Unit- 1
Intersection of Solid

Whenever two or more solids combine, a definite curve is seen at their intersection. This curve is called the curve of intersection (COI).

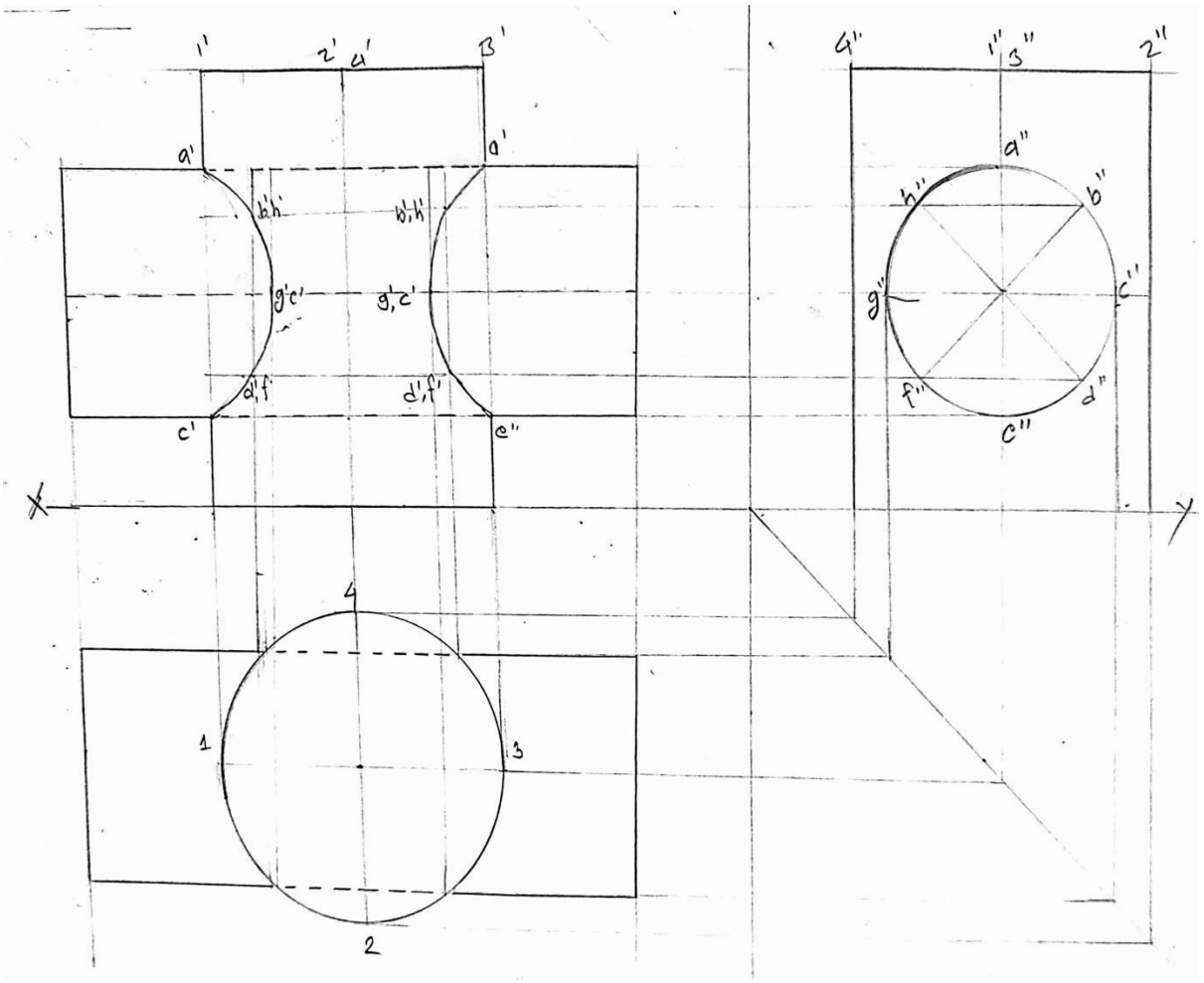
CASES OF INTERSECTION

The cases of intersection depend on the type of intersecting solids and the manner in which they intersect. Two intersecting solids may be of the same type (e.g., prism and prism) or of different types (e.g., prism and pyramid).

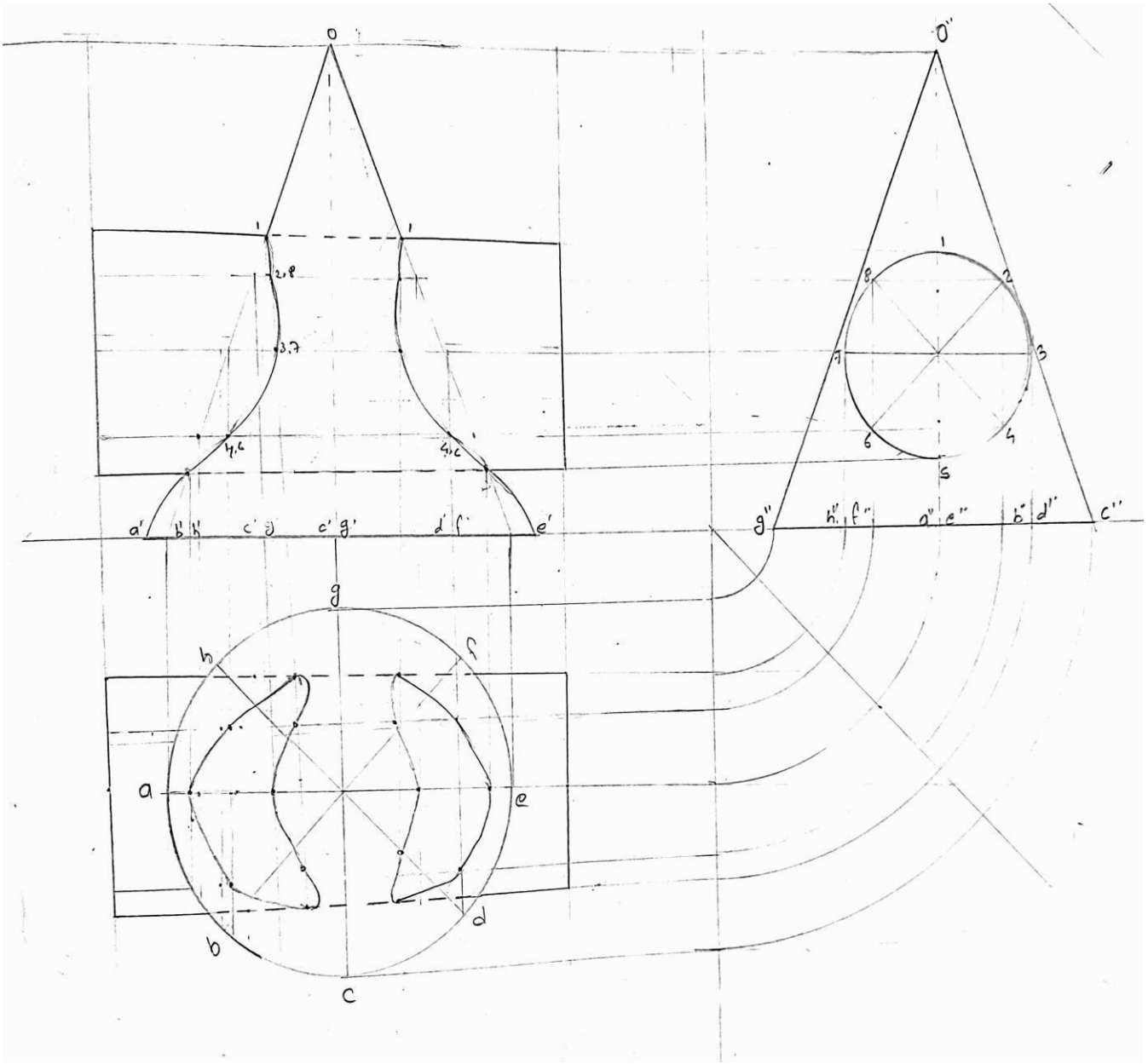
Intersection

The type of intersection created depends on the types of geometric forms, which can be two- or threedimensional. Intersections must be represented on multiview drawings correctly and clearly. For example, when a conical and a cylindrical shape intersect, the type of intersection that occurs depends on their sizes and on the angle of intersection relative to their axes. The line of intersection is determined using auxiliary views and cutting

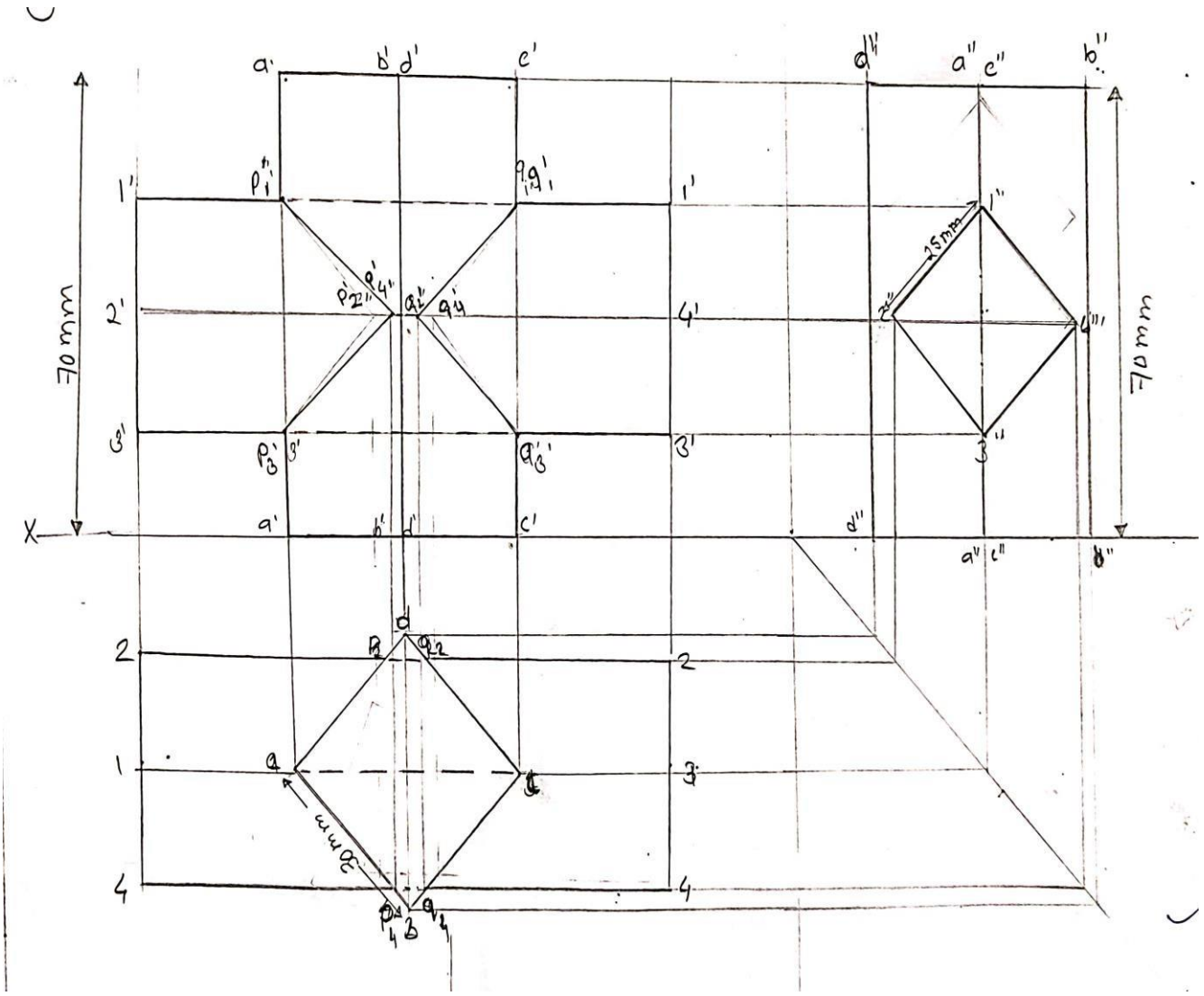
1. A cylinder 50mm dia. and 70mm axis is completely penetrated by another of 40 mm dia. and 70 mm axis horizontally Both axes intersect & bisect each other. Draw projections showing curves of intersections.



2. A vertical cone, base diameter 75 mm and axis 100 mm long, is completely penetrated by a cylinder of 45 mm diameter. The axis of the cylinder is parallel to Hp and Vp and intersects axis of the cone at a point 28 mm above the base. Draw projections showing curves of intersection.



Q.3 A sq. prism 30 mm base sides and 70mm axis is completely penetrated by another square prism of 25 mm sides and 70 mm axis, horizontally. Both axes Intersects & bisect each other. All faces of prisms are equally inclined to VP. Draw projections showing curves of intersections.



Rituraj Chandrakar

Unit - 2

"Development of surfaces"

In industrial world, an engineer is frequently confronted with problems where the development of surfaces of an object has to be made to help him to go ahead with the design and manufacturing processes. For example, in sheet metal work, it plays a vital role, thus enabling a mechanic to cut proper size of the plate from the development and then to fold at proper places to form the desired objects, namely, boilers, boxes, buckets, packing boxes, chimneys, hoppers, air-conditioning ducts etc.

"The development of surface of an object means the unrolling and unfolding of all surfaces of the object on a plane."

"If the surface of a solid is laid out on a plain surface, the shape thus obtained is called the development of that solid." In other words, the development of a solid is the shape of a plain sheet that by proper folding could be converted into the shape of the concerned solid.

Importance of Development:

Knowledge of development is very useful in sheet metal work, construction of storage vessels, chemical vessels, boilers, and chimneys. Such vessels are manufactured from plates that are cut according to these developments and then properly bend into desired shaped. The joints are then welded or riveted.

Principle of Development:

Every line on the development should show the true length of the corresponding line on the surface which is developed.

Methods of Development:

(a) Parallel-line development

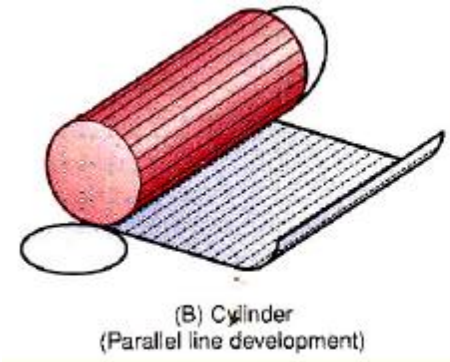
(b) Radial-line development

(c) Triangulation development

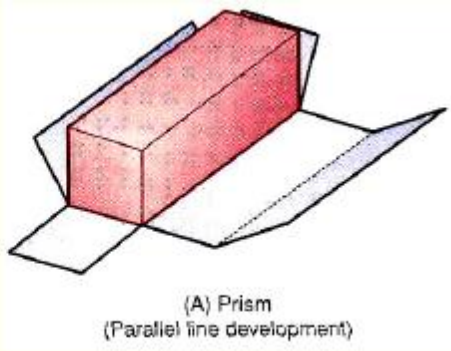
(d) Approximate development

Parallel-line Method:

It is used for developing prisms and single curved surfaces like cylinders, in which all the edges/generation of lateral surfaces are parallel in each other.



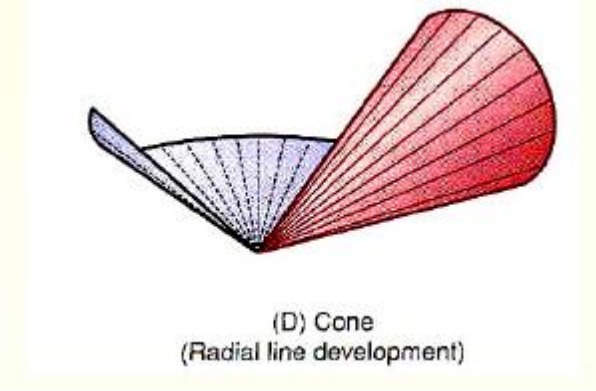
(B) Cylinder
(Parallel line development)



(A) Prism
(Parallel line development)

Radial-line Method:

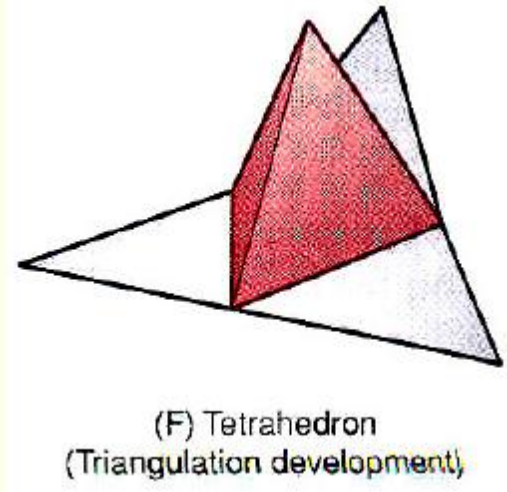
It is employed for pyramids and single curved surfaces like cones in which the apex is taken as centre and the slant edge or generator as radius of its development.



(D) Cone
(Radial line development)

Triangulation Method:

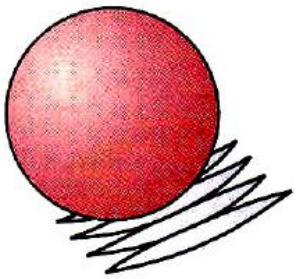
It is used for developing transition pieces.



(F) Tetrahedron
(Triangulation development)

Approximate Method:

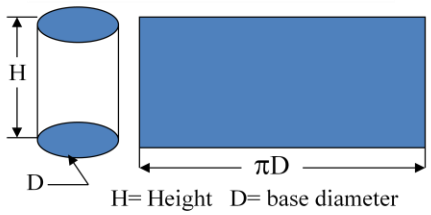
It is employed for double curved surfaces like spheres, as they are theoretically not possible to develop. The surface of the sphere is developed by approximate method. When the surface is cut by a series of cutting planes, the cut surfaces is called a zone.



(E) Sphere
(Approximate development)

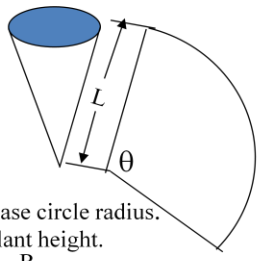
Development of lateral surfaces of different solids.
(Lateral surface is the surface excluding top & base)

Cylinder: A Rectangle



H= Height D= base diameter

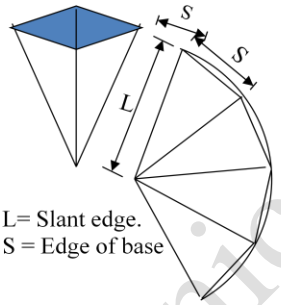
Cone: (Sector of circle)



R=Base circle radius.
L=Slant height.

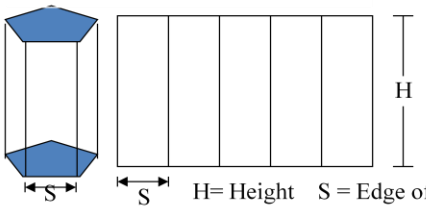
$$\theta = \frac{R}{L} \times 360^0$$

Pyramids: (No.of triangles)



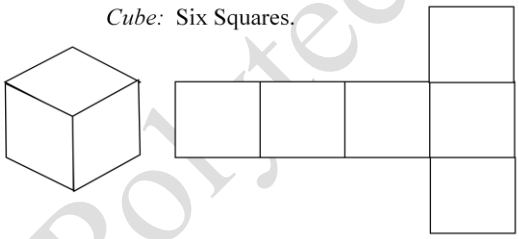
L= Slant edge.
S = Edge of base

Prisms: No.of Rectangles

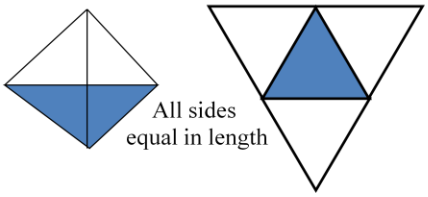


H= Height S = Edge of base

Cube: Six Squares.



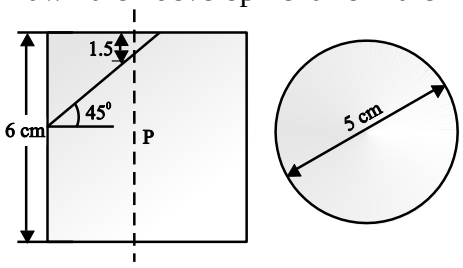
Tetrahedron: Four Equilateral Triangles



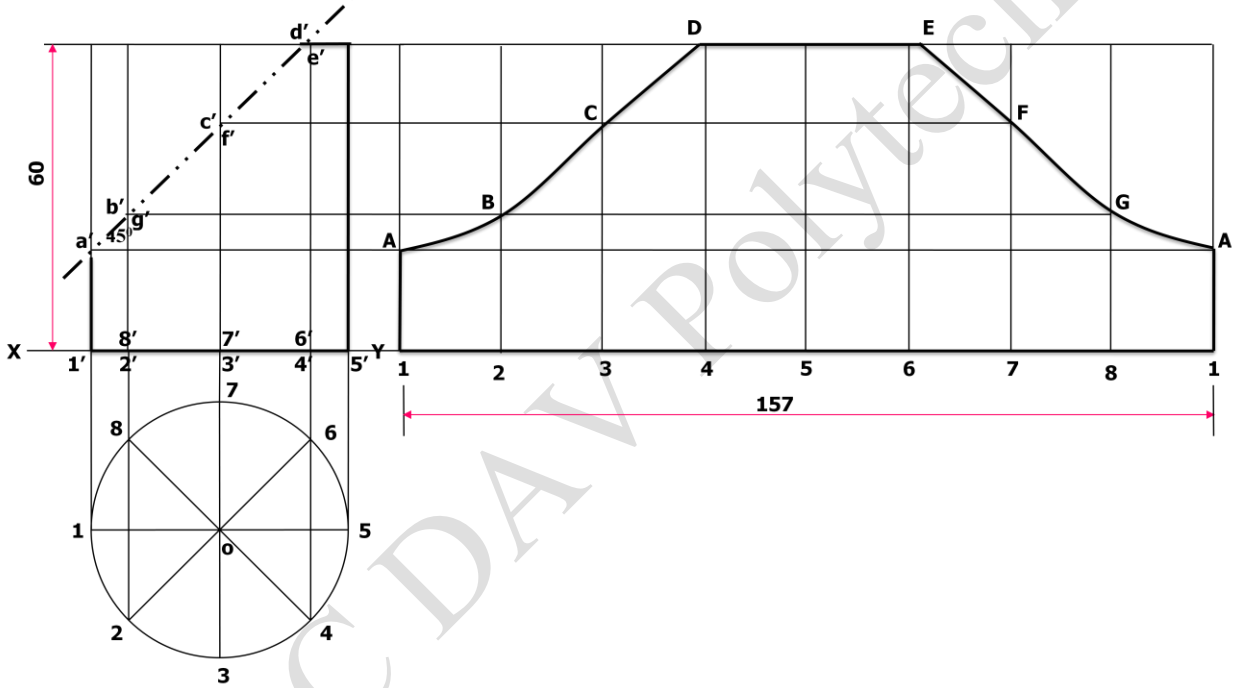
All sides
equal in length

Development of cylinders

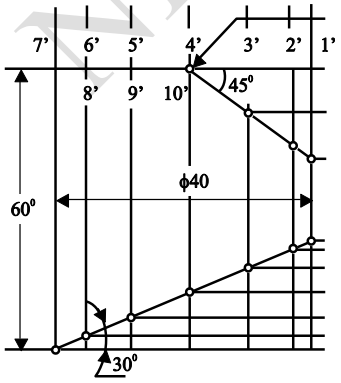
Q. Draw the development of the lateral surface P of the cylinder shown figure.



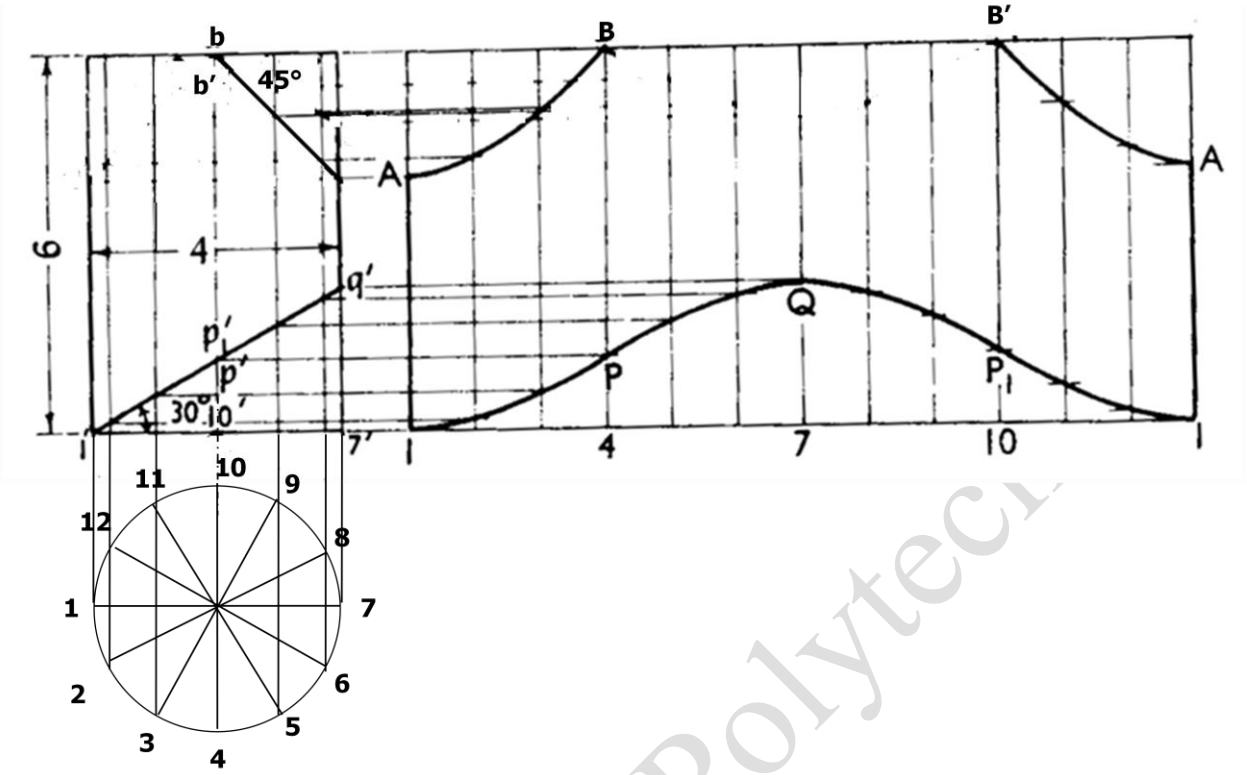
Solution :-



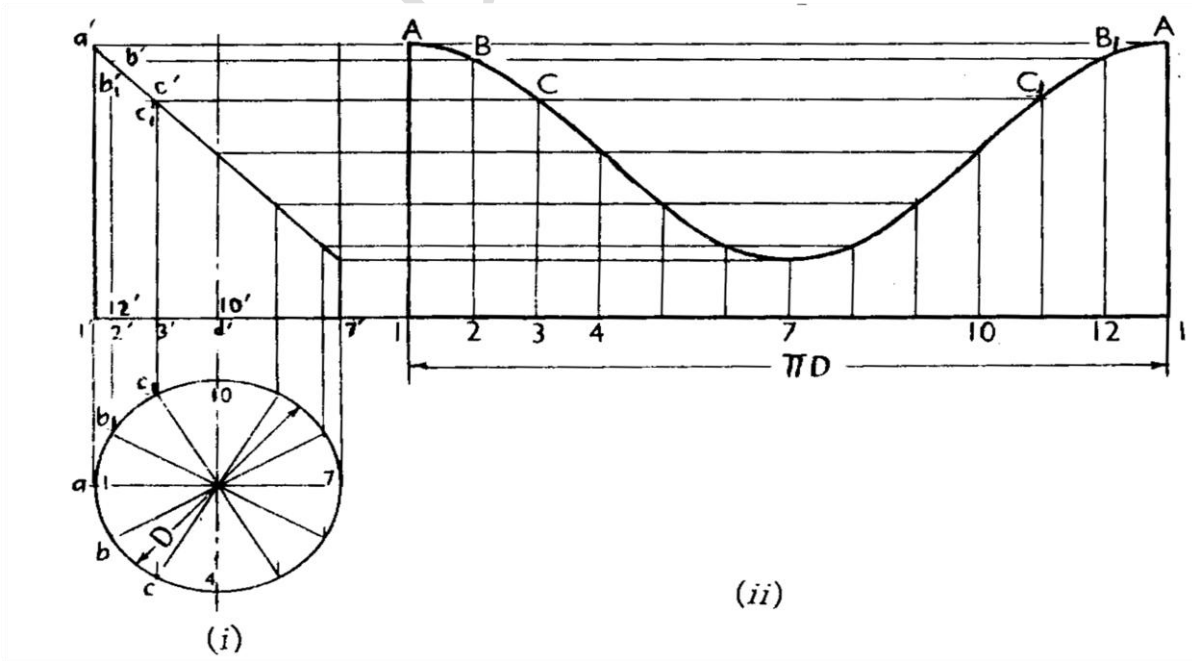
Q. A right circular, diameter of base 40 mm and height 60 mm is truncated at its two ends by two different section planes perpendicular to VP & inclined to HP as shown. Develop lateral surface of truncated cylinder.



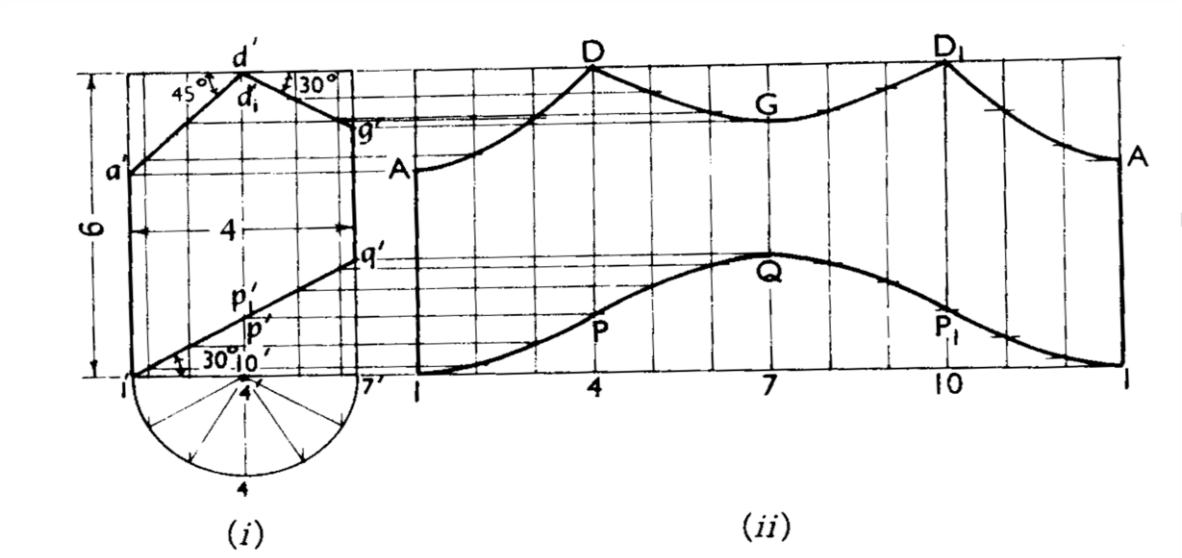
Solution :-



Problem :



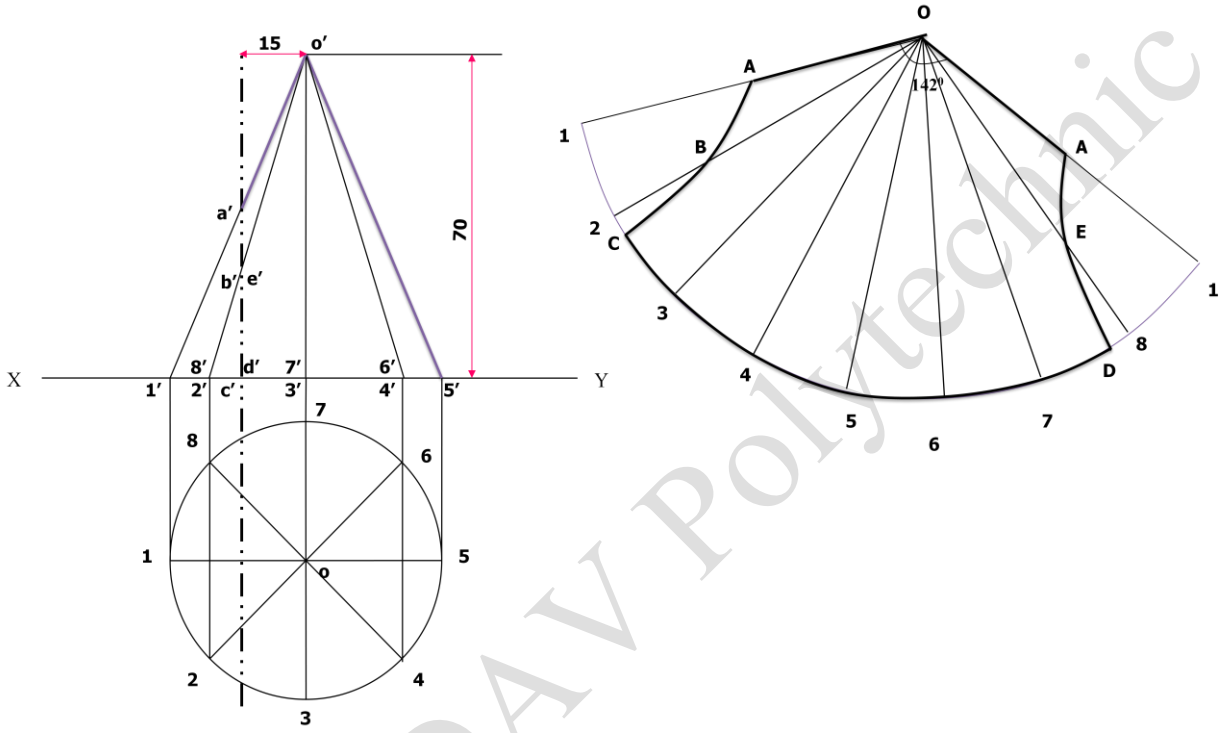
Problem :



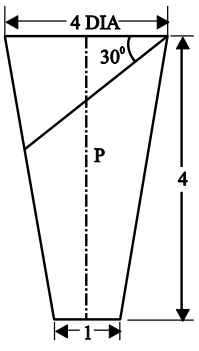
Development of cone

Q. A cone of base diameter 60 mm and height 70 mm is resting on its base on HP. It is cut by a plane perpendicular to both the HP and VP at a distance 15 mm to the left of the axis. Draw the development of the lateral surface of the height remaining portion**.**

