1.1 INTRODUCTION

Engineering drawing known as the language of engineers is widely used means of communication among the designer, engineers, draftsmen and craftmen in the industry. The translation of ideas into practice without the use of graphic language is really beyond imagination. The word graphics means dealing with the expression of ideas by drawing lines or curves on a surface.

Like other languages, drawing is also a language that can be learned and used like other languages. Basically, engineering drawing is the graphic language of engineers. It is a graphic representation of thinking, planning and language of every technical person who uses to communicate his ideas clearly to other engineers.

Before starting manufacturing or construction work, product to be developed or plan of housing society respectively is firstly drawn on a rough paper. The purpose of this drawing is to define physical shape completely and accurately of particular object by means of lines etc., regarding the object.

In the age of automation, engineering drawing has grown keeps and bounds. Without the fundamental knowledge of technical drawing, a student would not be successful in an industry. Engineering drawing has plenty of applications, especially in modern industries.

Application: Engineering drawing has plenty of applications for machines, automobiles, aeronautics, chemical, marine, electronics, computer and in electrical engineering which gives the correct shape and size along with the dimensional tolerances for understanding of a particular component.

In civil and architecture, engineering drawing is used to draw the plan and elevation of buildings, and structure work.

The application of engineering drawing in electrical, electronics, instrumentation and computer science are many, e.g., to prepare electrical wiring drawing, printed circuit board drawing, installation drawing, process drawing and pictorial drawing.

1.2 DRAWING INSTRUMENTS

Drawing instruments are used to prepare drawings easily and accurately. A neat and clean drawing is prepared by the help of good quality drawing instruments. The following are the drawing instruments commonly used in the industry.

1.2.1 Drawing Board

Drawing board is used for fixing the drawing sheet by means of a tape as shown in Fig. 1.1. It should be made of well seasoned soft wood of yellow pine. This wood should be free of knots and oily grains. Its surface should be perfectly smooth. Drawing boards are available in different sizes in the market. As per IS 1444:1989 the sizes of drawing boards are given in Table 1.1

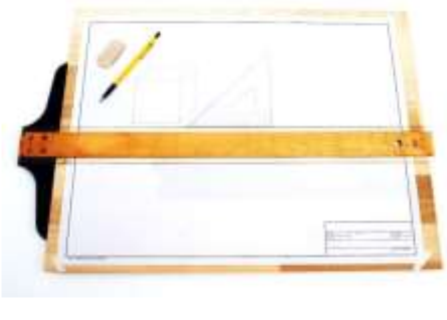


Fig. 1.1

Table 1.1 Drawing Board Size

S.No.	Designation	Size (in mm)
1.	B_0	1500 × 1000 × 25
2.	B_1	1000 × 700 × 25
3.	B_2	700 × 500 × 15
4.	B_3	500 × 350 × 15
5.	B_4	350 × 250 × 15

1.2.2 Drawing Sheet

A variety of drawing sheets are available in the market. Generally drawing sheets are of A_0 size and the other sizes can be obtained by cutting the A_0 size sheet as shown in Fig 1.2.

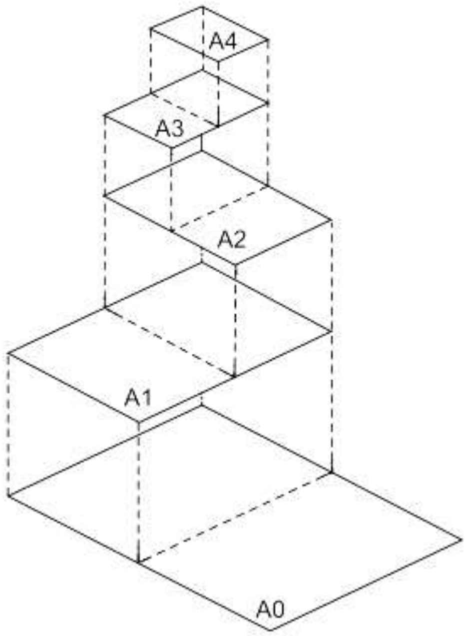


Fig 1.2. Drawing sheet sizes

There are six standard sizes for drawing sheetsspecified by the Indian Standard Institution. The preferred sizes of the sheet as selected from IS 10711:1983 are given in Table 1.2.

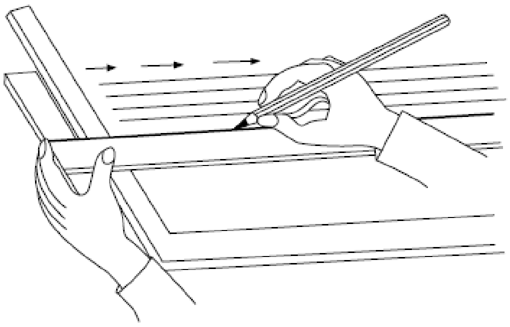
Table 1.2 Drawing Sheet Size

S.No.	Designation	size (in mm)
1.	A_0	841 × 1189
2.	A_1	594 × 841
3.	A_2	420 × 594
4.	A_3	297 × 420
5.	A_4	210 × 297
6.	A_5	148 × 210

For the practice of engineering students, A2 (420 × 594) size drawing sheet is recommended.

1.2.3 Tee-Square

The tee-square should be made of well seasoned hard wood, such as teak as shown in Fig. 1.3.



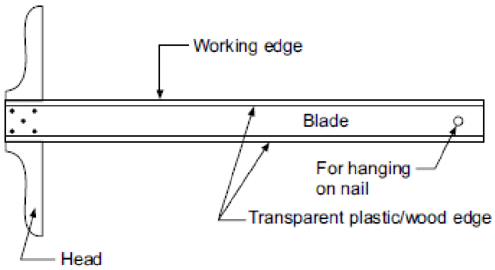


Fig.1.3. Tee-Square

1.2.4 Set-Squares

Set-squares are used in combination for drawing all straight lines except the horizontal lines which are usually drawn with T-square as shown in Fig. 1.4. It is made of transparent sheet of celluloid or plastic material in various sizes. They are available in the shape of triangle with one right angle corner. The set-square of 45º triangle and 30º-60º triangle of 200 mm and 250 mm length are available in the market for ordinary work. Two set-squares used simultaneously along with the tee-square produce lines for making angles of 15º, 30º, 45º, 60º, 75º, 90º and 105º etc. A circle can be divided in 6, 8, 12 and 24 parts by using set-squares. Hatching lines are also drawn by set squares.

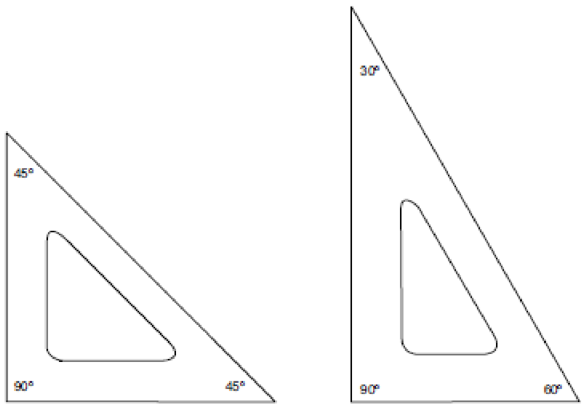


Fig.1.4.

1.3 DRAWING INSTRUMENT BOX

A standard set of drawing instrument box is used by engineering students, containing large compass, bow compass, large divider, bow divider, inking pen and pencil lead etc. as shown in Fig. 1.5.

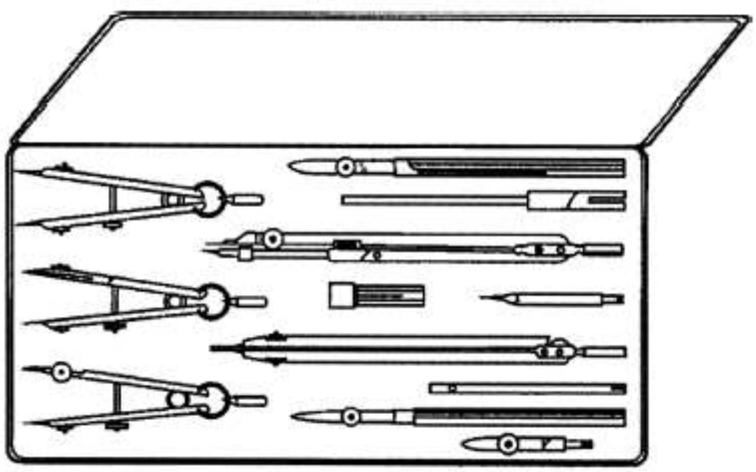


Fig.1.5

Drawing instruments are made of nickel, silver with a silvery lustre on the surface and are corrosion resistant. The other parts like divider point, ruling pen, nibs and spring parts are made of hard steel.

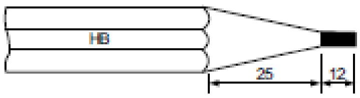
1.4 OTHER MISCELLANEOUS INSTRUMENTS

1.4.1 Drawing Pencils

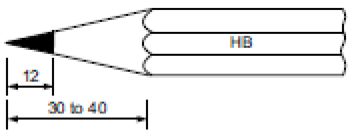
Drawing pencils are used for preparing the drawing of an object. The quality and neatness of the drawing depends upon the quality of the pencil used. Pencils are made of graphite, mixed with varying quantities of clay to produce different degree of hardness covering with ordinary wood. Various grades of the pencil to be used depend upon the type of the lines required as shown in Fig. 1.6.

Engineering students are recommended to use Kohinoor and Apsara brand pencils for better result. There are two ways for using the pencil to prepare the drawing:

1. Chisel edge pencil



2. Conical pointed pencil.



1.4.2 Scotch or Plastic Tape

Plastic tape is used for fixing the drawing sheet on the drawing board, before starting the work. It is made of transparent material and available in rolls of varying sizes and length.

Fig. 1.6

1.4.3 Eraser

A soft colourless and good quality rubber is used for erasing or rubbing unnecessary lines in the drawing. Frequent use of rubber should be avoided and rubbing should be dusted off by dusting cloth as shown in Fig. 1.7.

Fig. 1.7 Eraser

1.4.4 French Curve

French curve is used to draw irregular curves and arcs in the drawing. It is made of transparent plastic material and available in different shapes and sizes. Its edge must be perfectly smooth as shown in Fig. 1.8.

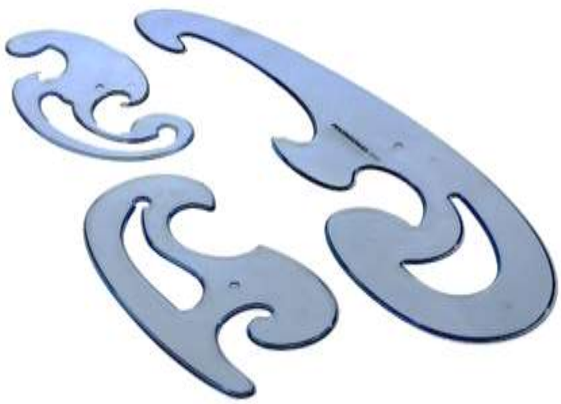
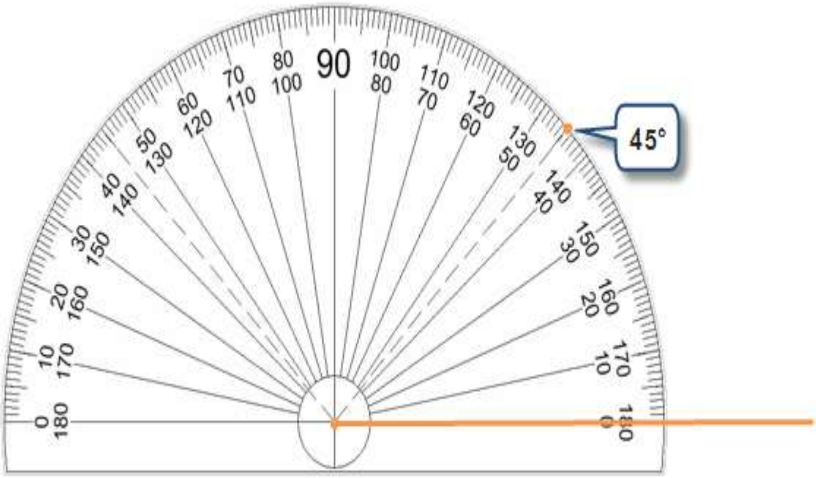


Fig. 1.8 French curve

1.4.5 Protractor



Protractor

1.5 LAYOUT OF DRAWING SHEET

The layout of drawing sheet is an important function of engineering drawing. The engineering student must know the standard rules for the selection of suitable scale, margin space, title block and part list etc. on the drawing sheet as shown in Fig. 1.9 and Fig. 1.10, according to IS 46 : 2003. Fig.1.11 shows the method of installing the drawing sheet.

The border line is drawn around a sheet by HB pencil. It is usually drawn at a distance of 30 mm from left hand side and 20 mm for the other three sides. The extra space which is kept on the left hand side is used for filing and binding purpose. For engineering students practice purpose, layout of drawing sheet is given below.

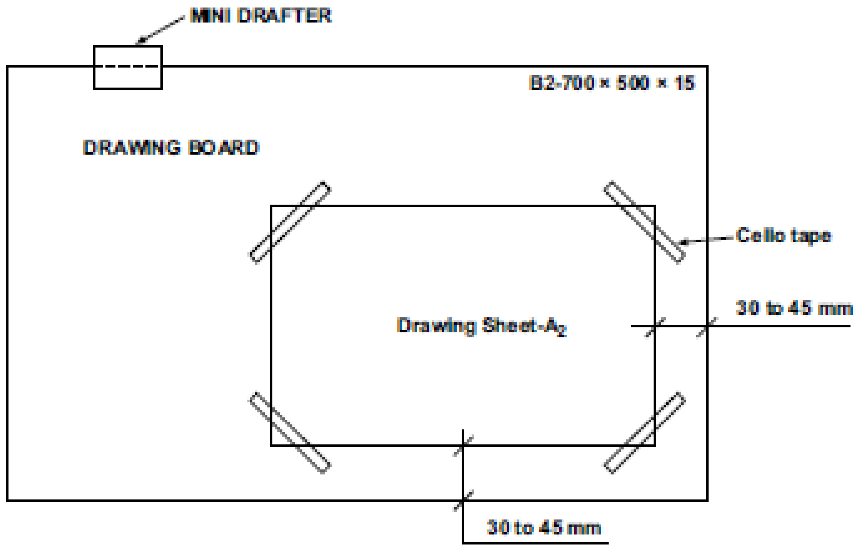


Fig. 1.9

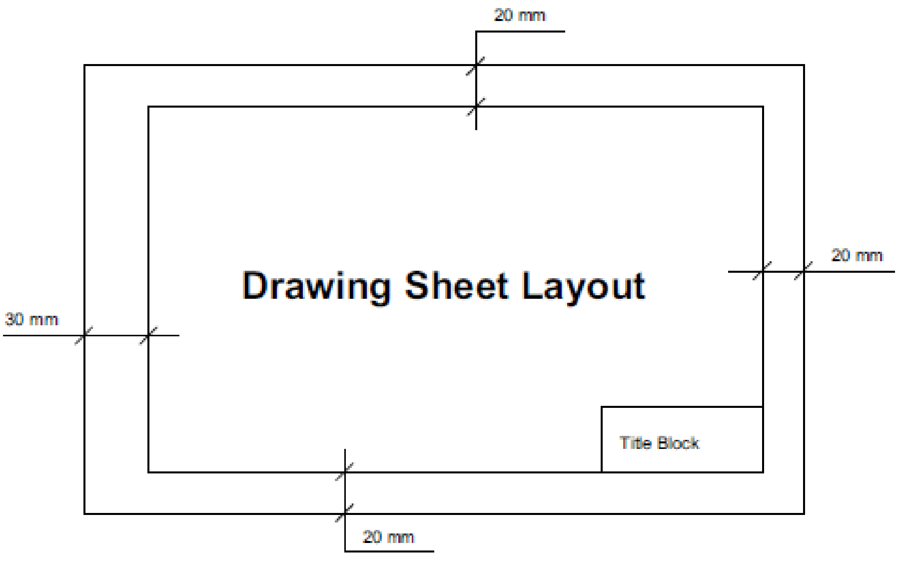


Fig. 1.10

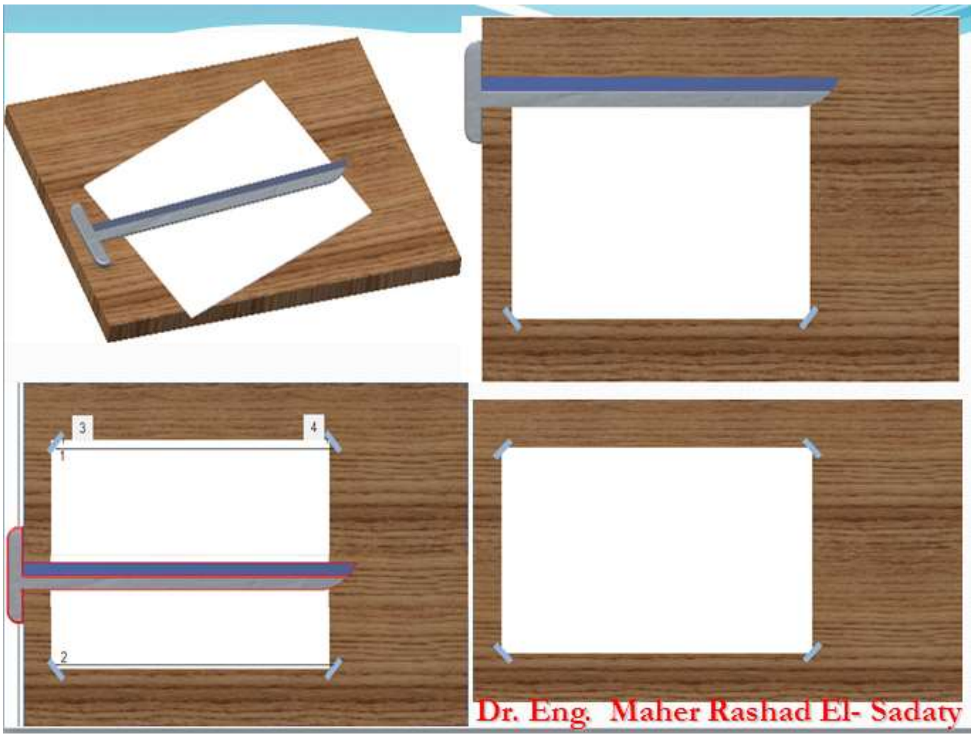


Fig. 1.11 Method of installing the drawing sheet

1.5.1 Title Block

Different types of title blocks are used in industrial as well as in engineering colleges. For all sizes of drawing sheets 65 × 185 mm size of title block is commonly used.

The title block provides the following information:

- Name of the institute or firm:
- Title of drawing:
- Sheet No:
- Scale:
- Symbol (1st angle or 3rd angle projection):
- Drawn by/Name:
- Class:
- Roll No:

- Starting date:
- Completion date:
- Checked by:

The different types of title blocks are shown in Fig. 1.12.

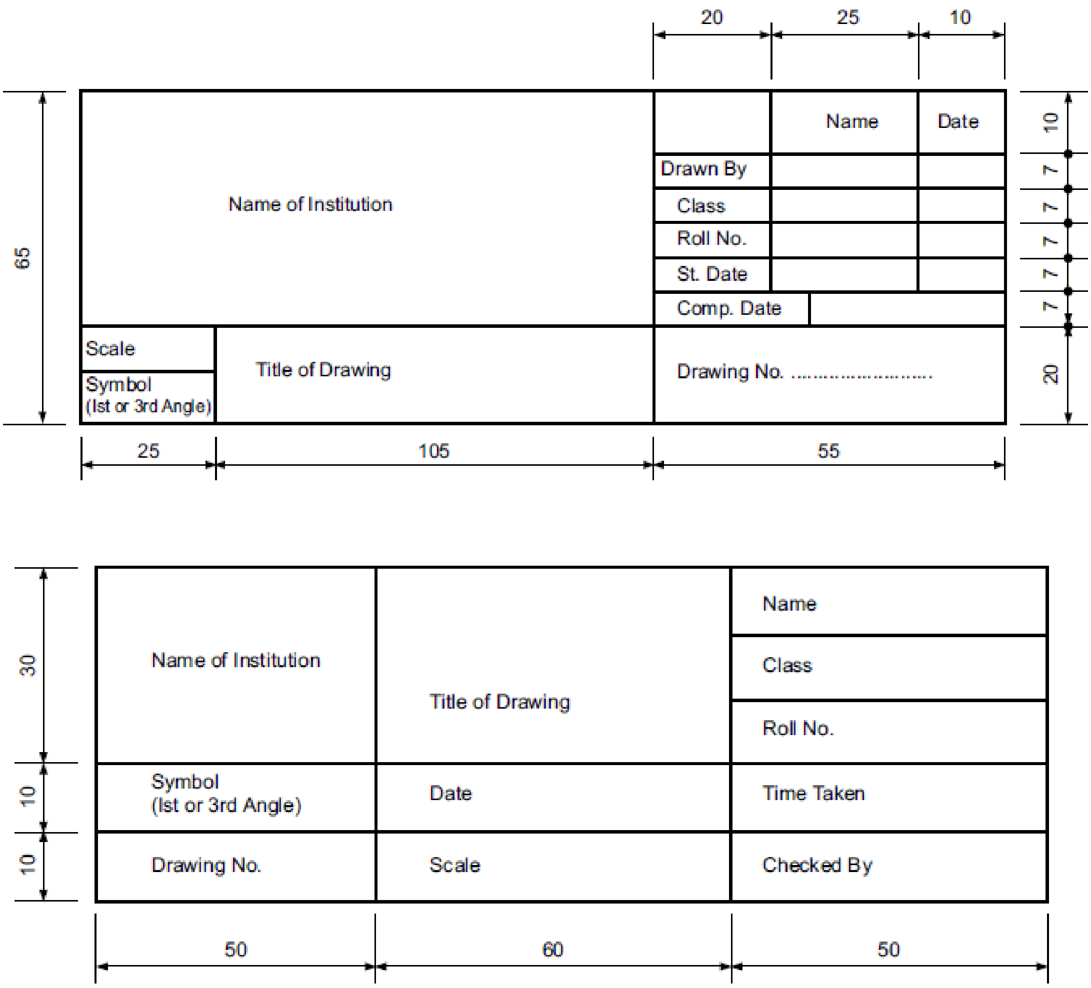


Fig. 1.12 Title block

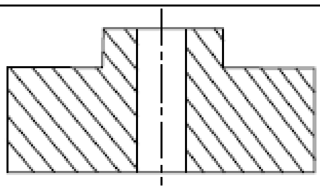
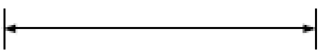
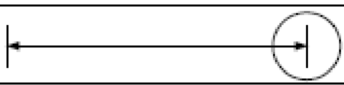
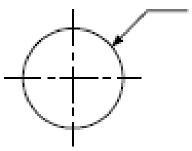
Chapter 2

Types of Lines and Free Hand Sketching

2.1 LINES

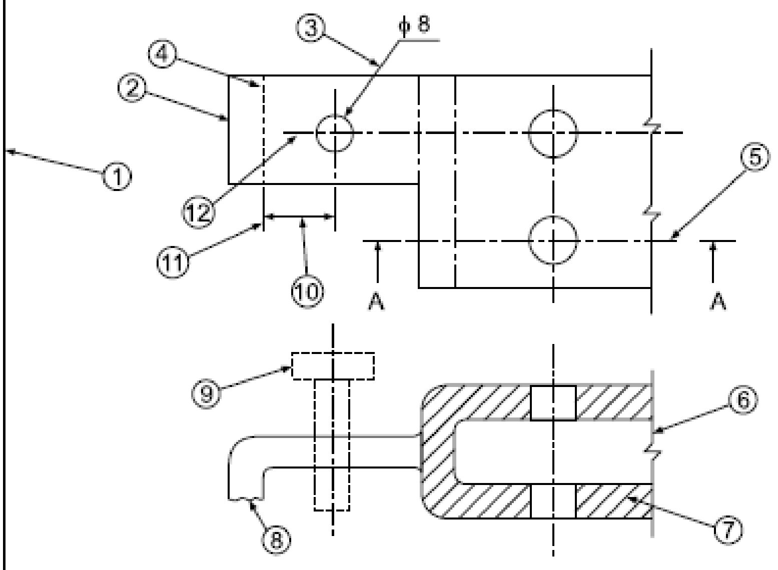
Different types of lines are used for different purposes in engineering drawing as described by S.P. 46-1988 which are known as an "ALPHABET OF LINES". The following types of lines should be used as given below in Table 2.1.

Table 2.1

S. No.	TYPES OF LINES	DESCRIPTION	THICKNESS OF LINE IN MM	GRADE OF PENCIL
1.		Border line	0.8	HB
2.		Visible line	0.6	H
3.		Center line	0.3	2H
4.		Hidden line	0.3	2H
5.		Cutting plane line	0.6	H
6.		Short break line	0.6	H
7.		Long break line	0.4	2H
8.		Section line	0.4	2H
9.		Dimension line	0.4	2H
10.		Extension line	0.4	2H
11.		Leader line	0.4	2H

Detailed description and uses of various lines are given in table 2.2.

Table 2.2 Different Types of Lines



1. BORDER LINE
2. VISIBLE LINE
3. LEADER LINE
4. HIDDEN LINE
5. CUTTING PLANE LINE (A — A)
6. LONG BREAK LINE
7. SECTION LINNING
8. SHORT BREAK LINE
9. ADJACENT PARTS. ALTERNATE POSITION OR REPEAT LINE
10. DIMENSION LINE
11. EXTENSION LINE
12. CENTRE LINE

2.2 TYPES OF LINES

2.2.1 Border Line

It is a thick continuous line used to draw boundary lines on the drawing sheet and title block lines at the bottom of drawing sheet, as shown in Table 2.1.

2.2.2 Visible Line

Outlines of parts in finished drawing are represented by thick lines. It is a continuous line which is also known as object line.

2.2.3 Centre Line

Centre line is used to locate the centre of arcs, circles and cylindrical objects. It should be thin, long and short dashes are evenly spaced in a proportion of 4:1 to 6:1.

2.2.4 Hidden Line

Hidden lines are used, where viewing surface of an object is not visible. Hidden line is represented by short dashes evenly spaced.

2.2.5 Cutting Plane Line

Cutting plane line is thin and long chain line which is thick at the ends. Cutting planes are designated by capital letters, with arrows indicating the direction for viewing section. It is just like a centre line.

2.2.6 Short Break Line

Short break line is drawn free hand for short breaks. It may be used on both details and assembly drawing. It is a thick curved line.

2.2.7 Long Break Line

Long break line represented by thin ruled straight lines with evenly spaced free hand zigzag, is used to shortening of long parts, which are the same throughout.

2.2.8 Section Line

Section line indicates plane cut in section view. These lines are usually drawn at an angle of 45º to the axis, with a spacing of 2 mm for small size drawing and 3 mm for large size drawings.

2.2.9 Dimension Line

Dimension line should be terminated by arrow head touching the extension lines on both ends. It is thin continuous line broken in the centre to insert the dimension.

2.2.10 Extension Line

Extension lines are projected from the outlines of the object and are usually perpendicular to the dimension line at a distance 2 mm from the outline of the object.

2.2.11 Leader Line

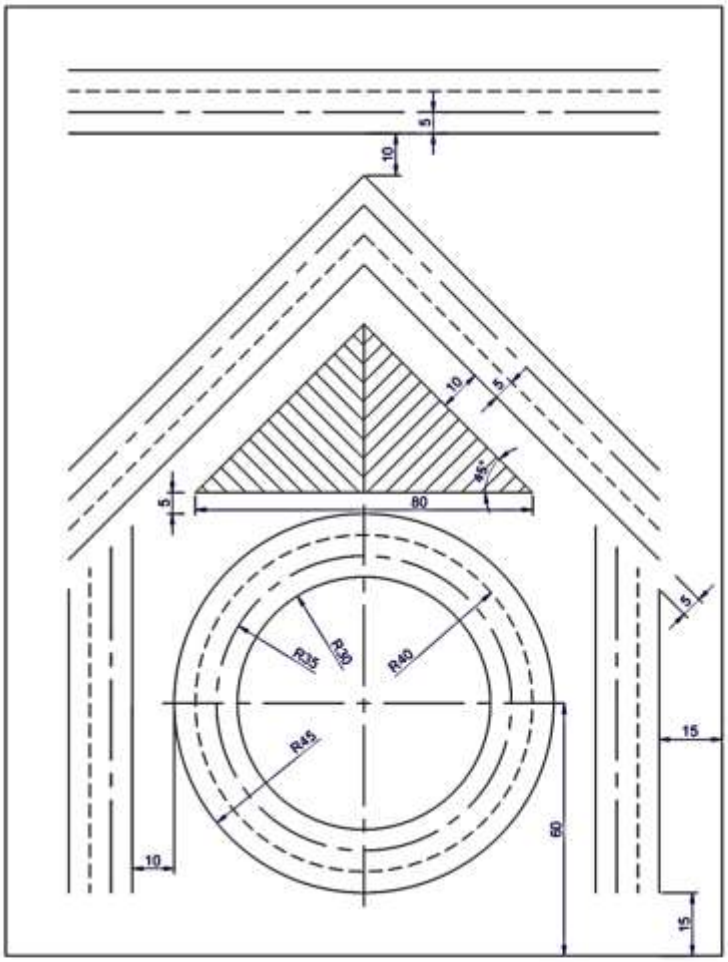
Leader lines contain numerals and indicate size of objects, and are generally 3 mm long. The angle of the leader line is not less than 30º.

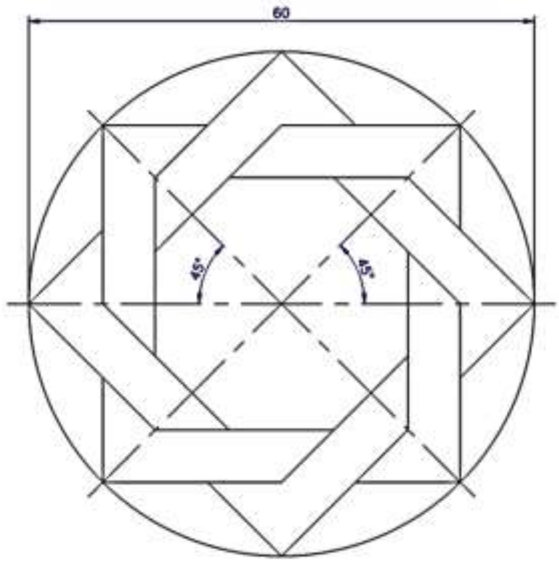
Basic Line Types

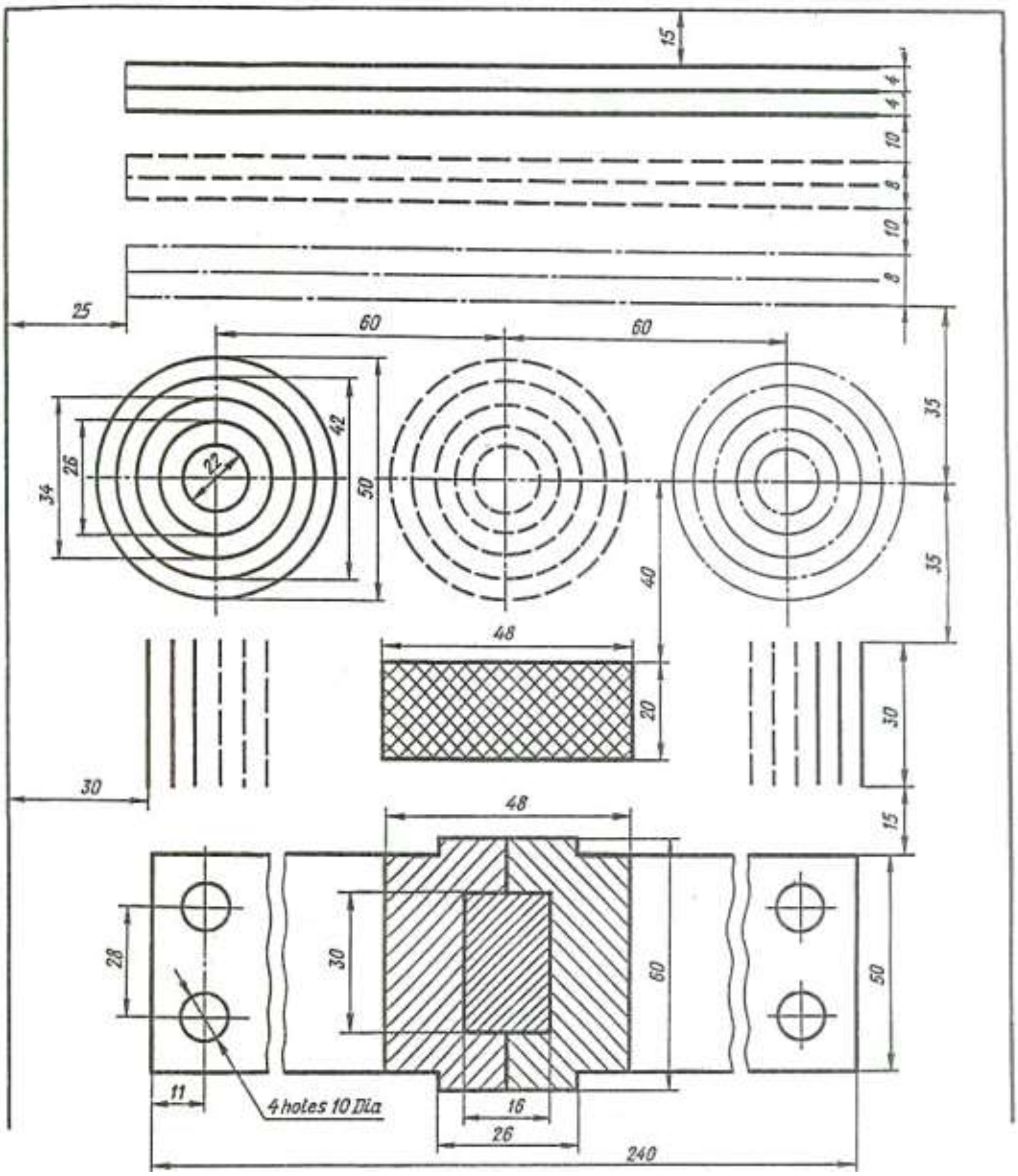
Types of Lines	Appearance	Name according to application
Continuous thick line		Visible line
Continuous thin line		Dimension line Extension line Leader line
Dash thick line		Hidden line
Chain thin line		Center line

NOTE : We will learn other types of line in later chapters.

Example:



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2.3 FREE HAND SKETCHING

Free hand sketching is an essential quality of a good engineer. Free hand sketching is used by the engineers as the first step to the preparation of a scale drawing i.e., a drawing drawn with the aid of instruments. In other words, a drawing prepared without the use of instruments is known as free hand sketching. An engineer expresses his/her ideas initially in the form of sketches which are later converted into drawing. In training, as in professional work, time can be saved by working freehand instead of working with instruments, because by using this method more problems can be solved in the alotted time. A freehand drawing is mostly made in correct proportion without the use of scale. Freehand sketching is never drawn on small scale. A good sketch should be achieved by continuous practice. The following instruments are required for free hand sketching:

(*i*) Pencil (A sharp conical pointed pencil)

(*ii*) Eraser

(*iii*) Paper

2.4 SKETCHING STRAIGHT LINES

2.4.1 Horizontal Lines

Horizontal lines are drawn from left to right neatly. The pencil is held with freedom and not too close to the lead point. Horizontal lines are drawn with the hand shifting to the position (shown in Fig. 2.1), using a wrist motion for short lines and a forearm motion for longer lines. Following procedures are used for drawing horizontal lines:

(*i*) Mark the starting and end point of the lines.

(*ii*) Position the arm for trial movement from left to right and hold the pencil at about 30 mm distance from the lead point.

(iii) Keep your eyes on the end point to which the line is to follow and sketch it with short and very light strokes.

(iv) Finally darken the line in a single stroke of pencil in correct direction as in Fig. 2.1.

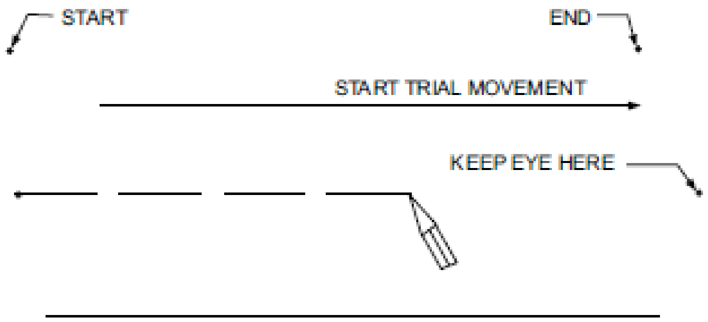


Fig. 2.1 Sketching Horizontal Lines

2.4.2 Vertical Lines

Vertical lines are drawn downward with a finger movement in a series of overlapping strokes as shown in Fig. 2.2.

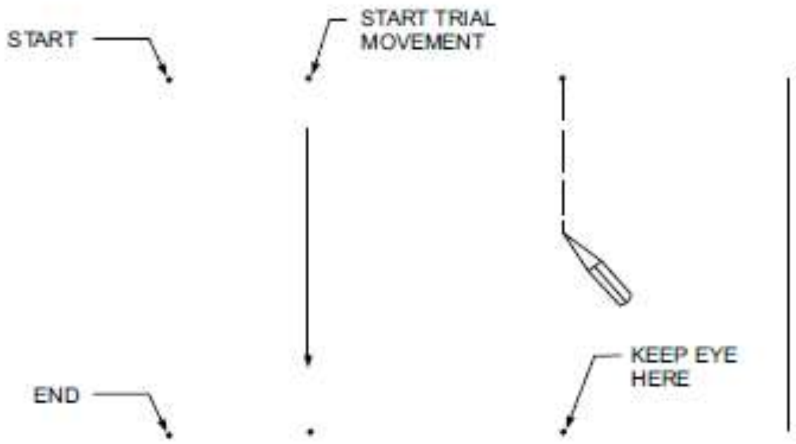


Fig. 2.2 Sketching Vertical Lines

2.4.3 Inclined Lines

Inclined lines running downward from right to left are drawn with the same movement as vertical lines, but the paper may be turned and the line may be drawn vertically as shown in Fig. 2.3.

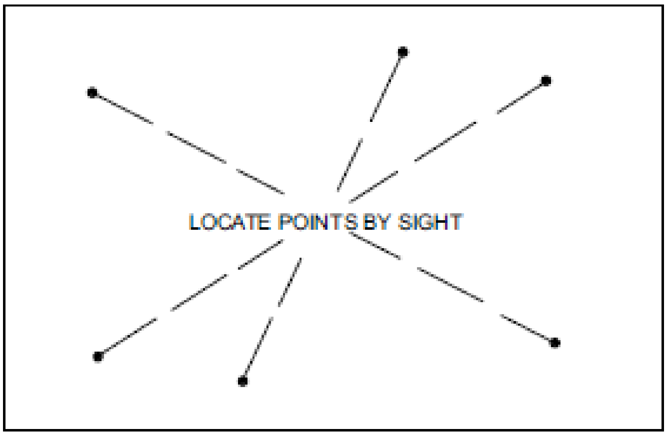


Fig. 2.3 Sketching Inclined Lines

2.5 SKETCHING OF CIRCLES

A circle can be drawn by marking radii on each side of the centre line. A more accurate method of sketching the circles is to locate a number of points through which the curve should pass as shown in Fig. 2.4.

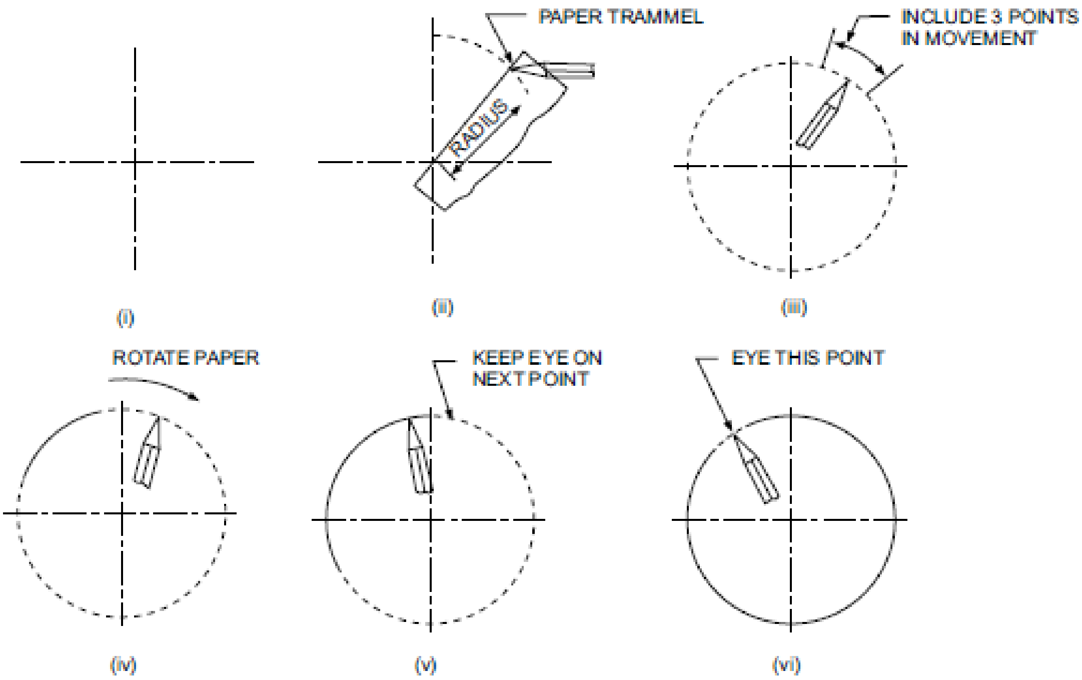


Fig. 2.4 Sketching of Circles

Following procedures should be adopted in order to drawn circles:

1. First of all sketch centre lines, horizontal and vertical lines.

2. Mark the points on these lines at radial distance from the centre by using paper as a trammer or judging by the eyes Fig. 2.4(*ii*)

3. Sketch first light arc and complete the circle as in Fig. 2.4(*iii*).

Large circles may be sketched by adding a few extra lines as shown in Fig. 2.4(*iv*). An easier method is to mark number of points by means of trammer and complete the circle as described above. While, a circle of small radius can be sketched with in a square. Sketch the circumscribing square and mark the diagonal as shown in Fig. 2.4(*v*). Mark the mid-point of the sides of the square and sketch the circles through these points to complete the circles respectively as shown in Fig. 2.4(*vi*).

2.6 SKETCHING OF ELLIPSES

For sketching an ellipse, it should be noted that an ellipse has a centre, through which all its diameters pass. The largest diameter is called the major axis and shortest diam eter is called the minor axis. The ellipse is always symmetrical in shape about both of these axes. Sketch one fourth of the ellipse by drawing arc as shown in Fig. 2.5(i), (ii) and then complete the ellipse.

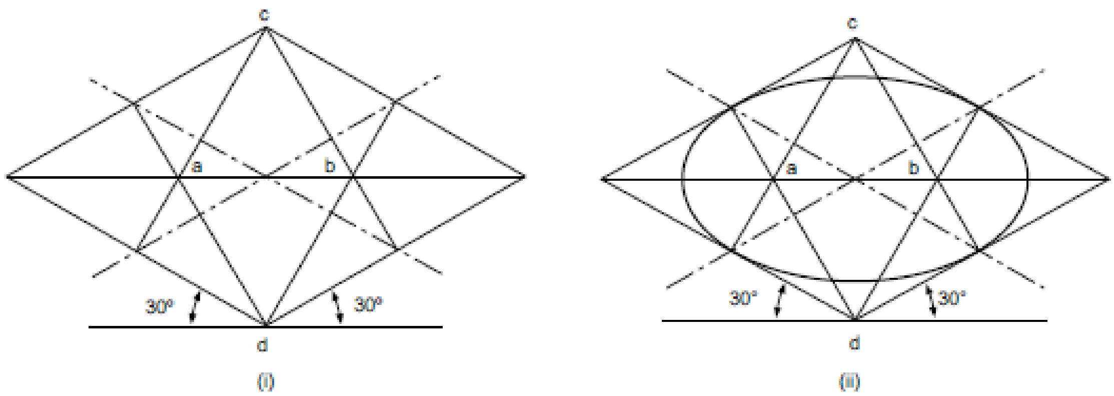


Fig. 2.5 Sketching of Ellipses

2.7 SKETCHING OF ORTHOGRAPHIC

In Orthographic sketching, all the necessary views i.e. front view, top view and side view are drawn. The procedure followed in making a sketch of an object is almost the same as that in drawing with instruments. The procedure followed in making an orthographic sketch is as follows:

1. Study the object carefully and decide the view which shows the best shape of an object.

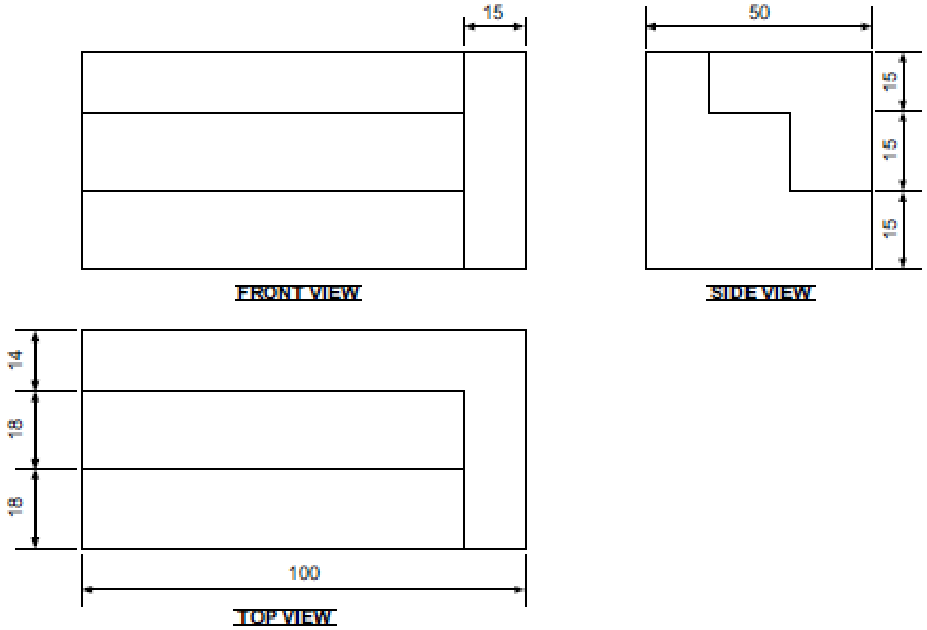


Fig. 2.6 Sketching of Orthographic

2. Draw the rectangle by faint lines in which the view is to be sketched as in Fig. 2.6.

3. Sketch the centre lines and hidden lines of an object if required by faint or construct lines.

4. Complete all the views of an object and rub the construction lines.

5. Dimensioning the object and add the notes where required and complete the drawing.

2.8 SKETCHING OF ISOMETRIC

The isometric sketches are prepared on a plain paper. Use the box method to draw the isometric sketches. The following procedure should be followed for making the isometric sketches.

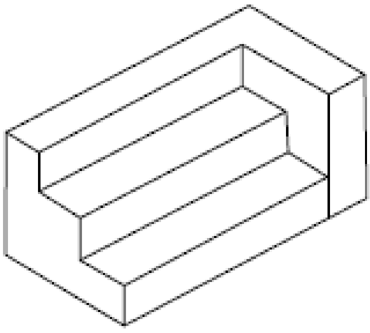
1. Study the orthographic view carefully and decide the position in which it should be drawn.

2. Draw the orthographic view in a box, to show its length, breadth and height.

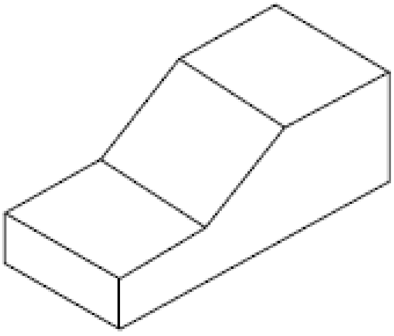
3. Isometric lines and darkens the required lines.

4. Sketch the required hidden lines.

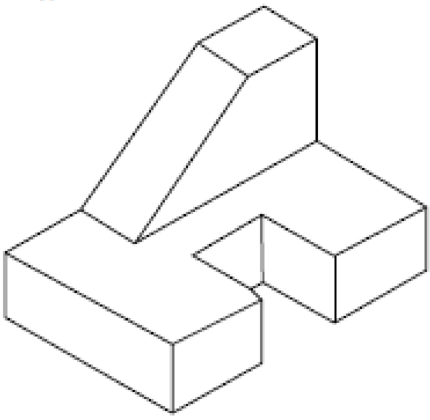
5. Add the necessary dimensions, notes etc. as shown in Fig. 2.7.



(i)



(ii)



(iii)

Fig. 2.7 Sketching of Isometric

2.9 LETTERING AND METHODS OF DIMENSIONING

2.9.1 Lettering

Lettering is an important part of engineering drawing which provides the complete information about size of an object and appearance required.

Writing of titles, sub-titles, dimensions and other relative details on drawing should be lettered with freehand. A good practice of freehand lettering improves the quality of drawing and is also executed neatly, uniformly and rapidly. The use of instruments for lettering consumes more time as compared with freehand lettering. Both the vertical (up-right) and sloping (italic) letters can be lettered freehand as well as by instruments. Capital letters should be used for all purpose except where lower case letters are accepted in international usage for abbreviations. A good practice of lettering is required which may be achieved by continuous efforts. Normally two types of lettering are commonly used by engineers which are:

1. Single stroke letters
2. Double stroke letters.

But another important type of lettering is the gothic style of lettering which is commonly used by draftsmen as well as the engineering students for writing title block and other features. It may be performed with single stroke without lifting the pencil.

2.9.2 SINGLE STROKE LETTERS

Single stroke letters are used universally in engineering drawing. The Bureau of Indian Standards ISO 9609 : 2001/ ISO 3098-0 : 1997 replaced by SP: 46-2003 also recommended the use of single stroke letters. The term “Single stroke” means the uniform width of letter is obtained in single stroke of pencil. Single stroke letters are of two types.

1. **Single Stroke Vertical Letter**

Single stroke vertical letters and numerals are written in a vertically upward direction without the use of drawing instruments as shown in Fig.

2.8. For writing the single stroke vertical letters and numerals, first draw the faint guide lines at a distance equal to the height of letters. These guide lines should be drawn by light grade pencil (2H). The height of letters and numerals are written in 1.8, 2.5, 3.5, 5, 7, 10, 14 and 20 mm. The ratio of height to width is 7: 4 for single stroke vertical letters except M and W which is of 7:5 ratios. The ratio of height to width of all the numerals is also written in 7: 4 except 1. There is no hard and fast rule for spacing between two letters. Generally the gap between two letters is taken as one unit. The spacing between two words is generally taken as 1 to 1.5 times their height and spacing in between two sentences is twice the height of letters as shown in Fig. 2.9.

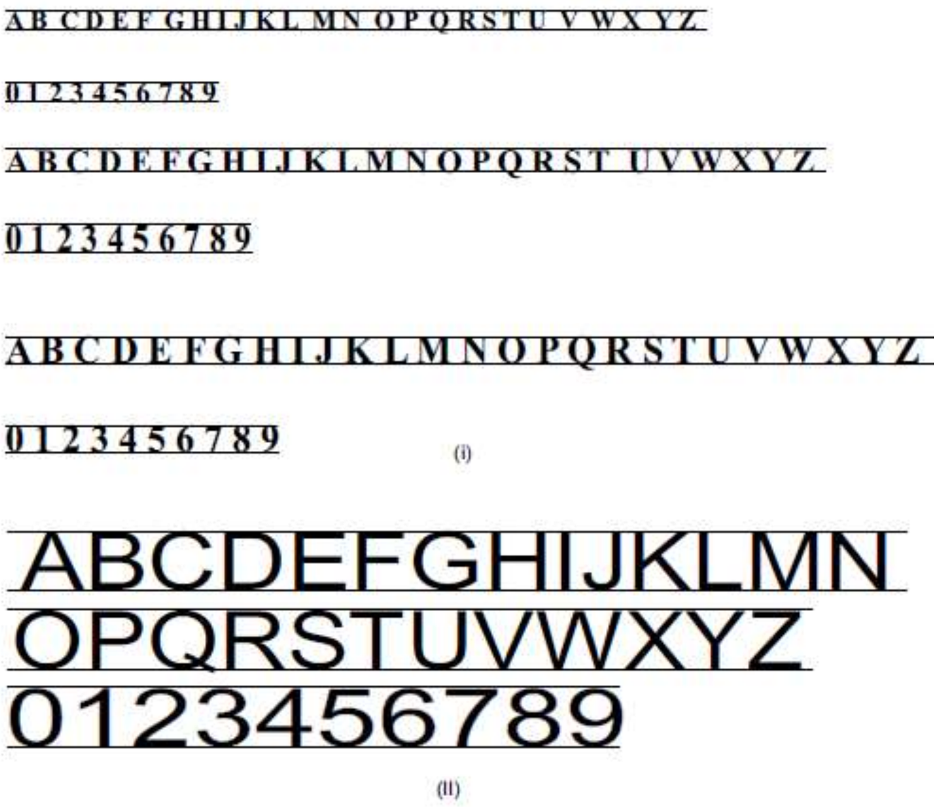


Fig. 2.8

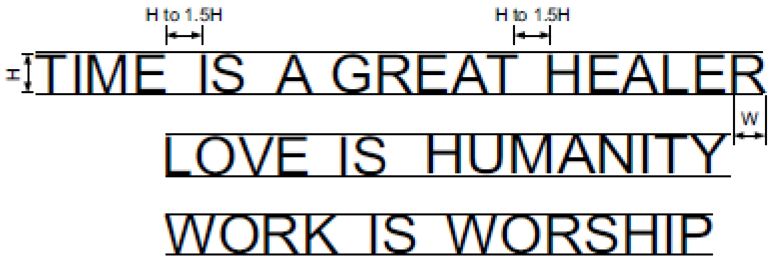


Fig. 2.9

2. THE HEIGHT OF LETTERS AND NUMERALS

The height of letters and numerals recommended by IS 9609 (Part O): 2001/ISO 3098-0: 1997 for different purposes according to drawing size are shown in Table 2.3.

Table 2.3

S.No.	Purpose	Height of Letters and Numerals in MM
1.	Title of drawing and drawing number in title block	6, 8, 10, 12
2.	Sub-titles and headings	3, 4, 5, 6
3.	Notes, material list, dimensioning and the tolerances	2, 3, 4, 5

2.9.3 METHODS OF DIMENSIONING

It is the art of describing the size of an object by supplying the complete information stating length, breadth, height, angle, arcs, size and position of holes etc. Lines, symbols, figure and notes are used for this purpose.

The elements of dimensioning are the projection lines or extension lines, dimension lines, leader lines etc. as shown in Fig. 2.10. The dimension without any unit is considered in mm

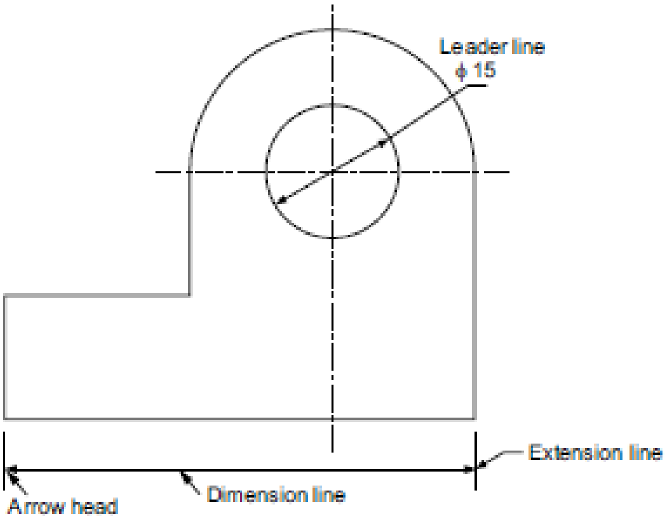


Fig. 2.10

2.9.4 NOTATION OF DIMENSIONING

The notation of dimensioning consists of arrow heads, leader lines, dimension lines, extension lines, symbols, notes etc. According to IS: SP 46-2003 these notations are explained below.

1. **Arrowheads**

They should be of isosceles triangular shape, the length (L) of which is three times of its width (w). It is placed at each end of dimension line by touching the extension line. The space between arrowheads is filled up with HB pencil as shown in Fig. 2.11.

