

COURSE CODE:	<i>MCE 301</i>
COURSE TITLE:	<i>Machine Drawing</i>
NUMBER OF UNITS:	<i>1 Units</i>
COURSE DURATION:	<i>Two hours per week</i>

COURSE DETAILS:

Course Coordinator:	Engr. Dr. Bukola Olalekan Bolaji <i>B.Eng., M.Eng., PhD</i>
Email:	bolajibo@unaab.edu.ng , bobbolaji2007@yahoo.com
Office Location:	HOD's Office, MCE Department, COLENG
Other Lecturers:	None

COURSE CONTENT:

Intersection of solids, simple and complex assembly drawings – orthographic and sectional views. Isometric drawings of assemblies – universal couplings, clutches, bearings and seals. Exploded drawings of assemblies – pumps, valves, internal combustion engines and parts. Interchangeability of machine parts. Installation drawings erection diagrams. Computer-aided drawings for design and manufacture. Drawing project.

COURSE REQUIREMENTS:

This is a compulsory course for all 300 level students in the Department of Mechanical Engineering. In view of this, students are expected to participate in all the course activities and have minimum of 75% attendance to be able to write the final examination.

READING LIST:

1. Cecil, H., Jensen, J. and Helsel, D. Engineering Drawing and Design. New York: McGraw-Hill, 1995.
2. Folkestad, J., Schetz, K. and Madsen D.A. Engineering Drawing and Design. Delmar Cengage Learning, 2001.
3. Green, P. The Mechanical Engineering Drawing. CreateSpace, 2007.
4. Madsen D.A. Engineering Drawing and Design (Drafting Series). Delmar Cengage Learning, 2001.
5. Robert, J.F. and Charles J.V. Engineering Drawing and Graphic Technology. Thomas Ewing French.

LECTURE NOTES

INTERSECTION OF SURFACE

INTRODUCTION

A machine part or structure of any kind consists of a number of geometric shapes arranged to produce the desired form. In these geometric shapes when combined or interlocked, the surface of these basic shapes meets in line of intersection either in the form of straight or curved lines at the joint. This line is common to both the surfaces.

When a form is to be constructed of sheet metal, fabricated parts or bonded material, an accurate representation of intersecting surfaces becomes important because of weldability at the joint, since the component part must fit accurately and smoothly for optimum functioning and appearance.

The line or curve of intersection depends upon the nature of meeting surfaces, if two plane surfaces meet the line of intersection in a straight line. If out of the two surfaces, one is curved, then the line of intersection will also be a curve.

The students must be able to construct the lines of intersections in orthographic views, as it is of considerable importance.

METHODS OF DETERMINATION OF LINE OR CURVE OF INTERSECTION

There are three methods of determination of line or curve of intersection of two interpenetrating solids:

1. Line method
2. Cutting plane method
3. Sphere method

1. **Line method:** number of lines/generators or edges are drawn on the lateral surface of the penetrating solid in the area of intersection or limited surfaces of the solid. The lines of intersection are obtained by finding the existing piercing points of lines /generators or edges of one solid with the surfaces of the second solid, and then the existing piercing points of the line/generators or edges in the second solid with the surfaces of the first. On joining these points in sequence, we can obtain the line/curve of intersection.

2. Cutting plane method: both solids are assumed to be cut by a number of cutting planes/slices to cut both solid in the region of intersection. These cutting planes/slices may be perpendicular to the H.P., V.P., or oblique plane. The intersections of these lines of section obtained from each surface by a plane will give one or more points satisfying the points of intersecting line/curve.

Certain important critical points called the key points shall be selected and the cutting planes should be passed through these points. The curve at this point usually takes a turn. These points will also give the highest and the lowest points on the on the intersection curve. Some arbitrary points may also be chosen in order to locate the intermediate points of the curve of intersection to establish the the flow of the curve.

3. Sphere method: this is one of the unique one view methods which may be used for finding the lines of intersection of two curved surfaces when the following conditions exist:

1. The surfaces have circular right sections
2. The axes intersect
3. The axes appear true length in the same view

This solution is based on the fact a sphere of sufficient diameter, centred on the axis of the surface of the revolution such as the circular cylindrical surface, intersects the surface in the circles which appears edgewise in a view that shows the true length of the axis of the cylinder. A sphere of same diameter as the cylinder would be tangent to the surface and contact the surface only in one circle.

COMBINATION OF SOLIDS

Among the solids, there can be many combinations for intersection, such as

1. Cylinder to cylinder, prism, cone and sphere
2. Prism to prism, cone, pyramid and sphere
3. Cone to cone.

INTERSECTION OF TWO PRISMS

Faces of the prism are plane surfaces. The line of intersection for these two solids is a closed figure, composed of number of straight lines. It is obtained by locating that point, at which edges of one prism intersect the edges of the faces of the other prism and then joining them in correct sequence. Line of intersection will appear in one view only in the case of cylinder and prism intersection.

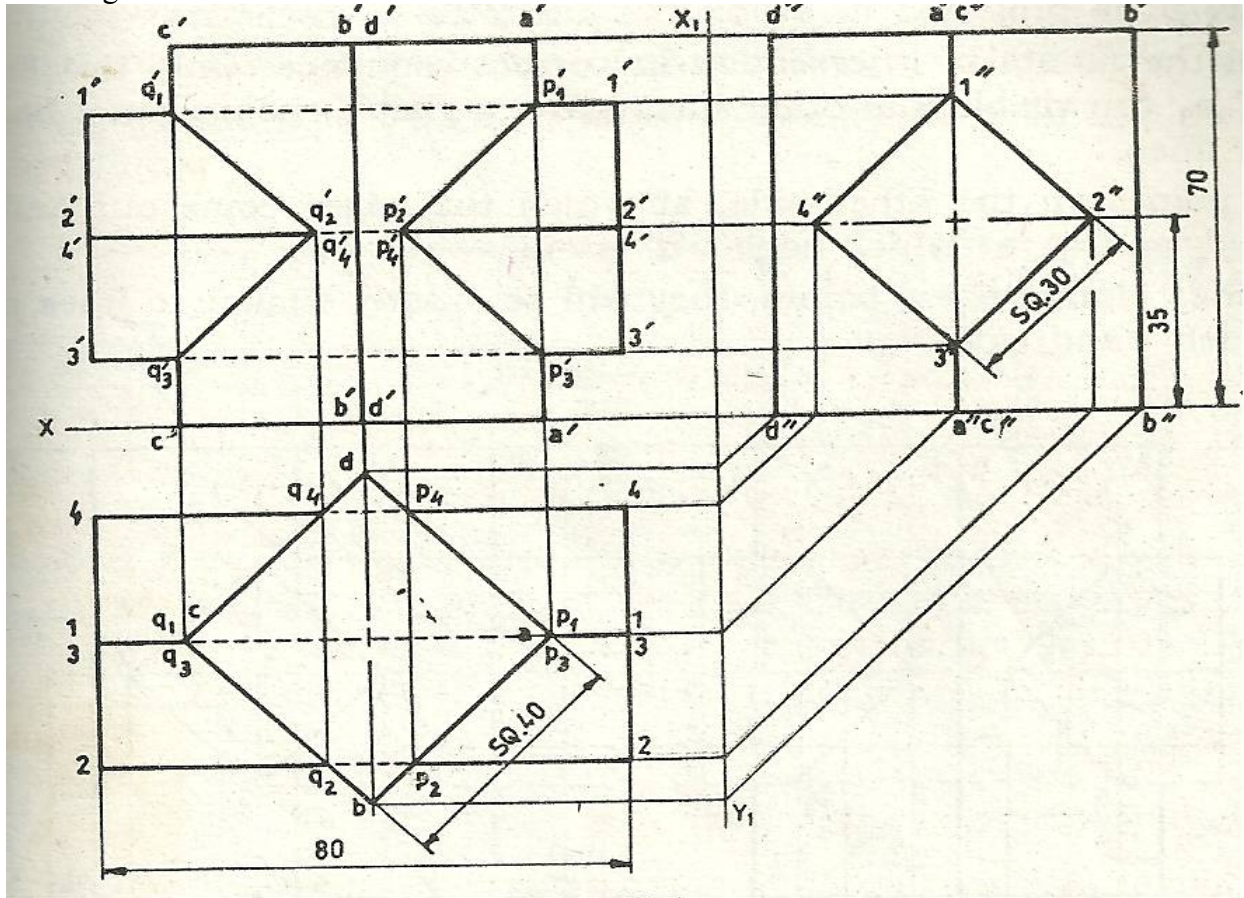
Example 1

A vertical square prism, base 40 mm side and length 70 mm, is completely penetrated by a horizontal square prism, base 30 mm side and axis length 80 mm, so that their axes intersect. The axis of the horizontal prism is parallel to the V.P., while the faces of the two prisms are equally inclined to the V.P. Draw the projections of the solids showing lines of intersection.

Solution:

- (i) First draw the views of the vertical square prism and then the views of the horizontal prisms. Number the edges of both the prisms as A, B, C, D, for the vertical prism and 1,2,3,4 for horizontal one
- (ii) Mark the common points of the edges as 1'', 2'', 3'', and 4'' in the side view and then project these edges in plan as p₁, p₂, p₃, and p₄, also q₁, q₂, q₃, and q₄,
- (iii) Lines 1-1 and 3-3 intersect the edges of the vertical prism in points p lines 2-2 and 4-4 intersecting the faces at p and p respectively.
Project this point vertically on the corresponding lines in the front view. For example p₂, is projected to p₂¹. Draw line p₁¹, p₂¹, p₃¹. Lines p₁¹p₄¹ and p₃¹p₄¹ coincide with the front line. These lines show the line of intersection
Lines q₁¹q₂¹ and q₂¹q₃¹ on the other side are obtained in the same manner
Note that lines for the hidden portions of the edges are shown by dashed lines.
The portions p₁p₂ and q₂q₃ of vertical edges a' a' and c' c' do not exist and hence must be removed or kept fainter.

Drawing



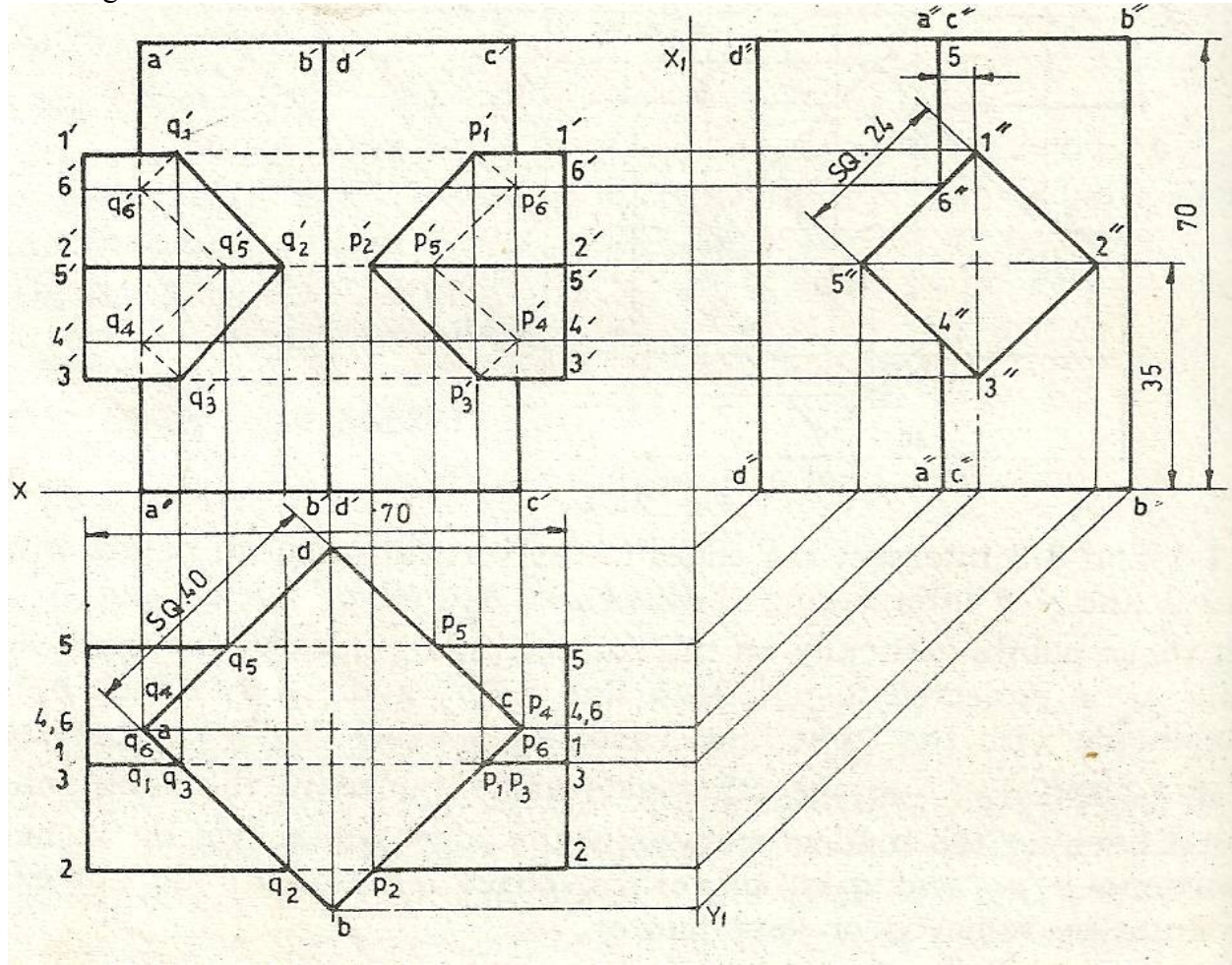
Example 2

A vertical square prism, base 40 mm side and 70 mm, is completely penetrated by a horizontal square prism, base 24 mm side and axis 70 mm, so that their axes are 5 mm apart. The axis of the horizontal prism is parallel to the V.P. Draw the projections of the prism showing lines of intersection

Solution

- (i) Draw the views of the vertical prism first and then the views of horizontal prism. Name out the edges 1,2,3, and 5 in the side view as 1'',2'',3'', and 5'',. Also points 4'', and 6'' on vertical edge a'' c''
- (ii) Project these points in the top view on the faces of vertical prism.
- (iii) The exact positions of these points along the length of the prism may now be determined by projecting them on the corresponding lines in the front view. For example, p₂ is projected to p'₂ on the line 2'2'.
- (iv) Join all the points of intersection in correct sequence. Only two lines p₁'p₂' are visible and other lines other not visible, hence must be shown by dashed lines.
- (v) Locate points on the other sides, at which the edges come out and also two point q₄ q₆ at which edges a'a' is cut.
- (vi) Draw lines joining these points, they will be exactly similar to lines p₁'p₂', etc,on the left hand side.

Drawing:



Example 3

A vertical square prism, base 40 mm side and height 72 mm, has a face inclined at 30° to the V.P. It is completely penetrated by another square prism, base 30 mm side and axis 80 mm long, faces of which are equally inclined to the V.P. The axes of the two prism are parallel to the V.P. and bisect each other at right angles. Draw the projections showing lines of intersection.

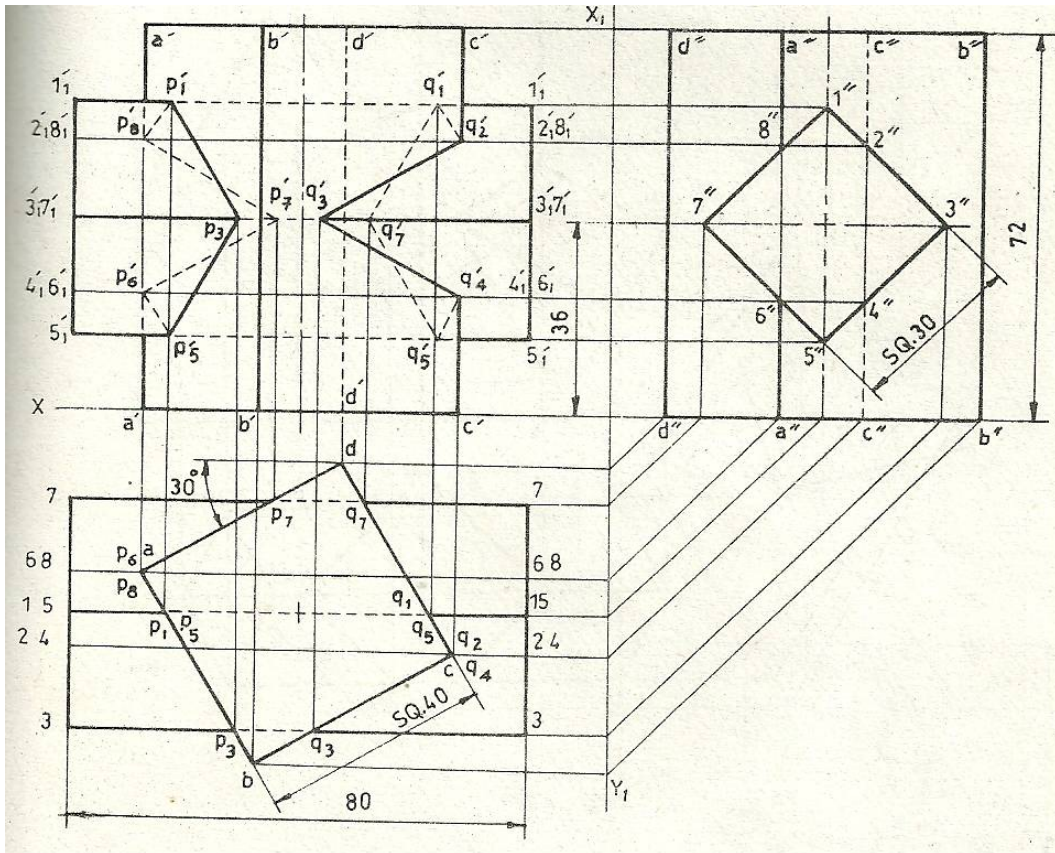
Adopt the same method as explained in problem 2 above

(i) Draw the three views of the vertical prism first and then the views of the horizontal projection. Name out the edges of horizontal prism in the side view as 1'', 2'', 3'', 5'' and 7'' and point 2'' and 4'' on the edge c''c'' and then 6'' and 8'' on the edge a''a''

(ii) project these points in the top view on the faces of vertical prism. From the top view, project them in the front view on the corresponding lines. For example, p3 and q3 are projected as p'3 and q'3 on line 3'1'1'. Similarly, project all the points and join them in sequence to obtain the lines of intersection.

