dimension lines.

- (c) Projection line should be drawn perpendicular to the part being dimensioned. (May be drawn obliquely, but parallel to each other and must be in contact with that part).
- (d) Dimension line and projection line should not cross each other (unless it is unavoidable)
- (e) Dimension line should not be shown broken, even if the part to which it refers is shown broken (shown in diagram).
- (f) A centre line or a outline of the part should not be used as a dimension line, but may be used in place of projection line.

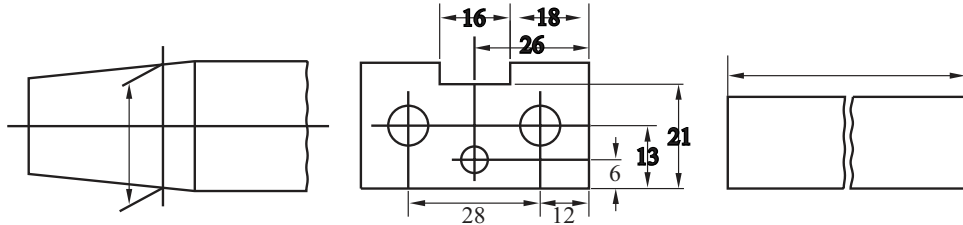


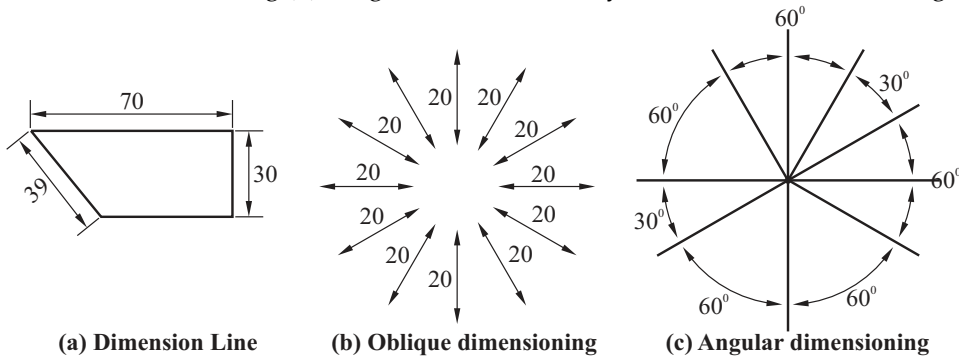
Fig. Method of dimensioning

3. **Method of indication of dimensions :** Dimension should be placed in such a way that they are not crossed or separated by any other line and should be shown on drawing in character of sufficient size, to ensure complete legibility.

Dimension should be indicated on drawing according to one of the following method (but only one method should be used on any drawing)

Method I : Aligned system,

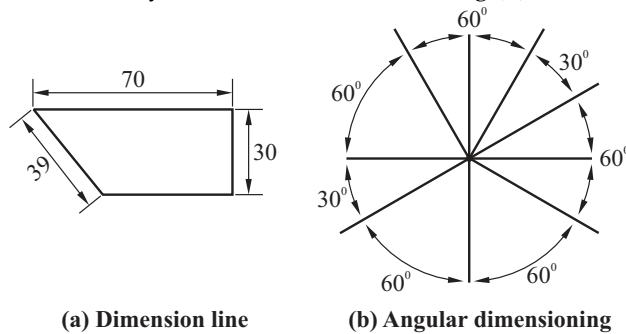
Dimension should be placed parallel to their dimension lines and preferably near the middle above and clear of the dimension lines (fig (a)). It may be written so that they can be read from the bottom or from the right side of the drawing. Dimension on oblique dimension lines should be oriented as shown fig.(b). Angular dimension may be oriented as shown in fig.(c).



Method II : Uni-directional system,

Dimension should be indicated so that they can be read from the bottom of drawing only. Non-horizontal dimensions lines are interrupted, preferably near the middle, for insertion of dimension (fig. (a))

Angular dimension may be oriented as shown in fig.(b)



Following indications (symbols) are used with dimension to reveal the shape indication and to improve drawing interpretation.

- (i) ϕ : Diameter
- (ii) $S\phi$: Spherical diameter
- (iii) R : Radius
- (iv) SR : Spherical Radius
- (v) \square : Square

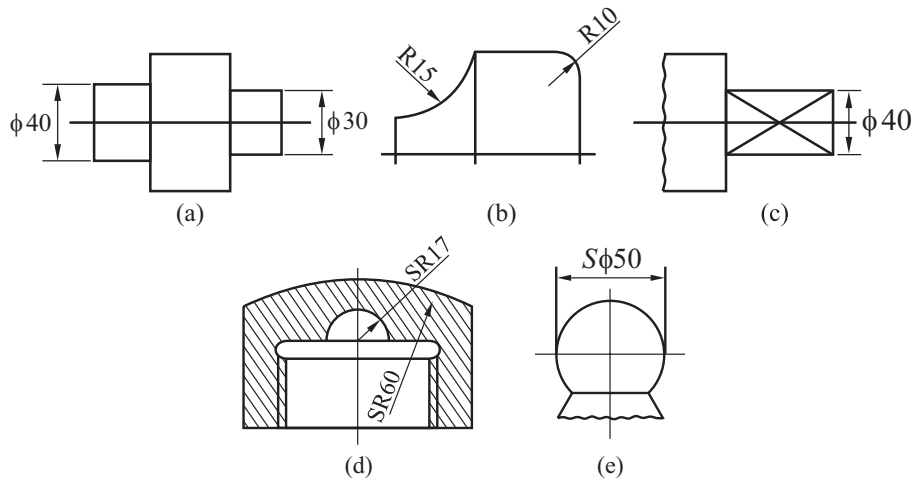


Fig. Shape identification symbols

4. **Arrangement of dimensions :** On a drawing arrangement of dimensions must indicate clearly the design purpose.

Way of arranging the dimensions :

- (i) **Chain dimension :** Chains of single dimensions should be used only where the possible accumulation of tolerance does not endanger the functional requirement of the part.