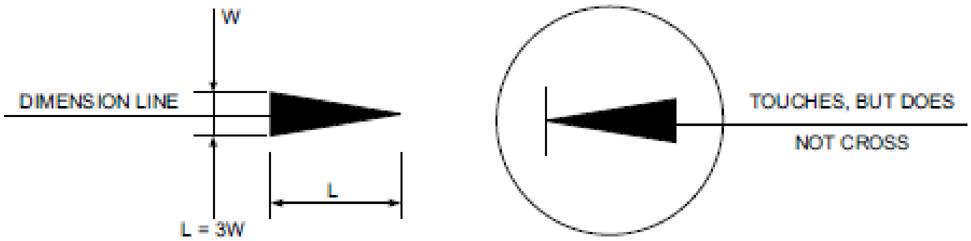


Fig. 2.11

2. Leader Line

A leader line is a thin continuous line containing notes and terminated by an arrow heads touching the line to be pointed out at an angle of 30º or 45º. The straight horizontal line of about 3 mm is used for writing dimensions or notes as shown in Fig. 2.12.

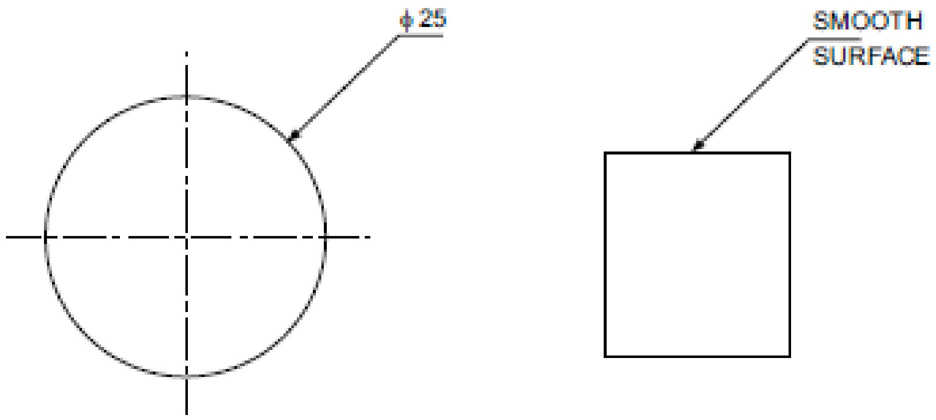


Fig. 2.12

3. Dimension Line

A dimension line is a thin continuous line. It should be terminated by arrow heads touching the extension lines on both ends. Dimension lines are light lines as shown in Fig. 2.13.

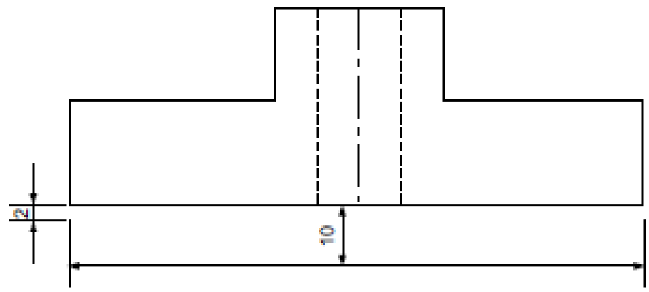


Fig. 2.13

4. Extension Lines

An extension line is a thin continuous line, extended from the outline of the objects. The extension line extends about 3 mm beyond the dimension line and the gap between an outline and the extension line is about 2 mm as shown in Fig. 2.13.

2.10 GENERAL PRINCIPLES OF DIMENSIONING

According to IS:SP46-2003 following principles of dimensioning are recommended:

1. Each feature is dimensioned and positioned only once.

2. All the necessary dimensions of the parts must be written on drawing sheet to show the correct functioning of each part.

3. Avoid the unnecessary dimensioning; every dimension should be given in one view only. Avoid repeating in second view.

4. Each feature is dimensioned and positioned where its shape shows.

5. Dimension should be given in the view which shows relative feature, more clearly.

6. Mark the dimensions outside the view only.

7. Crossing of dimension lines should be avoided. It should be placed in such a way that they do not cross each other as shown in Fig. 2.14.

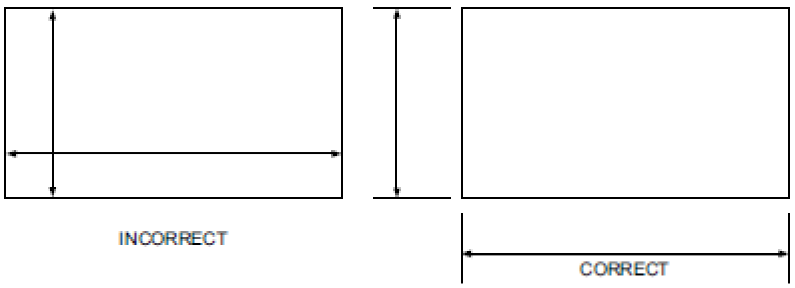


Fig. 2.14

8. Dimension should not be placed too close to each other.

9. Each drawing shall use the same unit for all dimensions.

10. Dimensioning to a centre line should be avoided except when the centre line passes through the centre of a hole as shown in Fig. 2.15.

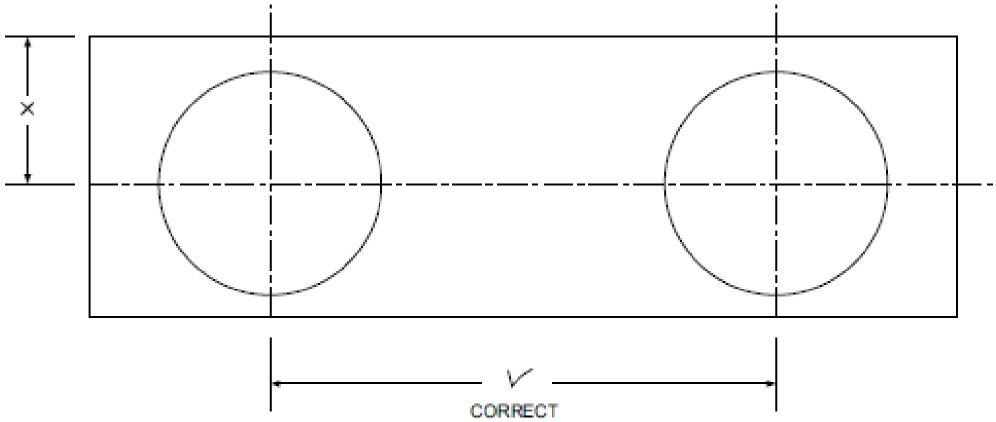


Fig. 2.15

11. Dimensioning for narrow space are shown in Fig. 2.16.

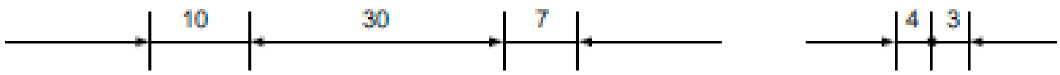


Fig. 2.16

12. Dimension should be taken from visible outline instead of hidden lines as shown in Fig. 2.17.

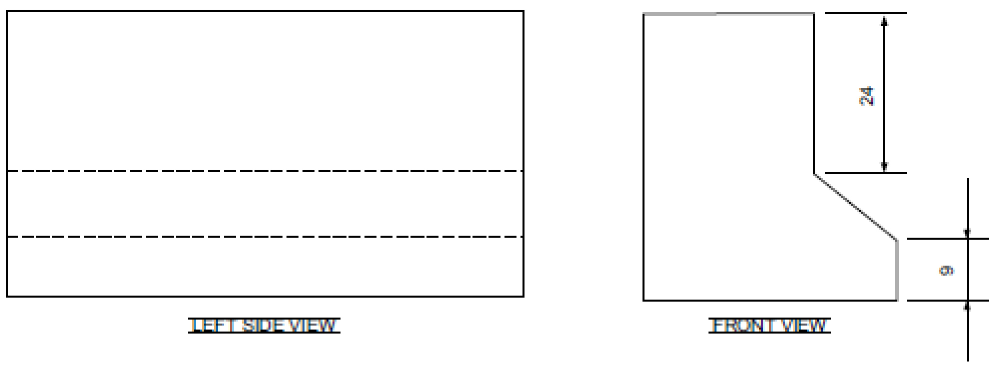


Fig. 2.17

13. Avoid the crossing of dimension lines with extension lines as shown in Fig. 2.18.

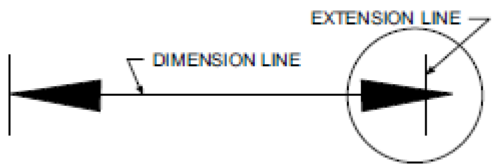


Fig. 2.18

3.11DIMENSIONING TECHNIQUES FOR COMMON FEATURES

1. Circles

Circles of different sizes should be dimensioned and diameter should be denoted by "φ", as per IS; SP46–2003 as shown in Fig. 2.19.

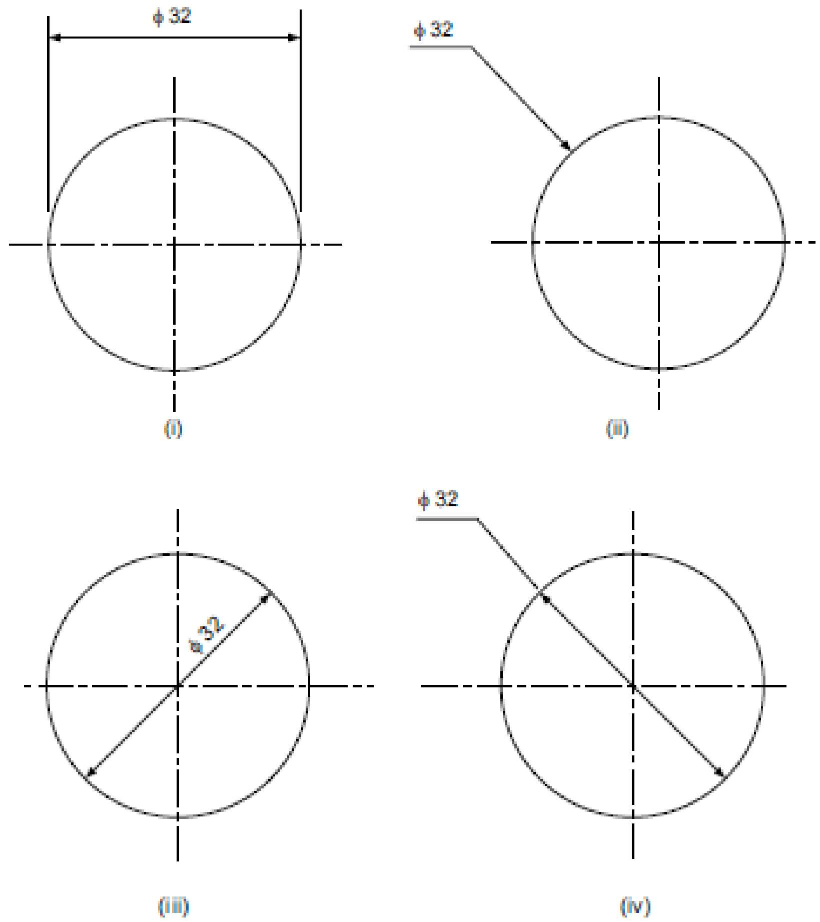


Fig. 2.19

2. Arcs

Arcs should be dimensioned by their radius which is shown preferably outside the line of the object. It is denoted by "*R*" as shown in Fig. 2.20.

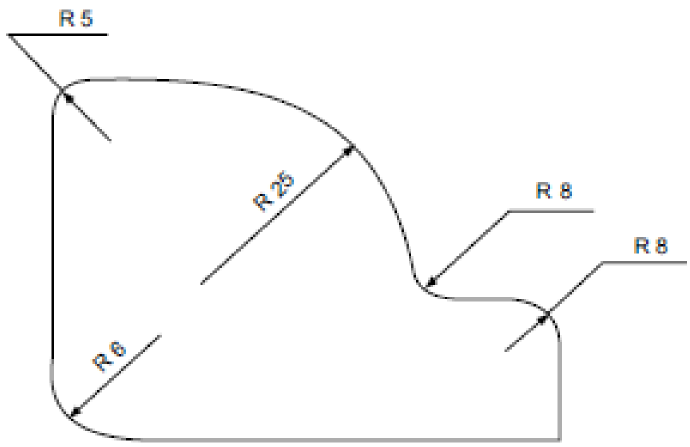


Fig. 2.20

3. Angles

Dimensions of angles and chords are expressed by degrees, on the arc swing from vertex as shown in Fig. 2.21.

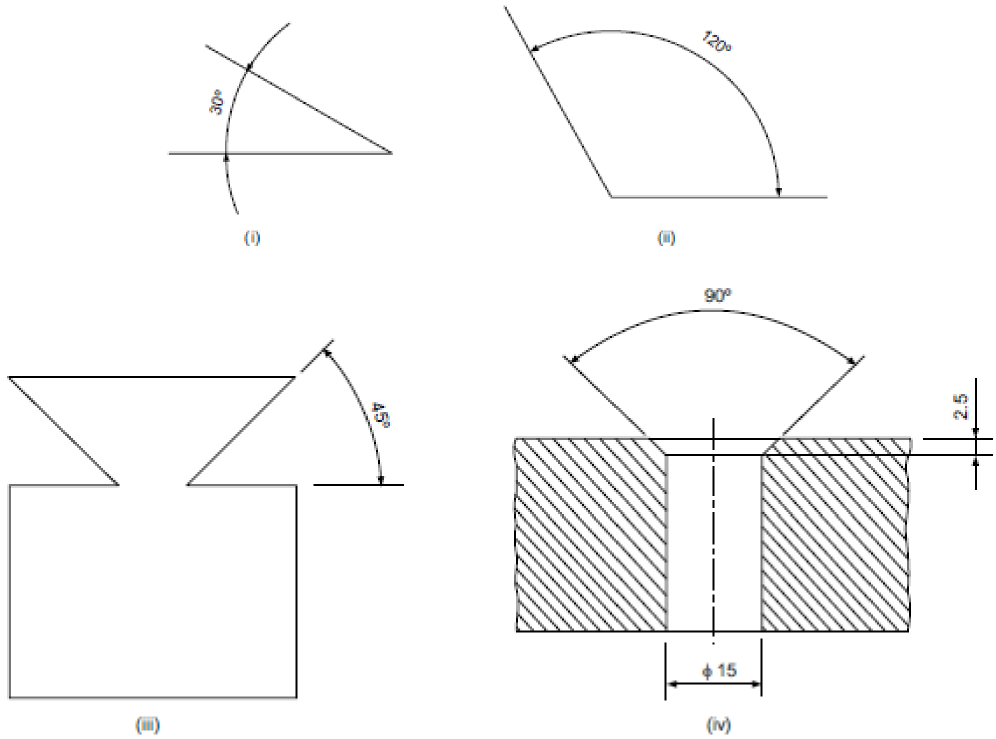


Fig. 2.21

4. Chamfers

Chamfers may be dimensioned by notes as shown in Fig. 2.22.

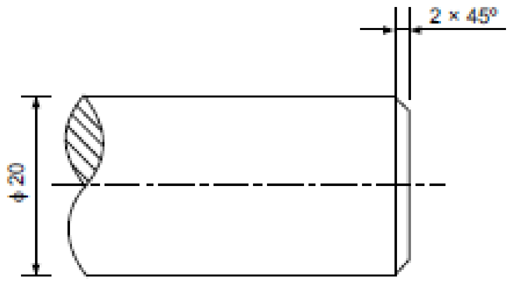


Fig. 2.22 Chamfer

5. Holes

The methods of dimensioning holes are shown in Fig. 2.23.

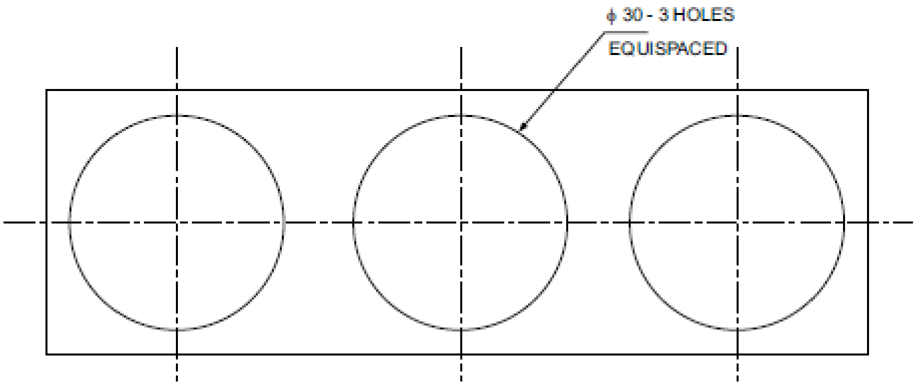


Fig. 2.23

6. Tapers

Dimensioning of tapered objects are shown in Fig. 2.24.

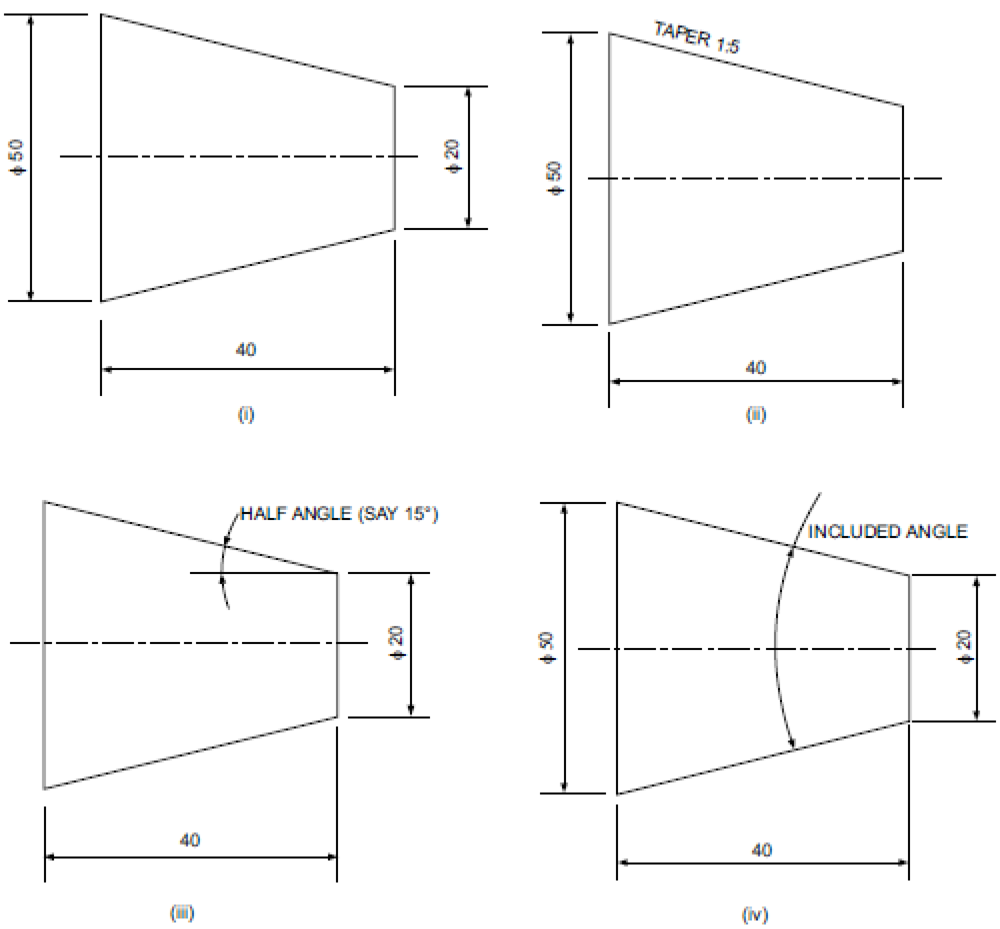


Fig. 2.24

7. Chain Dimensioning

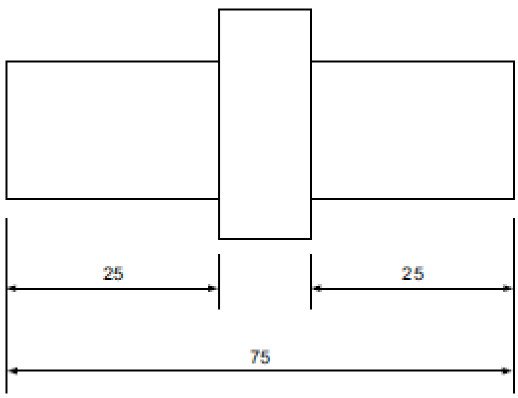


Fig. 2.25

8. Parallel Dimensioning

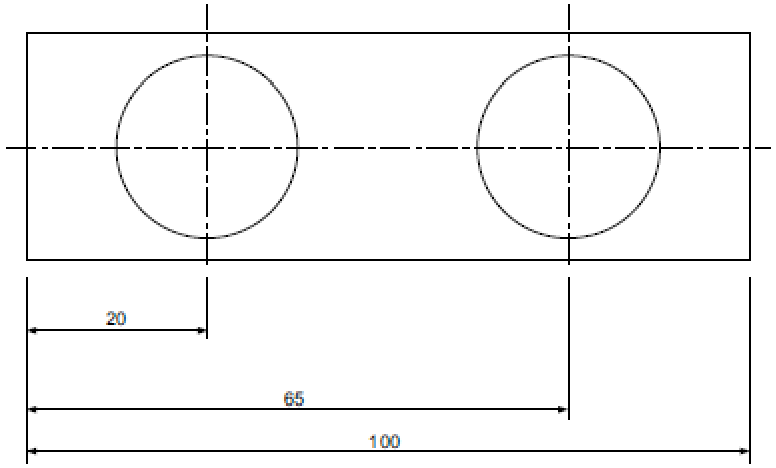


Fig. 2.26

9. Combined Dimensioning

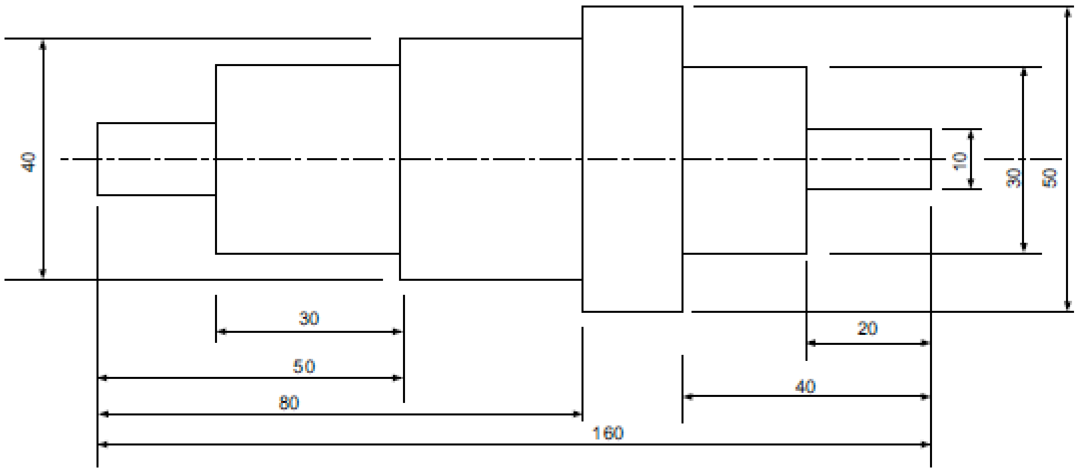
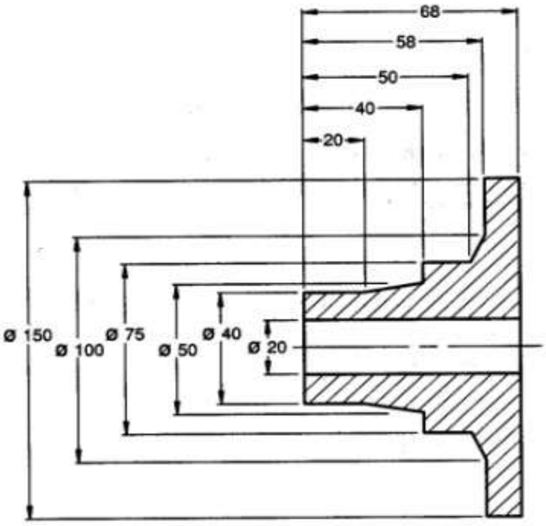


Fig. 2.27

Staggering Dimensions



- Put the lesser dimensions closer to the part.
- Try to reference dimensions from one surface
  - This will depend on the part and how the tolerances are based.

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Extension Line Practices

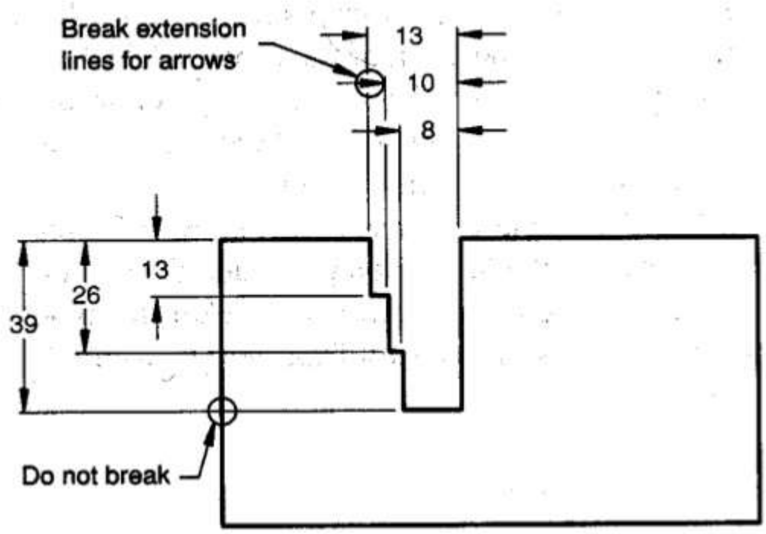

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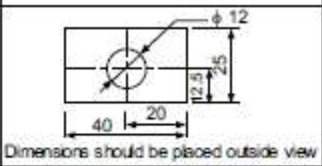
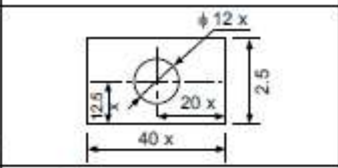
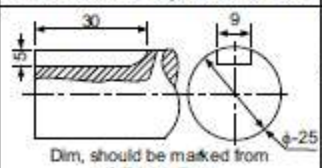
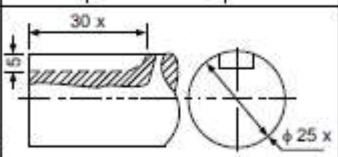
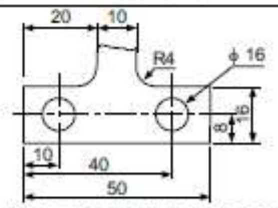
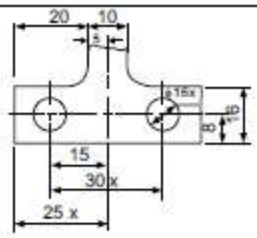
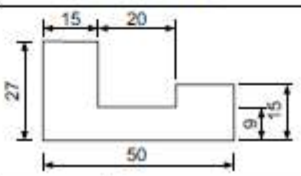
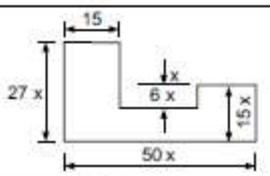
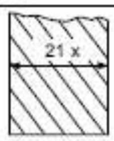
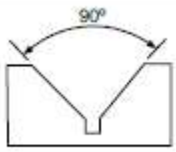
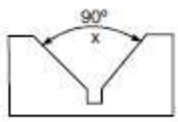
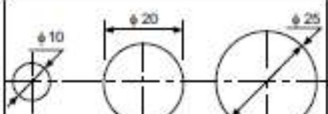
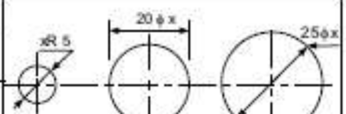
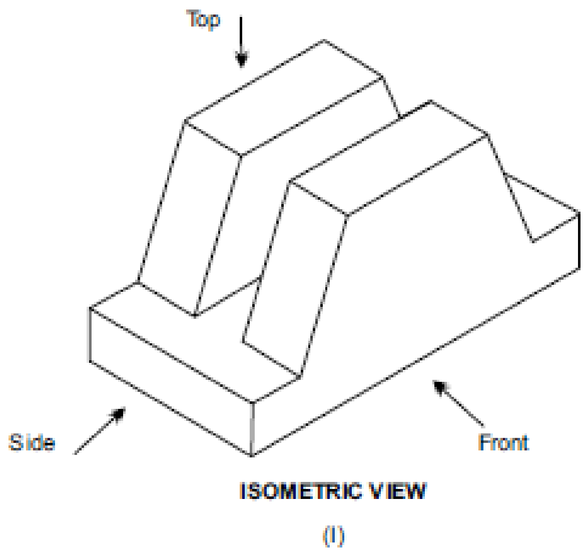
Table 2.4 shows correct and incorrect dimensioning

Table 2.4

	CORRECT	INCORRECT	REASONS FOR INCORRECT
(i)	 Dimensions should be placed outside view		1. Arrow head not proportionate. 2. Hole dimension shown in figure. Leader line not ending horizontally. 3. Dimension '40' is too close.
(ii)	 Dim, should be marked from visible outlines		1. A key-way is shown with a dotted line where the dimensions are placed. 2. Leader line for the shaft diameter is drawn horizontally touching the boundary line.
(iii)	 Dimensions should be given from the outlines (finished surfaces) or a centre line of a hole		1. Dimensions are given from the mid-line of the object. 2. Dimensions of holes are shown inside the figure. 3. Dimensions are shown in vertical line. 4. Smaller dimensions (25 mm) precedes the larger dimensions (30 mm) 5. Fillet radius is not shown.
(iv)			1. Dimension lines are used as extension. 2. Dimensions are placed insidethe view. 3. Dimension 27 and 50 not written according to aligned system.
(v)			Section-lines overlap the dimension 21.
(vi)			The outlines of the object are used as the extension lines.
(vii)			1. Smaller circle is designated with radius. 2. Convention 'ϕ' for diameter is placed after dimension. 3. Leader has arrow and it is drawn horizontal.

Problem 1. Fig.2.28 (*i*) and (*ii*) show the pictorial and orthographic views of an object. **Complete the dimension of the given object.**



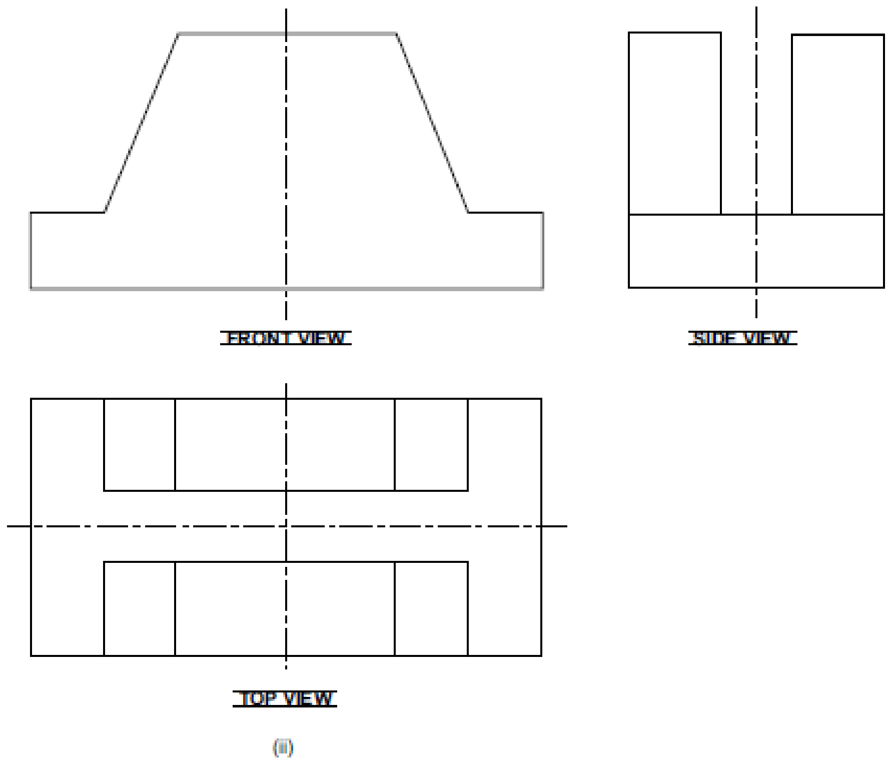
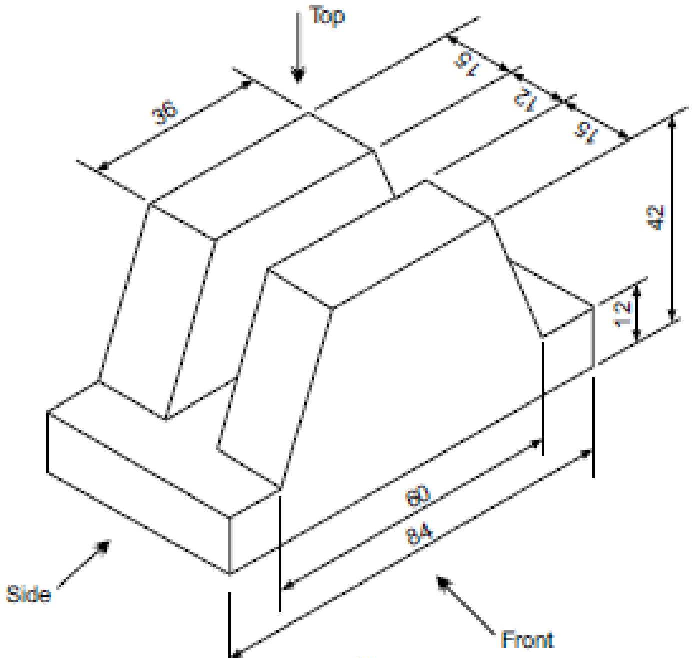
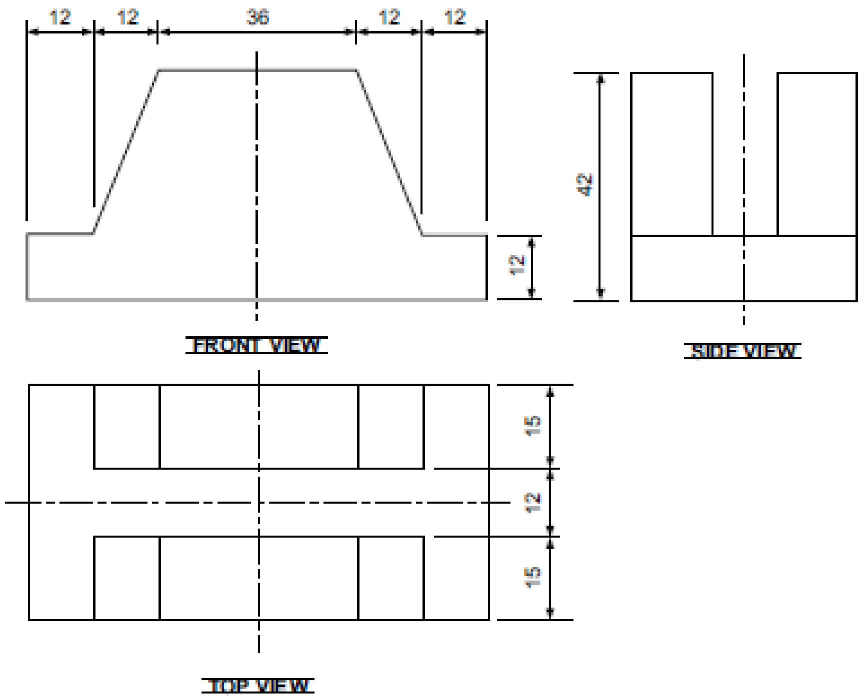


Fig. 2.28

Solution:


Top
36
15
12
15
42
12
60
84
Side
Front

(i)


12
12
36
12
12
42
12
FRONT VIEW
SIDE VIEW
15
12
15
TOP VIEW

Problem 2: Fig.2.29 (*i*) and (*ii*) show the isometric view and orthographic view of an object. Complete the dimensioning of the given object.

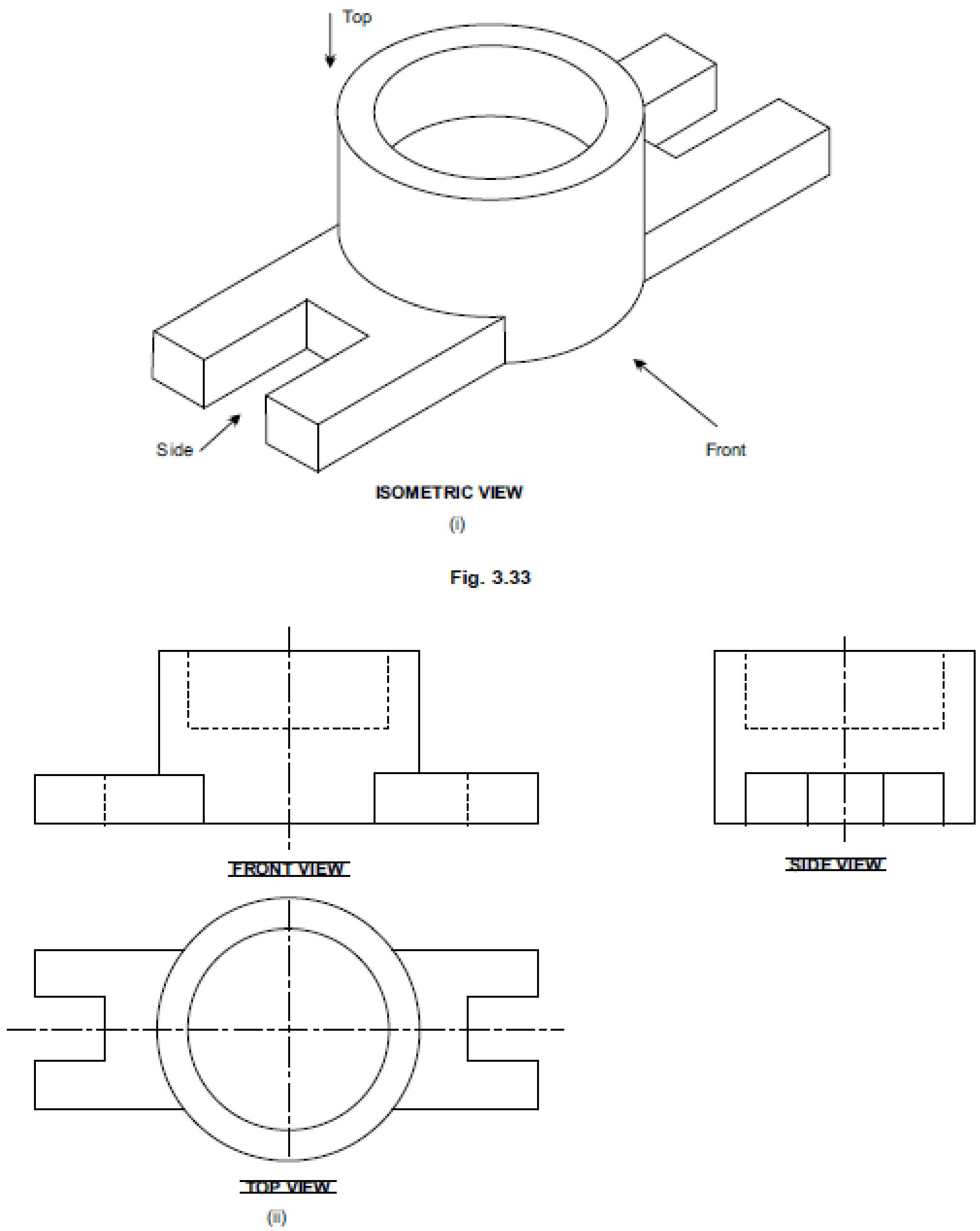
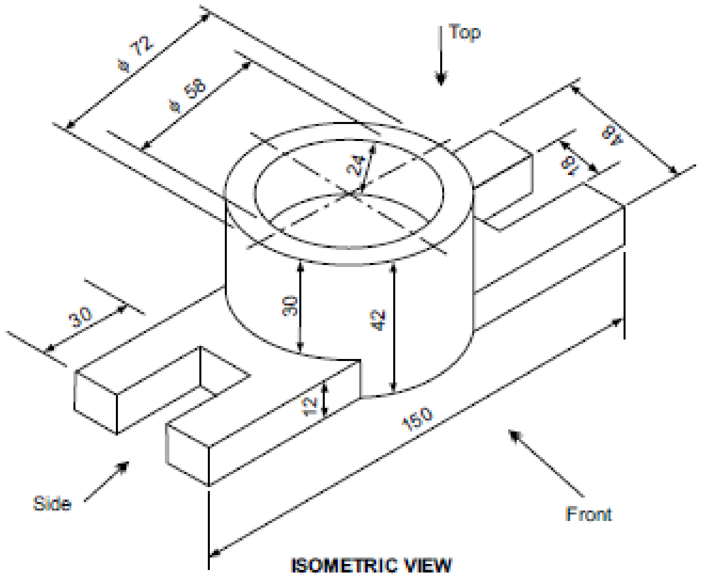
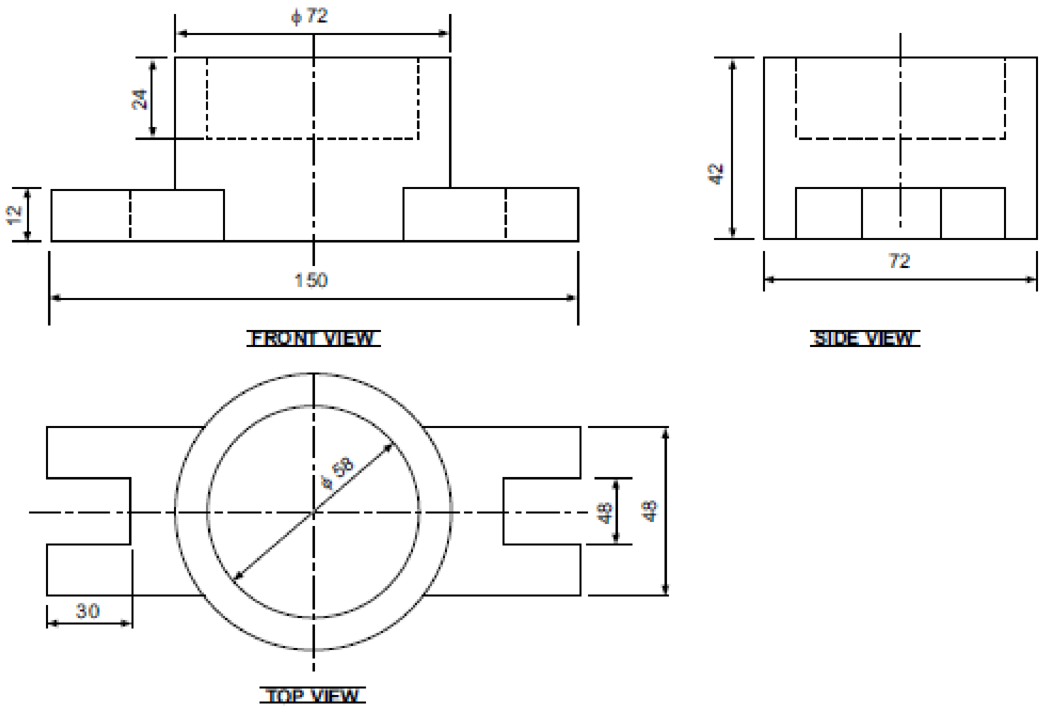


Fig. 2.29

Solution:



ISOMETRIC VIEW
(i)



(ii)

Chapter 3

Principles of Engineering Drawing and Geometrical Construction

Principles of Engineering Drawing and Geometrical Construction

3.1 INTRODUCTION

Geometry is the basis of all technical drawings. The knowledge of the principles of geometric construction and its applications are essential to an engineer. An engineer, must know how to draw various types of lines which can be a straight line, a circle, an arc of circle, a circular curve etc. This chapter provides information as well as deals with the drawing of polygons and noncircular curves like an ellipse, a parabola or a hyperbola.

3.2 TERMS USED IN GEOMETRICAL CONSTRUCTION

Definitions:

1. **Surface**

A plane surface has length and breadth but no thickness as shown in Fig. 3.1.

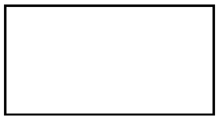


Fig.3.1

2. **Point**

A point is that which has position but has no magnitude. It is simply represented by a small dot as shown in Fig. 3.2.

Fig.3.2

3. **Line**

A line is that which has length but no thickness e.g., the boundary of a surface is a line.

(*i*) **Straight Line:** The shortest distance between two points is known as straight line as shown Fig. 3.3.

Fig.3.3

Fig.3.4

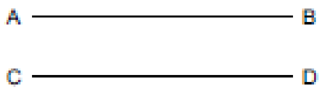
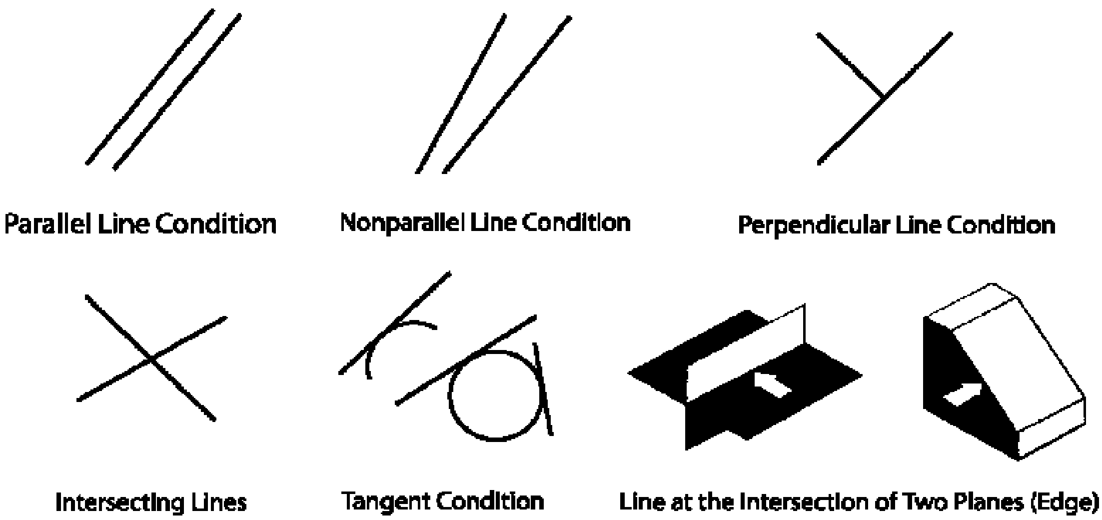


Fig.3.5

Relationship of one line to another line or arc



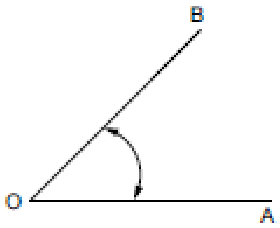
(*ii*) **Curved Line:** A curved line is that which does not lie in straight direction as shown in Fig. 3.4.

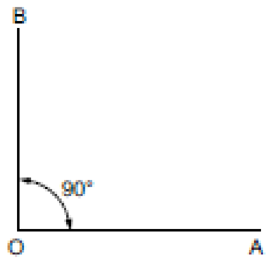
(*iii*) **Parallel Lines:** Parallel lines are those lines, which fall equal distance apart and never meet to each other, if they are extended in any direction as shown in Fig. 3.5.

4. **Angle**

An angle is formed between two intersecting lines, drawn from the same point as shown in Fig. 3.6.

(*i*) **Right Angle:** A right angle is the inclination between two perpendicular lines or an angle of 90º as shown in Fig. 3.7.





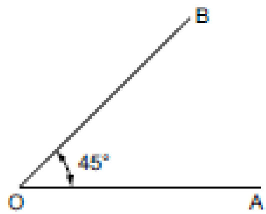


Fig. 3.6 **Fig. 3.7** **Fig. 3.8**

(*ii*) **Acute Angle:** An angle which is less than 90º is known as an acute angle as shown in Fig. 3.8.

(*iii*) **Obtuse Angle:** An angle which is greater than 90º then it is known as obtuse angle as shown in Fig. 3.9.

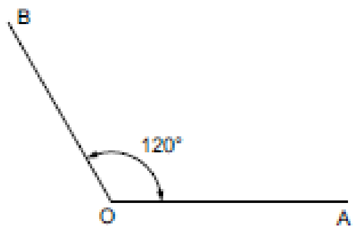
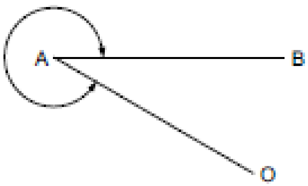


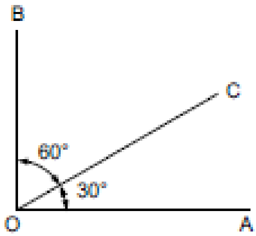
Fig. 3.9

(*iv*) **Reflex Angle:** An angle which is greater than two right angles then it is known as reflex angle as shown in Fig. 3.10.

(*v*) **Complementary Angles:** If the sum of two adjacent angles is equal to one right angle, they are known as complementary angles as shown in Fig. 3.11.

(*vi*) **Supplementary Angles:** If the sum of two adjacent angles is equal to two right angles, they are known as supplementary angles as shown in Fig. 3.12.





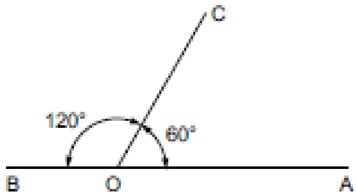


Fig. 3.10 **Fig. 3.11** **Fig. 3.12**

3.3 POLYGON

A polygon is a plane figure bounded by more than four sides. If the sides and angles of a polygon are equal, then it is known as a regular polygon and if they are unequal, then it is known as an irregular polygon. The sum of external angles of a regular polygon is 360º and each external angle is 360º*N*. Where *N* is the number of sides.

Polygons are named according to the number of their sides and angles are given below:

(*i*) **Regular Pentagon:** A polygon having five equal sides is known as a regular pentagon as shown in Fig. 3.13.

(*ii*) **Regular Hexagon:** A polygon having six equal sides is known as regular hexagon as shown in Fig. 3.14.

(iii) **Regular Heptagon:** A polygon having seven equal sides is known as regular heptagon as shown in Fig. 3.15.

(iv) **Regular Octagon:** A polygon having eight equal sides is known as regular octagon as shown in Fig. 3.16.

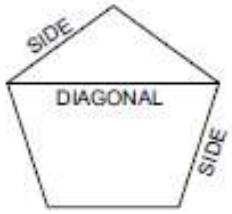


Fig. 3.13

Fig. 3.14

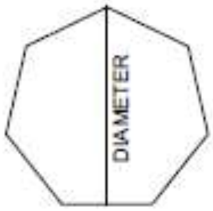


Fig. 3.15

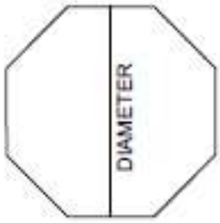


Fig. 3.16

(v) **Regular Nonagon:** A polygon having nine equal sides is known as regular nonagon as shown in Fig. 3.17.

(vi) **Regular Decagon:** A polygon having ten equal sides is known as regular decagon as shown in Fig. 3.18.

(vii) **Regular Undecagon:** A polygon having eleven equal sides is known as regular undecagon as shown in Fig. 3.19.

(viii) **Regular Duodecagon:** A polygon having twelve equal sides is known as regular duodecagon as shown in Fig. 3.20.

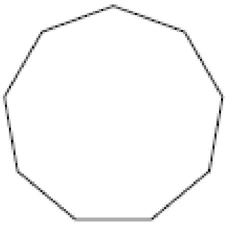


Fig. 3.17

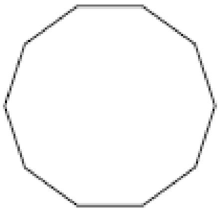


Fig. 3.18

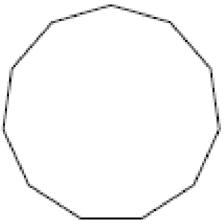


Fig. 3.19

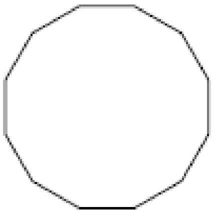


Fig. 3.20

3.4 BISECTING A STRAIGHT LINE

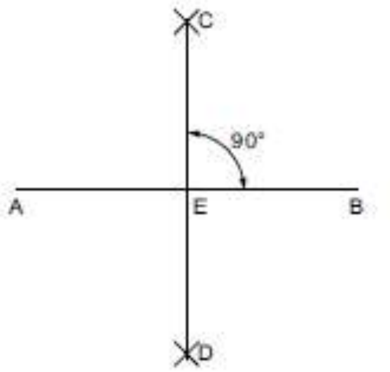
Problem 1. To bisect a given straight line.

Solution. (*i*) Let *AB* be a given straight line. With centre *A* and radius greater than half *AB*, draw arcs on both the sides of *AB*.

(*ii*) With centre *B* and the same radius, draw arcs on both the sides of *AB*, intersecting the previous arcs at *C* and *D*.

(*iii*) Draw a line joining *C* and *D* intersecting *AB* at *E*.

(*iv*) Then *CD* bisects the line *AB* at *E*. Thus, *AE* = *EB* = 0.5 *AB* as shown in Fig. 3.21.



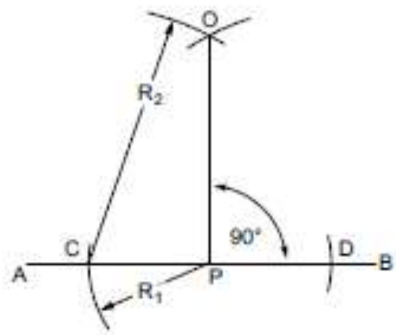
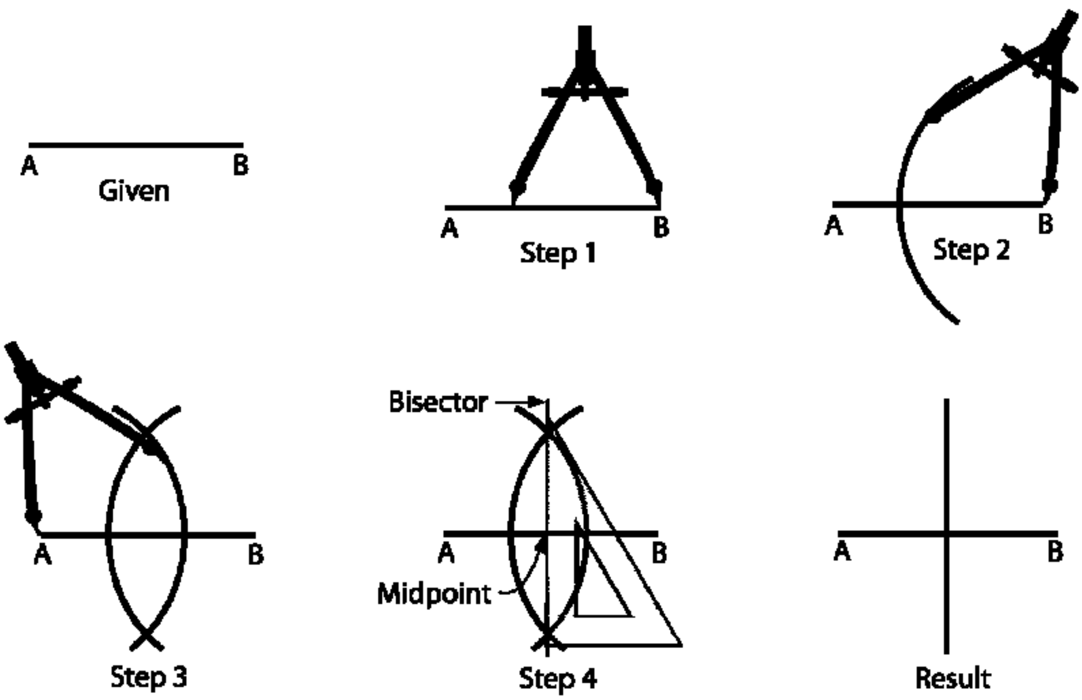


Fig. 3.21 **Fig. 3.22**

Bisecting a line



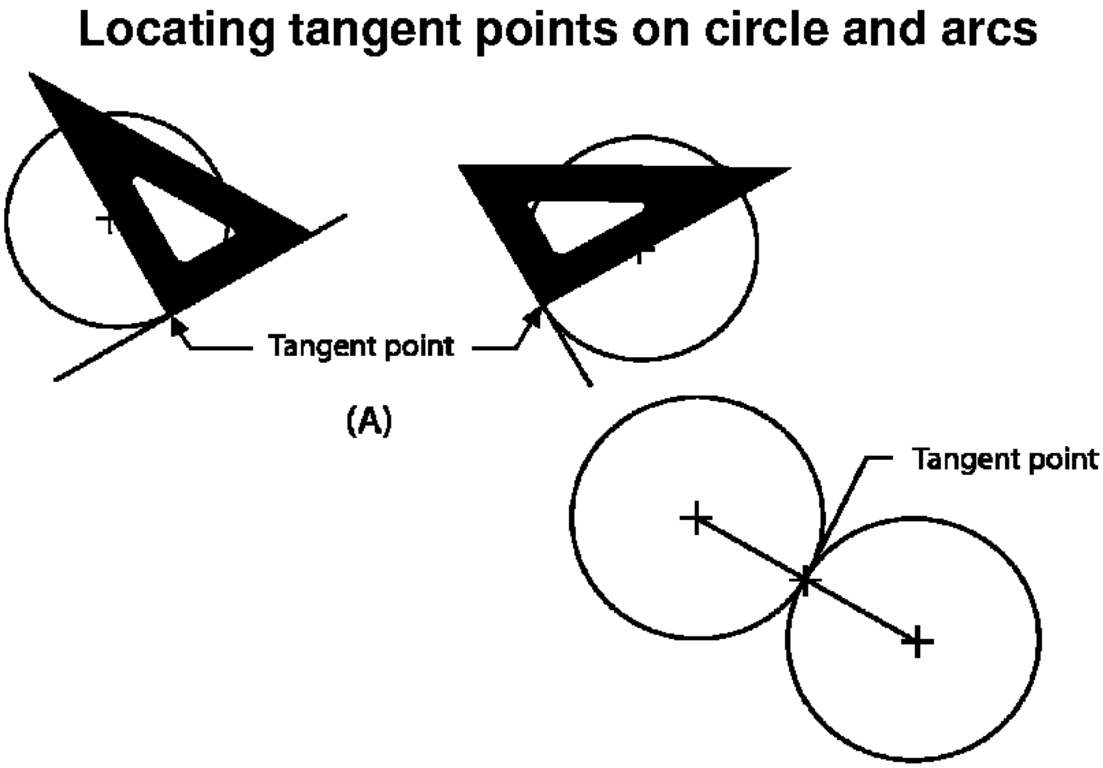
Problem 2. To draw a perpendicular to given line from a given point.