(iii) Lines q'2-q'3-q'5 are visible, others are invisible and, hence shown by dashed lines

Drawing:



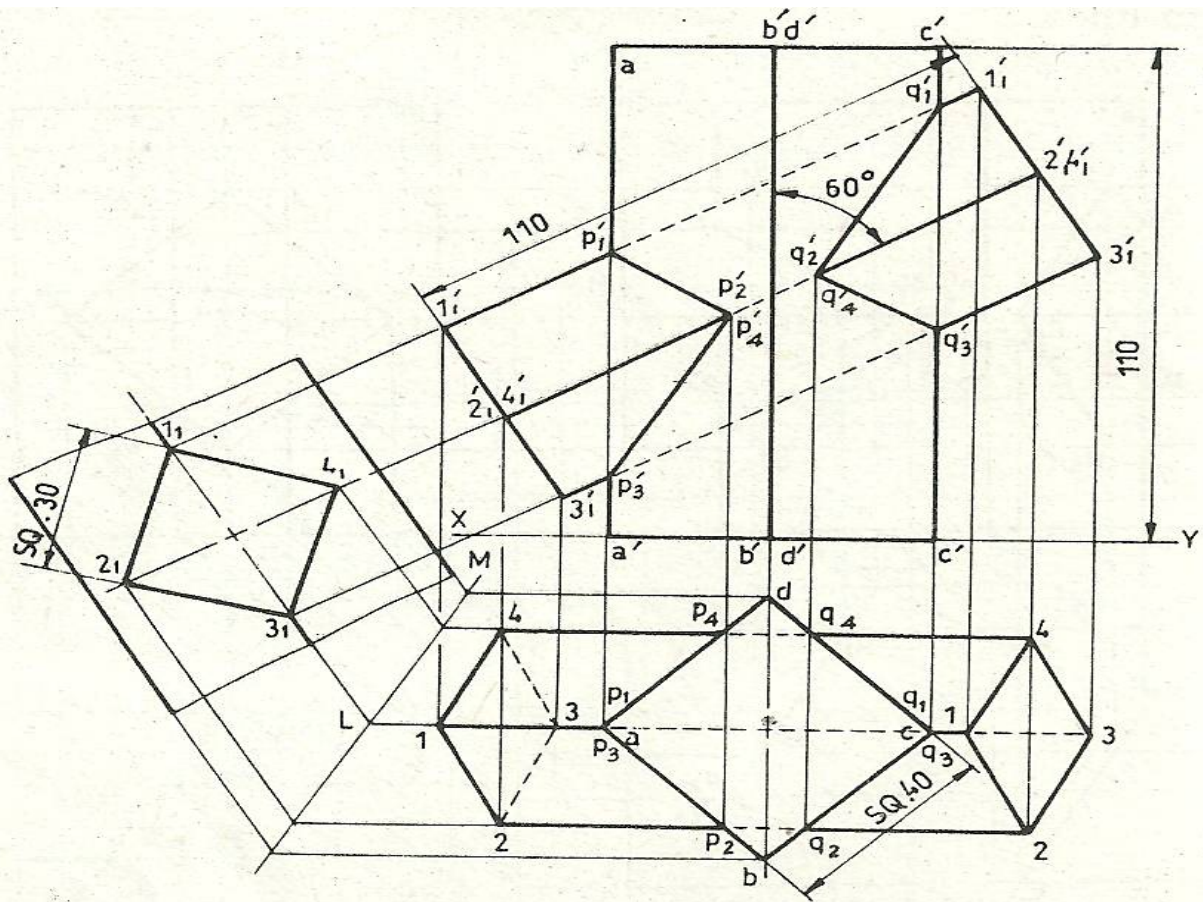
Example 4

A vertical square prism, base 40 mm side and height 110 mm, is completely penetrated by another square prism of base side 30 mm and length 110 mm. Faces of both the prisms are equally inclined to the V.P. The axis of the intersecting prism is intersecting the axis of the vertical prism inclined at 60° to it. Draw the projections showing lines of intersection

Solution:

- (i) Draw the two views of the vertical prism. Then draw the auxiliary view of the intersecting prism perpendicular to the axis and also the partial auxiliary view of the vertical prism.
- (ii) Name the edges of the intersecting prism in the auxiliary view as $1_1, 2_1, 3_1,$ and 4_1 . project these points in the top view on the on the face of the vertical prism as p_1, p_3, p_2 and p_4 . project these points on the corresponding lines in the front view. For example, p_2 is projected as p'_2 online $2'_1 2'_1$. similarly, project all the points and join them in correct sequence. similarly, obtain the other sets of points q'_1, q'_2, q'_4 and q'_3 and complete the view.
- (iii) Project the points of end faces $1'_1, 2'_1, 4'_1$ and $3'_1$ as points 1, 2, 4 and 3 on the corresponding lines and join them to complete the views.

Drawing:



Exercise 1

A vertical square prism, base 40 mm side and height 110 mm is intersected by another square prism, base 24 mm side and length 110 mm, the axis of which is parallel to the V.P. and inclined at 30° to the H.P. or at 60° to the axis of vertical prism. The axes of the two prisms are 6 mm apart and their faces are equally inclined to the V.P. Draw the projections showing lines of intersection Adopt the same method as explained in problem 21.4:

Exercise 2

A vertical square prism, base 40 mm side, height 120 mm, is intersected by another square prism, base 40 mm side and length 120 mm, the axis of which is parallel to the V.P. and inclined at 30° to the H.P. The axes of the two prisms are 7 mm apart and their faces are equally inclined to the V.P. Draw the projections showing lines of intersection. Adopt the same method, as explained in problem 4 above

Exercise 3

A square pipe of 30 mm side has a similar branch of 24 mm side .The axis of the main pipe is vertical and that of the branch pipe is parallel to the V.P. and inclined at 45° to the H.P. at a distance of 20 mm from the face of the vertical prism .The axes of the two pipe are 5 mm apart and their faces are equally inclined to the V.P. Draw the projections of the pipe, showing lines of intersection Also develop the surfaces of both the pipes. The line of intersection between two pipes is obtained in the same manner as explained in problem 2 and as shown in 4

The development for both the pipes is laid out parallel to the edges of the respective pipes. The construction of the development is clearly shown in the diagram.

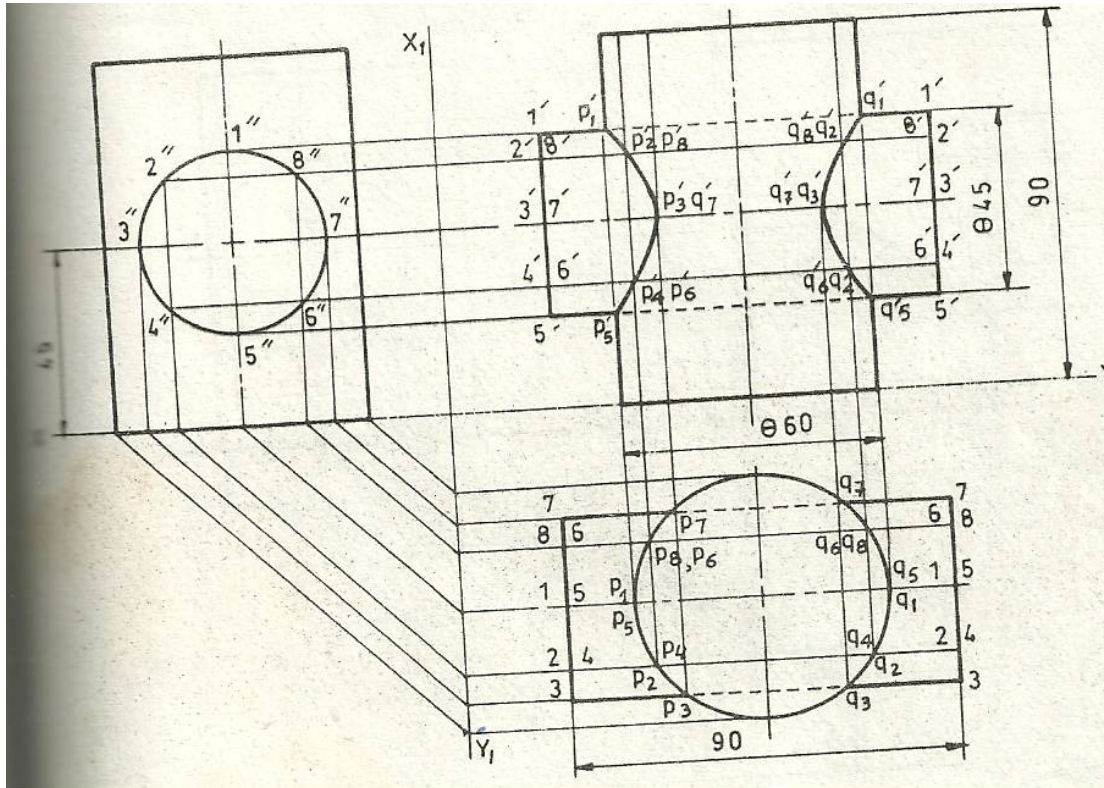
INTERSECTION OF TWO CYLINDERS

As the cylinders have their lateral surfaces curved, the line of intersection between them will also be a curved line and will appear in only one view. Points of curved line may be located by any one of the two methods. For accurate curve, deal with key points, and other points may be considered to know the flow of the curve.

Example 5

A vertical cylinder of 60 mm diameter and axis 90 mm, is completely penetrated by another cylinder of 45 mm diameter and axis 90 mm, their axis bisecting each other at right angles. Draw their projections showing curves of penetration, assume the axis of penetrating cylinder to be parallel to the V.P.

Draw the three views of the vertical cylinder first and then of the horizontal cylinder. show lines for eight generators in the horizontal cylinder in all the three news.



(a) Line method : mark points 1'',2'' etc, in the side view and project them on the top view on the surfaces of vertical cylinder as p₁, p₂ etc, and project them to p'₁, p'₂, etc, on corresponding lines 1'-1',2'-2', etc in the front view

Draw the required curve on both sides of axis through points thus located. Here 1'',3'',5'' and 7'' are key points and 2'',4'',6'' and 8'' are intermediate points.

(b) Cutting plane method: Assume a series of horizontal or vertical cutting planes through the common surfaces, cutting both the cylinders. If horizontal cutting planes are considered, sections of horizontal cylinder will be rectangles, while those of vertical cylinder will always be the circle of

the same diameter as its own. Points at which sides of the rectangle intersect will lie on the curve of intersection.

If vertical cutting planes are introduced, the sections of both the cylinders are rectangles. Points at which sides of rectangles intersect, will lie on the curve of intersection as shown in Fig. 21.8 (b) above.

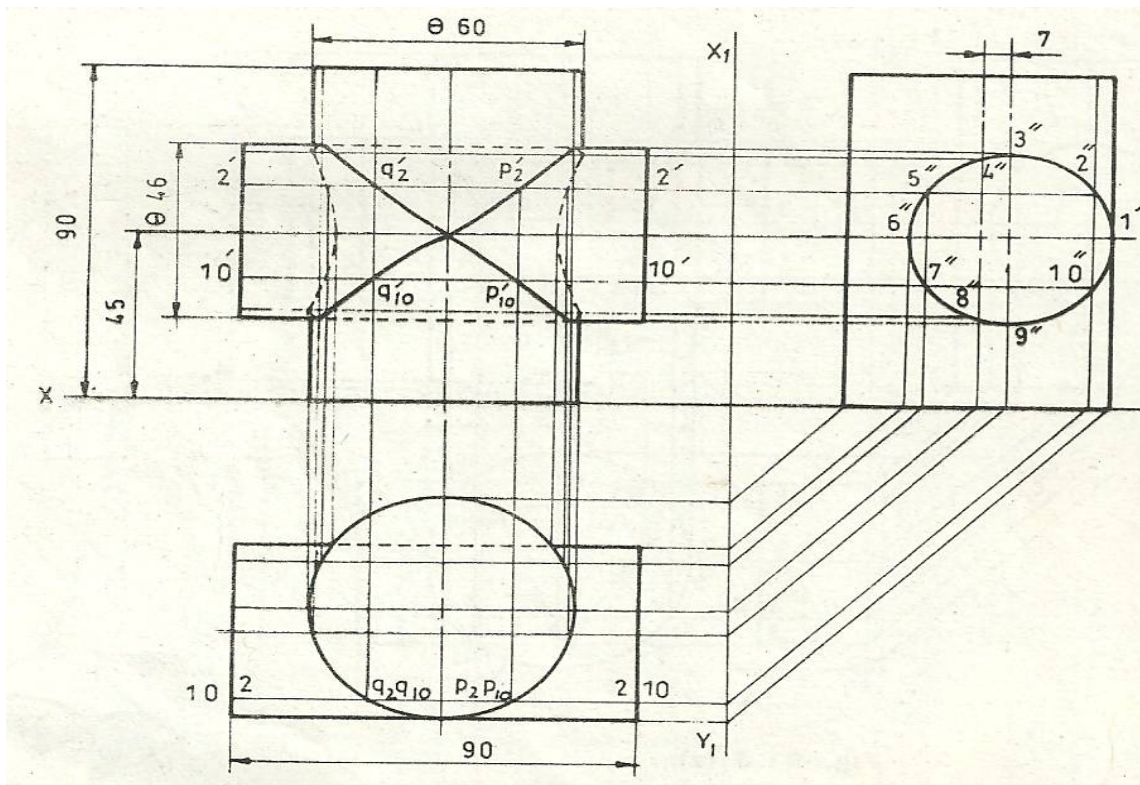
Example 6

(i) A vertical cylinder, of 60 mm diameter, axis 90 mm long, is penetrated by another cylinder of 46 mm diameter, length 90 mm, the axis of which is parallel to both the H.P. and the V.P. The two axes are 7 mm apart. Draw the projections showing curves of intersection.

(ii) Draw the projections of both the cylinders. Locate key points 1", 3", 4", 6", 8", 9" in the side view. Also, locate intermediate points 2", 5", 7" and 10" in the same view

(iii) Project these points on the curved surfaces of the vertical cylinder in the top view, which is a circle.

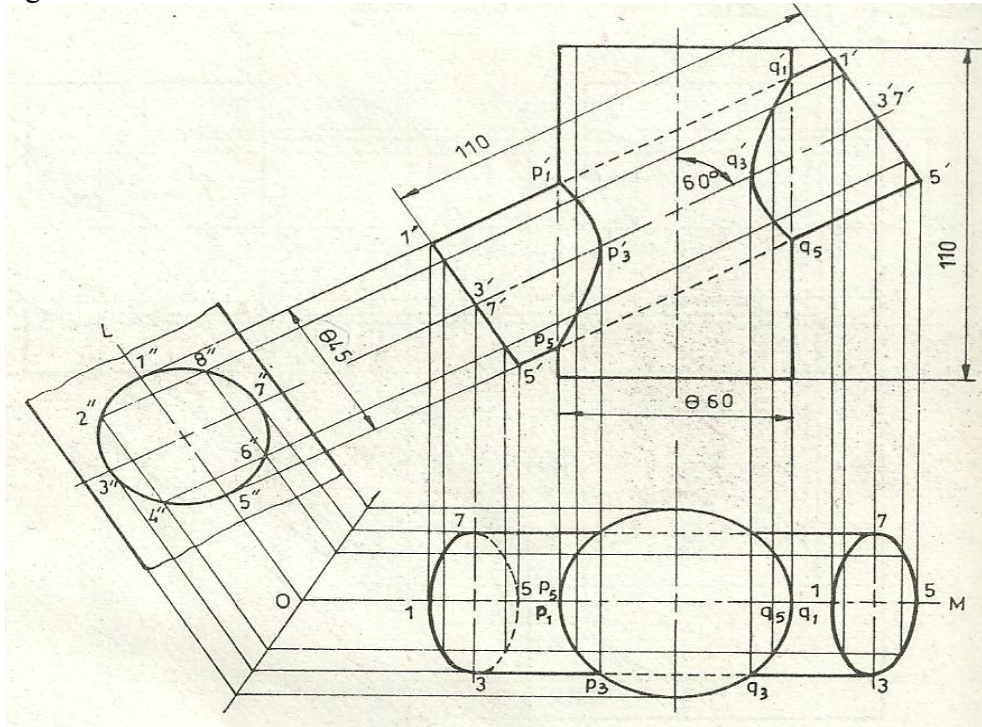
(iv) Project these points in the front view on the corresponding generators. For example, p_2-q_2 from top view is projected up in front view as $p'_2q'_2$ on $2'-2'$. Similarly, project all points in the front view. Join the points in the correct sequence to obtain the curve of intersection on both sides of the axis.



Example 7

A cylinder of 60 mm diameter, length 110 mm having its axis vertical, is penetrated by another cylinder of 45 mm diameter, length 110 mm. The axis of penetrating cylinder is parallel to V.P. and bisect the axis of vertical cylinder, making an angle of 60° with it. Draw the projections showing curves of intersection.