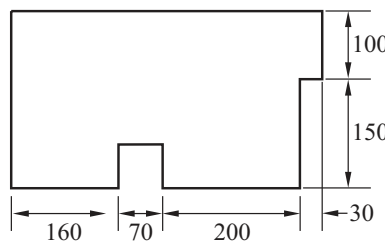


Fig. Chain dimensioning

- (ii) **Parallel dimensioning :** Number of dimensions lines parallel to one another and spaced out are used where a number of dimensions have a common datum features. [fig.(a)]

- (iii) **Super imposed running dimensions :** This types of simplified parallel dimensions may be used where there are space limitation [fig.(b)].

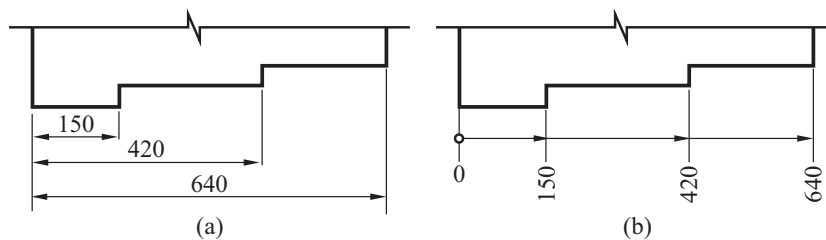
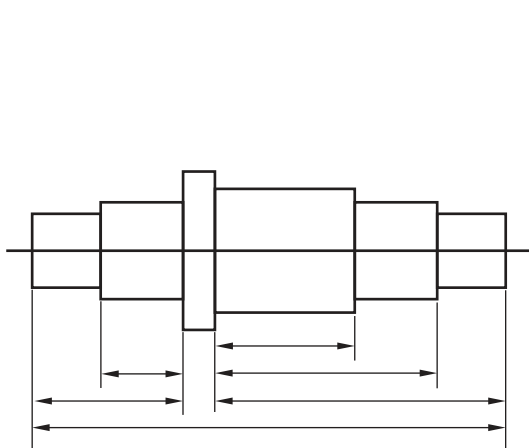


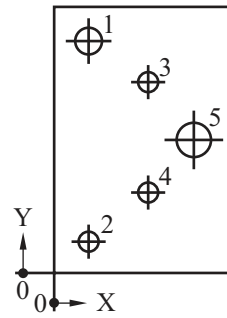
Fig. Parallel dimensioning and super imposed running dimension

(iv) **Combined dimensions** : It is simultaneous use of chain and parallel dimensions [fig.(c)]

(v) **Coordinate dimensions** : Size of the holes and their coordinate may be indicated directly on the drawing, or they may be conveniently presented in a tabular form as shown in fig.(d).



(c) Combined dimensioning



	X	Y	ϕ
1	20	160	15.5
2	20	20	13.5
3	60	120	11
4	60	60	13.5
5	100	90	26
6			
7			
8			
9			
10			

(d) Co-ordinate dimensioning

5. **Special indications of :**

- (i) Diameter
- (ii) Chords, angles, radii, arcs etc.
- (iii) Equi-distance features

All of above are shown by the means of diagram :

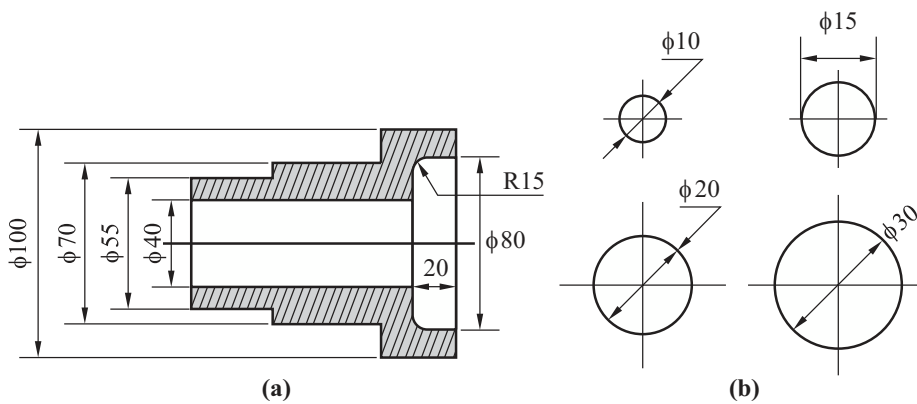


Fig. Dimensioning of diameters

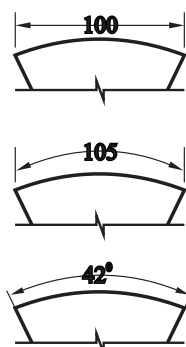


Fig. Dimensioning of chords, arcs and angles

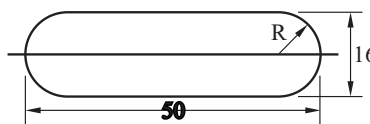


Fig. Dimensioning of radius

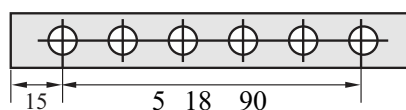
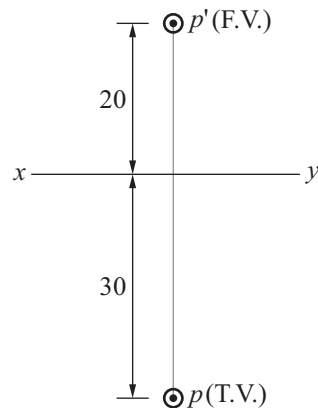


Fig. Dimensioning equidistant features

Q.1 (b) A point P is 20 mm above H.P. and 30 mm in front of V.P. Draw its projection.

Ans.