Solution. (*i*) Let *AB* be a given line and *P* is the point in it.

(*ii*) With *P* as a centre, draw an arc cutting *AB* at *C* and *D* at any convenient radius *R*1.

(*iii*) With any radius *R*2 greater than *R*1 and centres *C* and *D*, draw an arcs intersecting each other at *O* as shown in Fig. 3.22.

(*iv*) Draw a line joining *P* and *O*. Then the line *PO* is the required perpendicular.



3.5 TO DIVIDE A LINE INTO ANY NUMBER OF EQUAL PARTS

Problem 3. Divide a given line of 90 mm length into seven equal parts.

Solution. (*i*) Draw a line *AB* of 90 mm length, and divide into seven equal parts.

(*ii*) Draw a line *AC* of any length inclined at any convenient angle to *AB*, i.e., 30° to 40°C.

(*iii*) From *A* and along *AC*, cut-off seven equal divisions of any convenient length by the help of divider.

(*iv*) Draw a line joining *B* and 7 and with the help of minidrafter draw a line through 1, 2, 3, etc. parallel to *B*7 intersecting *AB* at points 1′, 2′, 3′ ... etc. into seven equal parts as shown in Fig. 3.23.

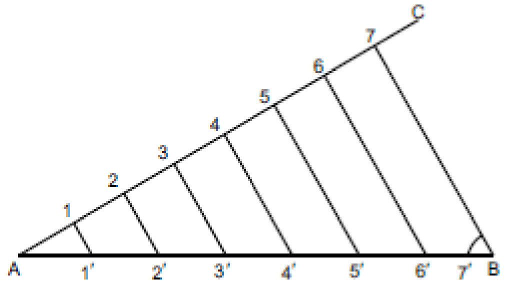


Fig. 3.23

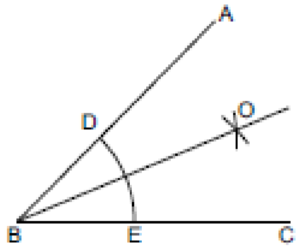


Fig. 3.24

3.6 TO BISECT AN ANGLE BETWEEN TWO GIVEN LINES

Problem 4. To bisect a given angle between two lines.

Solution. Let *ABC* be the given angle, to be bisected.

(*i*) With *B* as a centre and with any convenient radius, draw an arc cutting *AB* and *BC* at *D* and *E* as shown in Fig. 4.24.

(*ii*) With centres *D* and *E*, taking any convenient radius, draw arcs intersecting each other at *O*.

(*iii*) Draw a line joining *B* and *O*, then *BO* is the bisector of the angle *ABC* as shown in Fig. 3.24.

3.7 TO DRAW AN ARCS TANGENTIAL TO LINES

Problem 5. To draw an arc of given radius touching two straight lines perpendicular to each other. Taking *R* is the radius of arc.

Solution. Let *AB* and *AC* be the given lines and *R* is the given radius (*i*) With centre *A* and given radius *R*, draw arcs to cut *AB* at *M* and *AC* at *N*.

(*ii*) With *M* and *N* as a centres and radius *R*, draw arcs intersecting each other at *O*.

(*iii*) With *O* as a centre and the same radius *R*, draw the required tangent arc as shown in Fig. 3.25.

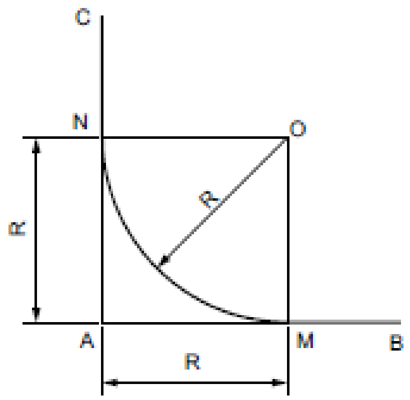


Fig. 3.25

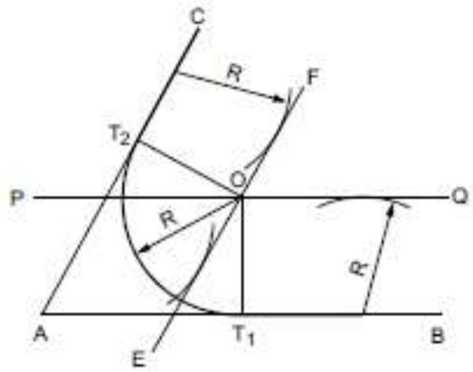


Fig. 3.26

Problem 6. To draw an arc touching two given straight lines which make an acute angle between them. Taking radius of arc is equal to *R*.

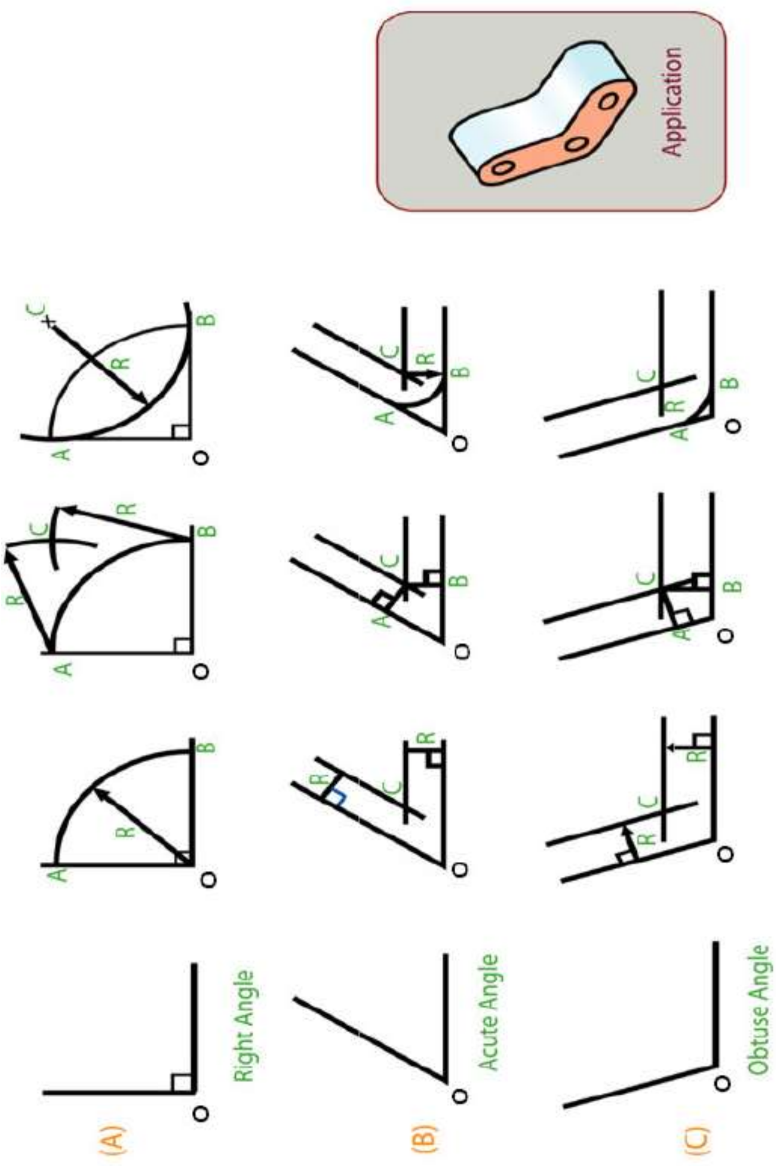
Solution. Let *AB* and *AC* be the given lines and *R* is the given centre.

(*i*) Draw a line *PQ* parallel to and at a distance equal to *R* from *AB*.

(*ii*) Draw a line *EF* parallel to and at a distance of *R* from *AC*, meeting *PQ* at *O*.

(*iii*) With *O* as a centre and *R* as a radius draw the tangent arc as shown in Fig. 3.26

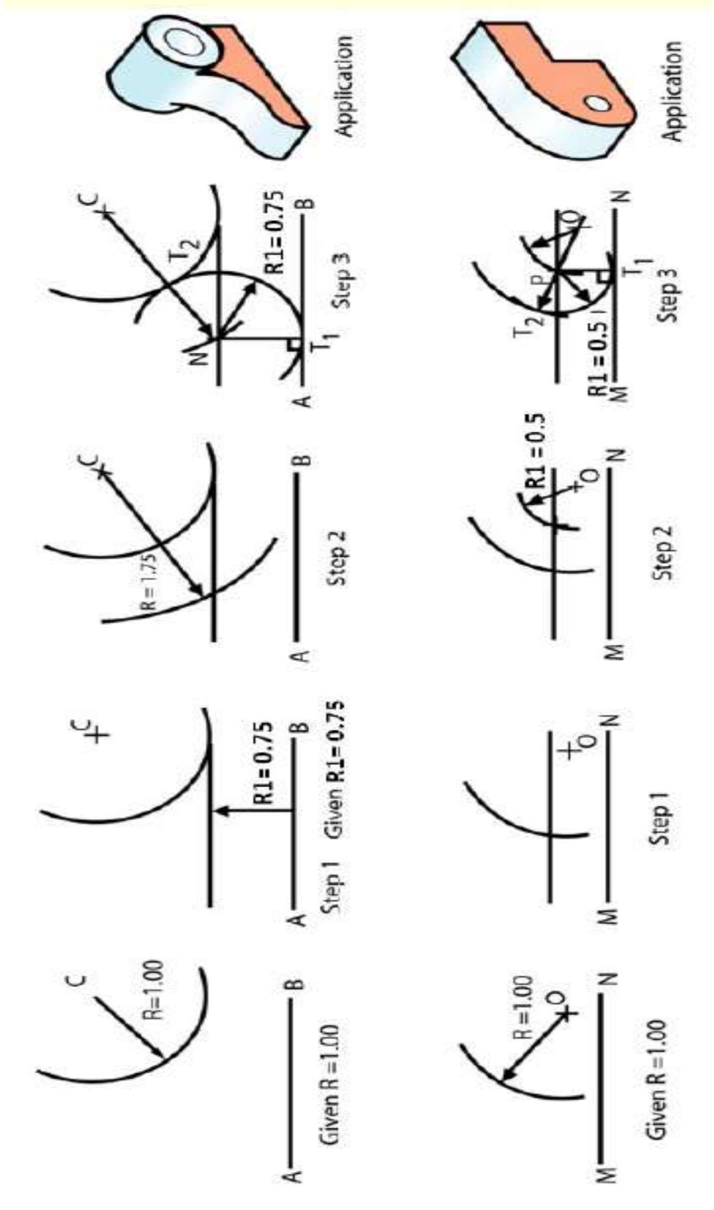
3.7.1 Drawing an arc, tangent to two lines



3.7.2 Drawing an arc, tangent to a line and an arc

(a) That do not intersect

(b) That intersect



3.8 CONSTRUCTION OF REGULAR PENTAGONS

Problem 7. Inscribe a pentagon in a circle of a given diameter.

Solution. (*i*) With centre *O*, draw a circle of a given diameter.

(*ii*) Draw diameters *AB* and *CD* perpendicular to each other.

(*iii*) Bisect *OB* and mark the point *N*. With centre *N* and radius *NC* draw an arc *CM*.

(*iv*)With centre *C* and radius *CM*, draw an arc cutting the circle in*S*and *R*.

(*v*) With centres *S* and *R* and the same radius, draw arcs cutting the circle in *P* and *Q* respectively.

(*vi*) Now join the intersecting points with each other to obtain the required pentagon as shown in Fig. 3.27.

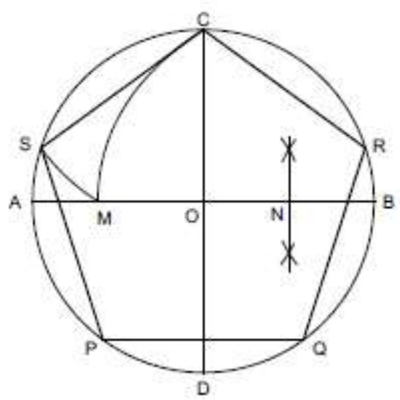


Fig. 3.27

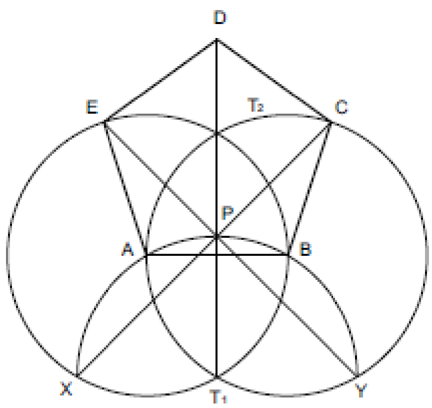


Fig. 3.28

Problem 8. Construct a regular pentagon, given the length of side *AB*.

Ist Method

Solution. (*i*) Draw a line *AB* of a given length.

(*ii*) Draw the circles with centres *A* and *B* with a radius equal to *AB*. The circles intersect each other at *T*1 and *T*2.

(*iii*) With centre *T*1 and radius equal to *AB*, draw a circle intersecting two circles at *X* and *Y*. The circle also intersects perpendicular bisector of *AB* at *P*. (*iv*) Joint *XP* and extend it to get a point *C* on the circle. Similarly, join *PY* and extend it at a point *E*.

(*v*) Join *AE*, *BC* with centre *E* and radius equal to *AB* draw an arc intersecting at *D*. Similarly, with centre *C* and radius *AB* draw another arc intersecting each other at *D*, and get the required pentagon *ABCDE* as shown in Fig. 3.28.

Second method

(*i*) Draw a line *AB* of a given length.

(*ii*) Draw an isosceles triangle *OAB* with *AB* as base and base angles of 54º. (*iii*) With *O* as centre and *OA* as a radius, draw a circle passing through *A* and *B*.

(*iv*) With *AB* as radius, intersect the circle successively at the points *C*, *D* and *E*. Join *BC*, *CD*, *DE* and *EA* to get the required pentagon *ABCDE* as shown in Fig. 3.29.

[*Note:* For the pentagon, angle subtended at the centre of the isosceles triangle = 360º/5= 72º. Hence, the base angle = (180º – 72º)/2 = 54º.

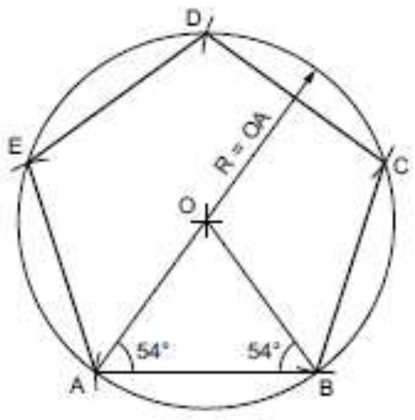


Fig.3.29

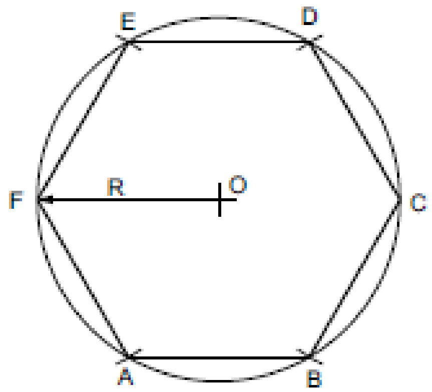


Fig.3.30

- **General method of drawing any polygon**

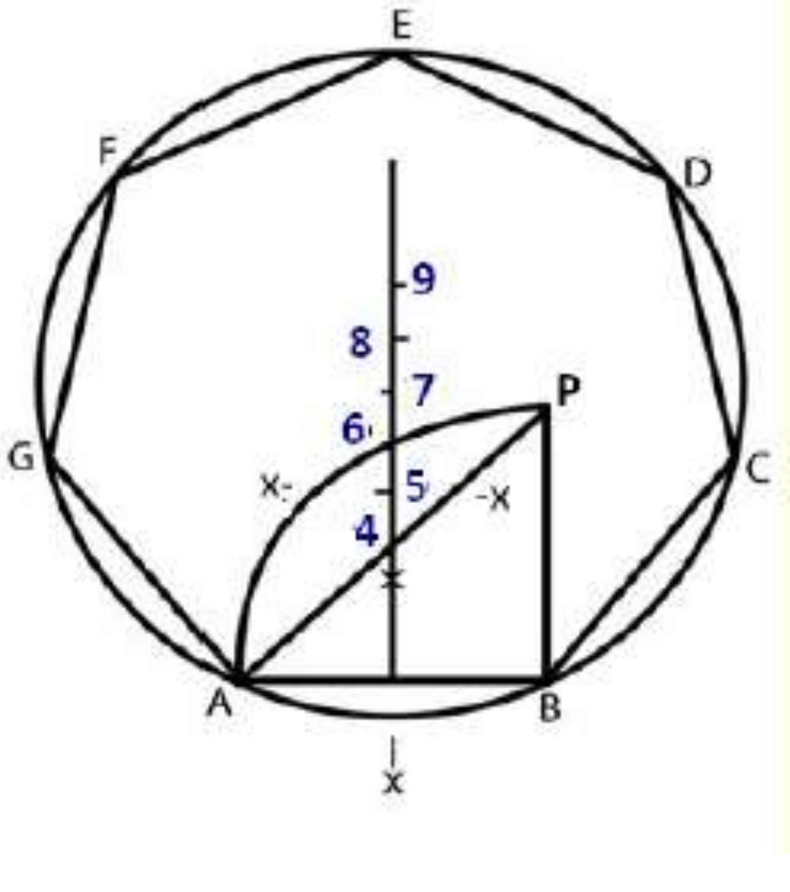
Draw AB = given length of polygon At B, Draw BP perpendicular & = AB

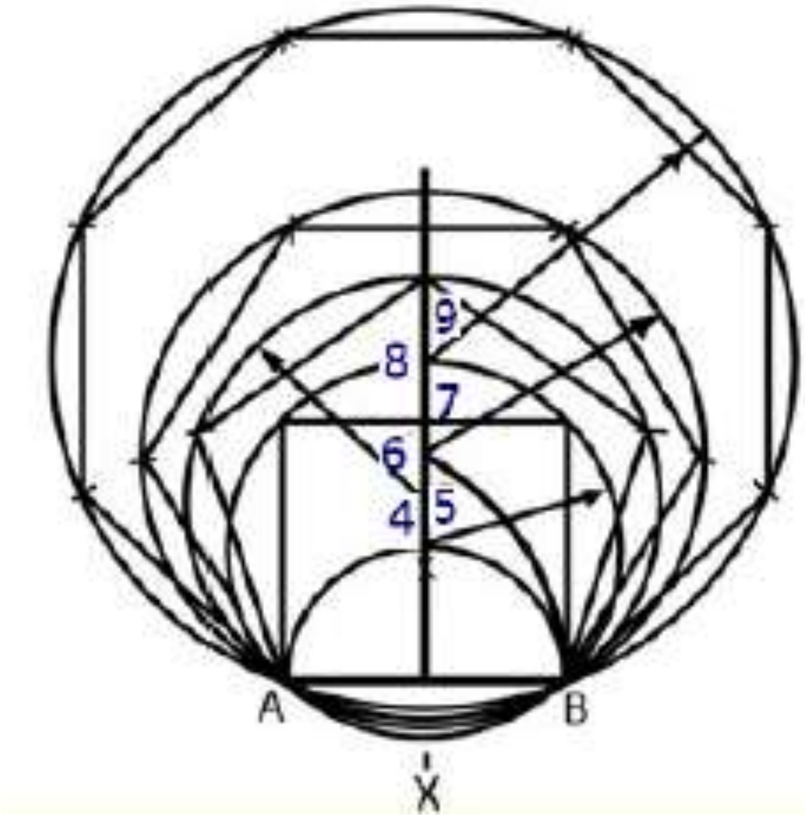
Draw Straight line AP With center B and radius AB, draw arc AP.

The perpendicular bisector of AB meets st.line AP and arc AP in 4 and 6 respectively.

Draw circles with centers as 4, 5,&6 and radii as 4B, 5B, & 6B and inscribe a square, pentagon, & hexagon in the respective circles.

Mark point 7, 8, etc with 6-7,7-8,etc. = 4-5 to get the centers of circles of heptagon and octagon, etc.





3.9 CONSTRUCTION OF REGULAR HEXAGON

Problem 9. Construct a regular hexagon, given length of one side *R*.

Solution. Ist Method (*i*) With centre *O* and the radius *R* draw a circle. Mark the radius *OF*.

(*ii*) With *F* as a centre and given side length as radius draw an arc to intersect the circle at *E*.

(*iii*) Similarly, mark the points *A*, *B*, *C* and *D* respectively.

(*iv*) Join the above division points in proper sequence, to obtained the required hexagon *ABCDEF* as shown in Fig. 3.30.

Second method

(*i*) Draw a line *AB* equal to the given length.

(*ii*) From *A*, draw a line *A*1 and *A*2 making 60º and 120º respectively with *AB* by help of mini drafter.

(*iii*) From *B*, draw a line *B*3 and *B*4 making 60º and 120º respectively with *AB* by the help of mini drafter.

(*iv*) From *O* the point of intersection of *A*1 and *B*3, draw a line parallel to *AB* and intersecting *A*2 at *F* and *B*4 at *C*.

(*v*) From *F*, draw a line parallel to *BC* and intersecting *B*3 at *E*.

(*vi*) From *C*, draw a line parallel to *AF* and intersecting *A*1 at *D*. Join the above division marks to obtain the required hexagon *ABCDEF* as shown in Fig. 3.31.

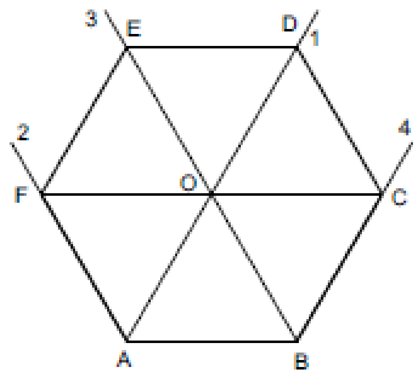


Fig.3.32

3.10 CONSTRUCTION OF REGULAR OCTAGON

Problem 10. Draw a regular octagon about a given circle.

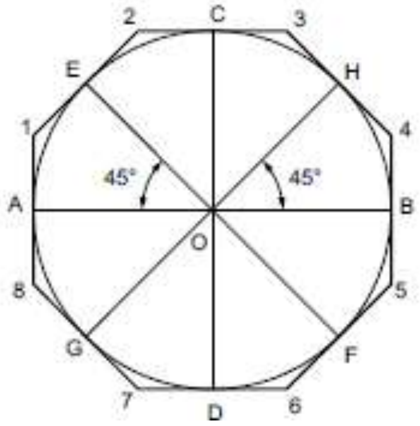
Ist method:

Solution: (*i*) With centre *O*, draw a circle of a given radius.

(*ii*) Draw the diameters *AB* and *CD* at right angles to each other.

(*iii*) Draw the diameters *EF* and *GH* inclined at an angle of 45º to *AB* or *CD*.

(*iv*) Draw eight tangents to the circle at the ends of the diameters *A*, *B*, *C*, ... *H* etc. respectively to intersect one another at the point 1, 2, 3 ... 8 etc. to complete the required octagon as shown in Fig. 3.32.



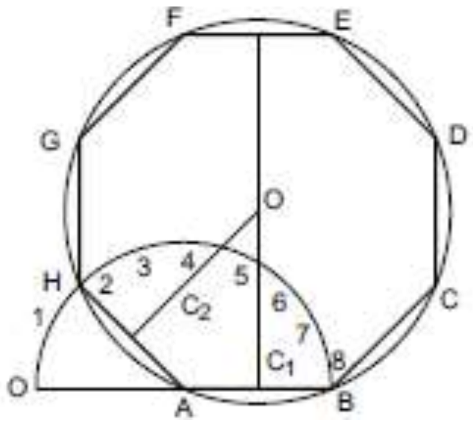


Fig.3.32 Fig.3.33

Second method

(*i*) Draw a line *AB* of a given length.

(*ii*) With centre *A*, draw a semi circle. Divide the semi circle into eight equal parts and mark 0, 1, 2, 3, 4, 5,6, 7 and 8 respectively and join *A*2.

(*iii*) Bisect *AB* and *A*2 at *C*1 and *C*2. Draw lines through *C*1 and *C*2 to intersect at *O*.

(*iv*) Draw a circle with centre *O* and radius equal to *OA* or *OB* or *O*2.

(*v*) Mark the remaining six sides equal to *AB* to complete the required octagon as shown in Fig. 3.33.

- INSCRIBE A REGULAR POLYGON OF ANY NUMBER OF SIDES

(say n = 5), in a circle

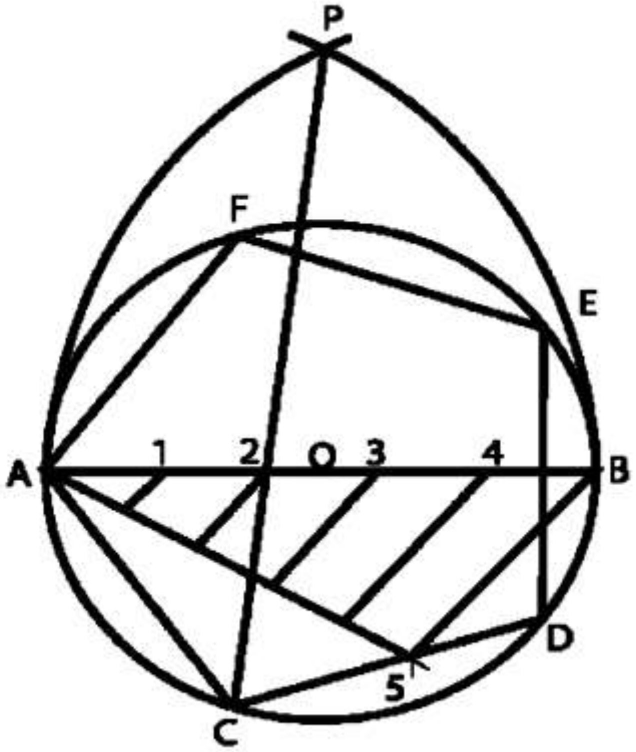
Draw the circle with diameterAB and Divide AB in to “n” equal parts

Number them.

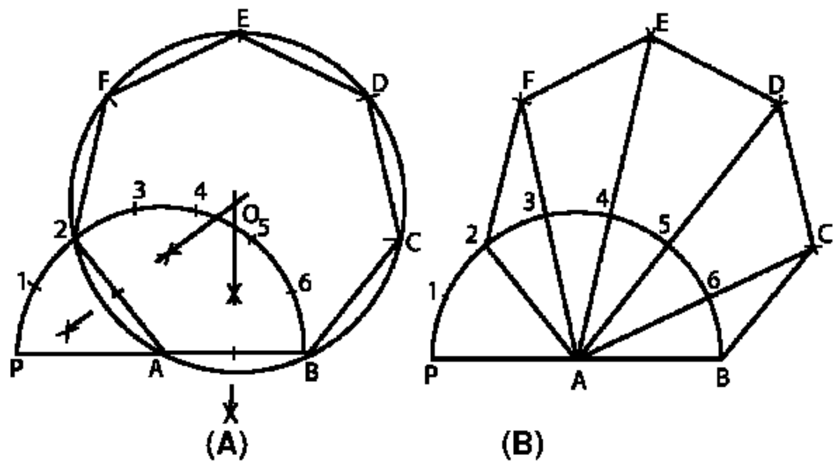
With center A & B and radius AB, draw arcs to intersect at P.

Draw line P2 and produce it to meet the circle at C.

AC is the length of the side of the polygon.



Construction of Regular Polygon of given length AB



Draw a line of length AB. With A as centre and radius AB, draw a semicircle.

With the divider, divide the semicircle into the number of sides of the polygon.

Draw a line joining A with the second division-point 2.

3.11 CONIC SECTION

3.11.1 Cone

A cone is a surface generated by the rotation of a straight line whose one end is in contact with a fixed point while the other end is in contact with a closed curve, not lying in the plane of the curve as shown in Fig. 3.34.

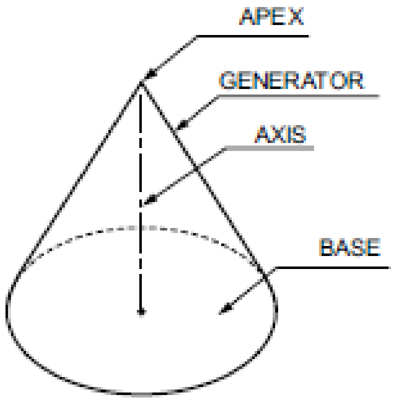


Fig. 3.34

(*i*) ***Vertex or Apex:*** The point of intersection of the axis and the generator is known as vertex or apex of the cone.

(*ii*) ***Generator:*** The revolving line which generates the surface of the cone is known as the generator as shown in Fig. 3.34.

(*iii*) ***Axis:*** The fixed line, about which the other line revolves, is known as the axis of the cone.

3.11.2 Conic Sections

The sections obtained by cutting off a right circular cone by section planes at different angles relative to its axis are known as conic sections.

The circle, ellipse, parabola, hyperbola are examples of conic section as shown in Fig. 3.35.

(*i*) ***Circle:*** When a cutting plane *AA* is perpendicular to the axis and cuts all the generators, the section obtained is known as circle as shown in Fig. 3.36.

(*ii*) ***Ellipse:*** When a cutting plane *BB* is inclined to the axis of the cone and cuts all the generators on one side of the apex, the section obtained is known as an ellipse as shown in Fig. 3.37.

(*iii*) ***Parabola:*** When a cutting plane *CC* is inclined to the axis of the cone and parallel to one of the generators, the section obtained is known parabola as shown in Fig. 3.38.

(*iv*) ***Hyperbola:*** When the cutting plane *DD* makes a smaller angle with the axis then that of the angle made by the generator of the cone, the section obtained is known as hyperbola as shown in Fig. 3.39.

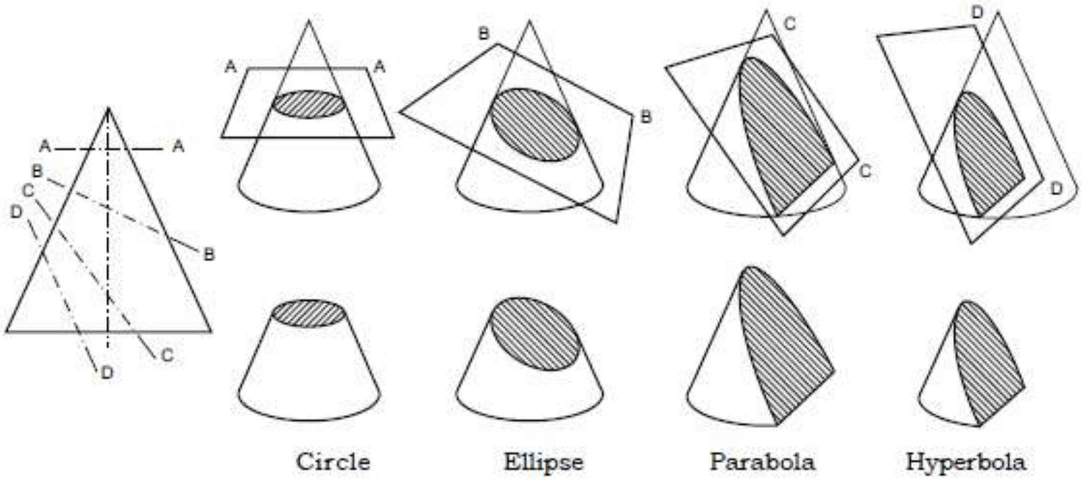


Fig.3.35 **Fig.3.36** **Fig.3.37** **Fig.3.38**

3.12 SCALES

3.12.1 INTRODUCTION

In most of the cases, drawing of big object can not be drawn in full size scale because the object may be too big. Similarly, the drawings of small object also cannot be prepared in full size. Hence it is necessary to draw them with suitable scale as per the drawing sheet. Therefore, scale may be defined as, "ratio of the linear dimension of an element of an object as represented in the original drawing to the real linear dimension of the same element of the object itself". In other words, the proportion by which a dimension is either reduced or increased in the drawing is known as scale. This scale is also known as draftsman scale.

5.2 SIZE OF SCALE

Scale may be represented by the following ways:

5.2.1 Full scale

A scale with the ratio of 1:1 is said to be full scale.

5.2.2 Enlarged scale

A scale where the ratio is larger than 1:1 is said to be larger as its ratio increases, e.g., watches, electronic devices etc.

5.2.3 Reducing scale

A scale where the ratio is smaller than 1:1 it is said to be smaller as its ratio decreases e.g. maps, building, structure etc.

The recommended scales for the use of technical drawings as per SP : 46–1988 are specified in Table 3.1.

Table 3.1

Category	Recommended Scales
Full scale	1 : 1
Enlarge scales	50 : 1 20 : 1 10 : 1 5 : 1 2 : 1
Reducing scales	1 : 2 1 : 5 1 : 10 1 : 20 1 : 50 1 : 100 1 : 500 1 : 1000 1 : 2000 1 : 5000 1 : 10000

5.3 Construction of an arc tangent of given radius to two given arcs:

• Given - Arcs of radii M and N. Draw an arc of radius AB units which is tangent to both the given arcs. Centers of the given arcs are inside the required tangent arc.

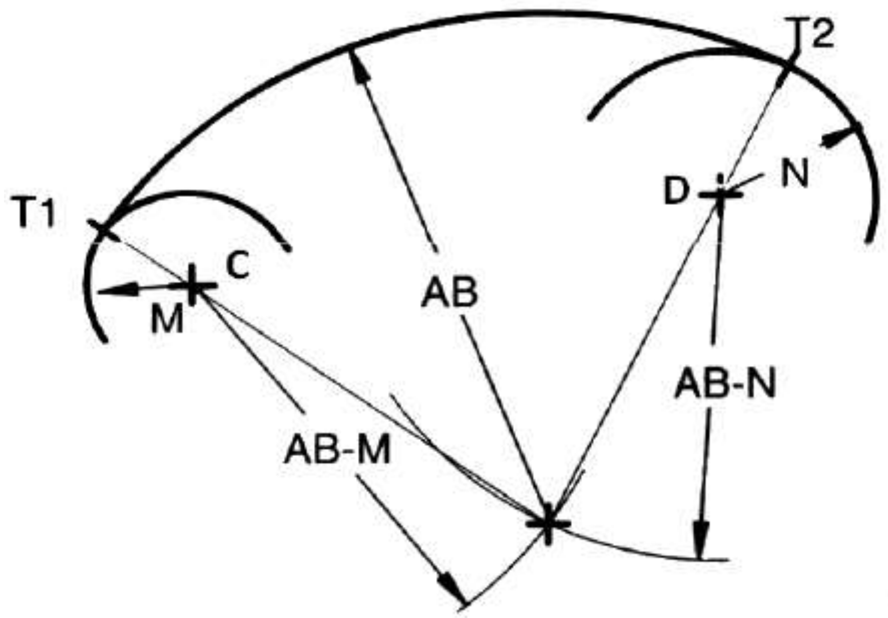
Steps:

From centers C and D of the given arcs, draw construction arcs of radii (AB – M) and (AB - N), respectively. With the intersection point as the center, draw an arc of radius

AB.

This arc will be tangent to the two given arcs.

Locate the tangent points T1 and T2.



5.5 Construction of line tangents to two circles (<u>Open belt</u>)

Given: Circles of radii R1 and R with centers O and P, respectively.

Steps:

With P as center and a radius equal to **(R-R1)** draw an arc.

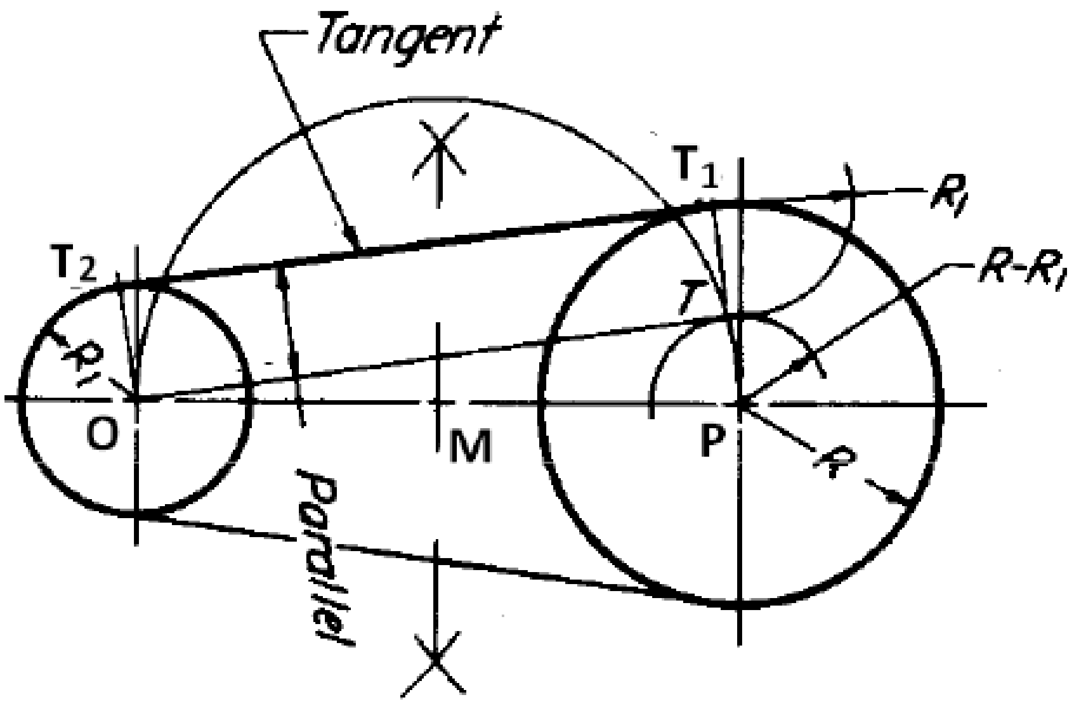
Locate the midpoint of **OP** as perpendicular bisector of **OP** as "**M**".

With **M** as centre and **Mo** as radius draw a semicircle.

Locate the intersection point **T** between the semicircle and the circle with radius (**R-R1**). draw a line **PT** and extend it to locate **T1**.

Draw **OT2** parallel to **PT1**.

The line **T1** to **T2** is the required tangent.



5.6 Construction of line tangents to two circles (<u>crossed belt</u>)

Given: Two circles of radii R1 and R with centers O and P, respectively.

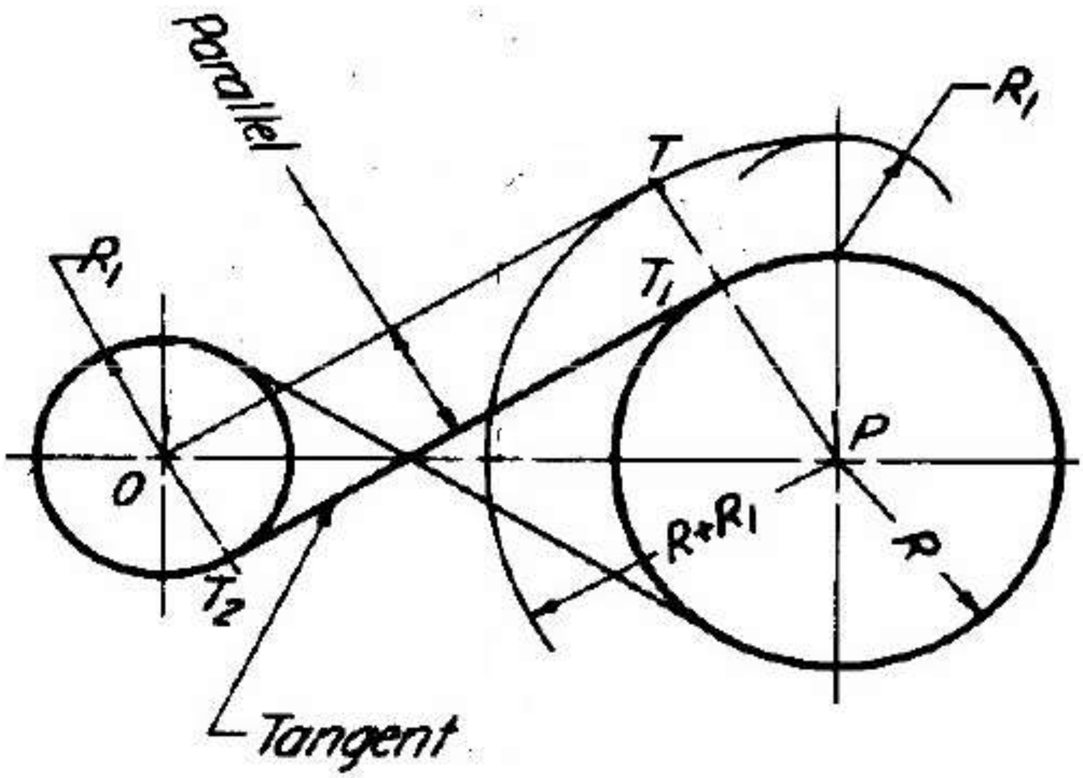
Steps:

Using **P** as a center and a radius equal to **(R+ R1)** draw an arc.

Through **O** draw a tangent to this arc. Draw a line PT cutting the circle at T1

Through O draw a line OT2 parallel to PT1.

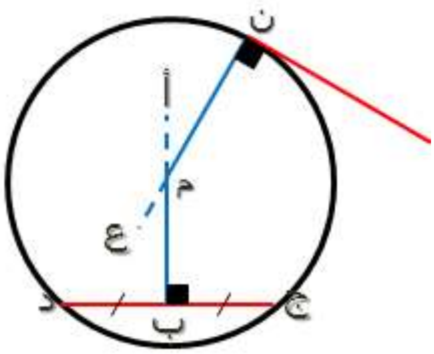
The line **T1T2** is the required tangent.



5.7 THE INVERTED CURVE

Before we start explaining the inverted curve, I want to review two important rules for the circle.

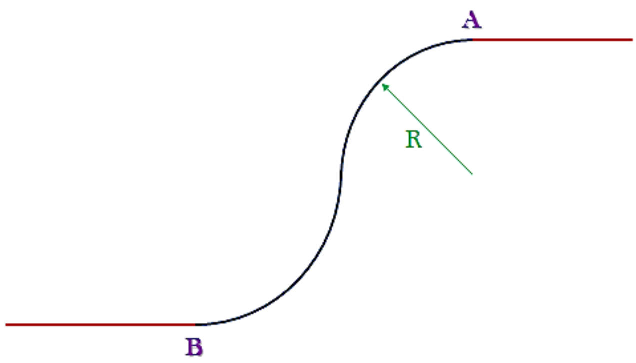
• Perpendicular to the tangent "N" from the point of contact "N" ... passing through the center "M" I mean, which line is drawn perpendicular to the tangent in the circle ... it will pass through the center of this circle That is, any chord in the circle "C D" ... the vertical equilibrium on it is "AB" ... also passes through the center "M"



To draw the inverted curve, the following must be specified:

1. The starting point and the end point. "of course, the two tangents remain parallel to them."

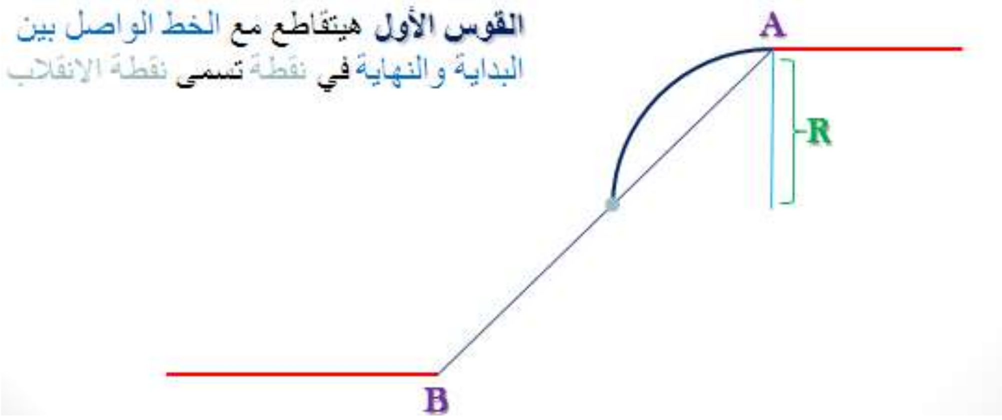
2. The inflection point or the radius of one of the arcs R1 or R2

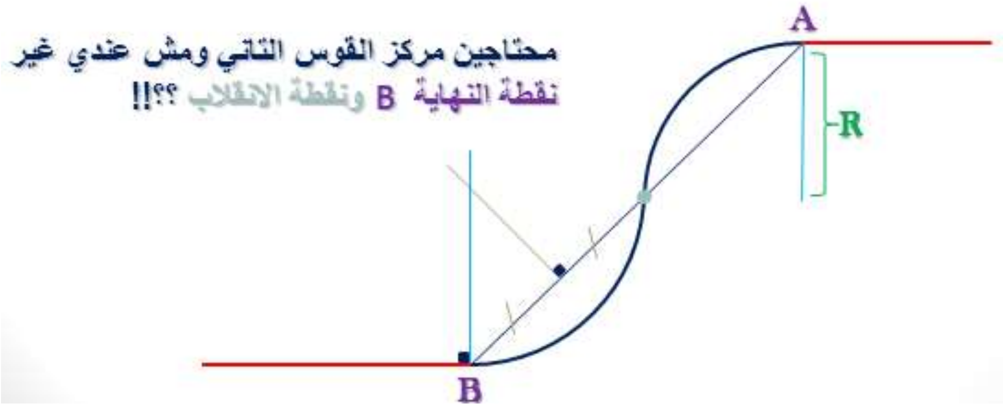


Steps of drawing an inverse curve:

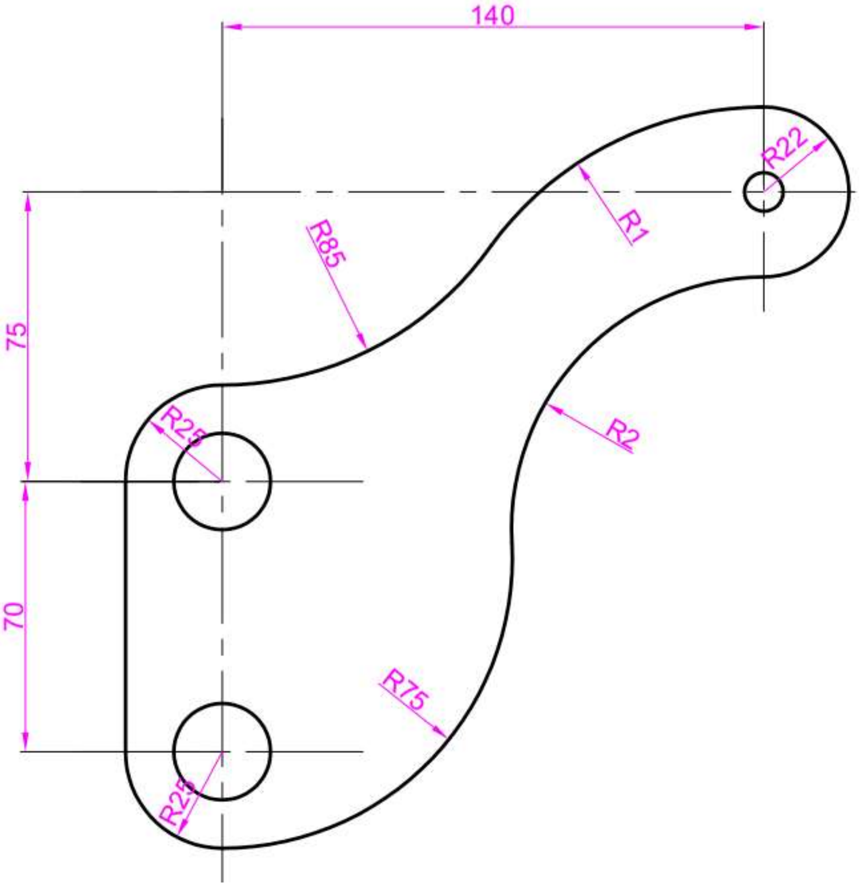
1- Connect the starting point A to the ending point B.