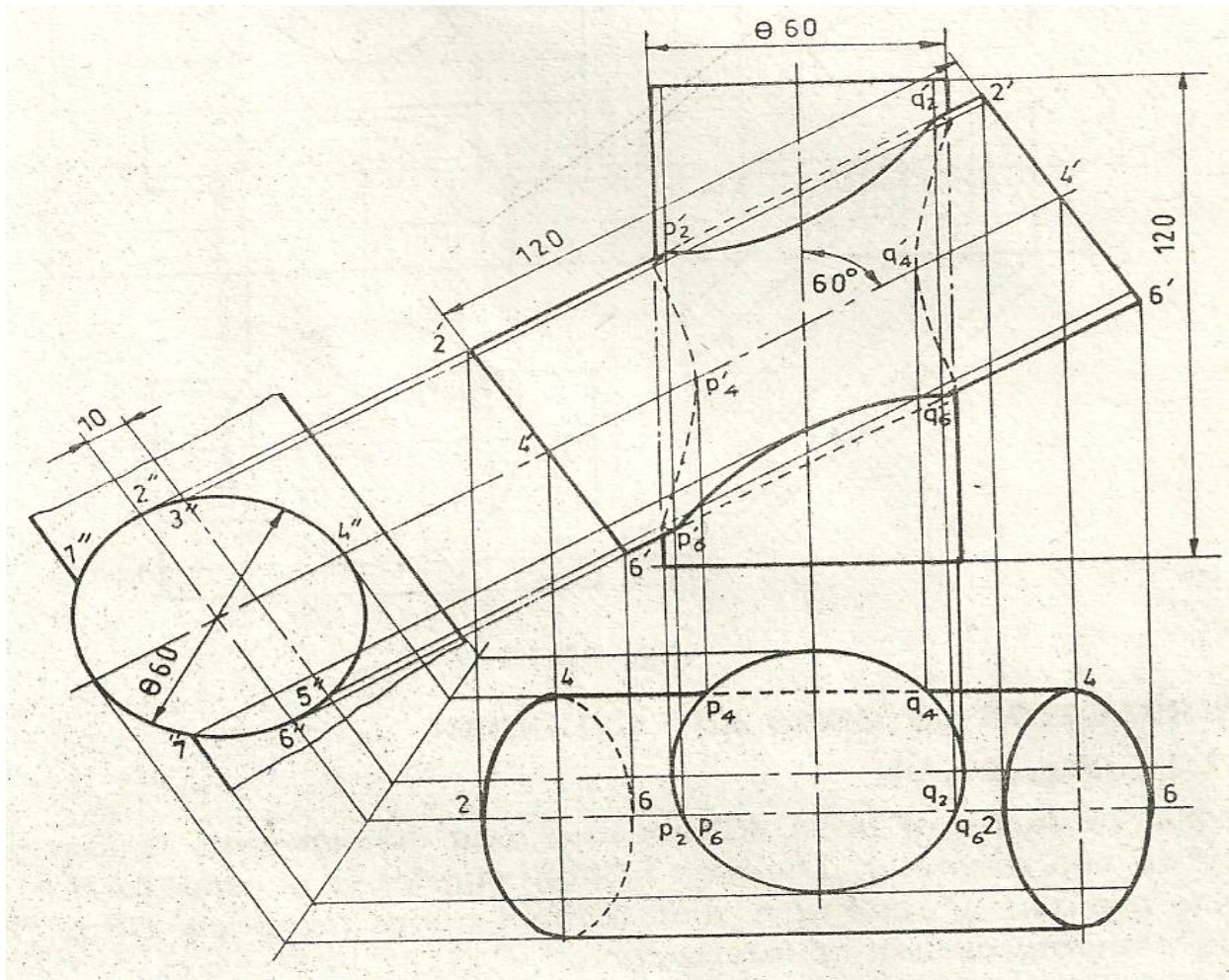
- (i) Draw the projections of vertical cylinder. Then, draw the front view and auxiliary view of intersecting cylinder perpendicular to the axis.
- (ii) Consider key point 1'',3'', 5'' and 7'' and intermediate points 2'',4'', 6'', and 8'', in the auxiliary view .
- (iii) Project the points from auxiliary view to the top view on the circular view of the vertical cylinder.
- (iv) From the top view, project these points vertically and locate on the corresponding elements of intersecting cylinder. For example, p₃ and q₃ from top view will be located as p'₃q'₃ on element 3'-3'. Similarly, locate other points and join them with smooth curve.
- (v) The end faces will be projected and eclipse in the top view, as are clearly shown in the figure:



Example 8

A cylinder of 60 mm: diameter, length of axis 120 mm, having its axis vertical, is penetrated by another cylinder of 60 mm diameter, length 120 mm. The axis of the penetrating cylinder is parallel to V.P. and is 10 mm away from the axis of vertical cylinder, making an angle of 60° with it. Draw the projections showing curves of intersection.

- i. Locate the key points in the auxiliary view. Project these points in the top view on the curved surface of the vertical cylinder. From top view, project the points in the front view on corresponding elements.
- ii. Complete the curve as Shown; care must be taken to locate points on/of hidden curve.



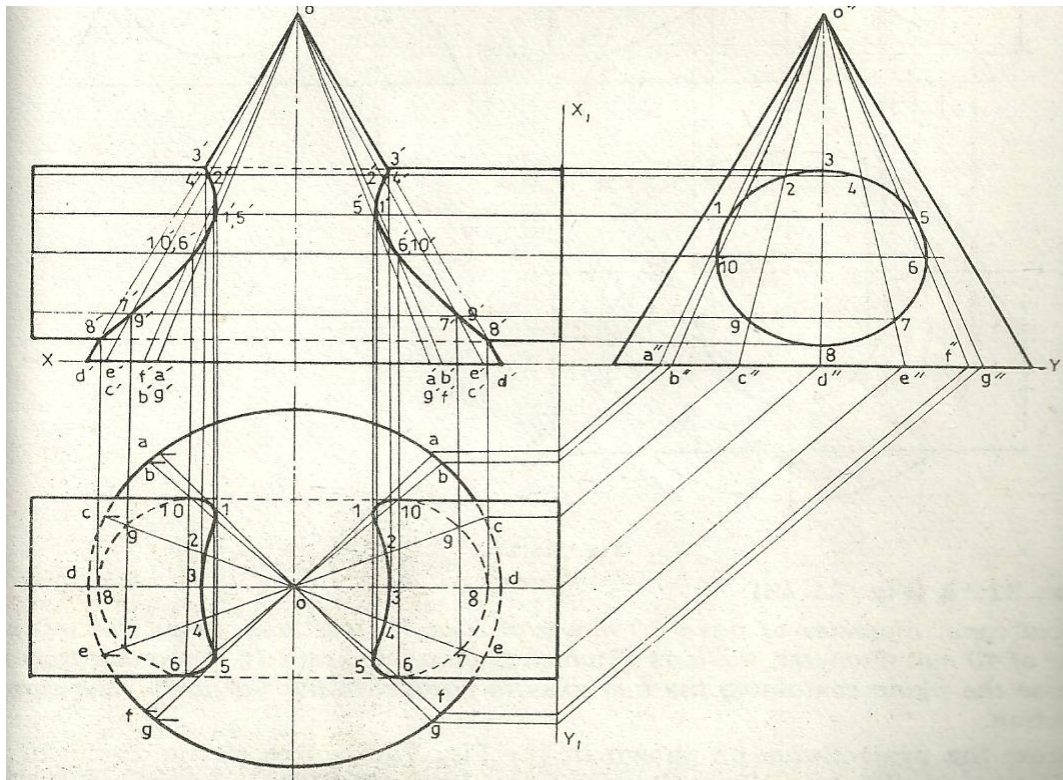
INTERSECTION OF CONE AND CYLINDER

Example 9

A vertical cone, of base 80 mm, axis 80 mm long, is completely penetrated by a cylinder of 40 mm diameter and 100 mm in length. The axis of the cylinder is parallel to H.P. and V.P. and intersects the axis of the cone at a point 25 mm above the base. Draw the projections of the solids showing curves of intersection.

- Draw the three views of the combination as per given dimensions. The surface of the cylinder will be seen as a circle in the side view.
- Draw numbers of lines (representing generators of the cone) $0''a''$, $0''b''$, etc, in the limits of the circle of cone and symmetrical on both sides of axis. Points where these lines intersect the circle lie on the curve of intersection.
- To project them in the front and top view, first project the lines in both the views, then locate the position of these points on them. Let us consider the line $0''c''$ in the side view. Locate its position $0c$ in the top view, as shown, and project its front view $o'c'$. Project points $2''$ and $9''$ to points $2'$ and $9'$ on $o'c'$ and from there, to 2 and 9 on oc in the top view.

- iv. Project all such points in the same manner and draw the required curve through intersecting them



Exercise 4

A vertical cone, base 80 mm diameter, axis 80 mm long, is penetrated by a horizontal cylinder of 40 mm diameter, 100 mm long, the axis of which is 25 mm above the base of the cone, 8 mm away from the axis of cone and parallel to V.P. Draw the projections showing the curve of intersection. Solve the problem by cutting plane method.

Exercise 5

A vertical cone, diameter of base 80 mm and axis 70 mm long, is penetrated by a cylinder of 40 mm diameter, the axis of which is parallel to and 10 mm away from that cone, also the plane containing the two axes is parallel to the V.P. Draw the curve of intersection.

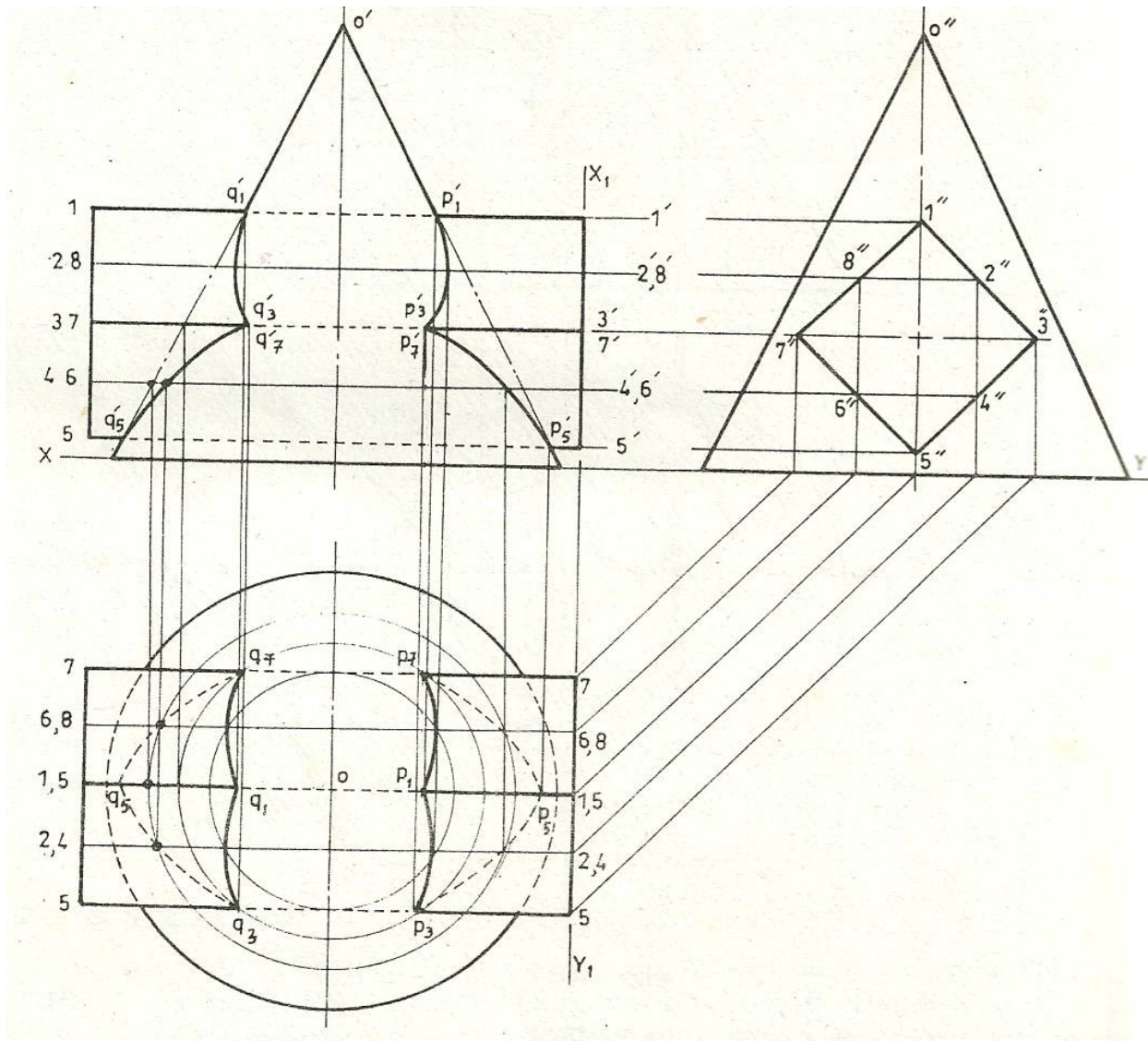
INTERSECTION OF CONE AND PRISM

Example 10

A vertical cone, of base 80 mm and axis 80 mm long, is completely penetrated by a square prism of 30 mm side and 100 mm of length. The axis of the prism is parallel to both H.P and V.P., its faces are equally inclined to H.P., and intersect the axis of the cone at a point 25 mm above the base. Draw the projections of the solids showing curve of intersection.

- i. Draw the projections of the combination as per given dimensions.
- ii. The prism will be seen as a square in the side view. Key points are 1'', 3'', 5'' and 7'' in the side view and 2, 4, 6 and 8'' are intermediate points, taken in the middle of respective sides of square.

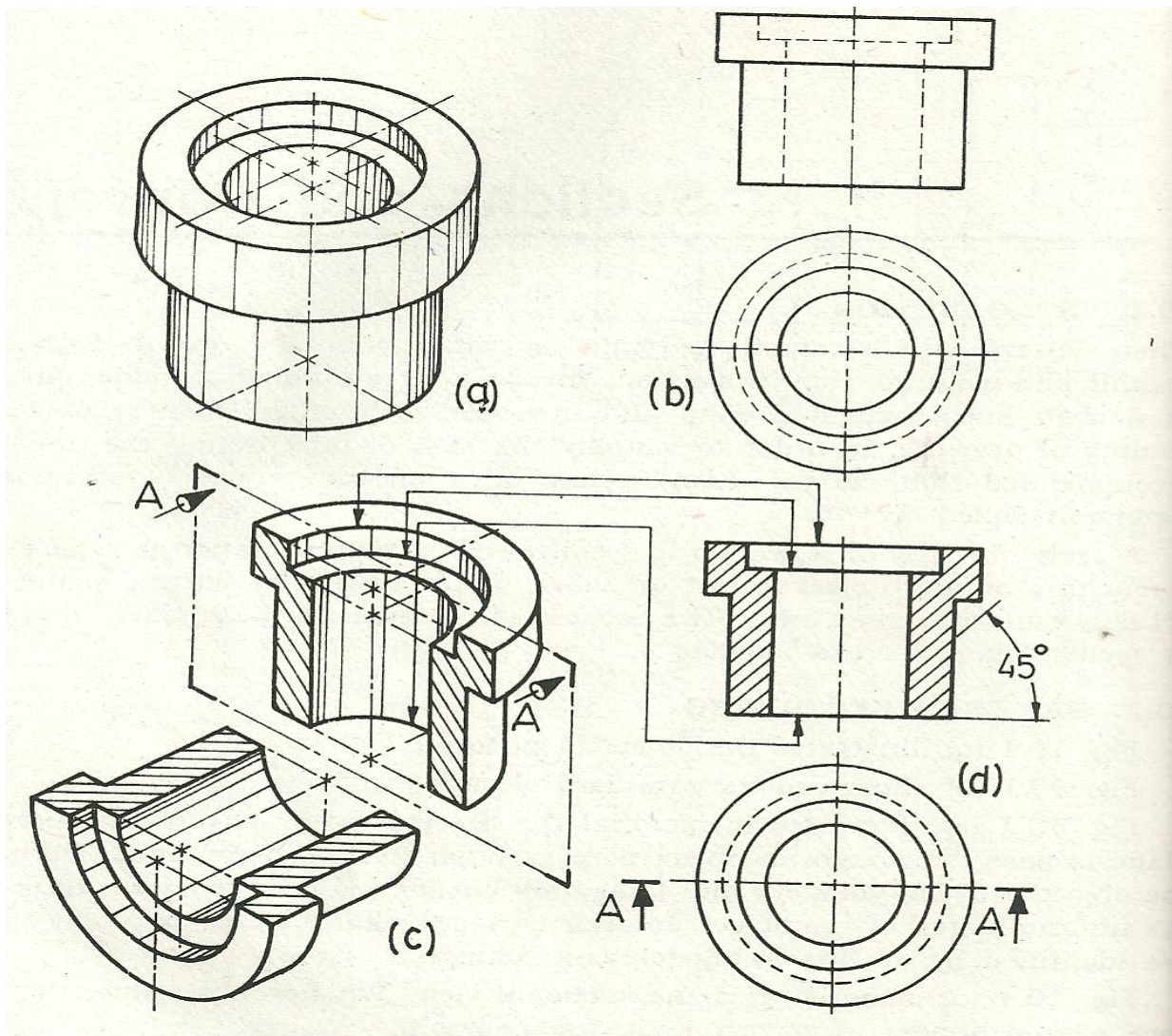
- iii. Pass horizontal cutting planes through these points. Horizontal section of cone will be a circle and that of the prism a rectangle, in top view.
- iv. Finally, solve the problem as per intersection of cone and cylinder.