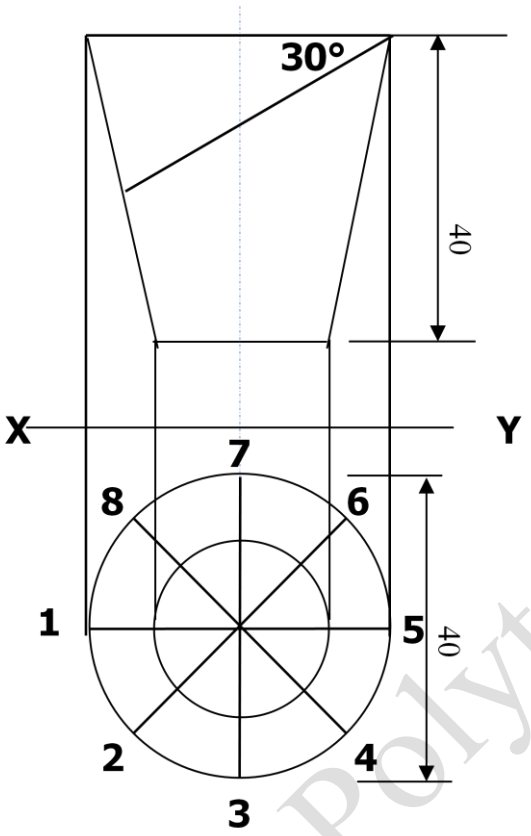
Solution :-

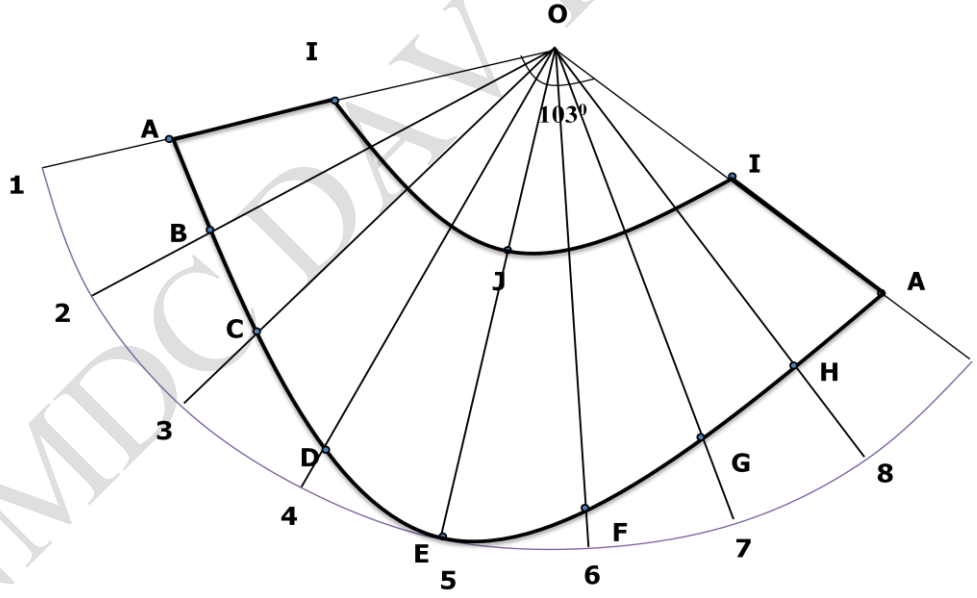


Q. Draw the development of the lateral surface of the part P of the cone front view of which is shown in figure 1 below. z



Solution :-

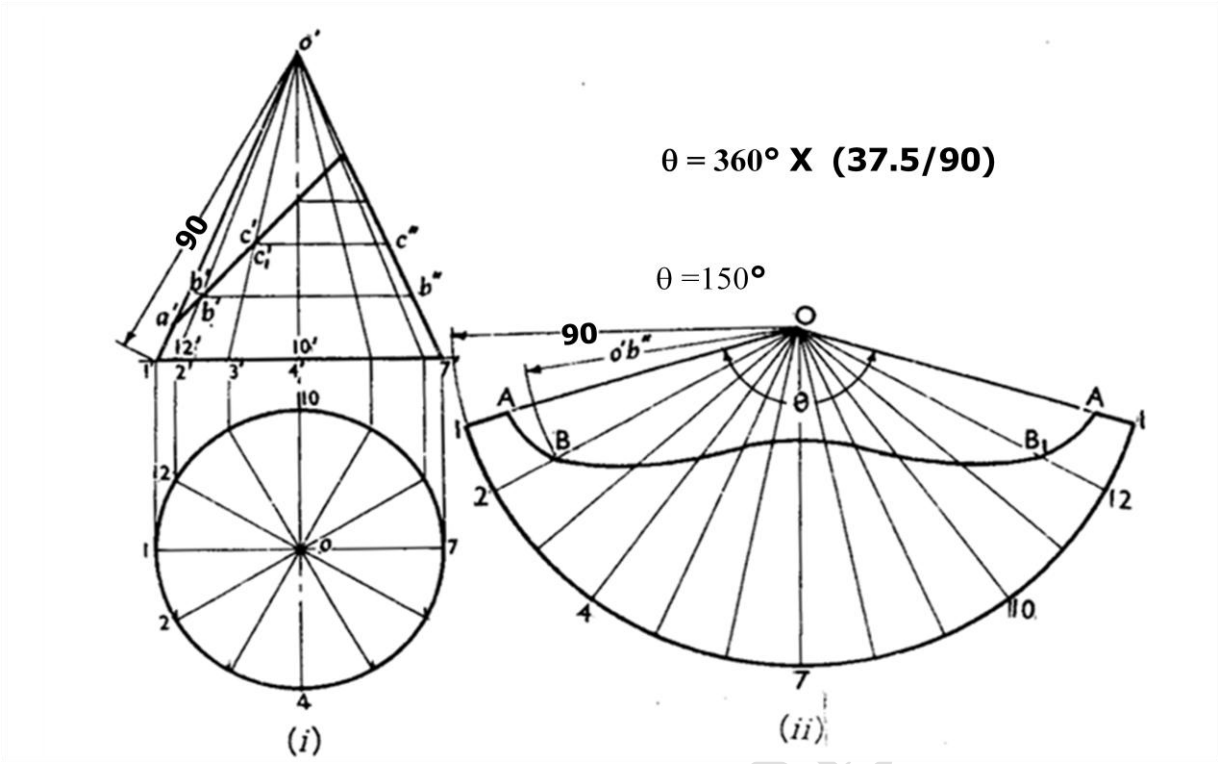




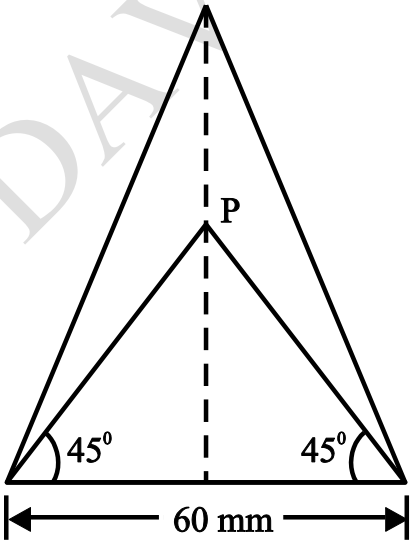
Q. A cone base 75 mm diameter and axis 80 mm long is resting on its base on the H.P. It is cut by a section plane perpendicular to the V.P. inclined 45^0 to the H.P. and cutting the axis at a point 35 mm from the apex. Draw the development of the lateral surface.

Solution :-

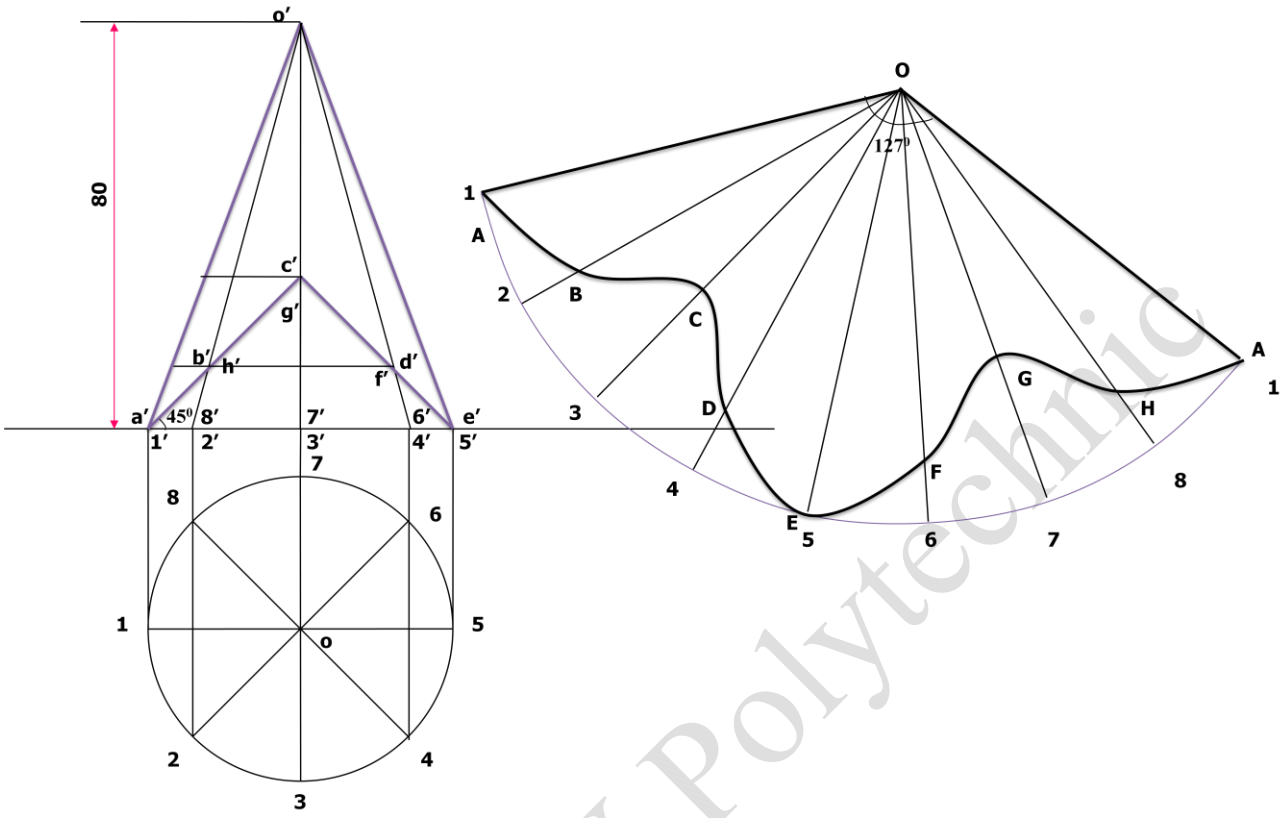
$$\theta = 360^{\circ} \times \frac{\text{radius of the base circle}}{\text{slant height}}$$



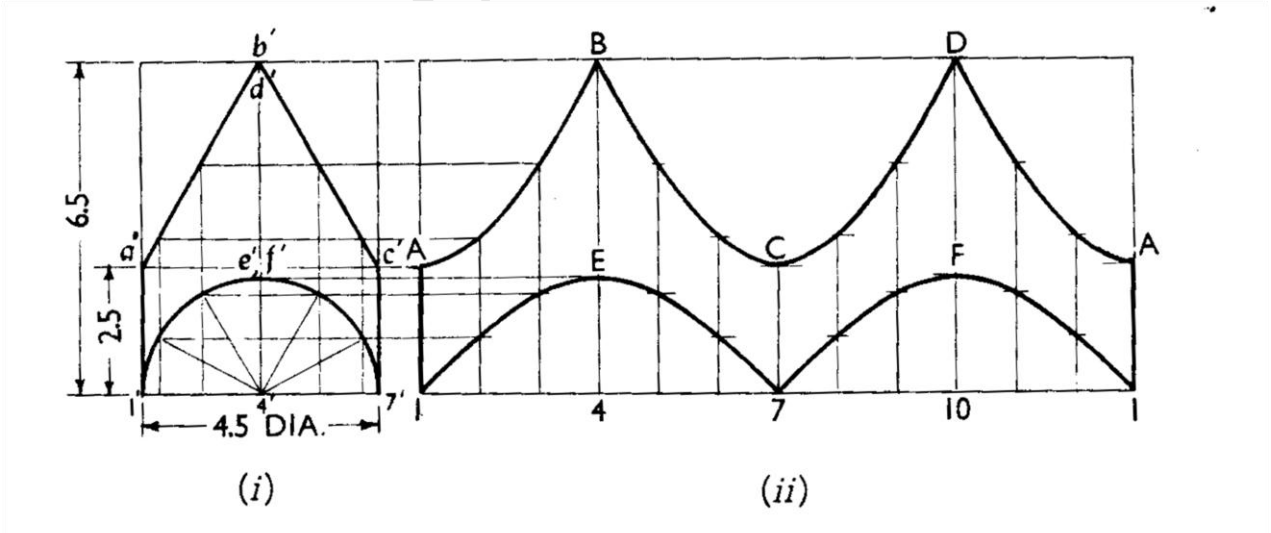
Q. Draw the development of the lateral surface of the part P of the cone. The front view of which is shown in figure.



Solution :-



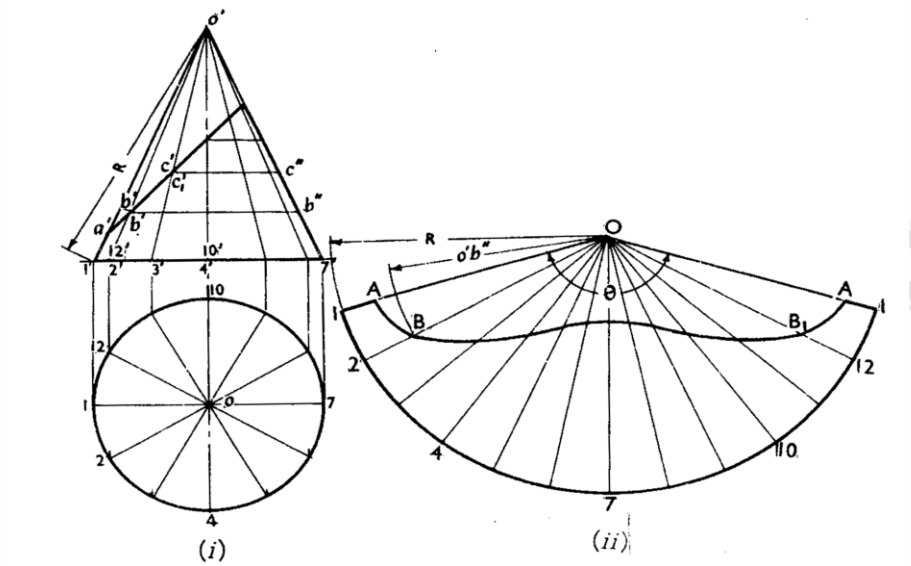
Problem :



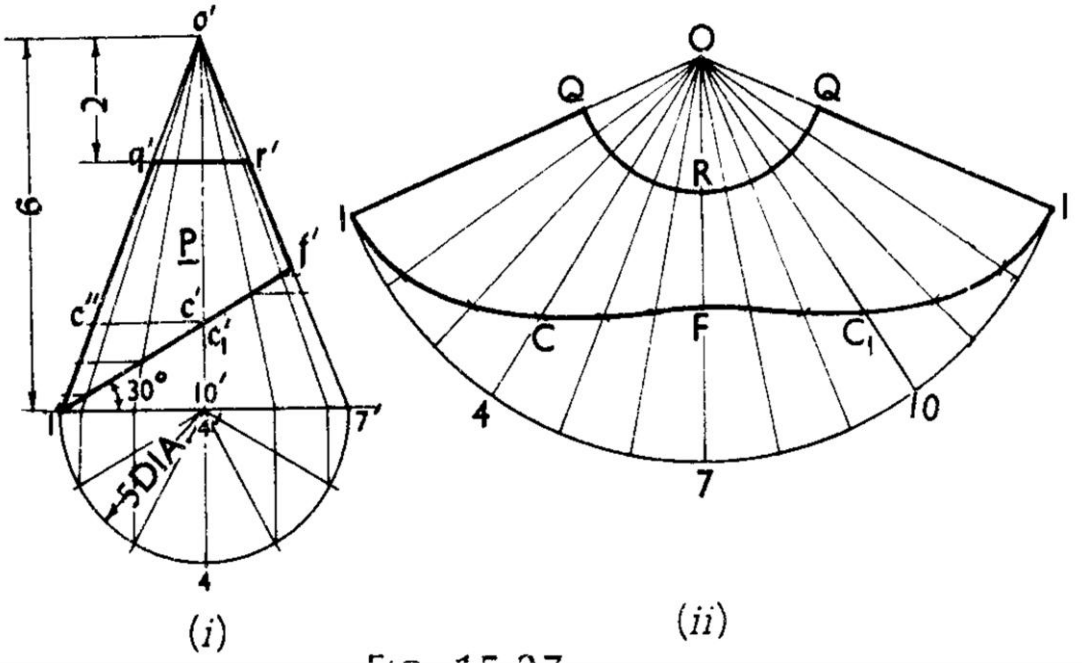
Calculate the subtended angle θ by the formula,

$$\theta = 360^\circ \times \frac{\text{radius of the base circle}}{\text{slant height}}$$

Problem :

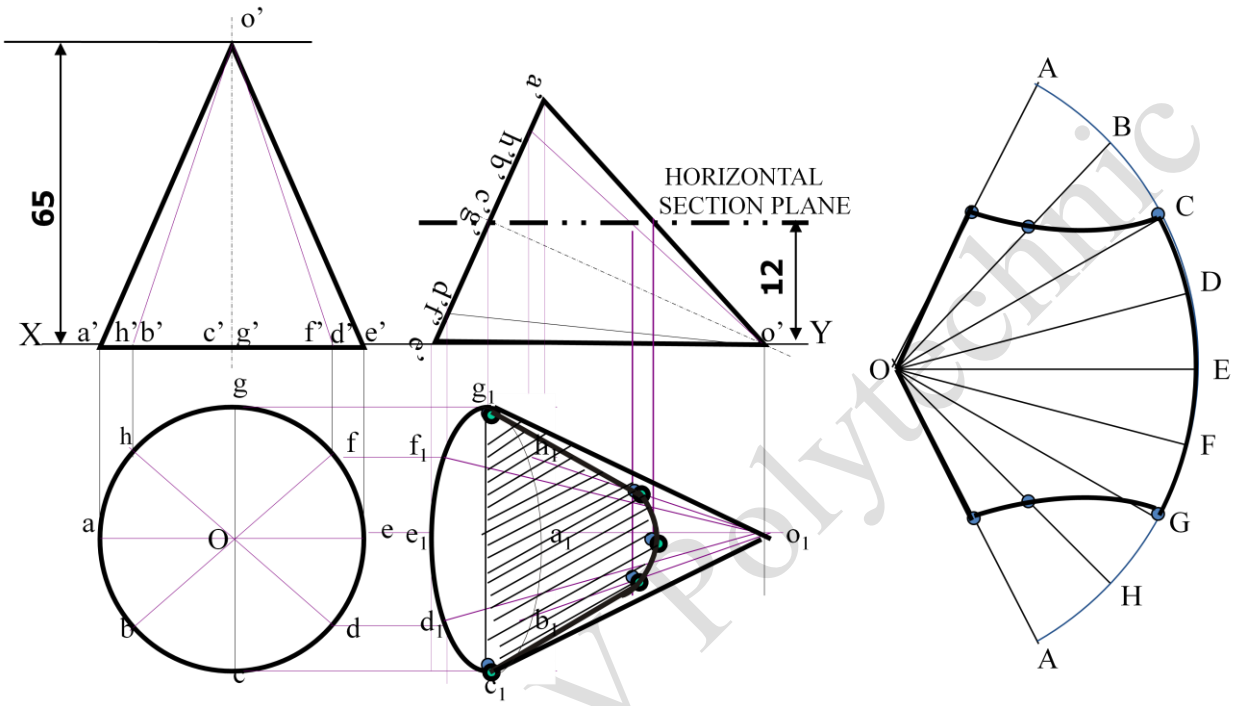


Problem :



Q. A cone, diameter of base 50 mm and axis 65 mm long, is lying on the H.P. on one of its generator with the axis parallel to the V.P. It is cut by a horizontal section plane 12 mm above the ground. Draw its front view and development of its surface.

Solution :-



Problem : A cone, 50 mm base diameter and 70 mm axis is standing on it's base on Hp. It cut by a section plane 45^0 inclined to Hp through base end of end generator. Draw projections, sectional views, true shape of section and development of surfaces of remaining solid.

Solution

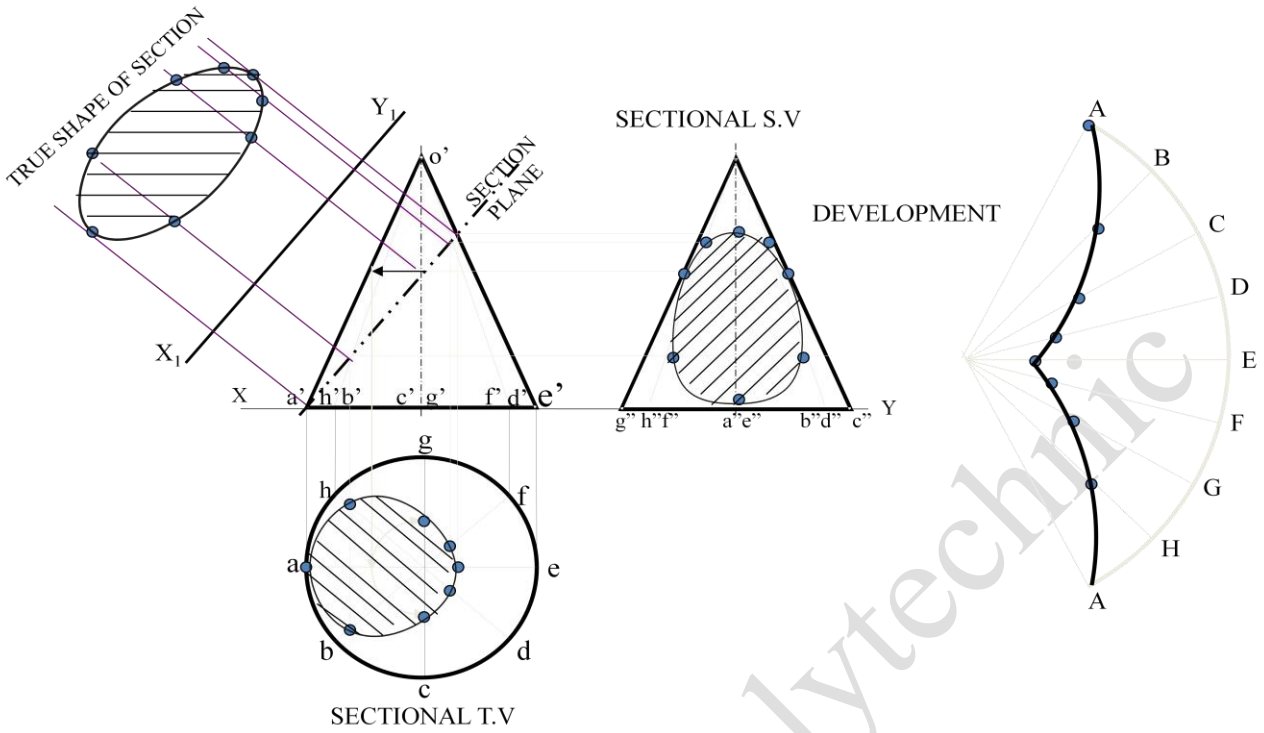
Steps:*for sectional views:*

Draw three views of standing cone.

Locate sec.plane in Fv as described.

Project points where generators are getting Cut on Tv & Sv as shown in illustration.Join those points in sequence and show Section lines in it.

Make remaining part of solid dark.



For True Shape:

Draw x_1y_1 // to sec. plane

Draw projectors on it from cut points.

Mark distances of points of Sectioned part from Tv, on above projectors from x_1y_1 and join in sequence.

Draw section lines in it.

It is required true shape.

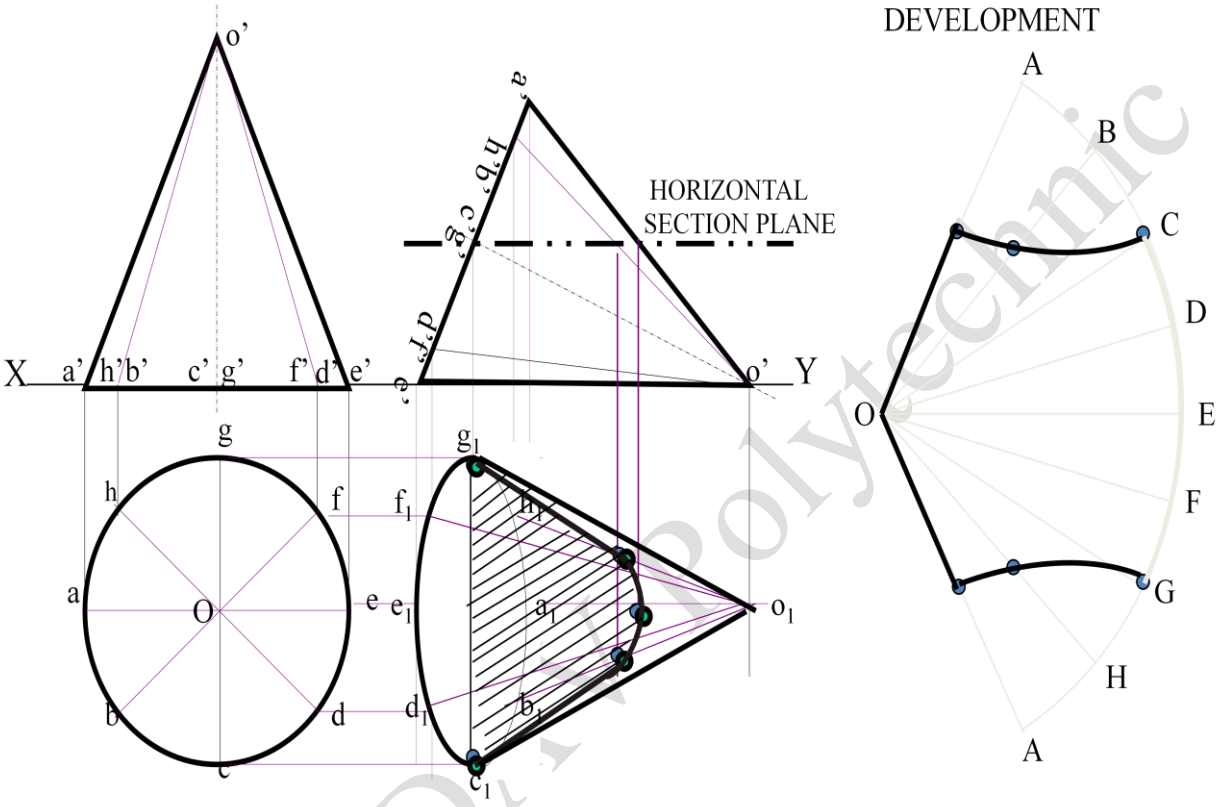
For Development:

Draw development of entire solid.

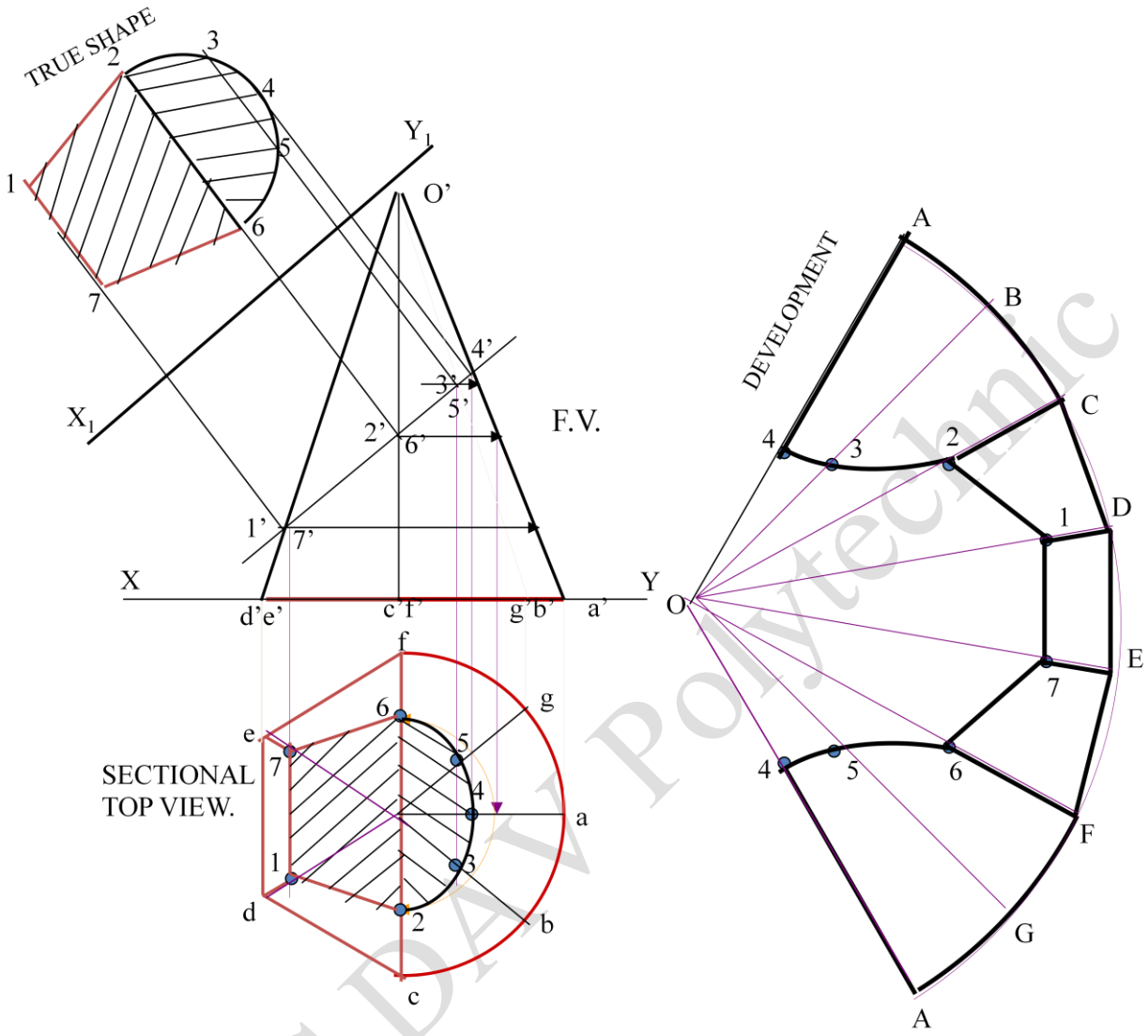
Name from cut-open edge i.e. A. in sequence as shown.Mark the cut points on respective edges.

Join them in sequence in curvature. Make existing parts dev.dark.

Problem : A cone 40mm diameter and 50 mm axis is resting on one generator on Hp(lying on Hp) which is // to Vp.. Draw it's projections.It is cut by a horizontal section plane through it's base center. Draw sectional TV, development of the surface of the remaining part of cone.



Problem : A solid composed of a half-cone and half- hexagonal pyramid is shown in figure.It is cut by a section plane 45^0 inclined to Hp, passing through mid-point of axis.Draw F.v., sectional T.v.,true shape of section and development of remaining part of the solid. (take radius of cone and each side of hexagon 30mm long and axis 70mm.)



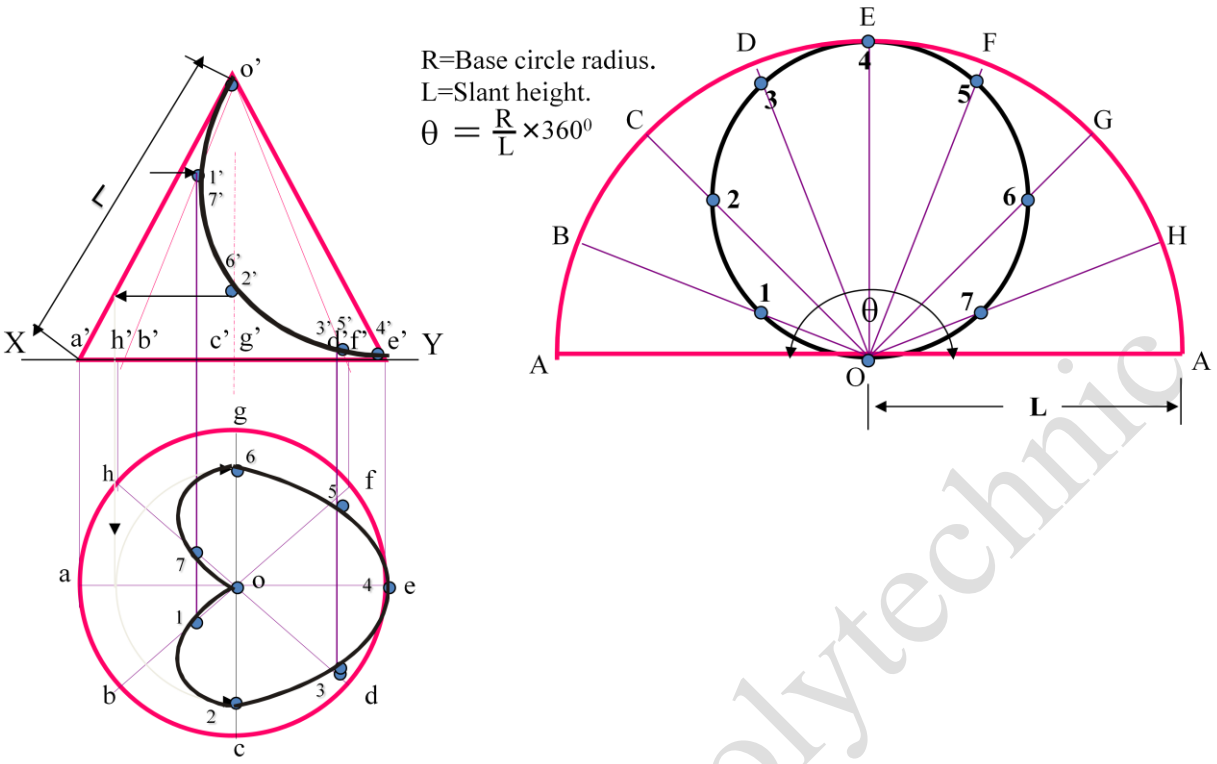
Problem : Draw a semicircle 0f 100 mm diameter and inscribe in it a largest circle. If the semicircle is development of a cone and inscribed circle is some curve on it, then draw the projections of cone showing that curve.

Solution Steps:

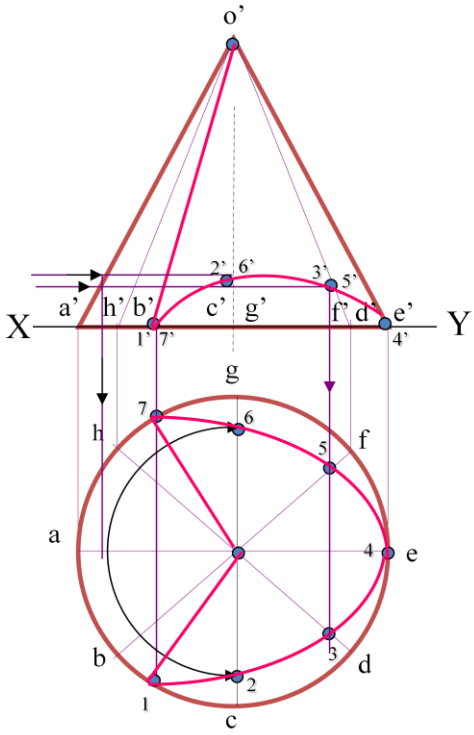
Draw semicircle of given diameter, divide it in 8 Parts and inscribe in it a largest circle as shown. Name intersecting points 1, 2, 3 etc. Semicircle being dev.of a cone it's radius is slant height of cone.(L)

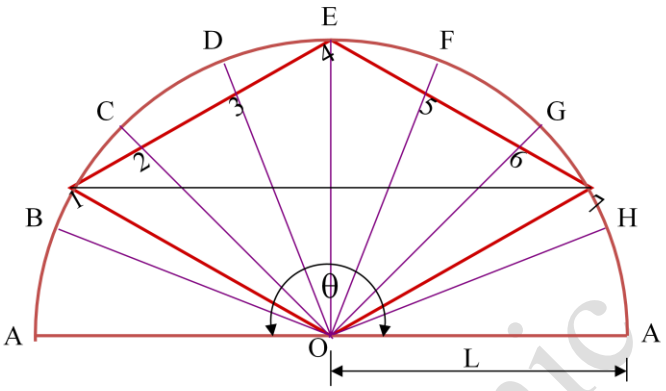
Then using above formula find R of base of cone. Using this data draw Fv & Tv of cone and form 8 generators and name.