2- We start from the arc whose radius is known, the radius R .. at the starting point in it is a tangent ... (We have agreed that the perpendicular to the tangent from the point of contact is hemmed in the center) .. The perpendicular line remains from A to the center. It remains if you measure the radius on it. R .. I scoured the center of the first arc ... and drow it

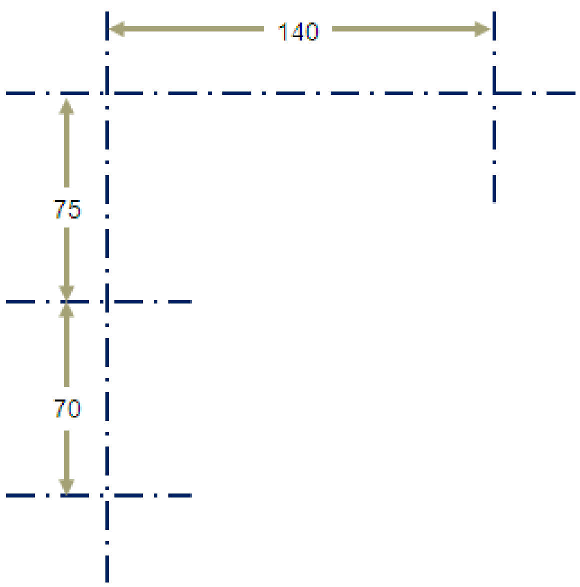


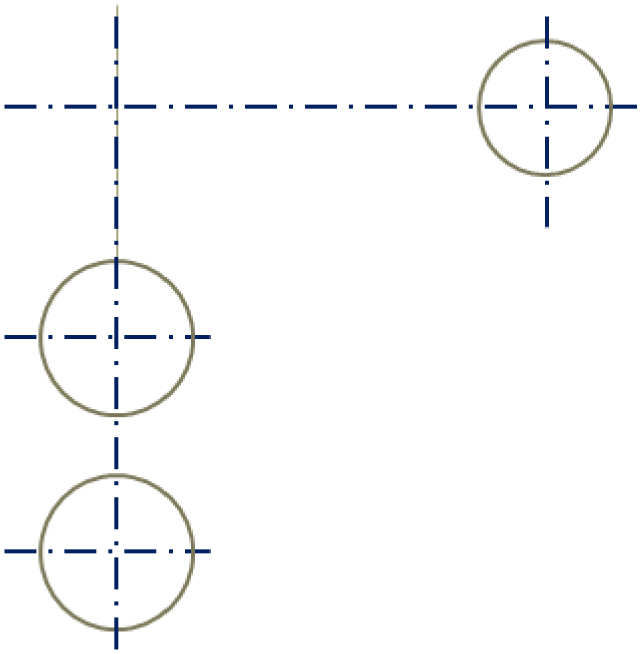


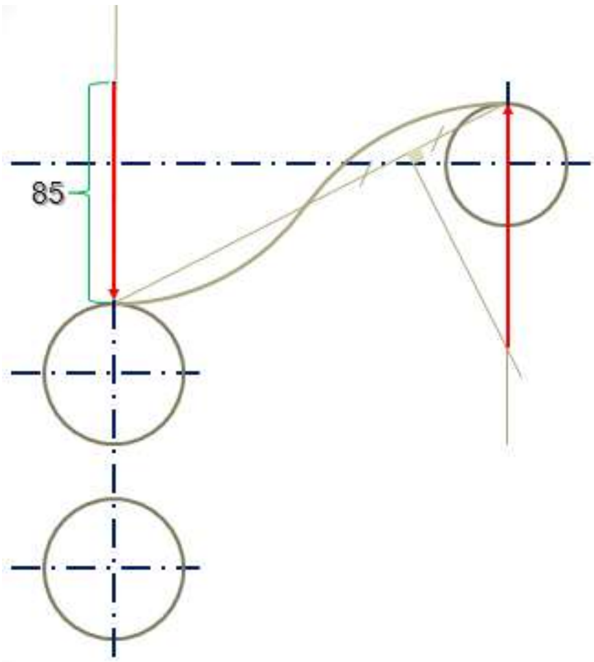
Example:

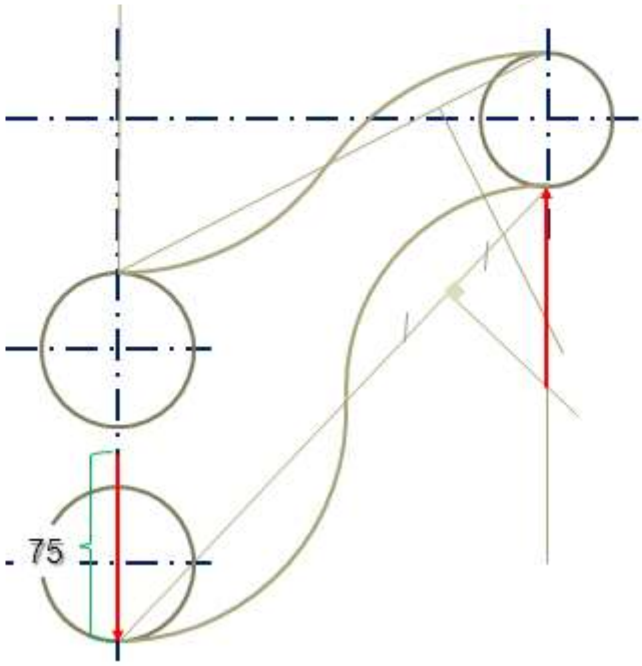


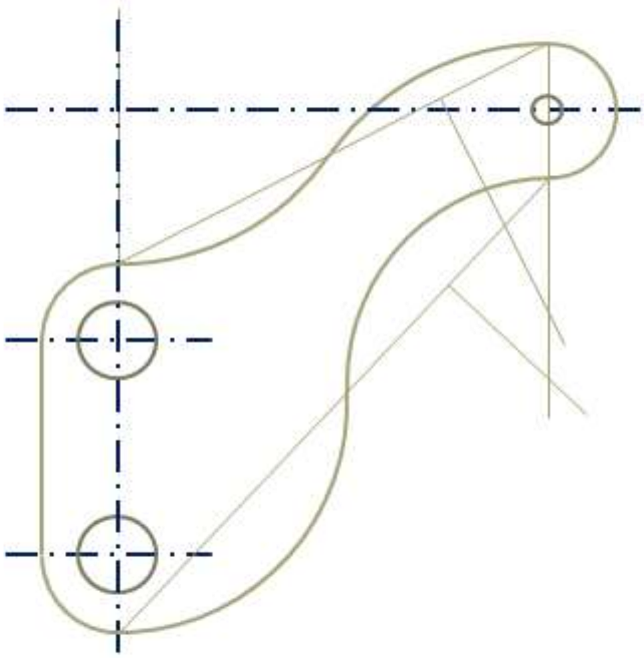
Solution :



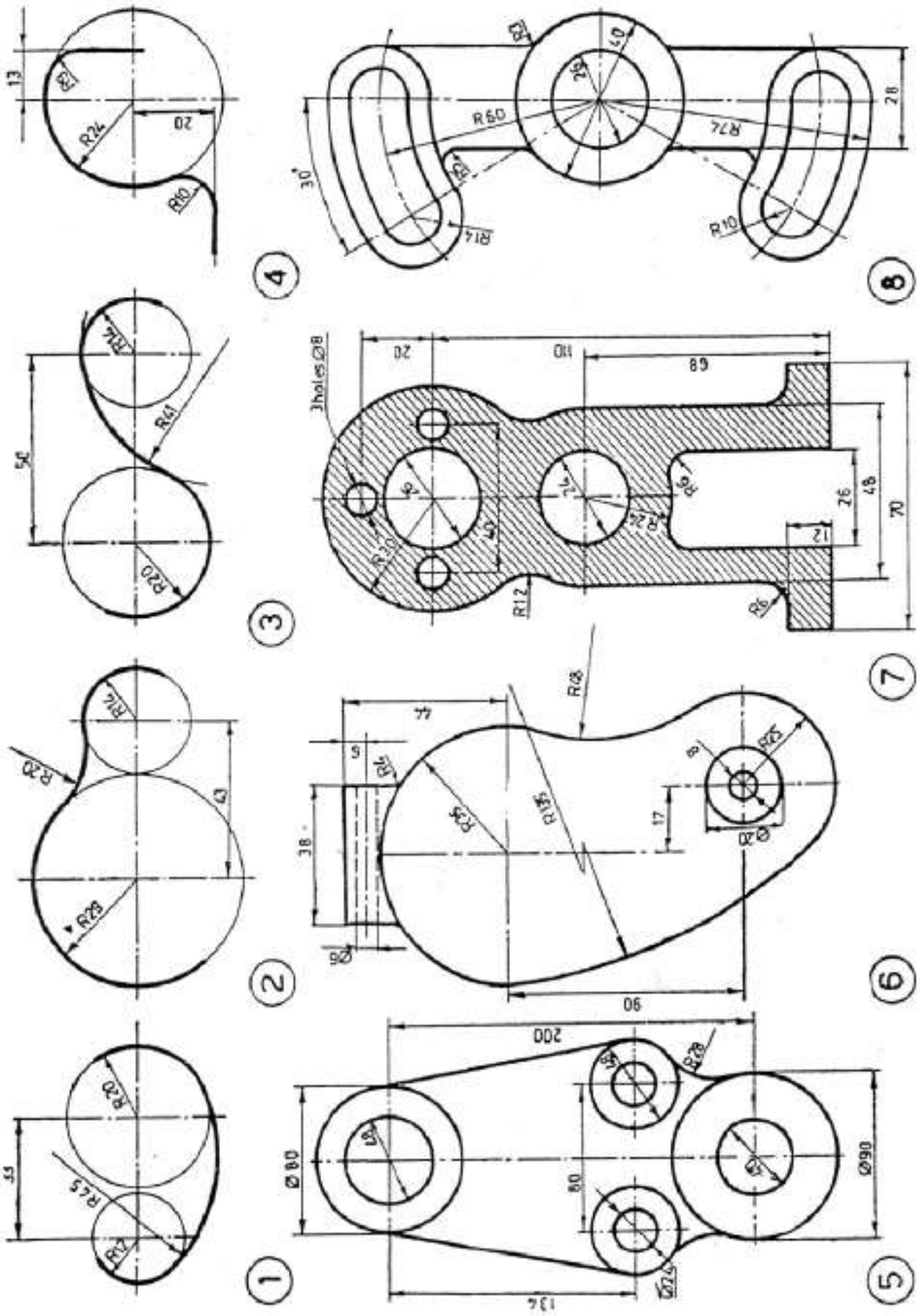


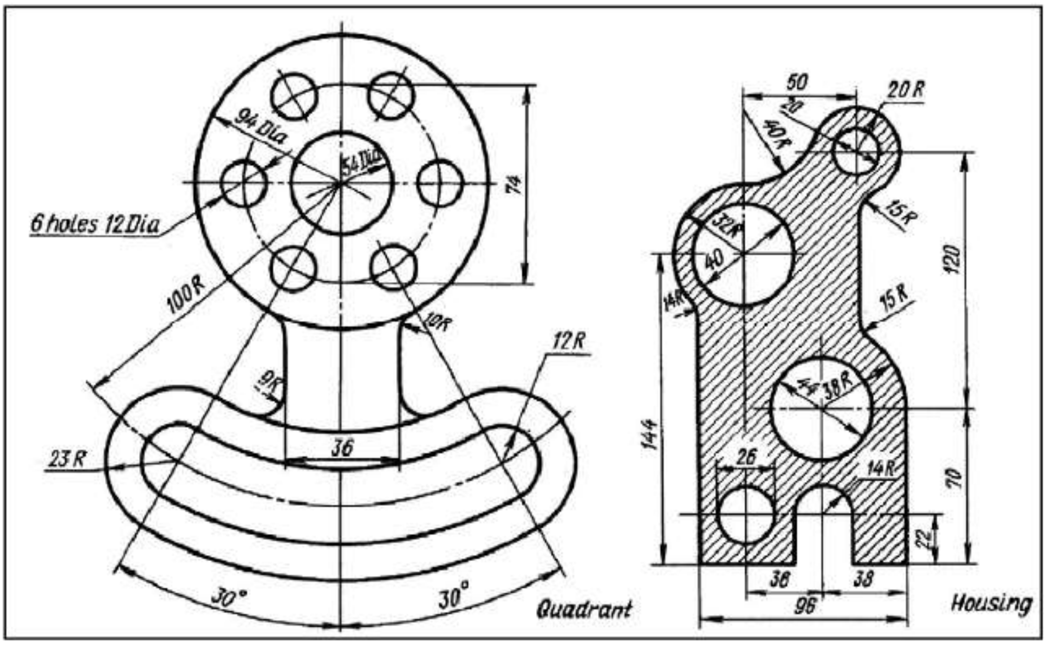

85


75

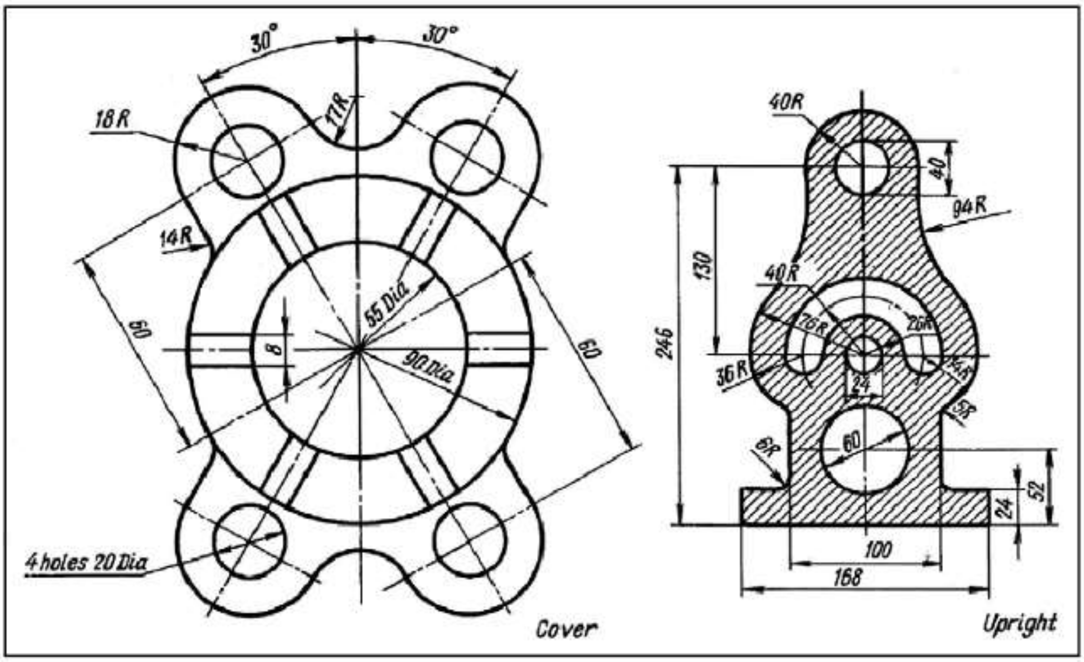


Exercises



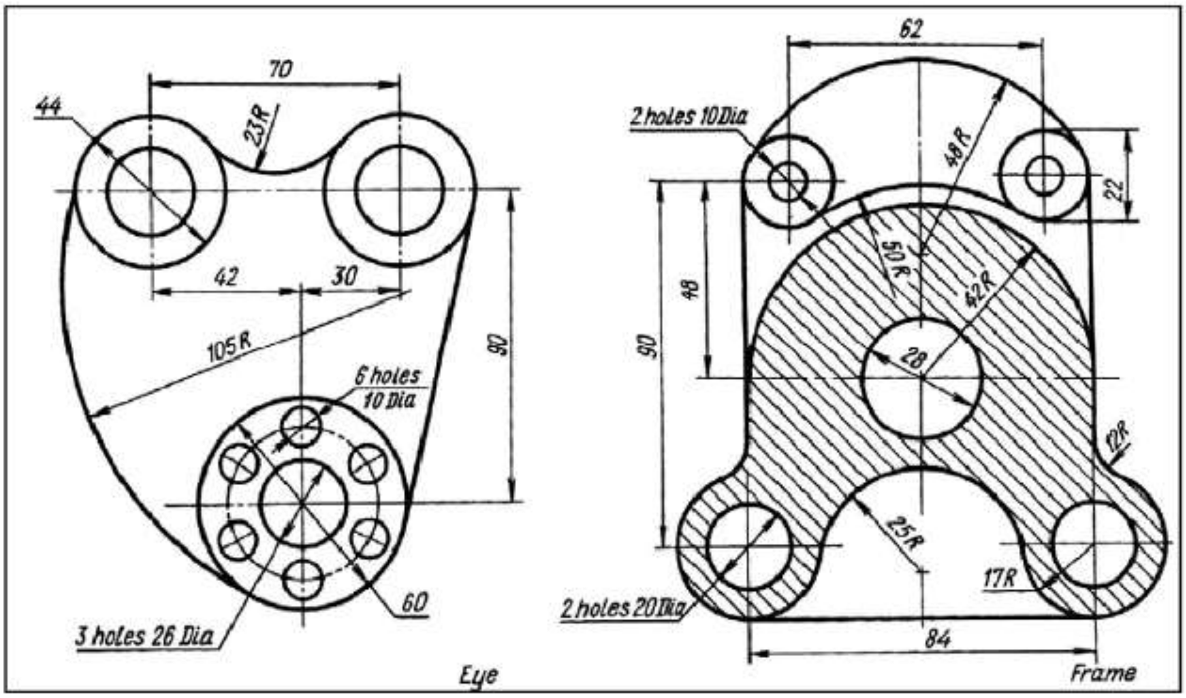


(a)

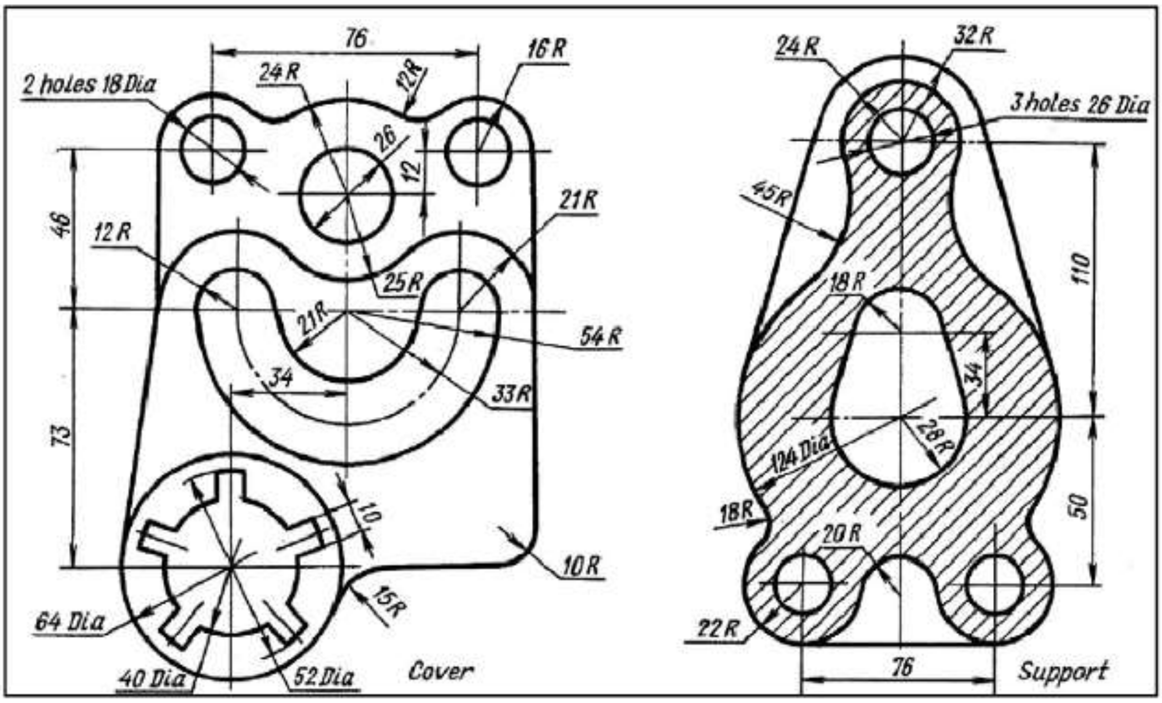


(b)

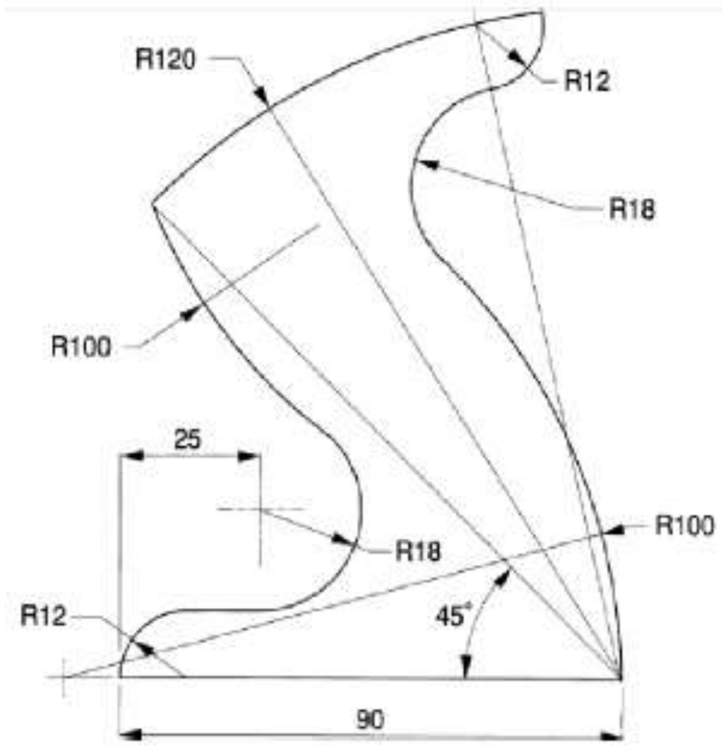
Dr. Eng. Maher Rashad El-Sadaty


70
44
23R
42
30
105R
90
6 holes
10 Dia
60
3 holes 26 Dia
Eye
62
2 holes 10 Dia
48R
22
50R
42R
48
90
28
12R
25R
17R
2 holes 20 Dia
84
Frame

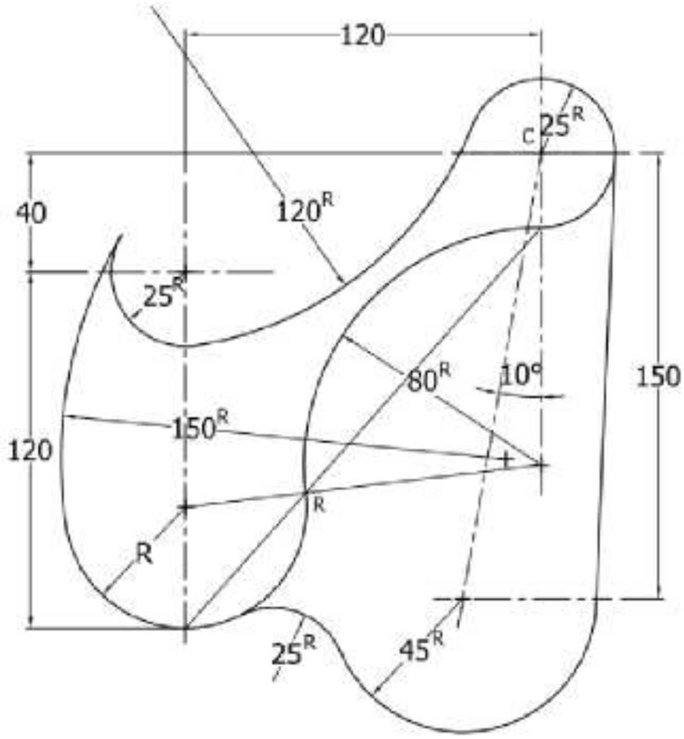
(a)


76
2 holes 18 Dia
24R
12R
16R
26
12
46
12R
21R
25R
21R
54R
34
33R
73
10
10R
15R
64 Dia
40 Dia
52 Dia
Cover
24R
32R
3 holes 26 Dia
45R
18R
110
34
28R
124 Dia
18R
20R
50
22R
76
Support

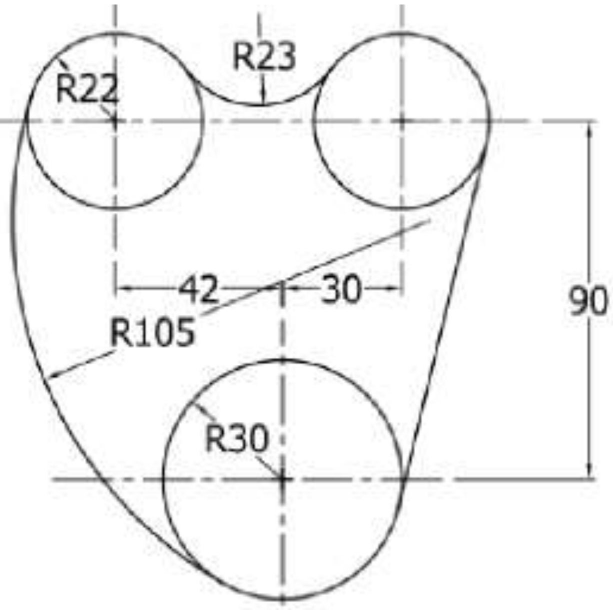
(b)


R120
R12
R18
R100
25
R18
R100
R12
45°
90

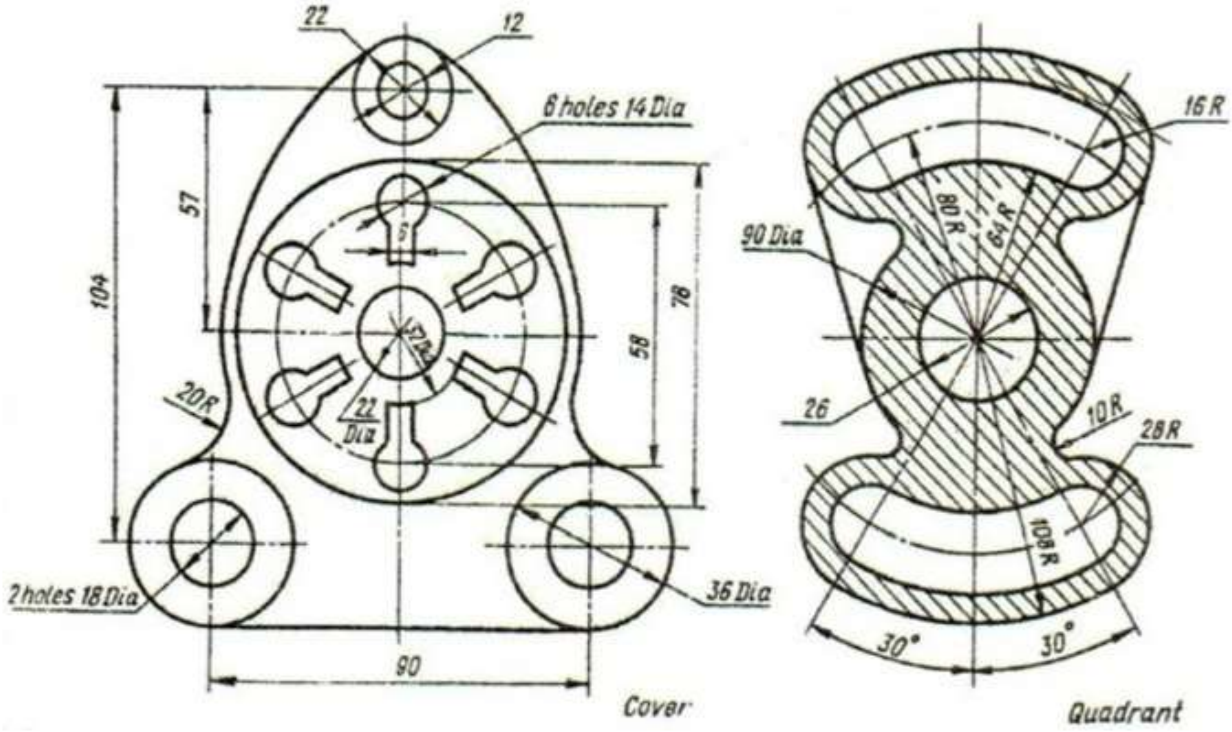
(a)


120
C
25R
40
120R
25R
80R
10°
150
150R
120
R
R
25R
45R

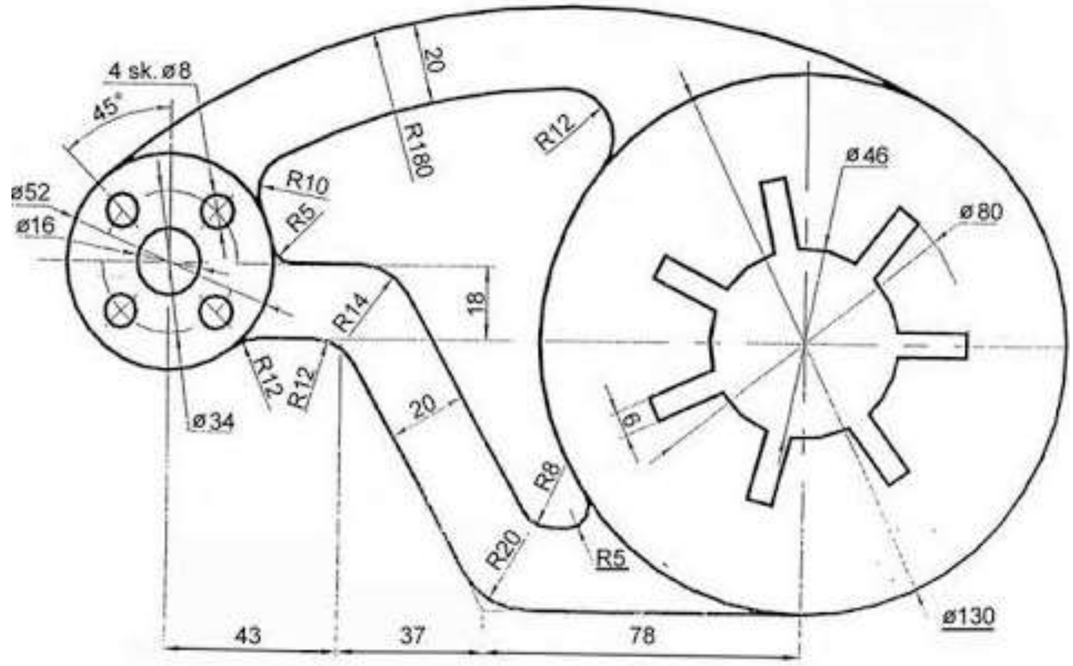
(b)


R22
R23
42
30
90
R105
R30

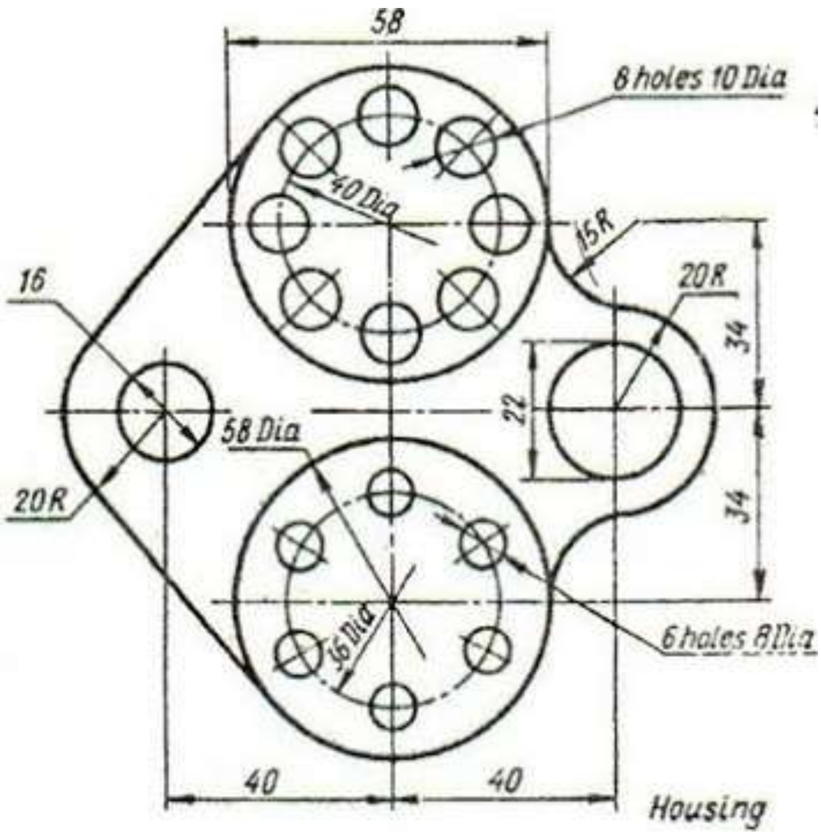
(a)


22
12
6 holes 14 Dia
57
104
78
58
20R
22 Dia
2 holes 18 Dia
36 Dia
90
Cover
16R
90 Dia
80R
64R
26
10R
28R
108R
30°
30°
Quadrant

(b)


4 sk. ø8
45°
ø52
ø16
ø34
R10
R5
R180
20
R12
R14
18
R12
R12
20
R8
R20
R5
ø46
ø80
9
ø130
43
37
78

(a)


58
8 holes 10 Dia
40 Dia
15R
20R
16
34
22
58 Dia
34
20R
36 Dia
6 holes 8 Dia
40
40
Housing

(b)

Chapter 4

Theory of Projection & Projection of Solids (Orthographic Projection)

4.1 INTRODUCTION

In technical drawings, the engineer is always concerned with the task of describing the shape of a solid on a sheet of paper, in order to represent the exact size of an object. It is possible by converting the three-dimensional object into two-dimensional projection which is known as orthographic projection. This chapter deals with orthographic projection and there fundamentals employed in engineering drawing.

4.2 PROJECTION

If straight lines are drawn from various points on the contour of an object to meet a plane, the figure obtained on the plane is called the projection of an object and object is said to be projected on the plane.

Definition: Projection is defined as the image produced by mapping a geometric representation of an object on a plane of projection. Different views of an object are drawn by taking projections on orthogonal planes.

A drawing of an object should consist of five things viz.:

1. Object 2. Projector 3. Plane of projection

4. Observer's eye. 5. Station point

4.3 METHODS OF PROJECTION

The following methods of projection are commonly used in engineering practice.

1. Orthographic projection 2. Isometric projections

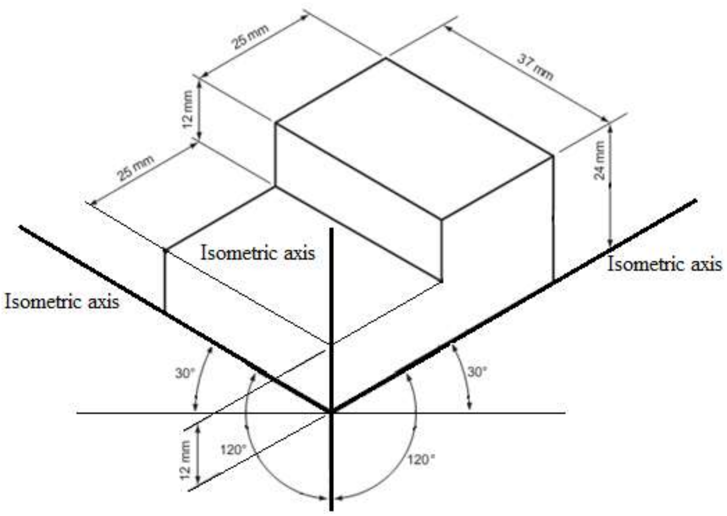
3. Oblique projections 4. Perspective projections

Isometric projections, oblique projections and perspective projections are known as pictorial views.

4.4 ORTHOGRAPHIC PROJECTION

The engineers must represent the object which appears as three-dimensional with dimensions such as width, height and depth on the drawing. Different views of an object are systematically arranged on the drawing to convey the necessary information such as front view, top view, side view etc. This type of drawing is called an *orthographic projection*.

The word orthographic is derived from Greek words, orthos, means “right angle” and graphikus means drawing lines. Basically it is the method of representing an object in two or more views on planes at right angle to each other by extending perpendiculars from the object to the planes. Let’s take an object and imagine that it is placed within the planes of projection which are transparent as shown in Fig. 4.1.



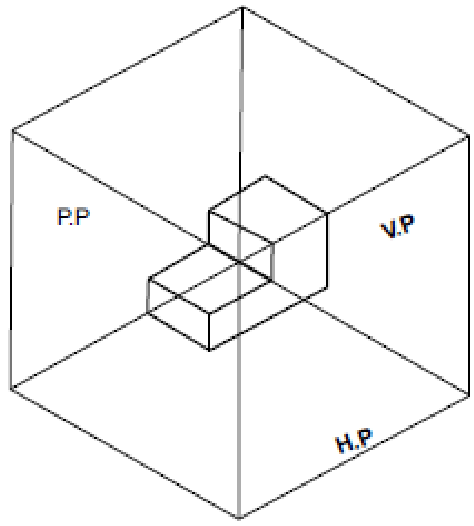
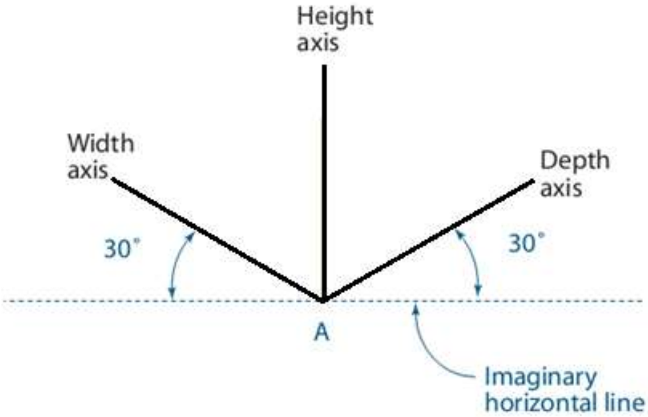


Fig. 4.1



Imagine that the views of the object are projected to the front, top and side of the transparent planes. Two different principal planes are used to get the projections of an object—the vertical plane (V.P.) and horizontal plane (H.P.) These planes intersect each other at right angles and the line of intersection is called axis of the planes. The projection on the vertical plane is called *elevation* or *front view* and the projection on horizontal plane is called *plan* or *top view*. The plane perpendicular to both horizontal and vertical is called profile plane (P.P.). The projection obtained on profile plane is called side view as shown in Fig. 13.2. The principles of orthographic projection can be applied in four different angles; first; second, third and fourth angle projection.

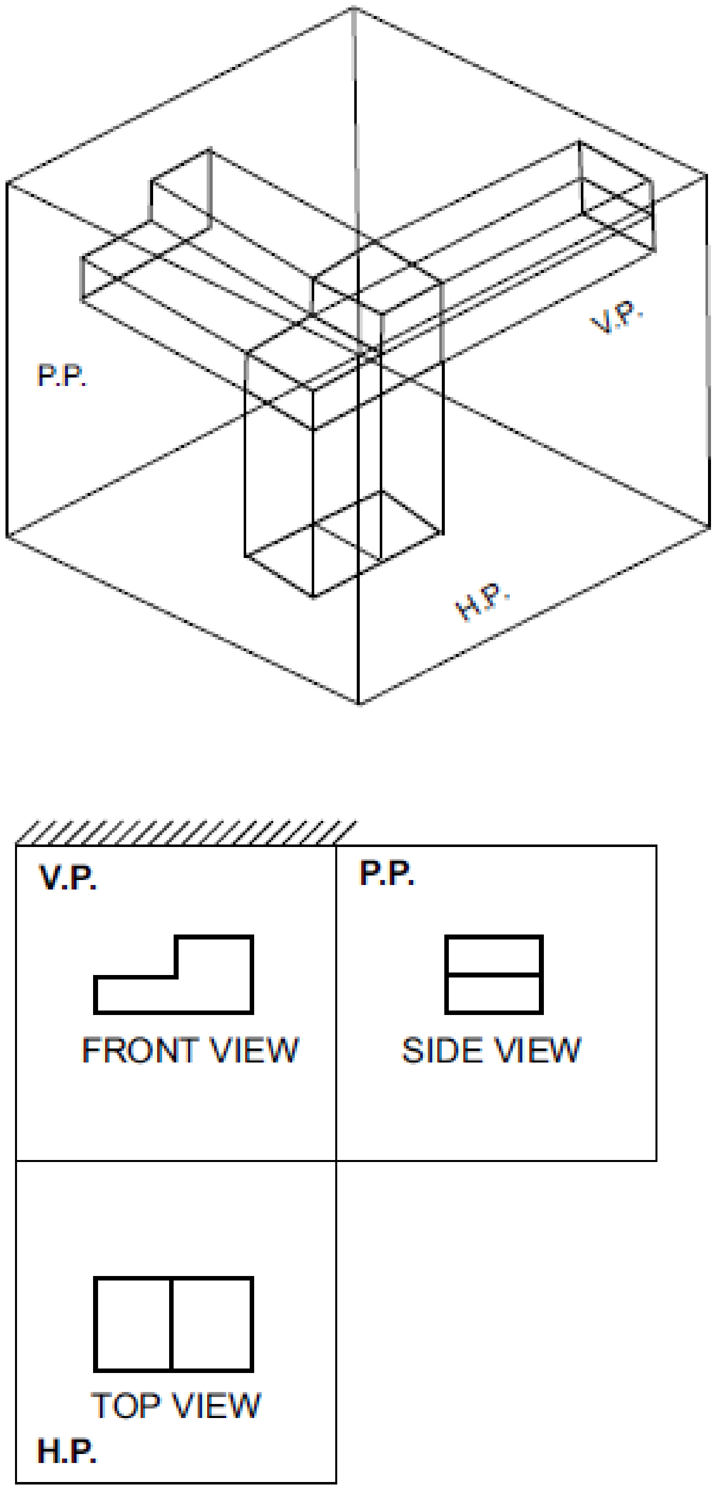


Fig. 4.2

4.5 TYPES OF ORTHOGRAPHIC PROJECTION

Following are the two types of orthographic projections used in engineering drawings.

4.5.1 First Angle Projection

In first angle projection, the object is assumed to be positioned in first quadrant as shown in Fig. 4.3(i). The object is placed between the observer and the plane of projection with top view being below the front view, because when horizontal plane is rotated in clockwise direction by an angle of 90º, it will become vertical. The relative positions of top view and front view are shown in Fig. 4.3(ii). Remember that, in first angle projection, the right side view goes to the left and left side view goes the right of front view.

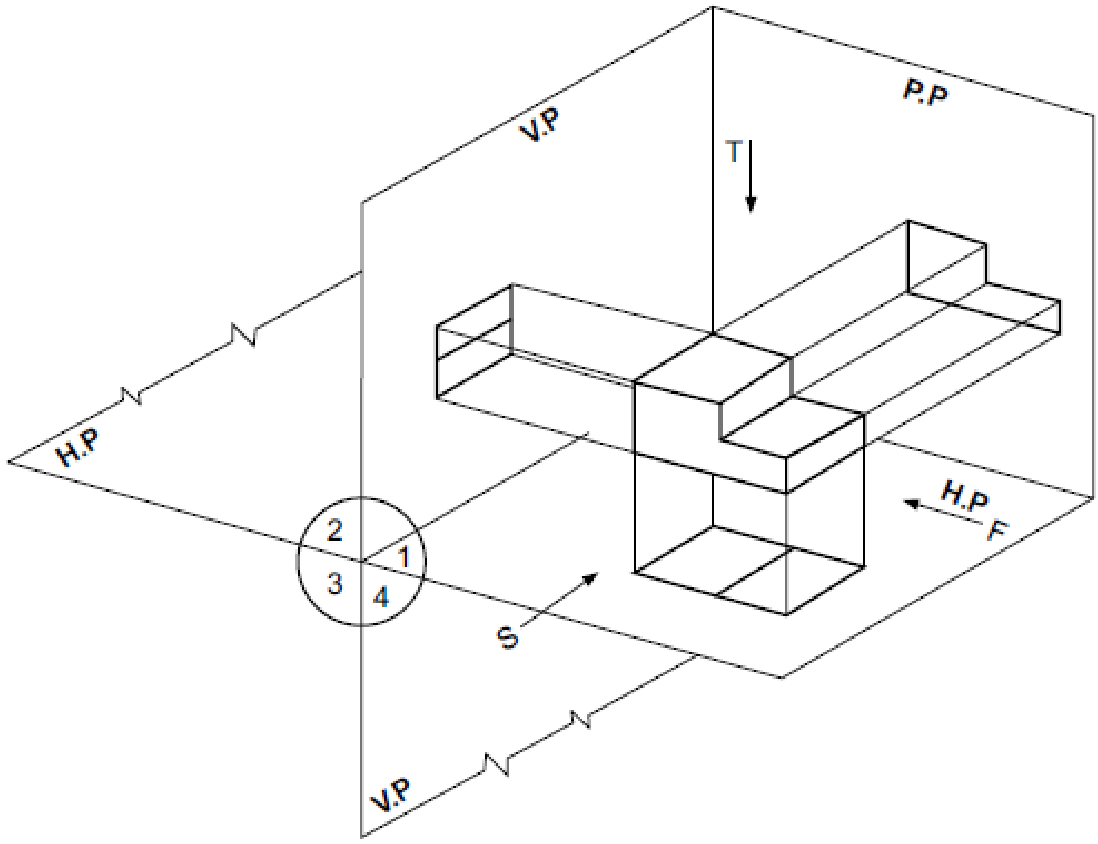


Fig. 4.3

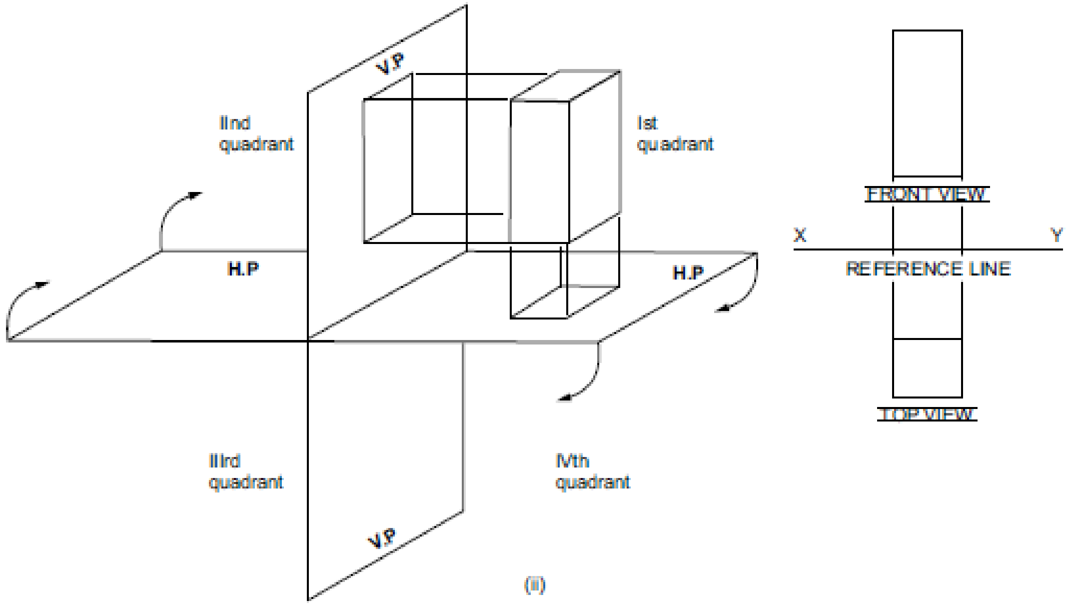
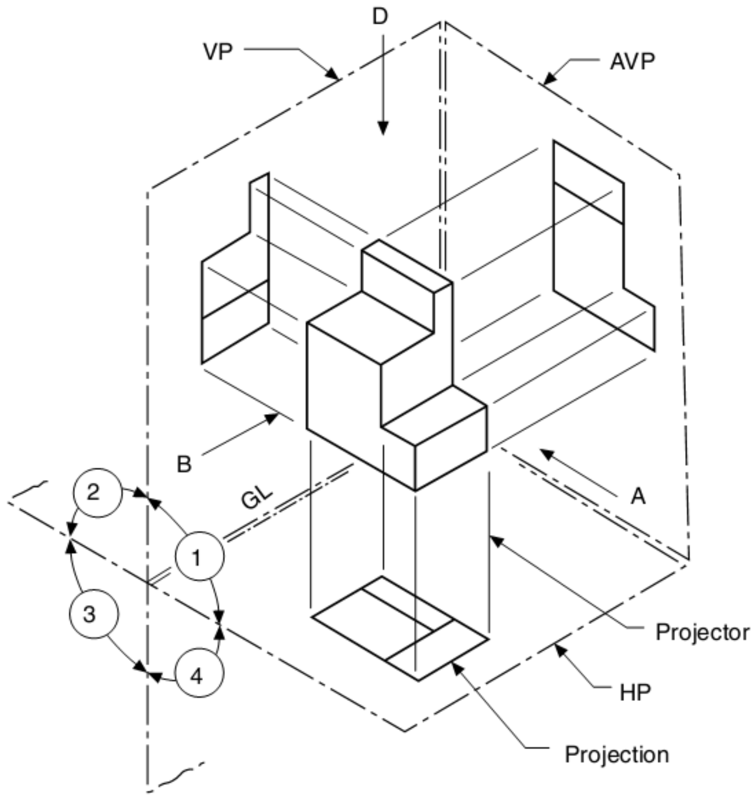


Fig. 4.3



First angle projection is commonly used in India, Europe and most of the world. This method of projection is recommended by the "Bureau of

Indian Standards” from 1991. The first angle and third angle projection symbol is shown in Fig. 4.4.

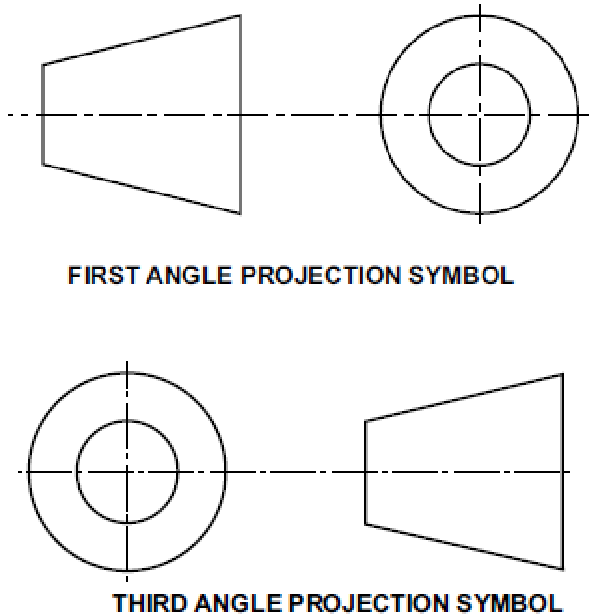


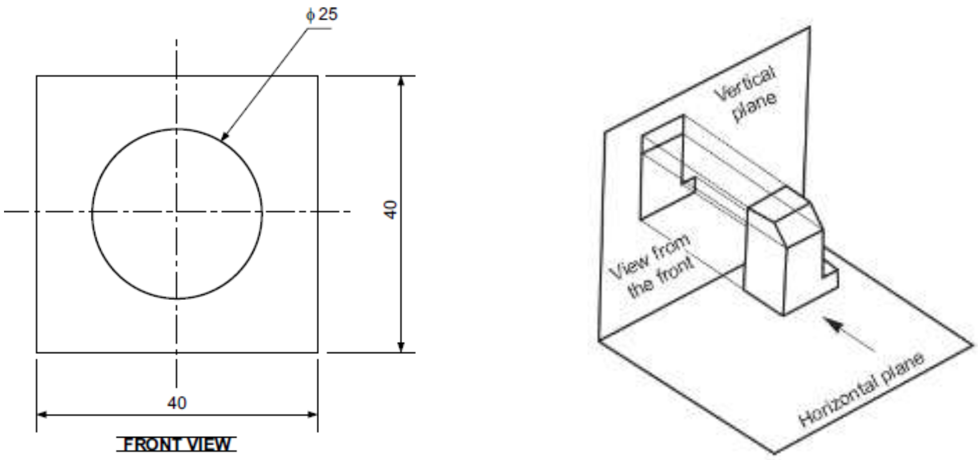
Fig. 4.4

4.6 SELECTION OF VIEWS

The selection of views to represent the object is very important in engineering drawing. The requirement of different views is as follows:

4.6.1 One View Drawing

In one view drawing only one view is required to describe certain objects completely as shown in Fig. 4.5.



4.6.2 Two View Drawing

In cylindrical and conical objects only two views are required for complete description as shown in Fig. 4.6.

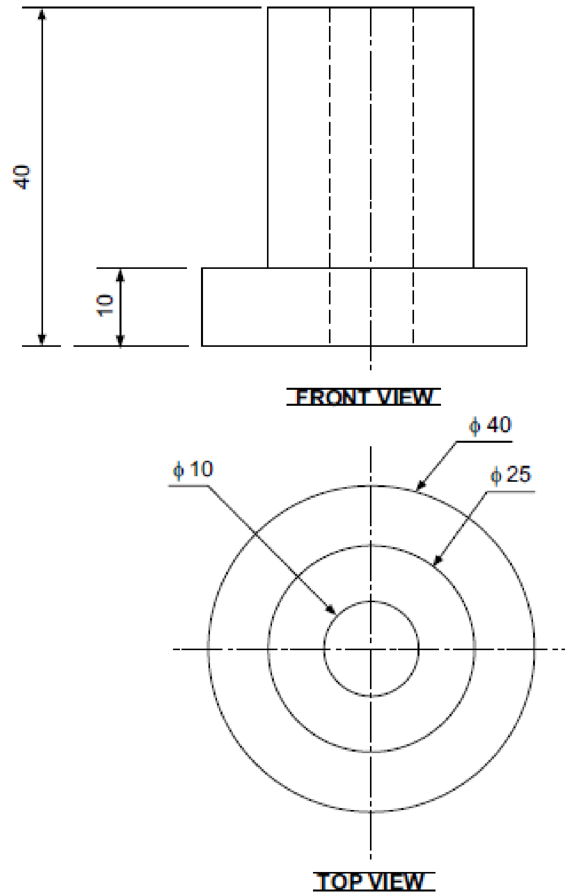


Fig. 4.6

4.6.3 Three view drawing

The object which can be described completely with the help of three views except irregular shape of an object as shown in Fig. 4.7. These three views are:

1. Front view 2. Side view 3. Top view

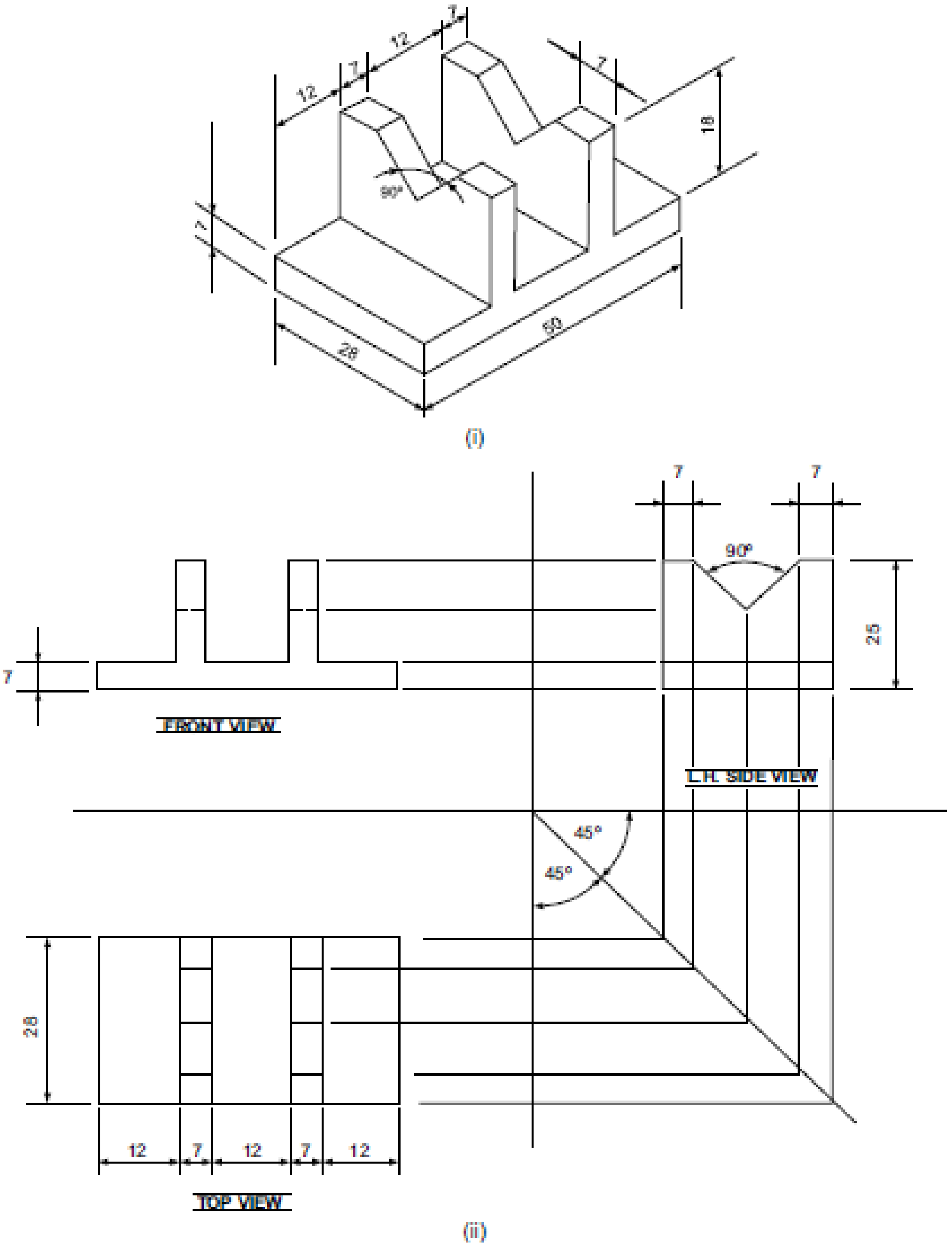


Fig. 4.7

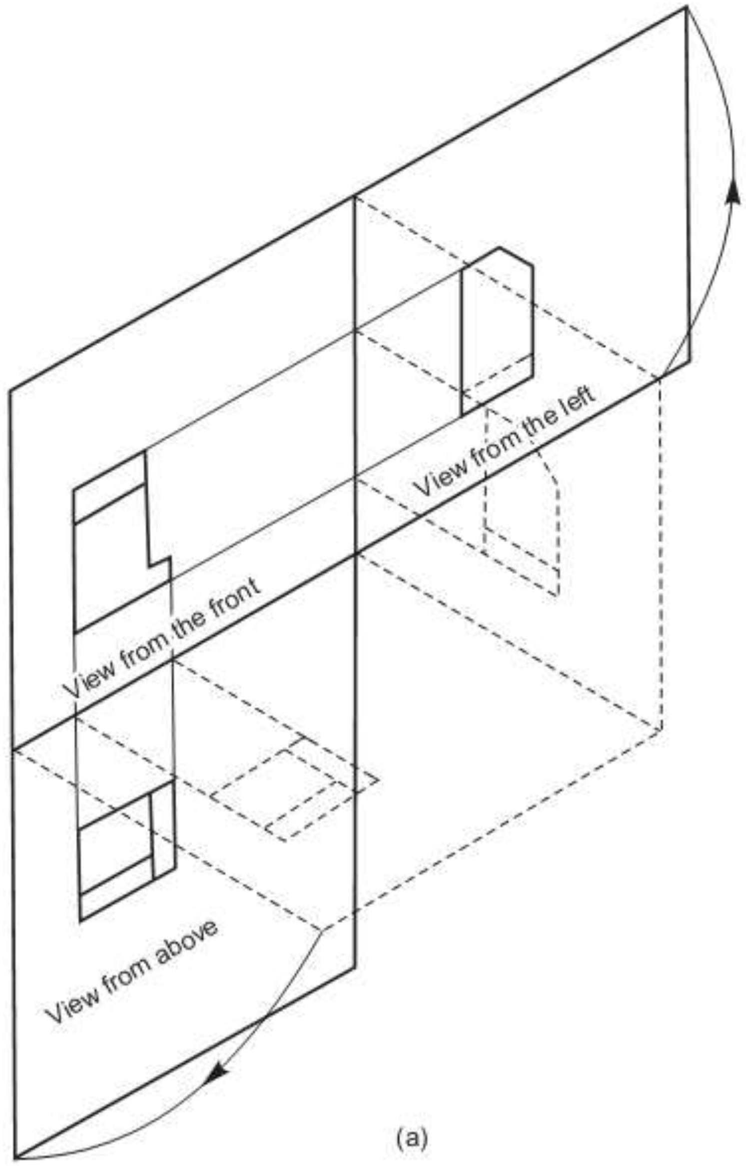
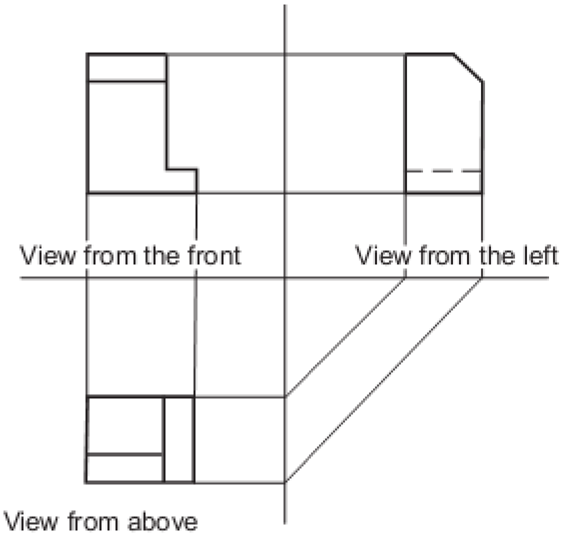
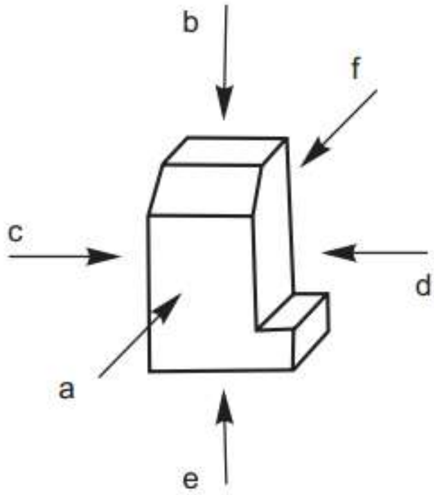


Fig. 4.7

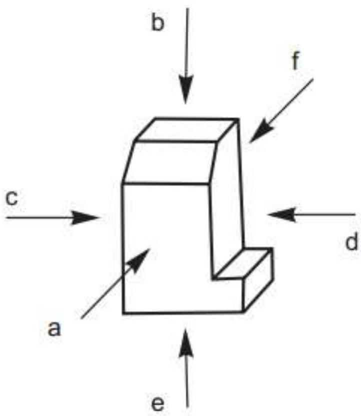




4.6.4 Six view drawing

Maximum six views are used to show the detailed informations of all the six sides of an object as shown in Fig. 4.8. These views are used for irregular shapes of the object on various faces as shown in Fig. 4.9.