SECTIONING

When objects with a complicated interior constitute construction are drawn both in orthographic and pictorial, their projections involve a large number of hidden details which overlap and intersect each other, thereby confusing the reading of the drawing, it is recommended that careful thought may be given to the use of sectional views commencing called sections.

A sectional view may or a section is a conventional representation in what a part of machine or any object is cut or sliced by an imaginary cutting plane (but not actually cut), or broken away. The exposed or cut surface of the material is indicated by section lining or cross-hatching. The cutting plane is denoted by long chain this line (thick at the ends).



If the notation or dictation between materials is not important on drawing, the general use (cast man) symbol is normally used. Therefore, it is recommended that in each case the material used should be indicated by a note in the bill of material. Reference letters are commonly used to correlate the view with the bill of materials.

In view of the fact that there is large number of materials used for making the object, it is desirable to use different conventional section to differentiate various materials.

RULES FOR SECTIONING

- (1) The edge of section cutting plane should be shown in the relevant view. The cutting plane consists of a Centre line with a thick short line each end and arrows pointing in the direction of viewing letters at each end of the section line should be included.
- (2) If it is obvious where the cutting plane should be there is no need to include it in the drawing e.g if the cutting plane is obviously along the Centre line of a plane for a section in the front view, the cutting plane line need not be include in the drawing.
- (3) The cut face of a section should be 'hatched with line at 450 and 3 or 4 mm apart.

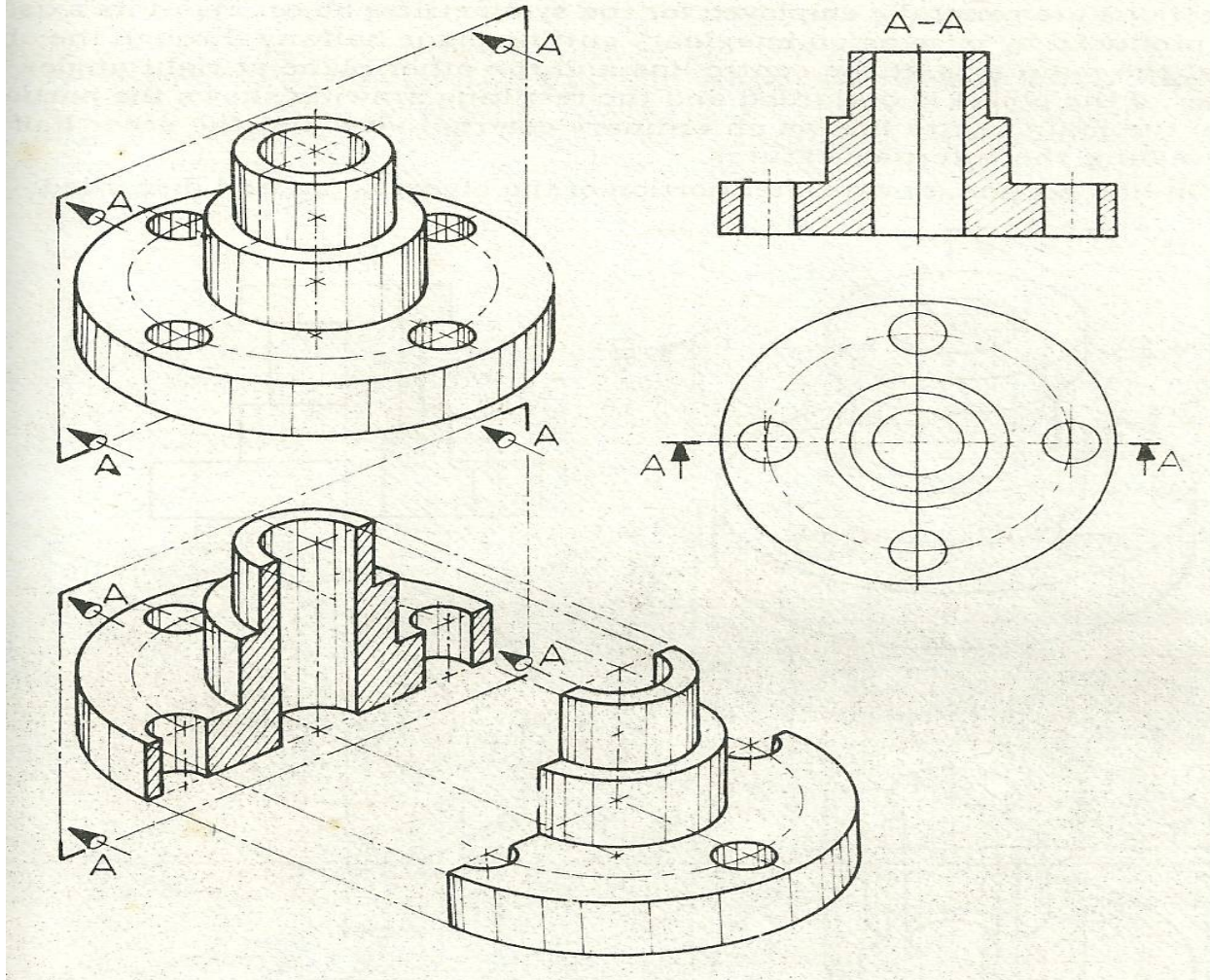
(4) If there are two adjacent cut faces in a section, the hatching of one should be at a different angle to that of the other. Angles other than 45° and spacing different to 3-4mm may be used when several faces are adjacent to each other.

TYPES OF SECTIONING

The following are the common types of sectioning

- (a) Full Section
- (b) Half Section
- (c) Off-set/Staggered Section
- (d) Revolved Section
- (e) Removed Section
- (f) Partial or broken out section

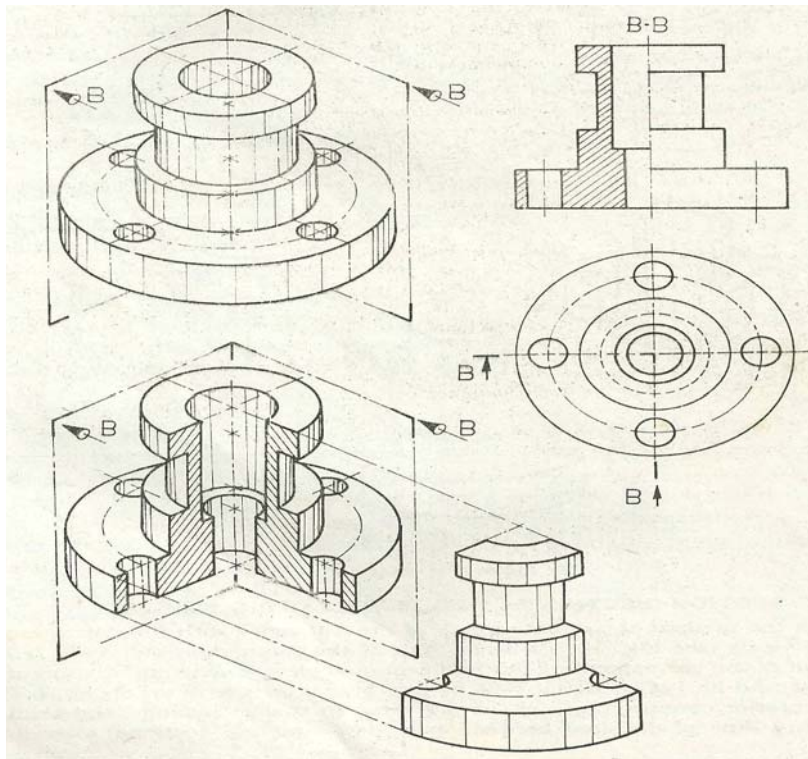
FULL SECTION: This method is used for majority of the sectioning work. In this method, a longitudinal or imaginary cutting plane extends through the entire object showing the view in section. Full section can be obtained by introducing an imaginary horizontal cutting.



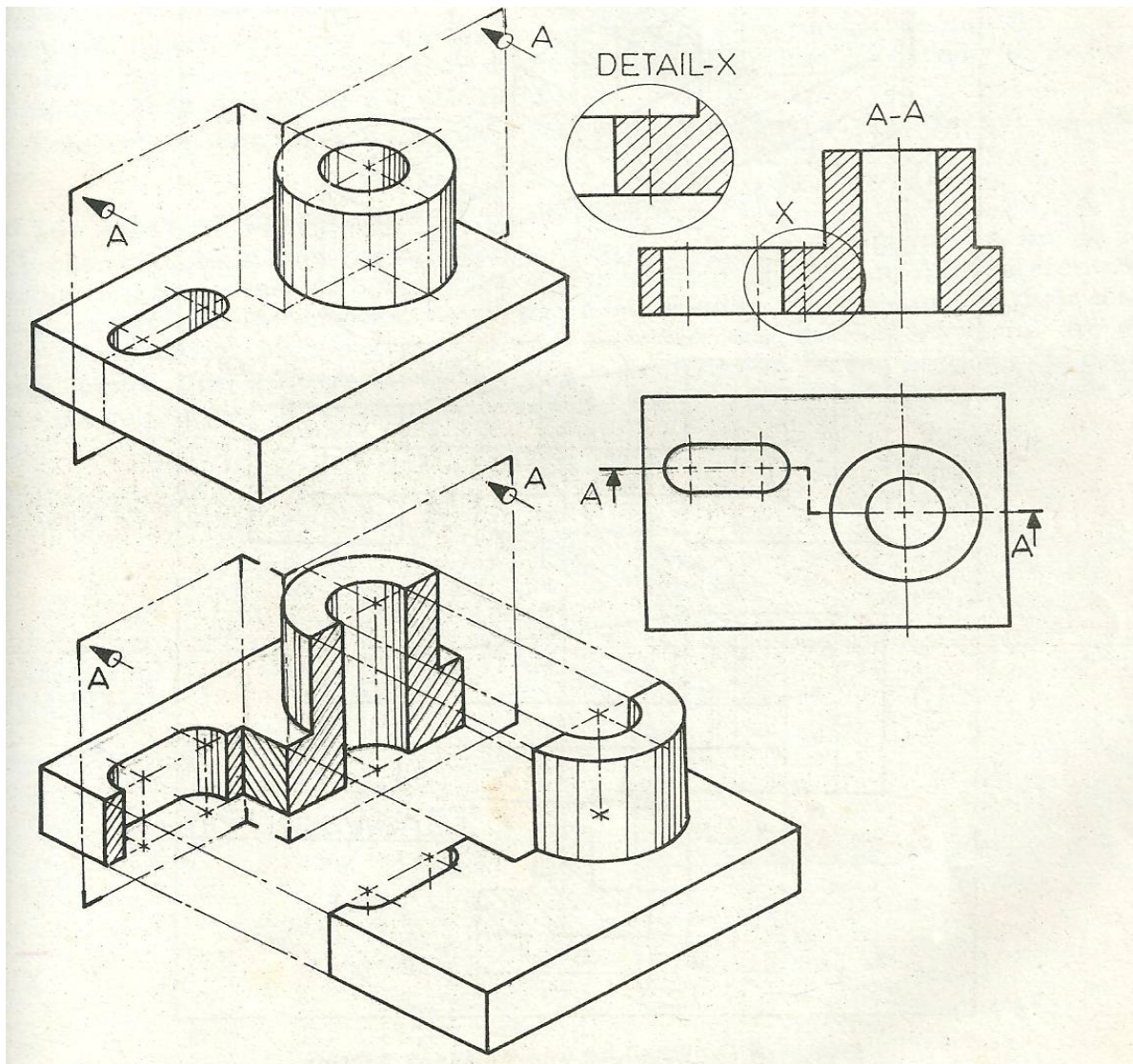
In doing this, we are imagining the object to be fractured in the places. The central parts of the object are not shown and the two other parts are brought together. The break-line showing the boundary of the break will depend upon the shape of the cross-section and the material of the object.

Plane crosses the entire object. In full section, half portion of the object is between observer and the cutting plane is imagined to be cut and removed.

HALF SECTION: This can also be referred to as **SECTION IN TWO INTERSECTION PLANES**. It is the method of combining half of the full views with the corresponding sectional view, in this type of section, half of the main right-half sectional view. Such combination of the half of the view and half of sectional view help to give a picture of an object both its exterior and interior construction in a single view. Half sections are generally employed for the symmetrical object about its axis. Half section is produced by passing on imaginary cutting plane halfway through object, stopping at the main axis at the centre line and the other plane at right side to it. One quarter of the object is discarded and the resulting drawing shows the portion on the side of the main centre line.



OFF-SET SECTION/STAGGERED SECTION



As you can observe from the diagram the cutting plane instead of pursuing its way continuously, changes its direction and pursues a zig-zag course through the object. In the figure, off-set plane A-A cuts its way as shown. The cut surface is brought out with cross-hatching.

REVOLVED SECTION: Also referred to as super imposed. If they are on the projection i.e if they are obtained by rotating the corresponding view until it is in the same plane to that of the corresponding view. We superimpose the figure of the section as it holes, recesses, hollows, ribs and other like forms can be represented with sufficient clarity by applying revolved sections.

REMOVED SECTION: The plane figure, resulting from the intersection of a cutting plane and the body of an object drawn on the extension of the cutting plane line is called the removed section of the object.

PARTIAL OR BROKEN OUT SECTION: this type of section is drawn when a full or half section is not necessary. Some solid objects without cavities are not represented in sections because

the exterior views give a sufficiently clear picture of their shape and constitution. But if such an object has a hole or recess partial sections are drawn. The portion is bounded by a thin irregular free-hand break line.

EXCEPTIONS TO THE GENERAL RULES

The following features are exempted from been sectioned. They are always shown by outside view, within a section. These are: Bolts, nut, washers, screw, spindles, webs and ribs, spokes of wheels, and fasteners of any type.

CONVENTIONAL BREAKS

In a situation where long parts of uniform cross-section connect be accommodated on the size of drawing sheet, we can use conventional symbols called conventional Breaks (on Fractures) to reduce the length of projection.

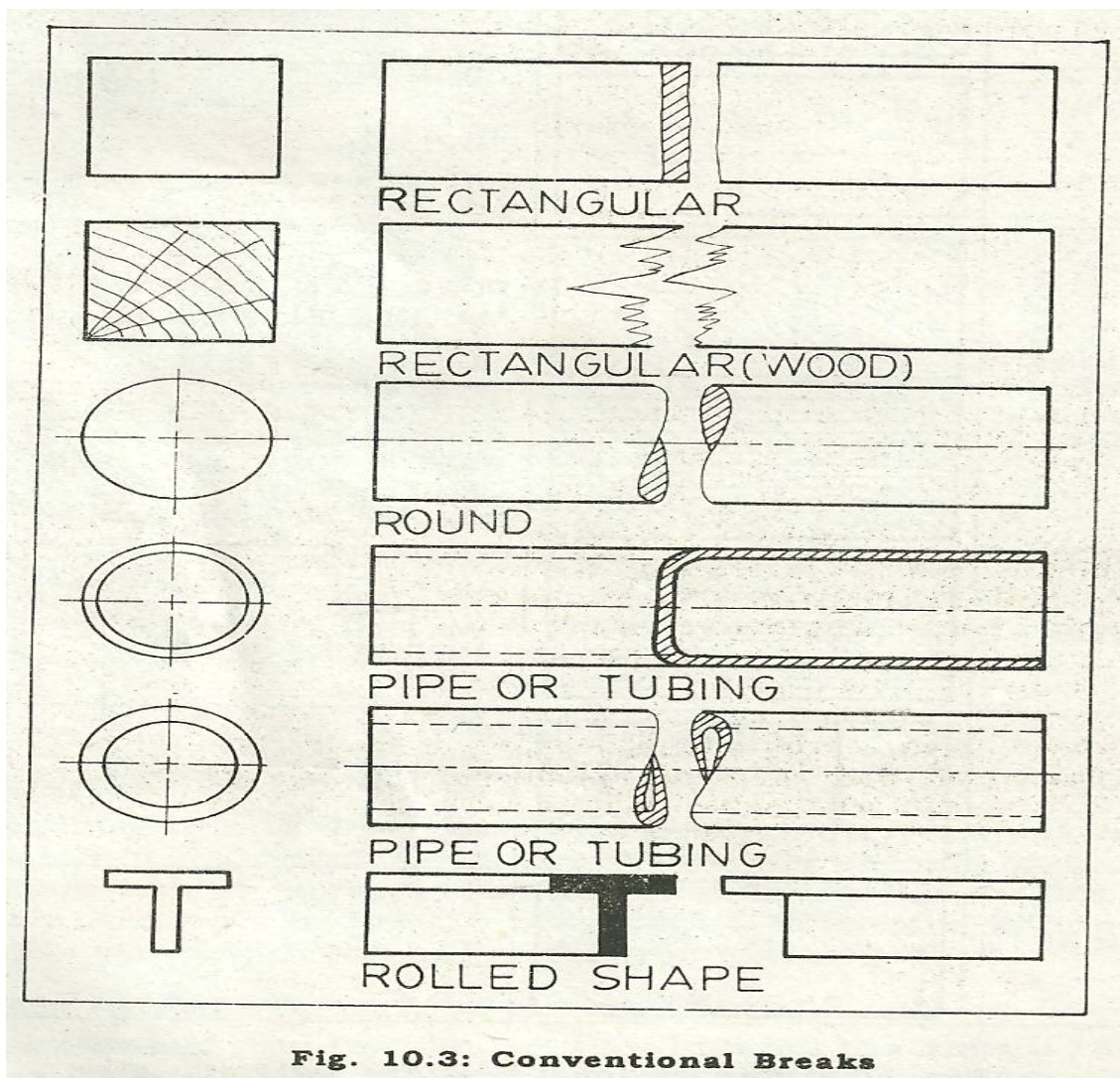


Fig. 10.3: Conventional Breaks

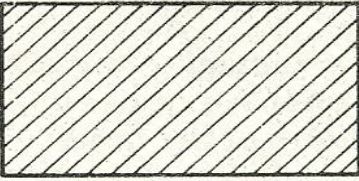
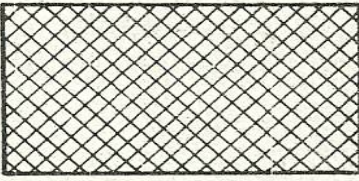
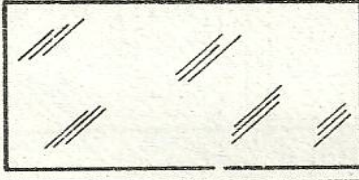
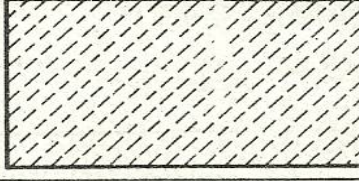
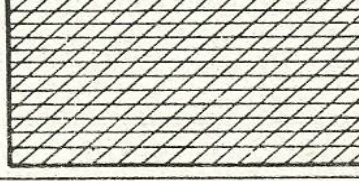
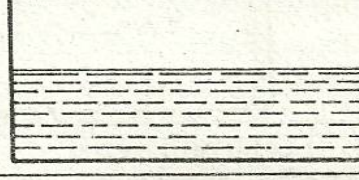
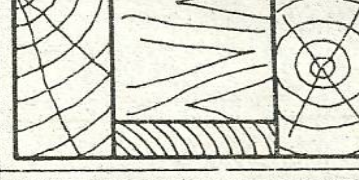