Take o -1 distance from dev.,mark on TL i.e.o'a' on Fv & bring on o'b' and name 1' Similarly locate all points on Fv. Then project all on Tv on respective generators and join by smooth curve.



Problem: Draw a semicircle 0f 100 mm diameter and inscribe in it a largest rhombus. If the semicircle is development of a cone and rhombus is some curve on it, then draw the projections of cone showing that curve.





R=Base circle radius.
L=Slant height.

$$\theta = \frac{R}{L} \times 360^0$$

Problem : A half cone of 50 mm base diameter, 70 mm axis, is standing on it's half base on HP with it's flat face parallel and nearer to VP.An inextensible string is wound round it's surface from one point of base circle and brought back to the same point.If the string is of *shortest length*, find it and show it on the projections of the cone.

Concept: A string wound from a point up to the same Point, of shortest length Must appear st. line on it's Development.

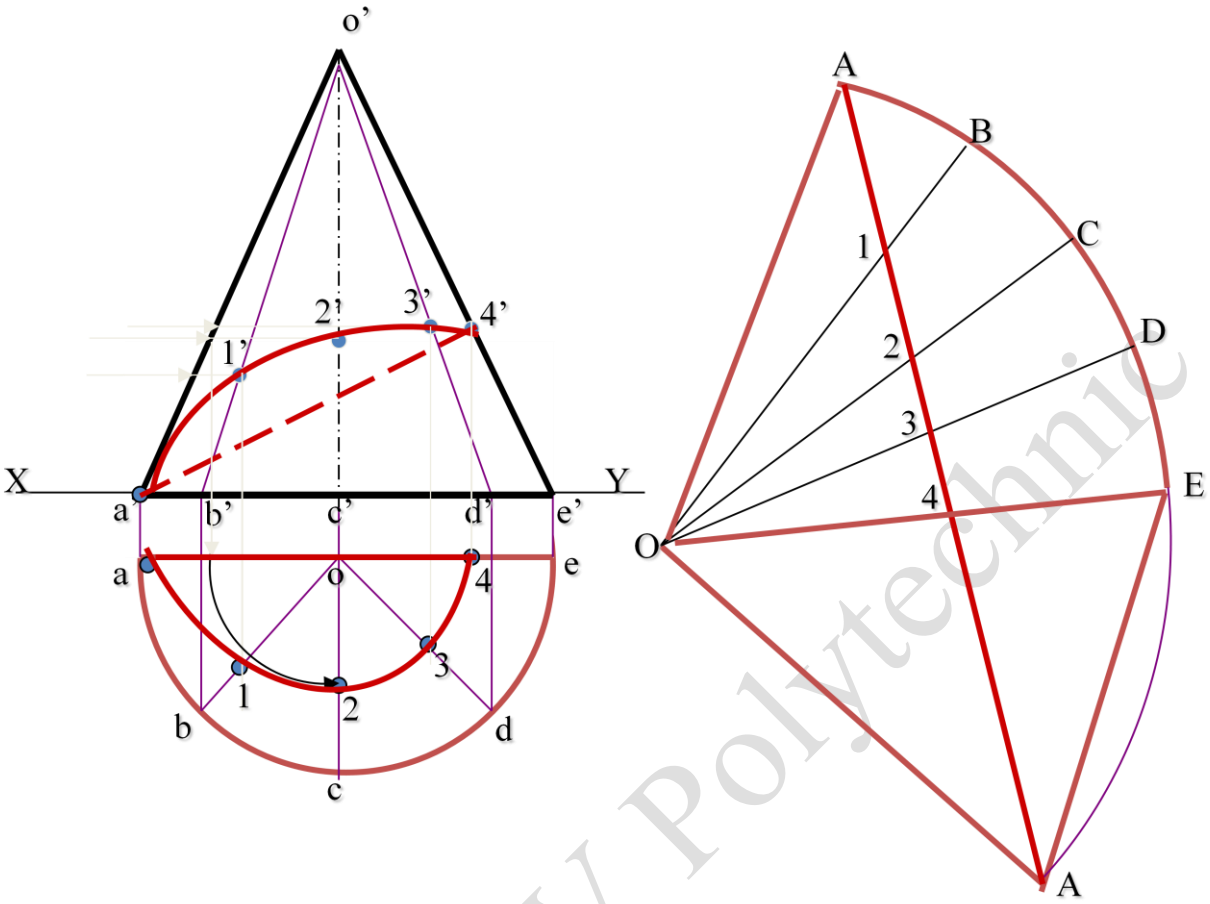
Solution steps:

Hence draw development,

Name it as usual and join A to A This is shortest Length of that string.

Further steps are as usual. On dev. Name the points of Intersections of this line with Different generators. Bring Those on Fv & Tv and join by smooth curves.

Draw 4' a' part of string dotted As it is on back side of cone.



Problem : A particle which is initially on base circle of a cone, standing on Hp, moves upwards and reaches apex in one complete turn around the cone. Draw it's path on projections of cone as well as on it's development.

Take base circle diameter 50 mm and axis 70 mm long.

It's a construction of curve Helix of one turn on cone*:*

Draw Fv & Tv & dev.as usual On all form generators & name.

Construction of curve Helix::

Show 8 generators on both views Divide axis also in same parts.

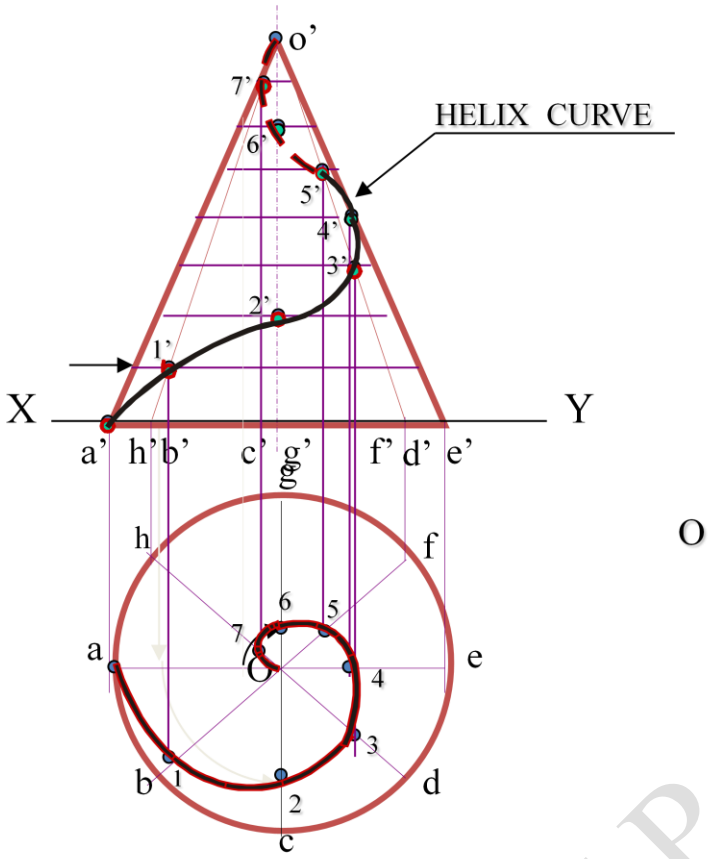
Draw horizontal lines from those points on both end generators.

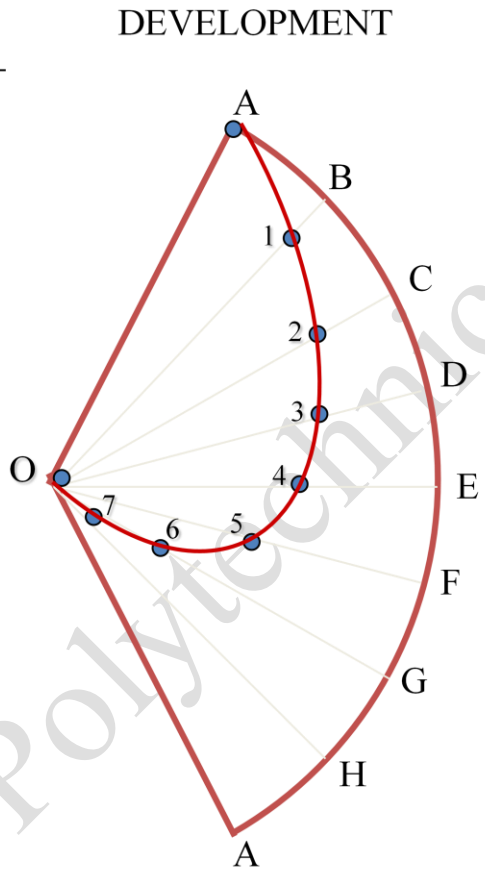
1' is a point where first horizontal Line & gen. b'o' intersect.

2' is a point where second horiz. Line & gen. c'o' intersect. In this way locate all points on Fv. Project all on Tv.Join in curvature.

For Development:

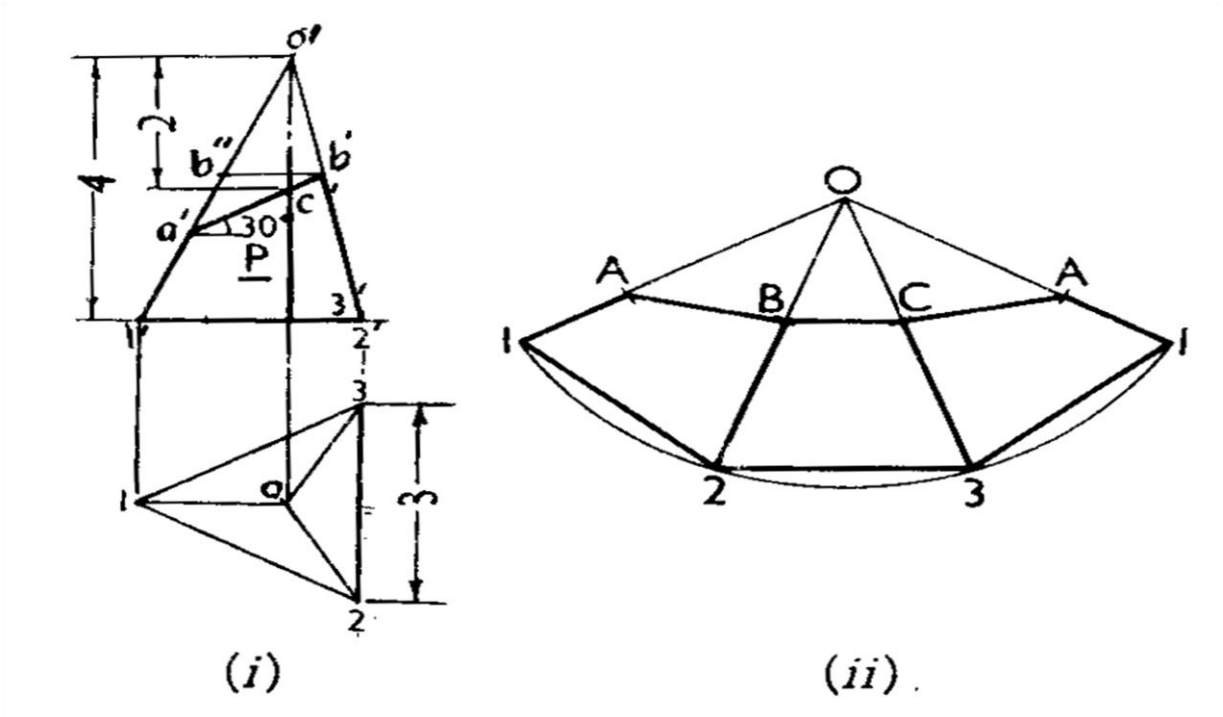
Then taking each points true Distance From resp. generator from apex, Mark on development & join.





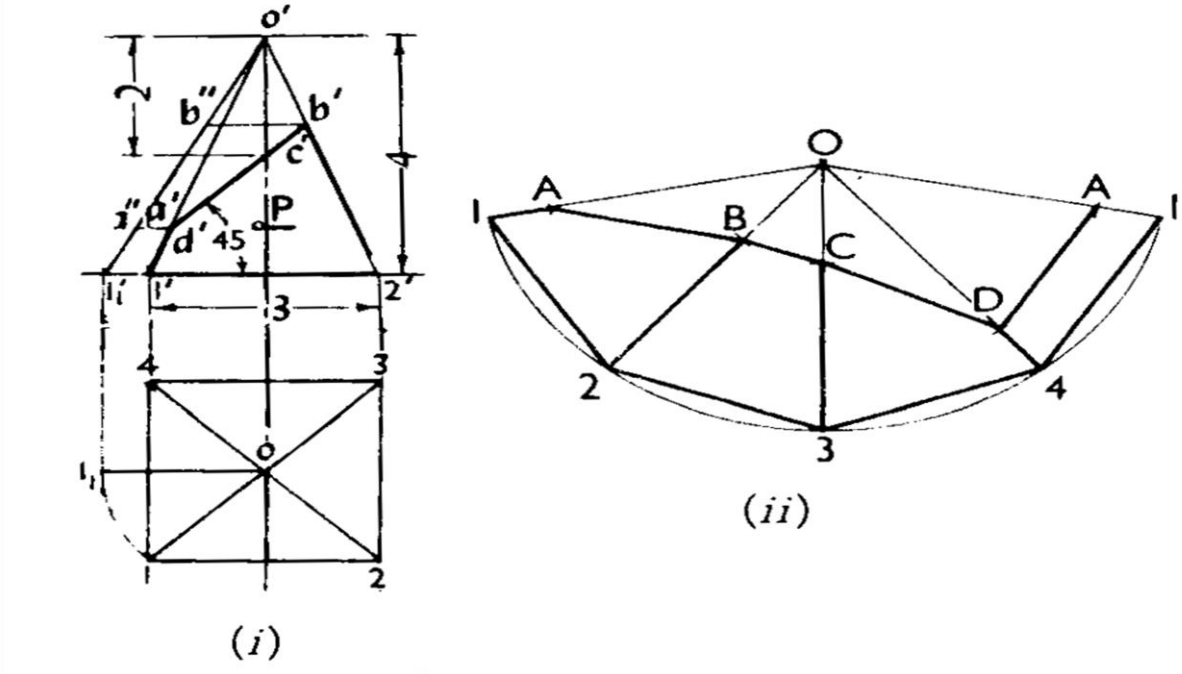
Development of pyramids

Problem :



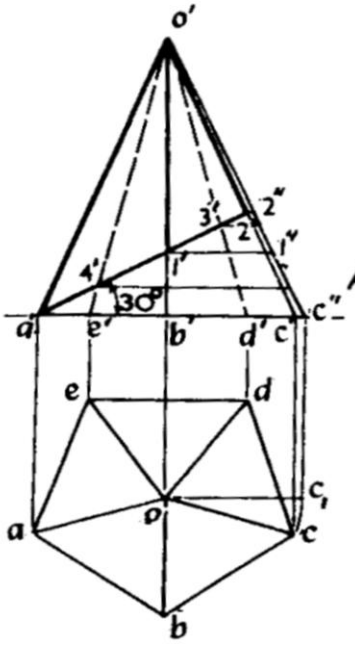
(i) (ii)

Problem :

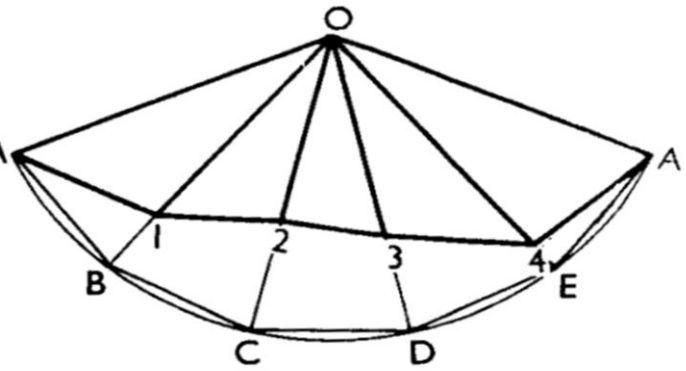


(i) (ii)

Problem :

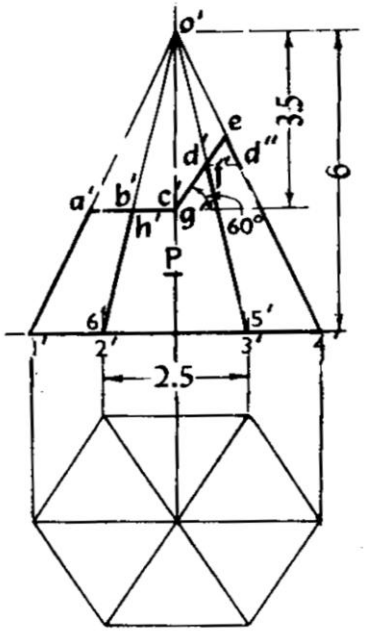


(i)

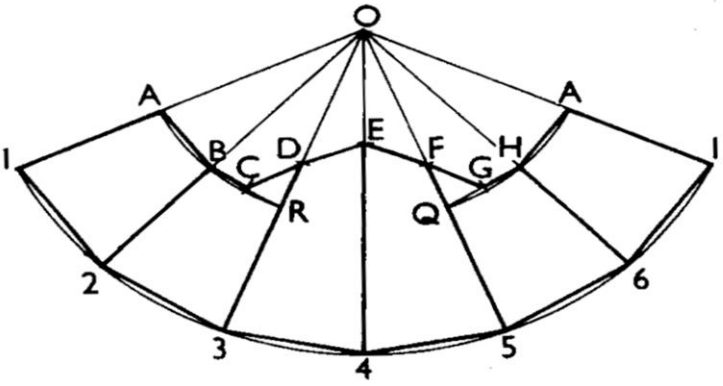


(ii)

Problem :



(i)

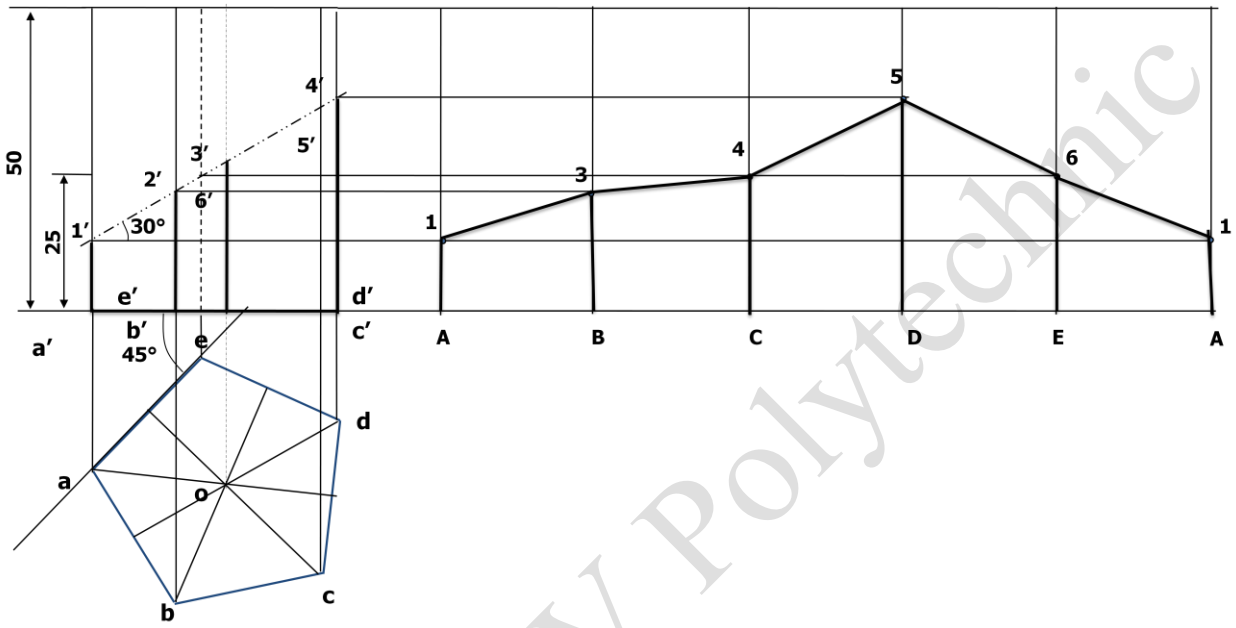


(ii)

Development of prism

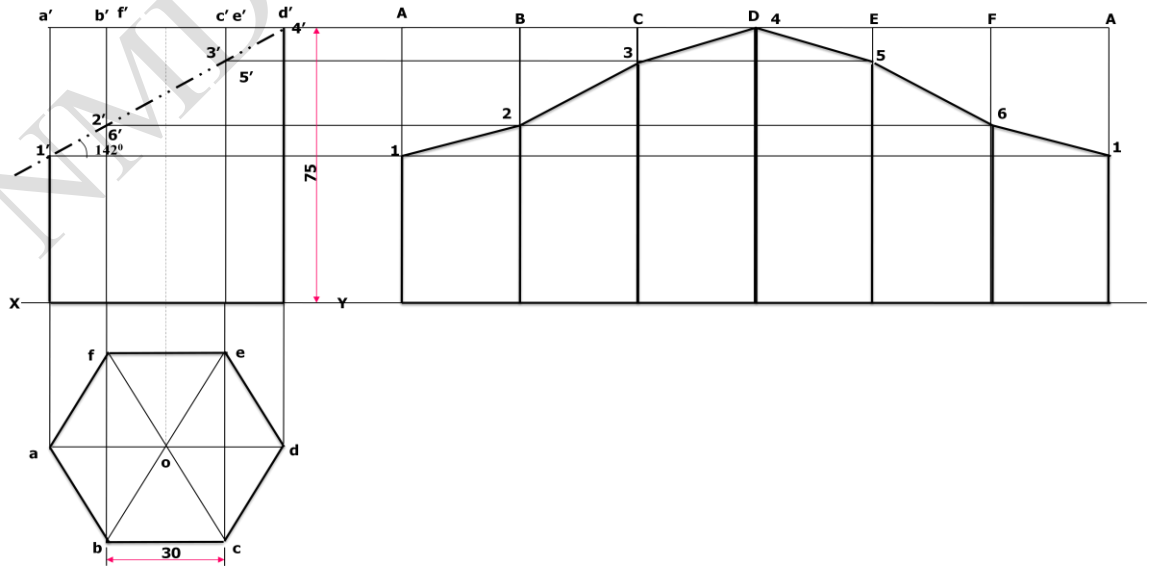
Q. A pentagonal prism of 25 mm base edges and 50 mm long, is resting on its base with and edge of base 45^0 to V.P. The prism is cut by a section plane V.T. inclined at 30^0 to H.P. and passes through a point 25 mm from the base along its axis. Develop the lateral surface of the truncated prism.

Solution :-

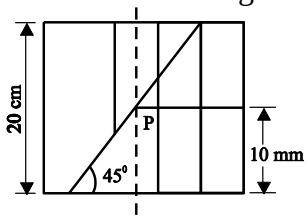


Q. A hexagonal prism of side of base 30 mm and axis 75 mm long, is resting on its base on HP such that, a rectangular face is parallel to VP. It is cut by a sectional plane, perpendicular to VP and inclined at 30^0 to HP. The section plane is passing through the top end of an extreme lateral edge of the prism. Draw the development of the lateral surface of the cut prism.

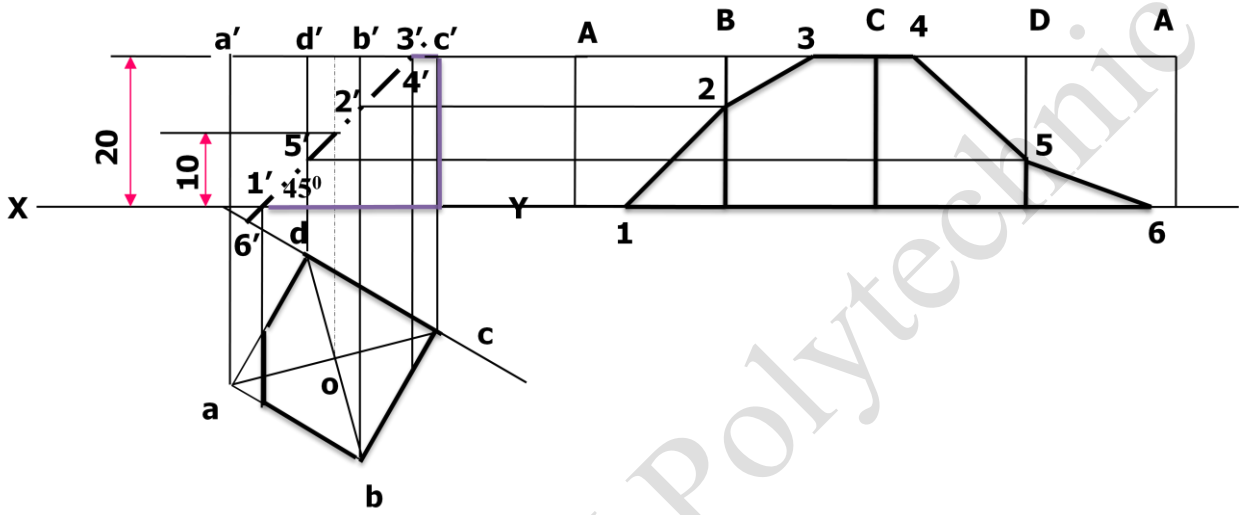
Solution :-



Q. Draw the level of the lateral surface of the part p of a cube, one vertical face inclined, 30^0 to the V.P., the front view of which is shown in figure.

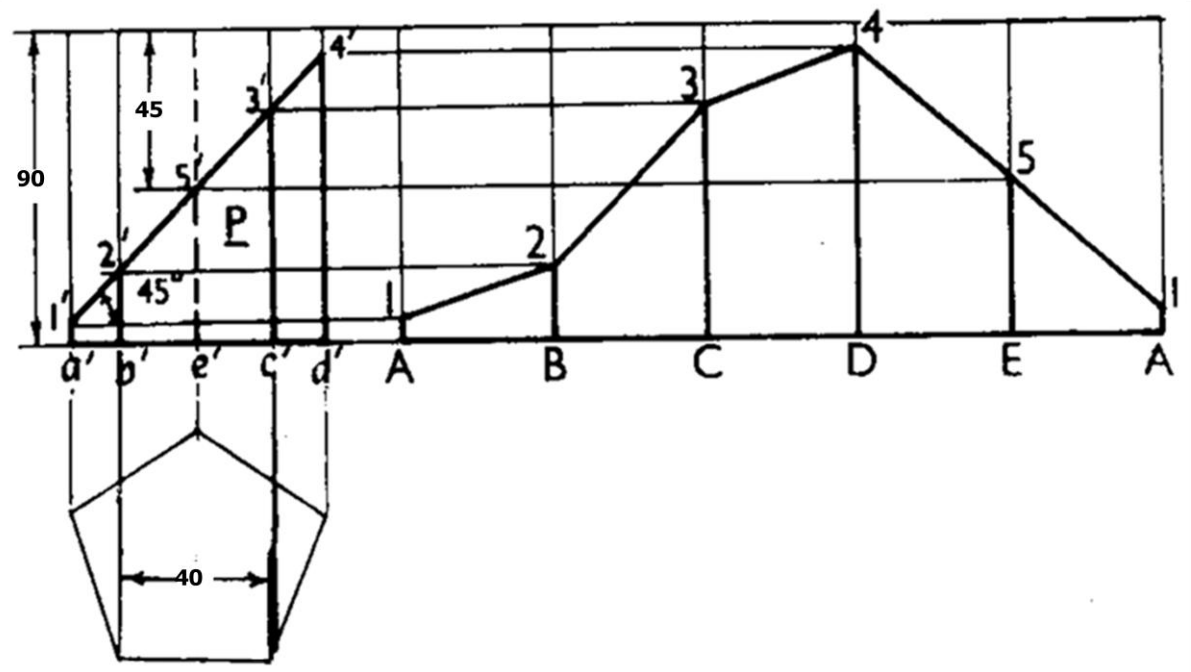


Solution :-

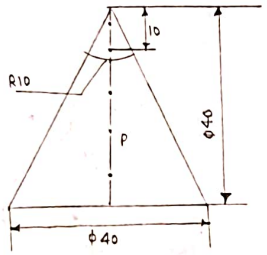


Q. Draw the development of lateral surface of a pentagonal prism with edge of base 40 mm and height 90 mm, kept on H.P. on its base with an edge of base parallel to V.P. when it is cut by an AIP, inclined at to H.P. and bisecting the axis of prism.

Solution :-

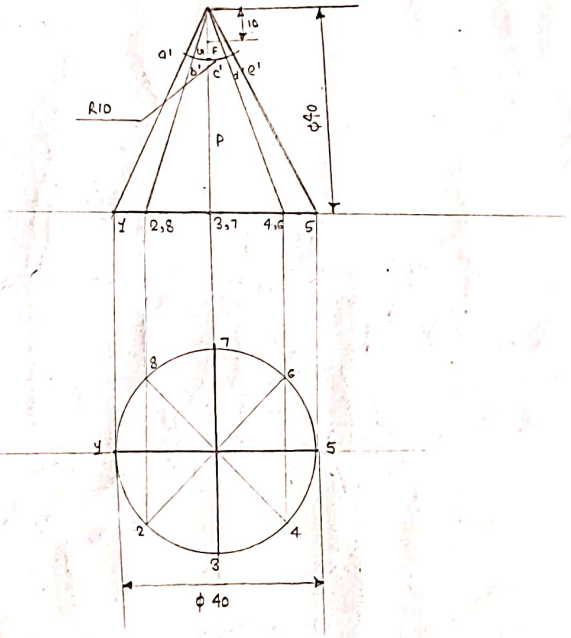


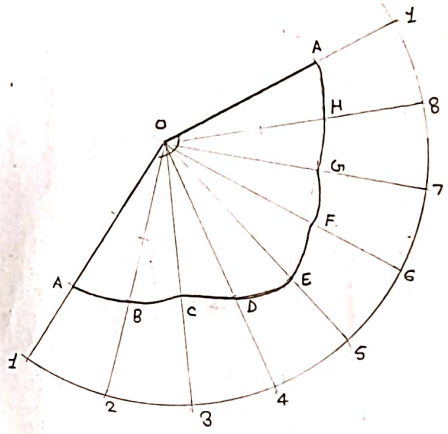
1. Draw the development of lateral surface of the part 'P' of each of cone, the front views of which are as shown in fig.



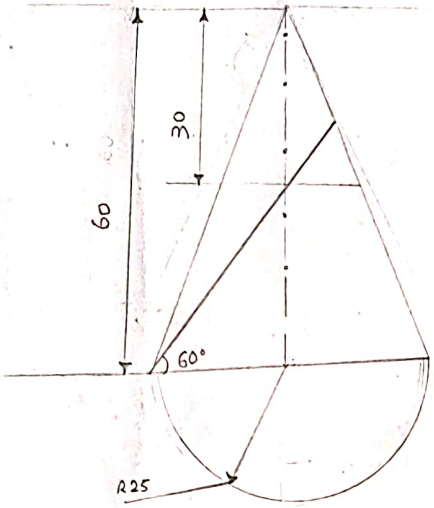
$$\frac{\cancel{360^{\circ}}^{40} \times \cancel{20}^{4}}{\cancel{45}_{9}} = 160^{\circ}$$

$$\frac{\cancel{160^{\circ}}}{\cancel{8}} = 20^{\circ}$$



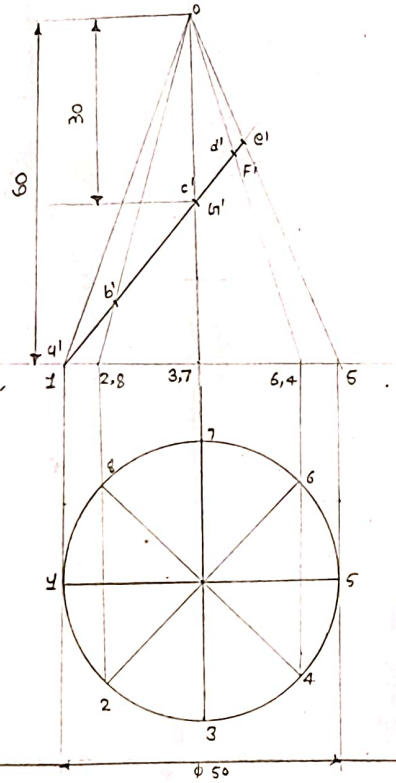


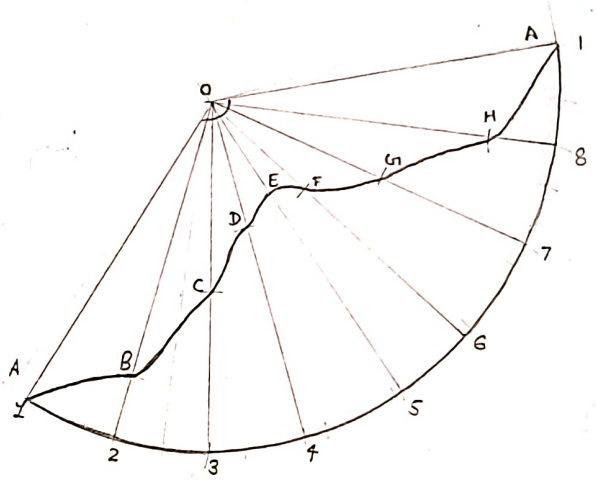
2. Draw the development of the lateral surface of the part of the cone shown in fig.