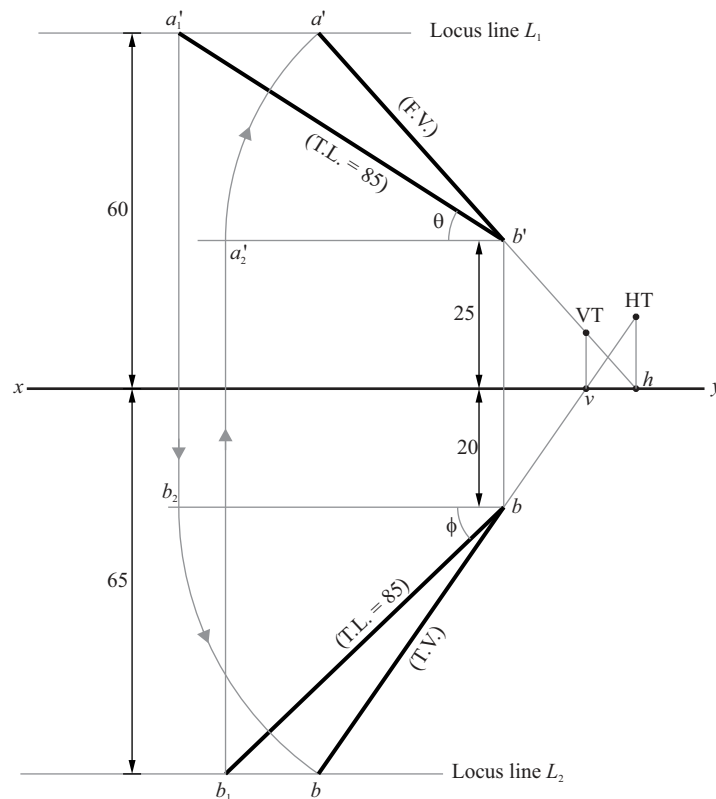


Q.1 (c) A line AB 85 mm long has its end A 60 mm above H.P. and 65 mm in front of V.P. The end B is 25 mm above H.P. and 20 mm in front of V.P. Draw the projections and find its inclinations with H.P. and V.P. Also mark its traces.

Ans. $AB = 85$ mm

End A : 65 mm above H.P. and 65 mm in front of V.P.

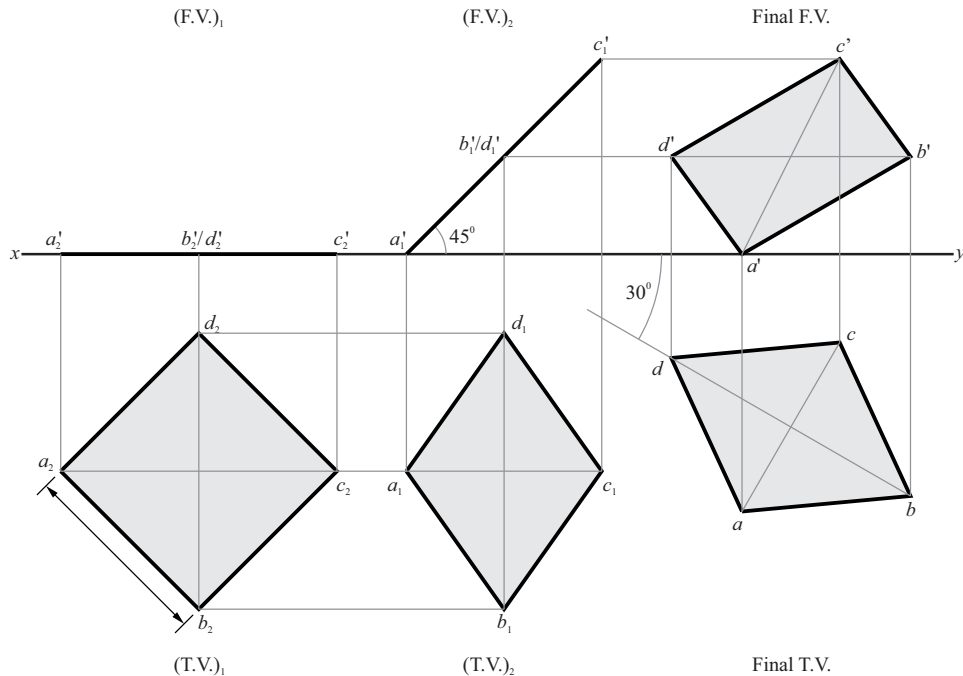
B : 25 mm above and 20 mm in front of V.P.



Steps of construction are same as previous problem.

Q.2 (a) A square plate $ABCD$ of side 30 mm is resting on H.P. on one of its corners and the diagonal AC inclined at 45° to H.P. The diagonal BD of the plate is inclined at 30° to V.P. and parallel to H.P. Draw its projections.

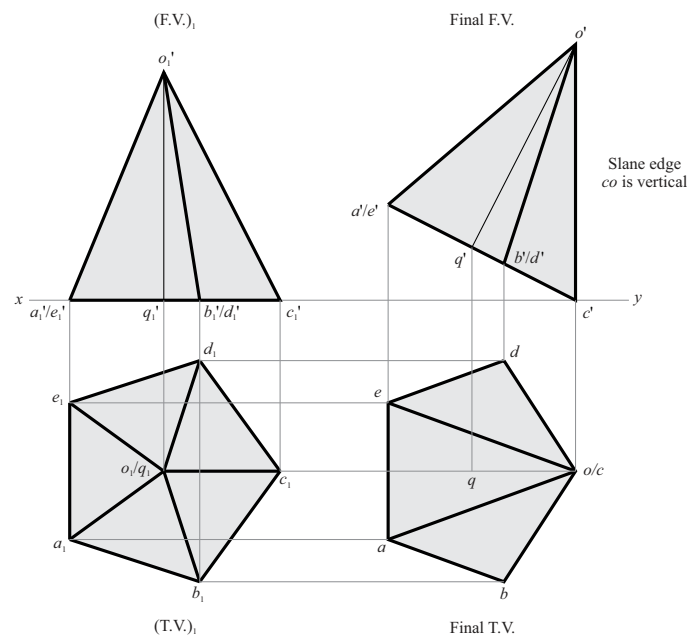
Ans.



Steps of construction is same as discussed earlier.

Q.2 (b) A pentagonal pyramid of base side 30 mm and axis length 50 mm is resting on H.P. on one of its base corner with its axis parallel to V.P. Draw its projections when the slant edge containing the resting corner is vertical.

Ans.