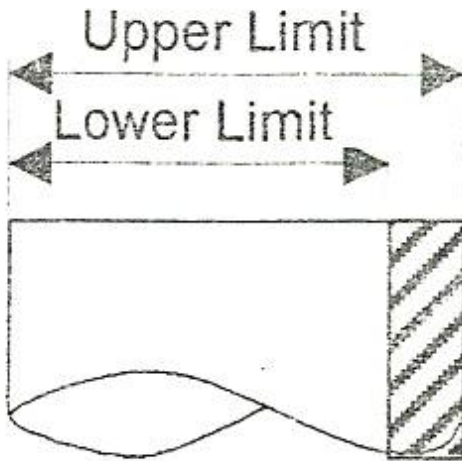
TYPE	CONVENTION	MATERIAL
METALS		STEEL, CAST IRON, COPPER AND ITS ALLOYS, ALUMINIUM AND ITS ALLOYS, ETC.
		LEAD, ZINK, TIN, WHITE-METAL, ETC.
GLASS		GLASS.
PACKING AND INSULATING MATERIALS		PORCELAIN, STONEWARE, MARBLE, SLATE, ETC.
		ASBESTOS, FIBRE, FELT, SYNTHETIC RESIN PRODUCT, PAPER, CORK, L INOLEUM, RUBBER, LEATHER, WAX, INSULATING AND FILLING MATERIALS.
LIQUIDS		WATER, OIL, PETROL, KEROSENE, ETC.
WOOD		WOOD, PLYWOOD, ETC.
CONCRETE		

Fig. 10.2: Conventional Sections for Materials

INTERCHANGEABILITY OF OBJECT

Interchangeability can be defined as the ability of component to be able to be replaced by another component of the same shape and size. The necessity for this in mass produce component has increased the importance of limit and tolerance on the sizes of parts. Suppose the size of a shaft to be exactly 35mm. the method of production will be very slow and the production will demand great skill and as such the product will be expensive. To prevent this, a suitable system of limit and fits has been introduced whereby all components are made to a specific size with narrow limits.

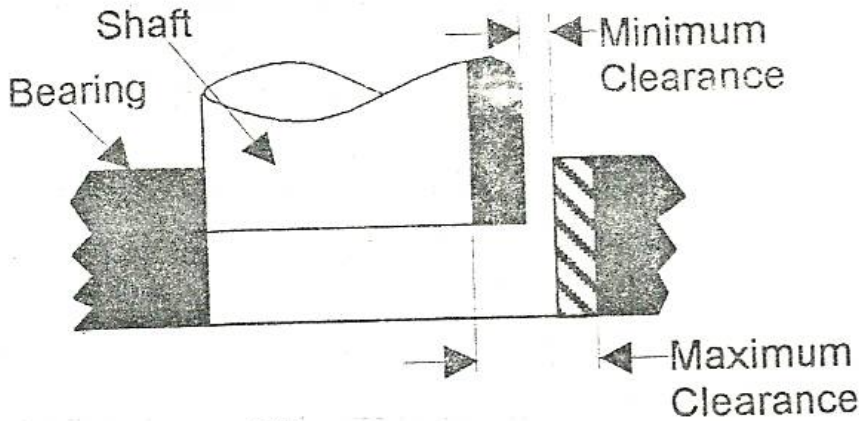
LIMITS: This can be defined as the two extreme permissible sizes for dimensions or allowance for manufacturing errors or the minimum and maximum size of a production. For example, with a nominal diameter of 50 mm, a slight error acceptable. The large dimension is called the upper limit while the smaller dimension is called the lower limit.



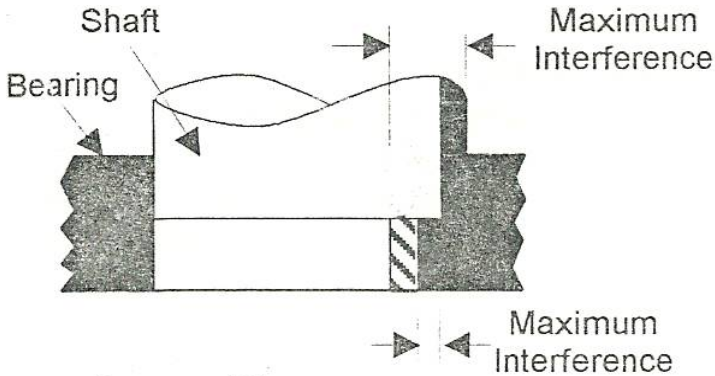
FITS: This can be defined as the relationship between an internal and external diameter of an object (i.e shaft and bearing). Most manufacturers will not allow parts to be too tight or too loose.

Types of Fits can be divided into three:

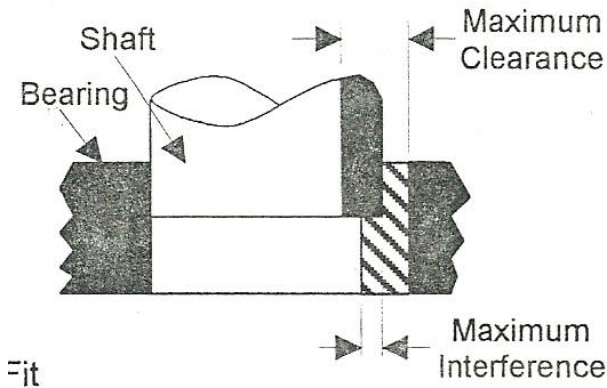
(a) **CLEARANCE FIT:** This is a case in which the shaft is always smaller than the hole into which it fits



(b) **INTERFERENCE FIT:** In this case, the shaft is bigger than the hole into which it fits.

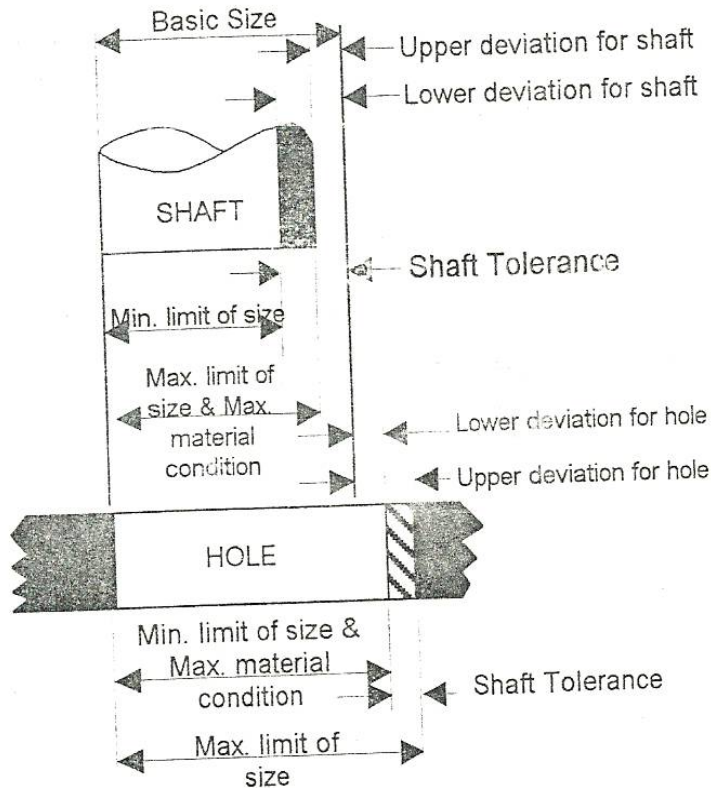


(b) **TRANSITION FIT:** this combines the possibility of having interference or clearance fit in one assemblies.



ELEMENTS OF INTER-CHANGEABLE SYSTEMS

- (1) **Basic Size:** This is the size in relation to which all limits of size are fixed and will be the same for both the male and female parts of the fits.
- (2) **Normal Size:** this is the size by which a component is reference to as a matter of convenience, i.e 25mm, 50mm, 60mm thread/hole/shaft.
- (3) **Actual Size:** is the measured size
- (4) **Limits of size:** These are the maximum and minimum possible sizes acceptable for a specific dimension.



(5) **Tolerance:** This is the total permissible variation in the size of a dimension, and is the difference between upper and the lower acceptable dimensions. There are two types:

(a) **Unilateral Tolerance:** When the permissible Tolerance is all negative or all positive. For example $40^{+0.01\text{mm}}$ for hole

$$-0.010\text{mm}$$

and $40^{-0.02\text{mm}}$ for shaft

(b) **Bilateral Tolerance:** When the permissible tolerance is partly +ve and partly -ve- e.g example $40^{+0.01\text{mm}}$ for hole

$$-0.010\text{mm}$$

(6) **Clearance:** This is the free space between the bearing and the shaft.

(7) **Allowance:** it concerns with the mating parts. It is the difference between the high limit of size of the shaft and the low limit of size of the mating hole. It may be +ve or - ve.

Types:

(a) **Minimum Allowance** – is the difference between d minimum dia of the bearing and maximum dia of d shaft.

(b) **Maximum Allowance** – is the difference between dia of bearing and the minimum dia of shaft.

Example: the limits shown on a drawing for mating hole and shaft are

$$\begin{array}{l} \text{example Shaft: } 20 \begin{array}{l} \text{upper limit} \\ -0.00\text{mm} \\ \text{lower limit} \\ -0.08\text{mm} \end{array} \\ \\ \text{Hole } 20 \begin{array}{l} +0.05\text{mm} \\ -0.15\text{mm} \end{array} \end{array}$$

Find the allowance and type of fit that exist between hole and shaft.

$$\begin{array}{l} \text{Solution: Maximum Allowance} = \text{biggest hole} - \text{smallest shaft} \\ \text{Minimum Allowance} = \text{smallest hole} - \text{biggest shaft} \end{array}$$

$$\text{Maximum} = 20.15 - 19.92$$

$$\text{Minimum} = 20.05 - 20$$

PICTORIAL DRAWING

1.1 INTRODUCTION

Engineers believe in three-dimensional objects and have been used to a system of describing these objects by referring to the length, breadth or depth and height.

In Engineering Drawing, there are two general methods of showing solid objects.

- (a) Pictorial drawing (i.e isometric and oblique)
- (b) Orthographic projection

Pictorial drawing is a three dimensional representation of an object that shows two or more views of the object very clearly and distinctly and gives a pictorial of perspective. It is widely used in engineering drawing, industrial design, architecture, technical illustration and advertisement.

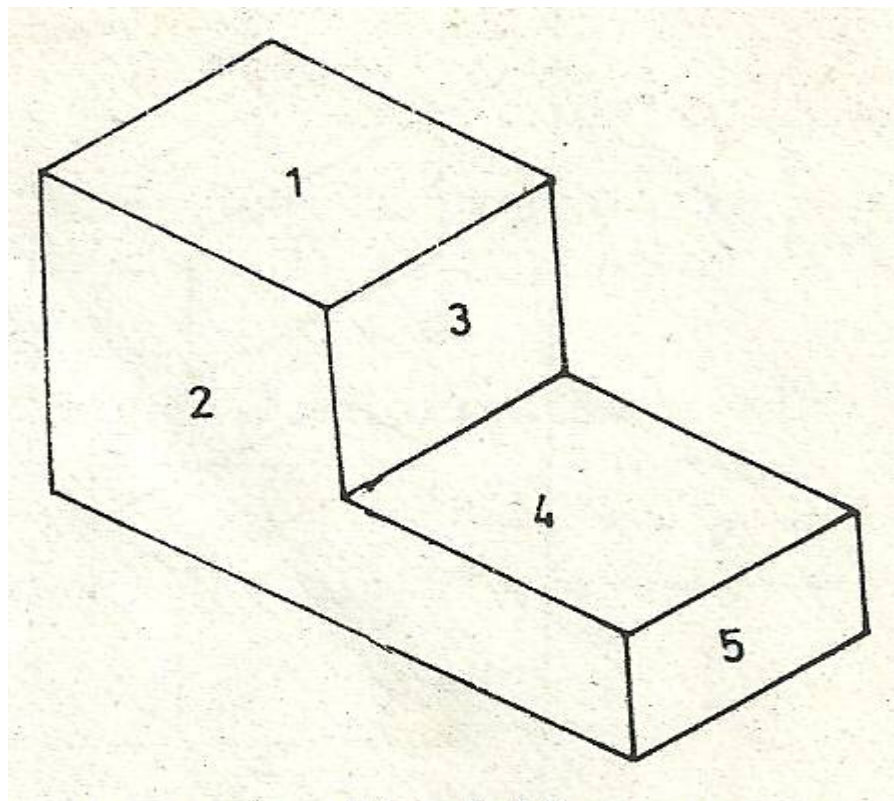
The most commonly used pictorial drawing methods are explained below:

- (a) Isometric Projection (i.e isometric and oblique)
- (b) Oblique Projection
- (c) Perspective Projection

Perspective projection gives the most realistic drawing in terms of vividity and naturality. It is however more difficult and tedious to draw than the others, hence, isometric and oblique projections are widely used in engineering while perspective is preferred in architecture.

1.2 ISOMETRIC PROJECTION

Isometric drawing is a method of producing an overall pictorial view of an object. It shows the three views of an object very clearly although with some distortions, for example, a circle appears as an ellipse. An isometric drawing is set out mainly on three isometric axes. All dimensions must be measured along these axes or on lines parallel to the axes.

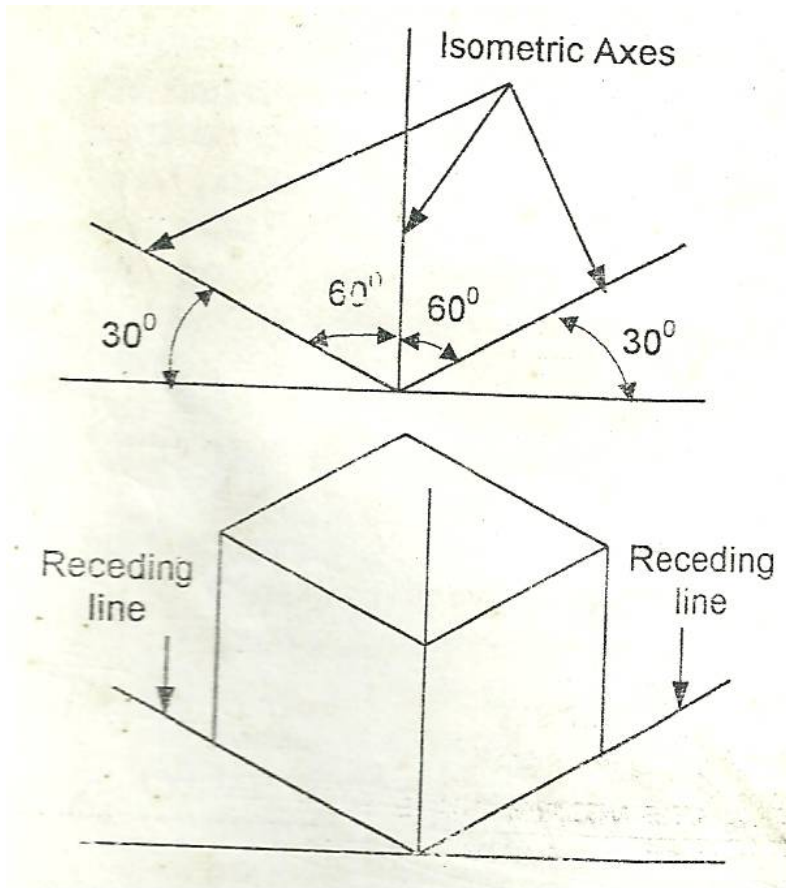


To draw a rectangular block for example, in isometric projection. First draw a horizontal line and then a vertical line and at the point of intersection draw lines (receding lines) one to the right and the other to the left an angle of 30° to the horizontal. The actual lengths of the three sides are marked and the remarked points.

It should be noted that all lines to the left are parallel to the receding line to the left and lines to the right are parallel to the receding line to the right.

1.2.1 OBJECTS COMPOSED ENTIRELY OF ISOMETRIC LINES

The objects are easily drawn as all the measurements in the orthographic views are scaled directly on the isometric view. The best method is to start off by first drawing a box, and then building up the shape of the object inside the box.