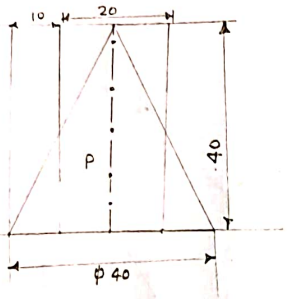
$$360^\circ \times \frac{25}{68} = 188^\circ$$

$$\frac{188}{8} = 17.25^\circ$$



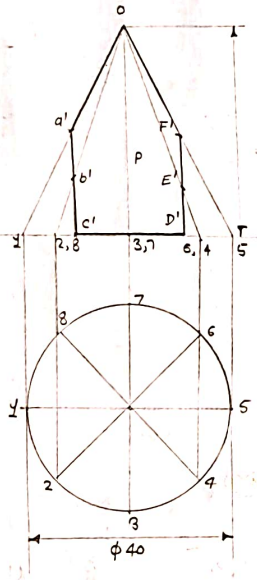


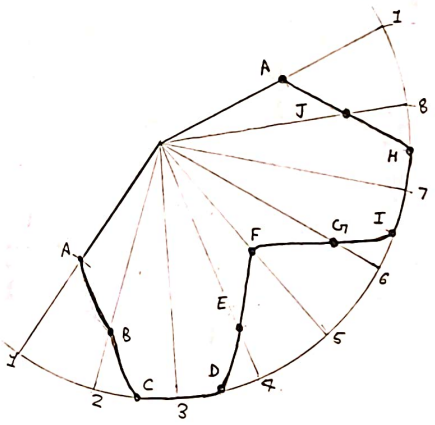
8. Draw the development of the lateral surface of the part P of a cone front view of which is shown in fig.



$$\frac{\cancel{360^\circ} \times \cancel{20}}{\cancel{45}} = 160^\circ$$

$$\frac{\cancel{160^\circ}}{\cancel{8}} = 20^\circ$$

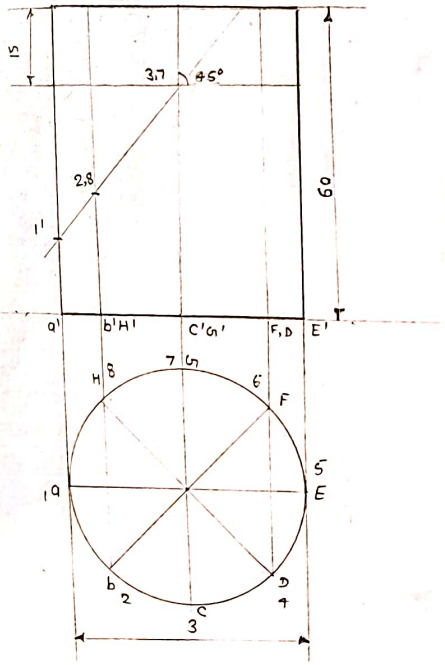


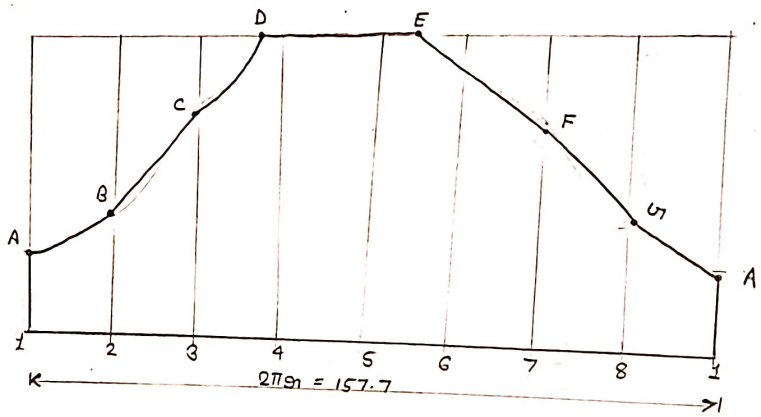


4. Draw the development of lateral surface of the part of the cylinder shown in fig.

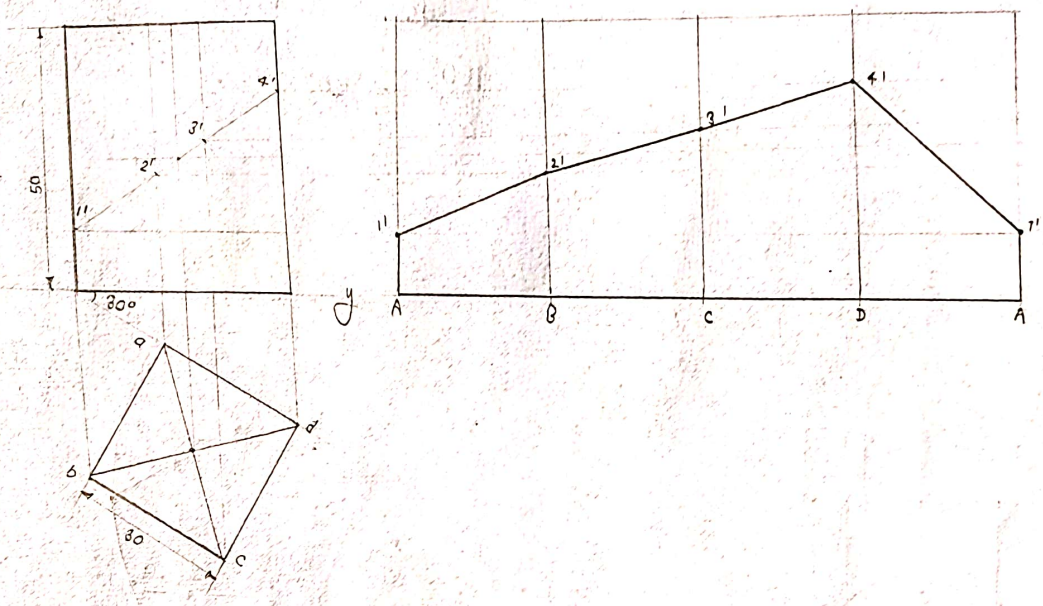
$2\pi r$ $\quad 2 \times \pi \times 25$

$\frac{167.07}{8} = 19.64$ mm



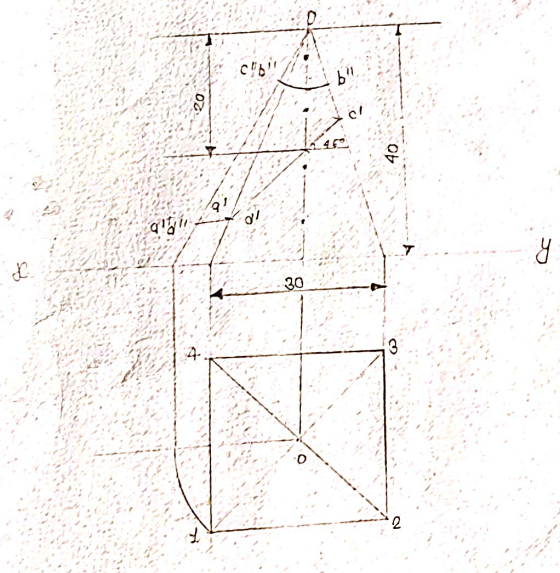


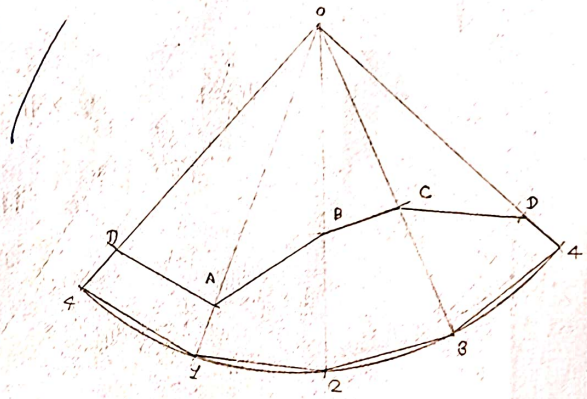
5. Draw the development of the lateral surface of the part 'P' of each of cylinders, the front views of which are as shown fig.



6. Draw the development of the lateral surface of the part of P of the square prism shown in fig.

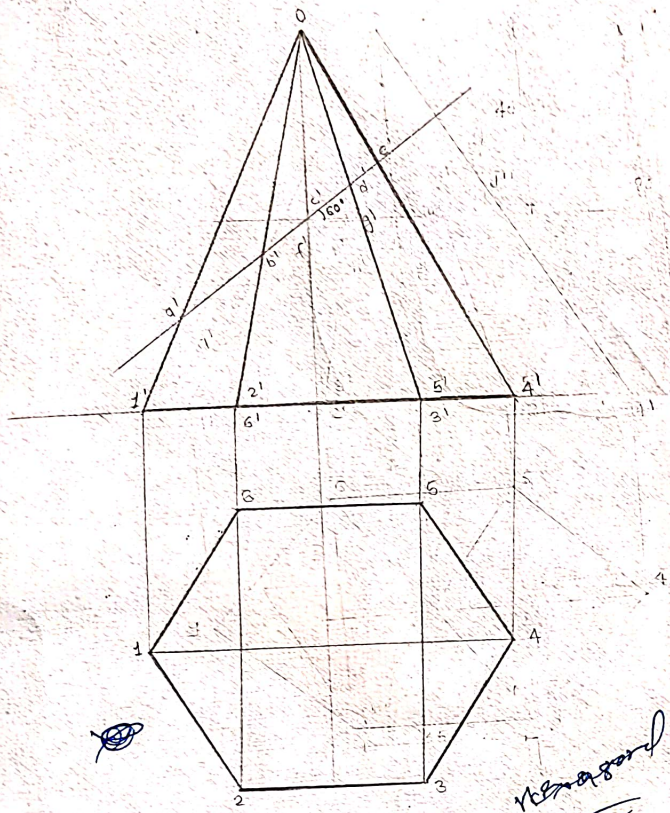
$$360^\circ \times \frac{15}{30} = 180^\circ, \quad = \frac{180^\circ}{4} = 45^\circ$$

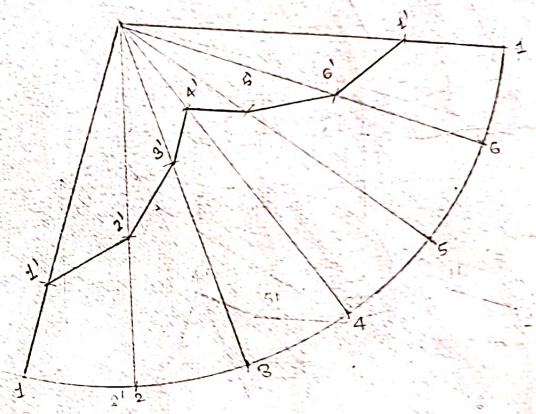




07. Draw the development of the lateral surface of the part 'P' of the hexagonal Pyramid shown in fig.

$$360° \times \frac{17}{35} = 174° \qquad \frac{174°}{6} = 29°$$





Problem 15-33. (fig. 15-35): *An air-conditioning duct of a square cross-section 70 mm × 70 mm connects a circular pipe of 40 mm diameter through the transition piece. Draw the projections and develop the lateral surface of the transition piece.*

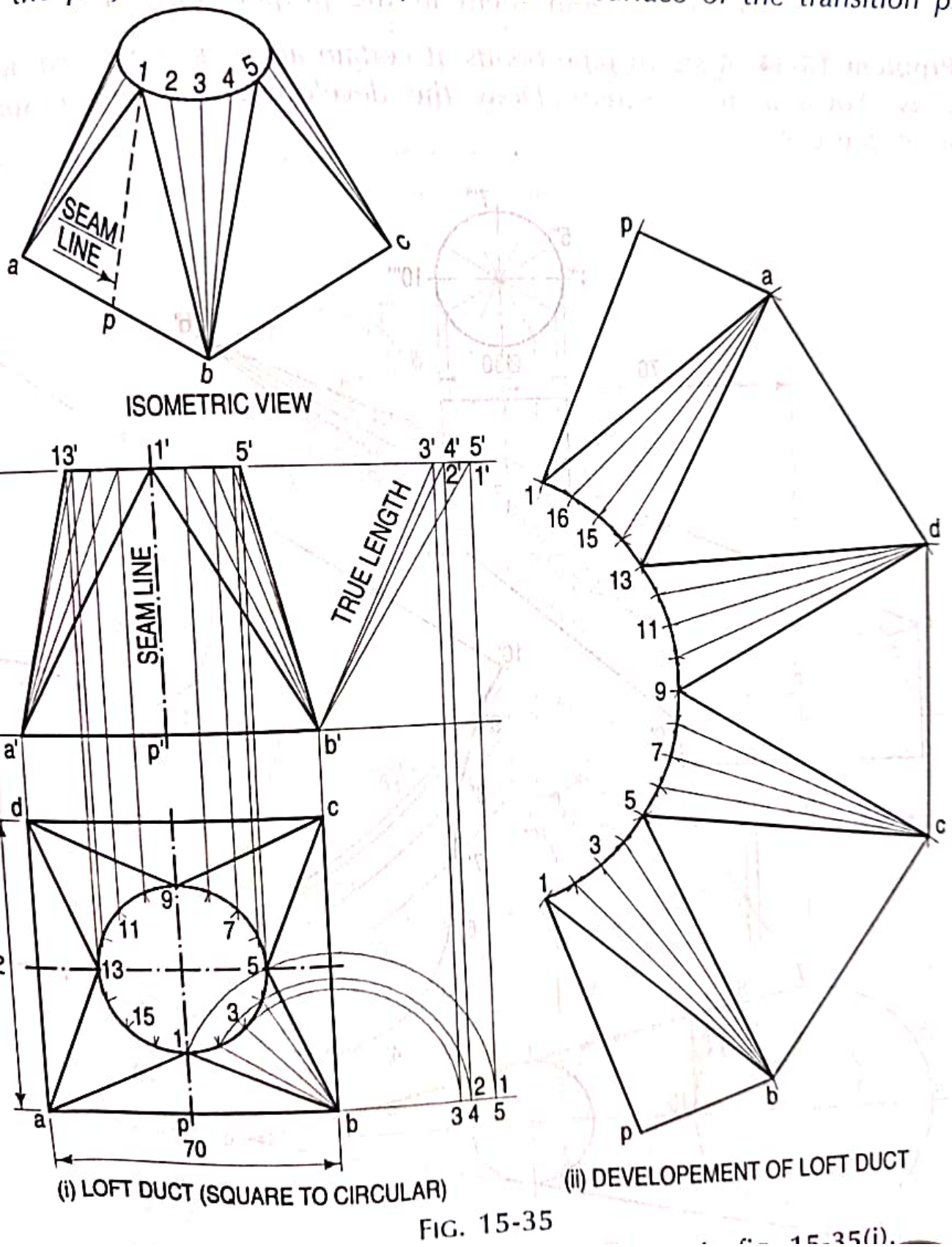
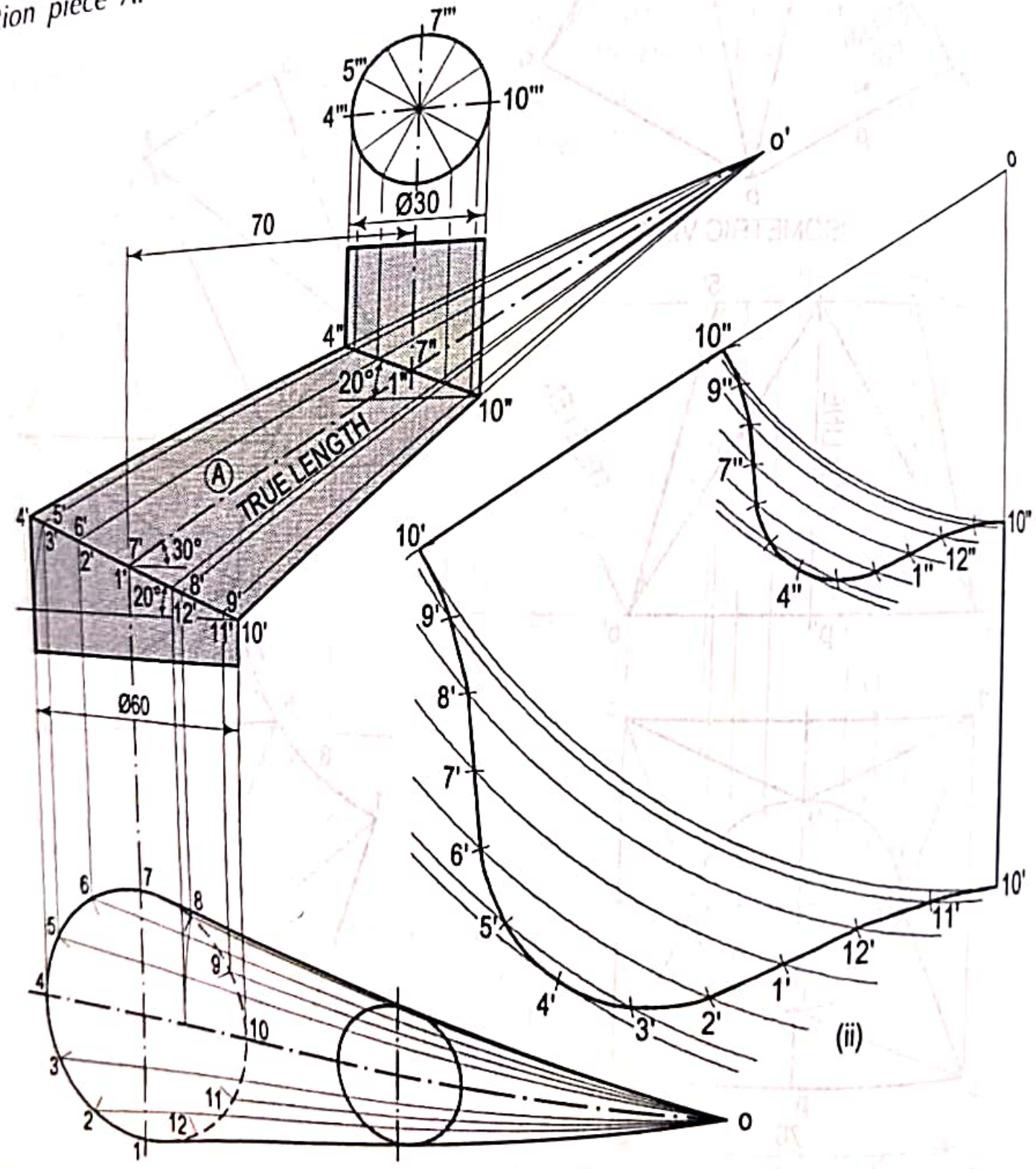


FIG. 15-35

arc drawn ... points 3', 4', 5' etc. join ...

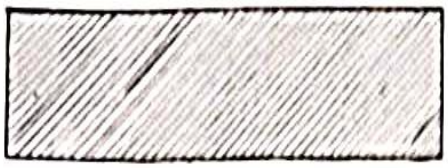
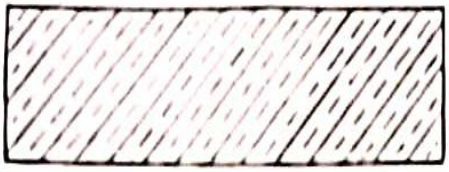
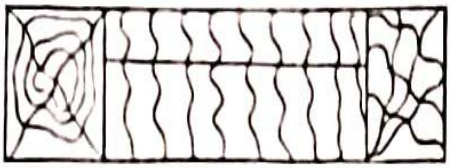
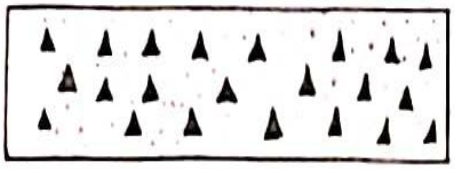
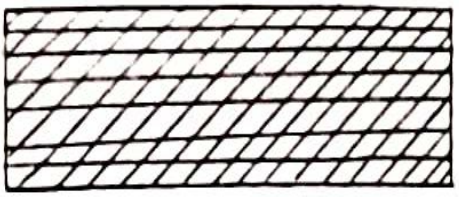
Problem 15-34. *A steam pipe bends at certain angle is connected by a transition piece as shown in fig. 15-36(i). Draw the development of lateral surface of the transition piece A.*



Unit 3

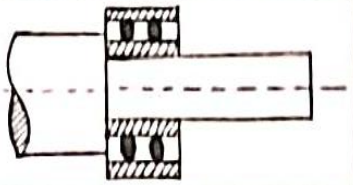
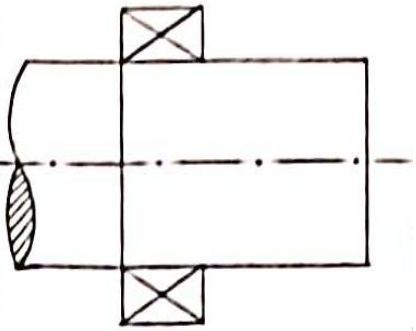
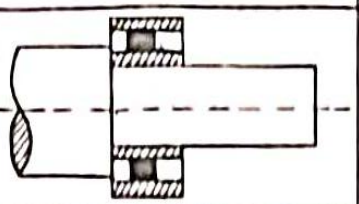
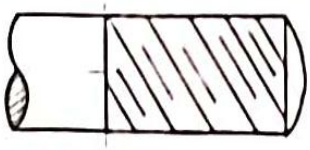
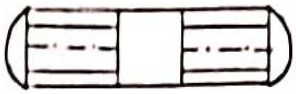
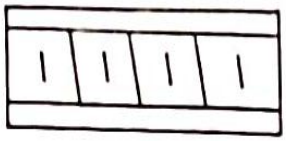
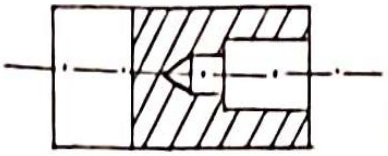
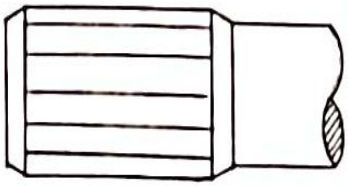
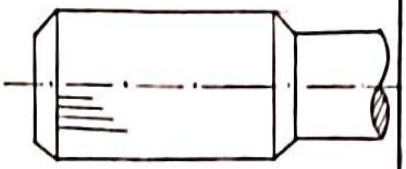
Conventional Representations

(Ex. -3.2)

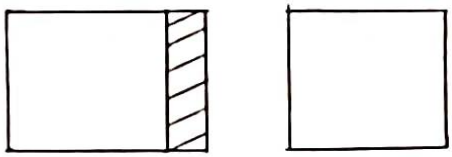
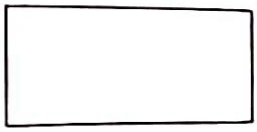
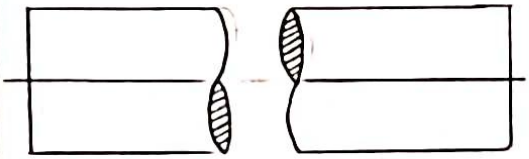
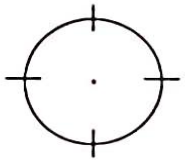
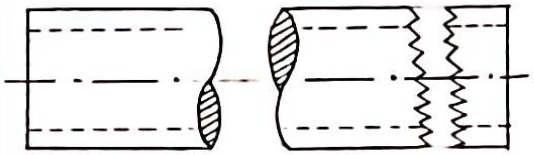
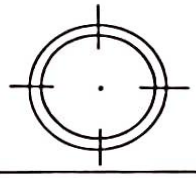
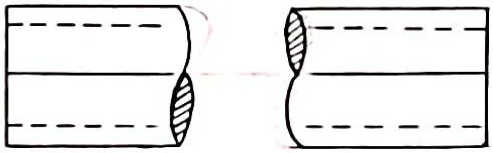
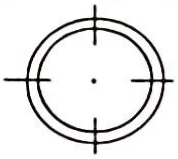
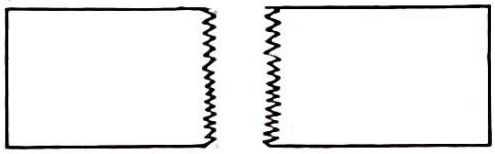
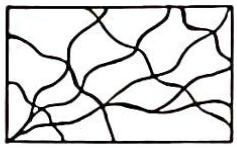
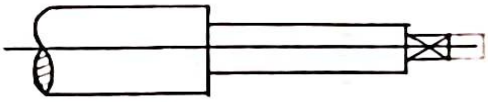
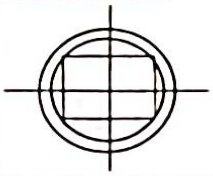
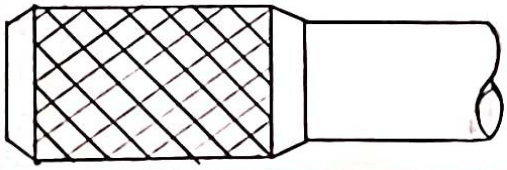
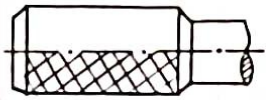
S.No.	Material	Convention
01.	Iron, cast Iron, Steel copper, Alluminium and its alloys	
02.	Brass, Bronze, Gun metal etc.	
03	Wood, plywood etc.	
04.	Concrete	
05.	Glass	
06.	Rubber	

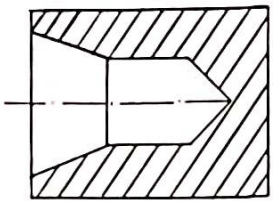
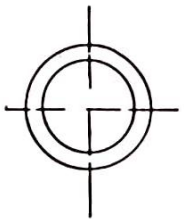
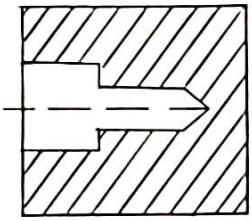
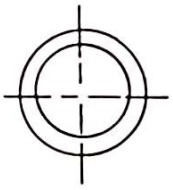
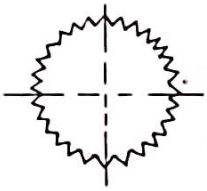
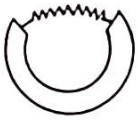
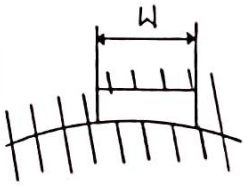
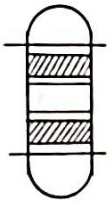
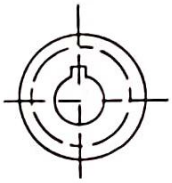
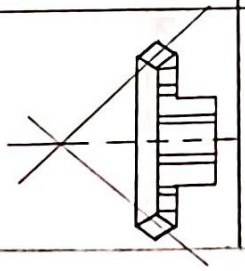
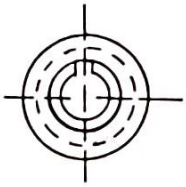
UNIT-03

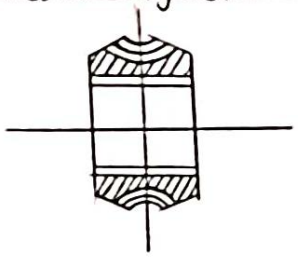
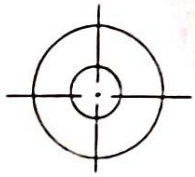
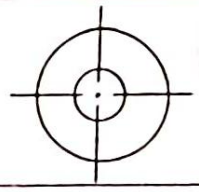
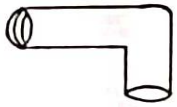
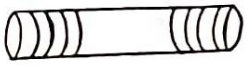
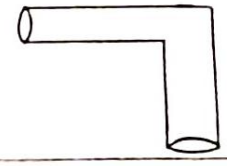
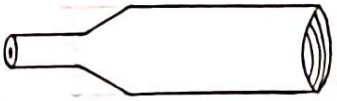
Conventional Representation

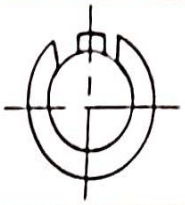
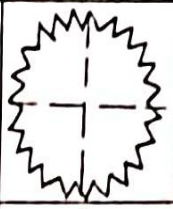
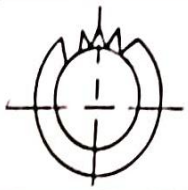
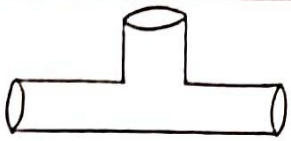
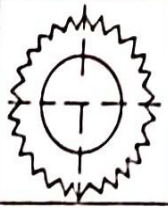
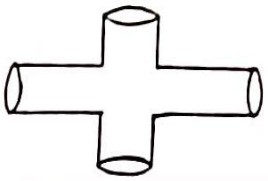
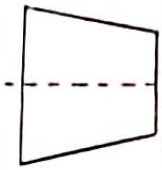
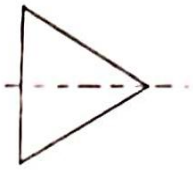
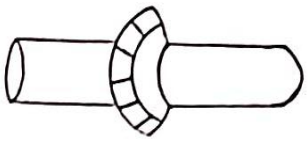
S.N.	Title	Actual projection	Convention
01.	Ball Bearing		
02.	Roller Bearing		
03.	External Theard		
04.	Internal Theard		
05.	Straight knurling		

(Ex. -3.3) Convention Break

S.No	Object	Convention	Symbols
01.	Rectangular section		
02.	pipe or tubing		
03.	Long brake in pipe		
04.	pipe or tubing		
05.	Wood rectangular section		
06.	Squre or Shalit		
07.	Diamond knurling		

S.No	Title	Actual Projection	Convention
01.	Counter Sunk		
02.	Counter Bore		
03.	Sprocket wheel		
04.	Saddle key		
05.	Spur Gear		
06.	Bevel Gear		

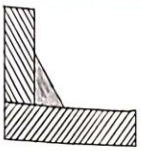
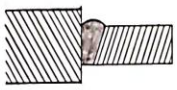
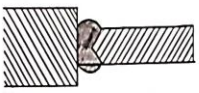
S.No.	Title	Actual Projection	Convention
01.	Warm wheel		
02.	flat and spring		
03.	Squre and spring		
04.	Socket		
05.	Elbow		
06.	Nipple		
07.	Bend		
08.	Reducer		

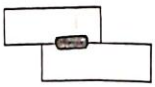
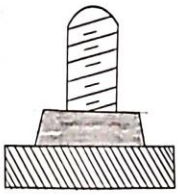
S.N.	Title	Actual projection	Convention	S.N.	Title	Accessories.
01.	Splined shaft			01.	plug	
02.	Serralted shaft			02.	TEE	
03	Chain wheel			03.	Cross	
04.	Taper			04.	union	

Unit 4

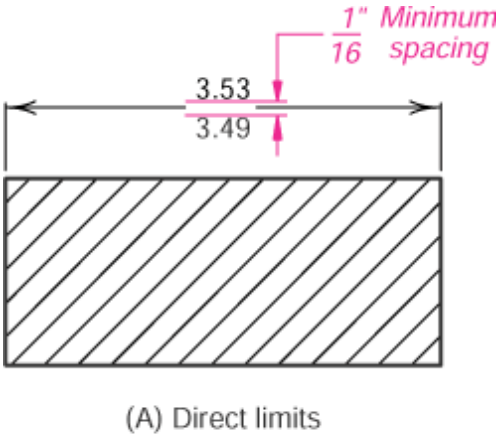
Limits-fits-tolerance, surface finish and welding symbols in Production drawing

Welding symbols

S. No.	Form of weld	Sectional Representation	Symbol
01	Fillet		
02	Square butt		
03	Single 'V' butt		
04	Double 'V' butt		
05	Single 'U' butt		
06	Double 'U' butt		
07	Single 'bevel' butt		
08	Double 'bevel' butt		
09	Single 'J' butt		
10	Double 'J' butt		
11	Bead edge or Seal		

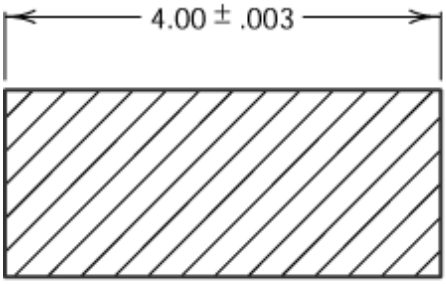
S.No.	Form of Weld	Sectional Representation	Symbols
12	Spot		
13	Stud		

Tolerance

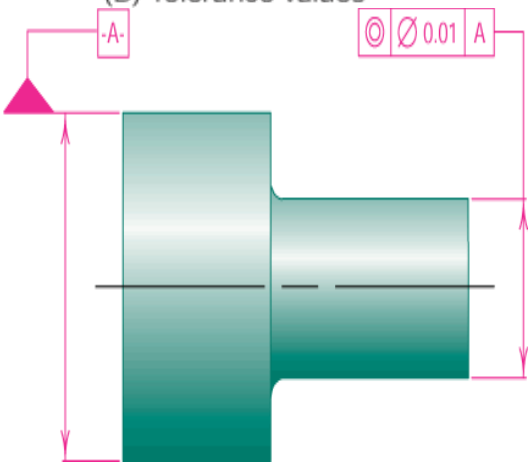


(A) Direct limits

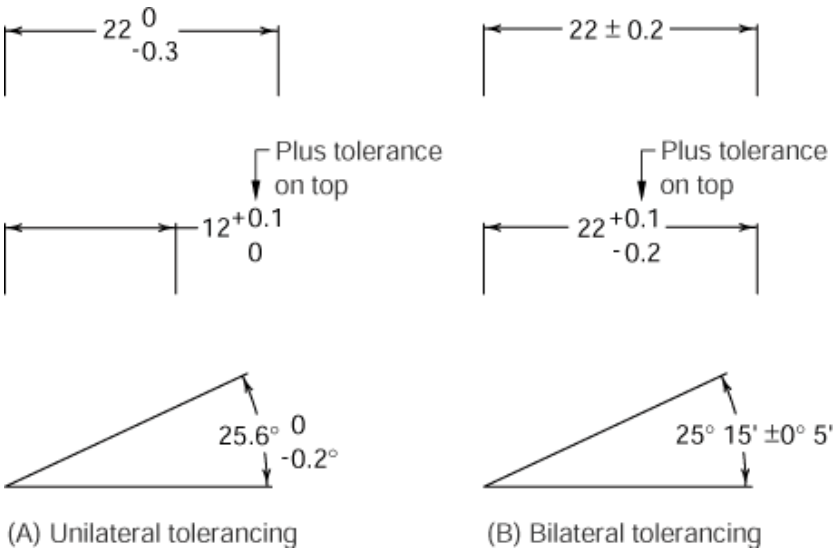
- **Tolerance** is the total amount a dimensionmay vary and is the difference between the**upper** (maximum) and **lower** (minimum) limits.
- Tolerances are used to control the amount of variation inherent in all manufactured parts. In particular, tolerances are assignedto mating parts in an assembly.
- One of the great advantages of usingtolerances is that it allows for **interchangeable parts,** thus permitting the replacement of individual parts.
- Tolerances are used in production drawings to control the manufacturingprocess more accurately and control the variation between parts.



(B) Tolerance values



- Tolerance representation
  - Direct limits or as tolerance values applied directly to a dimension.
  - Geometrictolerances
  - Notes referring tospecific condition.
  - Plus/Minus



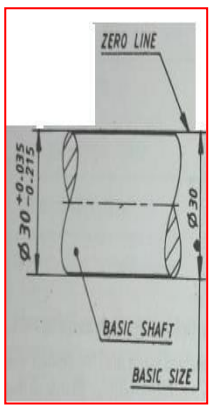
(A) Unilateral tolerancing (B) Bilateral tolerancing

Important terms

- **Limits** the maximum and minimum sizesshown by the toleranced dimension.
- **Allowance** is the minimum clearance or maximum interference between parts.
- **Tolerance** is the total variance in a dimension which is the difference between the upper and lower limits. The tolerance ofthe slot in Figure is .004" and the tolerance of the mating part is .002".

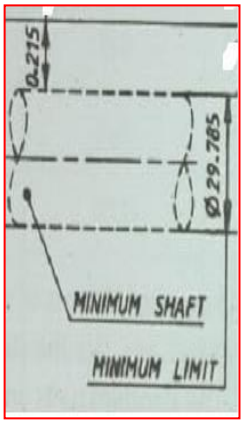
Definition of tolerance limits and fit.

- Interchangeable: A part picked at random must fit properly with its counterpart also picked randomly, and both of them must satisfy functionally.
- It is highly impossible to manufacture large number of identical parts accurately to the specified exact size economically due to the inherent limitations in men, material and machines.
- The cost of manufacture will be higher since greater care and skill is required while machining the part very close to the basic size.
- Therefore, it is inevitable to tolerate variations in the basic size so that the actual machined sizes may lie within the specified limits of variations.
- The amount of variation permitted for a basic size is called **tolerance**.
- The maximum and minimum permissible sizes within which the actual machined size lies are called **limits**.
- The functional relationship between the two adjacent parts achieved by the specified tolerances is called **fit**.