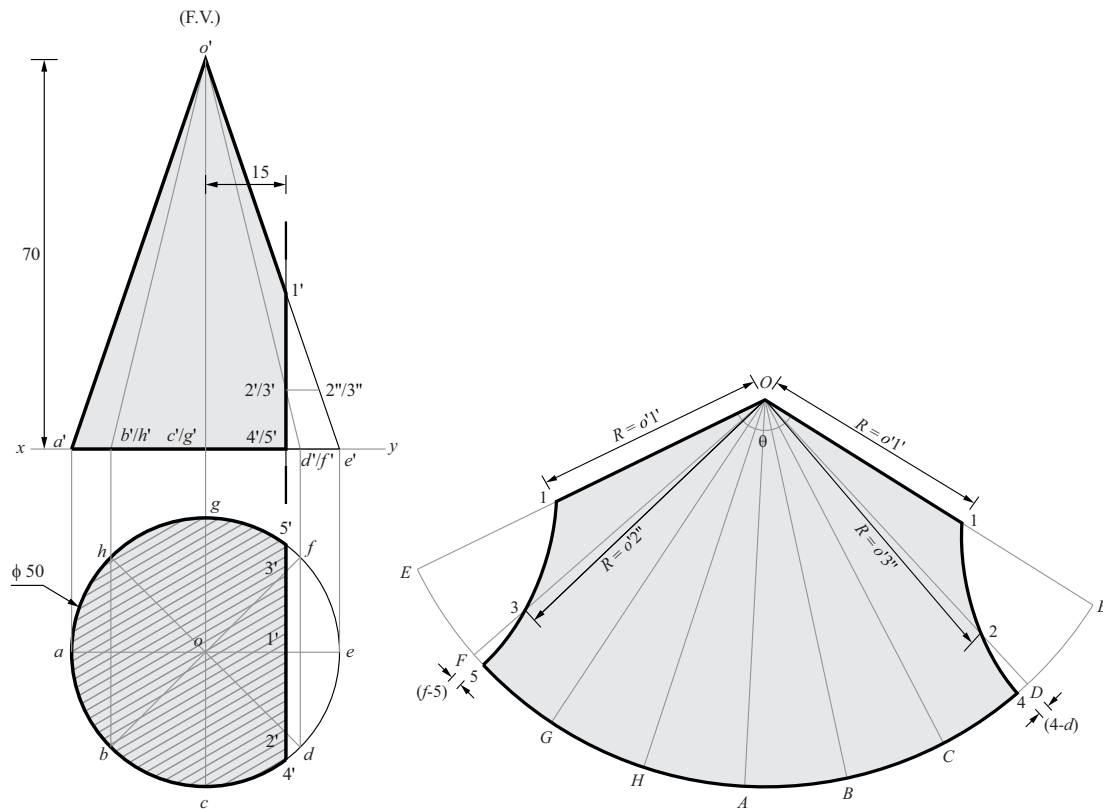
Steps :

- (i) Assuming axis of pentagonal pyramid is perpendicular to H.P. and resting on H.P. (with the help of base) draw $(T.V.)_1$ and $(F.V.)_1$ such that one base side (say $e_1 a_1$) perpendicular to V.P.

- (ii) Position the (F.V.)₁ such that slant edge of corner *c* or slant edge *co* perpendicular or vertical to *xy* (or H.P.). This is the final F.V.
- (iii) With the help of final F.V. and (T.V.)₁, draw its final T.V.

Q.3 (a) A cone of base diameter 50 mm and height 70 mm is resting on its base on H.P. It is cut by a plane perpendicular to both H.P. and V.P. at a distance of 15 mm to the right of the axis. Draw the development of the lateral surface of the left remaining portion.

Ans.



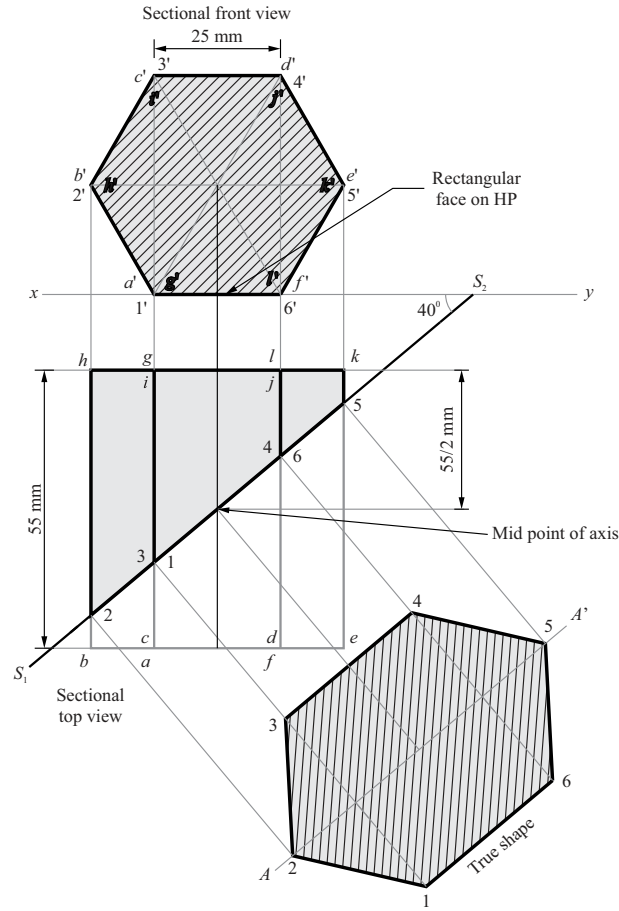
Steps :

Steps of construction are same as previous question. (Diagram explains its steps of construction)

Q.3 (b) A hexagonal prism of base side 25 mm and axis 55 mm long is resting on H.P. on one of its rectangular faces, with its axis perpendicular to V.P. It is cut by a plane inclined at 40° to V.P. and perpendicular to H.P. and is bisecting the axis of the prism. Draw its top view, sectional front view and true shape of the section.

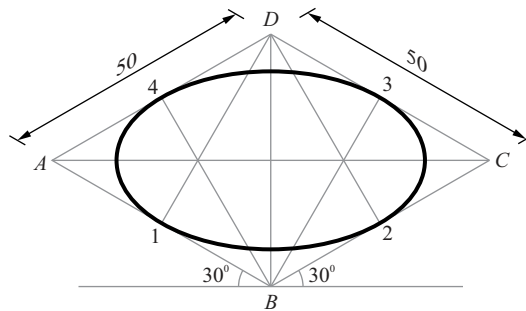
Ans. Steps of construction :

- (i) Front view is drawn first, as the hexagonal prism is resting on one of its rectangular faces in H.P.
Front view : Hexagon of side 25 mm.
- (ii) Top view is drawn by projecting all the points from front view; vertically downwards.
Top view : Hexagonal prism of base length equal to 'hk' and axis 55 mm long.
- (iii) A section plane *S₁S₂* is drawn inclined at 40° to the V.P., bisecting the axis into two halves in the top view.
- (iv) Sectional front view is drawn by projecting all the points from sectional top view.
- (v) True shape is drawn with the help of auxiliary plane *AA'* and by projecting all the points from sectional top view.