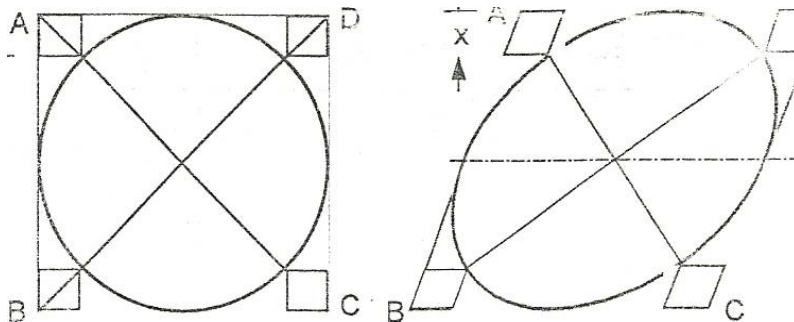


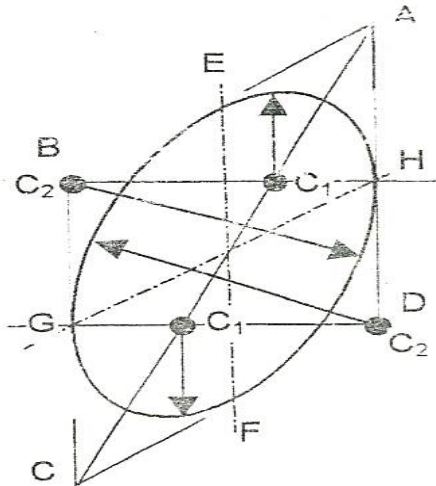
Drawing

Objects with non-isometric lines

Methods of Drawing with a curved Profile or an Isometric Circle

- (a) Method I (fig. 1.4)
 - (i) Draw a circle enclosed in a square ABCD
 - (ii) Draw the diagonals from corner
 - (iii) Draw a small shaped square at the corners of the square touching the circle's circumference.
 - (iv) The small corner squares as well as the other four midlines points on the circle's circumference formed basis for drawing the isometric circles.
 - (v) Draw the ellipse (isometric circle) with the aid of a French curve.





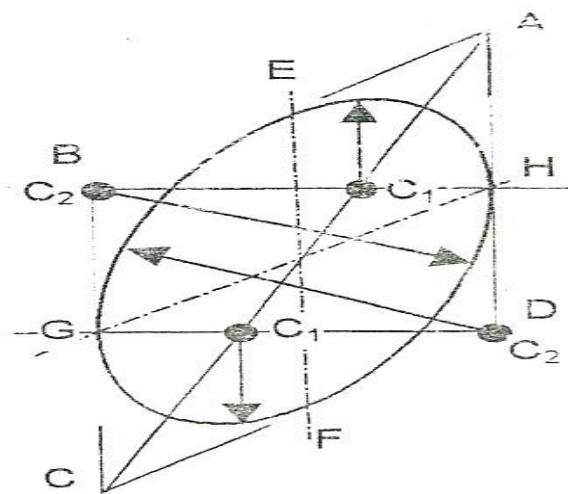
drawing
Fig. 1.4

(a) Method II Approximate Method (Fig. 1.5)

The ellipse occurs so frequently on isometric drawings that an approximate method using circular arcs is often used to draw the isometric circles.

- (i) Draw an isometric square ABCD with slides equal to the diameter of the required circle.
- (ii) Draw the centre lines EF and GH. Their intersection will be three centre of the square ABCD and the centre of the circle to be drawn.
- (iii) Draw the long diagonal AC of the square.
- (iv) Join B and D to the mid-points of the opposite sides. Where these lines intercept the long diagonal are two arc centre.
- (v) The other centers are the corners B and D of the isometric square.

The construction is facilitated if it is noted that all construction lines for the arc centers on the long diagonal are either horizontal or at 60° to the horizontal.



OBLIQUE PROJECTION

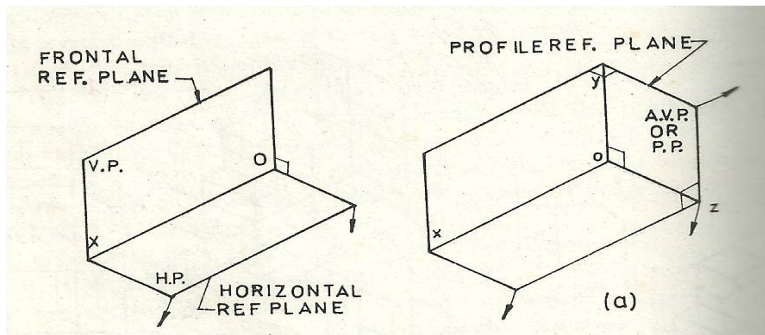
Oblique projection is the second type of pictorial drawing used. It gives an overall picture of an object. Oblique drawing is better used or explaining drawing to non-technical people. Since it is not as pleasing to the eyes as isometric drawings, it is less used.

A surface is parallel to the surface of the drawing sheet so it is easier for construction, and the surface carries the true shape and size or holes are usually placed on the surface that true shape and size since it can be drawn without distortion.

In an oblique projection we arrange axes to represent intersections of three mutually perpendicular faces of an object. Oblique drawings employed 30° , 45° , 60° as the receding edges instead of only 30° as used on isometric. In oblique projection the angle that is commonly used for the receding edges is 45° , the other two are always at 90° (i.e horizontal axis and vertical axis).

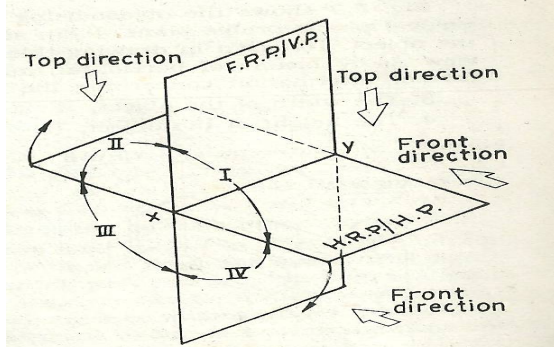
ORTHOGRAPHIC PROJECTION

Orthographic projection shows two or more views of an object in their true and sizes. Such views are quickly and easily drawn but can only be interpreted by someone with good knowledge of engineering drawing. We can build-up the projection of an object in our mind if we imagine parallel rays of light to strike an opaque rectangular block, the shadow so formed on a vertical plane positioned at right angle to the object is the projection of the particular side of the block. Such a projection can be made for the three views of an object.



Projection of an object can be made in any of the four quadrants of an angle but the first quadrants are more widely used because in the third quadrants their views overlap and this can lead to confusion. The view obtained in the first quadrant is called the first angle projection (English System) and in the third quadrant is the third angle projection (American System).

Projection on the vertical plane is the front view or elevation, on the horizontal plane is the plan view and on the auxiliary vertical plane is the side view.



FIRST ANGLE PROJECTION

VIEW

Front

POSITION

the main view, placed at the top of the sheet

Plan (Top view)
 Left-hand End view
 Left-hand End view
 Right-hand End view

Projected down from the front elevation
 Projected to the right of the front elevation
 Projected to the right of the front elevation
 Projected to the left of the front elevation













THIRD ANGLE PROJECTION








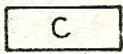

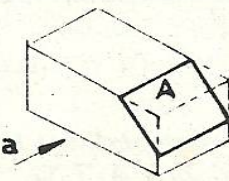
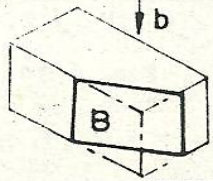
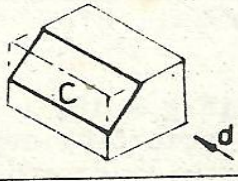
VIEW

Front Elevation
 Right-hand End view
 Left-hand End view

POSITION

Placed under the plan
 Projected to the left of the front elevation
 Projected to the right of the front elevation

NORMAL SURFACES		
TOP VIEW	FRONT VIEW	EITHER SIDE VIEW
HORIZONTAL LINE 	SURFACE 	VERTICAL LINE 
SURFACE 	HORIZONTAL LINE 	HOZIRONTAL LINE 
VERTICAL LINE 	VERTICAL LINE 	SURFACE 
		

INCLINED SURFACES		
TOP VIEW	FRONT VIEW	EITHER SIDE VIEW
FORESHORTENED SURFACE 	INCLINED LINE 	FORESHORTENED SURFACE 
INCLINED LINE 	FORESHORTENED SURFACE 	FORESHORTENED SURFACE 
FORESHORTENED SURFACE 	FORESHORTENED SURFACE 	INCLINED LINE 
		

PROJECTION SYMBOLS

It is necessary to indicate on the drawings the type of system of projection that has been used. This is done by a symbol constituting of an elevation and end view of frustum of a cone, the angle symbol is shown in fig. 2.3a and the their angle symbol in fig. 2.3b

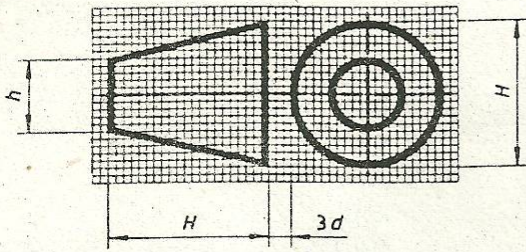


Fig. 8.26

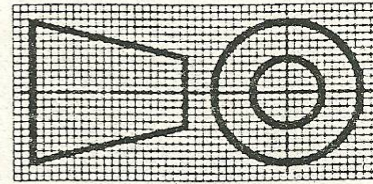


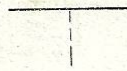
Fig. 8.27

8.17. HIDDEN LINE CONVENTIONS

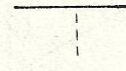
Superimposition of hidden detail on a drawing often leads to vagueness concerning

CORRECT

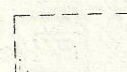
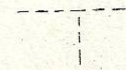
INCORRECT



(A) BEGIN WITH A DASH NOT WITH A SPACE



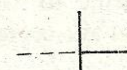
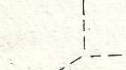
(B) DASHES INTERSECT WITH NO GAP BETWEEN



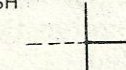
(C) DASHES INTERSECT WITH NO GAP BETWEEN



(D) THREE DASHES MEET AT THE CORNER



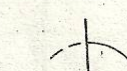
(E) AS A CONTINUATION OF A VISIBLE LINE, DASH LINE BEGINS WITH A SPACE



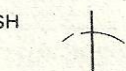
(F) INVISIBLE ARCS BEGIN WITH A DASH



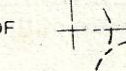
(G) VERY SMALL ARCS MAY BE MADE SOLID



(H) AS A CONTINUATION OF A VISIBLE ARC, DASH ARC BEGINS WITH A SPACE



(I) TWO DASH ARCS MEET AT THE POINT OF TANGENCY



The following examples will assist in understanding how objects are drawn in orthographic projections.

Example one

arrangement drawings. The drawing includes overall dimensions, installation details, overall weight/mass, weights of sub systems, and service supply details.

The general arrangement drawing includes references to the design documents. The drawing often also identifies relevant internal and external contract numbers. An example of a typical general arrangement drawing is a roller conveyor system comprising a number of conveyors with independent drives and guards.

The drawn separate assemblies and parts will be identified with leader lines to balloons which include the arrangement reference number linking to the list of arrangement drawings. Specifically installation drawing provides instructions of the installation requirements and often are based on the equipment location drawings and the piping arrangement drawings. The drawings often provide information on the sequence of equipment and piping installation by provision of a number of drawings of the same area at different stages of the installation.

Arrangement Drawing

Arrangement drawings represent self contained units used to make up the system drawn on the general arrangement drawing. Examples of arrangement drawings include drawings of assembled conveyers, drive systems, elevating units etc. The drawing should show in, at least three orthographic views, clear details to show all of the components used to make up the equipment items and how the component parts are located and fastened together.

Arrangement drawings include a table (parts list) identifying assemblies, fabrication drawings, detail drawings and proprietary items used to make up the equipment. Arrangement drawings include overall dimension, the weight/mass of the equipment drawn, the lifting points. All information needed to construct, test, lift, transport, and install the equipment should be provided in notes or as referenced documents.

The arrangement drawing may be a standard internal drawing which is repeatedly called up on different system general arrangement drawings. The drawn separate assemblies and parts will be identified with leader lines to balloons which include the item reference number linking to the parts list.

Detail Drawings

All individual items required to produce mechanical equipment need to be described in some detail to ensure that they are manufactured in accordance with the designers requirements. Proprietary items are selected from technical data sheets obtained from manufacturer /supplier. Items manufactured specifically for the application need to be made to detail drawings which include the geometry, material, heat treatment requirements, surface texture, size tolerances, geometric tolerances etc.

The detail drawing should include all of the necessary information to enable procurement, manufacture and should identify all of the relevant codes and standards. The item weight/mass should also be included for reference.

Depending on the level of detail, a detail drawing can comprise one drawing on a sheet or a number of separate drawings on one sheet. It is sometimes possible to combine the detail drawings onto the assembly drawing. The detail drawing must cross reference, both ways, to the parent assembly or arrangement drawing.

Fabrication Drawings

The fabrication drawing is a specific type of detail drawing. Some fabrication drawings are virtually assembly drawing e.g. when a number of items are assembled together as a fabrication.

The fabrication drawing generally includes a material parts list identifying all of the materials used to build up the fabrication. All weld details are included using the standard symbolic representation of welds as shown in BS EN 22553. All of the materials should be identified in accordance with the relevant standards and codes.

The fabrication drawing should clearly describe in notes or in referenced documents the heat treatment and stress relieving requirements prior to, during and following the completion of the fabrication processes. The dimensions and relevant linear and geometric tolerances should be indicated.

A fabrication drawing sometimes only includes the fabrication details, the final machining details are then shown on a separate drawing. It is equally acceptable to show all manufacturing information on one drawing.

The items used to make up the fabrication will be identified with leader lines to balloons which include the item reference number linking to the parts list. The listed items on a fabrication drawing do not identify items which can be disassembled, as on assembly and arrangement drawings. The numbering system should reflect this difference. Methods of numbering items on fabrication drawings include using lower case alphabet letters e.g a,b,c or optionally as sub units of the fabrication item number e.g 1/1, 1/2 1/3 ... or 1/a , 1/b, 1/c...

Item Identification

The method of identifying the parts must be clear and unambiguous. The equipment as represented on the general arrangement drawing and the sub-assemblies as shown on the arrangement and assembly drawing should be clearly identified with plant item numbers. The relevant drawing numbers are obtained by reference to the plant items list. Plant items are annotated by leader lines to a double balloon.

Typically a conveyor may have a plant item number e.g.H1040 and be shown on a drawing e.g. drawing number A0 12500.

The detail drawings are sub items of the arrangement drawings and are identified on the arrangement and assembly drawings. Typically an item say a conveyor frame may be identified from the conveyor plant item number e.g. H1040/3 . Optionally it may be identified using the arrangement drawing number e.g. A0 12500 /3. The frame will also have a discrete detail drawing number e.g A2 12503

The fabricated items which are based on sub-parts welded together should be identified as details but the individual sub-parts should be identified in a different way to avoid ambiguity. One option is to number the fabricated sub-parts alphabetically e.g a, b, c ...or as a combination of the fabrication detail number and the part number i.e 3/a , 3/b.... These sub-parts do not need to be identified as separate parts because following fabrication they will not exist as separate parts. If the sub-parts are complicated shapes or machined items and they cannot be described in sufficient detail on the fabrication drawing they should be drawn as separate detail drawings but still identified as sub-parts of the fabrication detail.

Block Diagram

This to show the essentials of an installation in sufficient block outline to indicate the main design features. This will use lines and simple geometric shape

Mass Flow Diagram

This is a simple block diagram which identifies the mass flow of "all" matter through the process. This document is essential in any process design and must be updated as the design progresses

Process Flow Diagram

The Flow diagram shows all the essential parts of the process and items of equipment in sufficient detail to enable the analysis and calculation of the physical characteristics of the system to be undertaken

A Process Flow Diagram is a diagram of a fluid flow system showing the equipment items connected by the major process pipes and containing data on the essential process control circuits or major process requirements. The drawing is not to scale and the the equipment items are represented by symbols. The main equipment items and flow streams should be identified and included in tables which identify process requirements in sufficient detail to enable production of the Piping and Instrument Diagrams (Engineering Line Drawings).

Process Data Sheets

The process data sheets (PSD) are generally A4 documents with a number of sheets. A PSD relates to a single item of equipment and contains the essential process data for initiating the detail design of an item. It includes the overall size, number, approximate geometry and identification of the connections, material of construction and the full range of operating conditions. The Process Data sheet generally includes a simple diagram.

The process design also generally includes for Instrument Process Data Sheets which provides a similar level of detail.

Circuit Diagrams

The circuit diagrams are used for pneumatic and hydraulic systems to show the various control valves, fluid pipeline equipment, and pneumatic /hydraulic line interfaces. The valve systems are specific to the fluid power industry.

The system and piping diagrams

This show detailed connections between the components and items of equipment. The main purpose of these diagrams is to identify pipe routings and initial pipe lengths for producing estimates.

Piping and Instrument Diagrams

The P & I diagram (Engineering Line Diagram) allows the design to progress from the "Process Flow Sheet / System and Piping diagram" to the final system design and installation stage. The P&ID are definitive and comprehensive diagrams showing all of the equipment, piping, valves and instrumentation. All items to be identified using a standard numbering systems. This normally entails having a unique plant item number for each item of equipment, valve, instrument and line. Ideally the line number should include a size, material and fluid contents identifier to enable the anyone reading the drawing to obtain this information without having to refer to other documents.