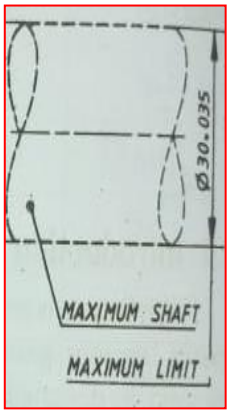


✓

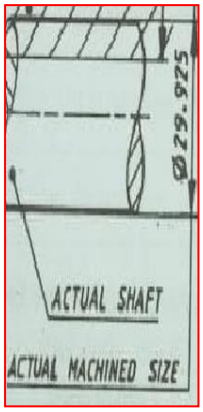
BASIC SHAFT



MINIMUM SHAFT



MAXIMUM SHAFT

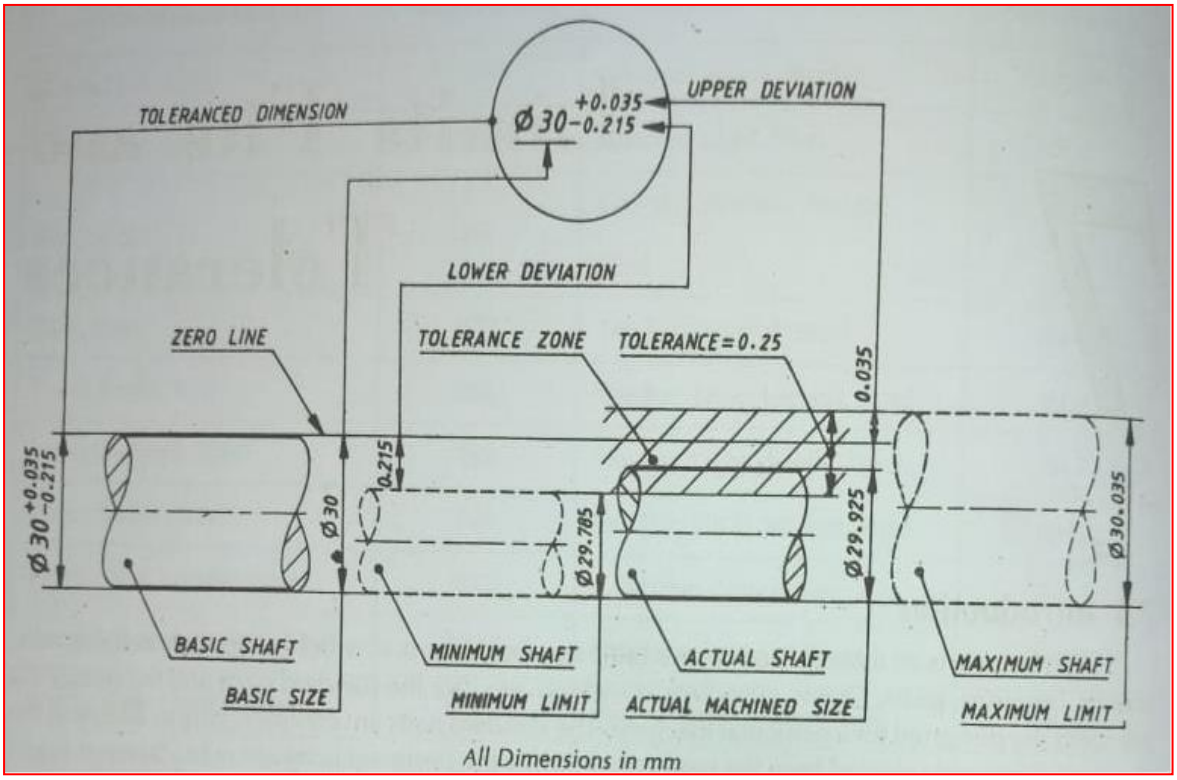


ACTUAL SHAFT

- **Basic size:** Theoretical size of a part derived from the design. The tolerances are always specified to

the basic size.

- The dimension 30 mm is the basic size.
- **Actual size:** Size actually obtained by the machining. The actual size of the shaft is ϕ29.925 mm.
- **Zero line:** Since the deviations are measured from the basic size, to indicate the deviations graphically: the basic shaft, the minimum shaft, the actual shaft and the maximum shaft are aligned at the bottom and a straight line, called zero line is drawn. Deviations above this line will be positive and below this line will be negative.



Limits: Two extreme permissible sizes between which the actual size lies.

Maximum limit: Maximum permissible size for a given basic size. From the fig, the maximum limit is ϕ30+0.035 = ϕ30.035 mm.

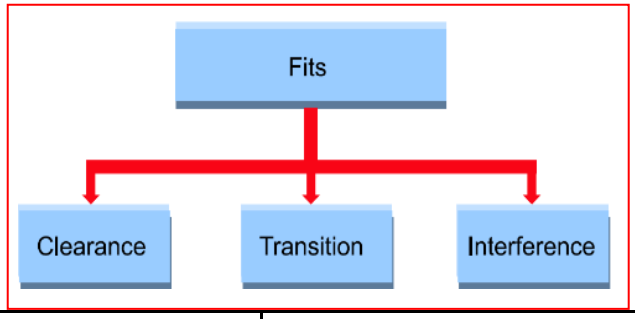
Minimum limit: Minimum permissible size for a given basic size. From the fig, the minimum limit is ϕ30-0.215 = ϕ29.785 mm.

Tolerance: Amount of variation permitted to a basic size. It is the **difference between the maximum andminimum limits of a basic size**.

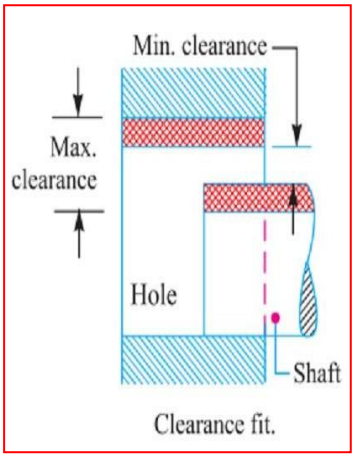
FITS

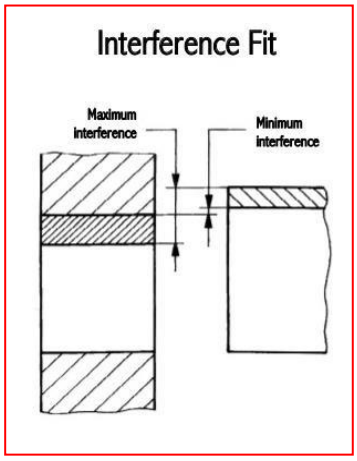
- The functional relationship between the two adjacent parts achieved by the specified tolerances is called **fit**.
- It is the relationship existing between the mating surfaces of the parts because of the differences in theirdimensions.
- It may be defined as the degree of tightness and looseness between two mating parts.
- In general, a **rod of circular section** and a **circular hole** are termed as ***shaft*** and ***hole*** respectively.
- For the sake of simplicity: even the non circular sections and the space containing or contained by two parallel faces of any part such as, thickness of a key and the width of the keyway or a slot, are also referred as***shaft*** and ***hole*** respectively.

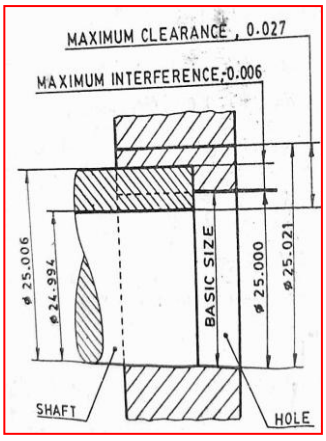
TYPES OF FIT



TYPE OF FIT	HOLE	SHAFT	CLEARANCE
Clearance fit	Smallest possible hole	Largest possible shaft	Positive
Interference fit	Largest possible hole	Smallest possible shaft	Negative
Transition fit	Largest possible hole	Smallest possible shaft	Positive
	Smallest possible hole	Largest possible shaft	Negative







CLEARANCE FIT

- In clearance fit, a clearance exists between the shaft and hole.
- It is defined as the fit established, when a **positive clearance** exists between the hole and the shaft.
- In this, the **clearance (i.e., positive clearance)** is due to the difference between the dimensions of the ***smallest possible hole*** and the ***largest possible shaft***.

TYPES OF CLEARANCE FIT

- **Loose Fit**

It is used between those mating parts where no precision is required. It provides minimum allowance and isused on loose pulleys, agricultural machineries etc.

- **Running Fit**

For a running fit, the dimension of shaft should be smaller enough to maintain a film of oil for lubrication. It isused in bearing pair etc.,

- **Slide Fit or Medium Fit**

It is used on those mating parts where great precision is required. It provides medium allowance and is used intool slides, slide valve, automobile parts, etc.

INTERFERENCE FIT

- In interference fit, a negative clearance exists between the shaft and hole.
- It is defined as the fit established, when a **negative clearance** exists between the hole and the shaft.
- In this, the **interference of the surfaces (i.e., negative clearance)** is due to the difference between the dimensions of the ***largest possible hole*** and the ***smallest possible shaft***.

TYPES OF INTERFERENCE FIT

- **Shrink Fit or Heavy Force Fit**

It refers to maximum negative allowance. In assembly of the hole and the shaft, the hole is expanded by heating and then rapidly cooled in its position. It is used in fitting of rims etc.,

- **Medium Force Fit**

These fits have medium negative allowance. Considerable pressure is required to assemble the

hole and the shaft. It is used in car wheels, armature of dynamos etc.,

- **Tight Fit or Force Fit**

One part can be assembled into the other with a hand hammer or by light pressure. A slight negative allowance exists between two mating parts. It gives a semi permanent fit and is used on a keyed pulley and shaft, rocker arm, etc.,

TRANSITION FIT

- In transition fit, a positive or a negative clearance exists between the shaft and hole.
- It is defined as the fit established, when the dimensions of the hole and the shaft are such that there exists a **positive clearance or** a **negative clearance** when the shaft is fitted into the hole.
- **Positive clearance** when the ***smallest possible shaft*** if fitted into the **largest possible hole**.

TYPES OF TRANSITION FIT

- **Push Fit or Snug Fit**

It refers to zero allowance and a light pressure is required in assembling the hole and the shaft. The moving parts show least vibration with this type of fit.

- **Force Fit or Shrink Fit**

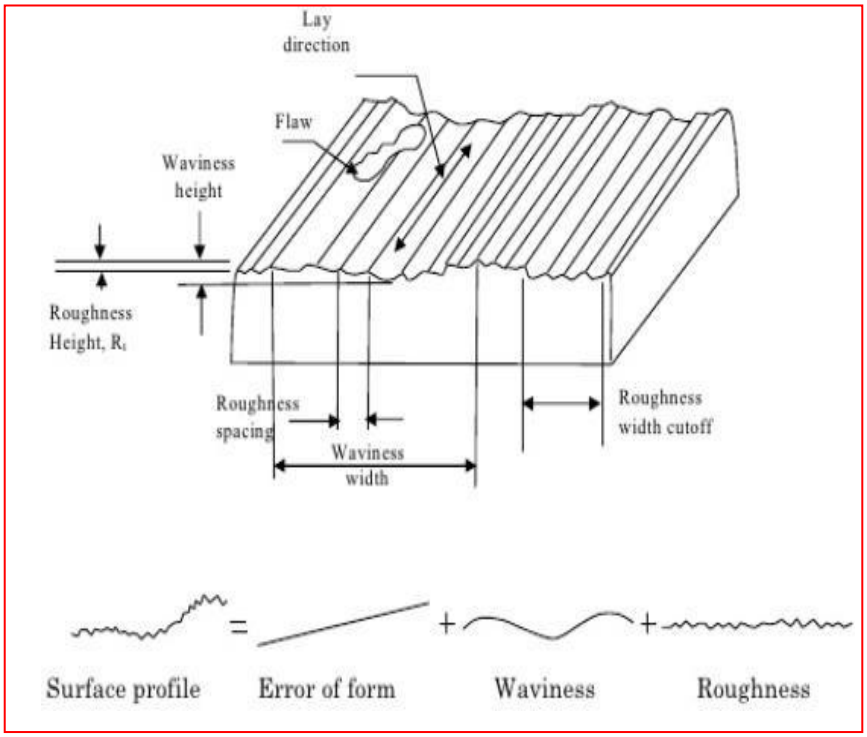
A force fit is used when the two mating parts are to be rigidly fixed so that one cannot move without the other.It either requires high pressure to force the shaft into the hole or the hole to be expanded by heating. It is used in railway wheels, etc.,

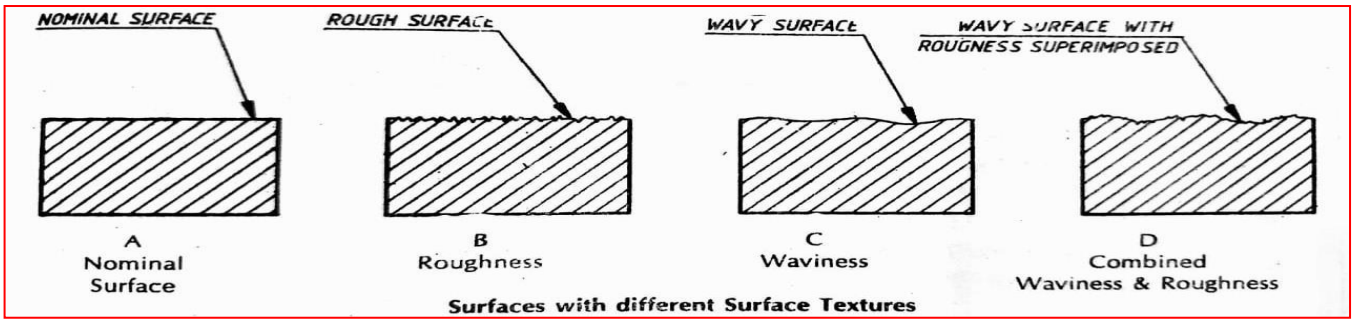
- **Wringing Fit**

A slight negative allowance exists between two mating parts in wringing fit. It requires pressure to force the shaft into the hole and gives a light assembly. It is used in fixing keys, pins, etc.

ROUGHNESS

- ✓ It is defined as the closely spaced, irregular deviation on a scale smaller than that of waviness.
- ✓ It is the micro irregularity of a surface produced by cutting tool or the abrasive grain action and the machine feed.
- ✓ Roughness may be superimposed on waviness.





Surfaces with different Surface Textures

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Unit 5

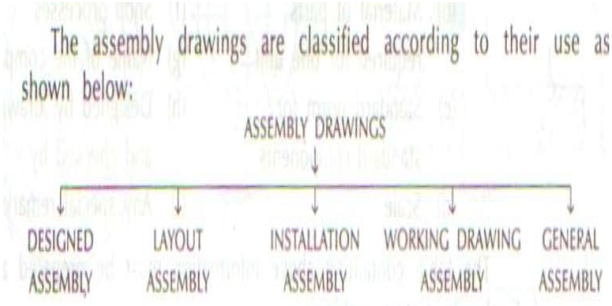
Assembly drawings

Introduction

• A drawing which displays the parts of a machine or a machine unit assembled in their relative working positions is known asassembly drawing.

• The assembly drawing would be such that it should satisfy: (i) Manufacturing requirements (ii) Operational requirements

(iii) Maintenance requirements.



Types of assembly Drawing

• Designed assembly: This assembly drawing is prepared at the design-stage on a larger scale. When a machine is designed, an assembly drawing or a design layout is first drawn to clearly visualize the performance, shape and clearances of various parts comprising the machine.

• Layout assembly: This is an assembly drawing showing how the parts are assembled with their basic proportions (dimensions). Installation assembly: This is prepared for the installation or erection of a machine. This is also sometimes known as an outline assembly. On this drawing, the location and dimensions of few important parts and overall dimensions of the assembled unit are indicated. This drawing provides useful information for assembling the machine, as this drawing reveals all parts of a machine in their correct working position.

• Working drawing assembly: A complete set of working drawings of a machine comprises of detailed drawings, giving all necessary information for the production of individual parts and assembly drawing showing the location of each part. The assembly drawing should be ready before the detailed drawings are acceptedas finished and the blue-prints are made.

• General assembly: It comprises of the detailed drawings of the individual parts, sub-assembly and the assembly drawings of the machine

• Detailed Assembly Drawing It is usually made for simple machines, comprising of a relatively

smaller number of simple parts. All the dimensions and information necessary for the construction of such parts and for the assembly of the parts are given directly on the assembly drawing. Separate views of specific parts in enlargements, showing the fitting of parts together, may also be drawn in addition to the regular assembly drawing.

• Sub- Assembly Drawing Many assemblies such as an automobile, lathe, etc., are assembled with many pre-assembled components as well as individual parts. These pre-assembled units are known as sub-assemblies. A sub-assembly drawing is an assembly drawing of a group of related parts, that form a part in a more complicated machine. Examples of such drawings are: lathe tail-stock, diesel engine fuel pump, carburettor, etc.

• Assembly Drawings for catalogues Special assembly drawings are prepared for company catalogues. These drawings show only the pertinent details and dimensions that would interest the potential buyer. Fig. 1.4 shows a typical catalogue drawing, showing the overall and principal dimensions.

• Exploded Assembly Drawing In some cases, exploded pictorial views are supplied to meet instruction manual requirements. These drawings generally find a place in the parts list section of a company instruction manual.

Norms to be observed in preparingassembly drawings

(i) Selection of views: The main or importantview which is usually in section should show all the individual parts and their relative locations. Additional views are shown only when they add necessary information.

(ii) Sectioning: The parts should be sectioned according to the requirements (i.e. half-sectionor partial section) to show important assembly details. Code of the BIS (SP:46-1988) for general engineering drawings must be observed

(iii) Dotted lines: The dotted lines should be omitted fromthe assembly drawing when a proper section is taken. If the view of a part is drawn by the half-section, then in un section portion of the view, the dotted lines may be drawn to clarify details of the part.

(iv) Dimensions: The overall dimensions and centre-to- centre distances showing the relationship of parts to the machine as a whole, are sometimes shown.

(V) Detailed dimensions are given on working assembly drawings when the detailed drawings are not prepared.

Sequences of preparing the assembly drawing

(i) Study functional requirements of each component and their inter relationship.

Learn the actual working of a machine.

(ii) Study carefully the views of each component in the detail drawing and decide the relative location of each part for the proper functioning ofthe machine.

(iii) Decide the mating dimensions between two components which are required to be assembled.

iv) Prepare free-hand sketch of the main view or an important view (generally front-elevation). Add additionalviews, if necessary.

(v) Select a suitable scale for the entire assemblydrawing.

(vi) Lay out the views of the assembly drawing so that itbecome easier to understand.

(vii) Prepare the bill of materials.

(viii) Label each component by the leader-line andnumber it.

(ix) Show overall dimensions.

(x) Draw the section-lines according to the convention

(xi) Show required fits and tolerances between the twomating components.

Bill of Materials

Bill of materials: Each part of the machine is identified on assembly drawing by the leader line and number, which are used in the detail drawing and in the bill of material. The height of the number may be approximately 5 mm and encircled by 9 mm diameter. Leader lines are drawn radially touching the respective parts.

The bill of materials also shows the following:

(a) Number of parts (e) Method of projection

(b) Material of parts (f) Shop processes required for one unit

(c) Standard norm for (g) Name of the company

(d) Scale (h) Designed by, drawn bystandard components and checked by

(i) Any special remark.

The table containing above information must be prepared as shownin the illustrative problems.

Suggested approach

Preparing an assembly form exploded view is easy task as clue to the position and sequence is available

For preparing from orthographic view ofthe individual components some skill is needed

The suggested approach is

- Functional Matching or Mapping

- Geometrical mapping
- Dimensional mapping

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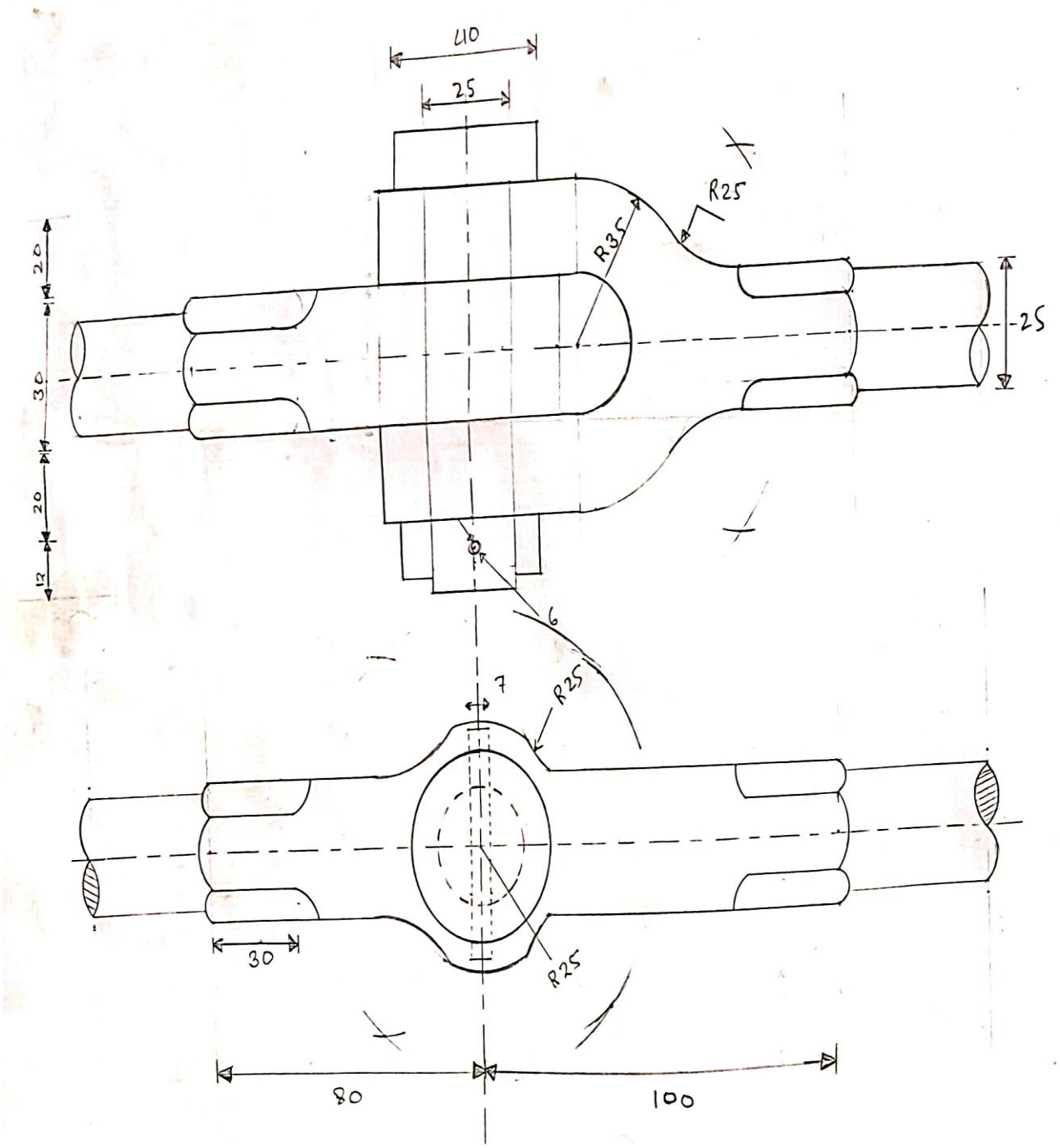
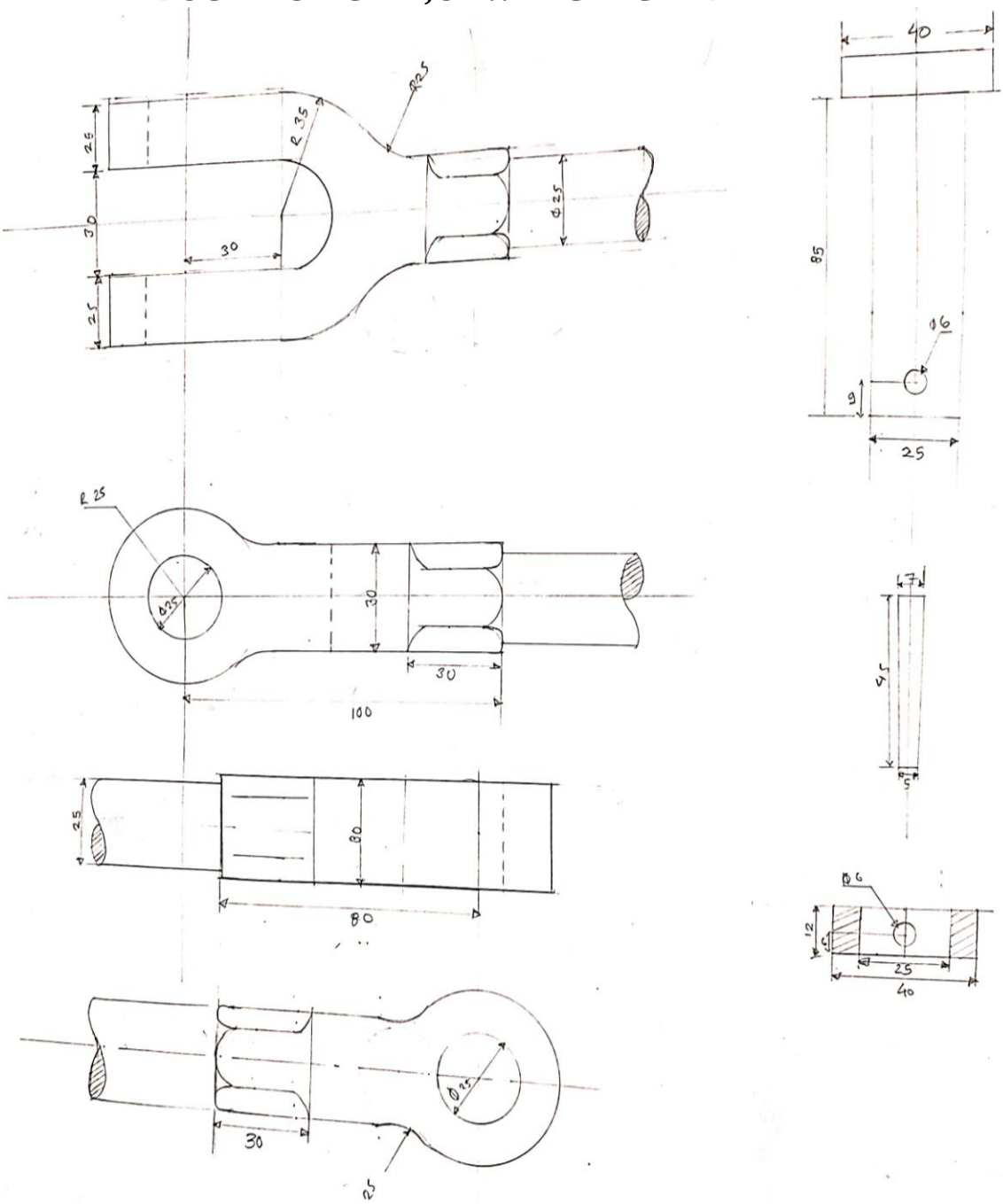
Q1. Draw Details from given assembly drawing. And also prepare Bill of material.



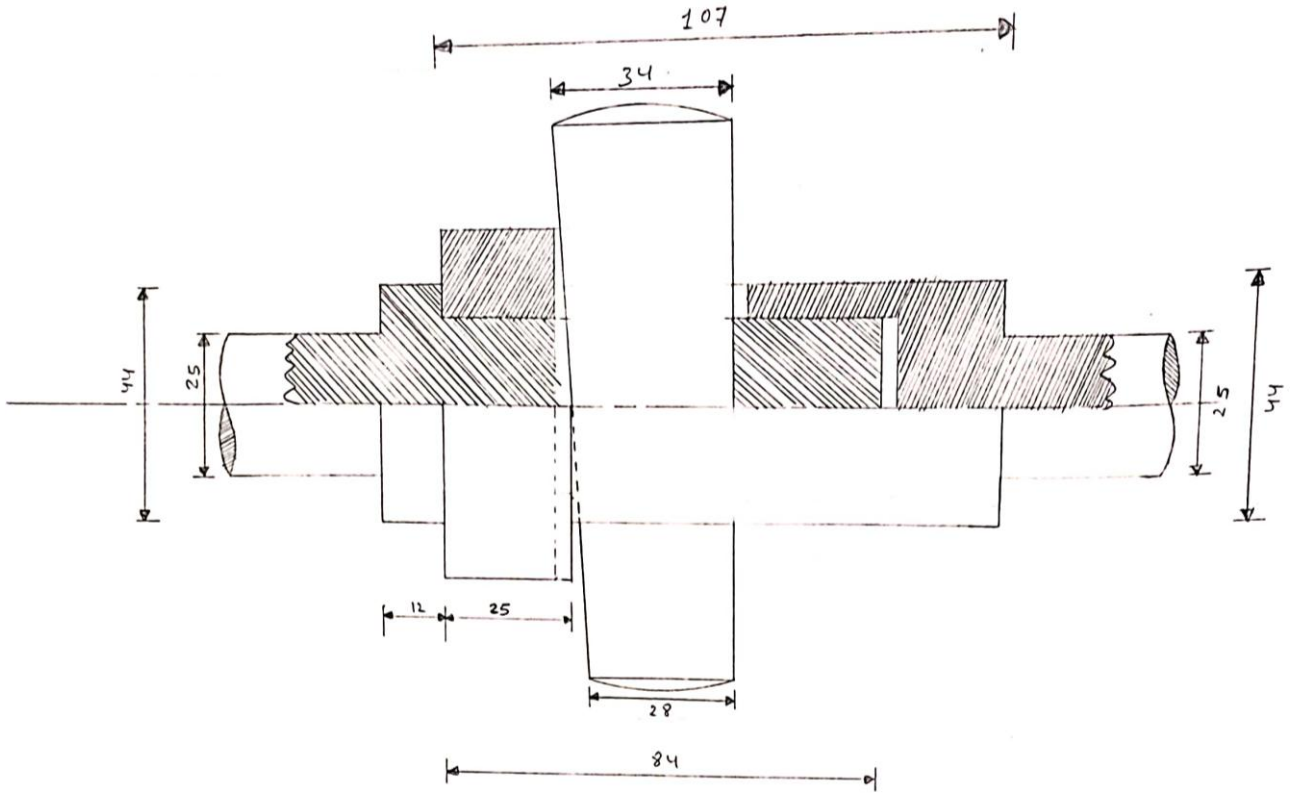
Ans.

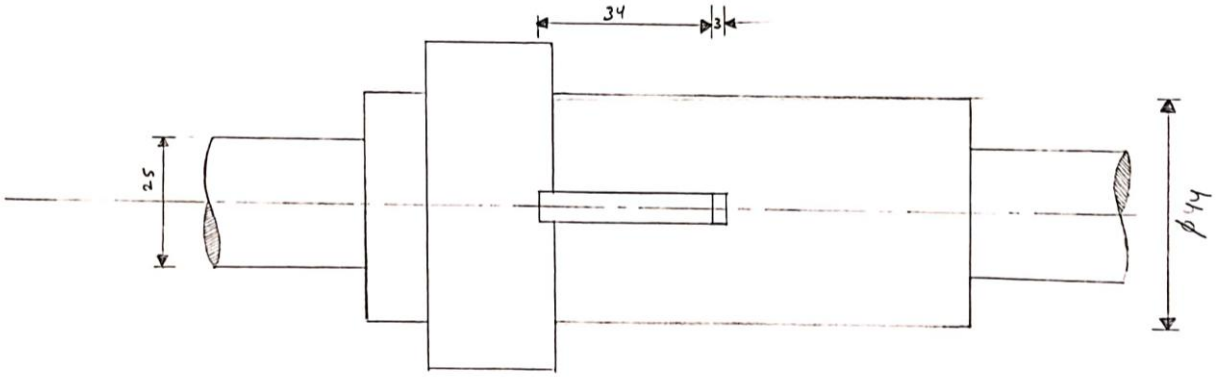


BILL OF MATERIALS

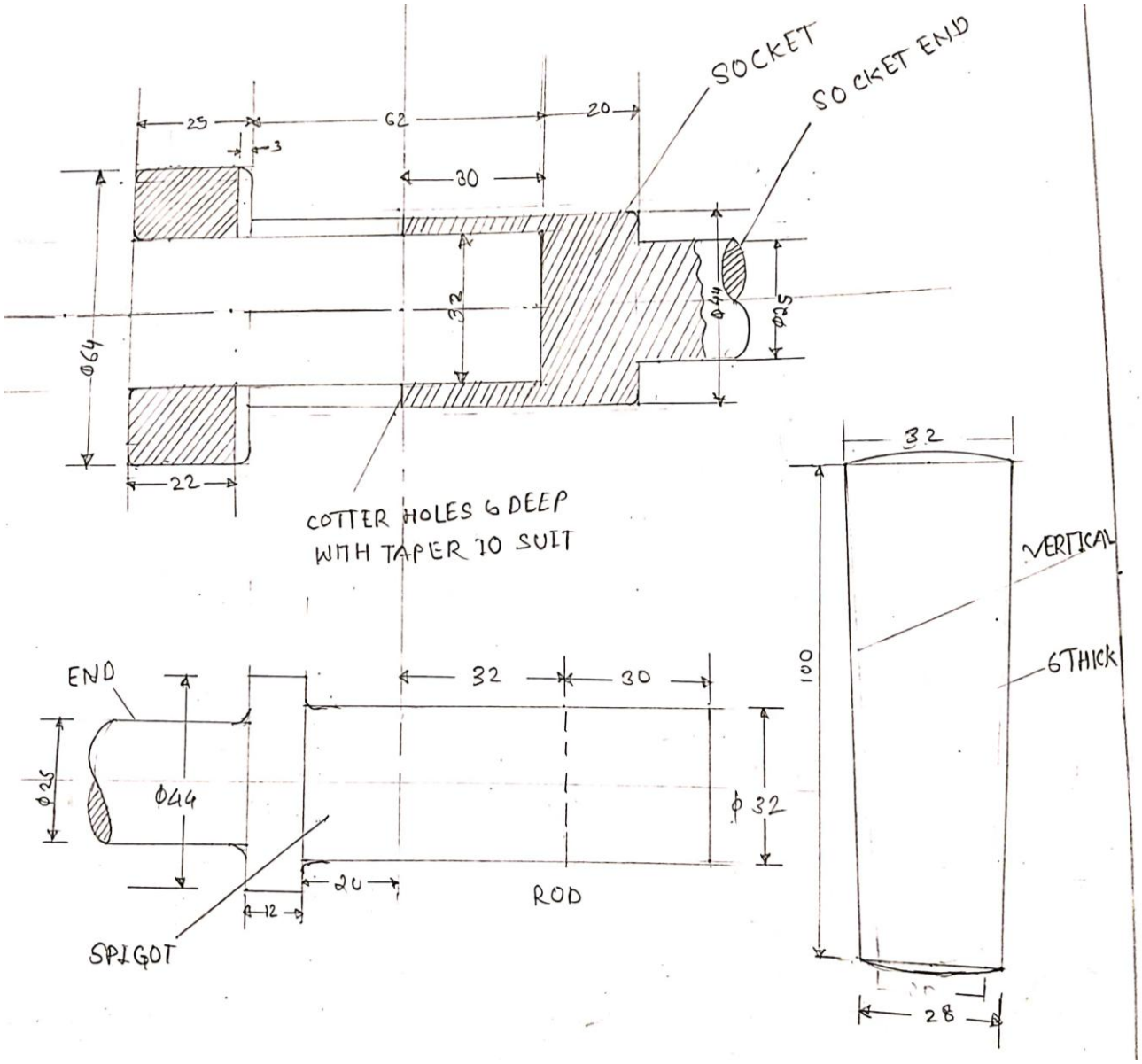
PART NO.	Description	Material	quantity
01	Fork	Structural steel	1
02	Eye	structural steel	1
03	Knuckle pin	structural steel	1
04	Colter	Structural steel	1
05	Taper pin	Structural steel	1