Q.4 (a) Draw the isometric view of a circular lamina of diameter 50 mm placed with its surface parallel to H.P.

Ans.



Its isometric view is an ellipse, draw ellipse by four centre method as shown in the diagram.

Q.4 (b) A tripod stand rests on H.P. One of its legs A.P. is 140 mm long and it makes an angle of 75° with H.P. The other two legs BQ and CR are 150 mm and 160 mm long respectively. The upper ends of all the three legs A, B and C are attached to horizontal circular plate of 60 mm diameter. The attachment is done to the circumference of the circle dividing it equally. In plan three legs appears as radial lines 40° apart with plan of the legs appearing perpendicular to V.P. Draw the projections of the complete tripod stand and find angle of other two legs with H.P. Also find the height of the horizontal circular plate from H.P.

Ans. AP = 140 mm
BQ = 150 mm
CR = 160 mm

Here,

$$pa = PA_1$$

$$cr = cr_1$$

$$bq = bq_1$$

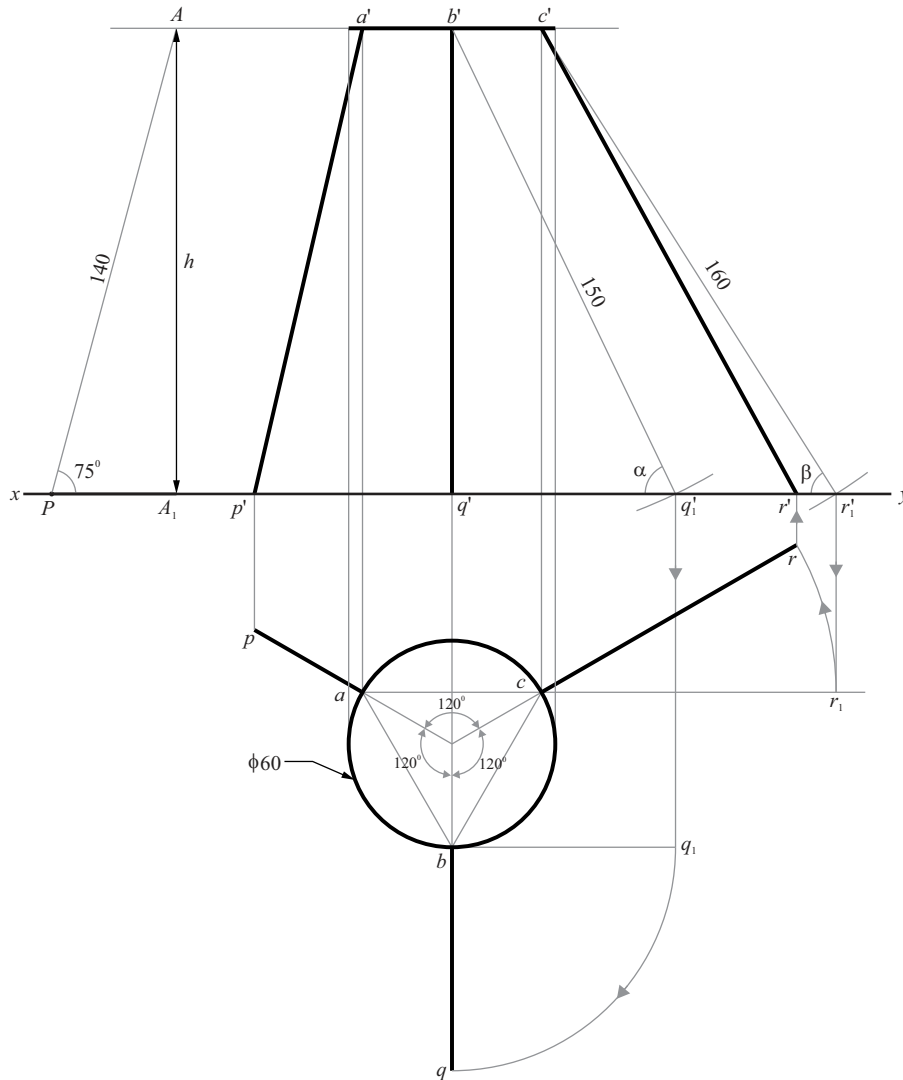
By measurement

$$h = 136 \text{ mm}$$

$$\alpha = 66^\circ$$

$$\beta = 58^\circ$$

Ans.



Q.5 (a) Explain the Cartesian and Polar coordinate systems of locating a point in the AutoCAD.

Ans. Absolute coordinate method : In the absolute coordinate method, the points are located with respect to the origin (0,0). The syntax is x, y .

Example : Draw a line diagram using absolute coordinate method.

To draw a line diagram using absolute coordinate method, follow the steps mentioned below:

1. Command : Limits (↵)

Reset model space limits :

Specify lower left corner or [ON/OFF] < 0.0000,0.0000 > : (↵)

Specify upper right corner < 12.0000, 9.0000 > : 120, 90 (↵)