1. Direction a = Front View = Elevation View
2. Direction b = Top View = Plan View
3. Direction c = Left Hand Side View
4. Direction d = Right Hand Side View
5. Direction e = Bottom View
6. Direction f = Rear View



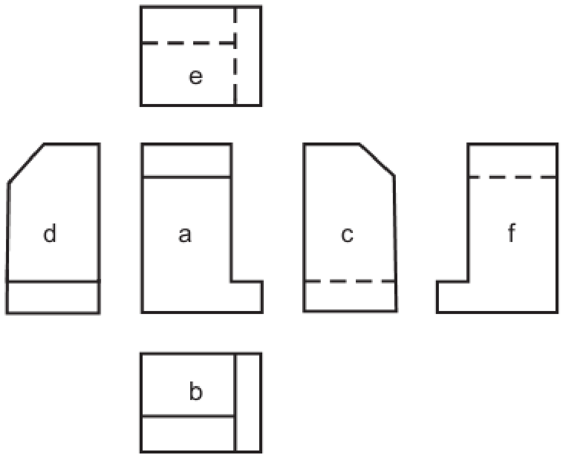

e
d
a
c
f
b

Fig. 4.9

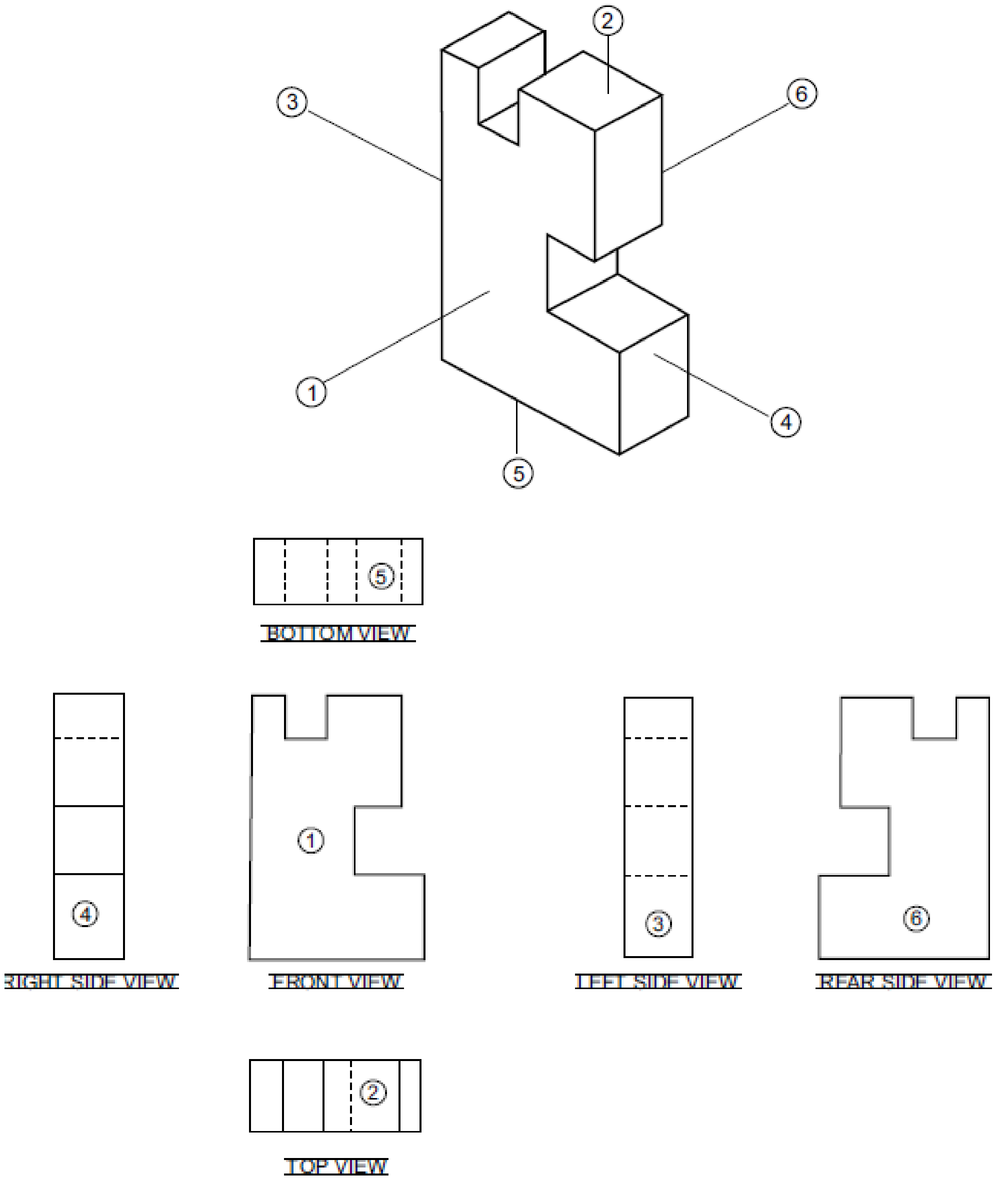
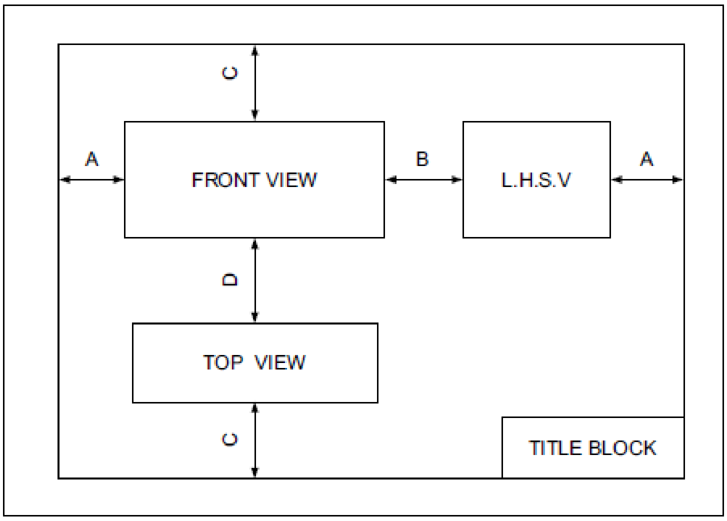

1
2
3
4
5
6
BOTTOM VIEW
RIGHT SIDE VIEW
FRONT VIEW
LEFT SIDE VIEW
REAR SIDE VIEW
TOP VIEW

Fig. 4.9

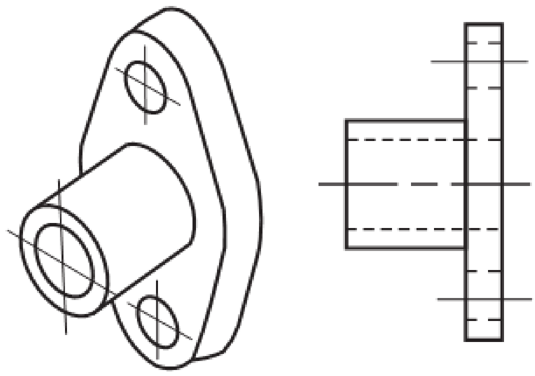
4.7 SPACING OF VIEWS

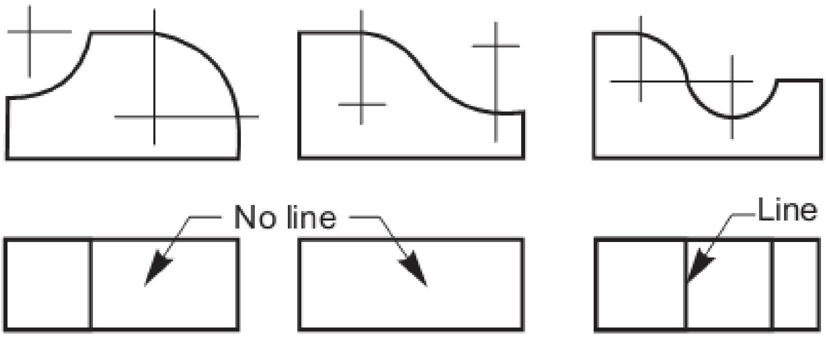
In technical drawings, spacing of views is very important factor before draw any object on the drawing sheet. It should be divided into suitable number of rectangles, if more than one view is required as shown in Figure below. Where A ≥ B and C ≥ D.

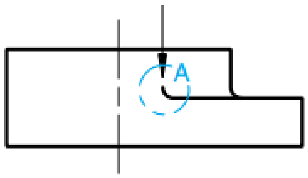


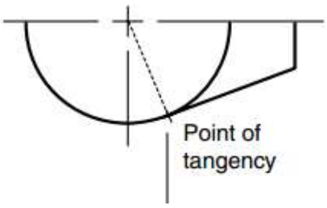
Ist ANGLE PROJECTION

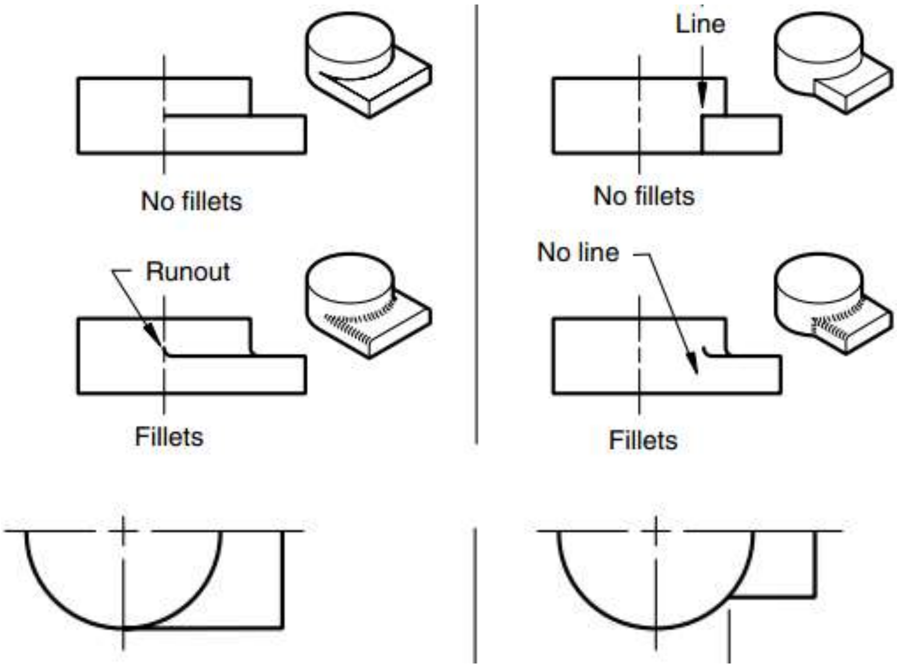
4.8 Hidden lines and Curved line

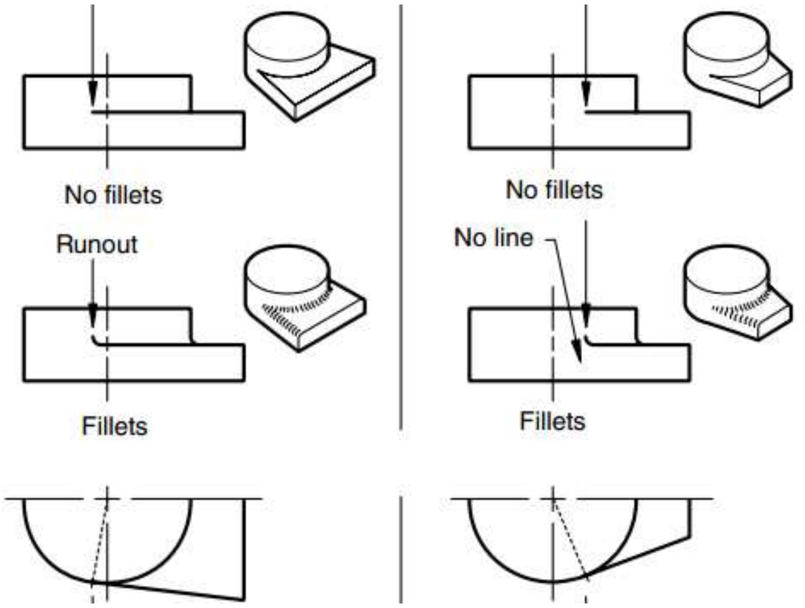


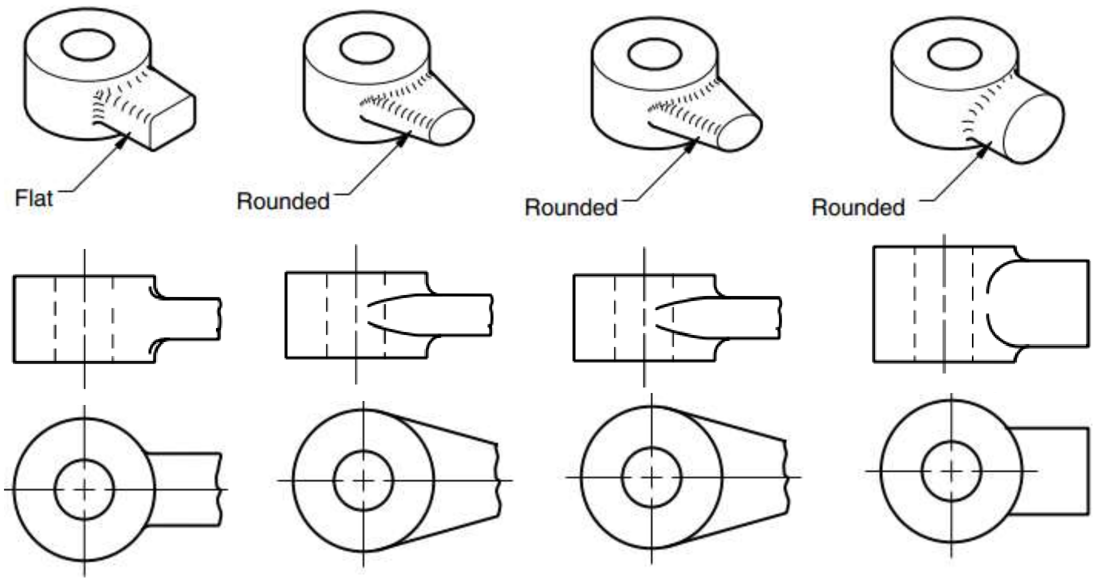

No line
Line

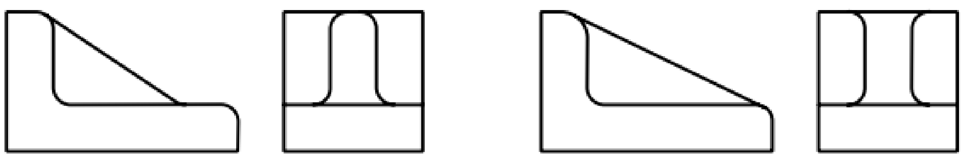

A


Point of
tangency


Line
No fillets
No fillets
Runout
No line
Fillets
Fillets


No fillets
Runout
Fillets
No fillets
No line
Fillets


Flat
Rounded
Rounded
Rounded



4.9 INTERSECTION OF SOLIDS

4.9.1 INTRODUCTION

Intersection occurs frequently in the design world. Therefore, a deep knowledge of it is must for designers and engineers.

The intersecting surfaces may be two plane surfaces or two curved surfaces of solids. The lateral surface of every solid taken as a whole is a curved surface. This surface may be made of only curved surface as in case of cylinders, cones etc. or of plane surface as in case of prisms, pyramids etc. In the former case the problem is said to be "intersection of surfaces" and in the latter case, it is known as interpenetration of solids.

When one solid penetrates the other, it is known as interpenetration of solids. In both the above cases the surfaces of the solids come in contact with another, the former is known as "the curve of intersection of surfaces" and the later "the curve of interpenetration".

It may be curved, straight or combination of curved and straight lines that occurs when geometrical surfaces such as cylinders, cones, prisms etc. intersect each other. In many engineering components such as different shapes of containers; tanks, machine casting, boiler shells, pipe joints etc, interpenetration of one part into another part may appear, hence the knowledge in intersection of their surface is required in order to fabricate those parts. The methods presented in this chapter are reorganized procedure for finding the more complicated lines of intersecting created by intersection of geometrical surface.

4.9.2 CLASSIFICATION OF INTERSECTING SURFACES

The intersecting surfaces are classified as follows:

1. **Intersection of two Plane Surfaces**

When two solids bounded by plane surfaces such as prism and pyramid, penetrate each other we get straight lines as their lines of intersections.

2. **Intersection of two Curved Surfaces**

When two solids, bounded by curved surfaces, such as cone and cylinder, intersect each other, the line of intersection is a tortuous curve.

3. **Intersection between a Plane Surface and a Curved Surface**

When two solids, one bounded by plane surface and other by curved surface, such as prism and cylinder, penetrate each other, the line of intersection is a curve.

4.10 METHODS OF DETERMINING THE LINE OF INTERSECTION

There are two methods of determining the line of intersection between surfaces of two interpenetrating solids.

1. **Line Method**

The process consists of drawing a number of lines on one surface and locating the points at which these lines intersect with the surface of the other solid. The points obtained lie on the line of intersection which may be either straight or curved.

2. **Cutting Plane Method**

This method is generally used to determine the line of intersection. The two solids, are assumed to be cut by a series of cutting planes which are so selected as to cut the surface of one of the solids in straight lines and that of the other in straight lines or circles.

Problem 1: A cylinder of 50 mm diameter and axis 70 mm long stands with its base on H.P. It is completely penetrated by a horizontal cylinder of 40 mm diameter and axis 80 mm long such that their axis bisect each other at right angles. The axis of the penetrating cylinder is parallel to

V.P as shown in Fig. 4.10(i). Draw the projections showing curves of intersection.

Solution. See Fig. 4.10(ii).

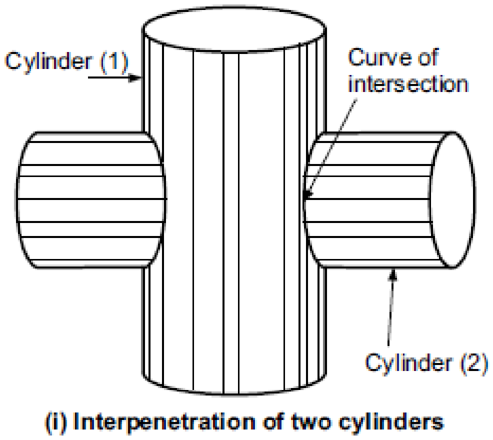


Fig. 4.10(i).

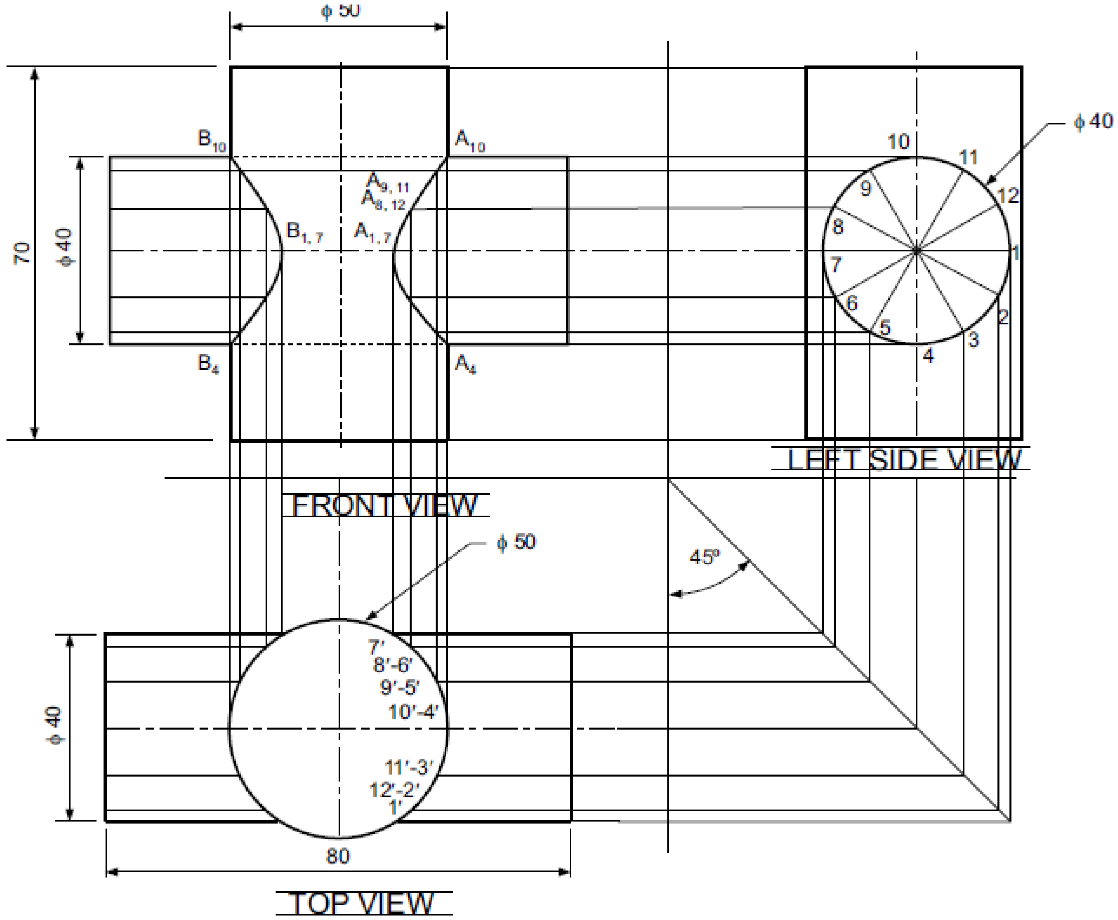


Fig. 4.10(ii).

Method Ist

Line Method

1. Draw the top, front and left side views of the vertical cylinder.

2. Draw the left side view of the horizontal cylinder as a circle of diameter 40 mm such that its centre is at the mid-point of the axis of the vertical cylinder. Divide the circle into 12 equal parts and mark point 1, 2, 3, ... etc.

3. Project these points on circle in the top view as 1', 2', 3' ... etc.

4. Transfer these points 1', 2', 3' ... from top view to front view by projectors so as to cut the corresponding points projected from the left side view.

5. Join all the intersecting points and draw a smooth curve as shown in Fig. 4.10(ii)

Method Second

Cutting Plane Method

Assume a series of horizontal section planes, passing through the generators of the horizontal cylinder to cut both the cylinders. For all the horizontal section planes, the sectional top view of the vertical cylinder will always be circle of 50 mm diameter. The cutting planes passing through the lines 2-12 as shown in Fig. 4.11. Points at which circle and rectangles intersect each other, will lie on the curve of intersection. P2, P12 are two such points at which the sides of the rectangle cut the circle. Other cutting plane may be 3-11, 4-10, 5-9 etc. and corresponding intersecting points are p3 and p11, p4 and p10, p5 and p9 etc. Join all the intersecting points and draw a smooth curve. Another intersecting curve on right side may be completed similarly.

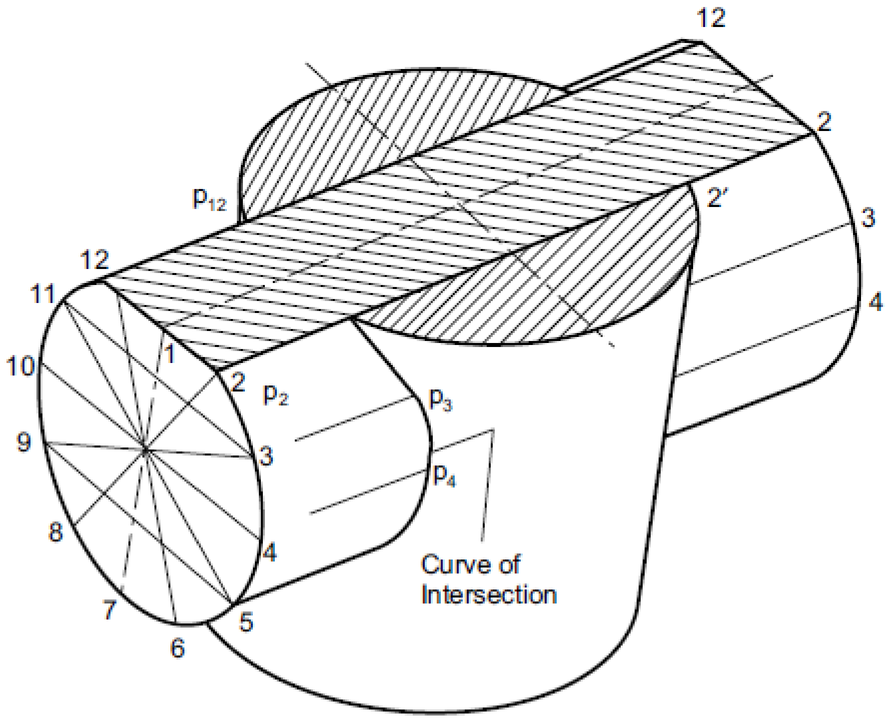
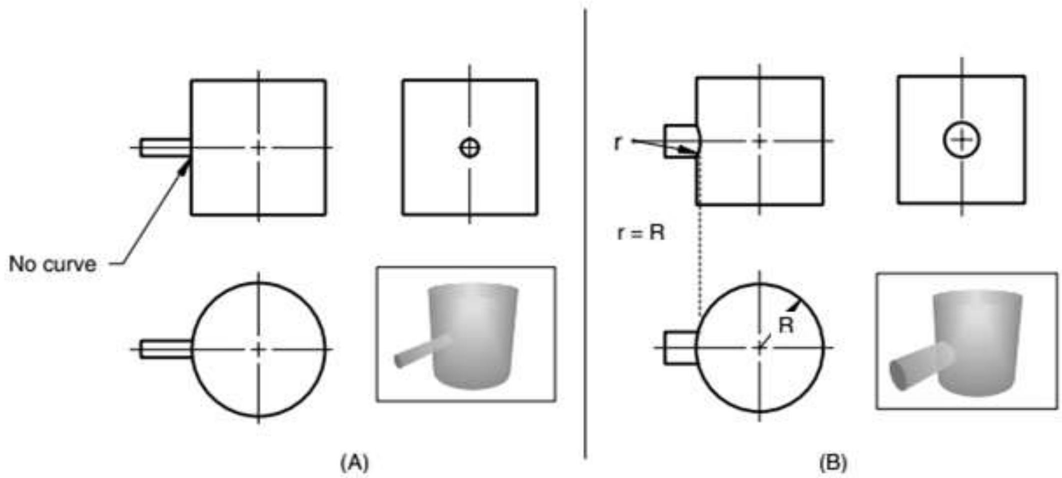
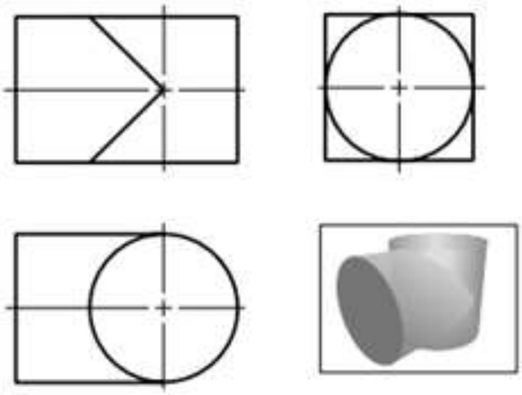
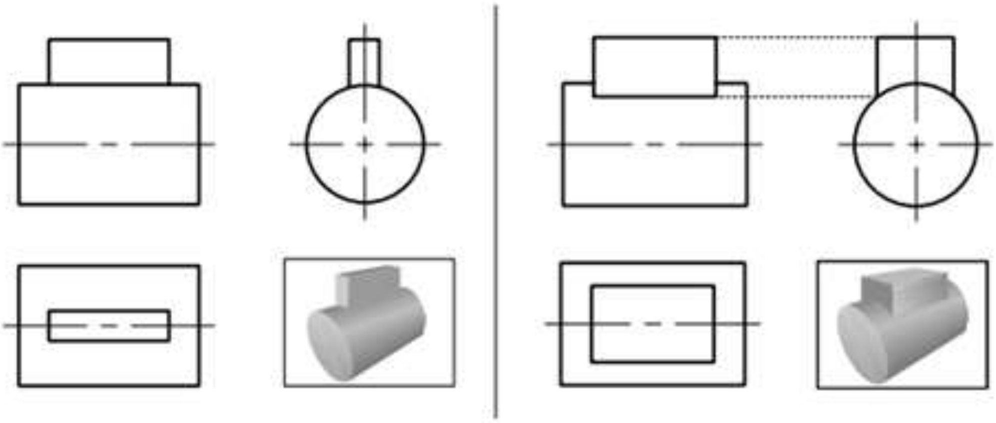

12
2
P12
2′
3
12
11
4
1
10
2
P2
P3
9
3
P4
8
4
Curve of
Intersection
7
5
6

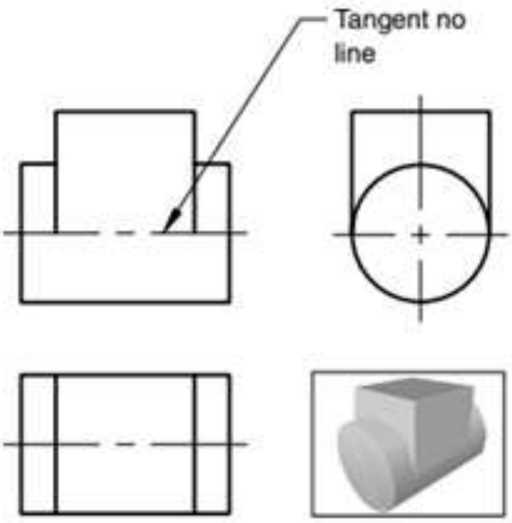
Fig. 4.11


No curve
r
r = R
R
(A)
(B)

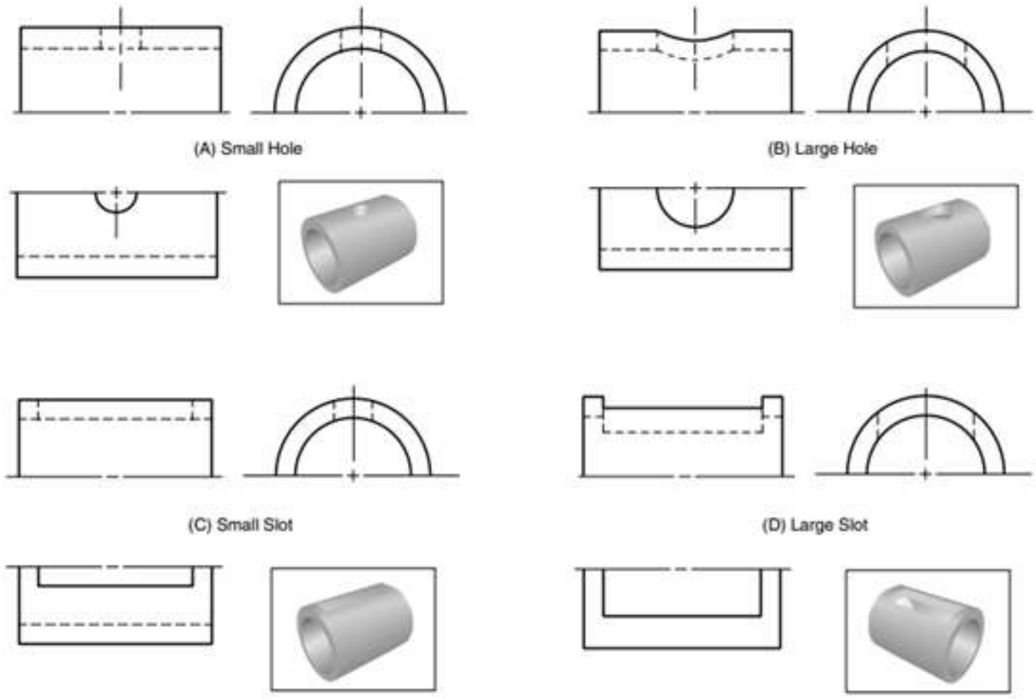


(C)





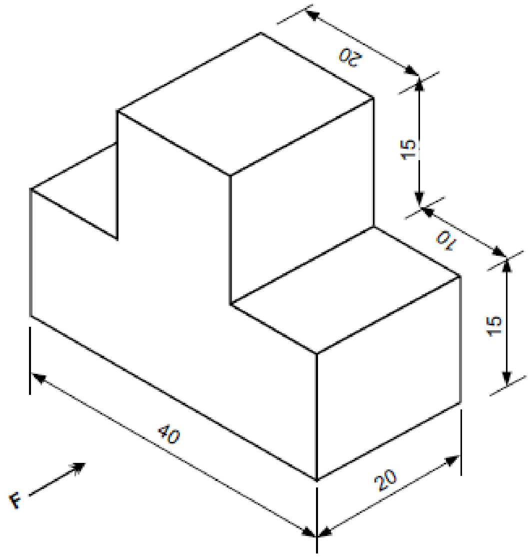
A cylinder and a hole

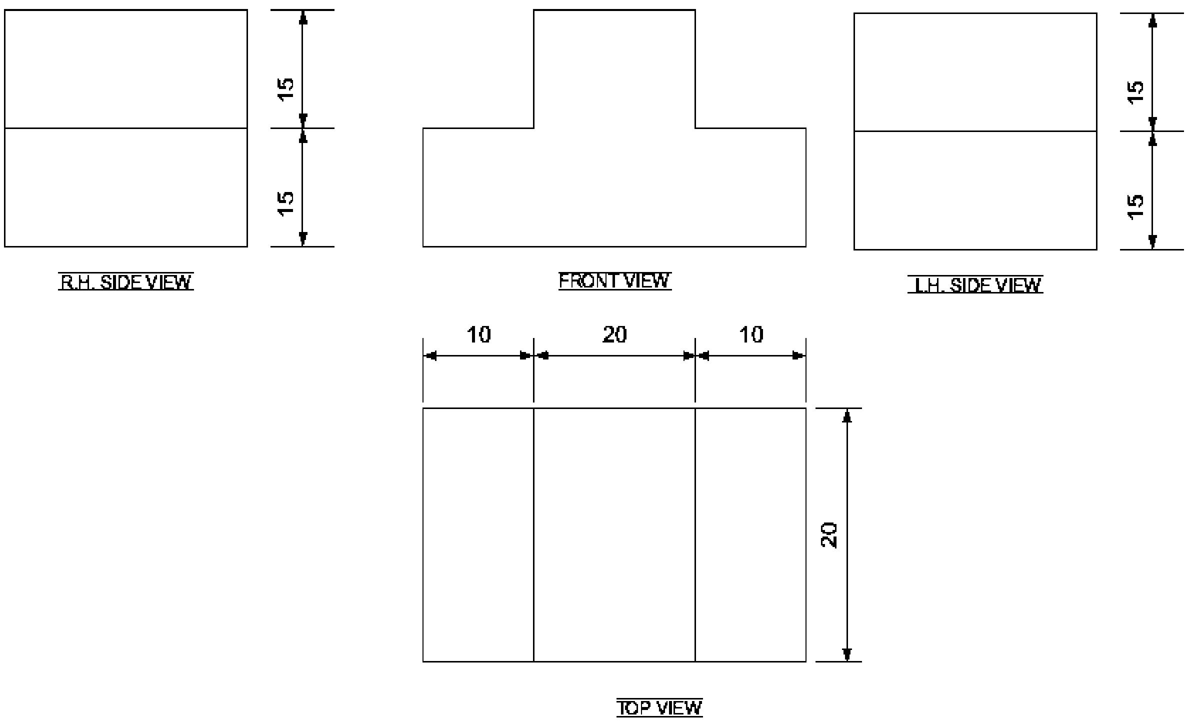


Solved Examples

Problem 1:

The figure below shows an isometric view of an object. Draw the following views: (Use Ist angle projection). 1. Front view (Elevation) 2. R.H. Side view 3. L.H. Side view 4. Top view (Plan)

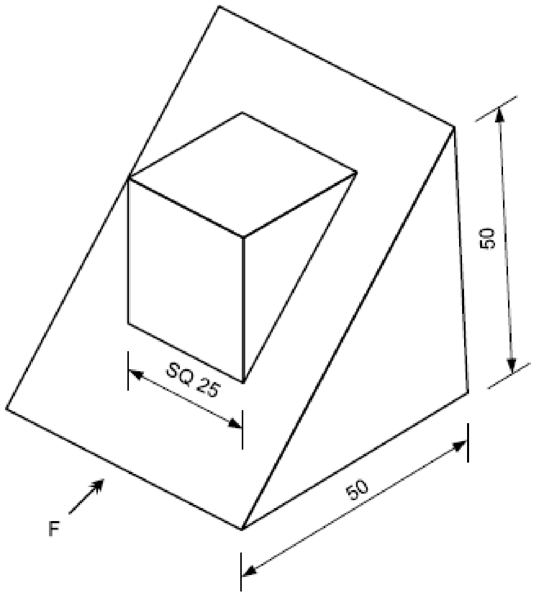


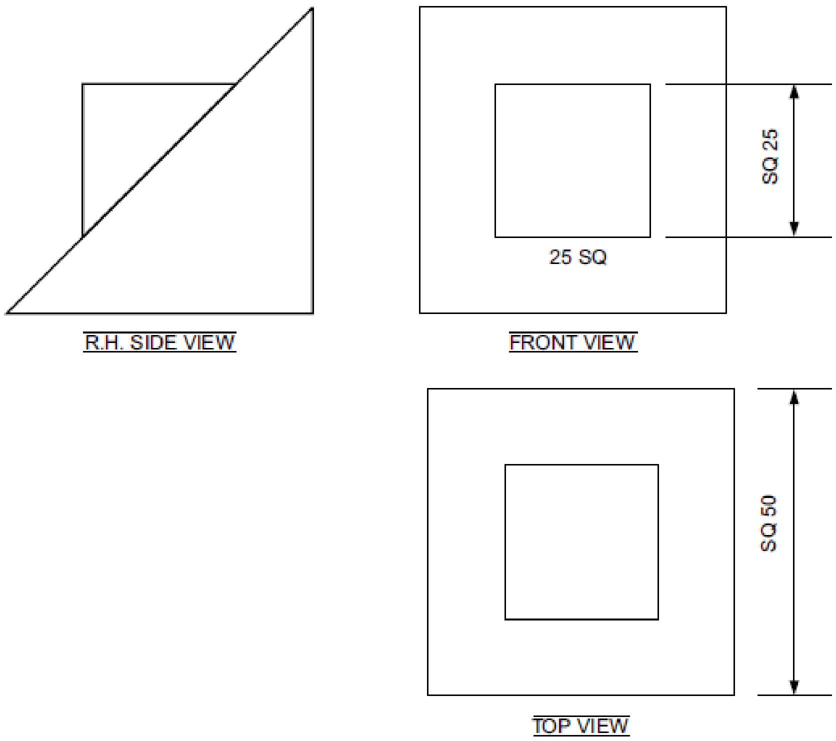


Problem 2

The figure below shows a square turncated prism. Draw the following views. (Use Ist angle projection).

1. Front view 2. R.H. Side view 3. Top view

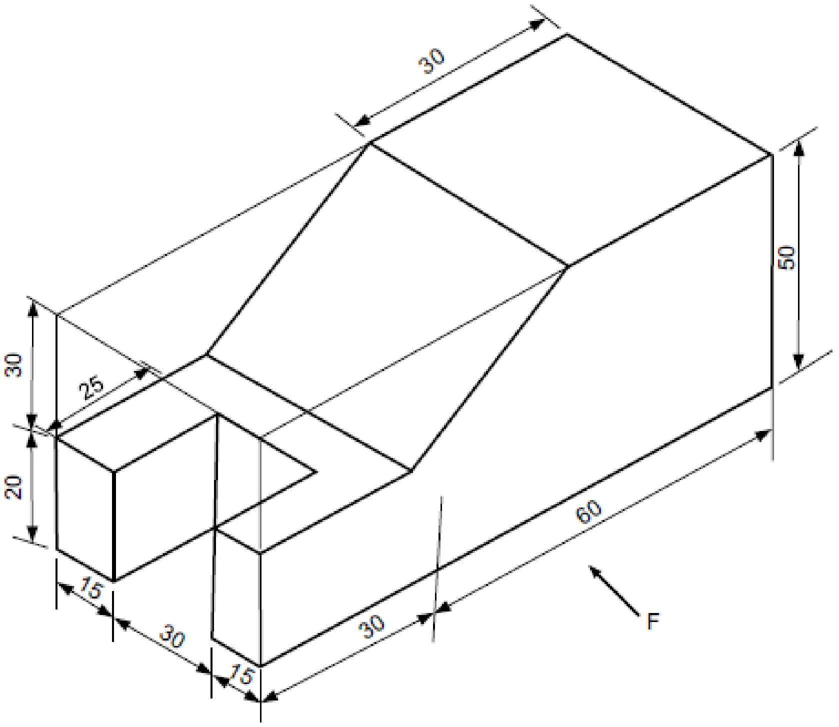


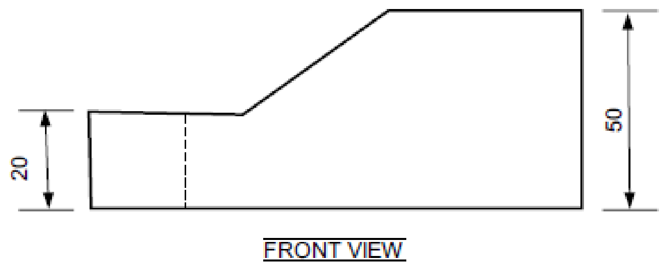


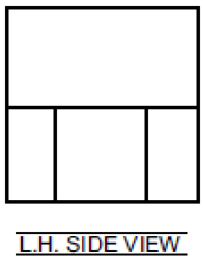
Problem 3

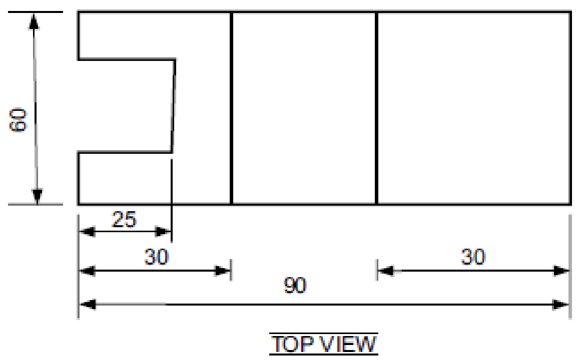
The figure below shows an isometric view of an object. Draw the following views: (Use I[st] angle projection).

1. Front view 2. L.H. Side view 3. Top view





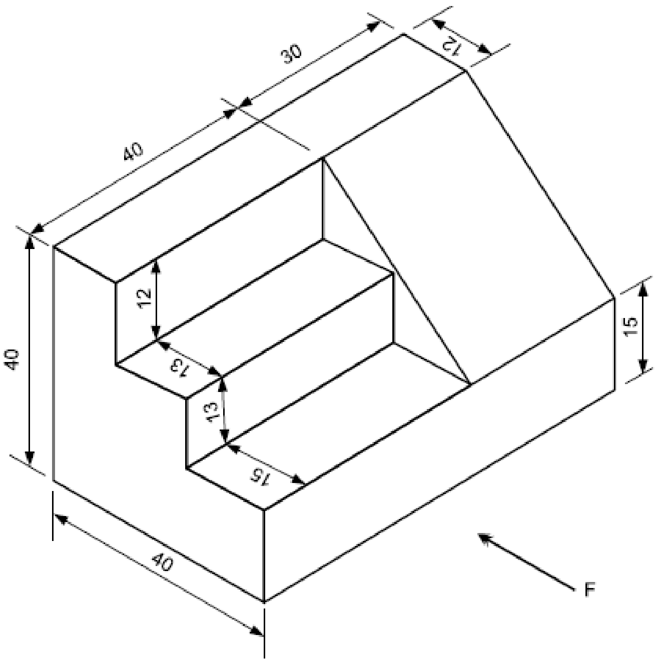


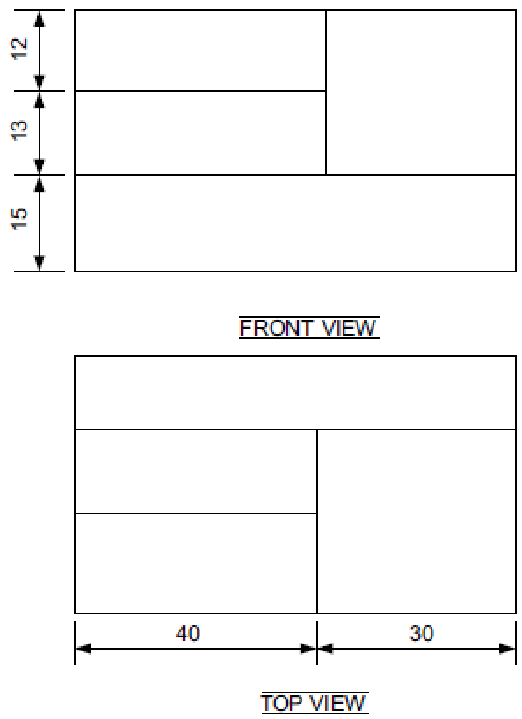


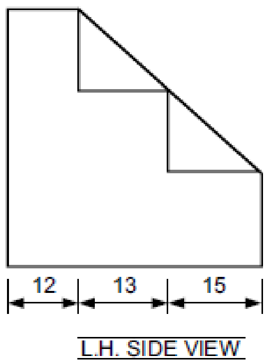
Problem 4

The figure below shows an isometric block of an object. Draw the following views: (Use I[st] angle projection).

1. Front view   2. L.H. Side view   3. Top view

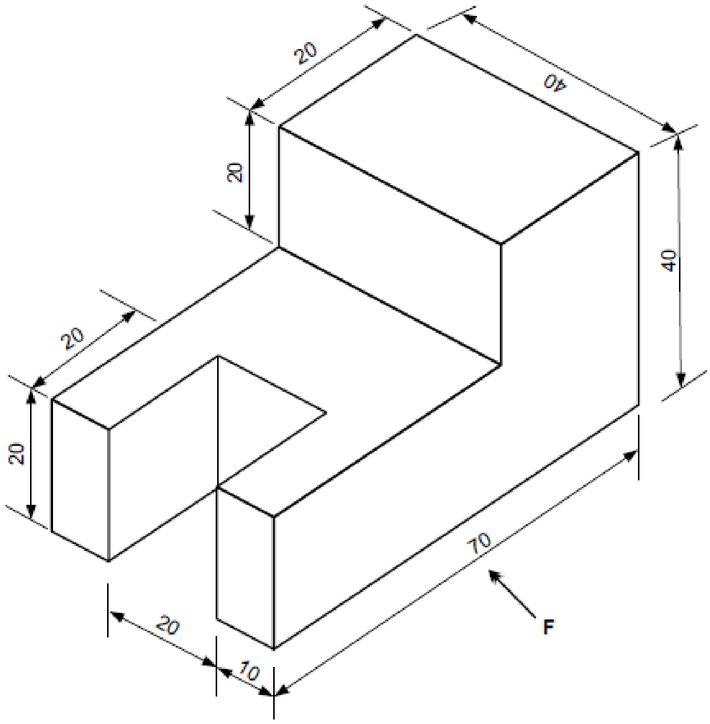


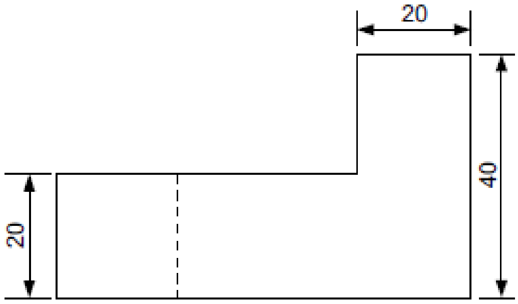


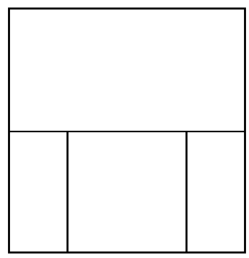


Problem 5

The figure below shows a pictorial view of an object. Draw the following views. 1. Front view 2. L.H. Side view 3. Top view

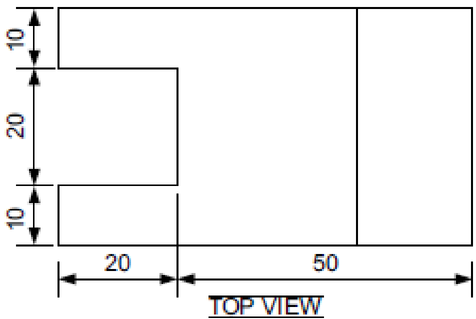






FRONT VIEW

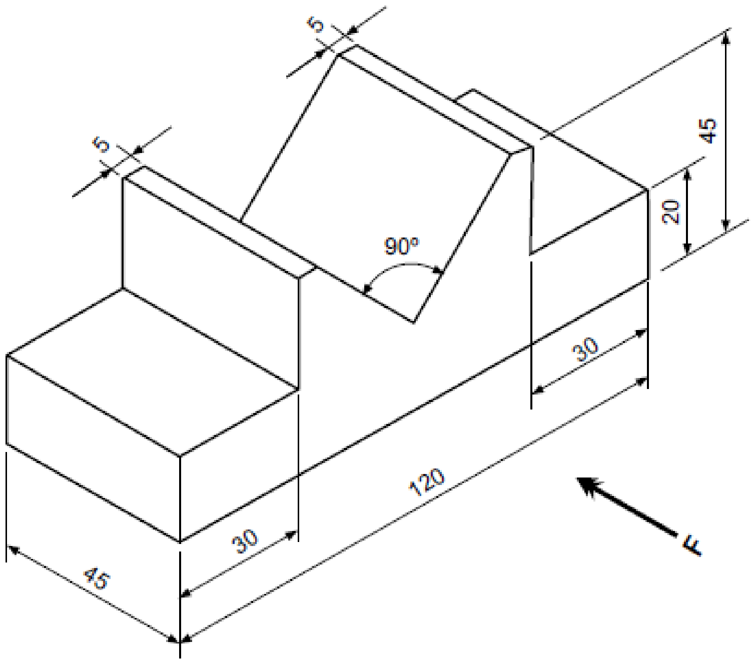
L.H. SIDE VIEW

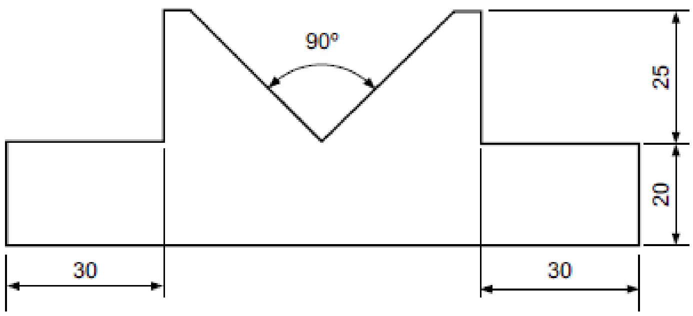


TOP VIEW

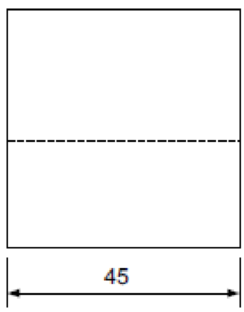
Problem 6

The figure below shows an isometric view of a block. Draw the following views. 1. Front view 2. L.H. Side view 3. Top view

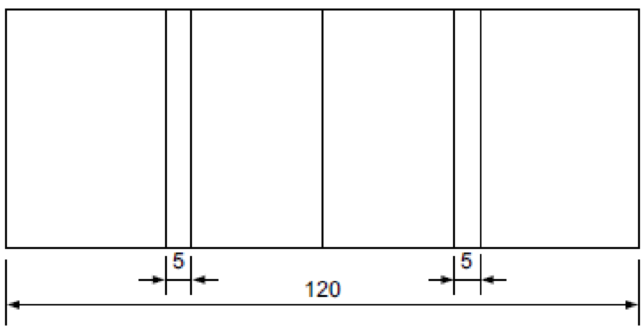




FRONT VIEW



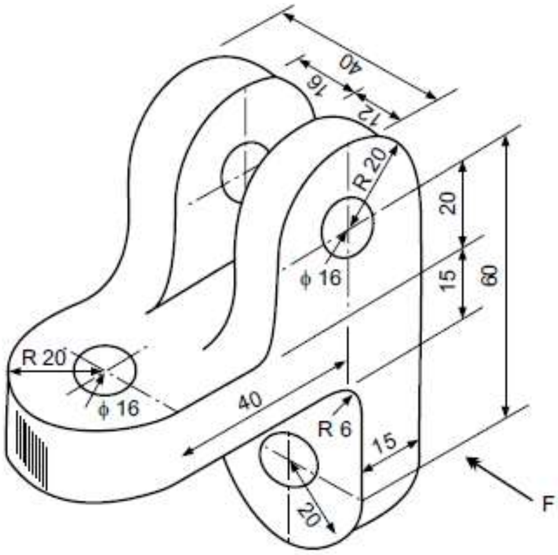
LEFT SIDE VIEW

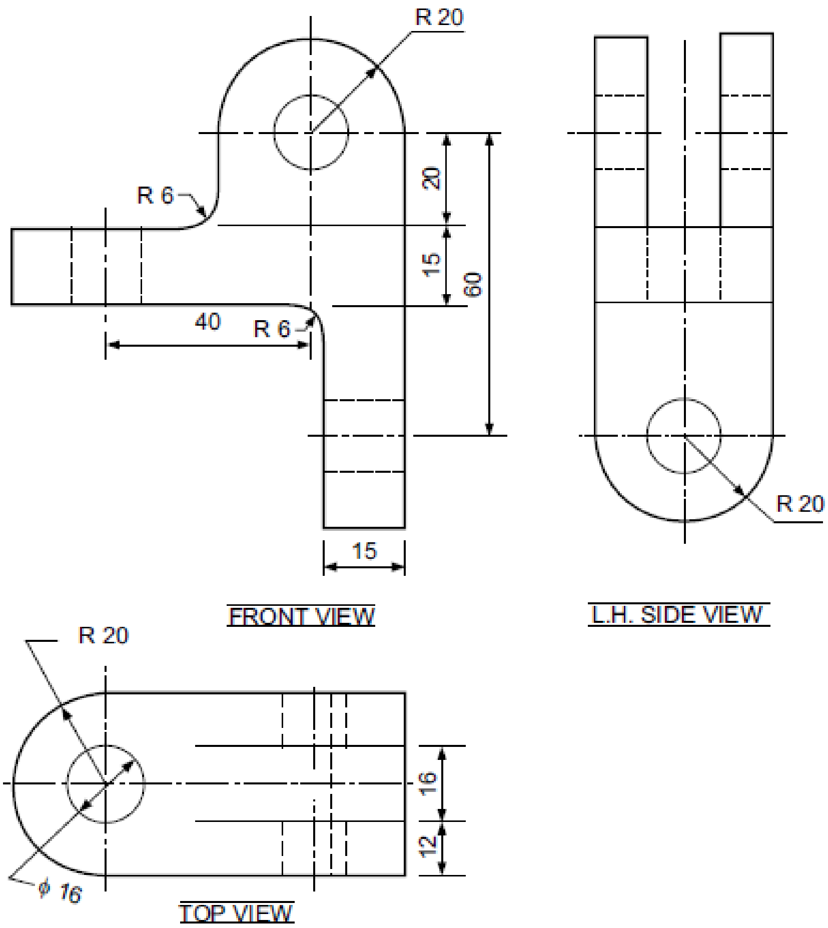


TOP VIEW

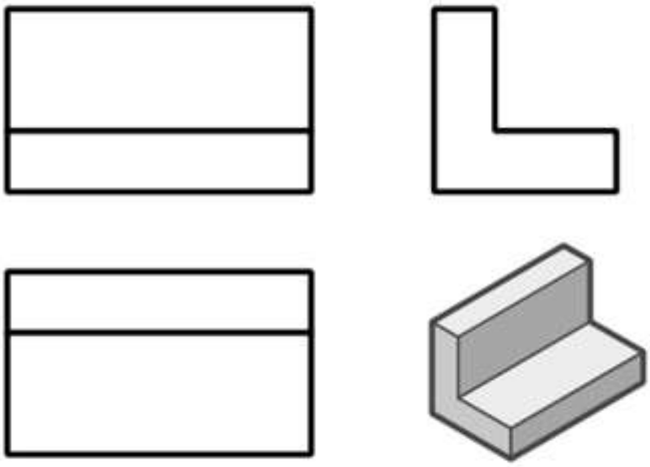
Problem 7

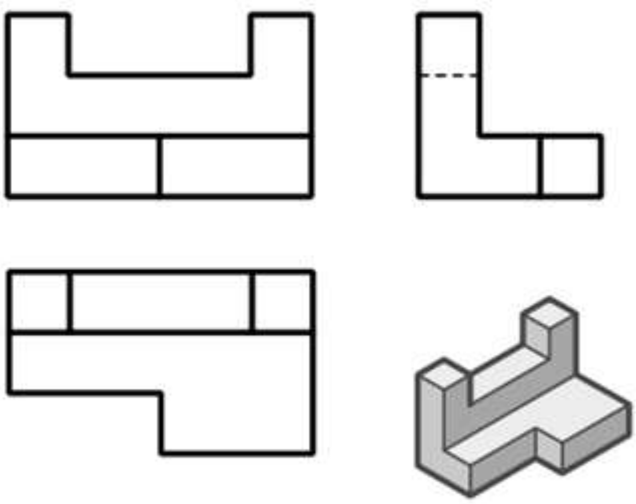
The figure below shows an isometric view of a block. Draw the following views. 1. Front view 2. L.H. Side view 3. Top view