Q. 2. Draw Details from given assembly drawing

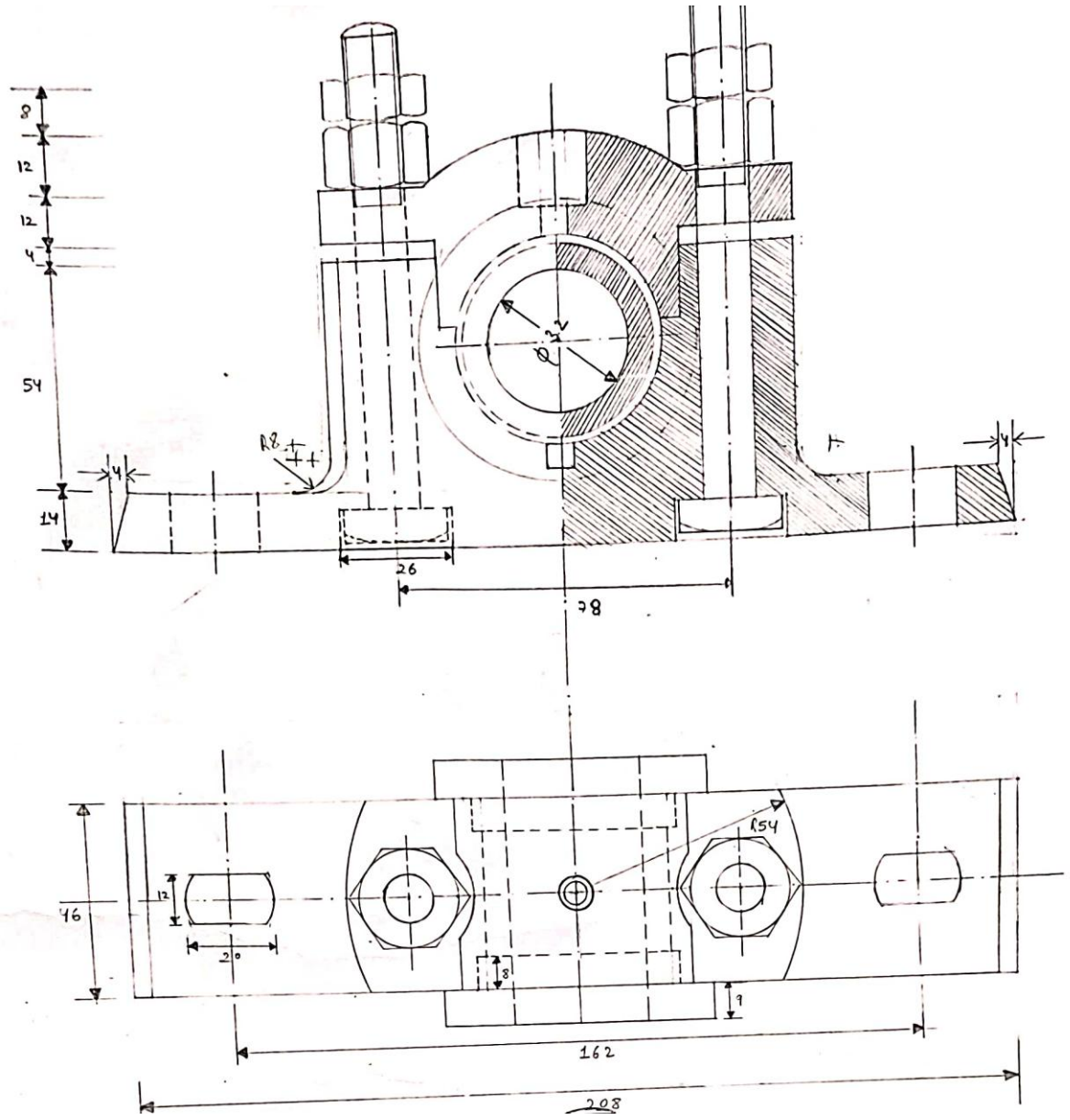




Ans :


SOCKET
SOCKET END
25
62
20
3
30
32
Φ44
Φ25
Φ64
22
COTTER HOLES 6 DEEP
WITH TAPER TO SUIT
32
VERTICAL
6THICK
100
END
32
30
Φ25
Φ44
Φ32
20
12
ROD
SPIGOT
28

Q. 3. Draw Details from given assembly drawing



Ans:

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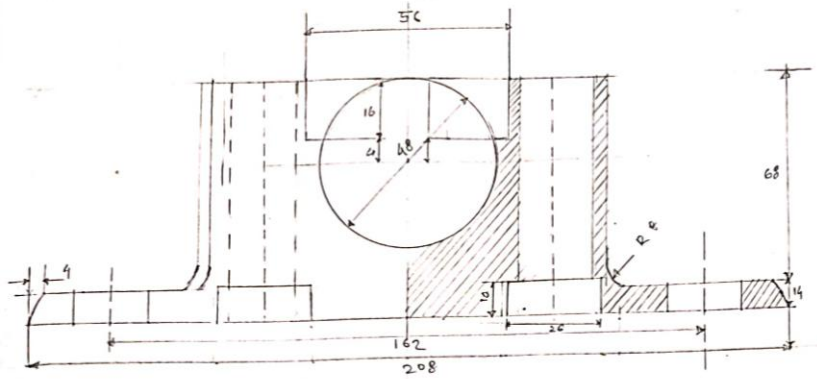
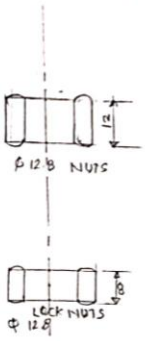
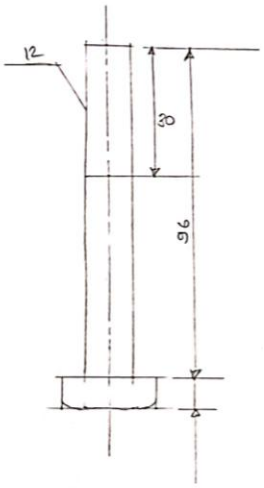
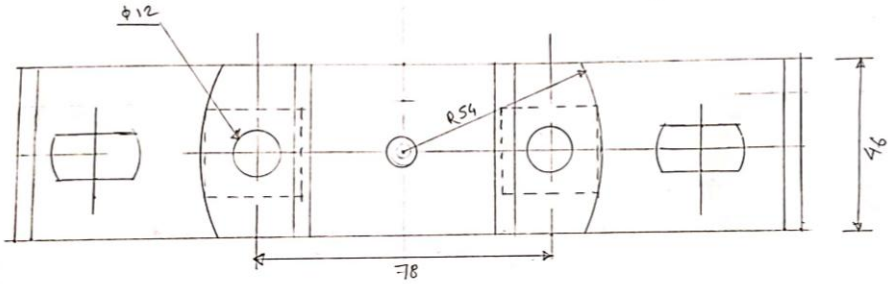
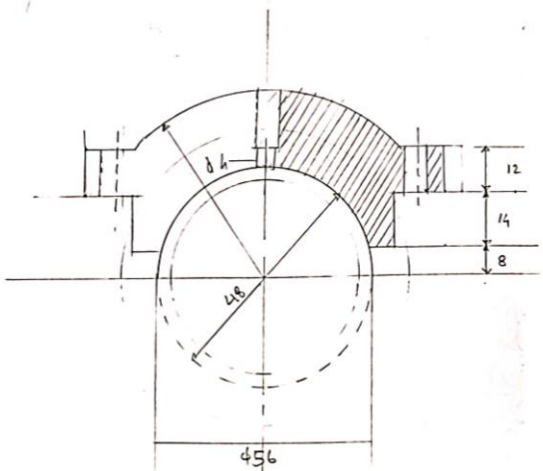
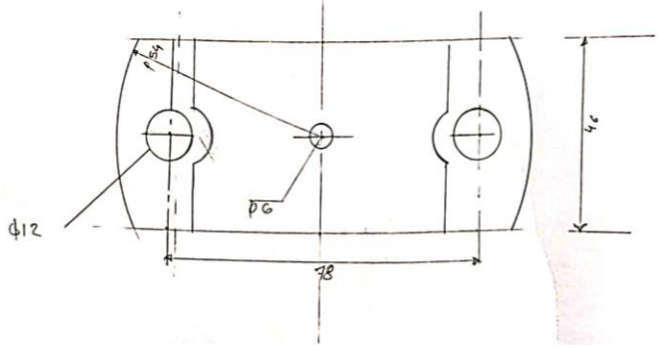




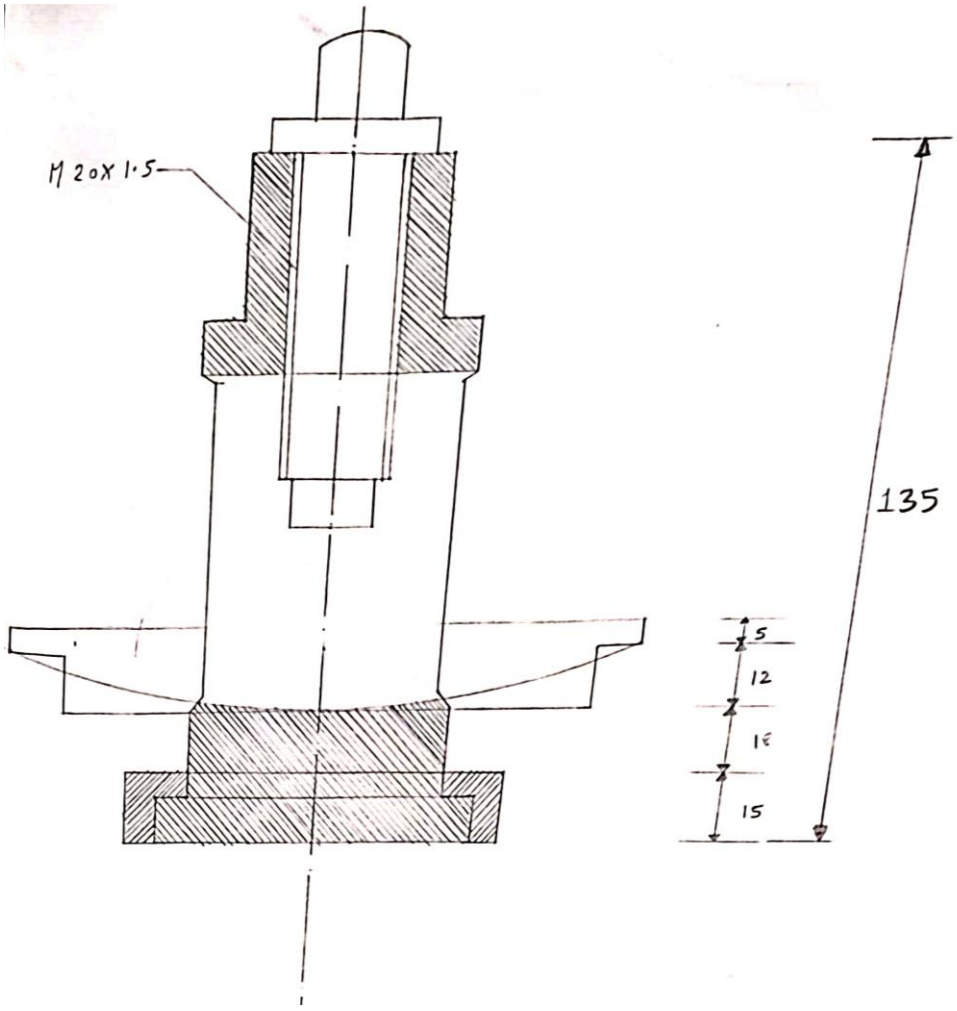




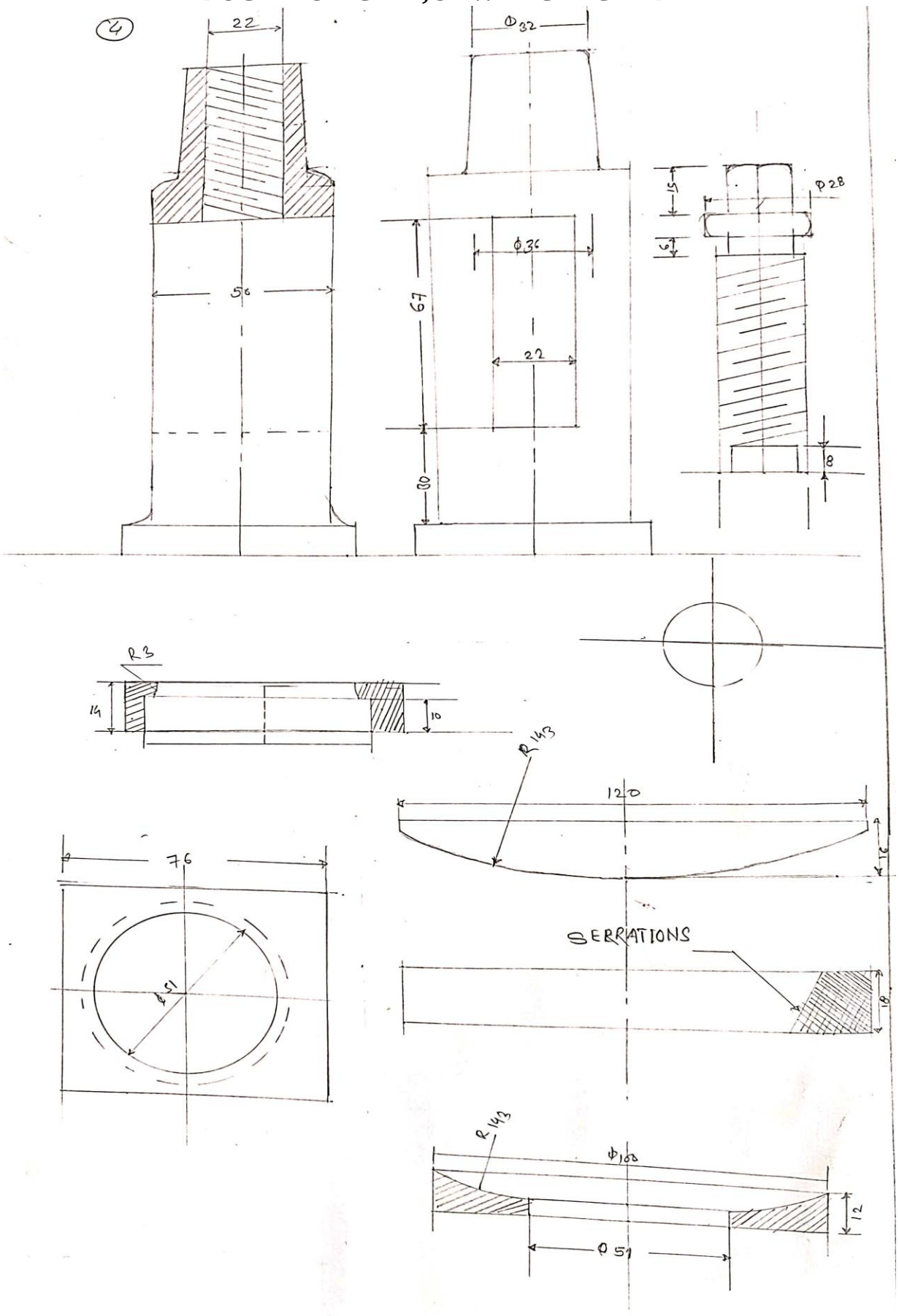




Q.4. Draw Details from given assembly drawing



Ans

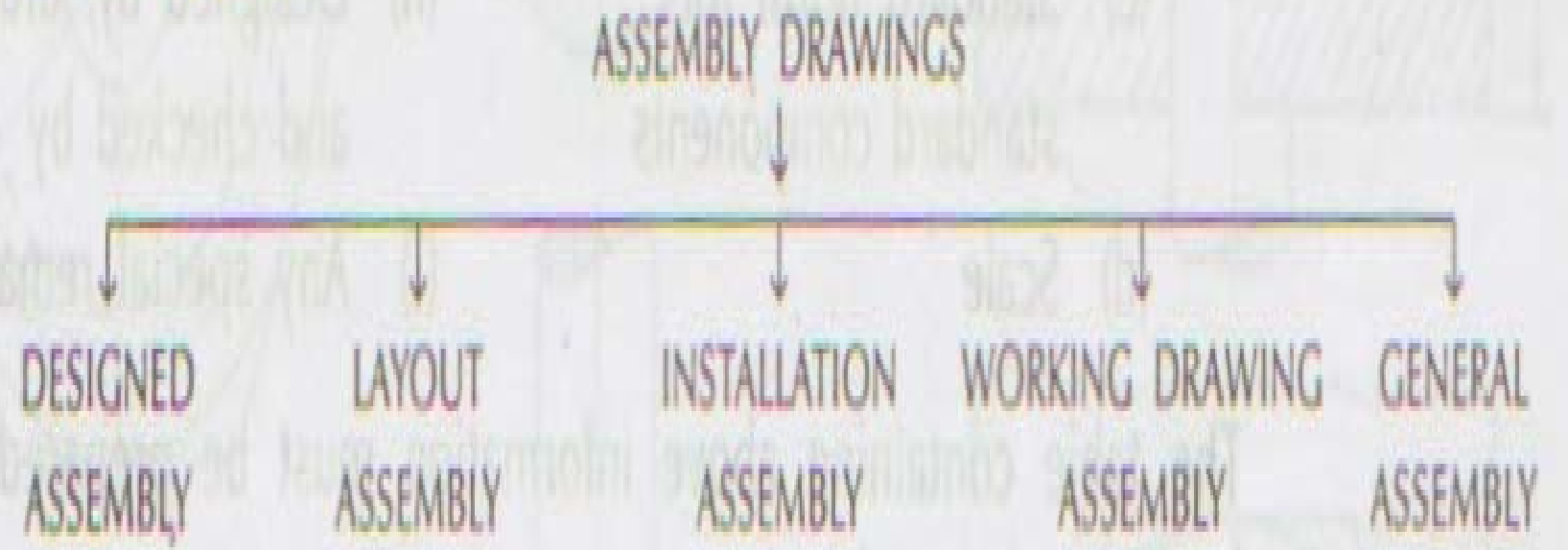

4
22
Φ32
56
67
Φ36
22
80
15
6
Φ28
8
R3
14
10
R143
120
31
76
Φ51
SERRATIONS
18
R143
Φ168
12
Φ51

Assembly drawings

Introduction

- A drawing which displays the parts of a machine or a machine unit assembled in their relative working positions is known as assembly drawing.
- The assembly drawing would be such that it should satisfy: (i) Manufacturing requirements (ii) Operational requirements (iii) Maintenance requirements.

The assembly drawings are classified according to their use as shown below:



- Designed assembly: This assembly drawing is prepared at the design-stage on a larger scale.
- Layout assembly: This is an assembly drawing showing how the parts are assembled with their basic proportions (dimensions).
- Installation assembly: This is prepared for the installation or erection of a machine. This is also sometimes known as an outline assembly.

- Working drawing assembly: A complete set of working drawings of a machine comprises of detailed drawings, giving all necessary information for the production of individual parts and assembly drawing showing the location of each part. The assembly drawing should be ready before the detailed drawings are accepted as finished and the blue-prints are made.
- General assembly: It comprises of the detailed drawings of the individual parts, sub-assembly and the assembly drawings of the machine

.

Norms to be observed in preparing assembly drawings

- (i) Selection of views: The main or important view which is usually in section should show all the individual parts and their relative locations. Additional views are shown only when they add necessary information.
- (ii) Sectioning: The parts should be sectioned according to the requirements (i.e. half-section or partial section) to show important assembly details. Code of the BIS (SP:46-1988) for general engineering drawings must be observed

- (iii) Dotted lines: The dotted lines should be omitted from the assembly drawing when a proper section is taken. If the view of a part is drawn by the half-section, then in un section portion of the view, the dotted lines may be drawn to clarify details of the part.
- (iv) Dimensions: The overall dimensions and centre-to-centre distances showing the relationship of parts to the machine as a whole, are sometimes shown.
- (V) Detailed dimensions are given on working assembly drawings when the detailed drawings are not prepared.

Bill of Materials

- Bill of materials: Each part of the machine is identified on assembly drawing by the leader line and number, which are used in the detail drawing and in the bill of material. The height of the number may be approximately 5 mm and encircled by 9 mm diameter. Leader lines are drawn radially touching the respective parts.
- The bill of materials also shows the following:
- (a) Number of parts
- (b) Material of parts required for one unit
- (c) Standard norm for standard components
- (d) Scale
- (e) Method of projection
- (f) Shop processes
- (g) Name of the company
- (h) Designed by, drawn by and checked by
- (i) Any special remark.
- The table containing above information must be prepared as shown in the illustrative problems.

Suggested approach

- Preparing an assembly form exploded view is easy task as clue to the position and sequence is available
- For preparing from orthographic view of the individual components some skill is needed

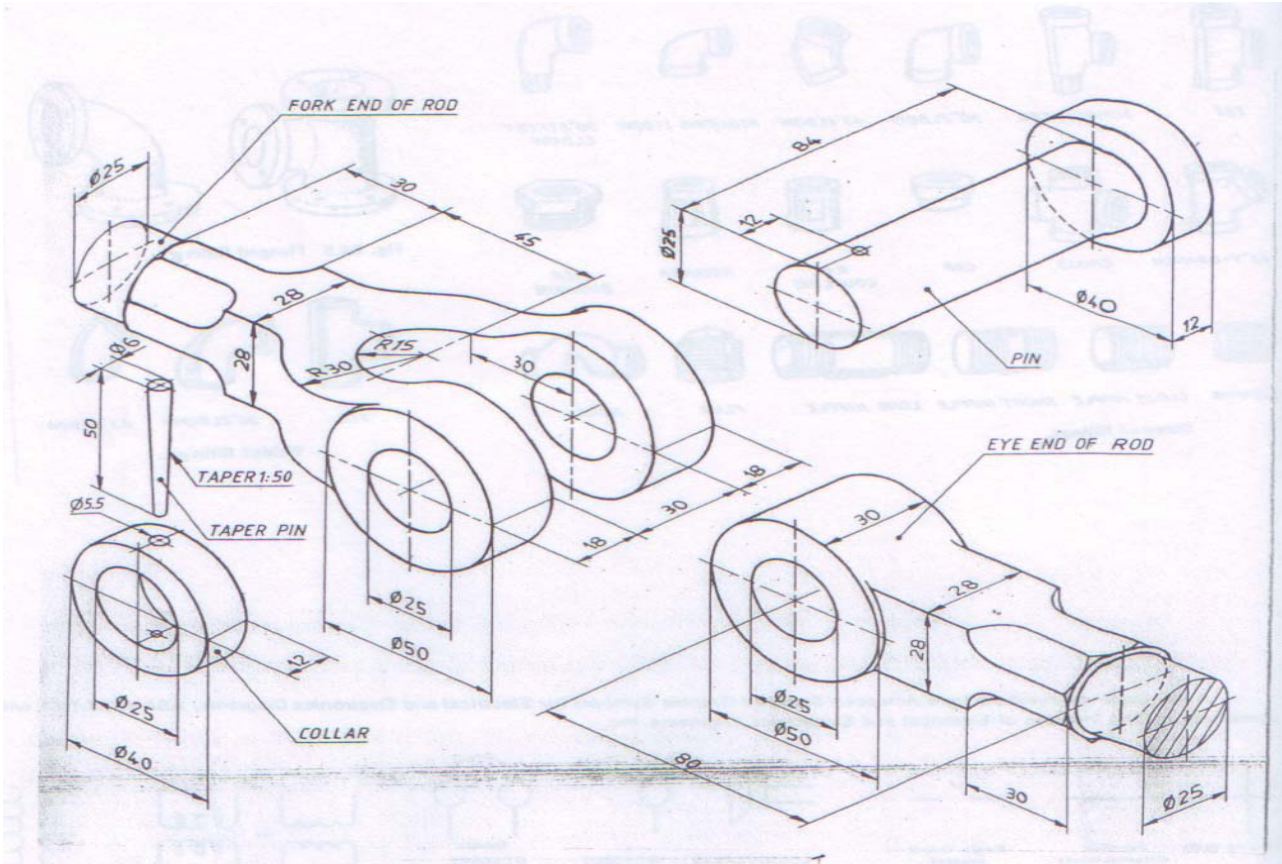


Fig. 4.1 Knuckle Joint

The suggested approach is

- Functional Matching or Mapping
- Geometrical mapping
- Dimensional mapping

Sequences of preparing the assembly drawing

- (i) Study functional requirements of each component and their inter relationship.
- Learn the actual working of a machine.
- (ii) Study carefully the views of each component in the detail drawing and decide the relative location of each part for the proper functioning of the machine.
- (iii) Decide the mating dimensions between two components which are required to be assembled.

- iv) Prepare free-hand sketch of the main view or an important view (generally front-elevation). Add additional views, if necessary.
- (v) Select a suitable scale for the entire assembly drawing.
- (vi) Lay out the views of the assembly drawing so that it become easier to understand.
- (vii) Prepare the bill of materials.
- (viii) Label each component by the leader-line and number it.
- (ix) Show overall dimensions.
- (x) Draw the section-lines according to the convention
- (xi) Show required fits and tolerances between the two mating components.

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Unit 7

Sectional Views

- Sectional drawings are multi view technical drawings that contain special views of a part or parts, views that reveal interior features.
- Used to improve clarity and reveal interior features of parts.
- Interior features of complicated assemblies.
- A primary reason for creating a section view is the elimination of hidden lines, so that a drawing can be more easily understood or visualized.
- Traditional section views are based on the use of an imaginary cutting plane that cuts through the object to reveal interior features.
- This imaginary cutting plane is controlled by the designer and can

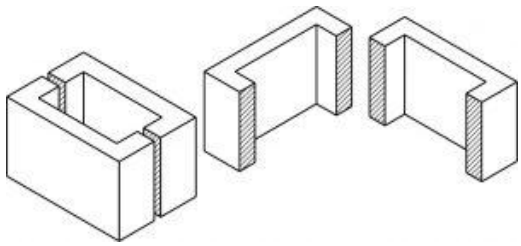
(a) go completely through the object (full section);
(b) go half-way through the object (half section);
(c) be bent to go through features that are not aligned (offset section); or
(d) go through part of the object (broken-out section).

You have learned that when making a multiview sketch, hidden edges and surfaces are usually shown with hidden (dash) lines.

When an object becomes more complex, as in the case of an automobile engine block, a clearer presentation of the interior can be made by sketching the object as it would look if it were cut apart. In that way, the many hidden lines on the sketch are eliminated.

The process of sketching the internal configuration of an object by showing it cut apart is known as sectioning. Sectioning is used frequently on a wide variety of Industrial drawings.

In this example, blocks A and B result after the block in figure has been "Sectioned". When you cut an apple in half you have sectioned it. Just as an apple can be sectioned any way you choose, so can an object in a sectional view of a drawing or sketch.

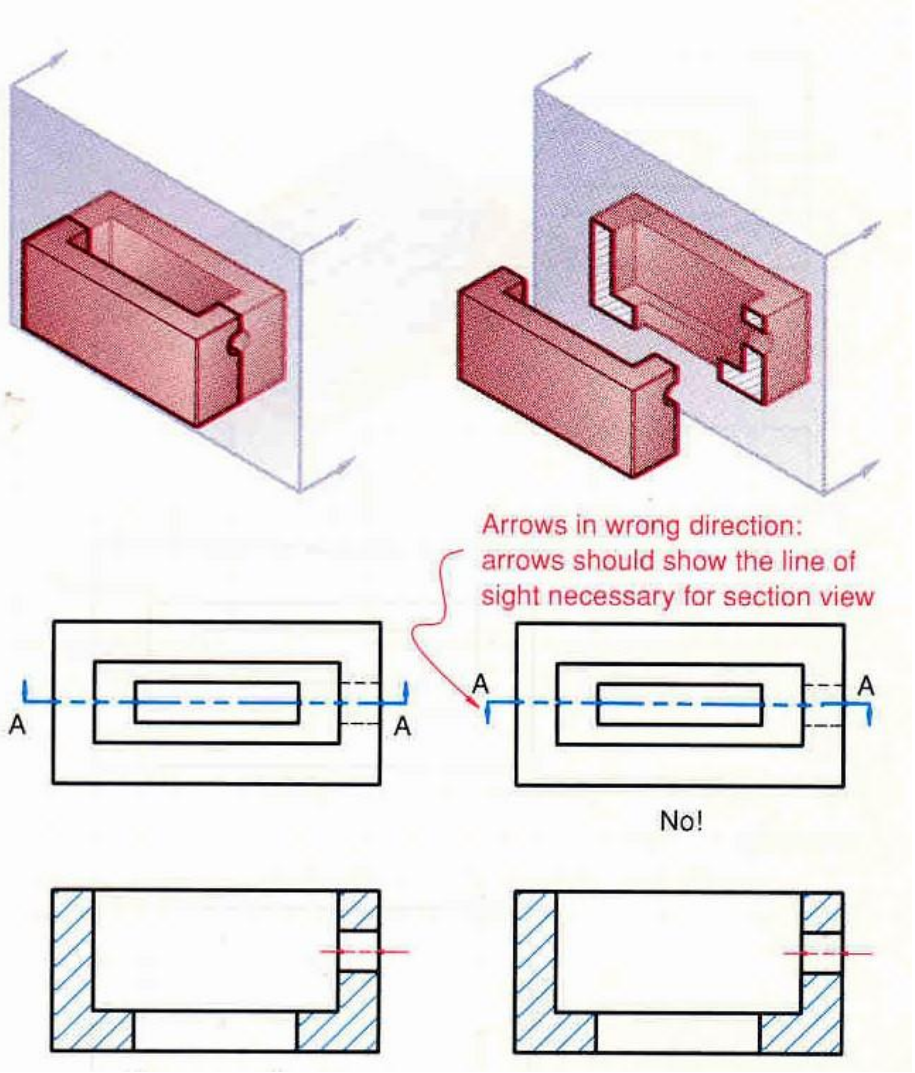


Cutting Plane

Which show where the cutting plane passes through the object, represent the *edge view* of the cutting plane and are drawn in the view(s) adjacent to the section view.

In the figure the cutting plane line is drawn in the top view, which is adjacent to the sectioned front view.

Cutting plane lines are ***thick*** **(0.7 mm) dashed lines**, that extend past the edge of the object **6 mm** and have line segments at each end drawn at **90 degrees and terminated with arrows**

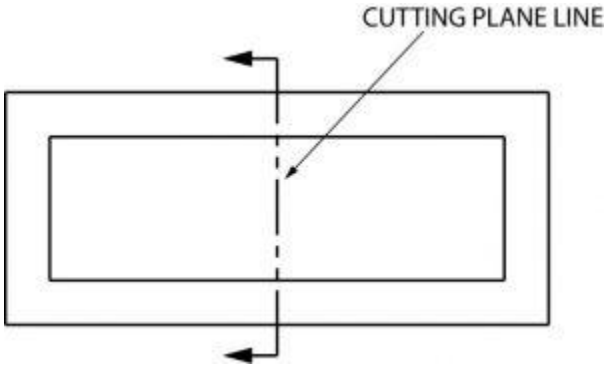


Types of Cutting Planes and TheirRepresentation

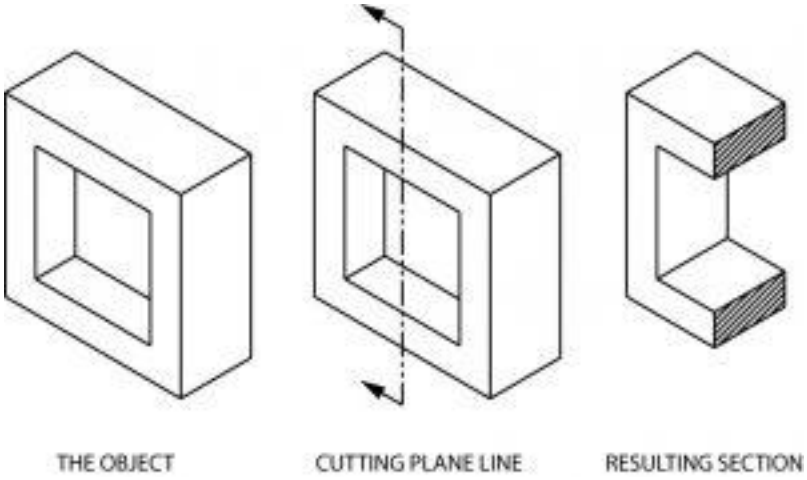
- Frontal or Vertical Cutting/ Section Plane
- Horizontal Cutting/ Section Planes
- Profile Cutting / Section Planes
- Auxiliary Section Plane
  - Auxiliary Inclined Plane (AIP)
  - Auxiliary Inclined Plane (AVP)
- Oblique Section Plane

Cutting Plane Line

A cutting plane is represented on a drawing by a cutting plane line. This is a heavy long-short-short-long kind of line terminated with arrows. The arrows in show the direction of view.



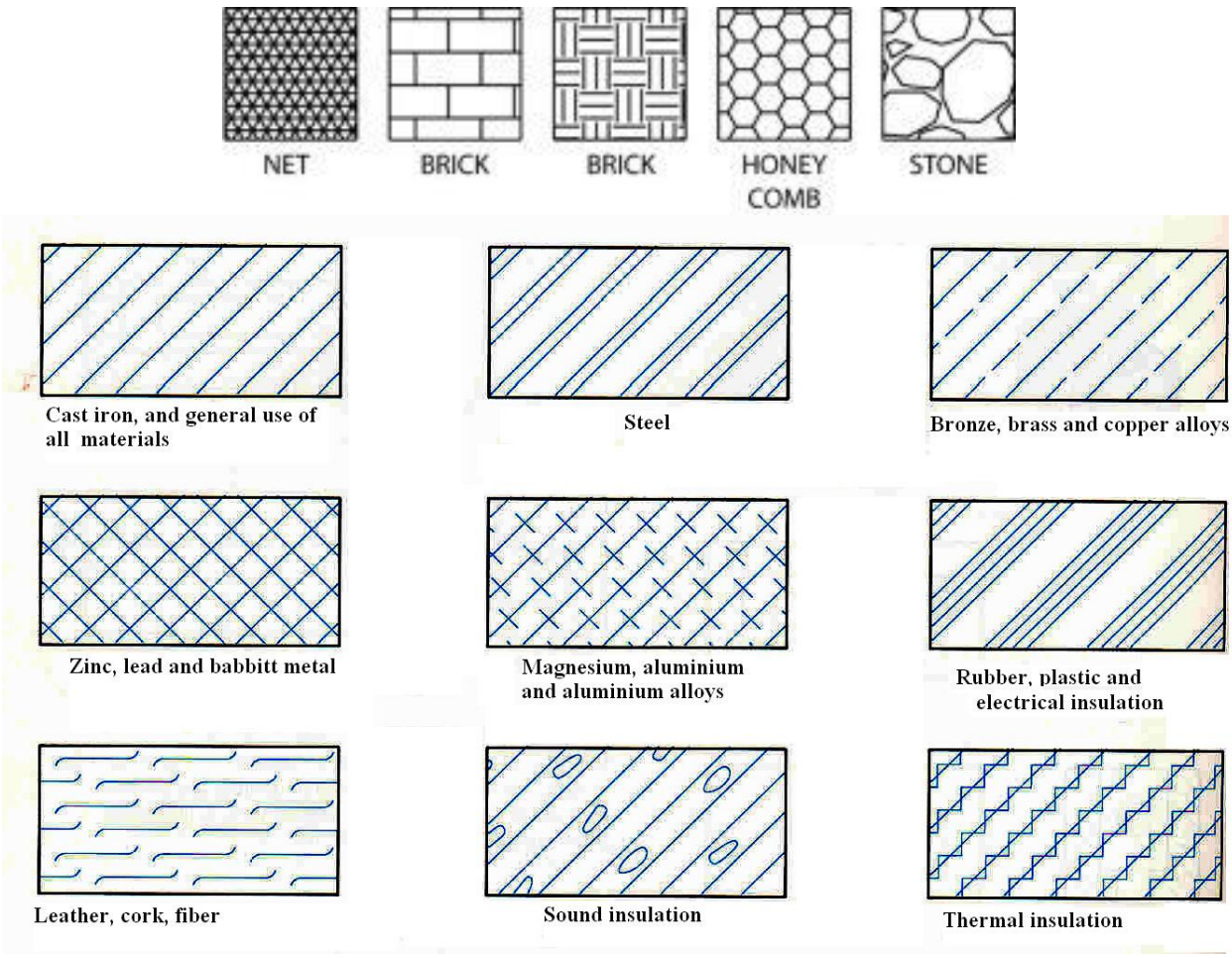
Once again, here is an graphic example of a cutting plane line and the section that develops from it.



Section Lining

The lines in the figure above, which look like saw marks, are called section lining. They are found on most sectional views, and indicate the surface which has been exposed by the cutting plane. Notice that the square hole in the object has no section lining, since it was not changed by sectioning.