An Engineering line should include.

- Mechanical equipment with names and numbers
- Instruments with identification and necessary interfaces with control loops
- Interconnecting piping, sizes and identification
- Valves with identifications
- Vents, drains, special fittings, sampling lines, reducers and increasers
- Flow directions
- Interface tags to other ELD's
- Control interfaces, inputs and outputs.
- Main plant interlocks.
- Identification of symbols used
- Reference list including identification of relevant PFD's

The symbols used to be in accordance with identified standards and should clearly indicate the type of component, the method of connection (screwed, flanged etc) and the status (Valves - Normally Closed, Normally Open). The method of operating equipment items should be clearly identified (electric motor, pneumatic actuator). It is not generally necessary to identify services and electrical supplies to the operators.

Mechanical Flow Diagram

This type of diagram is required to clearly show a system involving moving mechanical components which need to be interfaced and interlocked. All sequenced movements have to be identified including the method of initiating and terminating the movements. A mechanical flow diagram is generally a block diagram.

Equipment Location Diagram

The design process includes for provision of equipment location diagrams which identify is some detail the locations of all the equipment items used in the process. These diagrams also provide local floor loadings, static and dynamic and proposed fastening arrangements. This information is required to enable the structural design of the building to be progressed.

Piping Arrangement Drawings

These drawings are generally completed using 3 D Cad systems. They are either Isometric drawings or orthographic or a combination of the two. These drawings are completed to identify, in detail the pipe runs and to show the interfaces with the equipment and the various other services and to identify the access allowances.

Piping Isometrics

Each pipe line is detailed by at least one isometric. This need not be to scale but should show all material, size, weld, and fitting information. The isometric includes all pipe lengths and all necessary datums identifying the pipe location in the region where it is to be installed. The isometric is the pipe manufacturing drawing and includes a part list identifying all component parts of the pipe being detailed.

Assembly Drawings

The assembly /sub-assembly drawings are drawings of discrete sub-systems showing in some detail how the component items fit together. Typical assembly drawings include gearbox drawings, roller

drawings, guard system drawings.

The assembly drawing will generally include at least three orthographic views with sections as needed to clearly show all of the details and their relative positions. Overall and detail dimensions will be shown. The weight/mass of the assembly/sub-assembly will be noted. The drawing will include a parts list identifying all of the component details with quantities and materials and supply details. The assembly drawing will include a list of reference drawings and notes identifying the relevant codes and specifications and testing requirements.

The drawn separate items will be identified with leader lines to balloons which include the item reference number linking to the parts list.

ITEM REFERENCES

The items reference should be assigned in sequential order to each component/ part (see Assembled drawing Fig. 2.23).

Identical parts shown in the assembly should have the item reference (see Table 2.3)

ITEM	QUANTITY	DESCRIPTION	REFERENCE
1	1	Base	
2	1	Bottom housing	
3	1	Top Housing	
4	1	Bearing	
5	1	Filling Plug	
6	2	T-Bolt	
7	2	Hex nut	
8	4	Washer	
9	2	T-Bolt	
10	2	Castle nut	
11	2	Split pin	
12	1	Drain plug	

Item reference should be composed of Hindu-Arabic numerals only, and should be bold and comparatively of bigger height than other information recorded in drawing.

Sizes and layout of drawing sheets

The item references should be placed outside the drawing and should be connected by leader to the associated item (see Fig. 2.24 A, B and C). The item list can be recorded in the drawing or on the separate document.

Drawing

Fig. 2.23: Assembled Drawing

2.9 LAYOUT OF TITLE BLOCK FOR SCHOOLS AND COLLEGES

layout for the schools and colleges have to be designed especially for the convenience of the students (see Fig 2.25).

Students are advised to follow this practice throughout their course.

ORTHORGRAPHIC PROJECTION

Introduction

This chapter presents some essential projection fundamentals employed in engineering graphics. The basic projection theory topics introduced herein after are: perspective and parallel projection. The basic concepts are utilized by engineers, designers, and drafting personnel to prepare and interpret diagrams, production drawing and construction plans.

Additional graphic fundamentals like, sketching, axonometric projections, spatial visualization, auxiliary views, descriptive geometry, drafting standards are discussed in subsequent chapters.

PROJECTION THEORY

One of the most valuable means of communication in engineering graphics is projection theory. Projection theory is the body of the knowledge that comprises the set of rules and principles devised to analyse, draw and display spatial geometry on two-dimensional surfaces (sheet of paper). The main aim of projection is to describe the shape of the objects.

PROJECTION

Projection is defined as a visual, produced by mapping (from aggregation of points) a geometric representation of an object onto a viewing surface or plane (see Fig. 8.1).

Geometric projection involves establishing the relationship among a series of visual variables (e.g, line of sight, point of sight, picture planes, vanishing-point, horizon, etc). And spatial object to be represented or projected.

The following discussion will classify the types of projections available and their function. Some important projection theory terms will be defined.

CLASSIFICATION

Projection can be classified into two broad areas:

- (i) Perspective projections and
- (ii) Parallel projections.

PERSPECTIVE PROJECTION

A Perspective projection represents objects as perceived by the human eye (i.e., binocular vision). It is a pictorial drawing generated by the intersection of the observer's visual rays (lines of sight) converging on a picture plane. The observer's eye (station point or point of sight) is located at a finite (measurable) distance from the picture plane (see Fig. 8.1).

Perspective projections (also known as central projections) are classified based on the number of vanishing points used to draw them: one point, two point and three point.

This chapter will be dealt in detail in the foregoing chapter of perspective projection.

PARALLEL PROJECTION

Parallel projection portrays an object as a geometric abstraction of human visual reality. In Parallel projection, the observer is assumed to be at an infinite distance from the object (i.e., as opposed to finite distance-like perspectives). Since observer and object are so far apart, the visual rays are parallel to each other. These rays or lines-of-sight are used to project the object on to a standard drawing plane.

Parallel projections are unclassified by the angle formed between projection planes and the observer's line-of-sight. The forthcoming discussion covers the prevailing features of parallel projections. To guide the discussion the following summary is offered.

SUMMARY OF PARALLEL PROJECTION

1. **Orthographic-visual rays are at right-angle and plane**
 - (a) Multiview-display of standard 2-D views
 - (i) projective-object between observer and plane
 - (ii) Third-angle projection-plane between object and observer.
 - (b) Axonometric-visual rays are not at right angles with the projection plane
2. **Oblique-visual rays are not at right angle with the projection plane:**
 - (i) cavalier-45° with plane, receding lines in true length
 - (ii) Cabinet – 63°, 26' with plane, receding lines drawn half length
 - (iii) General oblique-any other angles with planes of projection
3. **Standard and special purpose:**
 - (a) sectional- projection plane cuts through solid geometry:
 - (i) Full section-plane passes fully through an object
 - (ii) Half section passes through and objects
 - (iii) Offset section passes halfway through
 - (iv) Other (rib, revolved, removed, broken out)
 - (b) Auxiliary-not parallel to the principal orthographic planes:
 - (i) Primary-perpendicular to one principal plane
 - (ii) Secondary-inclined to all principal planes

Let us now explain these concepts in detail chapterwise

ORTHOGRAPHIC PROJECTION

A three-dimensional object can be represented by projecting its views onto imaginary orthogonal planes as illustrated in Fig. 8.2.

The Greek word “orthogonious”-orthogonal means “composed of right angle”. Consequently, the projection method that utilizes parallel lines- of sight (projectors) at right angles (90°) to an imaginary plane to represent 3-D objects is named orthographic projection. This standard method allows the engineer to describe a specific object, in every detail on a 2-D plane.

MULTIVIEW DRAWING

This text is, however, concerned with class of orthographic projections called multiview drawing, in which the main purpose is to obtain views of an object on which true measurements could be made. The front face is oriented parallel to the projection plane so that the established orthographic view shows the width and height of the object. It will be noted however, that the depth dimension is shown in this front view. This view in itself cannot fully describe the object. An additional view on a projection plane perpendicular to the first is needed to give the depth.

Drawing

Illustrates the projections of the object on two perpendicular planes. It will be noted that the width dimension appears in both the view while the height appears in front view only. The depth appears only in top view.

Since it would be inconvenient to carry a drawing on planes at right angles to each other, therefore, a conventional means of showing several views on a single plane has been deployed. Suggests the manner in which the horizontal projection plane (H.P is rotated into the same as the front view (the vertical plane, V.P)

(a) Shows these views in the resulting conventional arrangement as they would be drawn on paper. Two such directly related views of an object are called adjacent views.

PLANES REFERENCE

Planes, used to obtain views, are called as principal reference planes. They are two and named as vertical reference plane and horizontal reference plane. See Fig. 8.5. They are set at right angles. Horizontal reference plane is also known as horizontal plane and denoted in short by H.R.P. and H.P. vertical reference plane is also known as frontal reference plane and is denoted by F.R.P or V.P

Orthographic projections of an object taken on H.R.P/H.P is generally know as top view of plan, and projection than F.R.P/V.P is called as front view or elevation. Any reference plane other than FRP/VP and HP is called on auxiliary plane. If front view and top view are not sufficient to describe the object fully, we have to obtain a third view on auxiliary plane. This plane is generally perpendicular to both the principal planes, and is called the profile reference plane. The projection obtained on this auxiliary plane is called a profile view or the side view. This is also known as end view.

Drawing.

SYSTEM OF ORTHOGRAPHIC PROJECTIONS

It is already stated that two principal planes are used in orthographic projection, one horizontal and other vertical, intersecting and dividing space up into four quadrants, or view respectively. Te until they coincide, the conventional is that the show views on plane surface, the H.P. and V. P are opened out, or rebated about the 1st quadrant must be opened.

For projections I 2nd, 3rd and 4th quadrants, the planes assumed to be transparent, the projections are drawn looking on the rabbeted 1st quadrant.

On opening the planes orthographic projection 2nd and 4th quadrant overlap each others and it becomes very difficult to read the drawing. Therefore, only 1st and 3rd quadrant are suitable for orthographic projections.

HOW IS A DRAWING MADE IN FIRST ANGLE PROJECTION

The object of engineering drawing is to define precisely a solid three dimensional on flat paper. In first angle (fig. 8.7) three planes of projection have been erected, and a solid object has been suspended between the planes.

The planes are actually drawing surfaces and the views drawn on the two vertical surfaces are usually referred to as front views or elevations (projection on V.P. is the front view and on top the p.p right side view). While the view on the horizontal surface is referred to as the top plan:

Diagram

Shows the object being viewed from direction A which is at right angle to the vertical principal plane the view seen is drawn on the plane beyond the object. In view, the following information concerning the object has been given

- i. The length of the, L.
- ii. The height of the object. H

Shows the object being viewed from direction B at right angles to end vertical plane (profile plane, P. P.), and the view seen is drawn on the plane beyond view A, by means of horizontal lines known as projection line. From this view further information concerning the object is given.

- (iii) The width of the object, W
- (iv) The height of the object, H.

Diagram

8.10 show the object being viewed from direction C, which is directly down, at right angles to the horizontal principal plane the view seen is drawn on the plane beyond the object.

- (v) The length of the object, L
- (vi) The width of the object, W.

The three views, now drawn completely define the shape, but drawing on three planes at right angles in the shape of the corner is inconvenient. Fig.11, the three connected planes, opened on the plane drawing as showing in Fig 8.12, result in three quarters of the flat sheet with three separated views, each one related to the other.

In laying out the planes flat, the projection lines transferring the sizes and points between the related views are seen to be horizontal or vertical parallel lines, but those between view B and the plan view C become broken, and the disjointed parts are at 90 to each other.

With dimensions added.

Show the three views in first angle projection as they would be drawn on a drawing sheet

HOW IS A DRAWING MADE IN THIRD ANGLE PROJECTOR?

The views drawn by this method differ from those in first angle projection only in the position of view relative to each other.

In fig.8.14, three transparent planes of projections have been erected (hinged) in third angle, and a solid object has been suspended between the planes. The object is viewed through the transparent planes, and the view is drawn. As in first angle projection, the views on the vertical surfaces are referred to as elevations/front elevation side elevation, and the view on the horizontal plane is referred to as the plan top view:

Diagram

Shows the object being viewed from direction A at right angles and through the vertical transparent plane. The view seen is drawn on this transparent plane. Note that the plane on which the view is drawn is adjacent to the object, and not beyond the object as in first-angle projection.

In this view, the following information concerning the object has been given.

1. The length of object, L
2. The height of the object, H.

As this single view A does not completely define the object, other views must be drawn.

Define the object, other views must be drawn.

To assist in drawing this view, the height (H) has been transferred from A by means of horizontal projection line. This view supplies the following information:

3. The width of the object, W
4. The height of the object, H.

Diagram

Fig. 8.17 shows the object viewed through the principal plane from above in the direction C; the view seen is drawn on the transparent plane.

To assist in drawing and relating the third view A. From view B, the width (W) has been projected round to the horizontal plane.

5. Length of the object, L
6. Width of the object, W

The three views, now drawn completely, define the shape and the planes are now opened out on the plane of drawing, resulting in three quarters of a flat sheet with three separated views related to each other (fig. 8.18)

When drawing the views on a drawing sheet, the information can be transferred between view B and view C, by means of 45 degree deflator or miter line (Fig. 8.19) as in first-angle projecting.

Shows three views in third-angle projection as they would be drawn on a drawing sheet. Not that the view be appear reversed. In third-angle projection, this is because views are positioned adjacent to the surfaces being illustrated, whereas in first-angle projection, the views of the surface illustrated are position adjacent to the surfaces being illustrated, whereas in first-angle projection, nterviewed.

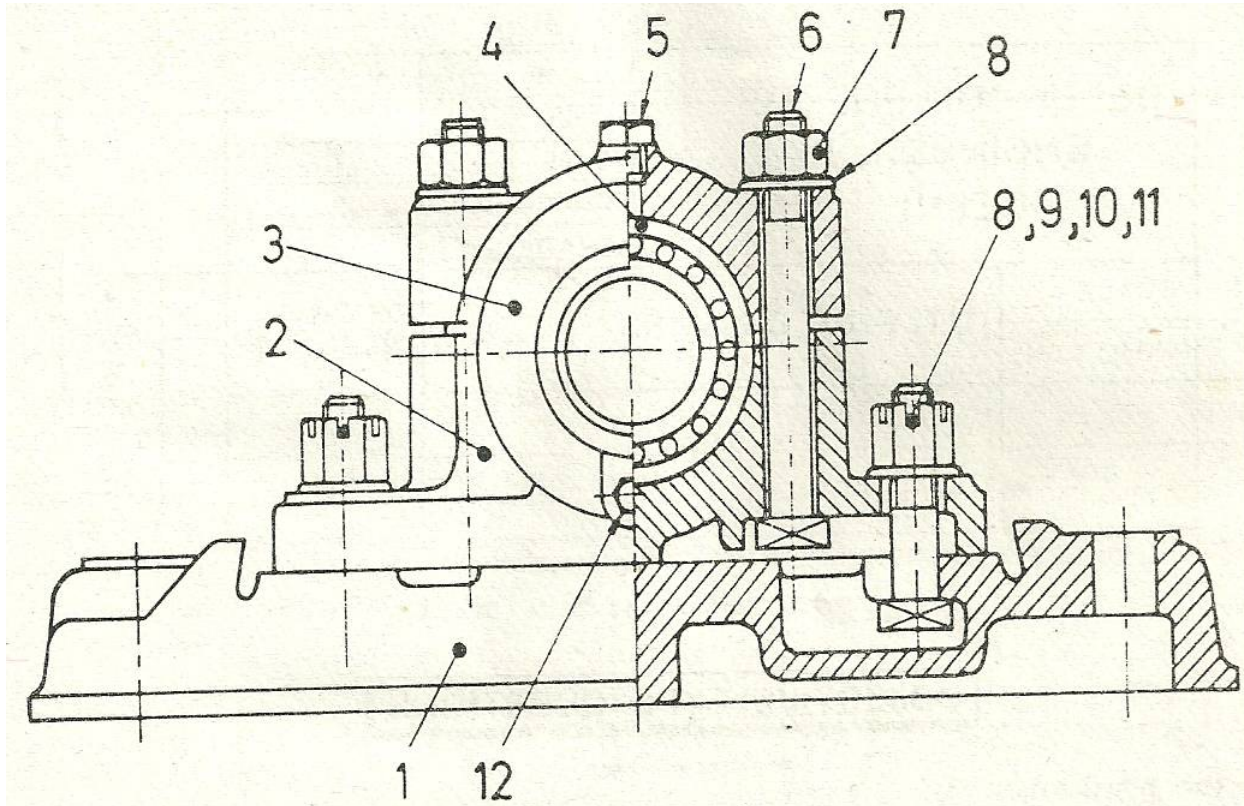
To show the object completely, the views in the directions a, b, c, d,e and f, may be necessary, in order of priority (see Fig 8. 21) and Table 8. 1.

BEARINGS

There are three principal classes of bearing, they are: journal bearings, thrust bearings and footstep or pivot bearings.

Journal Bearings

In journal bearings, the supporting pressure of the bearing is at right angles to the shaft axis



INTRODUCTION TO 2D AUTOCAD BASICS.

Learning Objectives:

- ◆ Create and Save AutoCAD® Drawing Files
- ◆ Use the AutoCAD® Visual Reference Commands
- ◆ Draw, Using the LINE and CIRCLE Commands
- ◆ Use the ERASE Command
- ◆ Define Positions Using the Basic Entry Methods
- ◆ Use the AutoCAD® Pan Real time Option and Plotting a drawing.