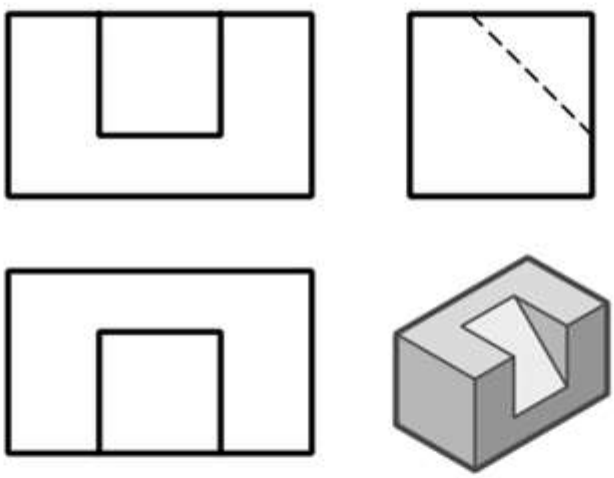


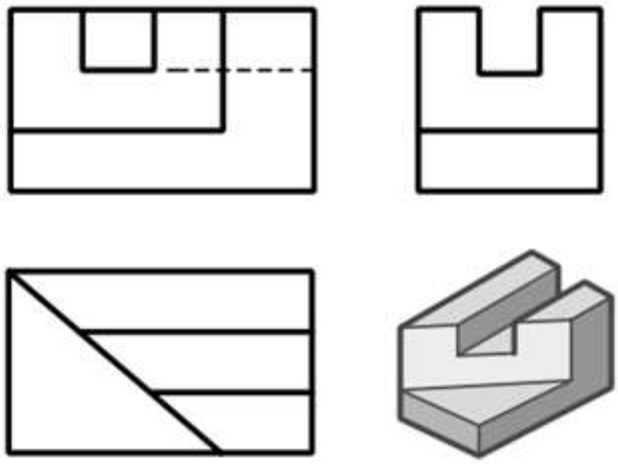


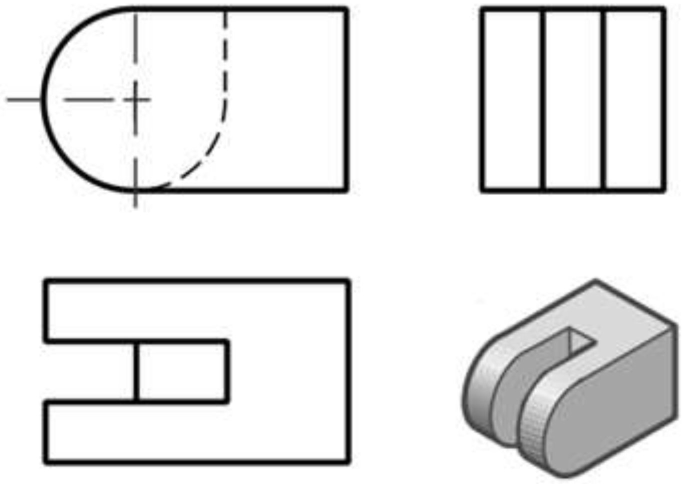
Addetinal Solved problems

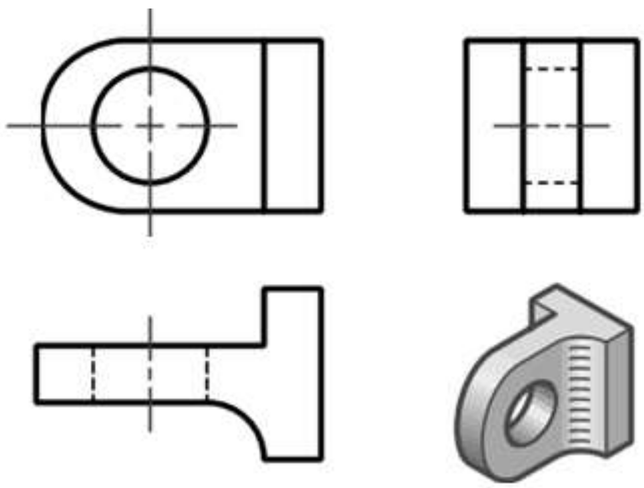


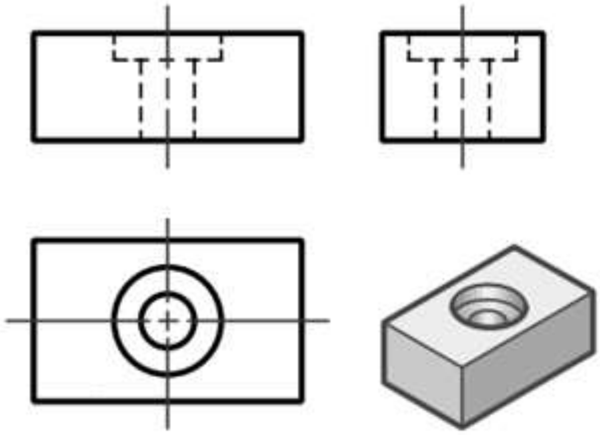


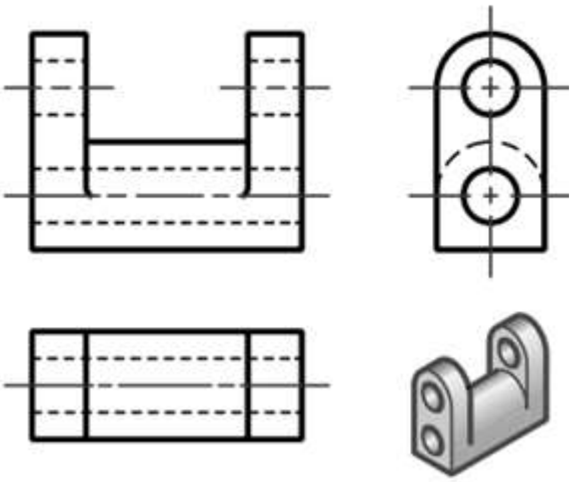


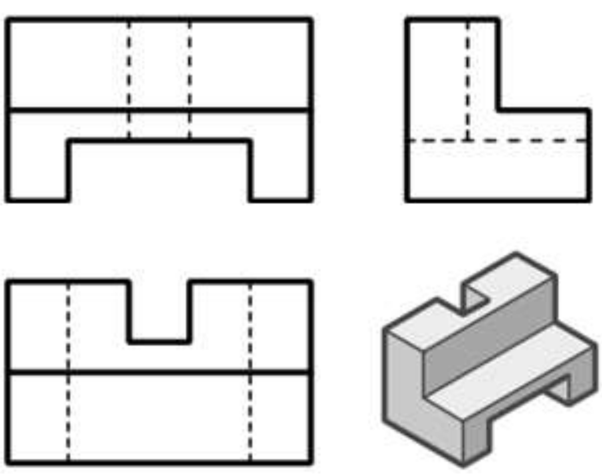






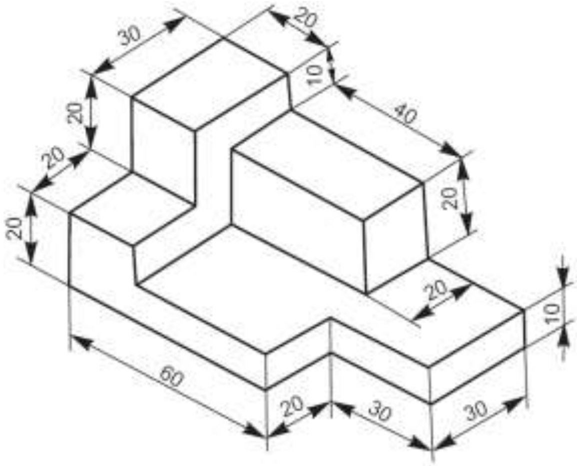


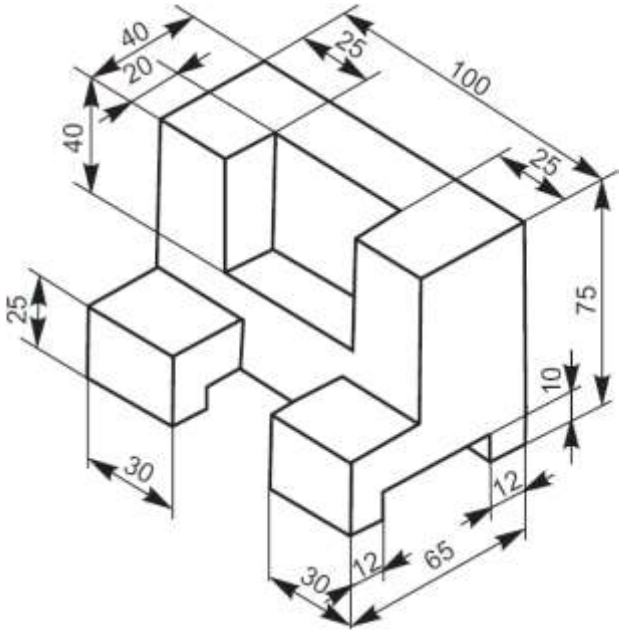


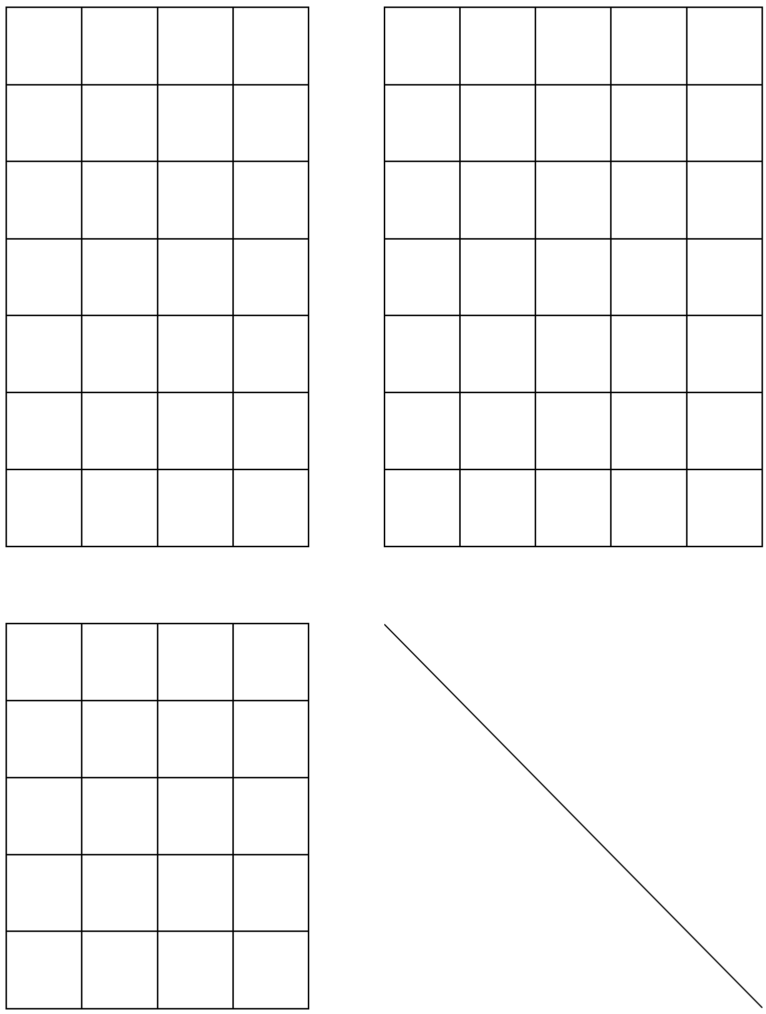


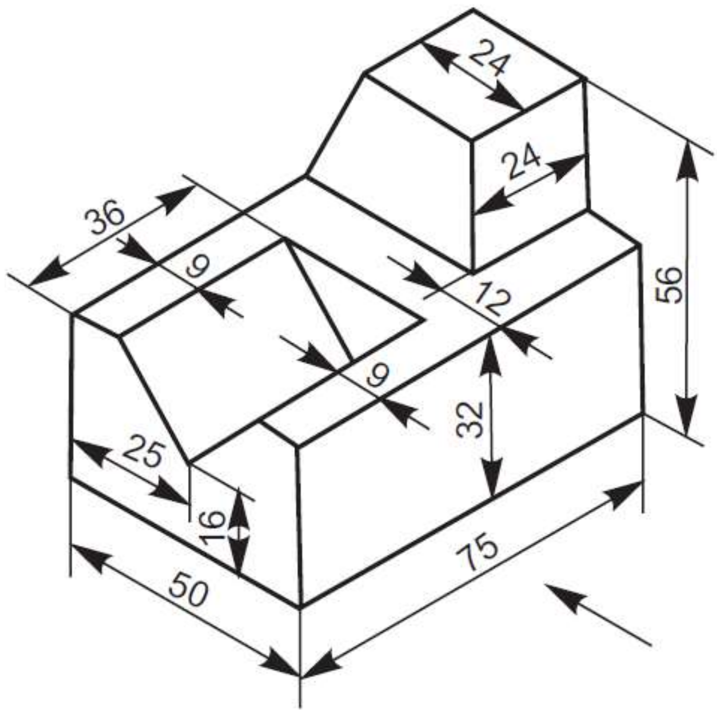
Exercises

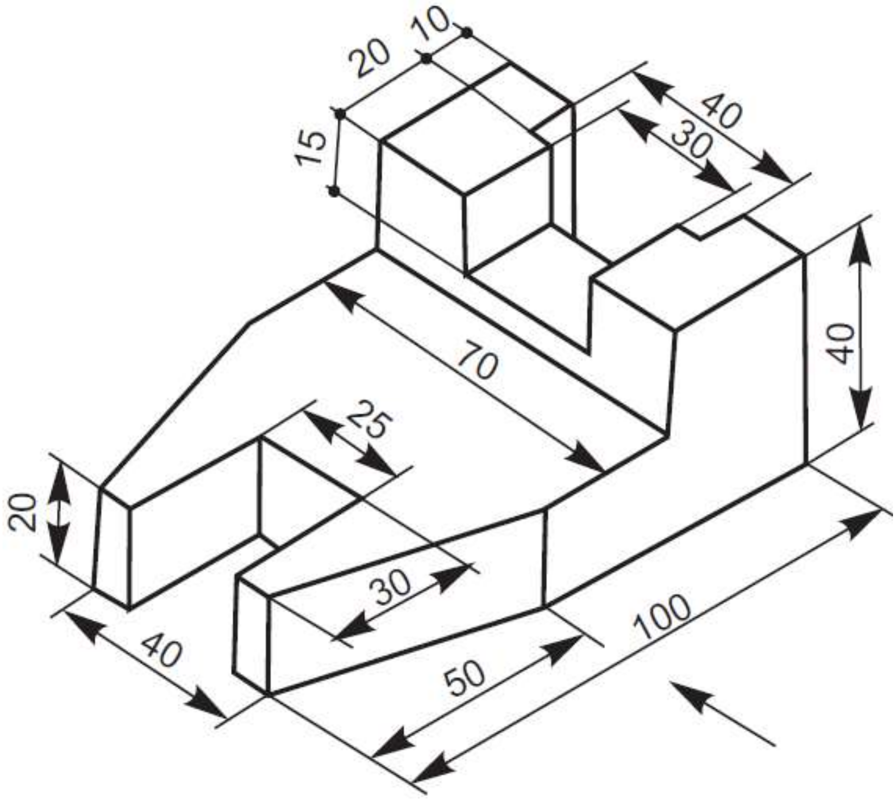
The pictorial views of different types of objects are shown in Figur. Draw the following views. 1. Front view 2. L.H. Side view 3. Top view




40
20
25
100
40
25
75
25
10
30
12
65
30
12




24
24
36
9
12
56
9
32
25
16
50
75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20
15
40
30
70
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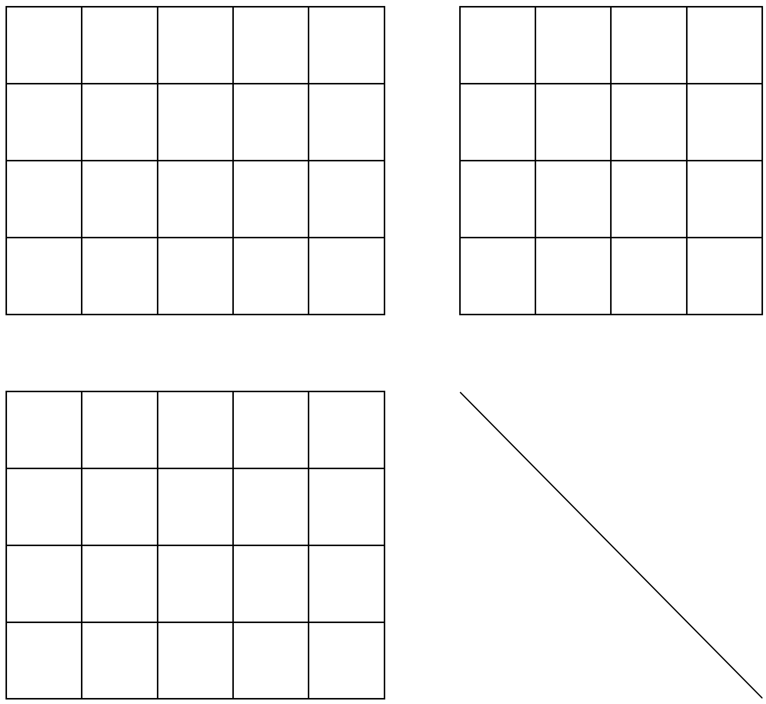
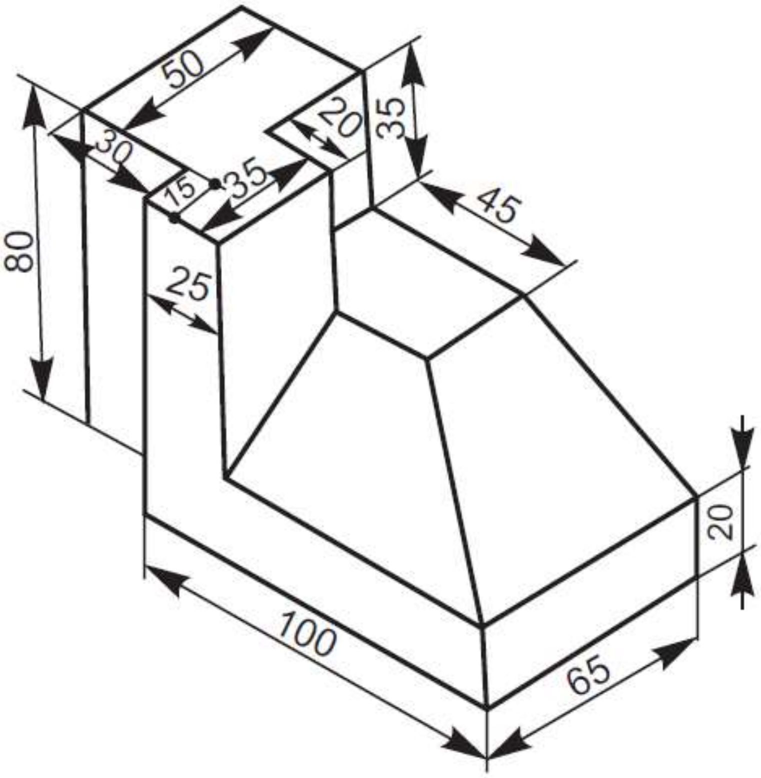
40
50
100
40

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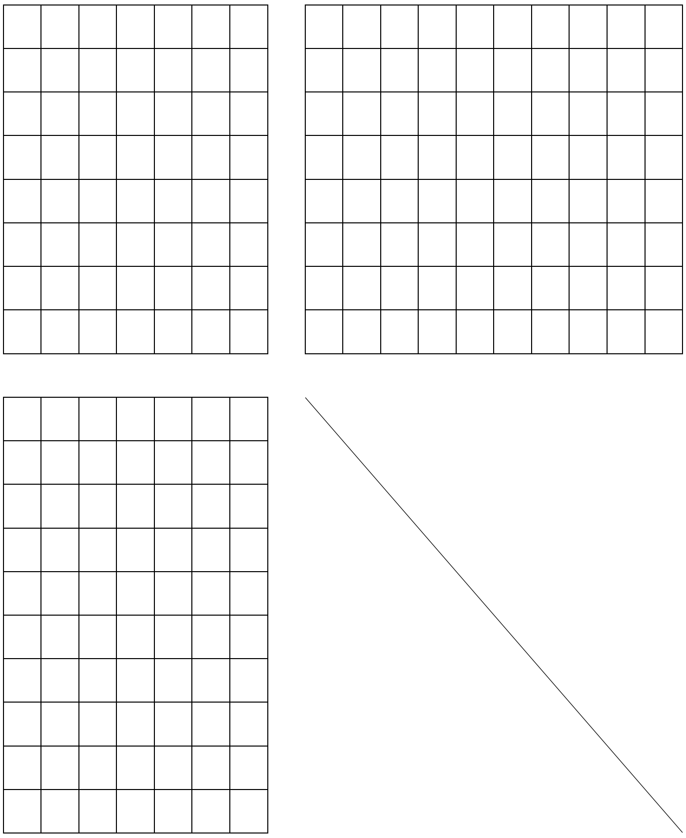
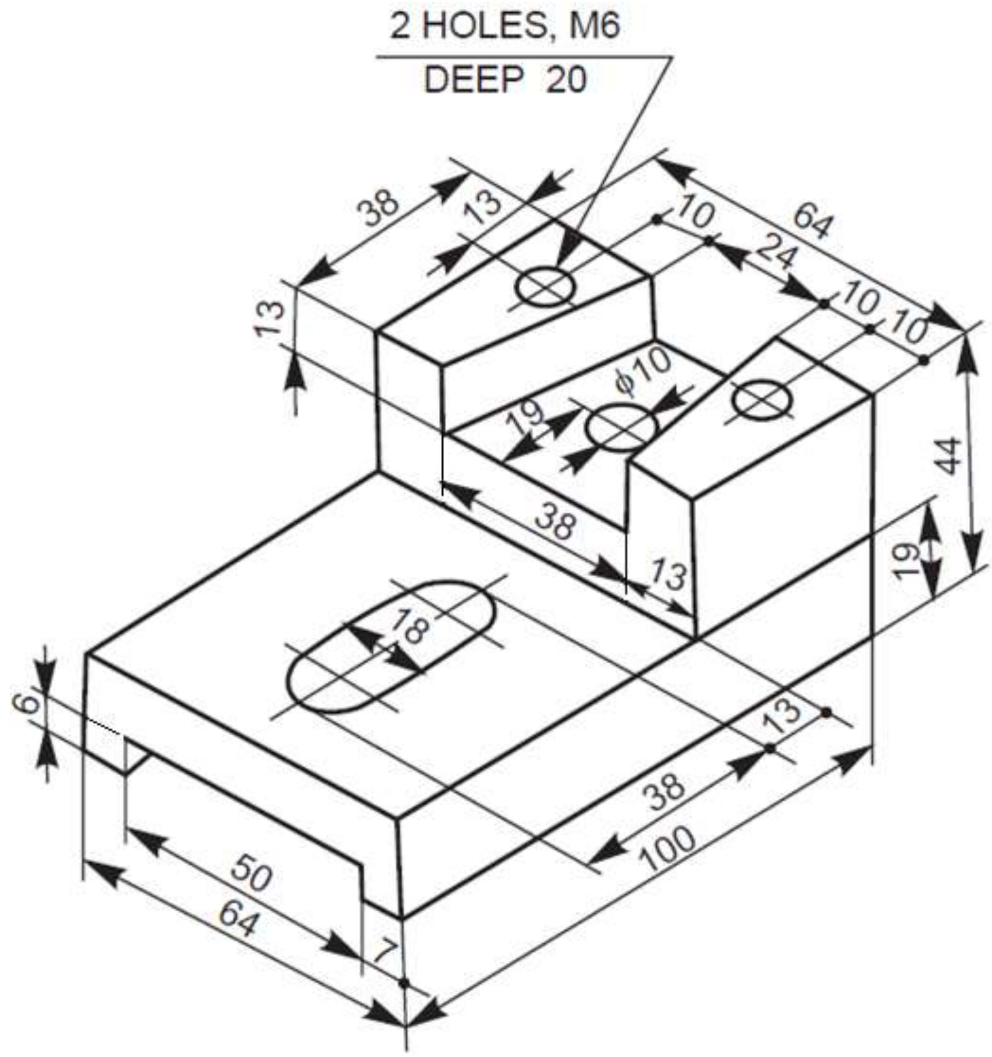
20
35
30
15
35
45
80
25
20
100
65




2 HOLES, M6
DEEP 20
38
13
10
64
24
10
10
13
44
φ10
19
38
13
19
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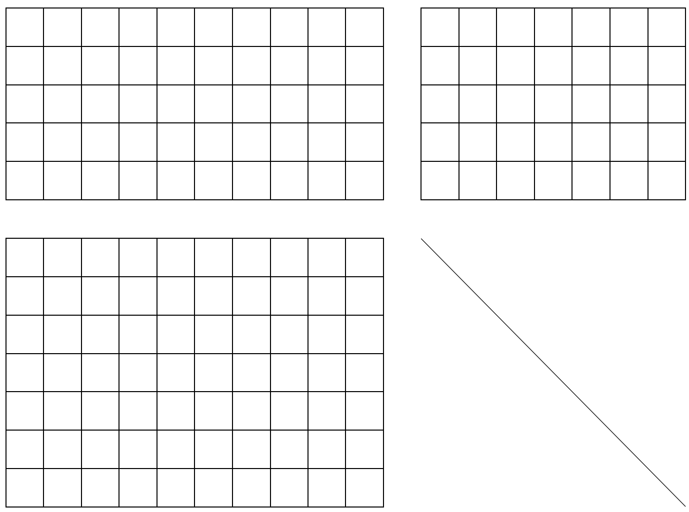
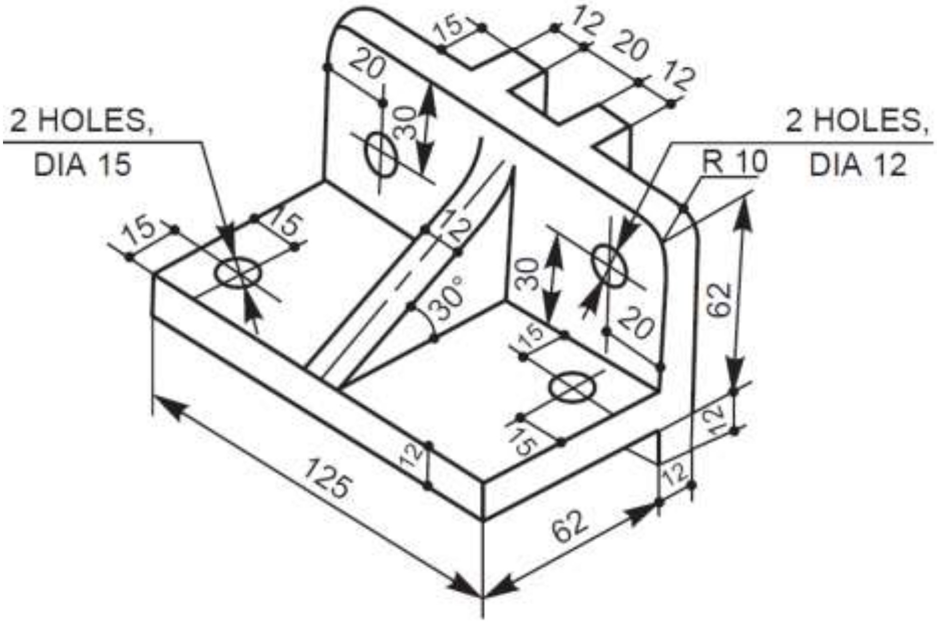
38
100
50
64
7




2 HOLES,
DIA 15
2 HOLES,
DIA 12
R 10
15
12
20
12
20
30
15
15
12
30°
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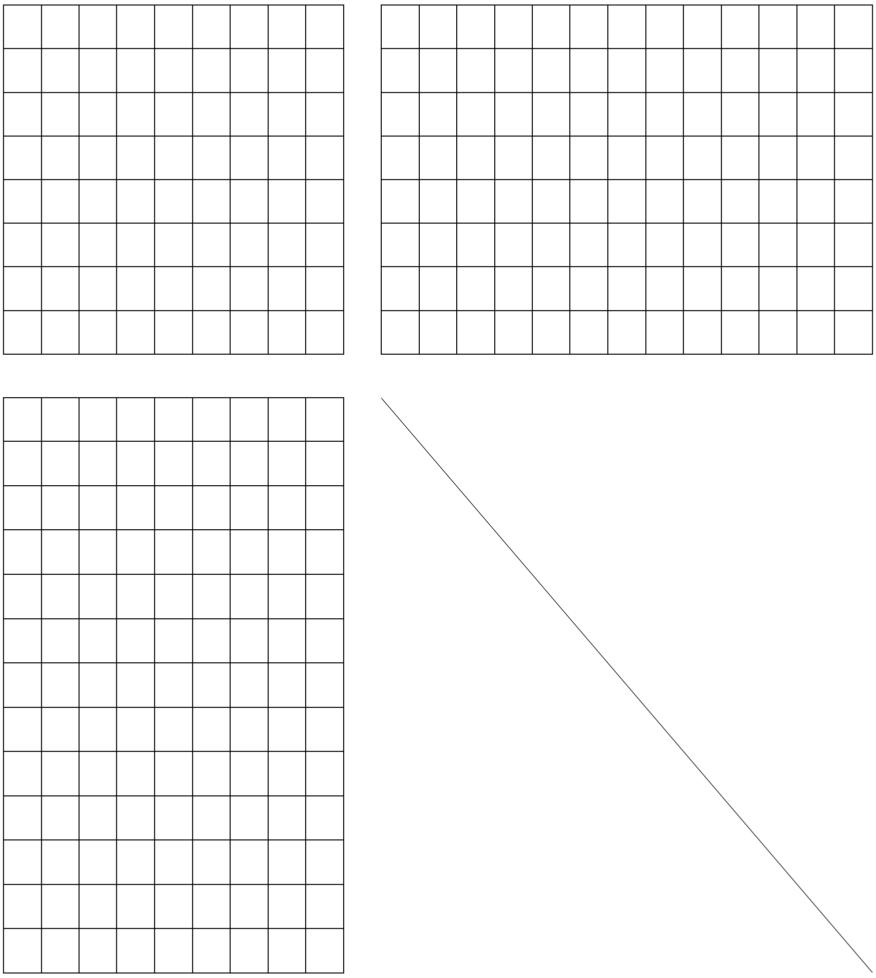
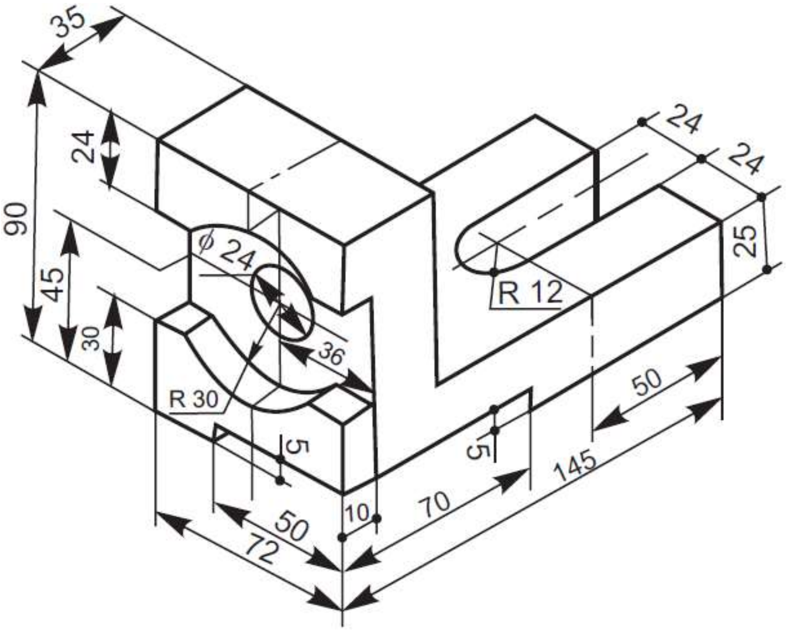
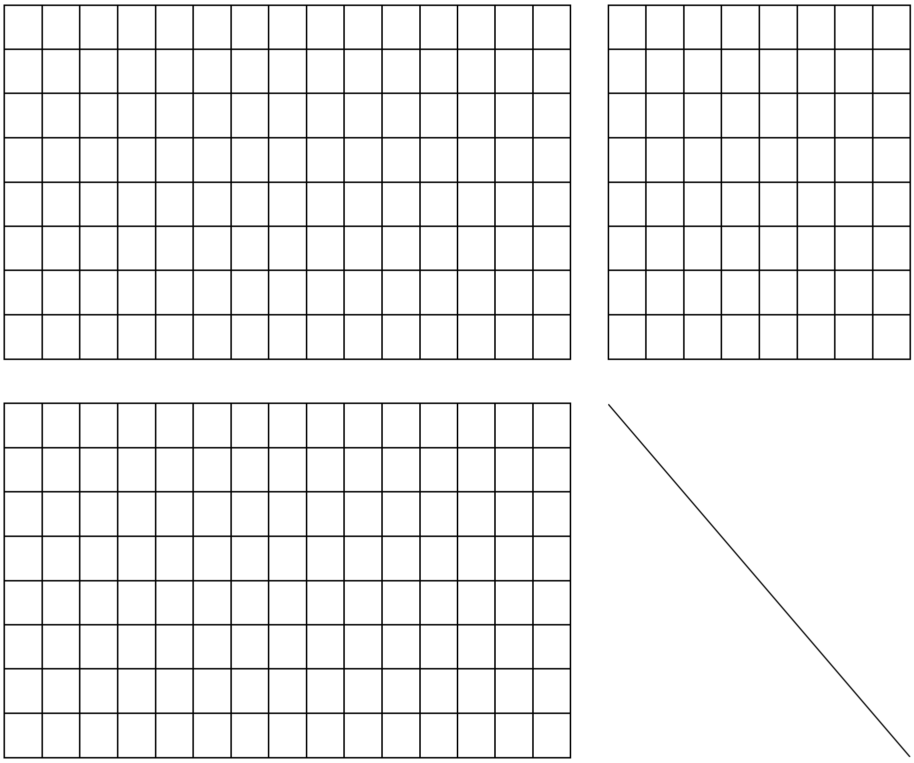
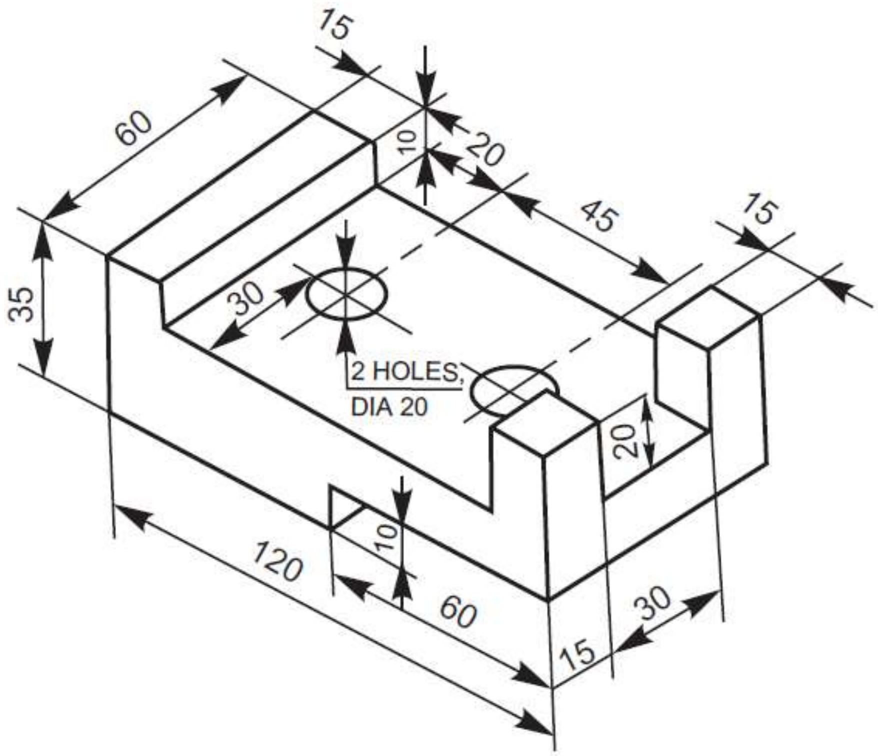
12
12
12
125
62




35
24
90
45
30
ϕ 24
36
R 30
5
50
72
10
70
145
50
5
R 12
24
24
25

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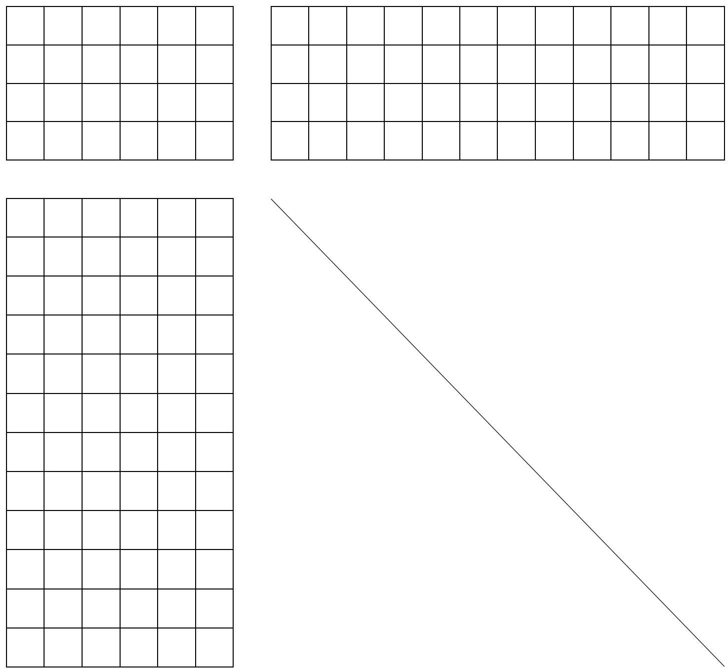
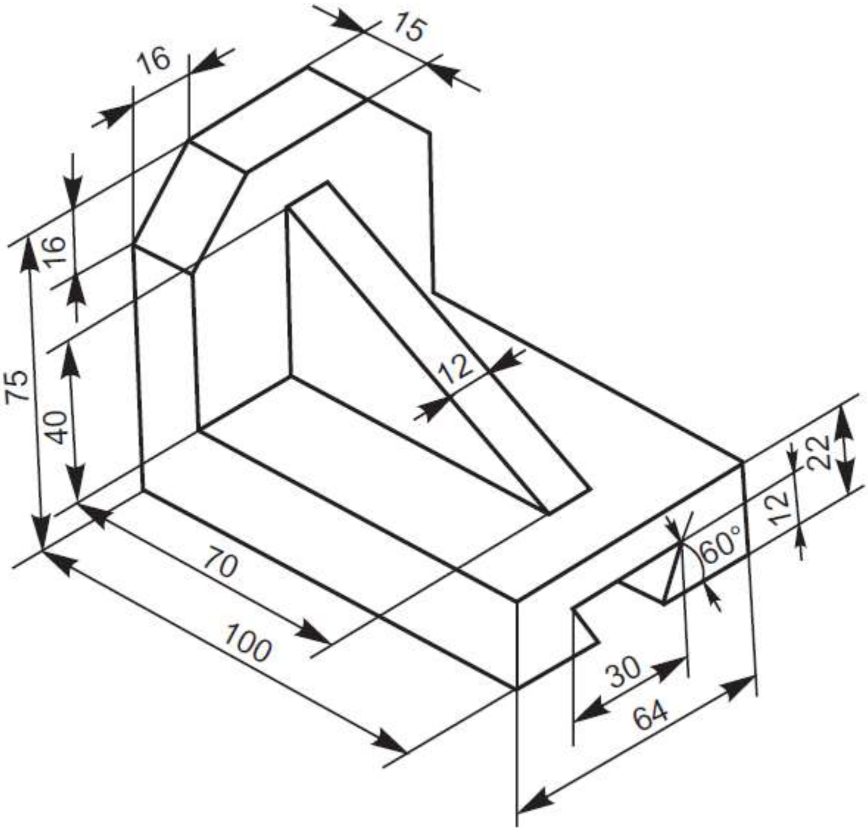
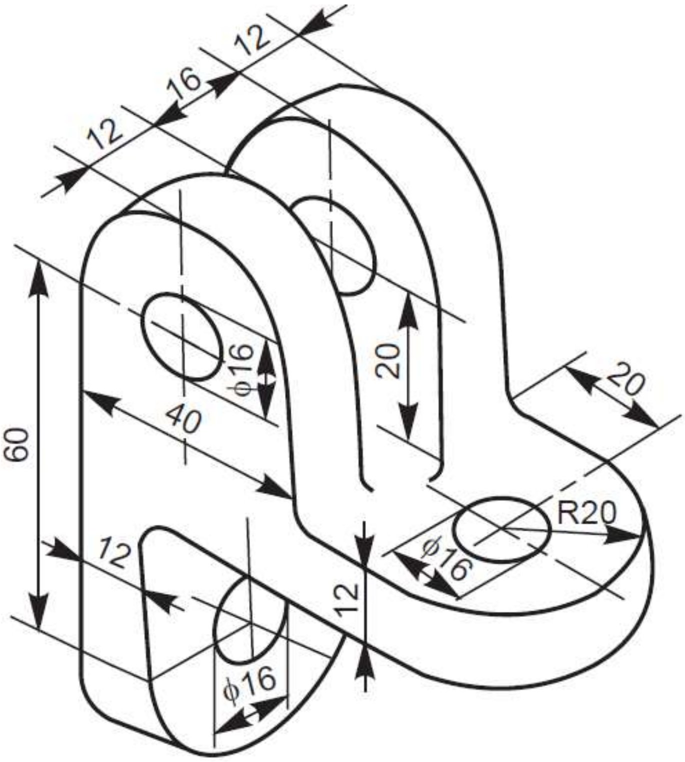
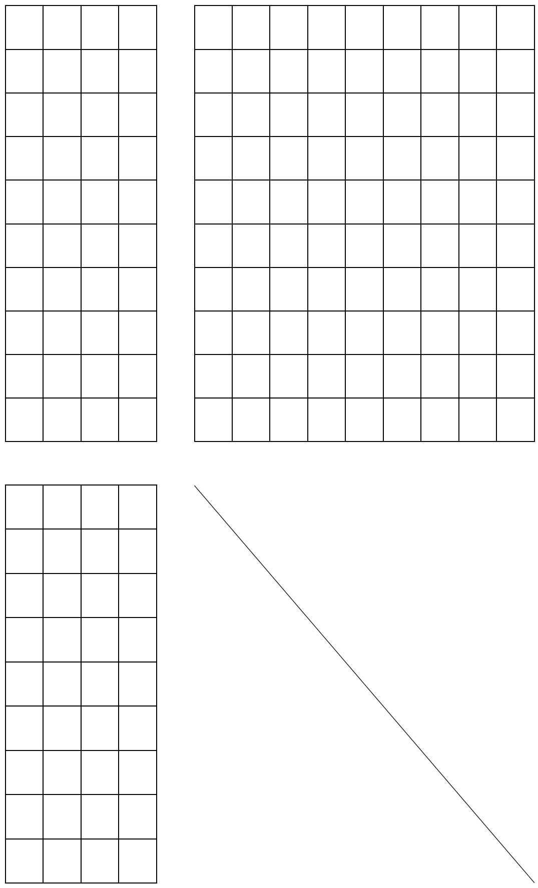
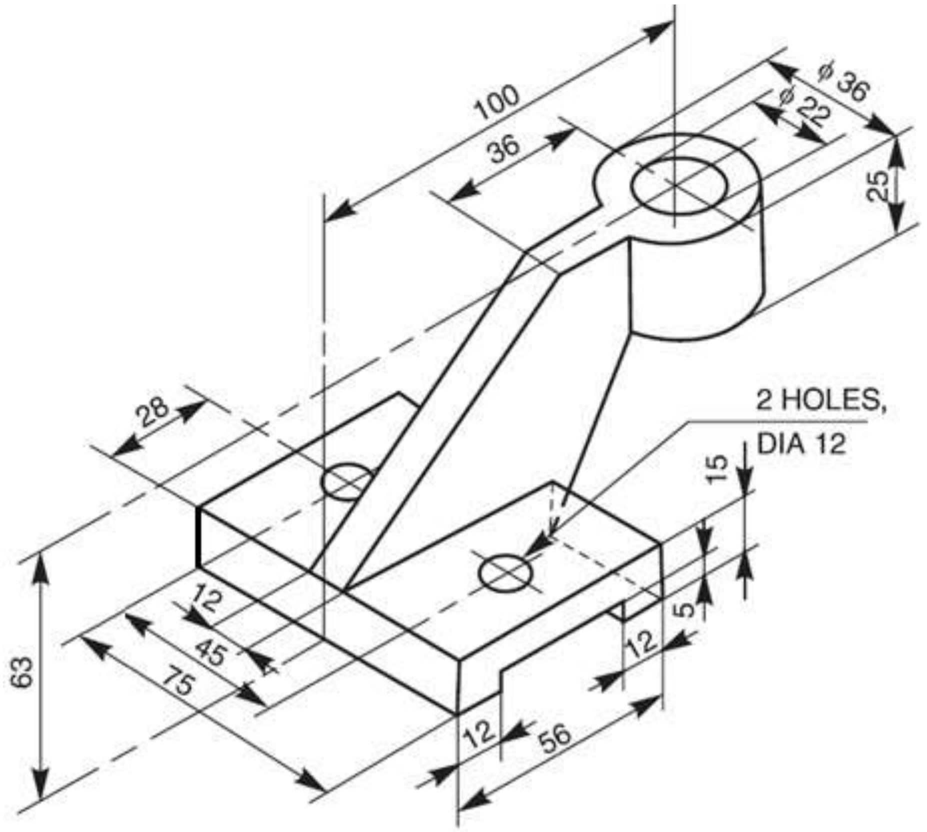
60
10
20
45
15
35
30
2 HOLES,
DIA 20
20
10
120
60
15
30

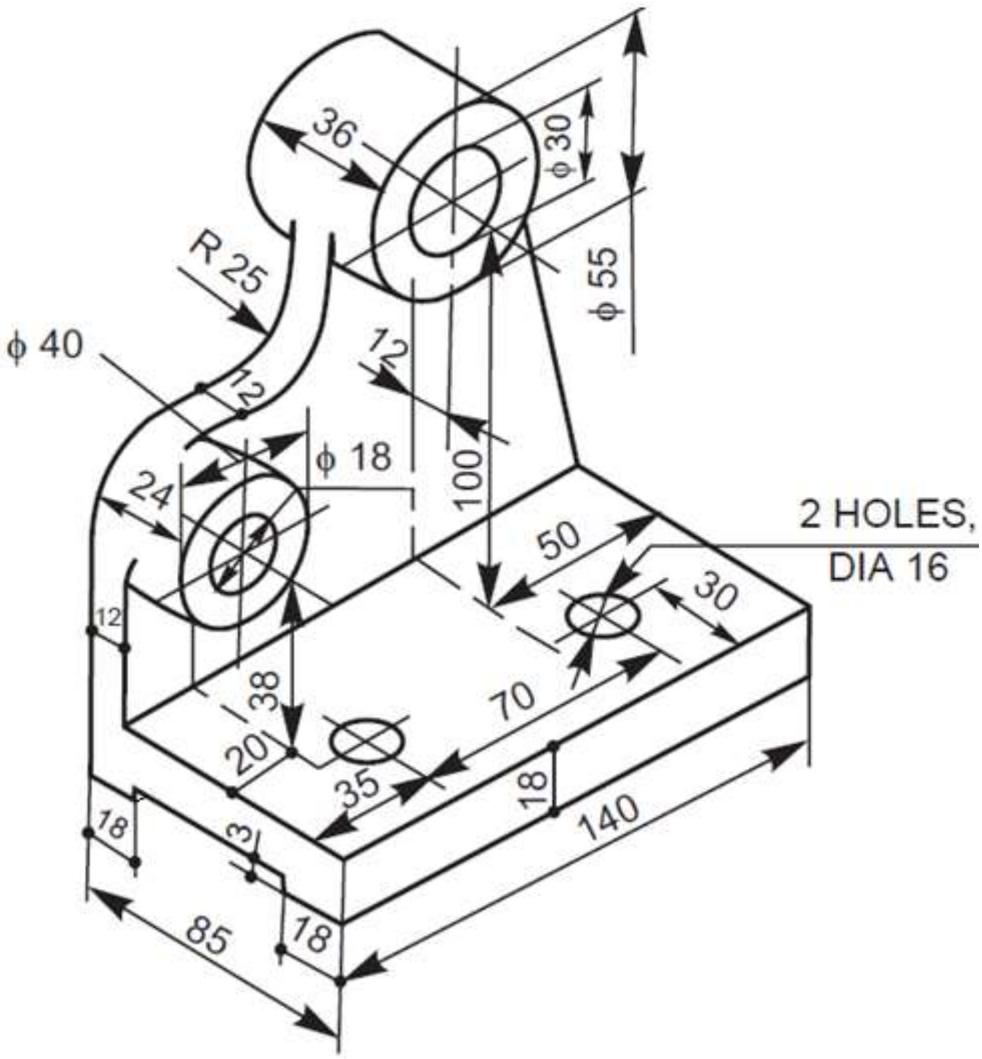


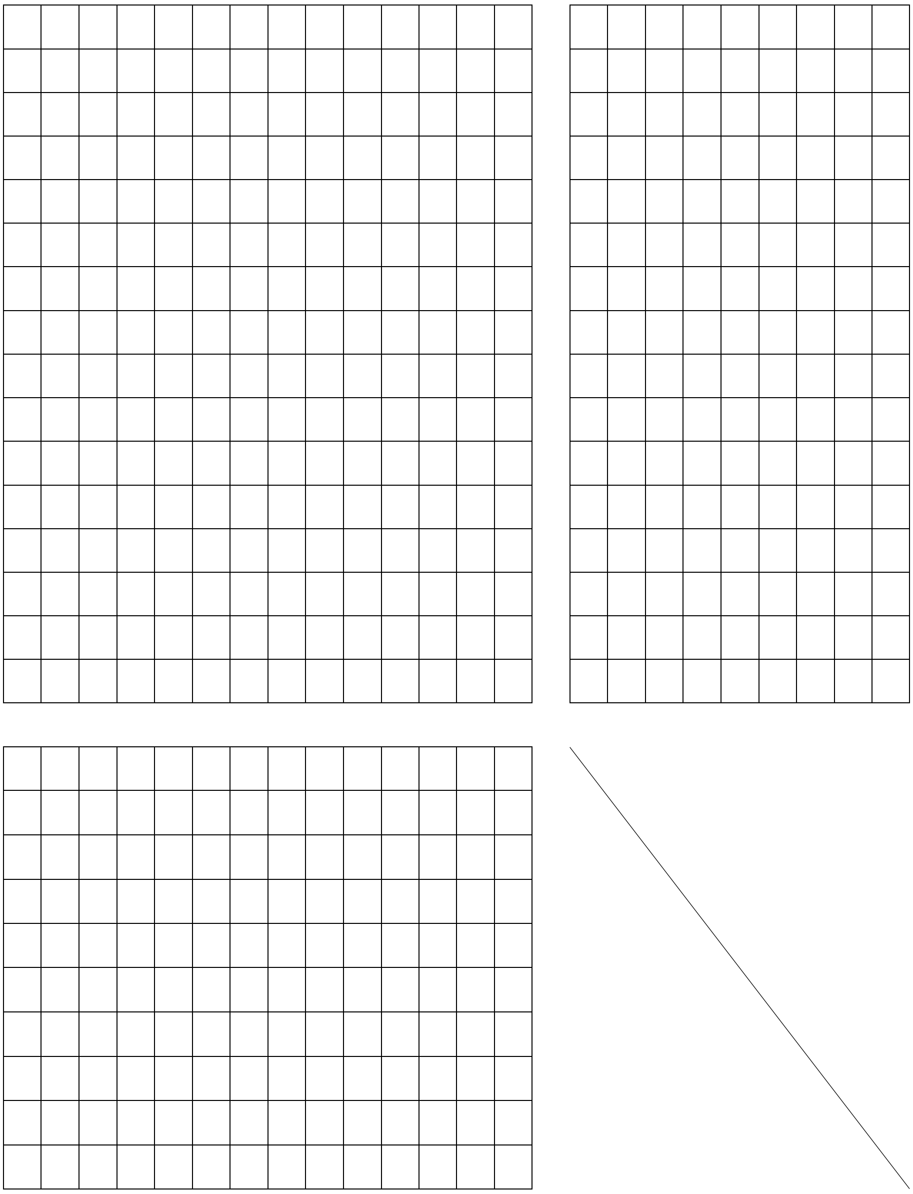

15
16
16
75
40
12
22
12
60°
70
100
30
64


12
16
12
20
ϕ16
40
60
20
R20
ϕ16
12
12
ϕ16



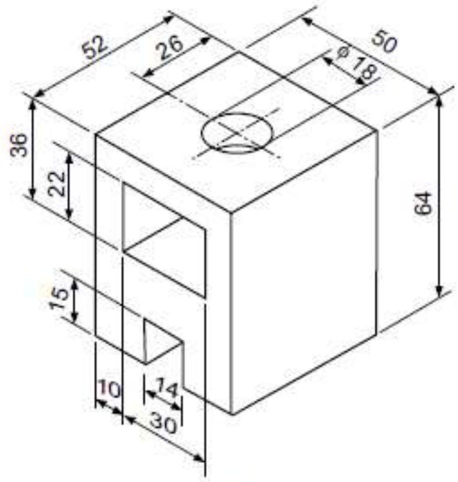

100
36
ϕ 36
ϕ 22
25
28
2 HOLES,
DIA 12
15
5
12
45
75
63
12
12
56
12


36
φ 30
φ 55
R 25
φ 40
12
12
φ 18
100
24
2 HOLES,
DIA 16
50
30
12
38
70
20
35
18
18
3
140
85
18

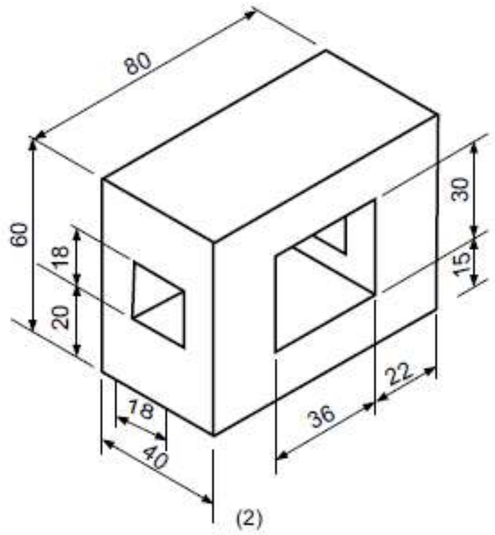


Additional Excirsises

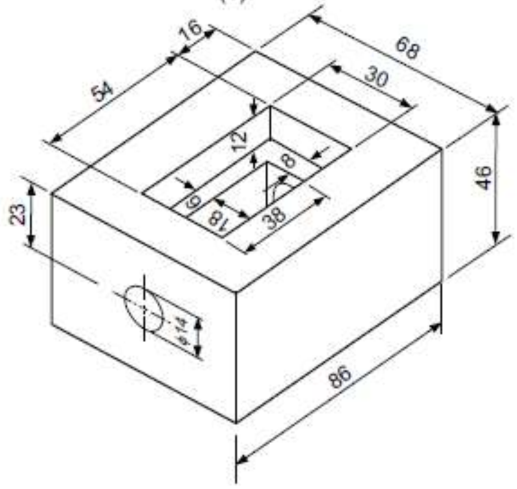
The pictorial views of different types of objects are shown in Fig. Draw the elevation, plan and side view, using the first angle projection.