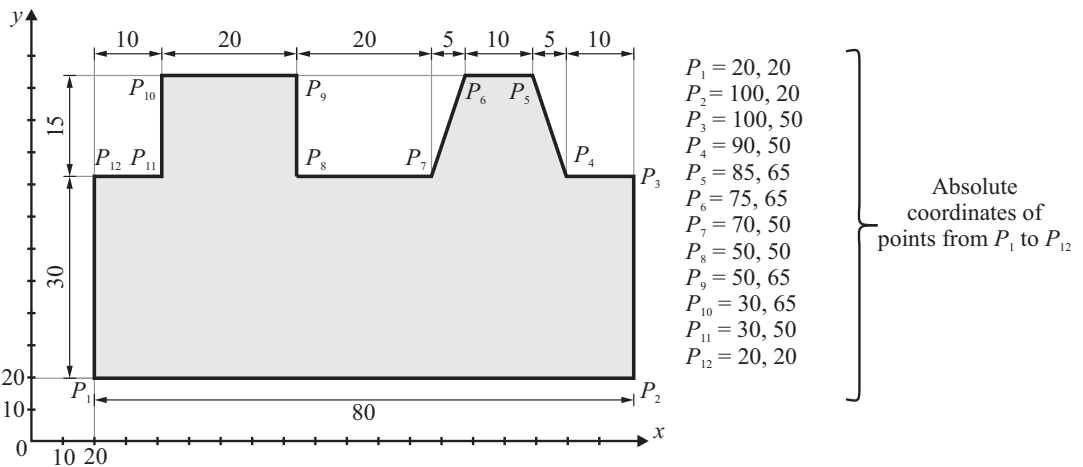
2. **Command :** Zoom (↵)

Specify corner of window, enter a scale factor (nx or npx) or [All | Center | Dynamic | Extents | Previous | Scale | Window] < real time > : ALL (↵)
Regenerating model.

3. **Command :** Line (↵)

Specify first point : 20, 20 (↵)
Specify next point : [Undo] : 100, 20 (↵)
Specify next point or [Undo] : 100, 50 (↵)
Specify next point or [Close/Undo] : 90, 50 (↵)
Specify next point or [Close/Undo] : 85, 65 (↵)
Specify next point or [Close/Undo] : 75, 65 (↵)
Specify next point or [Close/Undo] : 70, 50 (↵)
Specify next point or [Close/Undo] : 50, 50 (↵)
Specify next point or [Close/Undo] : 50, 65 (↵)
Specify next point or [Close/Undo] : 30, 65 (↵)
Specify next point or [Close/Undo] : 30, 50 (↵)
Specify next point or [Close/Undo] : 20, 50 (↵)
Specify next point or [Close/Undo] : C (↵)

4. Save the file as Module 1.DWG.



Relative coordinate method : In the relative coordinate method, the displacement along x and y axis are measured with reference to the previous point.

The syntax is @ x, y.

Example : Draw a line diagram using relative coordinate method.

To draw a line diagram using relative coordinate method, following steps are mentioned below :

1. **Command :** Limit (↵)

Reset model space limits :
Specify lower left corner or [ON/OFF] < 0.0000, 0.0000 > : (↵)
Specify upper right corner < 12.0000, 9.0000 > : 120, 90 (↵)

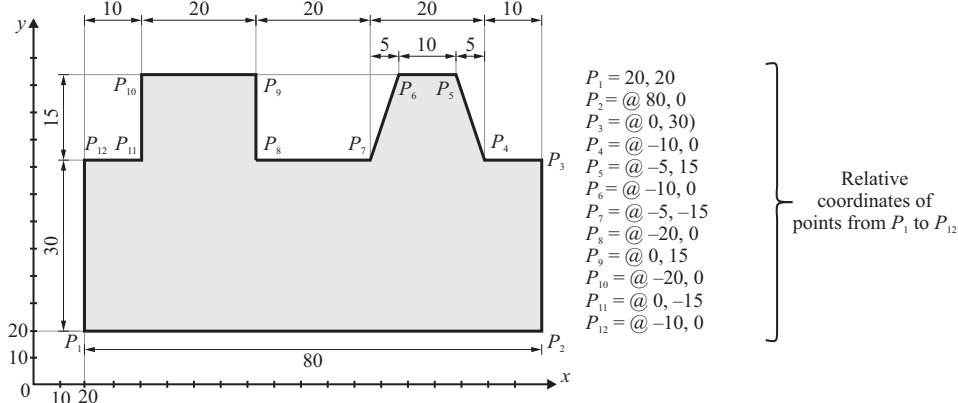
2. **Command :** Zoom (↵)

Specify corner of window, enter a scale factor (nx or npx) or [All/Center/Dynamics/Extents/Previous/Window/Scale] < real time > : ALL (↵)
Regenerating model.

3. Command : Line (↵)

Specify first point : 20, 20 (↵)
 Specify next point or [Undo] : @ 80, 0 (↵)
 Specify next point or [Undo] : @ 0, 30 (↵)
 Specify next point or [Undo] : @ -10, 0 (↵)
 Specify next point or [Close/Undo] @ -5, 15 (↵)
 Specify next point or [Close/Undo] @ -10, 0 (↵)
 Specify next point or [Close/Undo] @ -5, -15 (↵)
 Specify next point or [Close/Undo] @ -20, 0 (↵)
 Specify next point or [Close/Undo] @ 0, 15 (↵)
 Specify next point or [Close/Undo] @ -20, 0 (↵)
 Specify next point or [Close/Undo] @ 0, -15 (↵)
 Specify next point or [Close/Undo] @ -10, 0 (↵)
 Specify next point or [Close/Undo] C (↵)

4. Save the file as Module-2.DWG



Polar coordinate method : In the Polar coordinate method a point is located by defining the direct distance of the point from current point and the angle that the direct distance makes with the positive 'x' axis in counter clock wise direction. It's syntax is @ direct distance < angle.

Example : Draw a line diagram using Polar coordinate method.

To draw a line diagram using Polar coordinate method, following steps are mentioned below:

1. Command : Limits (↵)

Reset model space limits :

Specify lower left corner or [ON/OFF] < 0.0000, 0.0000 > : (↵)

Specify upper right corner or < 12.0000, 9.0000 > : 120, 90 (↵)

2. Command : Zoom (↵)

Specify corner of window, enter a scale factor (nx or npx) or [All/Center/Dynamic/Extents/Previous/Scale/Window/ < real time > : ALL (↵)

3. Command : Line (↵)

Specify first point : 20, 20↵

Specify next point or [Undo] : @ 80 < 0 (↵)

Specify next point or [Undo] : @ 30 < 90 (↵)

Specify next point or [Close/Undo] : @ 10 < 180 (↵)