Different kinds of section lining is used to identify different materials. When an object is made of a combination of materials, a variety of section lining symbols makes materials identification easier. Here are a few examples:



Section lines are very light. When sketching an object or part that requires a sectional view, they are drawn by eye at an angle of approximately 45 degrees, and are spaced about 1/8" apart. Since they are used to set off a section, they must be drawn with care.

It is best to use the symbol for the material being shown as a section on a sketch. If that symbol is not known, you may use the general purpose symbol, which is also the symbol for cast iron.

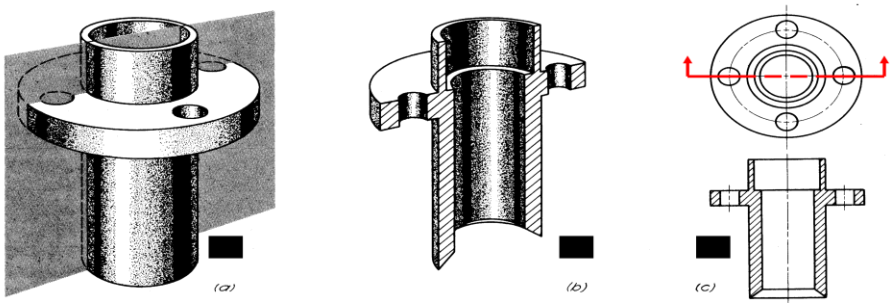
Types of Section Views

- Full sections
- Half sections
- Offset sections
- Broken-out sections
- Revolved sections
- Removed sections

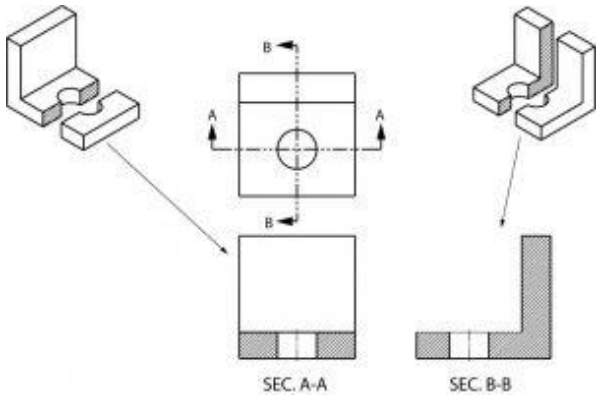
1. **Full Sections**

- In a full section view, the cutting plane cuts across the entire object
- Note that hidden lines become visible in a section view

When a cutting plane line passes entirely through an object, the resulting section is called a full section Fig illustrates a full section.



It is possible to section an object whenever a closer look intentionally is desired. Here is an object sectioned from two different directions.

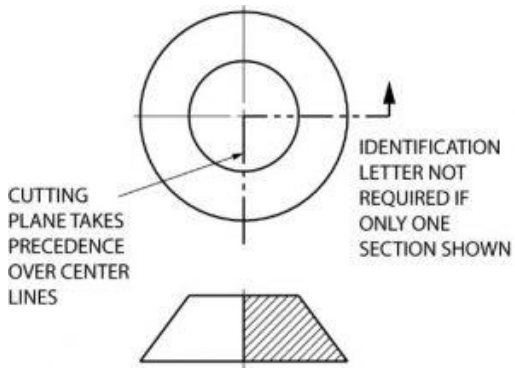


2. Half Sections

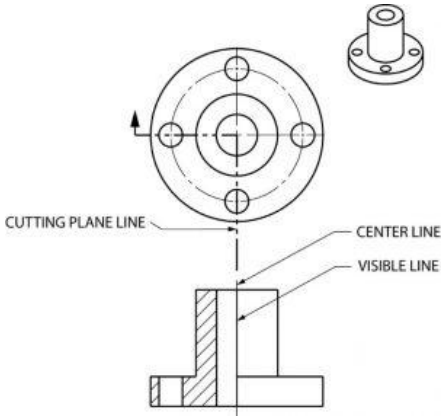
- **The cutting planes do not cut all the way through to the object.**
- **They cut only half way and intersect at the centerline.**

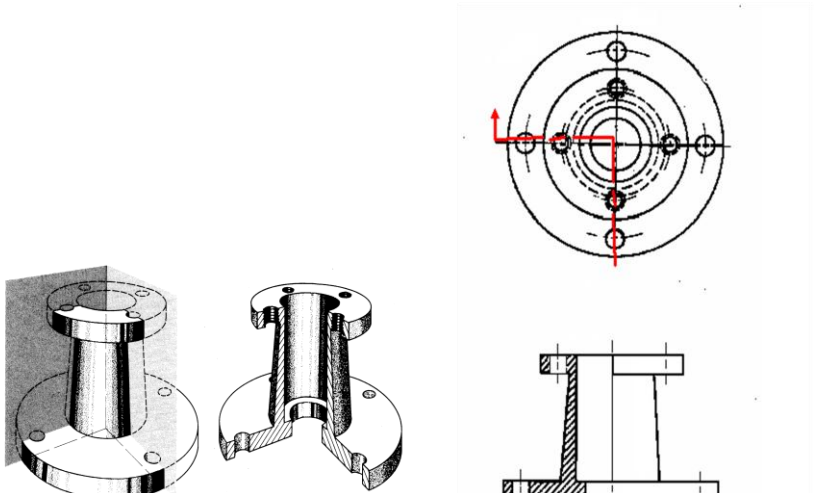
If the cutting plane is passed halfway through an object, and one-quarter of the object is removed, the resulting section is a half section. A half section has the advantage of showing both inside and outside configurations.

It is frequently used for symmetrical objects. Hidden lines are usually not shown on the un-sectioned half unless they are needed for clearness or for dimensioning purposes. As in all sectional drawings, the cutting plane take precedence over the center line.

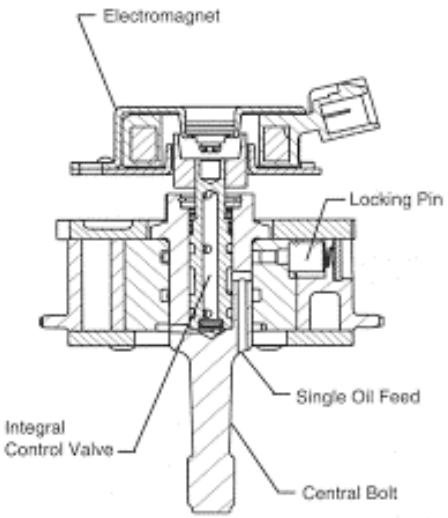


Here is another example of a half section. Remember that only one fourth of the object is removed with a half section, whereas half of the object is generally removed with a full section.





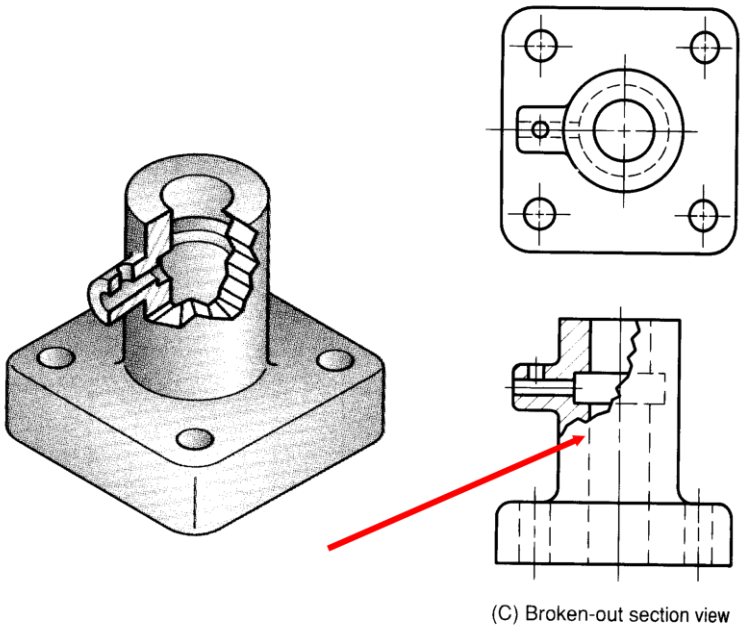
This manufacturer's drawing, using both full and half section, illustrates the advantages of sectional views. The different line directions indicate different parts and materials used in the assembly of this valve.



Section View

3. **Broken Out Sections**

In many cases only a small part of a view needs to be sectioned in order to show some internal detail. In the figure below, the broken out section is removed by a freehand break line. A cutting plane line does not need to be shown, since the location of the cut is obvious.



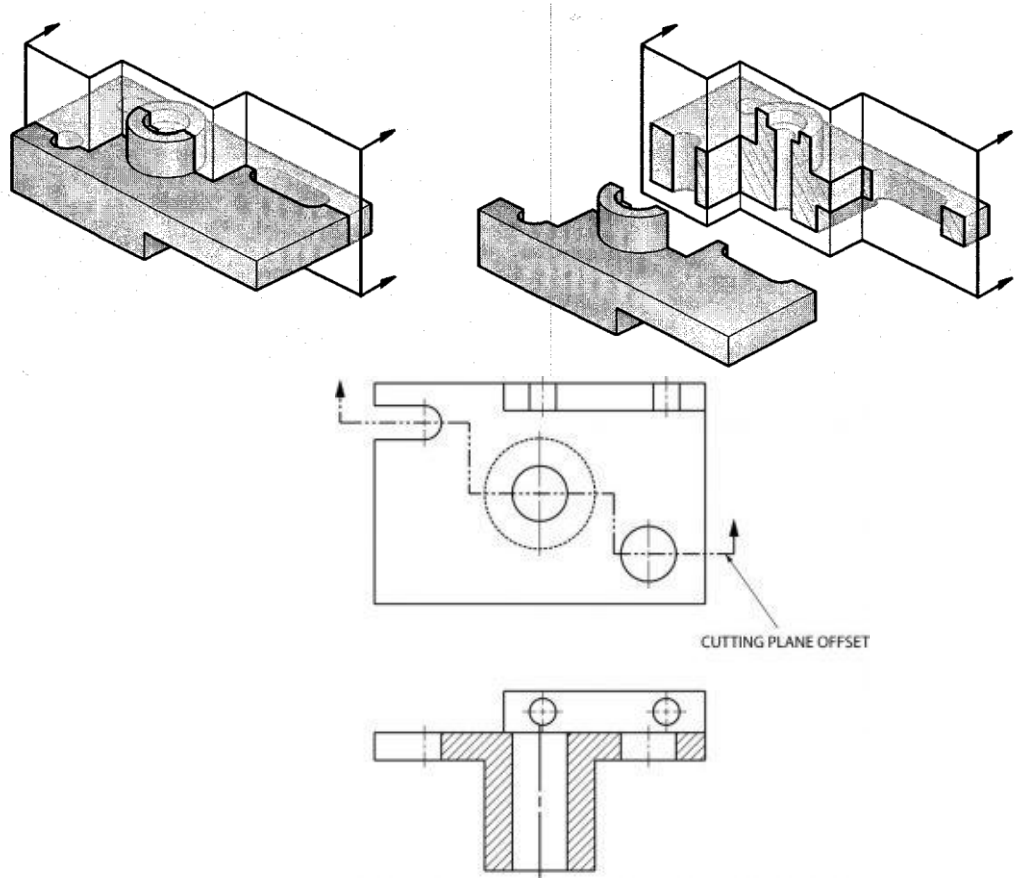
(C) Broken-out section view

4. **Revolved Sections**

A revolved section shows the shape of an object by rotating a section 90 degrees to face the viewer. The three revolved sections illustrated in the spear-like object of figure show the changes that take place in its shape.

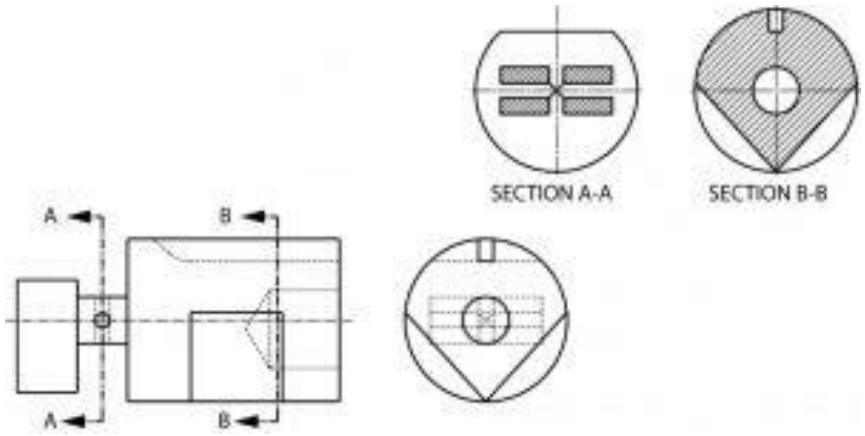
5. **Offset Sections**

An offset section is a means of including in a single section several features of an object that are not in a straight line. To do this, the cutting plane line is bent, or “OFFSET” to pass through the features of the part.



6. **Removed Sections**

A section removed from its normal projected position in the standard arrangement of views is called a "removed" section. Such sections are labeled SECTION A-A, SECTION B-B, etc., corresponding to the letter designation at the ends of the cutting plane line. Removed sections may be partial sections and are often drawn to a different scale.



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Definition of CAD (CAD की परिभाषा):-

Computer Aided Drafting (CAD) एक ऐसा Process है. जिसके द्वारा किसी Object का drawing बनाया जाता है और drawing बनाने के लिए Computer की सहायता ली जाती है। CAD Software एक आसान तथा उपयुक्त विधि है, जिसके द्वारा सामान्य या विशेष डिजाइन, मैकेनिकल तथा विद्युत इंजीनियरिंग में करते हैं।

Advantages of computer Aided Desing and Drafing (CAD):-

(i) CAD में उपलब्ध सुविधा से किसी Job का माडल develop करना तथा drafting करना बहुत ही सरल होता है।

(ii) CAD traditional विधि से fast तथा ज्यादा यथार्थ होता है।

(iii) CAD द्वारा विभिन्न diamention की गणना उसके मॉडल बनाये बिना ही Accuracy से की जा सकती है।

(iv) CAD में एक बार बनी Drawing को पुनः नहीं बनाया जाता है। इसे अल्प समय (Short time) में copy करके आगे का कार्य किया जाता है।

(v) माडल में संशोधन बहुत सरल होता है और भविष्य को देखते हुए Designer द्वारा दिये गये उत्पाद में आवश्यक परिवर्तन किया जा सकता है।

(vi) Traditional drawing विधि की तुलना में कैड द्वारा ड्रॉइंग की dimentions विशेषताओं तथा दूरी को संशोधित किया जा सकता है।

(vii) 3D देखने की Avalable सुविधा को व सभी direction (दिशा) में देखा जा सकता है।

(viii) Desing में Productivity (उत्पादकता) सुधार तथा Shoart lead time. लगता है।

(ix) किसी जटिल भाग का निरीक्षण, डॉक्यूमेंट बनाने में सहायता करता है,

(xi) यह निर्माण, योजना तथा निरी. में उपयोगी है।

(xi) Colluies भी use कर सकते Product को बेहतर ढंग से देखने के लिए।

List of various CAD softwares :-

1. Corel draw.
2. Microsoft office
3. Photo finish,
4. Point
5. Page makes,
6. Auto - CAD
7. Micro- station
8. Corel - CAD
9. Pro - E
10. Solidworks

Q.3 Functions of utility commands (setting Drawing Units, Limits, grid, and snap)

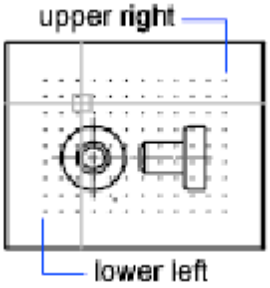
1. LIMITS: Allows changing the upper and lower limits of the drawing area while working on a drawing.

For example to set the screen for A3 size (420x297), following steps are to be carried out:

Command: limits ↵
ON/OFF/⟨lower left corner⟩ ⟨0.000, 0.000 or current⟩ : ↵
Upper right corner ⟨12.000, 9.000⟩ : 420,297 ↵
This will set the drawing screen of A3 size.

Sets and controls the limits of the grid display in the current Model or named layout.

Command entry: 'limits for transparent use

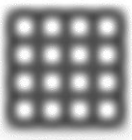


On Turns on limits checking. When limits checking are on, you cannot enter points outside the grid limits. Because limits checking tests only points that you enter, portions of objects such as circles can extend outside the grid limits.

Off Turns off limits checking but maintains the current values for the next time you turn on limits checking.

2. **GRIDS:** It displays a dot grid in the current view port.

Command: grid ↵
Grid spacing (x) or ON/OFF/Snap/Aspect/<current>: specify a value or enter an option.
Displays a grid pattern in the current viewport.



Toolbar: Status bar ➤ Grid

Grid Spacing (X) Sets the grid to the specified value. Entering **x** after the value sets the grid spacing to the specified value multiplied by the snap interval.

On Turns on the grid using the current spacing.

Off Turns off the grid.

Snap Sets the grid spacing to the snap interval specified by the SNAP command.

Major Specifies the frequency of major grid lines compared to minor grid lines. Grid lines rather than grid dots are displayed when SHADEMODE is set to Hidden.

3. SNAP- Sets the grid spacing to the current snap interval as set by the snap command.

Aspect- Sets the grid to a different spacing in x & y. It restricts cursor movement to specified intervals.

Command: snap ↵

Snap spacing or ON/OFF/Aspect/Rotate/Style/<current>: specify a distance, enter an option or press enter.

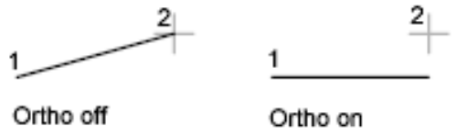
Spacing- Activates snap mode with the value you specify.

Rotate- Sets the rotation of the snap grid.

Style- format of the snap grid, standard or isometric.

4. ORTHO: Constrains cursor movement to the horizontal or vertical.

In the fig, a line is drawn using Ortho mode. Point 1 is the first point specified, and point 2 is the position of the cursor when the second point is specified.



Ortho mode is used when you specify an angle or distance by means of two points using a pointing device. In Ortho mode, cursor movement is constrained to the horizontal or vertical direction relative to the UCS.

5. Osnap

Sets running object snap modes. **Allows to select specify points on an object. e.g. endpoints, midpoints, intersection etc.**

Menu: Tools ➤ Drafting Settings

Toolbar: Status bar ➤ Osnap

Shortcut menu: Press Shift while right-clicking in the drawing area and choose Osnap Settings.

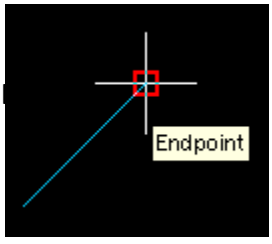
Command entry: F3

Command entry: 'osnap for transparent use

Function

The Object Snap tab of the Drafting Settings dialog box is displayed. If you enter -**osnap** at the Command prompt, the following prompts are displayed.

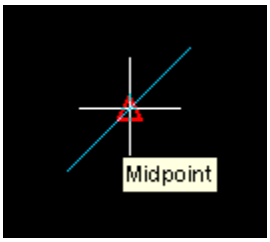
Endpoint



Toolbar	Object Snap 
Pull-down	Shift + Right Click Endpoint
Keyboard	END (when picking)

The Endpoint Osnap snaps to the end points of lines and arcs and to polyline vertices. This is one of the most useful and commonly used Osnaps.

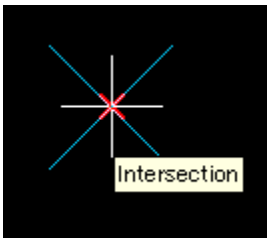
Midpoint



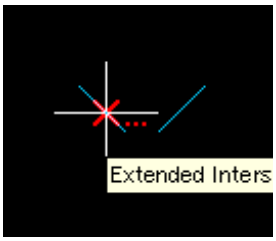
Toolbar	Object Snap 
Pull-down	Shift + Right Click Midpoint
Keyboard	MID (when picking)

The Midpoint Osnap snaps to the mid points of lines and arcs and to the mid point of polyline segments.

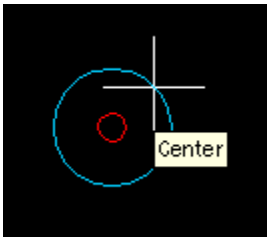
Intersection



Toolbar	Object Snap 
Pull-down	Shift + Right Click Intersection
Keyboard	INT (when picking)



Center



Toolbar	Object Snap 
Pull-down	Shift + Right Click Center
Keyboard	CEN (when picking)

The Center Osnap snaps to the centre of a circle, arc or polyline

arc segment. The cursor must pass over the circumference of the circle or the arc so that the centre can be found. This often causes some confusion for new users.

Editing Commands

1. Erase

Get rid of objects you do not want. To use:
1. Type or select the command
2. Select the objects you want to get rid of.
3. Press enter.

Note: A really quick way to erase is to select items and hit the Delete key.

2. COPY

Function: It duplicates objects. To copy objects within a drawing, specify a starting point and an endpoint of displacement.
Command: **COPY**
Select objects: **(pick object to copy)**
Select objects: ↵ **(to end selection)**
Specify base point or displacement, or [Multiple]: **(pick a point or M for multiple copies)**
Specify second point of displacement or <use first point as displacement>: **(pick a point)**

3. ARRAY

Function: It creates a multiple copies of the selected objects in a rectangular matrix (columns and rows) or a circular/polar pattern.

Command: array↵

Rectangular or polar array (R/P) <current>: enter an option or ↵

Option:

RECTANGULAR: – Creates an array defined by a number of rows and columns of copies of selected objects.

A **polar array** is defined by specifying a centre point about which the selected object is replicated, number of items and angle to fill. A positive angle value specifies counter-clockwise rotation and a negative value specifies clockwise rotation.

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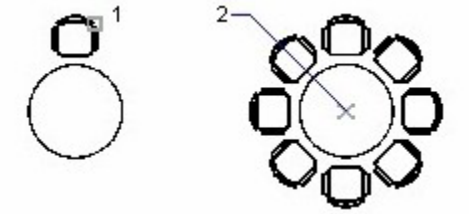
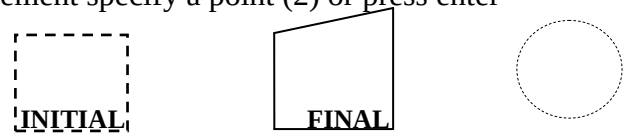


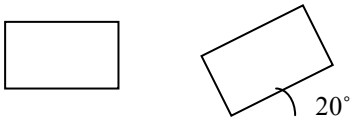
MOVE: This allows to move or displace objects at a specified distance in a specify direction.

At the command prompt, enter MOVE
Select object use an object selection method,
Base point or displacement specify a box pt.(1)
Second point or displacement specify a point (2) or press enter



ROTATE: It Rotate an object about a specified base point.

Defines horizontal line right side angle 0˚
At the command prompt, enter rotate
Selection objects: use an object selection method



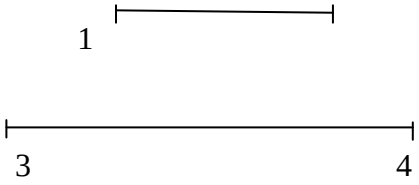
Base point: specify a point (1)
< Rotation angle > / Reference: specify an angle or enter Y

SCALE:-**This enlarges or reduces selected object equally in X & Y directions.**

At command prompt, enter SCALE
Select object: Use an object selection method.
Base point: Specify a point (1)
< Scale factor > / Reference: Specify a scale or enter Y

STRETCH:- This stretches object. It also stretches lines, arcs, elliptical arcs, rays, splines & polylines segments that cross the selection window.

At command prompt, enter STRETCH
Select object: - Use an object selection method (pt. 1 2)
2
Base point or displacement: Specify a point (3) or press Enter
Second point of displacement, specify a point (4) or press Enter



Basic Draw Commands

CIRCLE :- Draws circles of any size.
Command :- Circle (enter)
3P/2P/TTR/<center point> :- (pick a center point)
Diameter or <Radius> :- (Pick a point on the circle)

LINE :- Draws straight lines between two points
Command :- LINE (enter)
From Point :- (pick a point using the mouse)
To Point :- (Pick a point using the mouse)
To Point :- (Press return to end the command)

ARC :- Draws an arc (any part of a circle or curve) through three known points.
Command :- ARC (enter)
Center/ < Start point > :- (pick the first point on the arc)
Center/End/ < Second point > :- C
Center :- (pick the arc's center point)
Angle/Length of chord/ <End point > :- (pick the arc endpoint)

Explain the following EDIT commands :

(i) OFSET
(ii)FILLET
(iii)CHAMFER
(iv) EXTEND and
(v) ARRAY.

Ans: (i) OFSET

Function: It creates new objects that are parallel to or concentric with a selected object. The new object is drawn at a user defined distance (the offset) from the original and in a direction chosen by the user with a pick point.

Command: OFFSET
Specify offset distance or [Through]<1.0000>: (specify distance)
Select object to offset or <exit>: (select an object)
Specify point on side to offset: (pick a direction)
Select object to offset or <exit>: ↵ (to end or select another object to offset)

(ii)FILLET

Function: It connects two objects with a smoothly fitted arc of a specified radius. To fillet objects, specify the radius and change the trim mode if necessary before selecting the objects.

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 10.0000
Select first object or [Polyline/Radius/Trim/mUltiple]: **(enter r to specify radius)**
Specify fillet radius <10.0000>: **(enter the radius value)**
Select first object or [Polyline/Radius/Trim/mUltiple]: **(pick the first object)**
Select second object: **(pick the second object)**

(iii)CHAMFER

Function: It creates a chamfer between any two non-parallel lines or any two adjacent polyline segments. Usually the chamfer command is used to set the chamfer distances before drawing the chamfer. There are a number of options which can be used to control the way the chamfer command behaves: polyline, distance, angle, trim and method.

Command: **CHAMFER**
(TRIM mode) Current chamfer Dist1 = 10.0000, Dist2 = 10.0000
Select first line or [Polyline/Distance/Angle/Trim/Method/mUltiple]: **(enter D to specify the Distance option)**
Specify first chamfer distance <10.0000>: **(enter required distance)**
Specify second chamfer distance <10.0000>: (↵**specifying the first distance value or enter a different value)**
Select first line or [Polyline/Distance/Angle/Trim/Method/mUltiple]: **(pick an object)**
Select second line: **(pick the other object)**

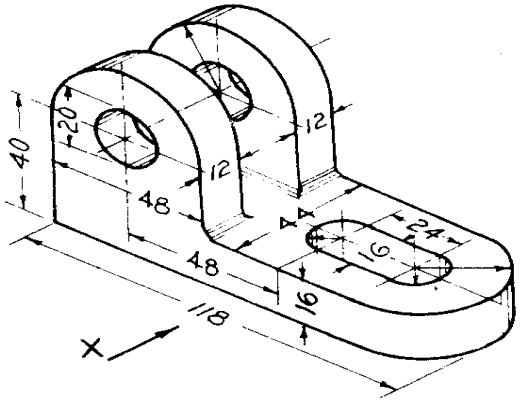
(iv) EXTEND

Function: It extends an object to meet another object precisely. To extend an object, specify boundary edge(s), the edge extension mode if necessary, before selecting the desired object.

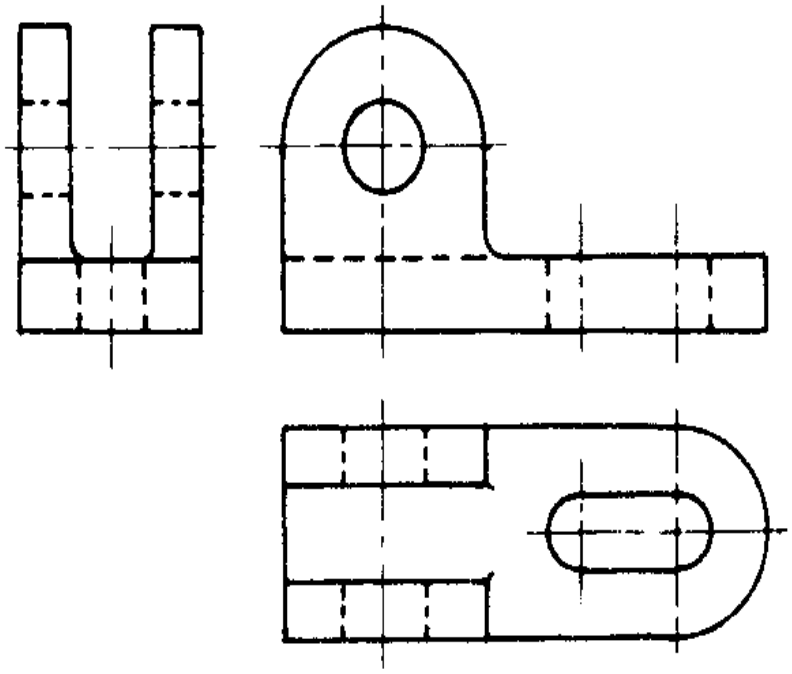
Command: **EXTEND**
Current settings: Projection=UCS, Edge=None Select boundary edges ...
Select objects: **(select the boundary edge)**
Select objects: ↵ **(to end boundary edge selection)**
Select object to extend or shift-select to trim or [Project/Edge/Undo]:**(pick the object to extend)**
Select object to extend or shift-select to trim or [Project/Edge/Undo]: ↵ **(to end)**

Orthographic Projection

Problem 20-5. *Draw the following views of the object shown pictorially in fig. 20-22(i). (i) Front view. (ii) Top view. (iii) Side view from the right.*



Ans



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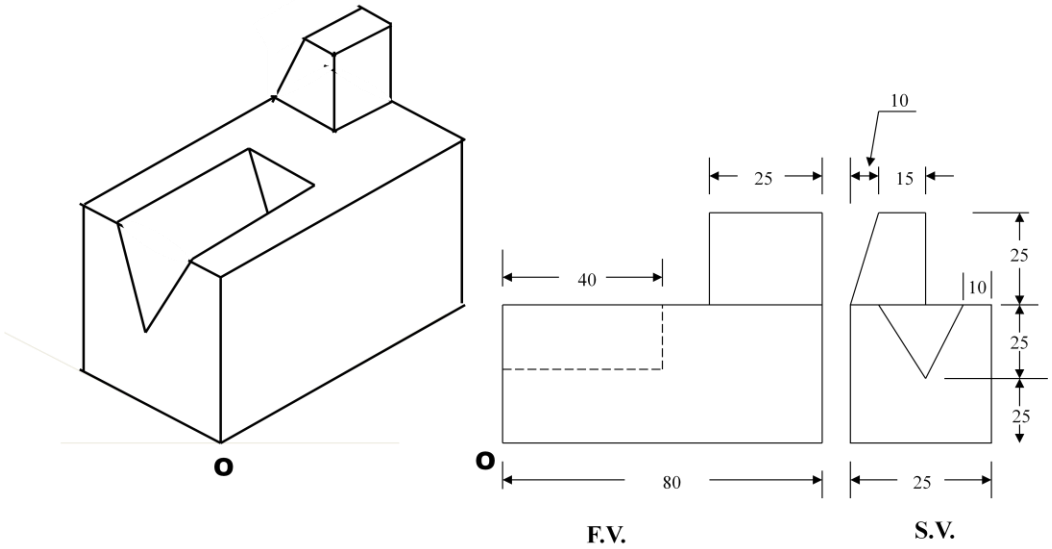
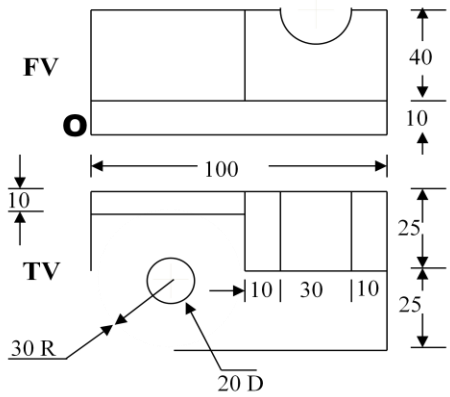
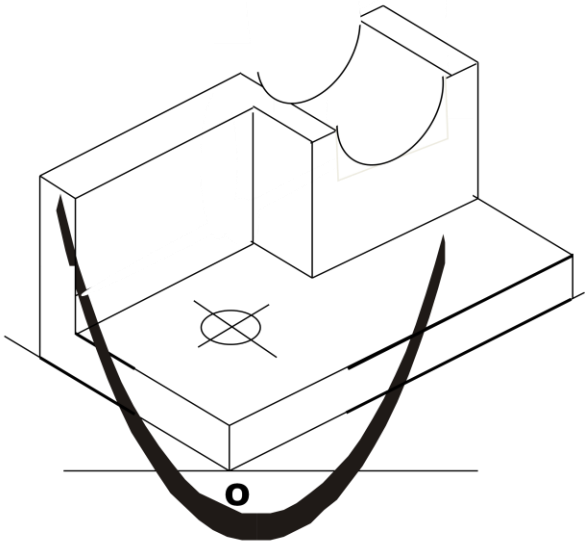
Q. 2

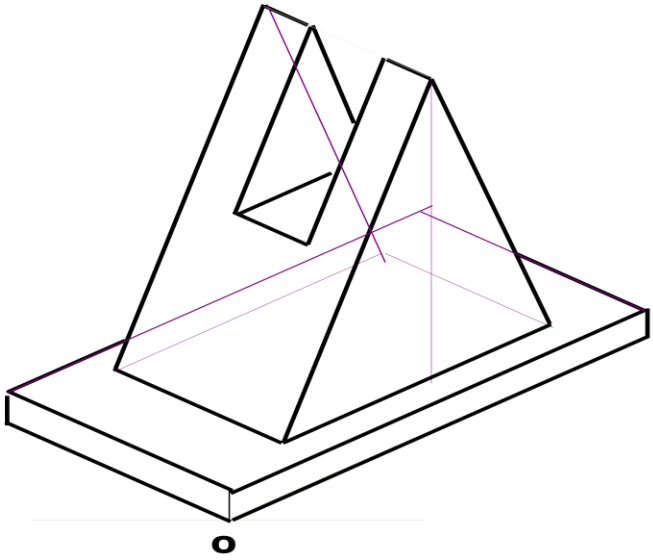


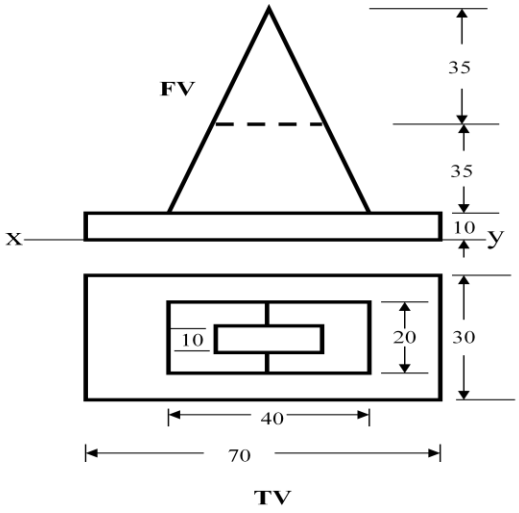
3.





4.