(1)



(2)



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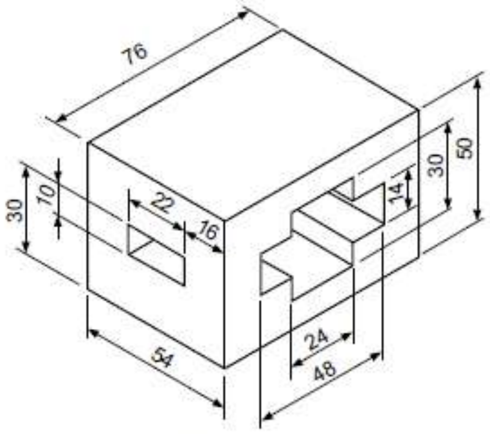

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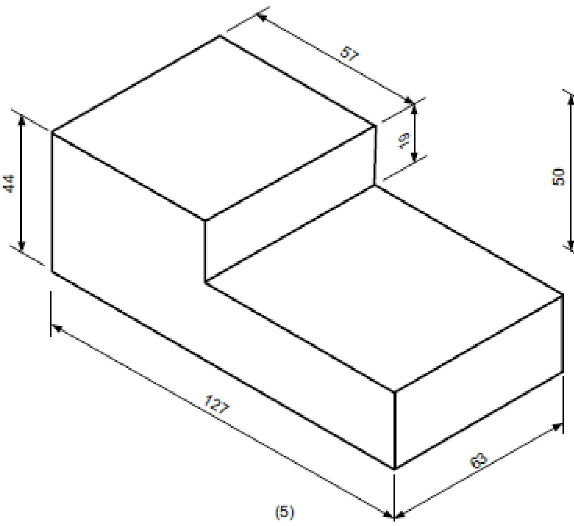
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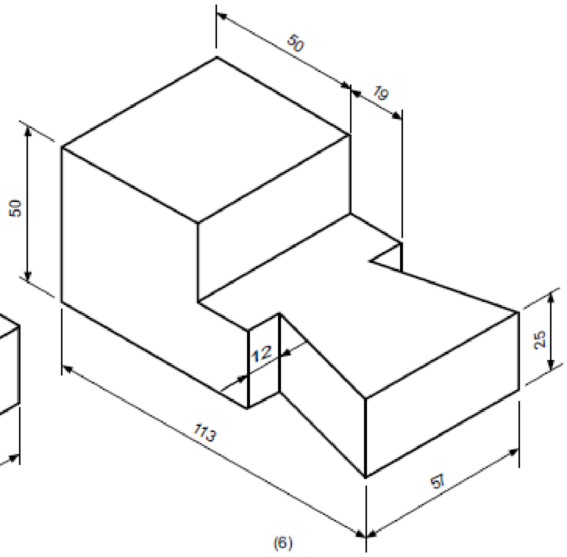
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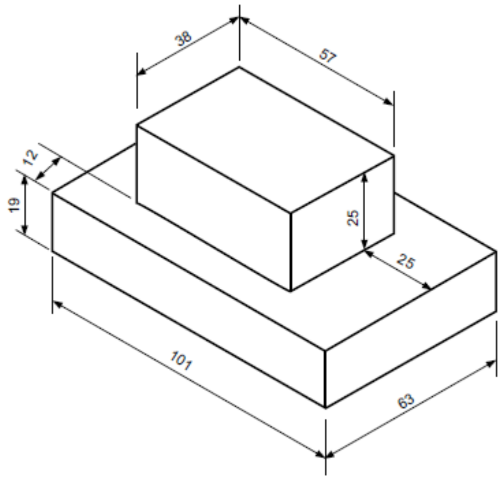
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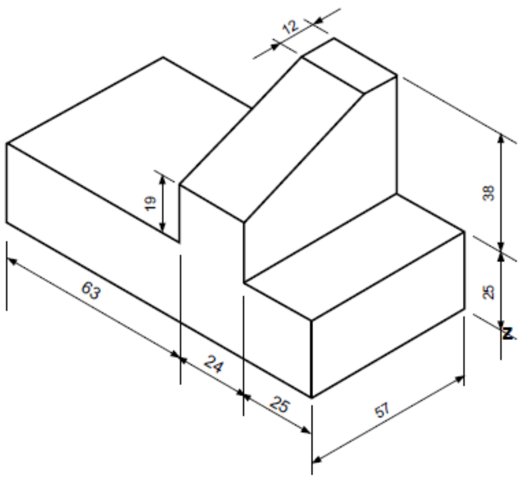
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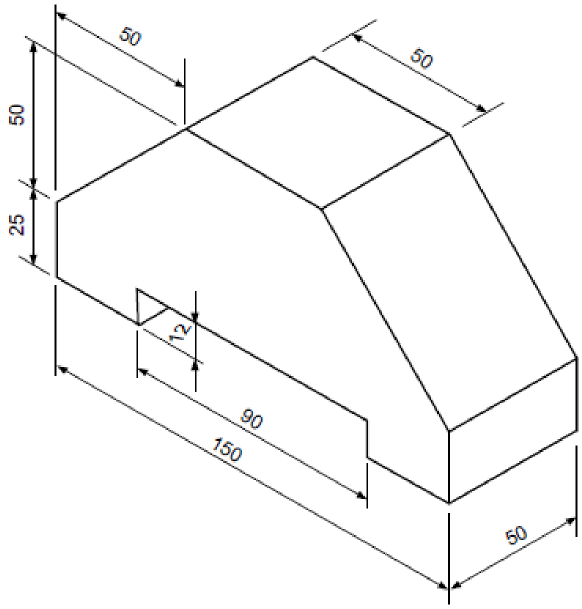
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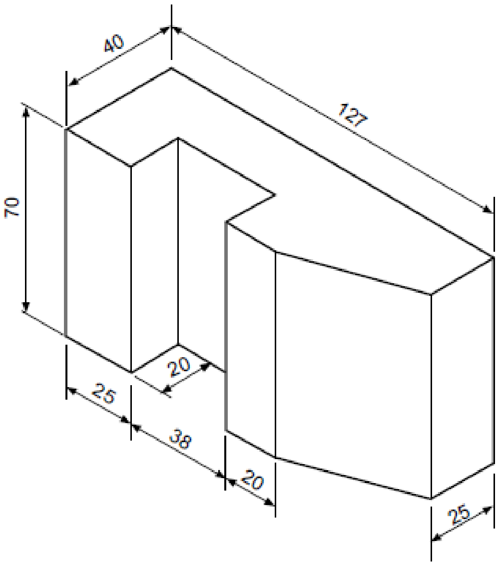
(9)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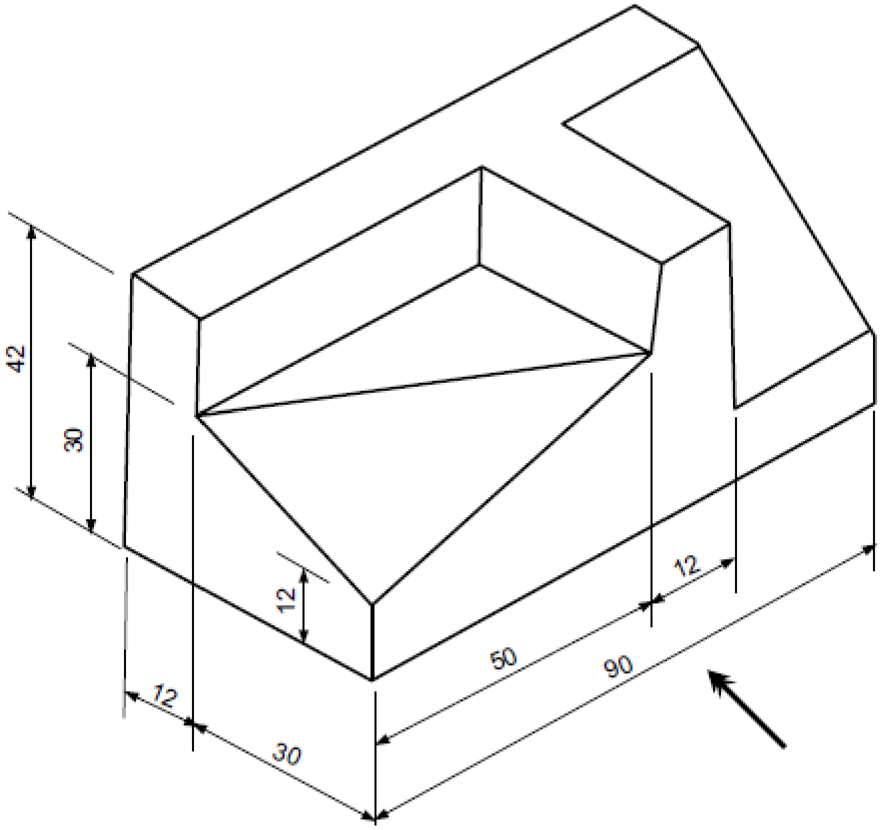
(10)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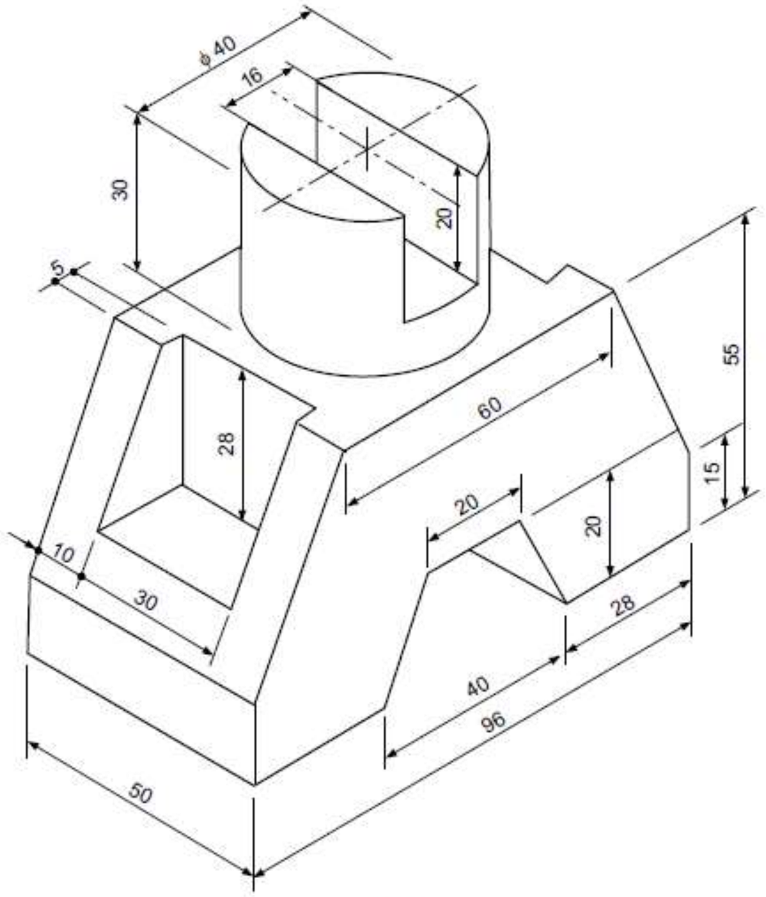
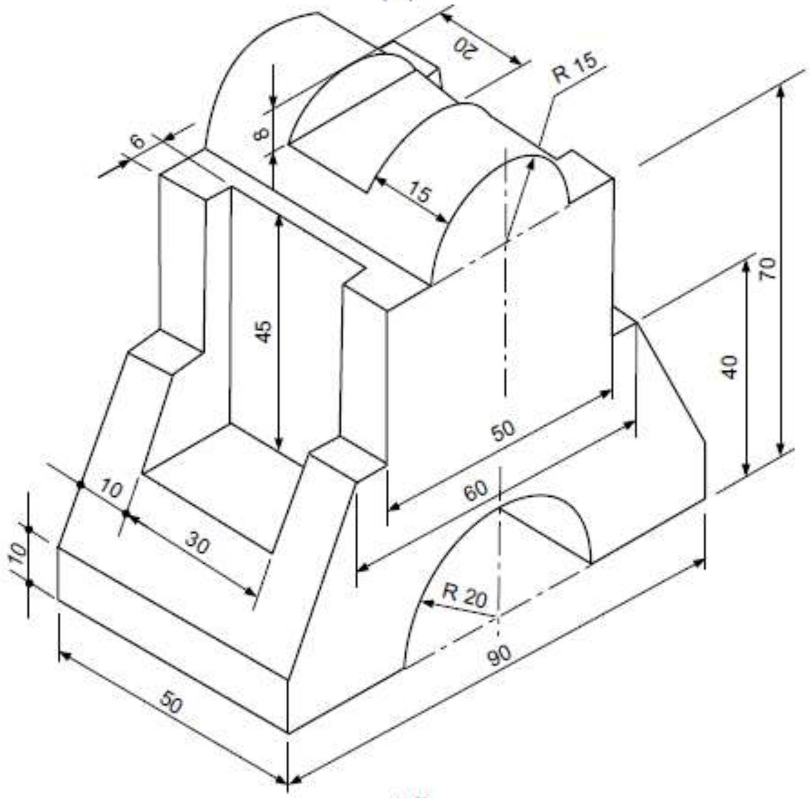

ϕ 40
16
30
20
5
28
55
60
15
20
20
10
30
28
40
96
50

(11)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R 15
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10
R 20
90
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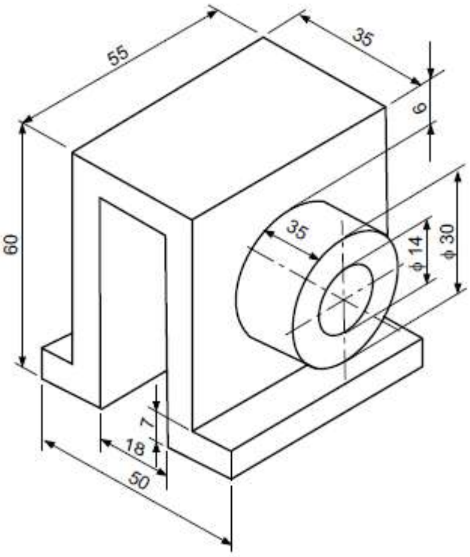
(12)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6
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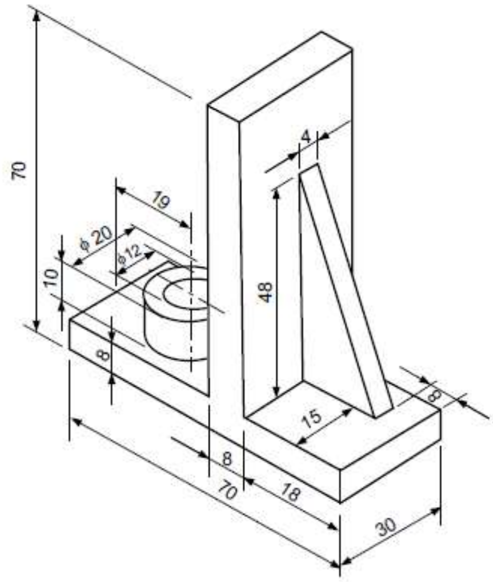
35
ϕ 14
ϕ 30
7
18
50

(13)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ϕ 20
ϕ12
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48
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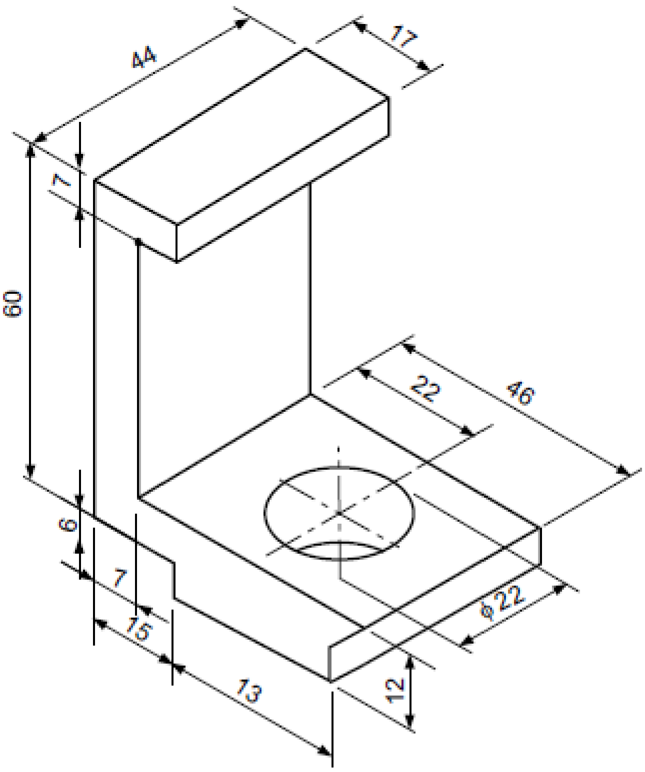
15
8
70
18
30

(14)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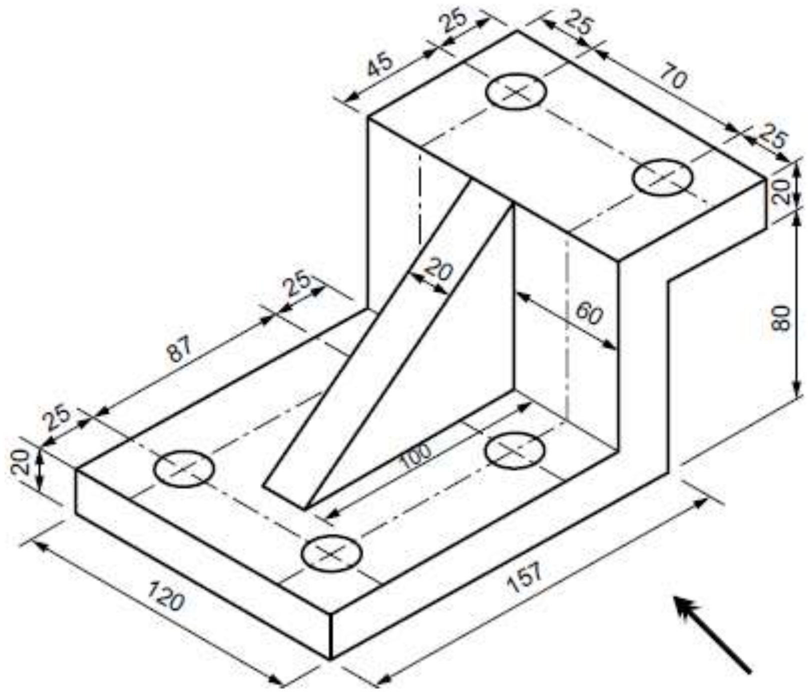
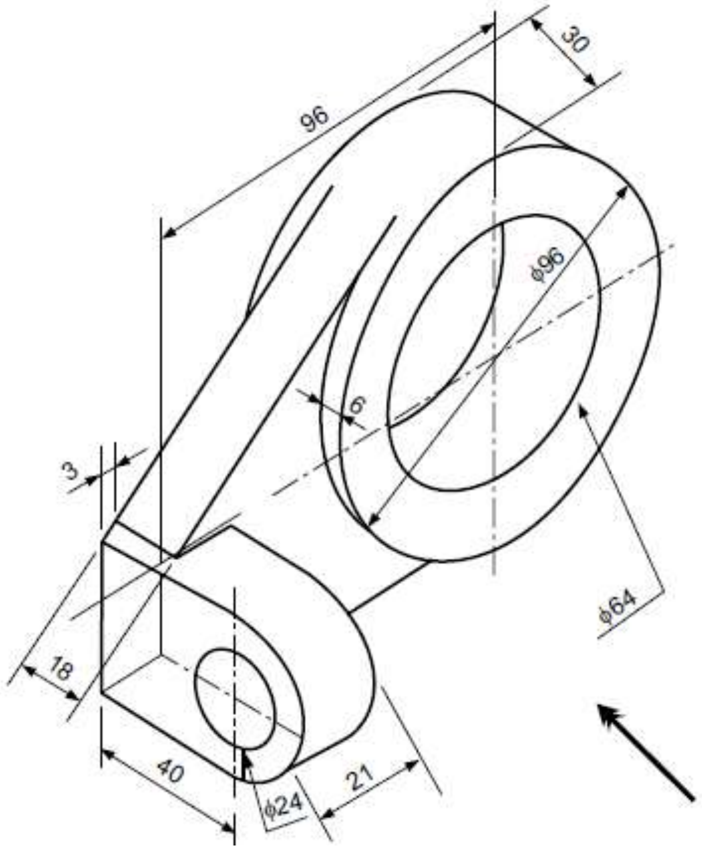
13
ϕ22
12

(15)





Dr. Eng. Maher Rashad El-Sadaty

Chapter 5

Isometric Projection

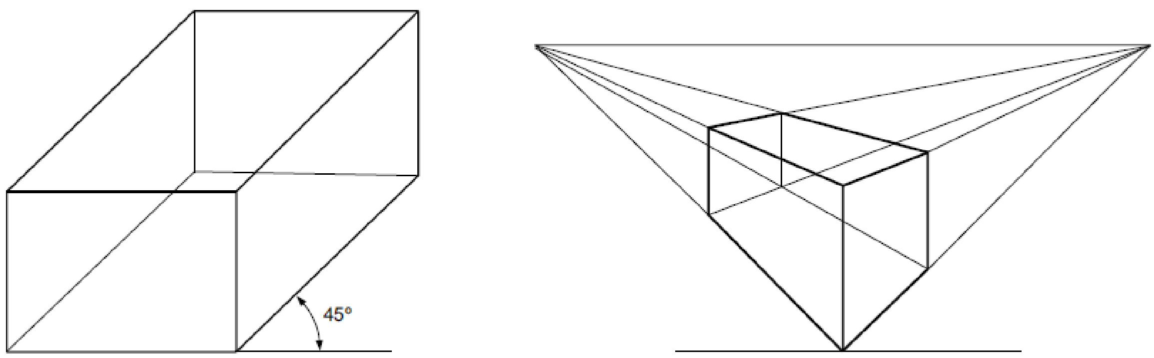
ISOMETRIC PROJECTION

5.1 INTRODUCTION

In engineering drawing, orthographic projection of a solid is best for showing the details of an object when a solid is resting in its simple position, the front view or top view taken separately, gives an incomplete idea of the object. Even, sometimes an experienced engineer gets puzzled when studying the orthographic projection of complicated parts. To avoid this confusion, a pictorial projection is the best method to show the object in one view only. Basically, pictorial projection represents three dimensional shape of an object and represents real things in one view only, which indicates length, breadth and height of the object. Therefore, the object is easily visualized from a pictorial projection than from its orthographic projection. The pictorial projection may be divided as:

1. Oblique projection (Fig. 5.1(i))
2. Perspective projection (Fig. 5.1(ii))
3. Axonometric projection.

In this chapter you will learn about the axonometric projections, which are commonly used in industries.



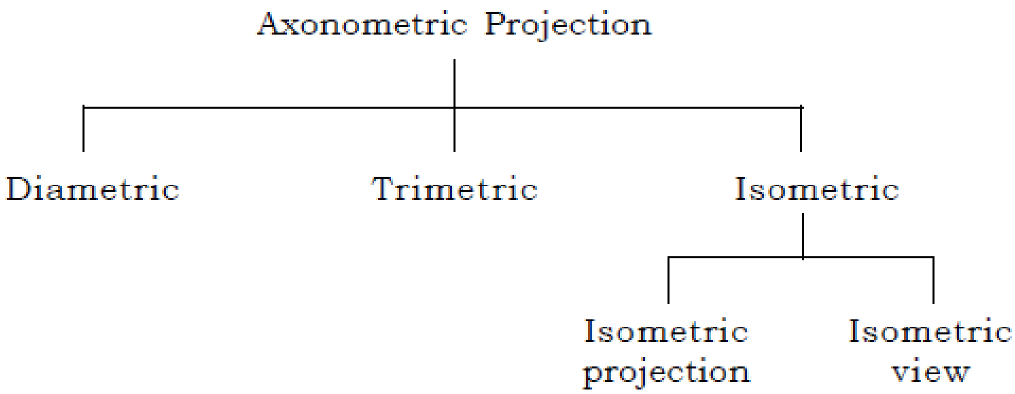
(i) Oblique Projection   (ii) Perspective Projection

Fig. 5.1

5.2 Axonometric Projection

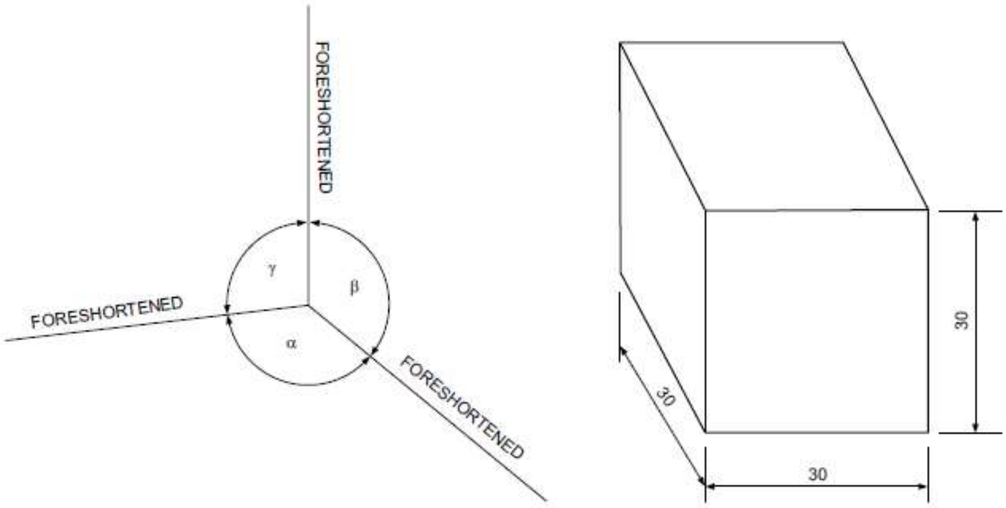
The word ‘axonometric projection means measuring along axis in which “axon” means axis while metron means measuring. Axonometric projections are commonly used to draw mechanical parts of an object for the clear picture of an object which are visualized from the orthographic projection. In this projection the object can be drawn at different angles and having the different length of edges.

Axonometric projections are classified as follows:



1. **Diametric Projection**

In diametric projection, only two faces are making equal angles, while the third angle is different one with the projection of plane as shown in **Fig.5.2**



2. Trimetric Projection

In trimetric projection, all the three faces are making different angles with the plane of projection as shown in Fig.5.3

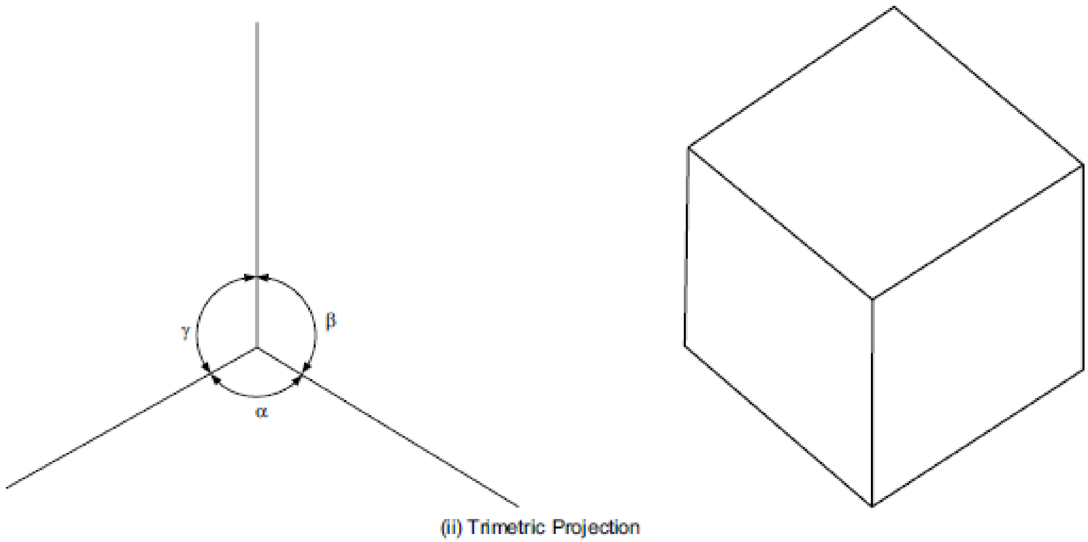
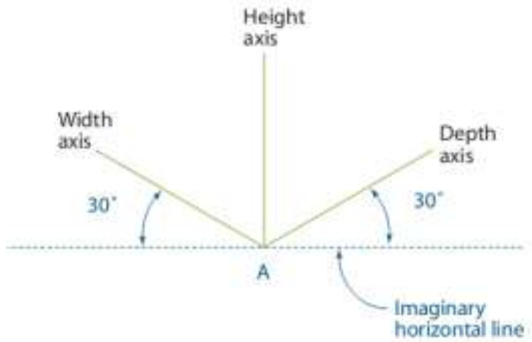


Fig.5.3

3. Isometric

In isometric, all the three faces are making equal angles with the plane of projectionas shown in Fig. It is a type of pictorial projection which is taken from the Greek word. ISO means equal METRON means measure. So, isometric means equal measure. Isometric are further divided into two types:

1. Isometric projection
2. Isometric view



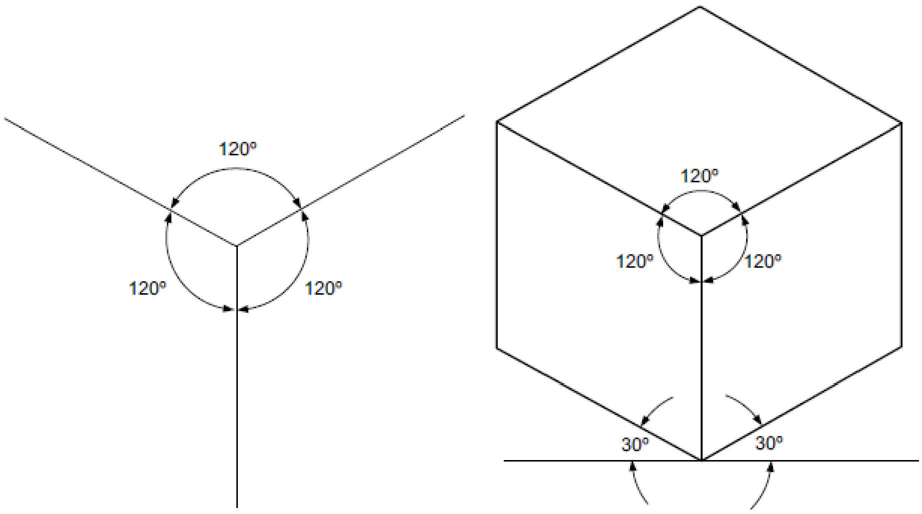


Fig.5.4

5.3 ISOMETRIC PROJECTION

The isometric projection is the most common pictorial representation used in industries where visualization of the three dimensions of a solid is required. The isometric projection, obtained on a plane when the object is so placed that all the three axes make equal angle with the plane of projection. In isometric projection, the dimensions are reduced by the isometric scale and these dimensions are reduced by multiplying 0.816. The principle involved in drawing an isometric projection can best be explained by drawing orthographic projection of a cube.

Example: A cube of 35 mm is placed on one of its corners on the ground with a solid diagonal perpendicular to V.P. It will be seen that the front view may be used to obtained will give the isometric projection of a given cube as shown in Fig. 5.5.

Isometric projections are commonly used in mechanical, electrical, chemical, automobile engineering to show the machine components.

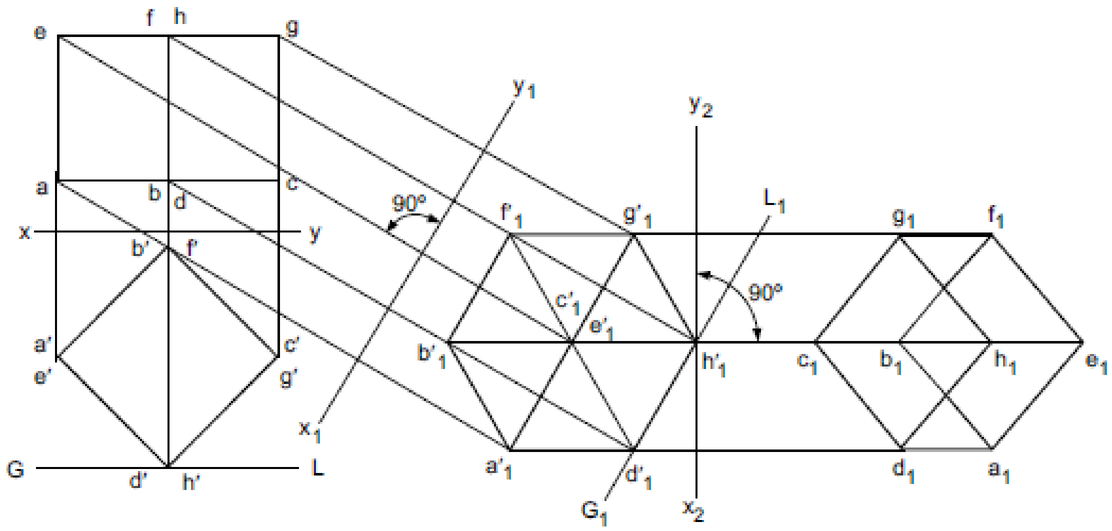


Fig.5.5

5.4 ISOMETRIC VIEW

The measurements of the size of an object are taken with the actual scale without reducing dimension by isometric scale. In isometric view, we are interested in the shape of an object rather than its size. So, to avoid confusion, the view drawn with the actual scale is known as isometric view as shown in Fig. 5.6.

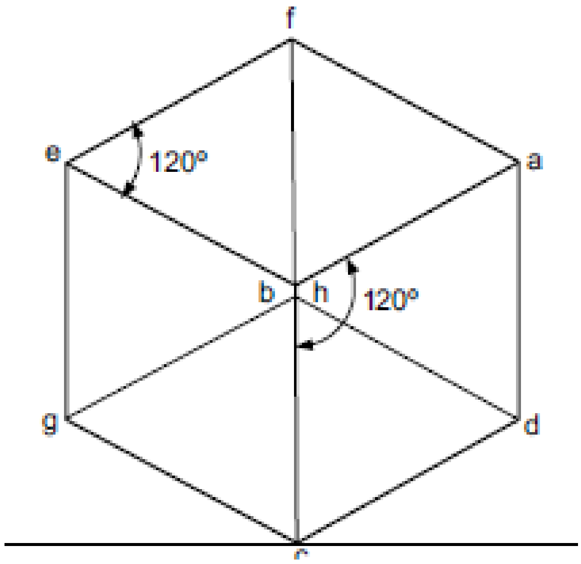


Fig.5.6

5.5 Methods of making an isometric projection or view

There are two methods can be used for making an isometric projection or view, if an object contains a number of non-isometric lines are as follows:

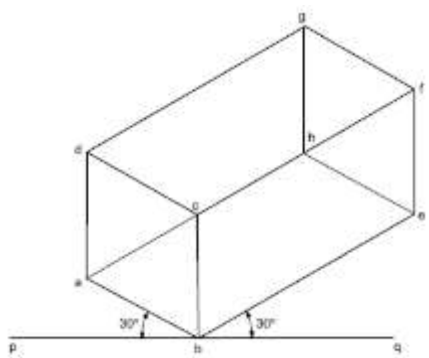
1. **Box Method (Enclosing Box)**

The isometric projection of a solid, such as cube, square or a rectangular prism are drawn directly when their edges are parallel to the three isometric axes

Step by step construction of box method is given below:

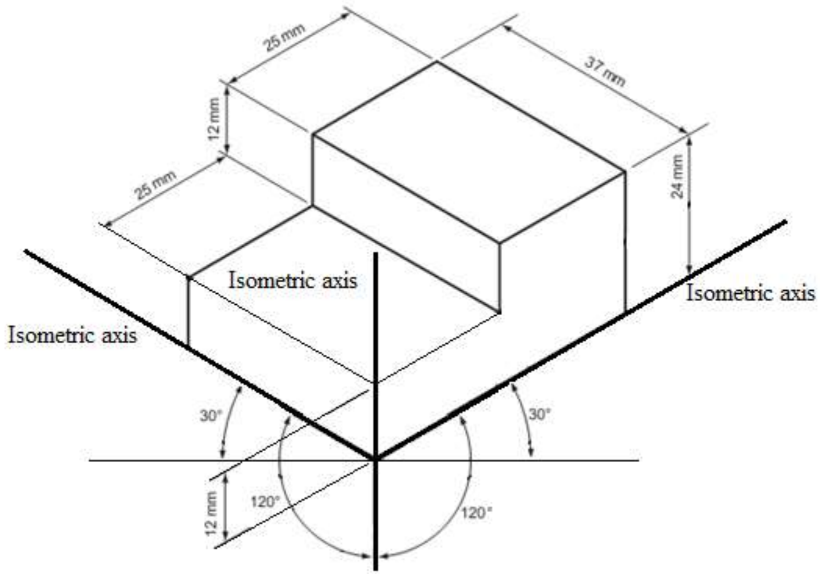
1. Draw a horizontal line *pq* and take a point *b* on the line *pq*. Through *b* draw the three isometric axes *ba*, *be* and *bc* with the help of minidrafter or set square, where *bc* is perpendicular to *pq* line. $\angle ebq = \angle abp = 30^o$.
2. Mark ba, be and bc the length, breadth and height of the object along the axis ba, be and bc respectively.
3. Through a, draw a line ad parallel to bc and through c, draw a line cd parallel to ba intersecting each other at a point d. Now through point e, draw a line ef parallel to bc and through c, draw a line cf parallel to be, which intersect each other at a point f.

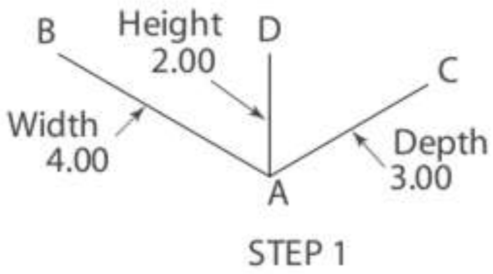
Similarly, through d draw a line dg parallel to be and through f draw a line fg parallel to ba, intersecting at g. Now complete the rectangular block.

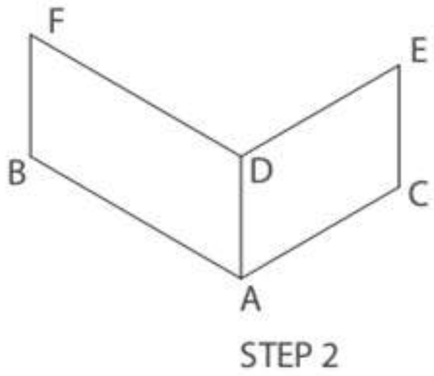


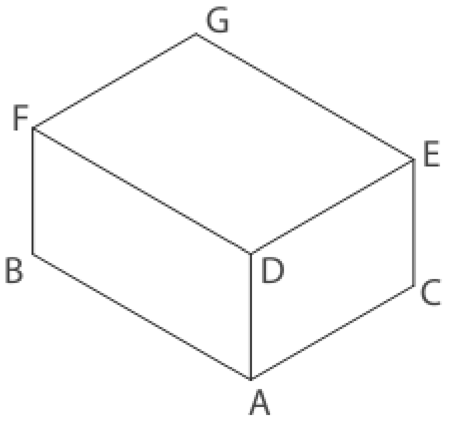
BOX METHOD

Example:

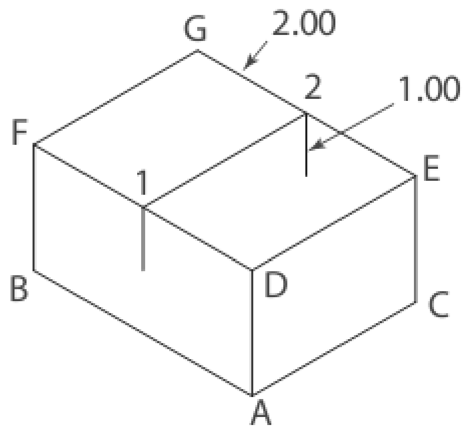

25 mm
37 mm
12 mm
25 mm
24 mm
Isometric axis
Isometric axis
Isometric axis
30°
30°
120°
120°
12 mm


B
Height
2.00
D
C
Width
4.00
Depth
3.00
A
STEP 1


F
E
B
D
C
A
STEP 2



STEP 3



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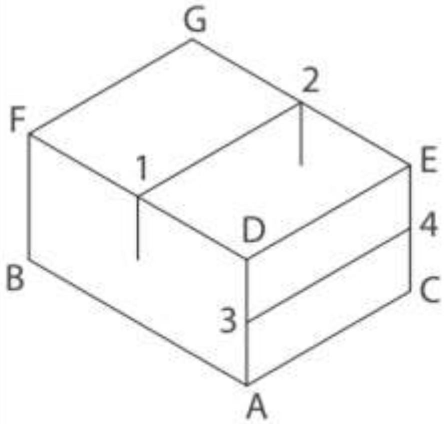

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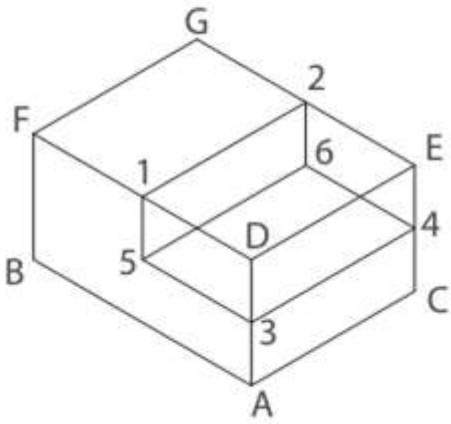

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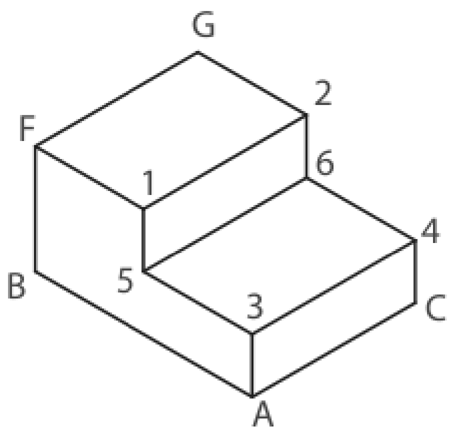


STEP 7

2. Off-Set Method

This method is used to draw isometric projection of the object which has neither nonisometric lines nor their ends lie in isometric plane. The isometric projection of pyramids and cones are generally drawn by co-ordinates or off-set method.

Step by step construction of off-set method is given below:

1. Draw the top and front view of the hexagonal pyramid.

2. Enclose the hexagon in a rectangle *pqrs* in the top view.

3. Draw the isometric view of the base of the pyramid in the parallelogram PQRS.

4. FC is an isometric line on which O1 lies. Hence mark O1 on the isometric line FC such that FO1 = *y*.

5. From O1 draw a vertical line and mark the apex O.

6. Join O with all the corners of the base of pyramid and complete the isometric view as shown in Fig. 5.8.

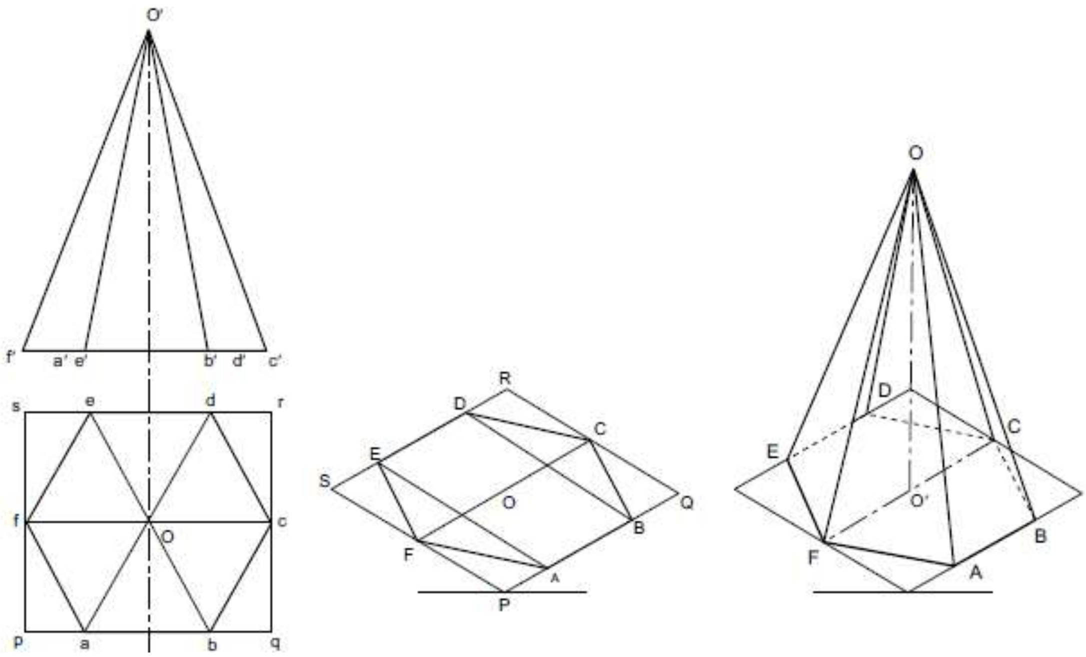
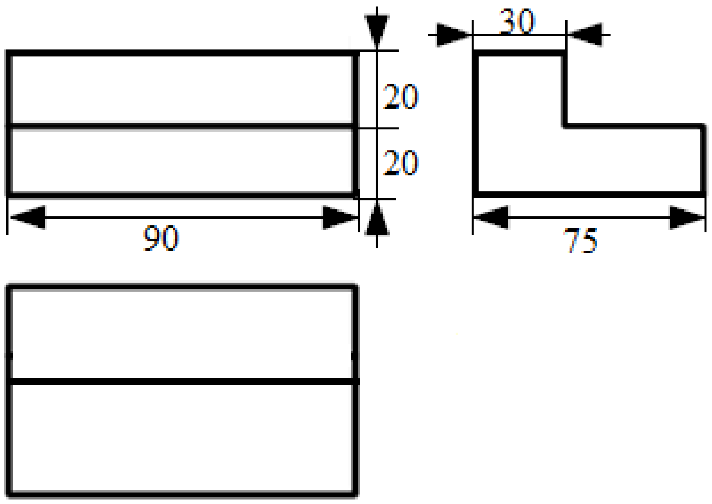
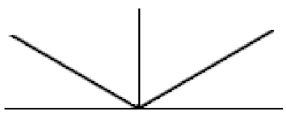


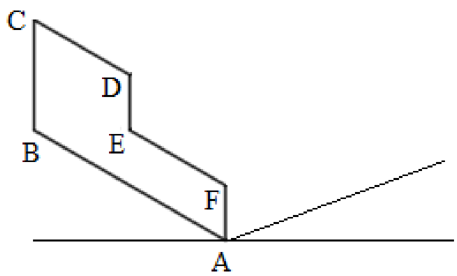
Fig. 5.8

Example:

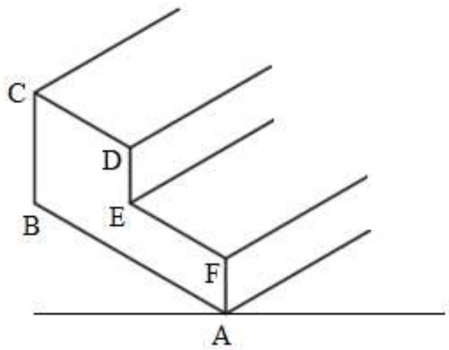




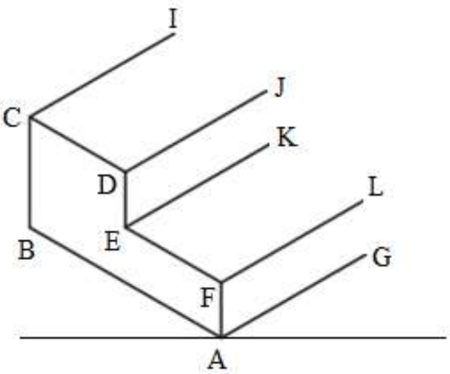
(a)



(b)

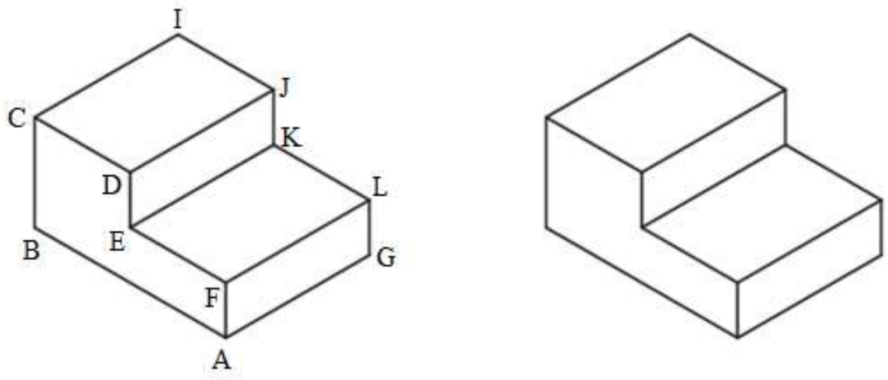


(c)



(d)

Fig. 5.8

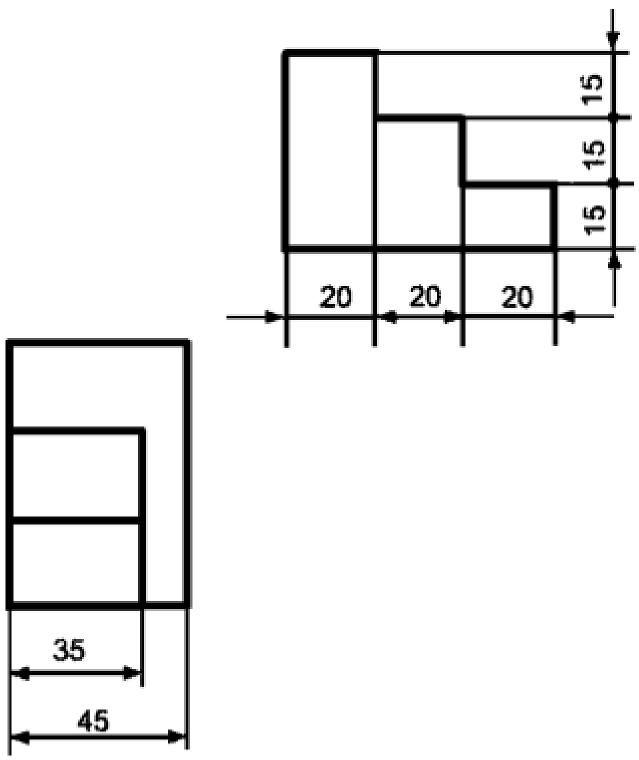


(e) (f)

Fig. 5.8

5.6 Isometric and Non-Isometric lines

1. **Isometric lines (Fig.5.9)**



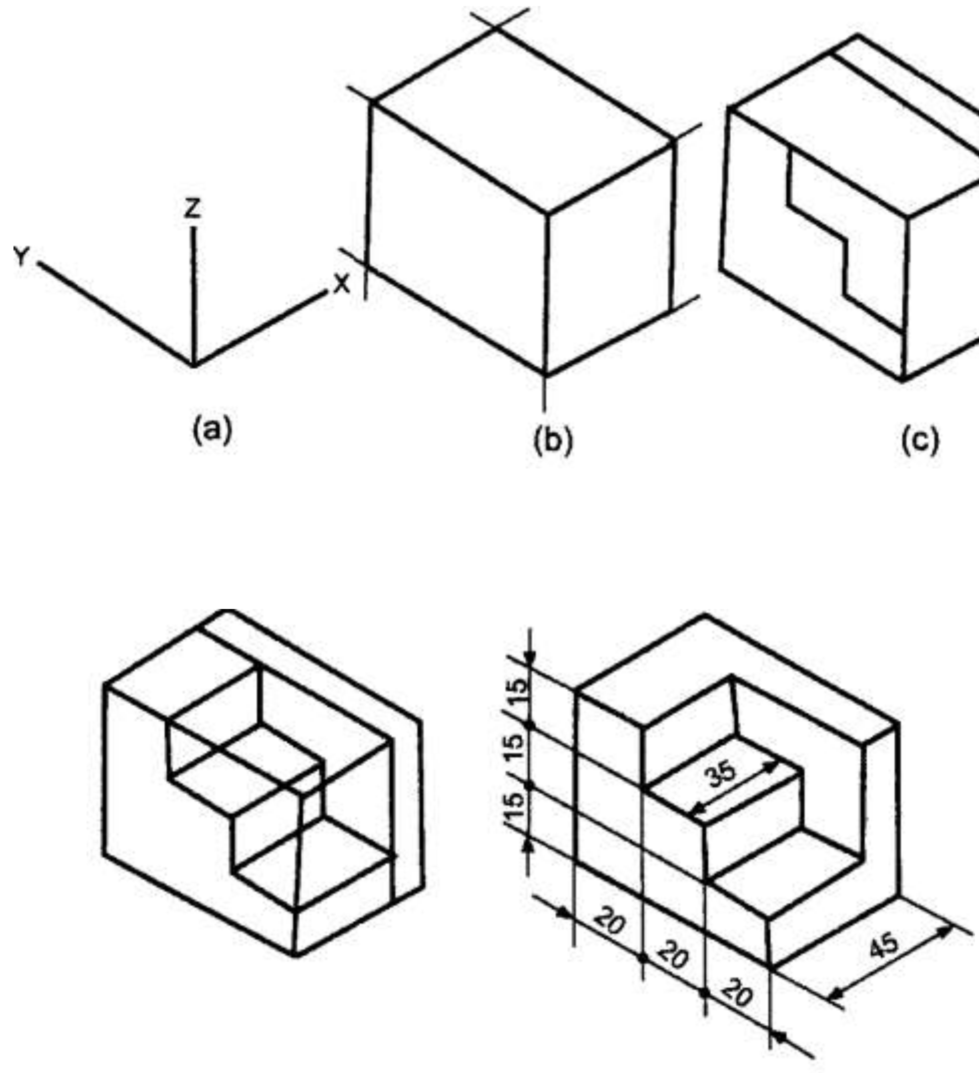
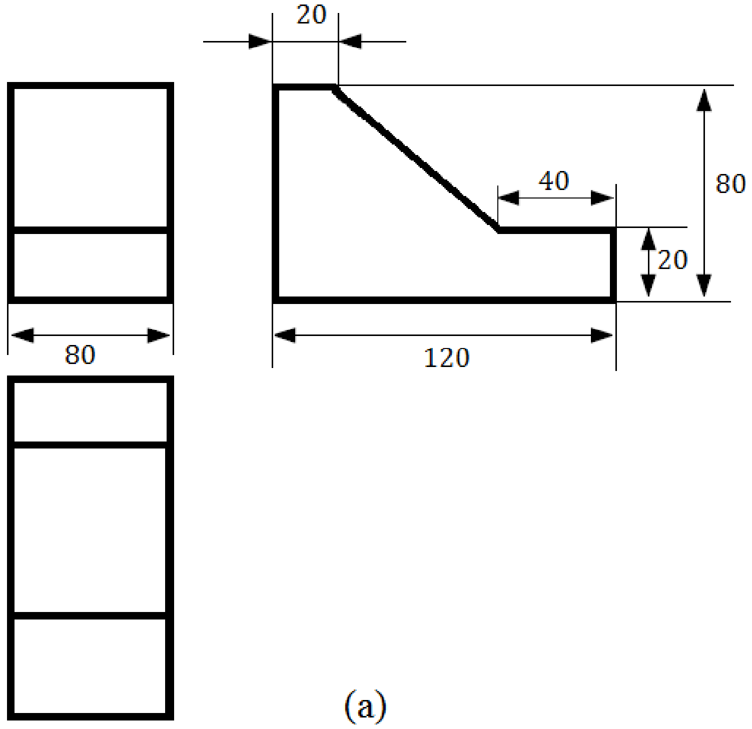
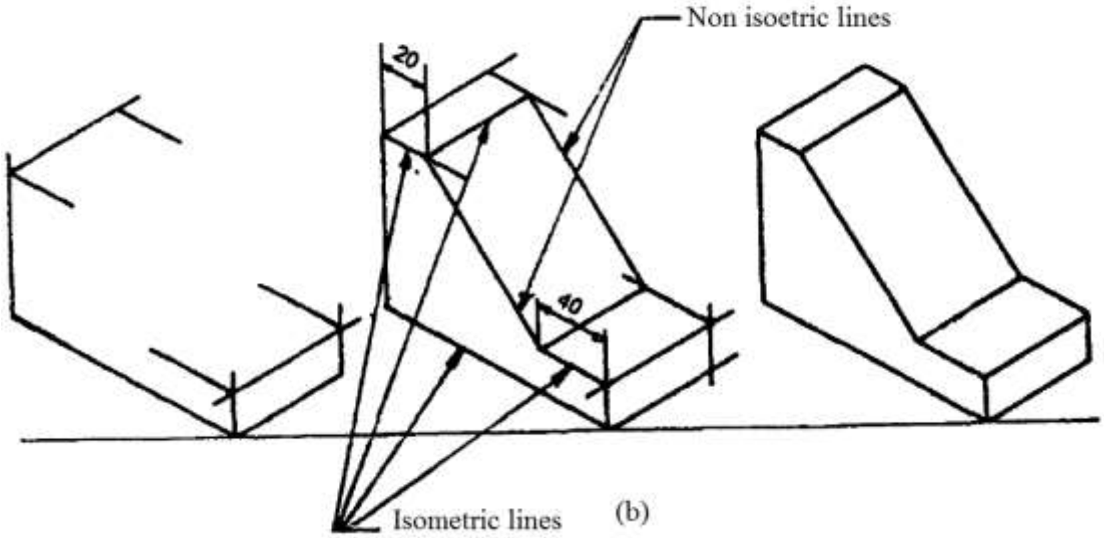


Fig. 5.9

2. Non-Ismetric line (Fig. 5.10)



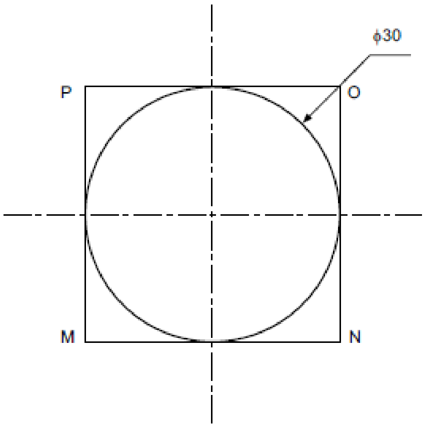
(a)



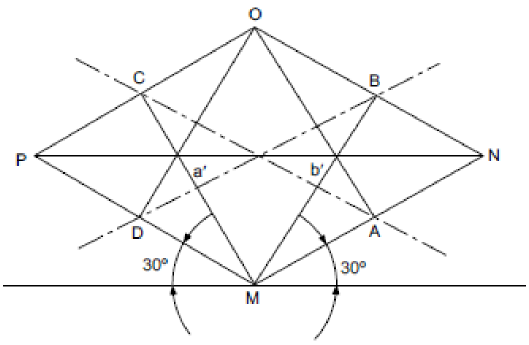
(b)

5.7 ISOMETRIC PROJECTION OF A CIRCLE

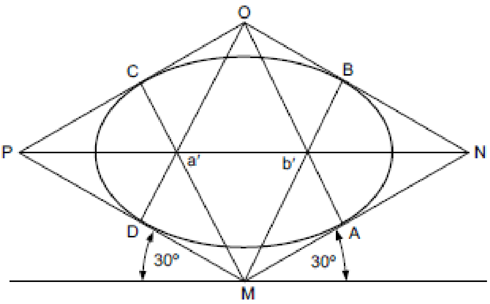
In isometric projection or view of a circle is seen like an egg shape or like an ellipse. It may be drawn by enclosing it in a square and locating number of points on it by offset-method. Four centre method is commonly used to draw an isometric projection or view of a circle.