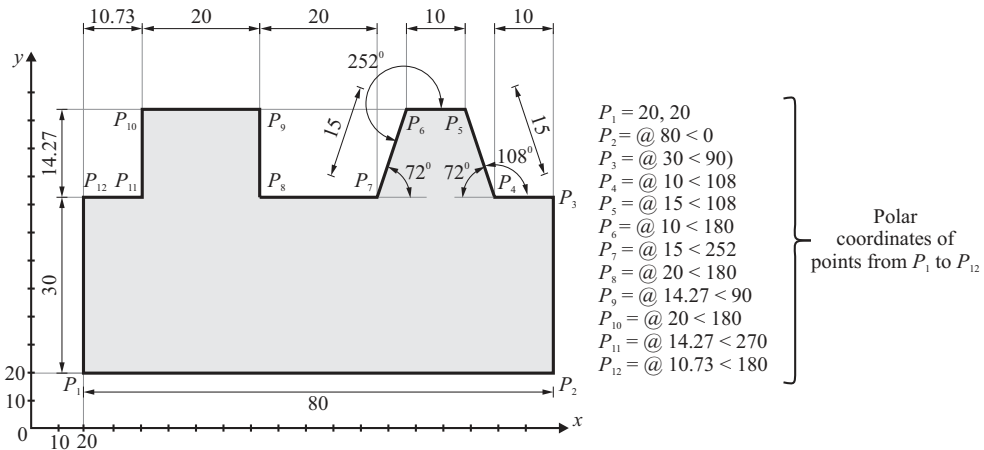
Specify next point or [Close/Undo] : @ 15 < 108 (↵)

Specify next point or [Close/Undo] : @ 10 < 180 (↵)

Specify next point or [Close/Undo] : @ 15 < 252 (↵)

- Specify next point or [Close/Undo] : @ 20 < 180 (↵)
- Specify next point or [Close/Undo] : @ 14.27 < 90 (↵)
- Specify next point or [Close/Undo] : @ 20 < 180 (↵)
- Specify next point or [Close/Undo] : @ 14.27 < 270 (↵)
- Specify next point or [Close/Undo] : @ 10.37 < 180 (↵)
- Specify next point or [Close/Undo] : C (↵)

4. Save the file as Module-3.DWG.



Q.5 (b) State a series of AutoCAD steps needed to draw an ellipse of given major and minor axes. How would you draw another ellipse parallel to it at a specified distance?

Ans. Steps required to draw an ellipse of given major and minor axes using AutoCAD are as follows :

- Command : ELLIPSE (↵)
- <Axis endpoint1> : 10, 40 (↵)
- <Axis endpoint2> : 60, 40 (↵)
- <Other axis distance>/Rotation : 35, 60 (↵)

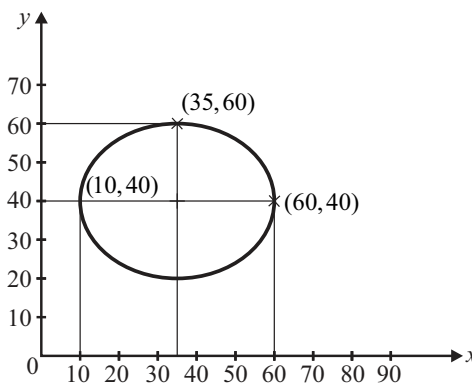


Fig. : After Ellipse command

Another ellipse may be drawn parallel to the above ellipse at a specified distance using 'OFFSET' command.

- Command : OFFSET (↵)
- Select object : Select the ellipse to offset with the help of mouse (↵)
- Specify base point : Select point (10, 40) with the help of mouse or enter 10, 40 (↵)
- Specify second point of displacement : 10, 90 (↵)

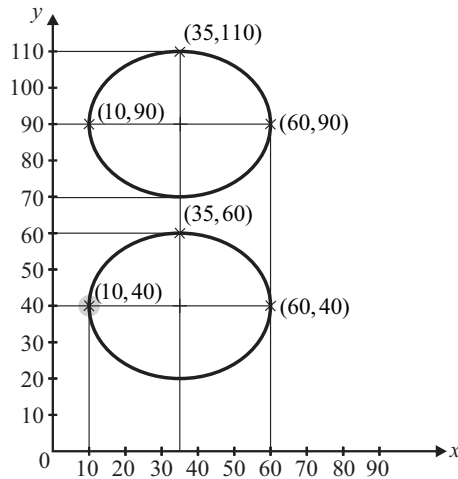


Fig. : After Offset command

Q.6 (a) Give the list of softwares, currently available for 2D solid modeling.

Ans. The various types of CAD softwares available in the market are :

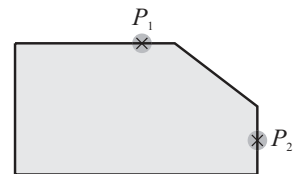
- AutoCAD
- Pro-E
- Catia
- Unigraphics
- ANSYS, etc.

Q.6 (b) Explain the use of fillet and chamfer tools in AutoCAD.

Ans. **The Chamfer Command :** The Chamfer command enables you to create a chamfer between any two non-parallel lines as in the illustration below or any two adjacent polyline segments. Usually, the Chamfer command is



Before Chamfer



After Chamfer

used to set the chamfer distances before drawing the chamfer. Follow the command sequence below where the chamfer distances are changed to 20 before the chamfer is made.

Command : CHAMFER

(TRIM mode) Current chamfer Dist1 = 10.0000, Dist2 = 10.0000

Select first line or [Polyline/Distance/Angle/Trim/Method] : D (to set distances)

Specify first chamfer distance <10.0000> : 20 (enter required distance)

Specify second chamfer distance <20.0000> : ↵ (first distance value or enter a different value)

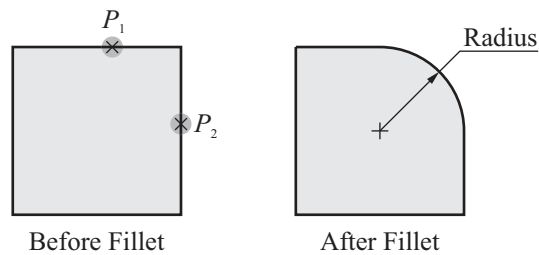
Select first line or [Polyline/Distance/Angle/Trim/Method] : (pick P1)

Select second line : (pick P2)

The chamfer is made and the command ends.