Introduction

Learning to use a CAD system is similar to learning a new language. It is necessary to begin with the basic alphabet and learn how to use it correctly and effectively through practice. This will require learning some new concepts and skills as well as learning a different vocabulary. All CAD systems create designs using basic geometric entities.

Many of the constructions used in technical designs are based upon two-dimensional planar geometry. The method and number of operations that are required to accomplish the constructions are different from one system to another. In order to become effective in using a CAD system, we must learn to create geometric entities quickly and accurately. In learning to use a CAD system, lines and circles are the first two, and perhaps the most important two, geometric entities that one should master the skills of creating and modifying. Straight lines and circles are used in almost all technical designs. In examining the different types of planar geometric entities, the importance of lines and circles becomes obvious. Triangles and polygons are planar figures bounded by straight lines. Ellipses and splines can be constructed by connecting arcs with different radii. As one gains some experience in creating lines and circles, similar procedures can be applied to create other geometric entities. In this chapter, the different ways of creating lines and circles in AutoCAD.

Starting Up AutoCAD® 2007

1. Select the AutoCAD 2007 option on the Program menu or select the AutoCAD 2007 icon on the Desktop. Once the program is loaded into memory, the AutoCAD 2007 drawing screen will appear on the screen.
2. Choose AutoCAD Classic as the initial workspace.
3. Select Don't Show me this again and click OK to continue.

Note that AutoCAD® automatically assigns generic name, Drawing X, as new drawings are created. In our example, AutoCAD® opened the graphics window using the default system units and assigned the drawing name Drawing1.

Drawing Units Setup

Every object we construct in a CAD system is measured in units. We should determine the value of the units within the CAD system before creating the first geometric entities.

1. In the pull-down menus, select:
[Format] _ [Units]
2. In the Drawing Units dialog box, set the Length Type to Decimal. This will set the measurement to the default English units, inches.
3. Set the Precision to two digits after the decimal point
4. Pick OK to exit the Drawing Units dialog box.

Drawing Area Setup

Next, we will set up the Drawing Limits; setting the Drawing Limits controls the extents of the display of the grid. It also serves as a visual reference that marks the working area. It can also be

used to prevent construction outside the grid limits and as a plot option that defines an area to be plotted/printed. Note that this setting does not limit the region for geometry construction.

1. In the pull-down menus, select:

[Format] _ [Drawing Limits]

2. In the command prompt area, near the bottom of the AutoCAD® drawing screen, the message “Reset Model Space Limits: Specify lower left corner or [On/Off] <0.00,0.00>:” is displayed. Press the ENTER key once to accept the default coordinates <0.00,0.00>.

3. In the command prompt area, the message “Specify upper right corner <0.00,0.00>:” is displayed. Press the ENTER key once to accept the default coordinates <12.00,9.00>.

4. On your own, move the graphic cursor near the upper-right corner inside the drawing area and note that the drawing area is unchanged. (The Drawing Limits command is used to set the drawing area; but the display will not be adjusted until a display command is used.)

5. In the pull-down menus, select:[View] _ [Zoom] _ [All]_ The Zoom All command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.

6. Move the graphic cursor near the upper right corner inside the drawing area and note that the display area is updated.

Using the Line Command

1. Click on the Info Palette option in the Help pull-down menu to activate the Quick Help option.

2. Move the graphics cursor to the first icon in the Draw toolbar. This icon is the Line icon. A help-tip box appears next to the cursor and a brief description of the icon is displayed at the bottom of the AutoCAD® drawing screen: “Creates Straight line segments: LINE.”

3. Select the icon by clicking once with the left-mouse-button, which will activate the Line command. _ Notice a brief explanation of the selected command is displayed in the Info Palette window. It is highly recommended that you read the explanations to gain some insights on the general procedure of using AutoCAD®.

4. In the Info Palette window, click To draw lines to open the AutoCAD® Help window and get a more detailed explanation on the procedure. The general procedure to create a line in AutoCAD® is displayed in the Info Palette window.

5. Click on Close button located at the upper left corner of the Info Palette window as shown.

6. In the command prompt area, near the bottom of the AutoCAD® drawing screen, the message “_line Specify first point:” is displayed. AutoCAD® expects us to identify the starting location of a straight line. Move the graphics cursor inside the graphics window and watch the display of the coordinates of the graphics cursor at the bottom of the AutoCAD® drawing screen. The three numbers represent the location of the cursor in the X, Y, and Z directions. We can treat the graphics window as if it was a piece of paper and we are using the graphics cursor as if it were a pencil with which to draw.

Visual Reference

The method we just used to create the freehand sketch is known as the interactive method, where we use the cursor to specify locations on the screen. This method is perhaps the fastest way to specify locations on the screen. However, it is rather difficult to try to create a line of a specific length by watching the displayed coordinates. It would be helpful to know what one inch or one meter looks like on the screen while we are creating entities.

AutoCAD® 2007 provides us with many tools to aid the construction of our designs. We will use the GRID and SNAP options to get a visual reference as to the size of objects and learn to restrict the movement of the cursor to a set increment on the screen.

The Status Bar area is located at the bottom of the AutoCAD® drawing screen. The words SNAP, GRID, ORTHO, POLAR, OSNAP, OTRACK, DUCS, LWT and MODEL appearing to the right of the coordinates are buttons that we can left-click to turn these special options ON and OFF. When the corresponding button is highlighted, the specific option is turned ON. These buttons act as toggle switches; each click of the button will toggle the option ON or OFF. Using the buttons is a quick and easy way to make changes to these drawing aid options. We can toggle the options on and off in the middle of another command.

GRID On

1. Left-click the GRID button in the Status Bar to turn On the GRID option. (Notice in the command prompt area, the message “<Grid on>” is also displayed.)
2. Move the cursor inside the graphics window, and estimate the distance in between the grid points by watching the coordinates display at the bottom of the screen.

Option Buttons

The GRID option creates a pattern of dots that extends over an area on the screen.

Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distance between them. The grid is not displayed in the plotted drawing. The default grid spacing, which means the distance in between two dots on the screen, is 0.5 inches. We can see that the sketched horizontal line in the sketch is about 4.0 inches long.

DYN Off

1. Left-click the DYN button in the Status Bar to turn Off the Dynamic Input option.

The DYN button allows the quick toggle of the Dynamic Input option.

SNAP On

1. Left-click the SNAP button in the Status Bar to turn On the SNAP option.
2. Move the cursor inside the graphics window, and move the cursor diagonally on the screen. Observe the movement of the cursor and watch the coordinates display at the bottom of the screen. The SNAP option controls an invisible rectangular grid that restricts cursor movement to specified intervals. When SNAP mode is on, the screen cursor and all input coordinates are snapped to the nearest point on the grid. The default snap interval is 0.5 inches, and aligned to the grid points on the screen.
3. Click on the Line icon in the Draw toolbar. In the command prompt area, the message “_line Specify first point:” is displayed.
4. On your own, create another sketch of the five-point star with the GRID and SNAP options switched ON.
5. Use the right-mouse-button and select Enter in the popup menu to end the Line command if you have not done so.

Using the ERASER Command

One of the advantages of using a CAD system is the ability to remove entities without leaving any marks. We will erase two of the lines using the Erase command.

1. Pick Erase in the Modify toolbar. (The icon is the first icon in the Modify toolbar. The icon is a picture of an eraser at the end of a pencil.) The message “Select objects” is displayed in the command prompt area and AutoCAD® awaits us to select the objects to erase.

2. Left-click the SNAP button on the Status Bar to turn off the SNAP option so that we can more easily move the cursor on top of objects. We can toggle the Status Bar options ON or OFF in the middle of another command.
3. Select any two lines on the screen; the selected lines are displayed as dashed lines as shown in the figure below.
4. Right-mouse-click once to accept the selections. The selected two lines are erased.

Repeating the Last Command

1. Inside the graphics window, click once with the right mouse-button to bring up the popup option menu.
2. Pick Repeat Erase, with the left-mouse-button, in the popup menu to repeat the last command. Notice the other options available in the popup menu.
3. Move the cursor to a location that is above and toward the left side of the entities on the screen. Left-mouse-click once to start a corner of a rubber-band window.
4. Move the cursor toward the right and below the entities, and then left-mouse click to enclose all the entities inside the selection window. Notice all entities that are inside the window are selected.
5. Inside the graphics window, right-mouse-click once to proceed with erasing the selected entities.

On your own, create a free-hand sketch of your choice using the Line command. Experiment with using the different commands we have discussed so far. Also try switching the GRID and SNAP options ON and OFF in the middle of a command.

Designs and drawings created in a CAD system are usually defined and stored using sets of points in what is called world space. In most CAD systems, the world space is defined using a three-dimensional Cartesian coordinate system. Three mutually perpendicular axes, usually referred to as the X-, Y-, and Z-axes, define this system. The intersection of the three coordinate axes forms a point called the origin. Any point in world space can then be defined as the distance from the origin in the X-, Y- and Z- directions. In most CAD systems, the directions of the arrows shown on the axes identify the positive sides of the coordinates.

A CAD file, which is the electronic version of the design, contains data that describe the entities created in the CAD system. Information such as the coordinate values in world space for all endpoints, center points, etc., along with the descriptions of the types of entities are all stored in the file. Knowing that AutoCAD stores designs by keeping coordinate data helps us understand the inputs required to create entities.

The icon near the bottom left corner of the default AutoCAD® graphics window shows the positive X-direction and positive Y-direction of the coordinate system that is active. In AutoCAD®, the coordinate system that is used to create entities is called the user coordinate system (UCS). By default, the user coordinate system is aligned to the world coordinate system (WCS). The world coordinate system is a coordinate system used by AutoCAD® as the basis for defining all objects and other coordinate systems defined by the users. We can think of the origin of the world coordinate system as a fixed point being used as a reference for all measurements.

The default orientation of the Z-axis can be considered as positive values in front of the monitor and negative values inside the monitor.

Changing to the 2D UCS Icon Display In AutoCAD ®2007.

The UCS icon is displayed in various ways to help us visualize the orientation of the drawing plane.

1. In the pull-down menus, select:

[View] _ [Display] _

[UCS Icon] _ [Properties]

2. In the UCS icon style section, switch to the 2D option as shown.

3. Click OK to accept the settings.

Note the W symbol in the UCS icon indicates the UCS is aligned to the world coordinate system.

Cartesian and Polar Coordinate Systems

In a two-dimensional space, a point can be represented using different coordinate systems. The point can be located, using a Cartesian coordinate system, as X and Y units away from the origin. The same point can also be located using the polar coordinate system, as r and θ units away from the origin.

For planar geometry, the polar coordinate system is very useful for certain applications. In the polar coordinate system, points are defined in terms of a radial distance, r, from the origin and an angle θ between the direction of r and the positive X axis. The default system for measuring angles in AutoCAD ® 2007 defines positive angular values as counter-clockwise from the positive X-axis.

Absolute and Relative Coordinates AutoCAD ® 2007 also allows us to use absolute and relative coordinates to quickly construct objects.

Absolute coordinate values are measured from the current coordinate system's origin point. Relative coordinate values are specified in relation to previous coordinates.

Note that the coordinate display area can also be used as a toggle switch; each left-mouse-click will toggle the coordinate display on or off.

In AutoCAD® 2007, the absolute coordinates and the relative coordinates can be used in conjunction with the Cartesian and polar coordinate systems. By default, AutoCAD® expects us to enter values in absolute Cartesian coordinates, distances measured from the current coordinate system's origin point. We can switch to using the relative coordinates by using the @ symbol. The @ symbol is used as the relative coordinates specifier, which means that we can specify the position of a point in relation to the previous point.

Defining Positions

In AutoCAD®, there are five methods for specifying the locations of points when we create planar geometric entities.

Interactive method: Use the cursor to select on the screen.

Absolute coordinates (Format: X,Y): Type the X and Y coordinates to locate the point on the current coordinate system relative to the origin.

Relative rectangular coordinates (Format: @X,Y): Type the X and Y coordinates relative to the last point.

Relative polar coordinates (Format: @Distance<angle): Type a distance and angle relative to the last point.

Direct Distance entry technique: Specify a second point by first moving the cursor to indicate direction and then entering a distance.

The Guide Plate

We will next create a mechanical design using the different coordinate entry methods.

- Use the Erase command and erase all entities on the screen before proceeding to the next section.
- The rule for creating CAD designs and drawings is that they should be created at full size using real-world units. The CAD database contains all the definitions of the geometric entities and the design is considered as a virtual, full-sized object. Only when a printer or plotter transfers the CAD design to paper is the design scaled to fit on a sheet. The tedious task of determining a scale factor so that the design will fit on a sheet of paper is taken care of by the CAD system. This allows the

designers and CAD operators to concentrate their attention on the more important issues – the design.

1. Select the Line command icon in the Draw toolbar. In the command prompt area, near the bottom of the AutoCAD® graphics window, the message “_line Specify first point:” is displayed. AutoCAD® expects us to identify the starting location of a straight line.

2. In the command prompt area, we will locate the starting point of our design at the origin of the world coordinate system. Command: _line Specify first point: 0,0 [ENTER]

(Type 0,0 in the command prompt area and press the [ENTER] key once.)

3. We will create a horizontal line by entering the absolute coordinates of the second point.

Specify next point or [Undo]: 5.5,0 [ENTER]

• The line we created is aligned to the bottom edge of the drawing window. Let us adjust the view of the line by using the Pan Real time command.

4. Click on the Pan Real time icon in the Standard toolbar area. The icon is the picture of a hand with four arrows. The Pan command enables us to move the view to a different position. This function acts as if you are using a video camera.

5. Move the cursor, which appears as a hand inside the graphics window, near the center of the drawing window, then push down the left-mouse-button and drag the display toward the right and top side until we can see the sketched line. (Notice the scroll bars can also be used to adjust viewing of the display.)

6. Press the [Esc] key to exit the Pan command. Notice that AutoCAD® goes back to the Line command.

7. We will create a vertical line by using the relative rectangular coordinates entry method, relative to the last point we specified:

Specify next point or [Close/Undo]: @0,2.5 [ENTER]

8. We can mix any of the entry methods in positioning the locations of the endpoints. Move the cursor to the Status Bar area, and turn ON the GRID and SNAP options.

9. Create the next line by picking the location, world coordinates (8,2.5), on the screen.

10. We will next use the relative polar coordinates entry method, relative to the last point we specified:

Specify next point or [Close/Undo]: @3<90 [ENTER]

(Distance is 3 inches with an angle of 90 degrees.)

SNAP and GRID On

11. Using the relative rectangular coordinates entry method to create the next line, we can imagine a reference coordinate system aligned at the previous point. Coordinates are measured along the two reference axes. Specify next point or [Close/Undo]: @-1.5,1 [ENTER]

(-1.5 and 1 inches are measured relative to the reference point.)

12. Move the cursor directly to the left of the last point and use the direct distance entry technique by entering 6.5 [ENTER].

13. For the last segment of the sketch, we can use the Close option to connect back to the starting point. Inside the graphics window, right-mouse-click and a popup menu appears on the screen.

14. Select Close with the left-mouse-button to connect back to the starting point and end the Line command. Reference Coordinate System aligned at the previous point

Creating Circles

The menus and toolbars in AutoCAD® 2007 are designed to allow the CAD operators to quickly activate the desired commands. Besides using the Draw toolbar, we can also select the different Draw commands through the pull-down menus.

1. In the pull-down menus, select:

[Draw] _ [Circle] _ [Center, Diameter]

Notice the different options available under the circle submenu:

- Center Point: Draws a circle based on a center point and a diameter or a radius.
- 2 Points: Draws a circle based on two endpoints of the diameter.
- 3 Points: Draws a circle based on three points on the circumference.
- TTR–Tangent, Tangent, Radius: Draws a circle with a specified radius tangent to two objects.
- TTT–Tangent, Tangent, Tangent: Draws a circle tangent to three objects.

AutoCAD® Fundamentals 1-23

2. In the command prompt area, the message “Specify center point for circle or [3P/2P/Ttr (tan tan radius):” is displayed. AutoCAD expects us to identify the location of a point or enter an option. We can use any of the four coordinate entry methods to identify the desired location. We will enter the world coordinates (2.5,3) as the center point for the first circle.

Specify center point for circle or [3P/2P/Ttr (tan tan radius): 2.5,3 [ENTER]

3. In the command prompt area, the message “Specify diameter of circle:” is displayed.

Specify diameter of circle: 2.5 [ENTER]

4. Inside the graphics window, right-mouse click to bring up the popup option menu.

5. Pick Repeat Center, Diameter with the left-mouse-button in the popup menu to repeat the last command.

6. Using the relative rectangular coordinates entry method, relative to the center-point coordinates of the first circle, we specify the location as (2.5,2).

Specify center point for circle or [3P/2P/Ttr (tan tan radius): @2.5,2 [ENTER]

1-24 AutoCAD® 2007 Tutorial

7. In the command prompt area, the message “Specify Diameter of circle: <2.50>” is displayed. The default option for the Circle command in AutoCAD® is to specify the radius and the last radius used is also displayed in brackets.

Specify Diameter of circle<2.50>: 1.5 [ENTER]

Saving the CAD Design

1. In the pull-down menus, select:

[File] _ [Save As]

Note the command can also be activated with quick-key combination of [Ctrl]+[Shift]+[S].

2. In the Save Drawing As dialog box, select the folder in which you want to store the CAD file and enter Guide Plate in the File name box.

3. Pick Save in the Save Drawing As dialog box to accept the selections and save the file.

To Exit AutoCAD® 2007

To exit AutoCAD® 2007, select File then choose Exit from the pull-down menu or type QUIT at the command prompt.

Note the command can also be activated with quick-key combination of [Ctrl]+[Q].

Plotting a Drawing

1. Click on File and then plot
2. Select printer type, paper size
3. Click on window and select the drawing

4. Click ok