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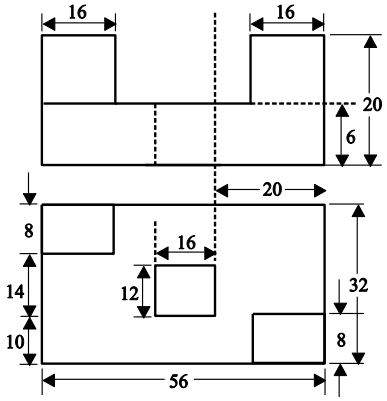
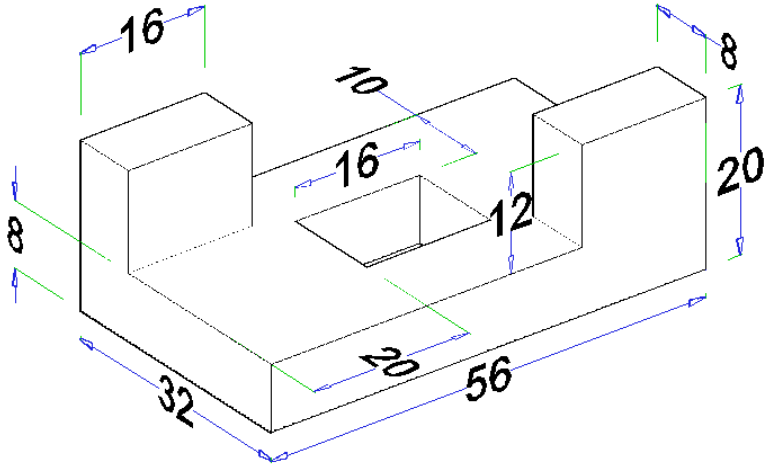
Isometric

Draw the isometric view.

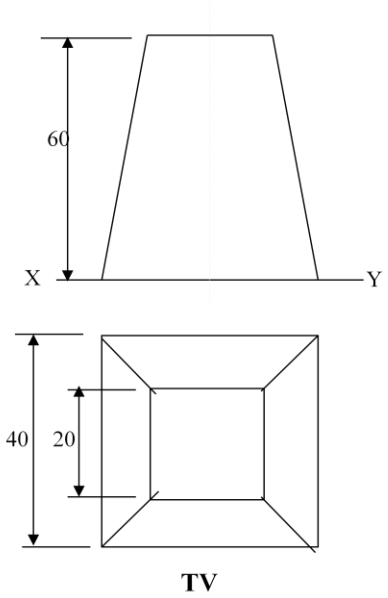


Sol:

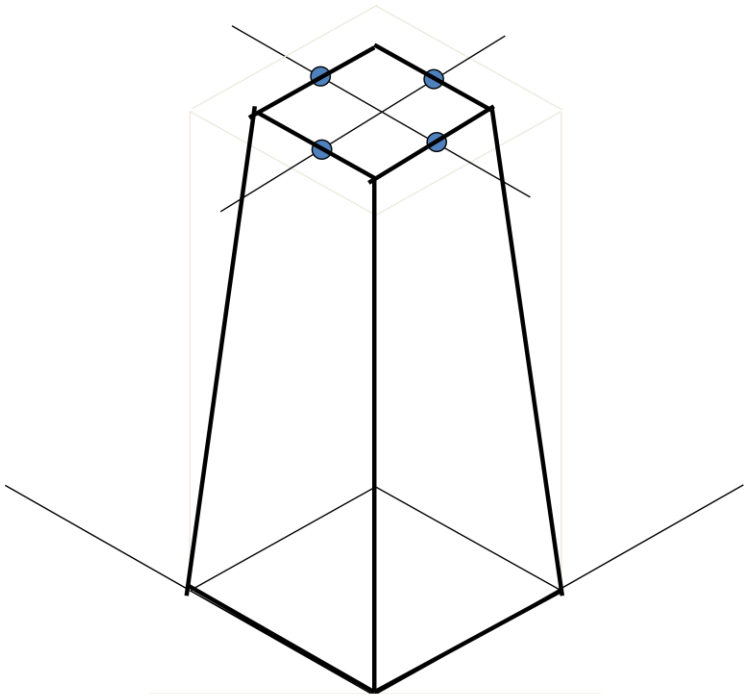


Q.3 Draw the isometric view of the frustum of a square pyramid side of base 50 mm and of tope square is 25 mm. the height of the frustum is 80 mm.

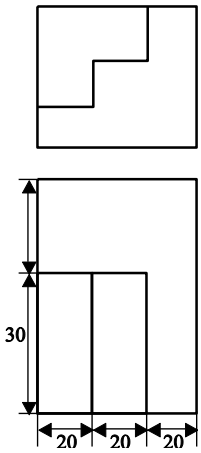
Solution :-



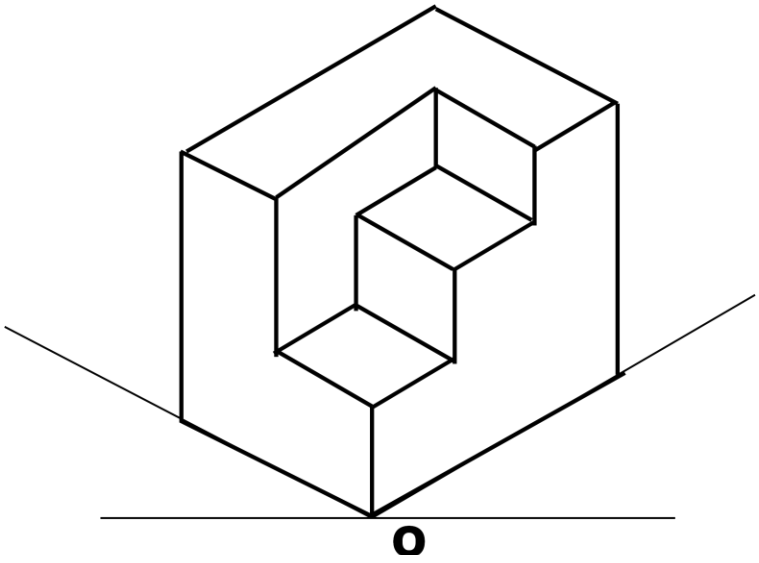
TV



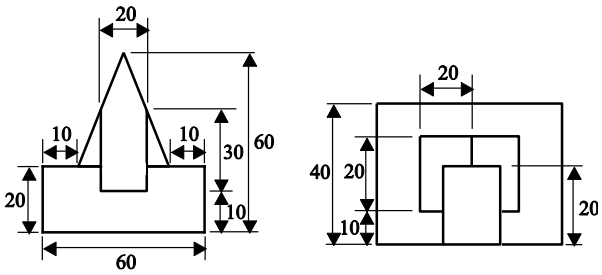
Q.4 Draw the isometric view of the given figure,



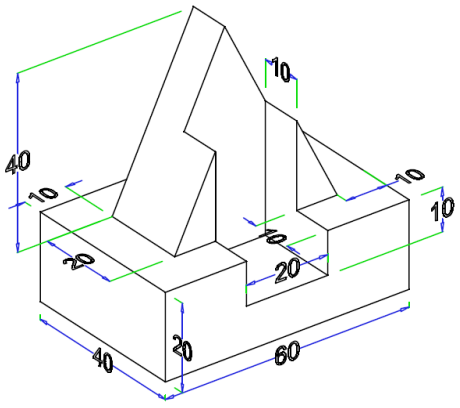
Solution :-



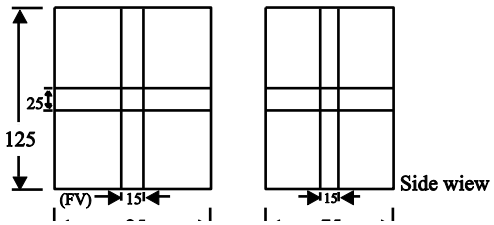
Q.11 Draw the isometric view for the given front view and top view.



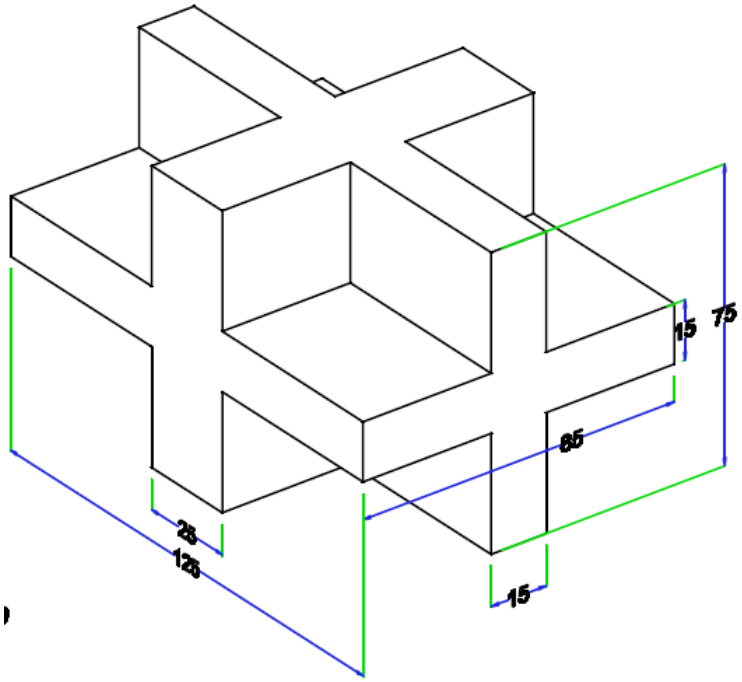
Solution :-



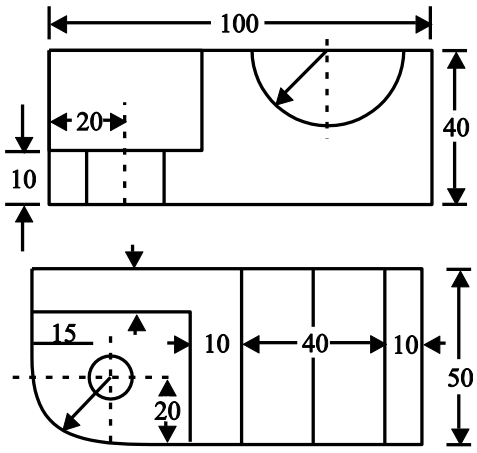
Q.19 Draw isometric view of machine part, whose orthographic views are given below.



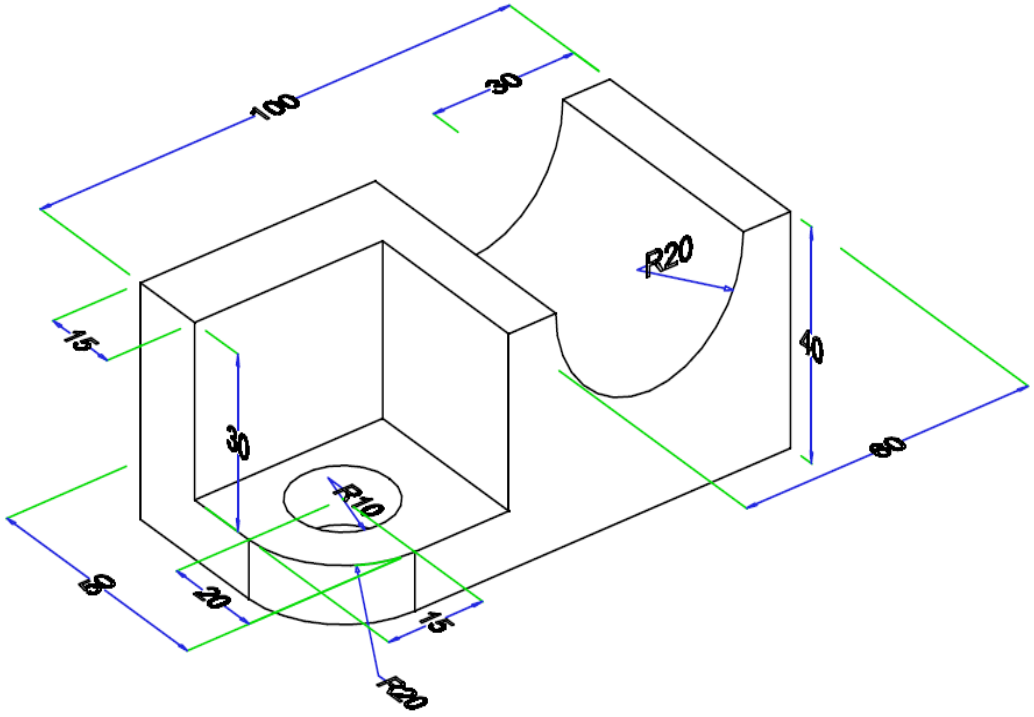
Solution :-



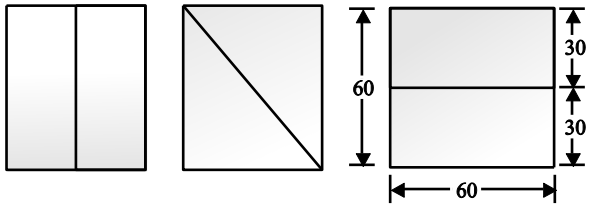
Q.23 Draw the isometric view of the casting shown in two views in figure.



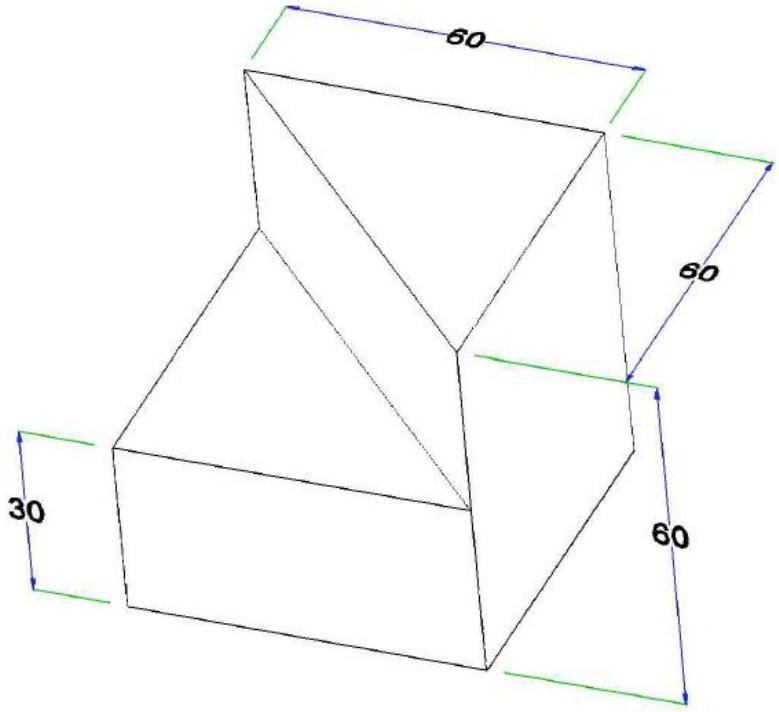
Solution :-



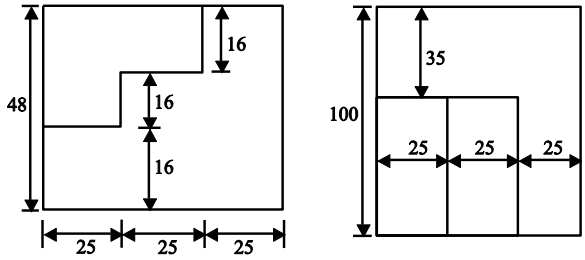
Q.27 Draw the isometric view of the drawing shown in figure.



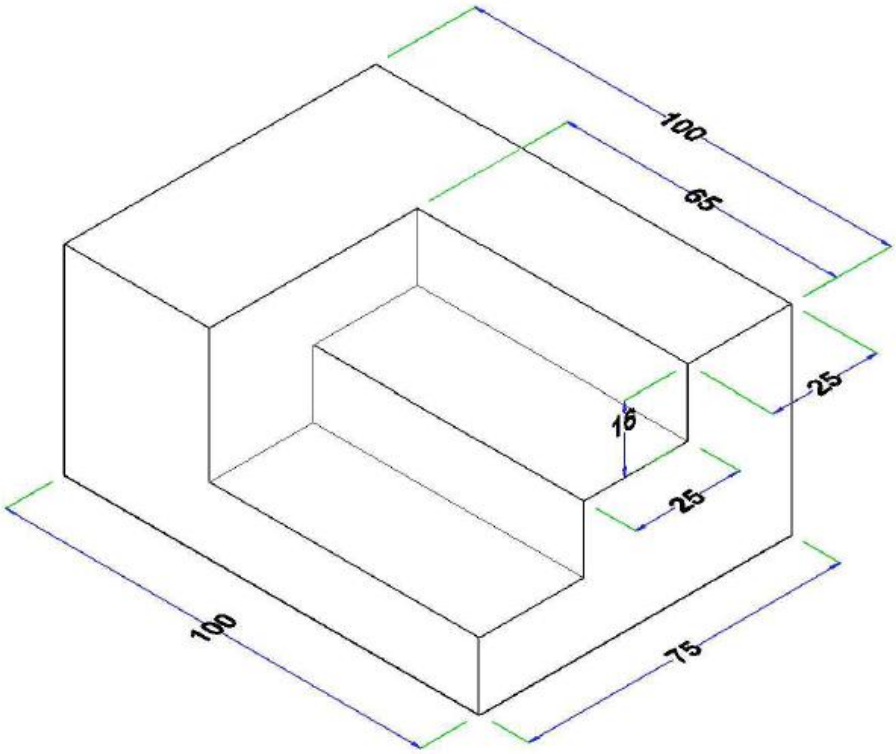
Solution :-



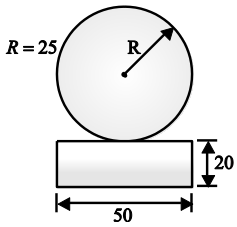
Q.30 Draw the isometric view of the casting shown in two views :



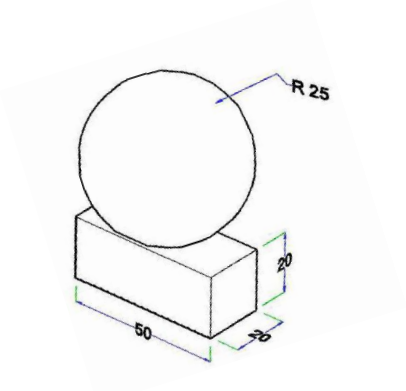
Solution :-



Q.31 Draw the isometric projection of a sphere, resting centrally on the top of a square prism, the front view of which is shown :

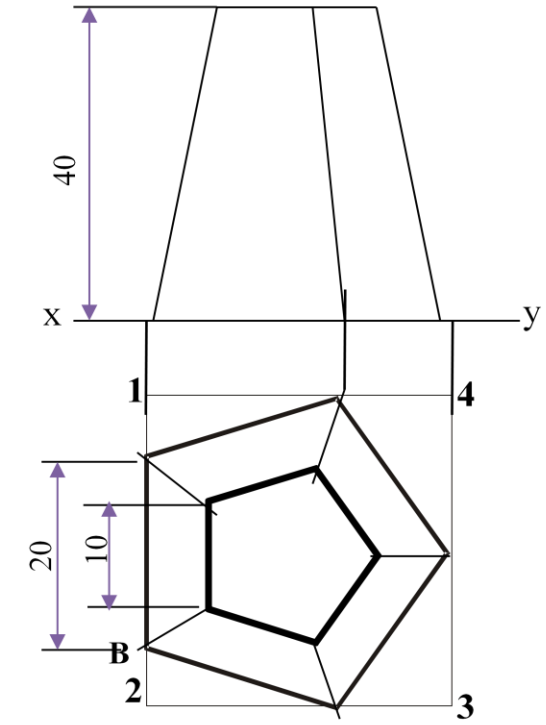


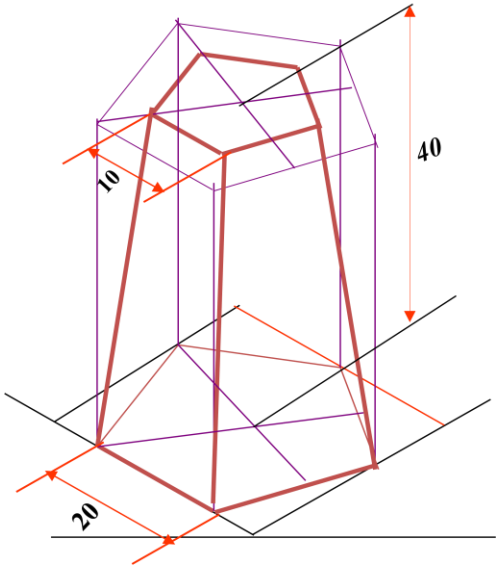
Solution :-



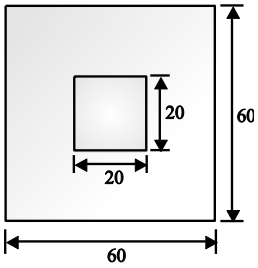
Q.34 Draw the isometric drawing of the frustum of a right regular pyramid, side of base hexagon is 20 mm and of the top hexagon is 10 mm and height of the frustum is 40 mm.

Solution :-

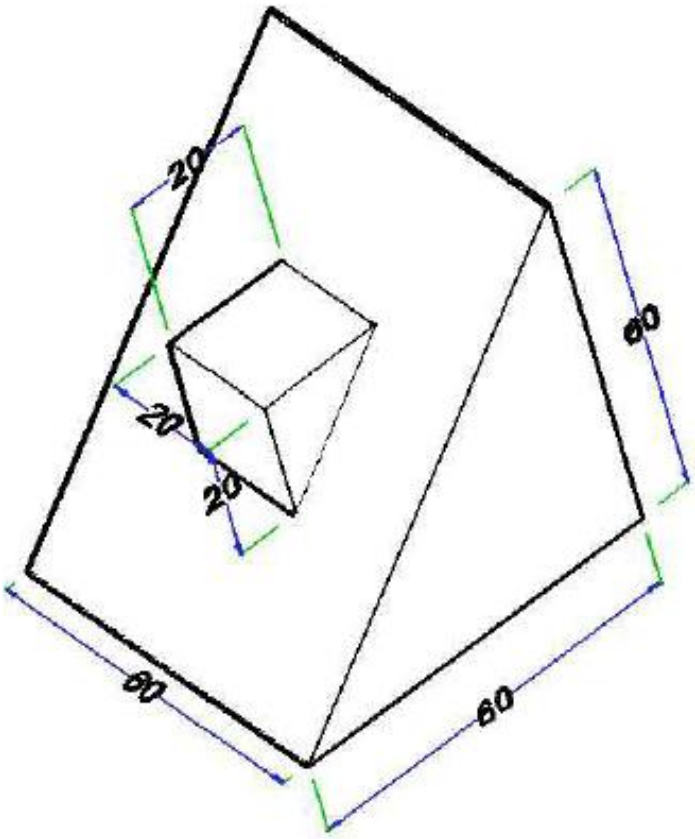




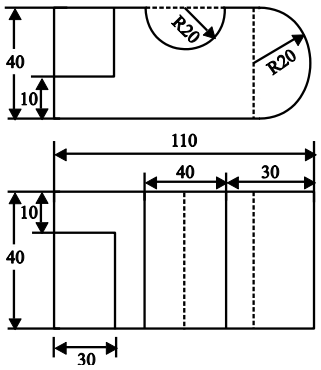
Q.35 Plan and elevation of an object are identical as shown in figure, draw the isometric view.



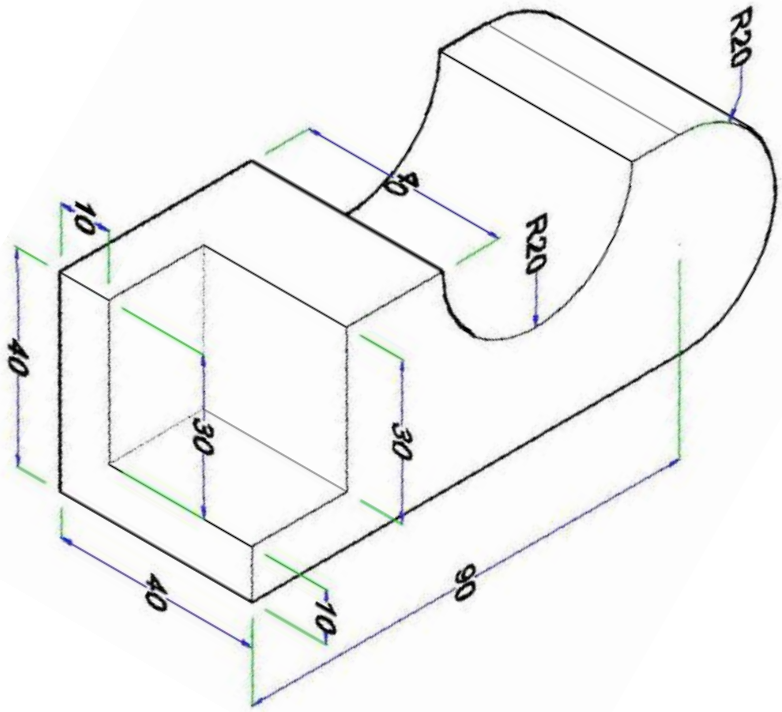
Solution :-



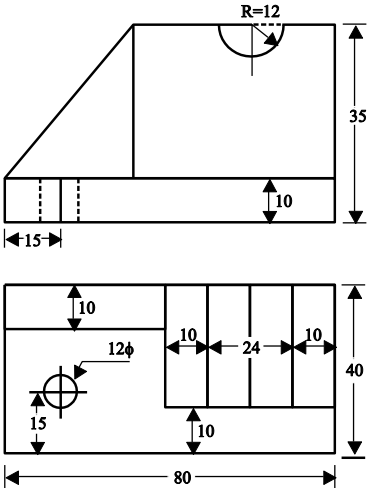
Q.38 Draw the isometric view from given orthographic projections.



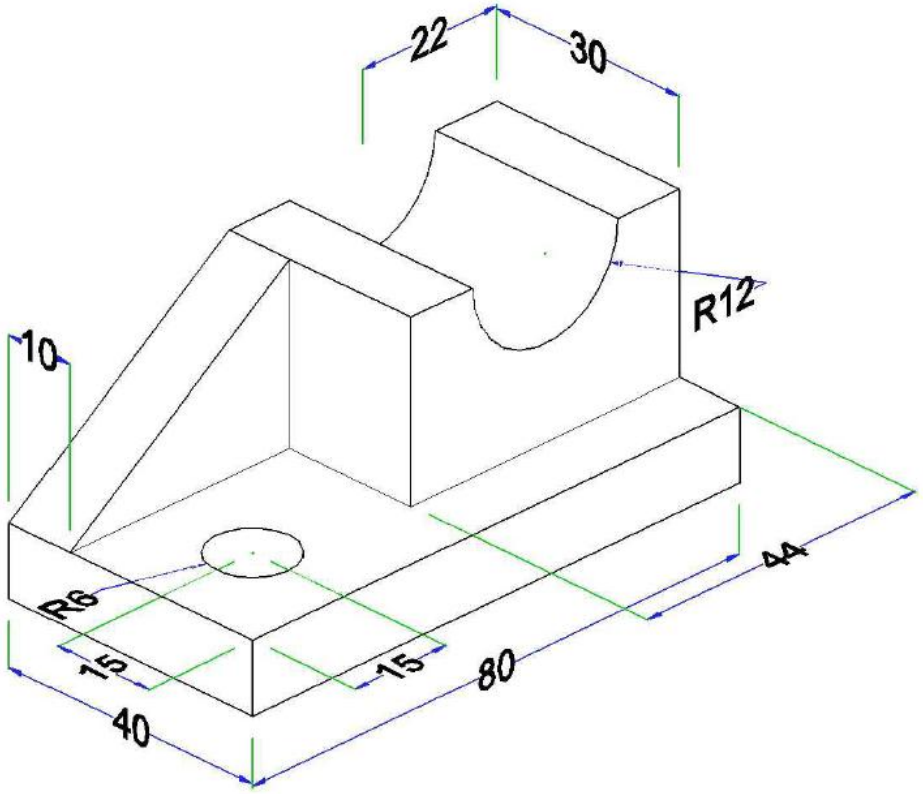
Solution :-



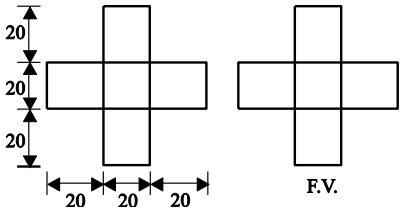
Q.41 Draw the isometric view of the object shown in two views in following figures.



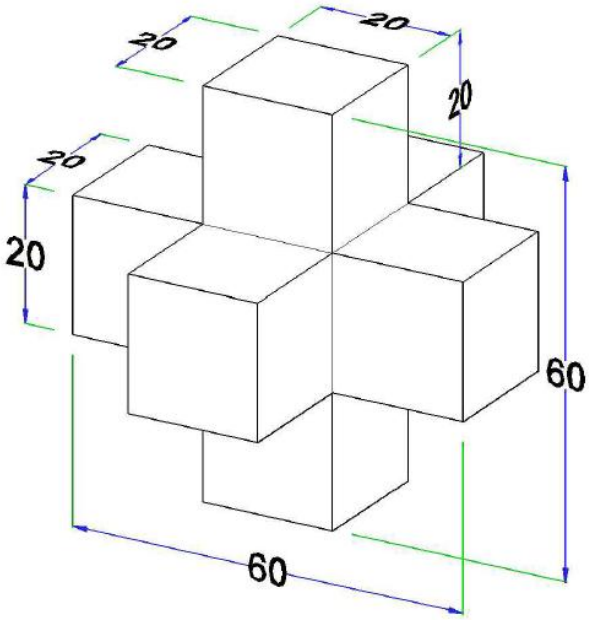
Solution :-



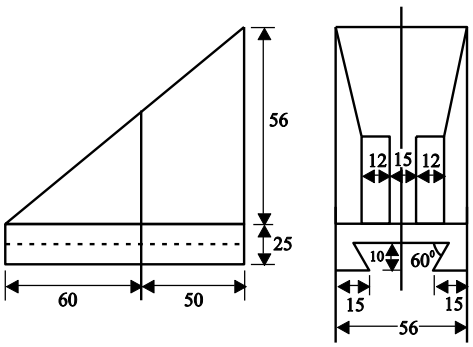
Q.46 Draw the isometric view of solid whose orthographic projection are given in below figure.



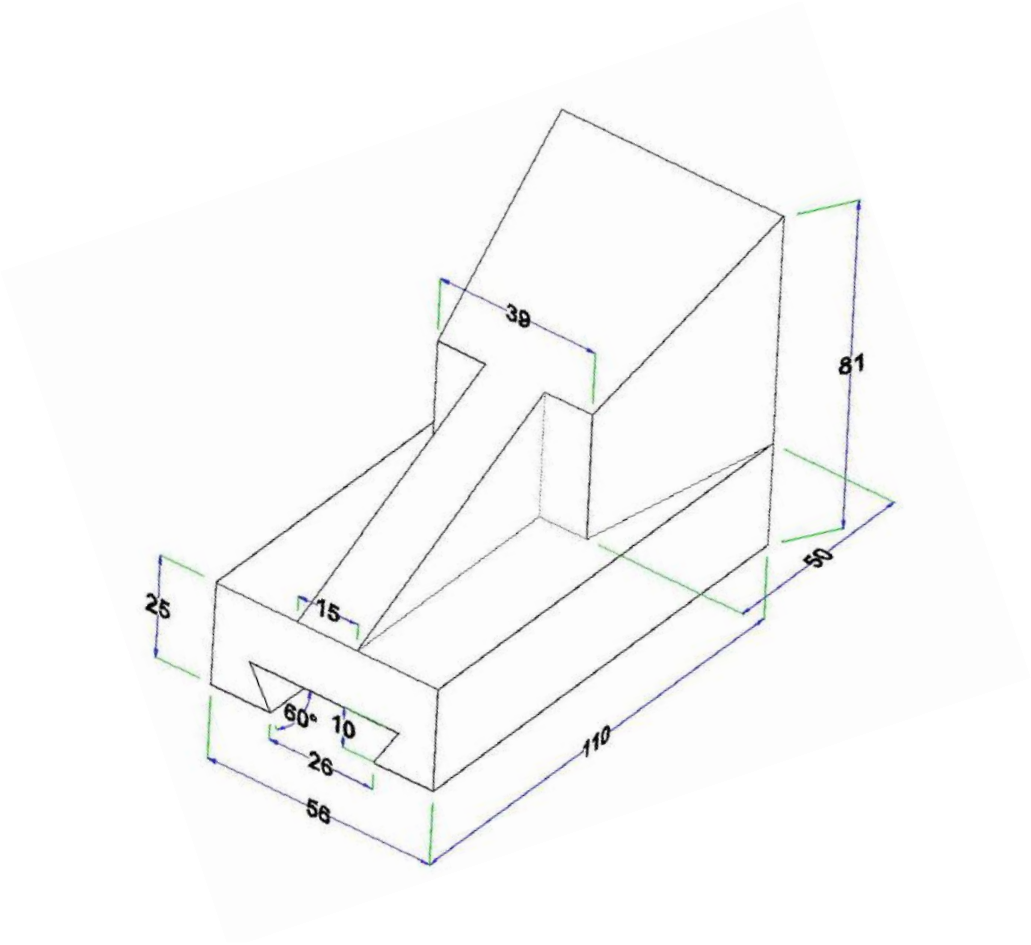
Solution :-



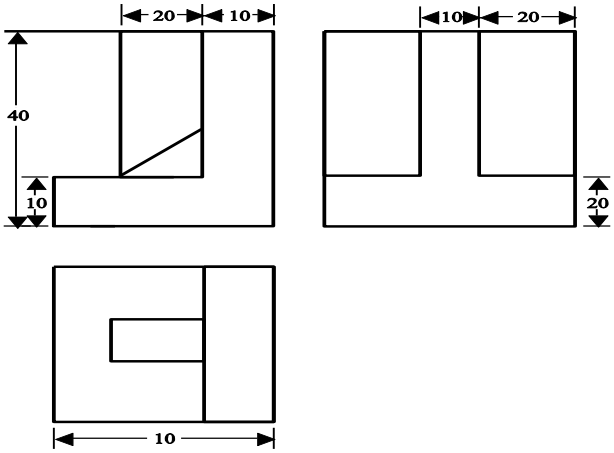
Q. 48 Draw the isometric of the figure given below.



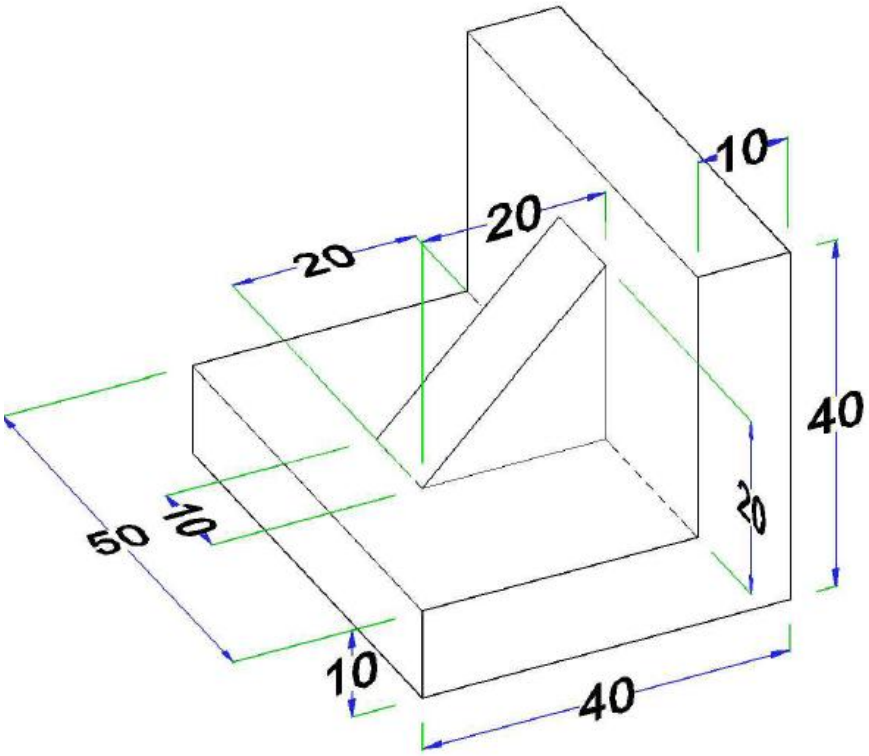
Solution :-



Q.51 Draw the isometric views of the Ribbed angle plate **.**



Solution :-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