Notice from the command sequence that there are a number of options which can be used to control the way the Chamfer command behaves. The Polyline option can be used to chamfer all vertexes of a polyline simultaneously. The Distance option allows you to specify the two chamfer distances. Angle allows the angle between the first line and the chamfer to be specified. Trim is used to control whether the original lines are trimmed to the chamfer or remain as they are. Finally, Method is used to toggle the command between Distance and Angle mode. When Angle mode is used, the chamfer is defined using one distance and an angle rather than two distances.

Fillet Command : The Fillet command is a very useful tool which allows you to draw an arc between two intersecting lines or adjacent polyline segments. You first need to use the command to set the required radius and then a second time to select the two lines.



Command Sequence

Command : FILLET

Current settings : Mode = TRIM, Radius = 10.0000

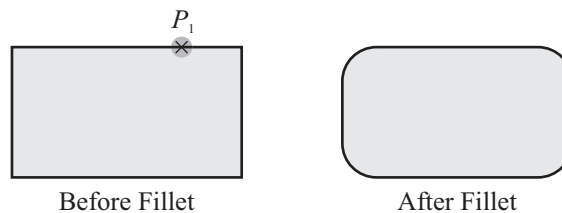
Select first object or [Polyline/Radius/Trim] : R

Specify fillet radius <10.000> : 25

Select first object or [Polyline/Radius/Trim] : (pick P1)

Select second object : (pick P2)

The Fillet command can also be used to fillet arcs and circles. The "Polyline" option also allows you to fillet all vertices of a polyline with a single command. It's worth experimenting with this command, it can save you lots of time and enables you to construct shapes which otherwise would be quite difficult.



For example, you can easily create the lozenge shape shown on the right from a simple rectangle. Since AutoCAD rectangles are just closed polylines, you can use the Polyline option of the Fillet command to fillet all polyline vertices simultaneously. Try this for yourself; draw a rectangle and then follow the command sequence below.

Command : FILLET

Current settings: Mode = TRIM, Radius = 10.0000

Select first object or [Polyline/Radius/Trim]: P

Select 2D polyline: (pick P1)

4 lines were filleted

Q.6 (c) Draw the projection of a cube of 30 mm edge resting on H.P. on one of its corner with solid diagonal vertical.

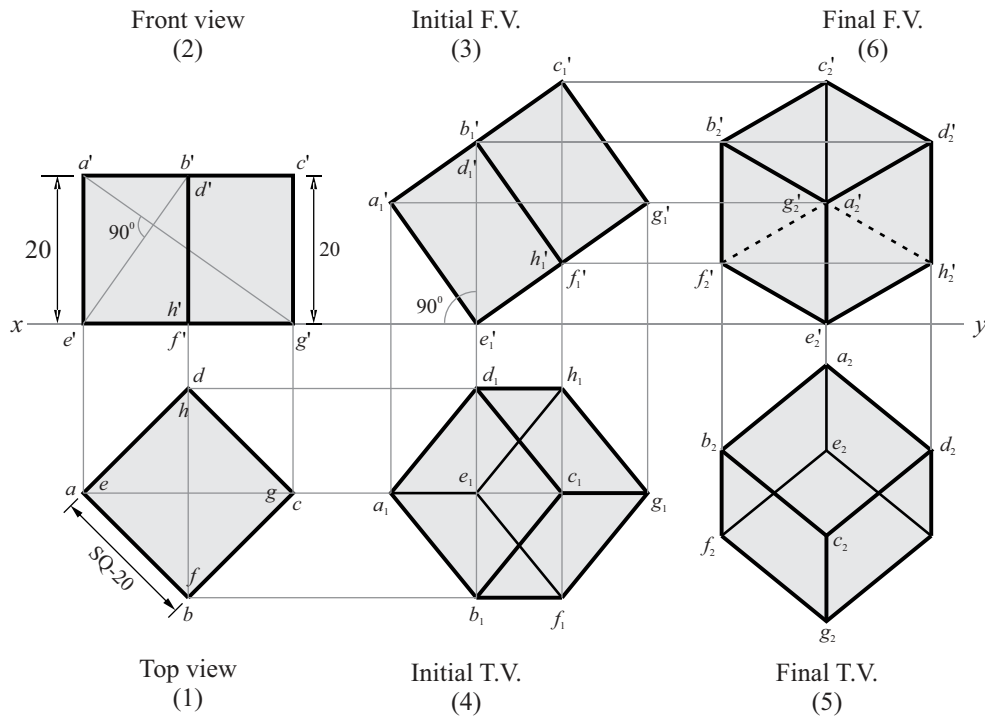
Ans. Steps :

(i) Initially assume that cube is resting on one of its face in H.P. such that its solid diagonal (ag or ec) is parallel to V.P.

Draw its top view (top view of a cube will be a square with all sides inclined 45° with xy). Project its front view.

(ii) Position the front view (inclined front view) such that its diagonal ag becomes parallel to H.P. (or xy line).

- (iii) Project its top view with the help of previous F.V.(3) and T.V.(1), as a initial top view(4).
- (iv) Position the initial top view(4) such that its solid diagonal $a_2 g_2$ perpendicular to V.P. This is final top view(5).
- (v) With the help of final top view(5) and initial front view(3), draw the final F.V.(6).

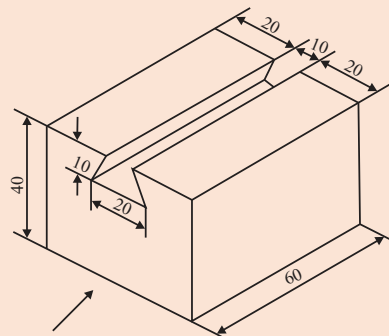


Q.7 (a) Differentiate between frustum of pyramid and truncated pyramid.

Ans. Frustum of a pyramid : When a horizontal section plane (plane perpendicular to V.P. and parallel to H.P.) cut the pyramid, which is resting on H.P. with its base, then frustum is obtained. Here all the slant sides are of equal length.

Truncated pyramid : When a section plane inclined to H.P. and perpendicular to V.P. cut the pyramid, which is resting on H.P. with its base, the truncated pyramid is obtained. Here all the slant sides are not of equal length.

Q.7 (b) Draw the front view, top view and right side view of the object shown in figure.



Ans. 1st angle projection :