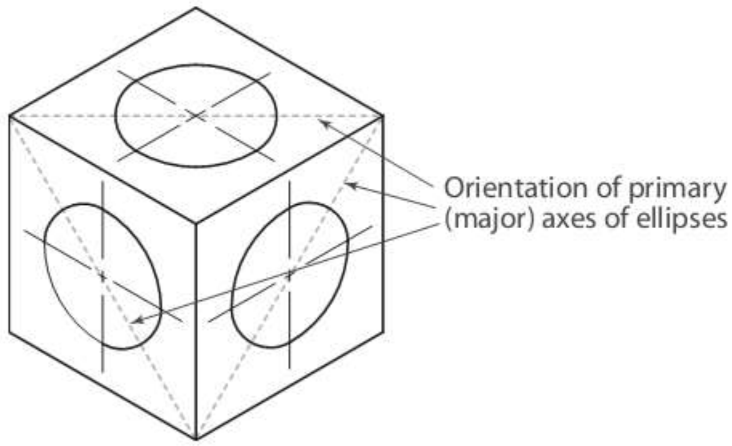
(i)



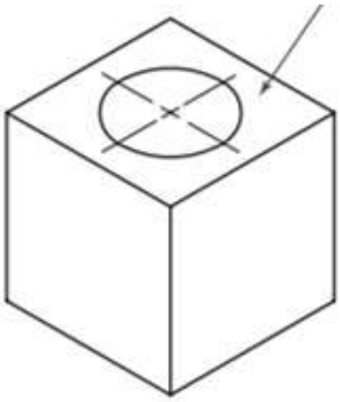
(ii)

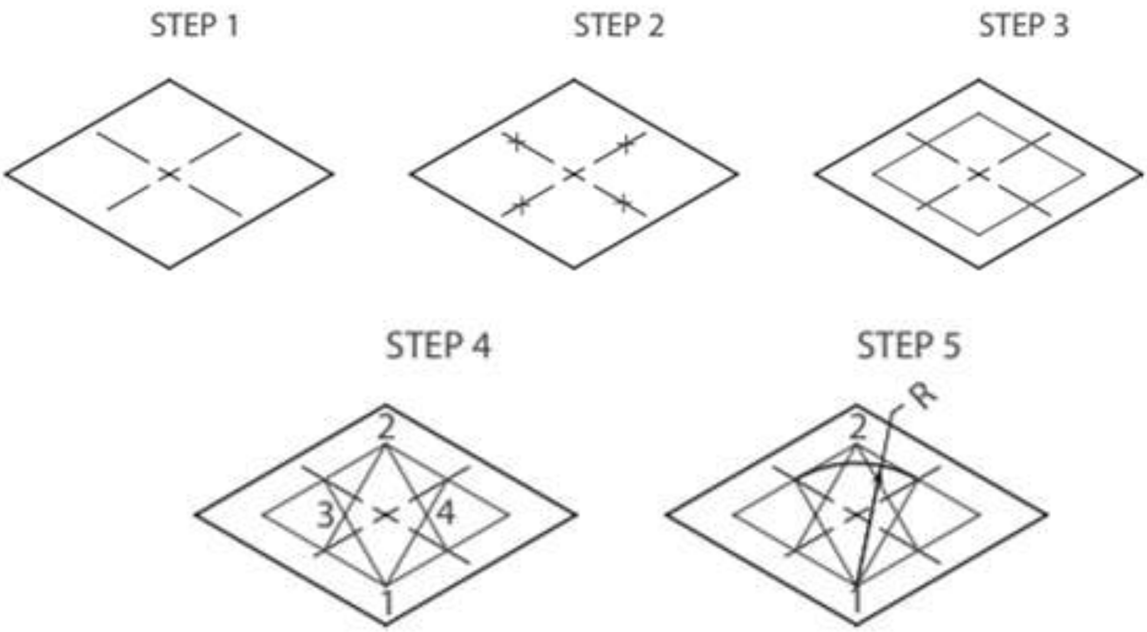


(iii)

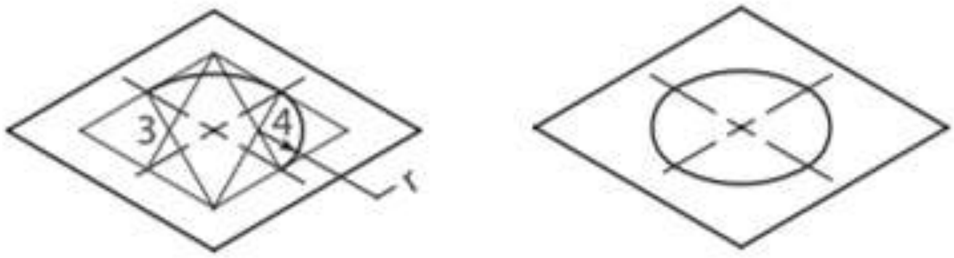


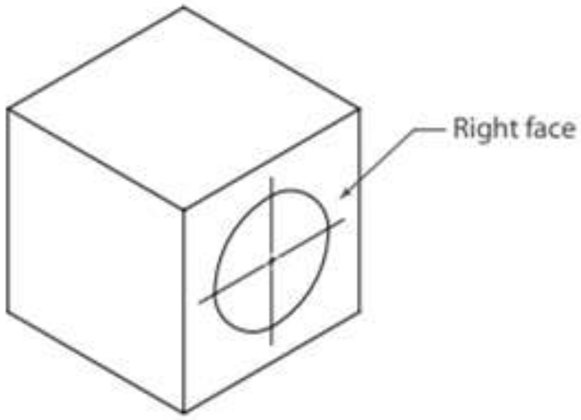
Circle drawing steps in all Views

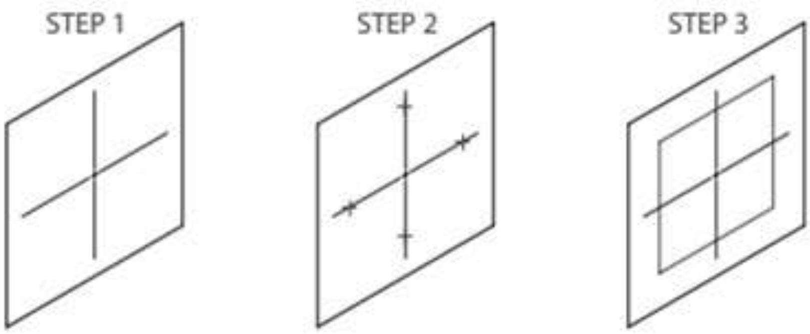


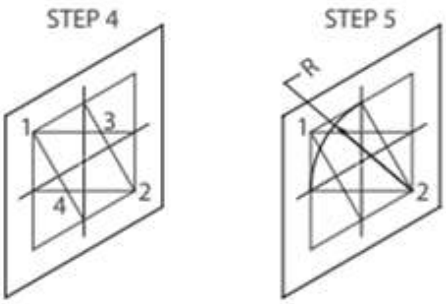


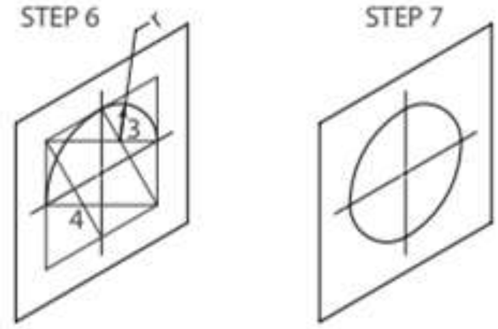
STEP 6 STEP 7

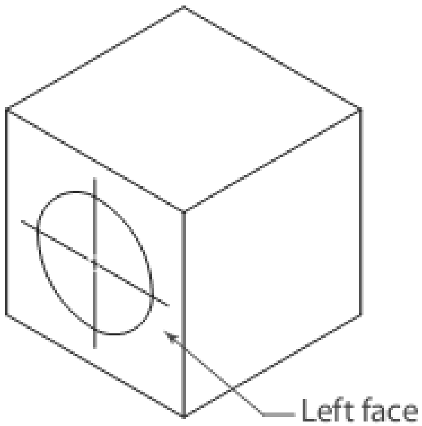


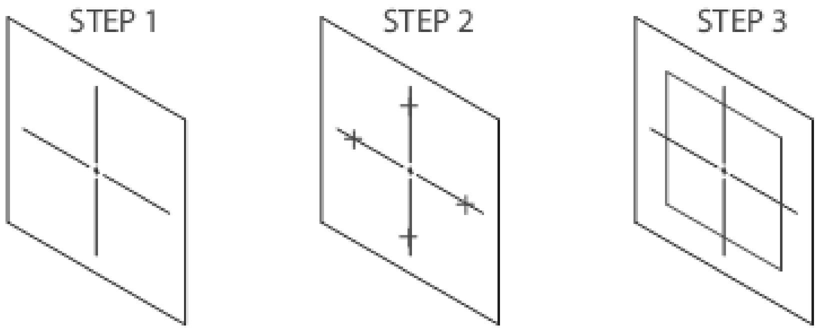


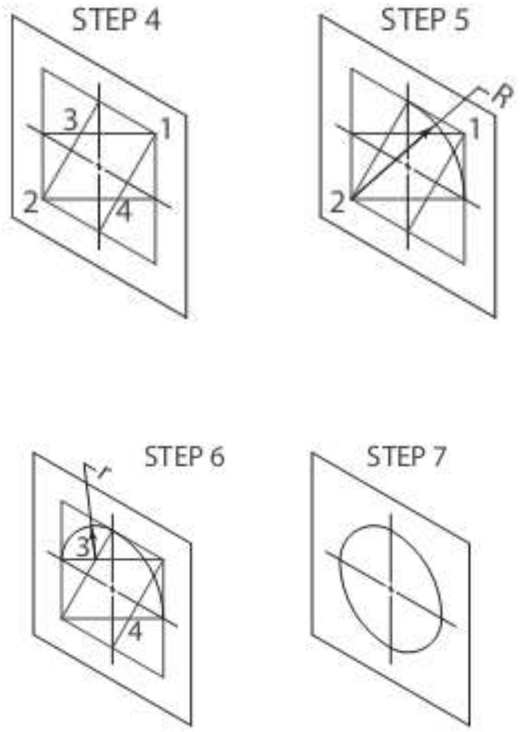





STEP 6
3
4
STEP 7


Left face


STEP 1
STEP 2
STEP 3


STEP 4
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STEP 5
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STEP 6
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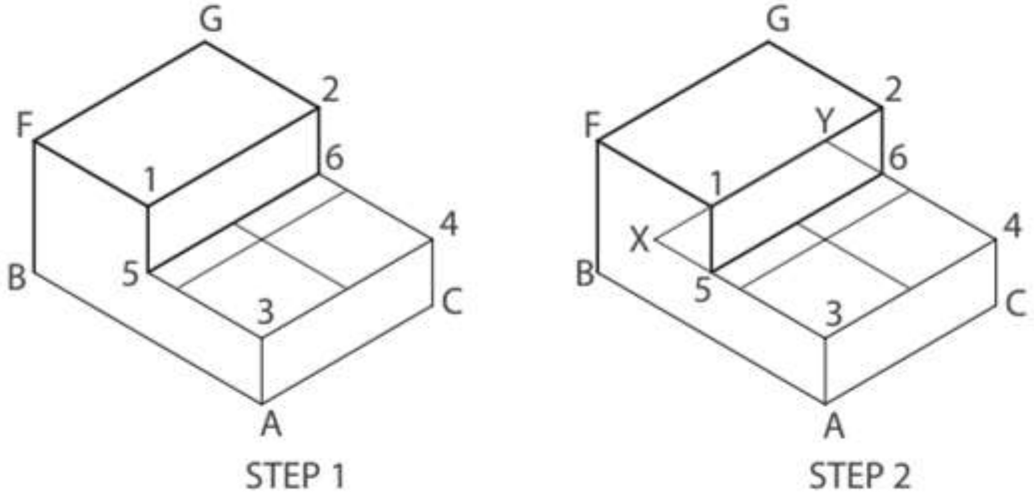
STEP 7

Example:


G
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B
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C
A
STEP 1
G
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Y
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X
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B
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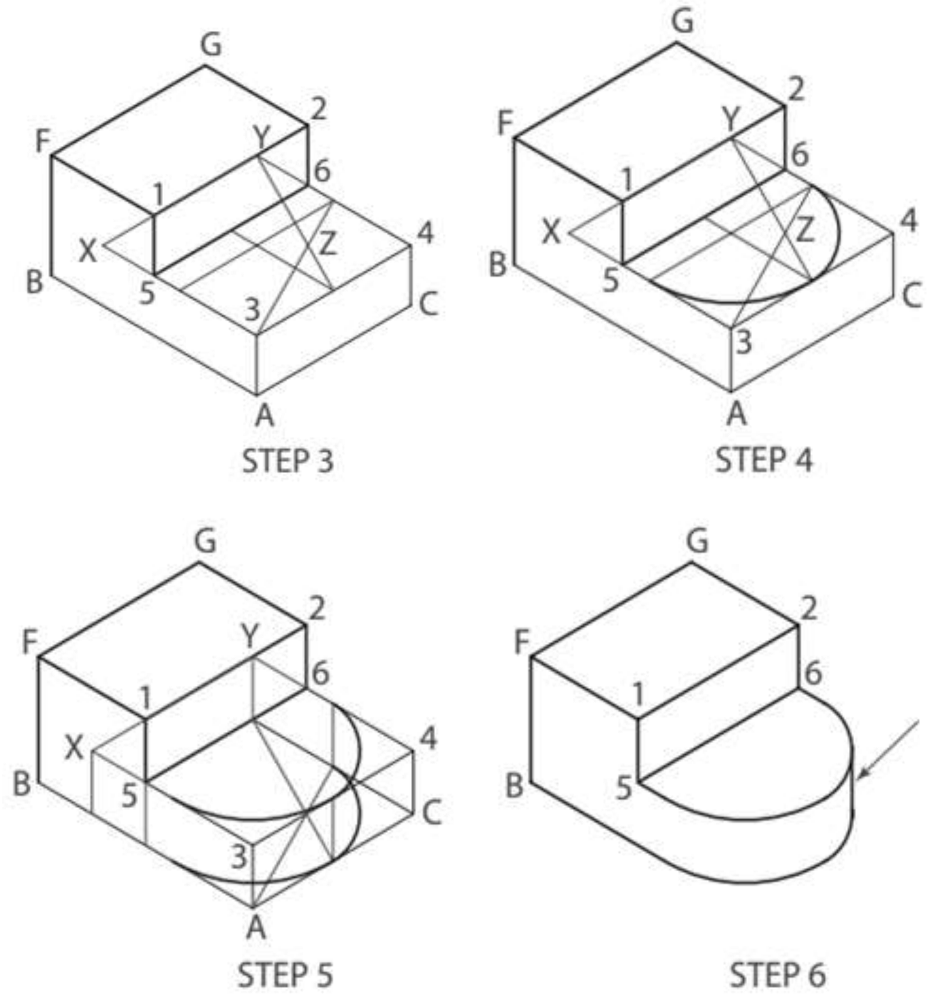
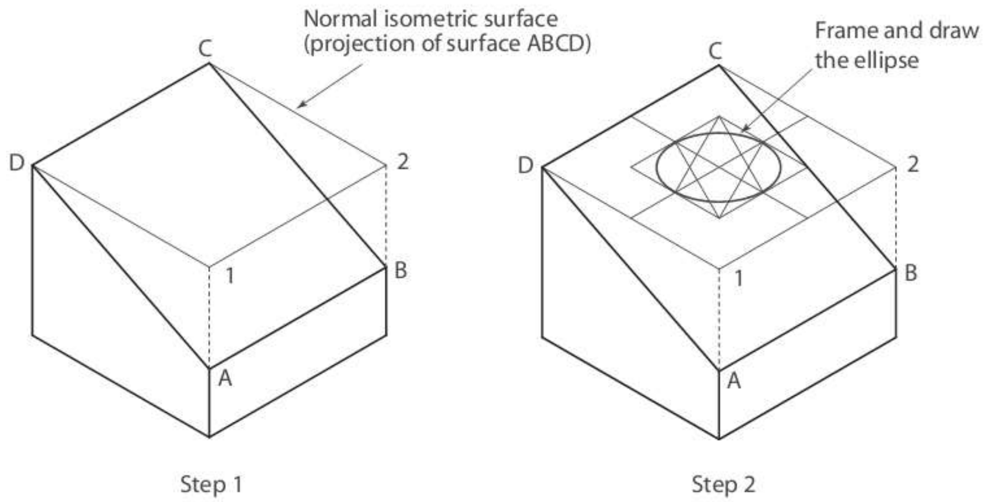
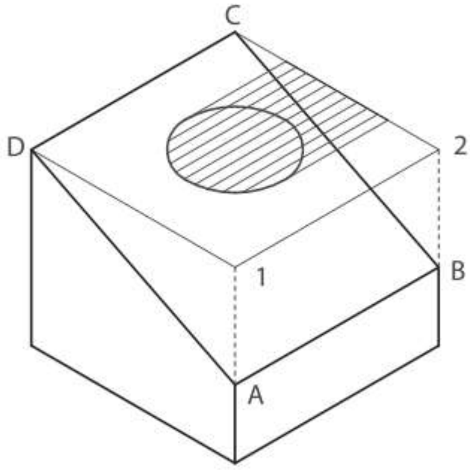
3
C
A
STEP 2



5.8 ISOMETRIC PROJECTION OF OBLIQUE CIRCLE





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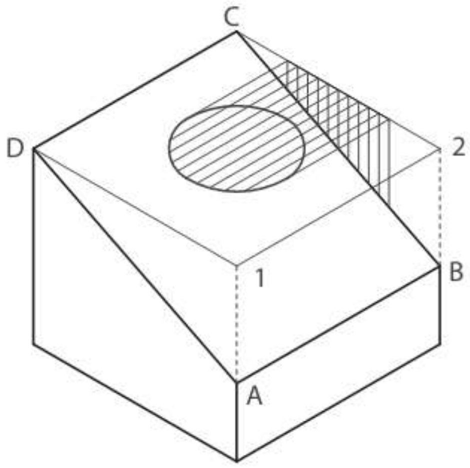

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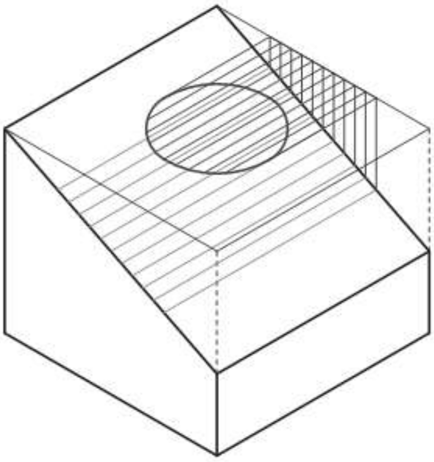

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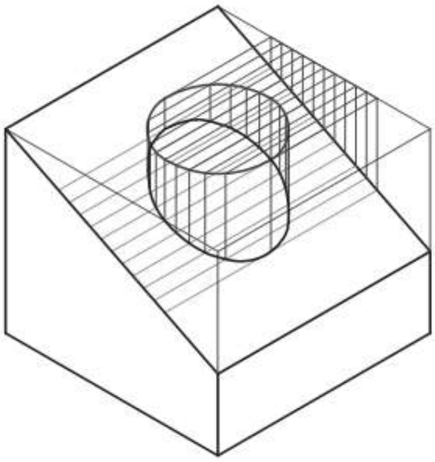
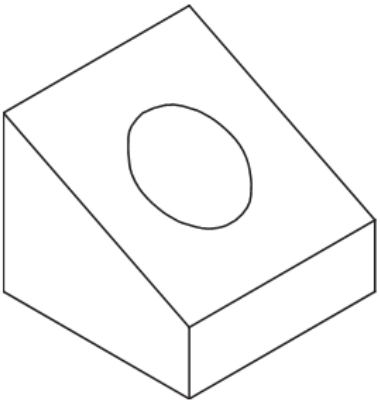
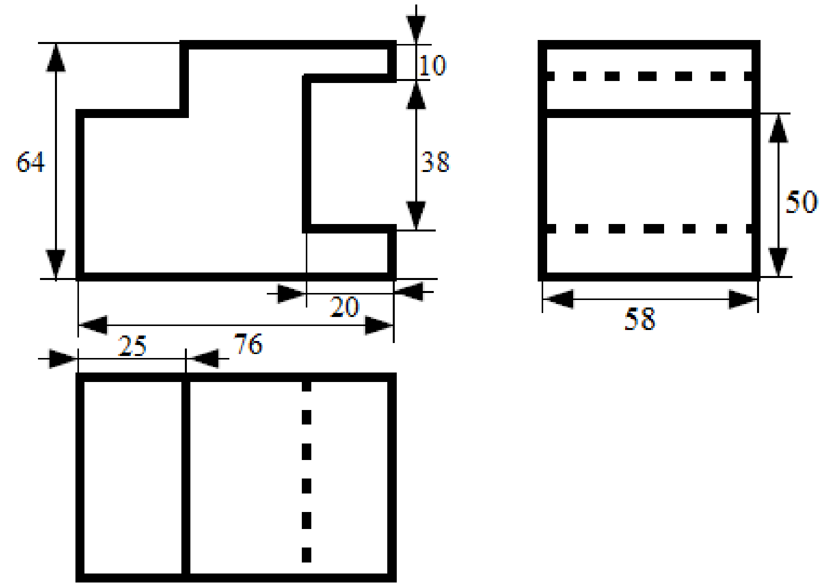
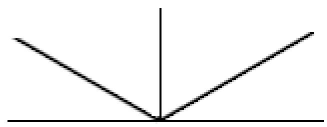


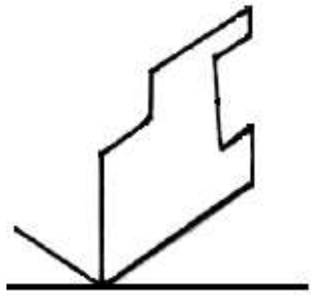
Step 6



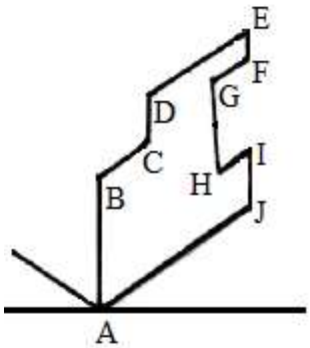




(a)



(b)



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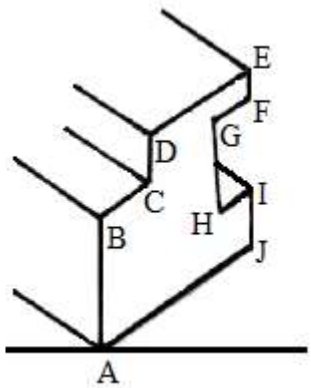

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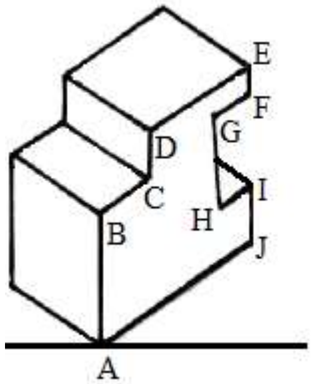

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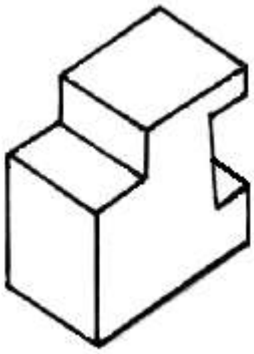

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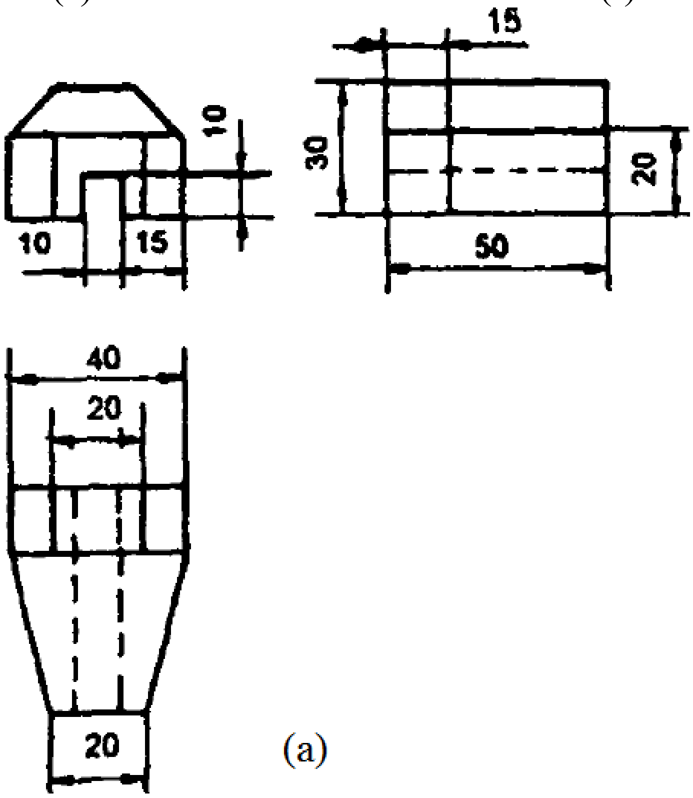
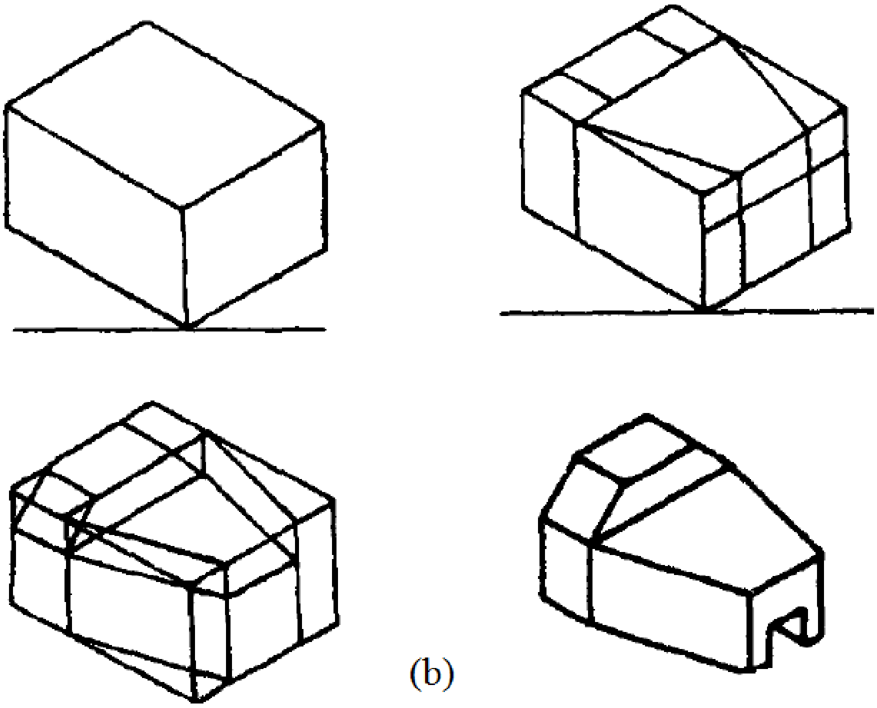


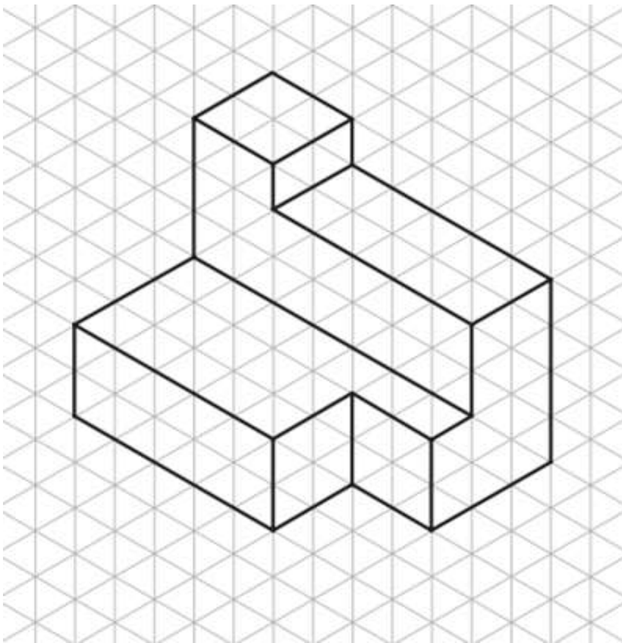
(a)

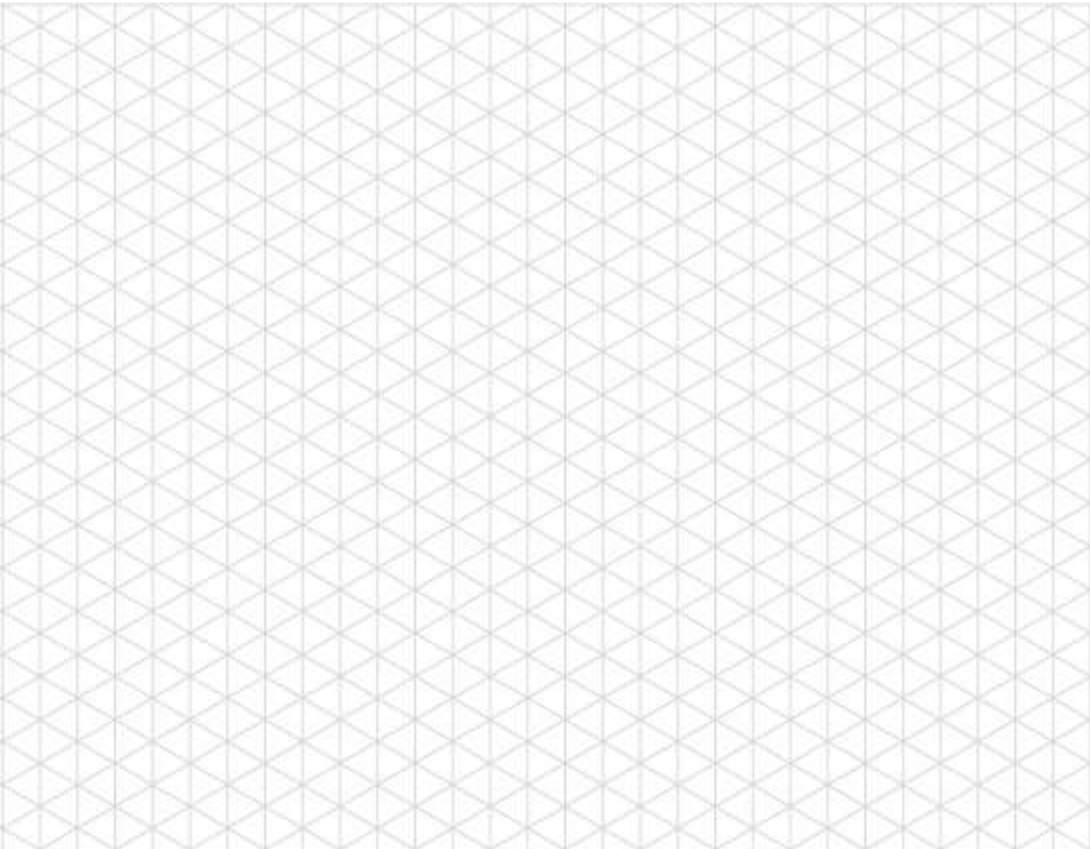


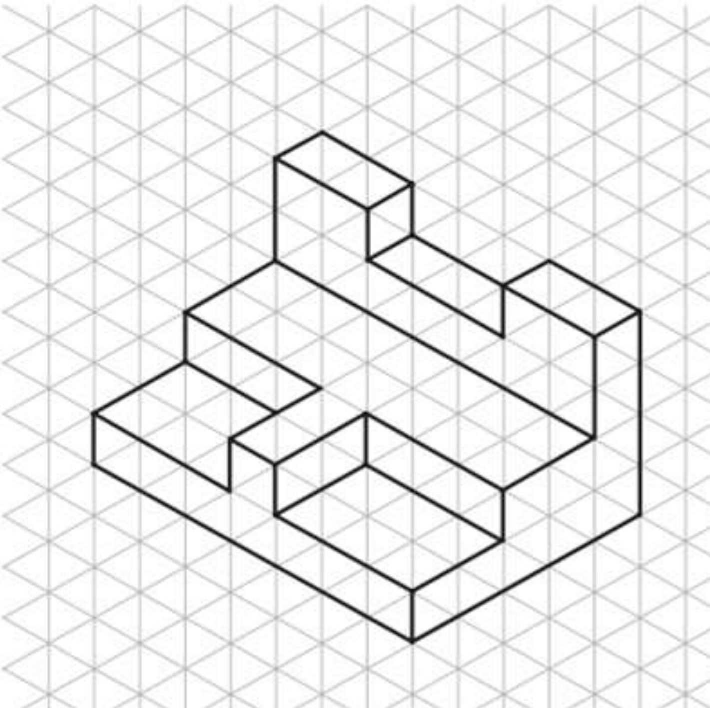
(b)

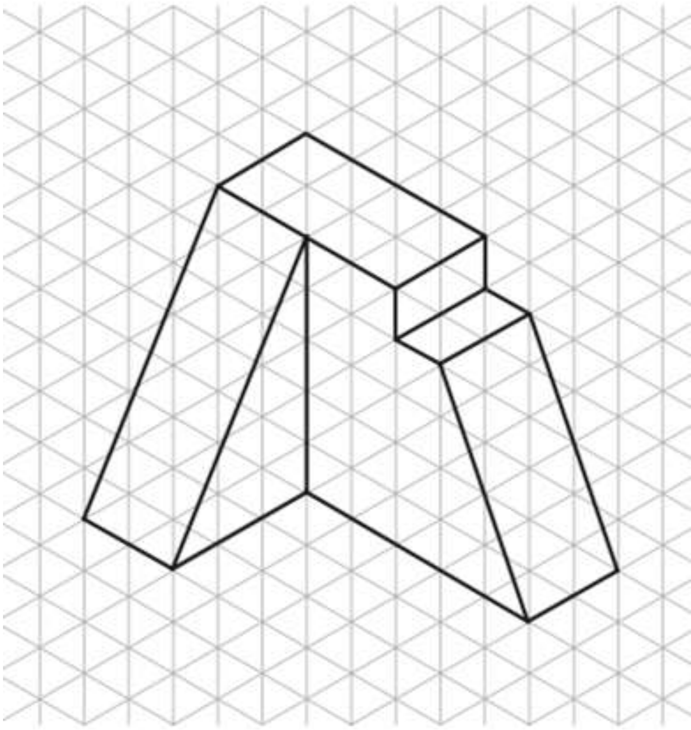
Exercises

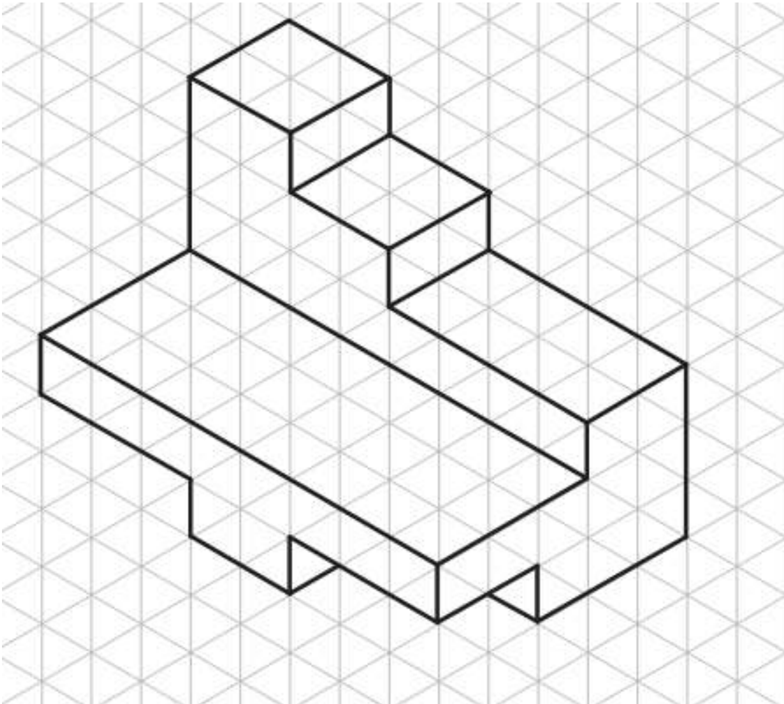
- Draw the following Isomtric of an object in the same paper.

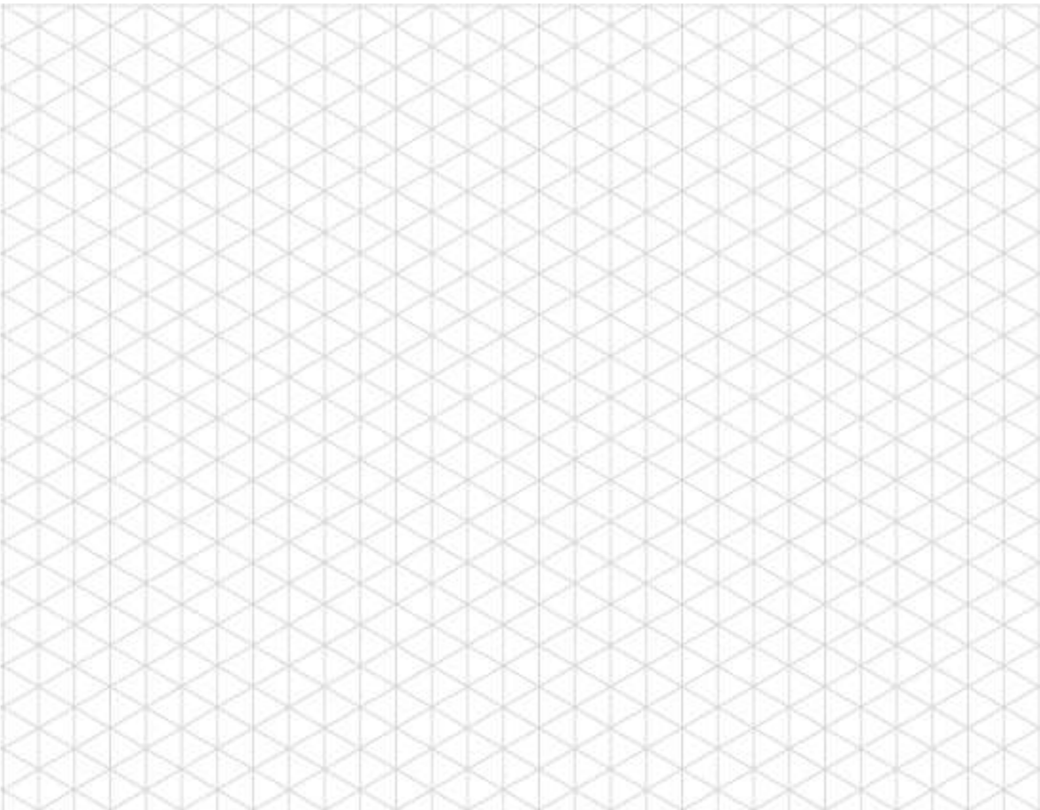


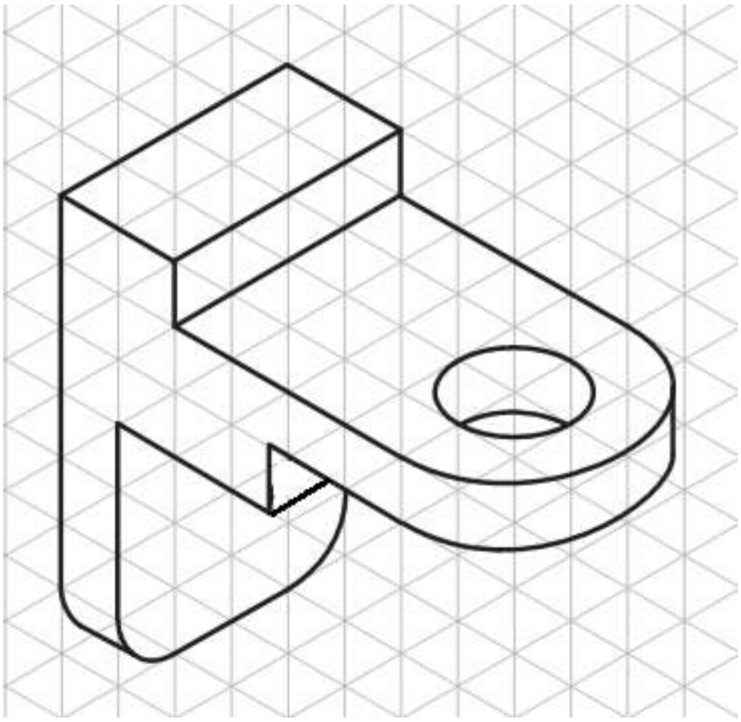


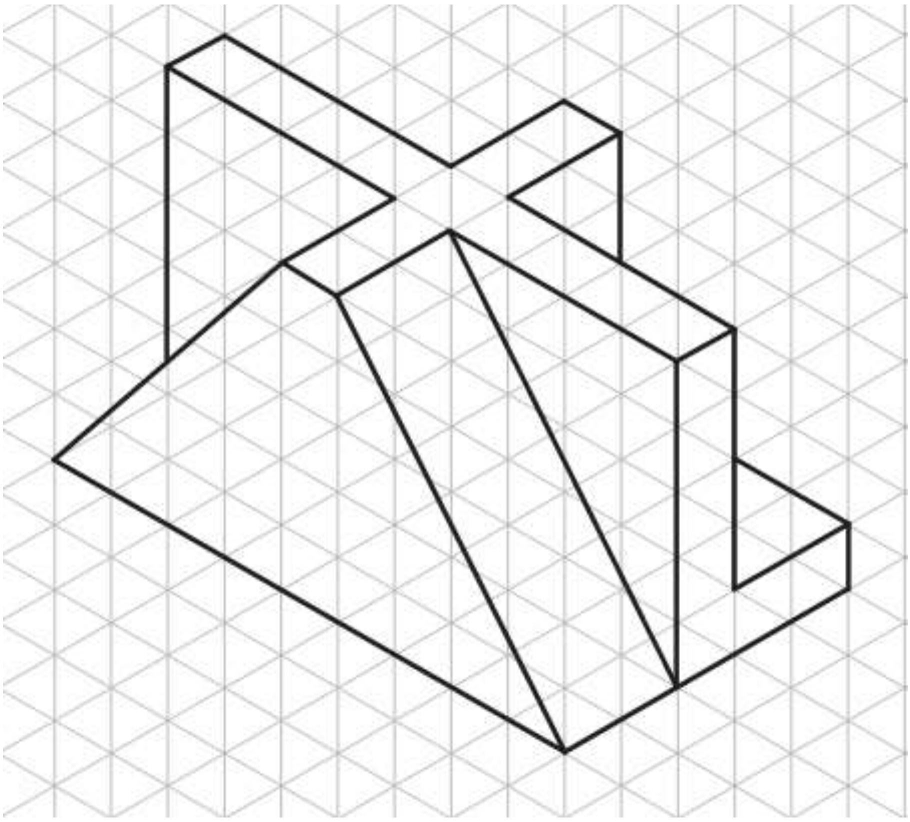


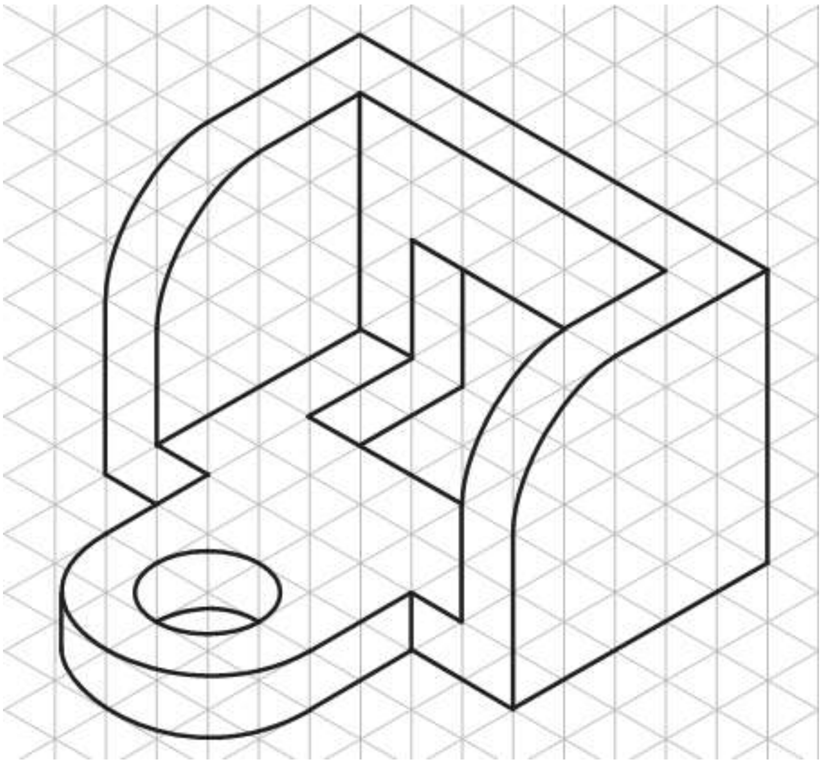




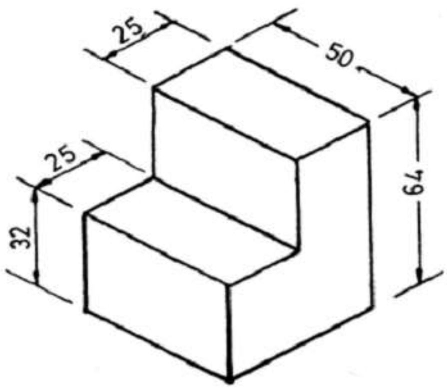


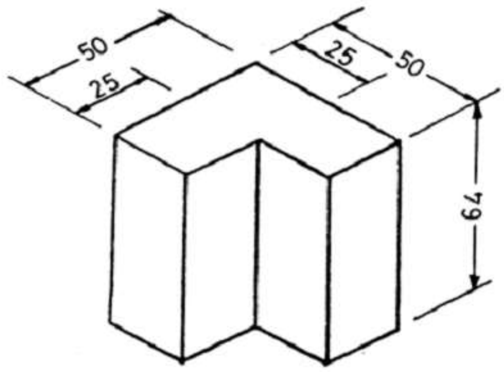


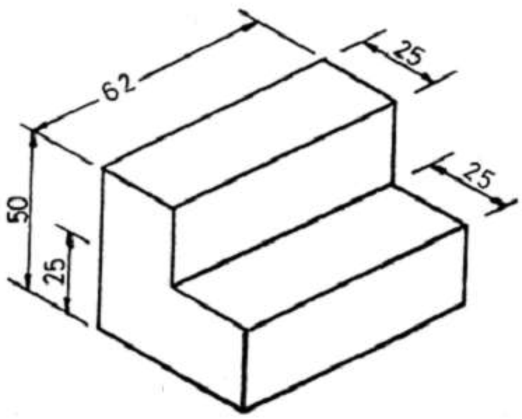


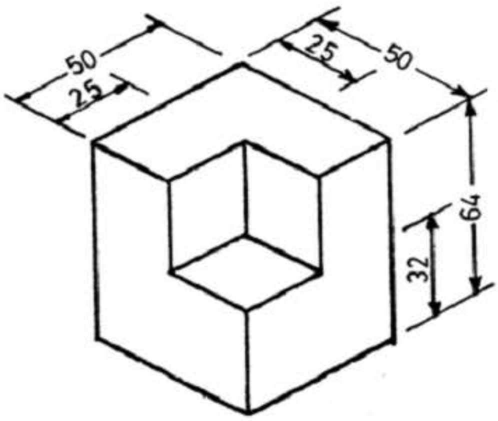


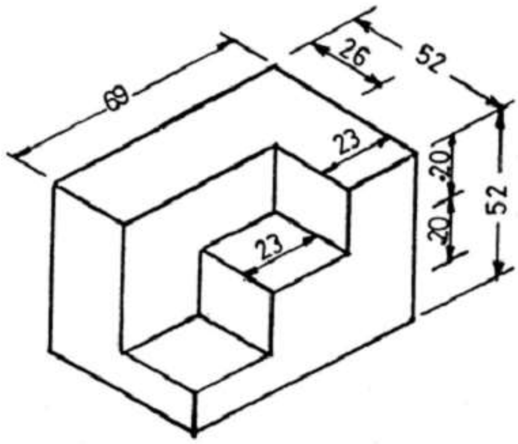
- Draw the following Isomtric of an object with free hand skhetch

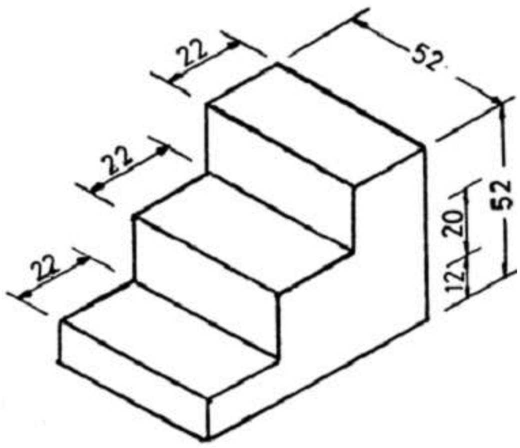


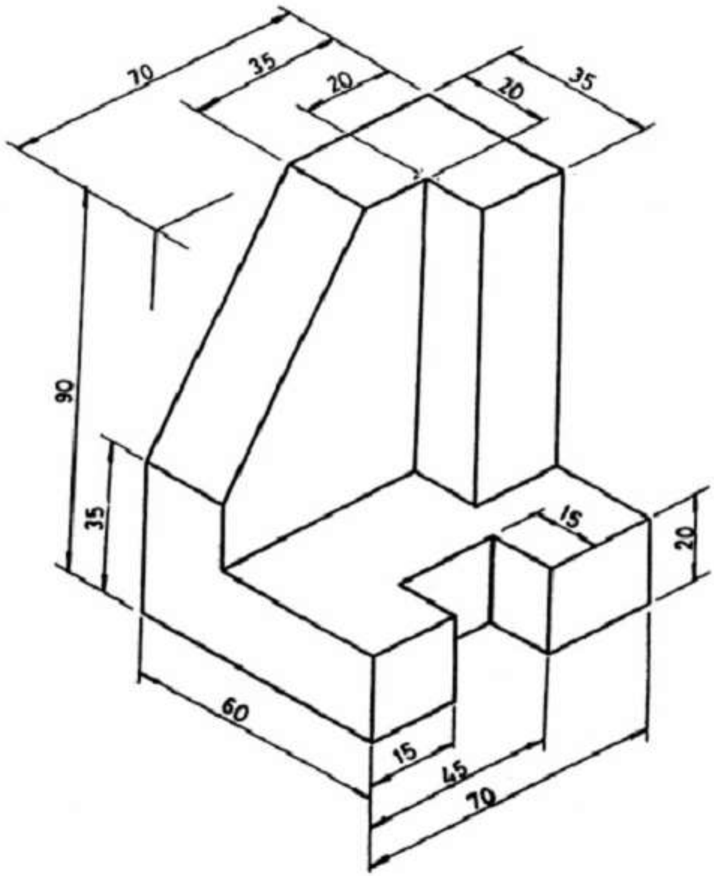








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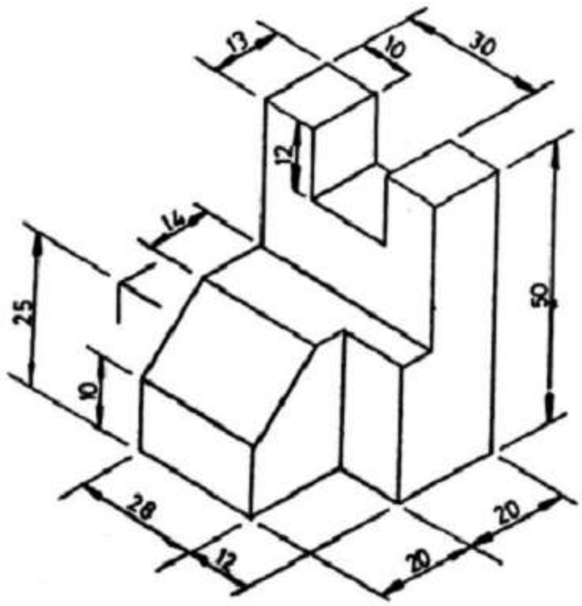
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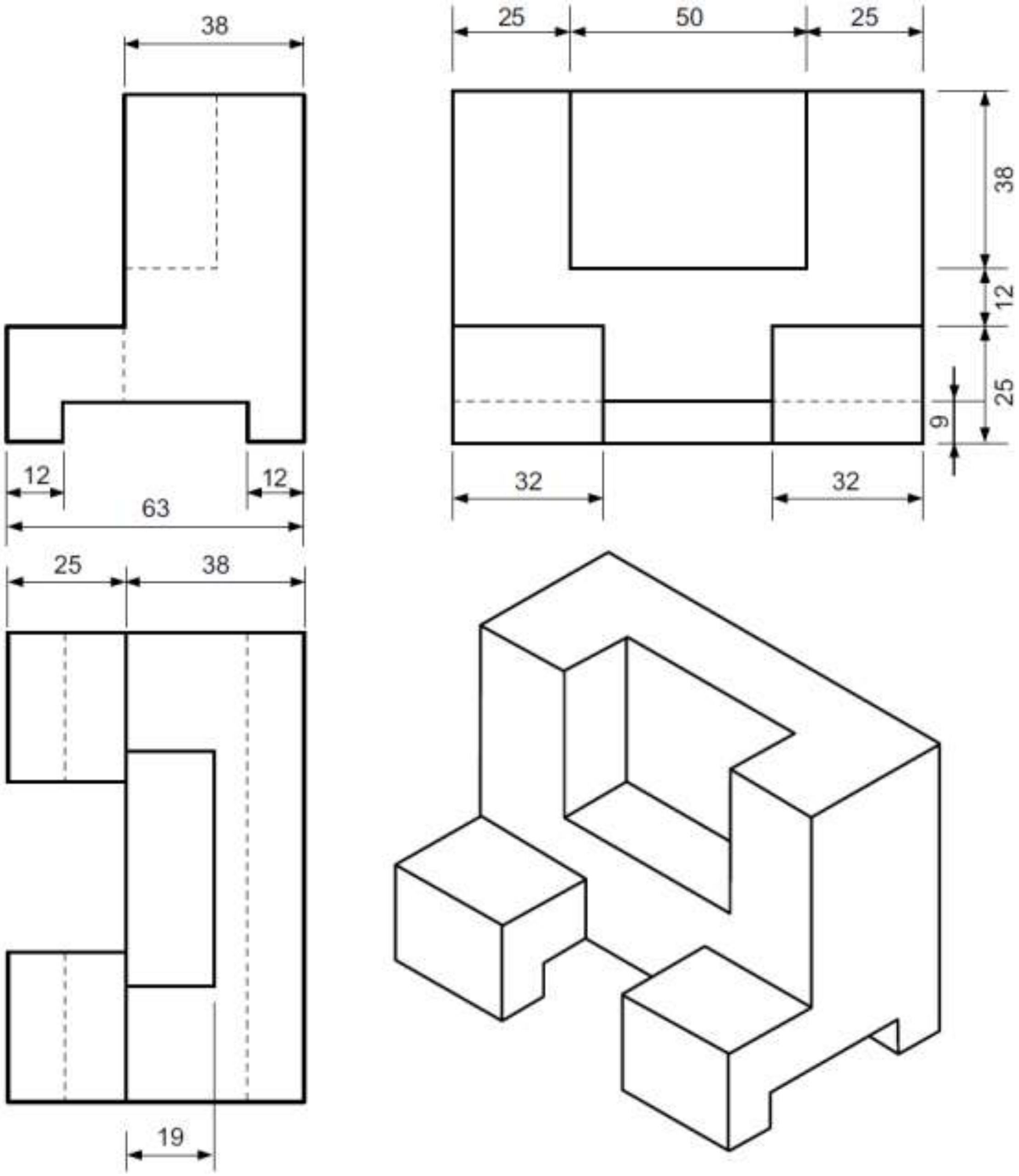
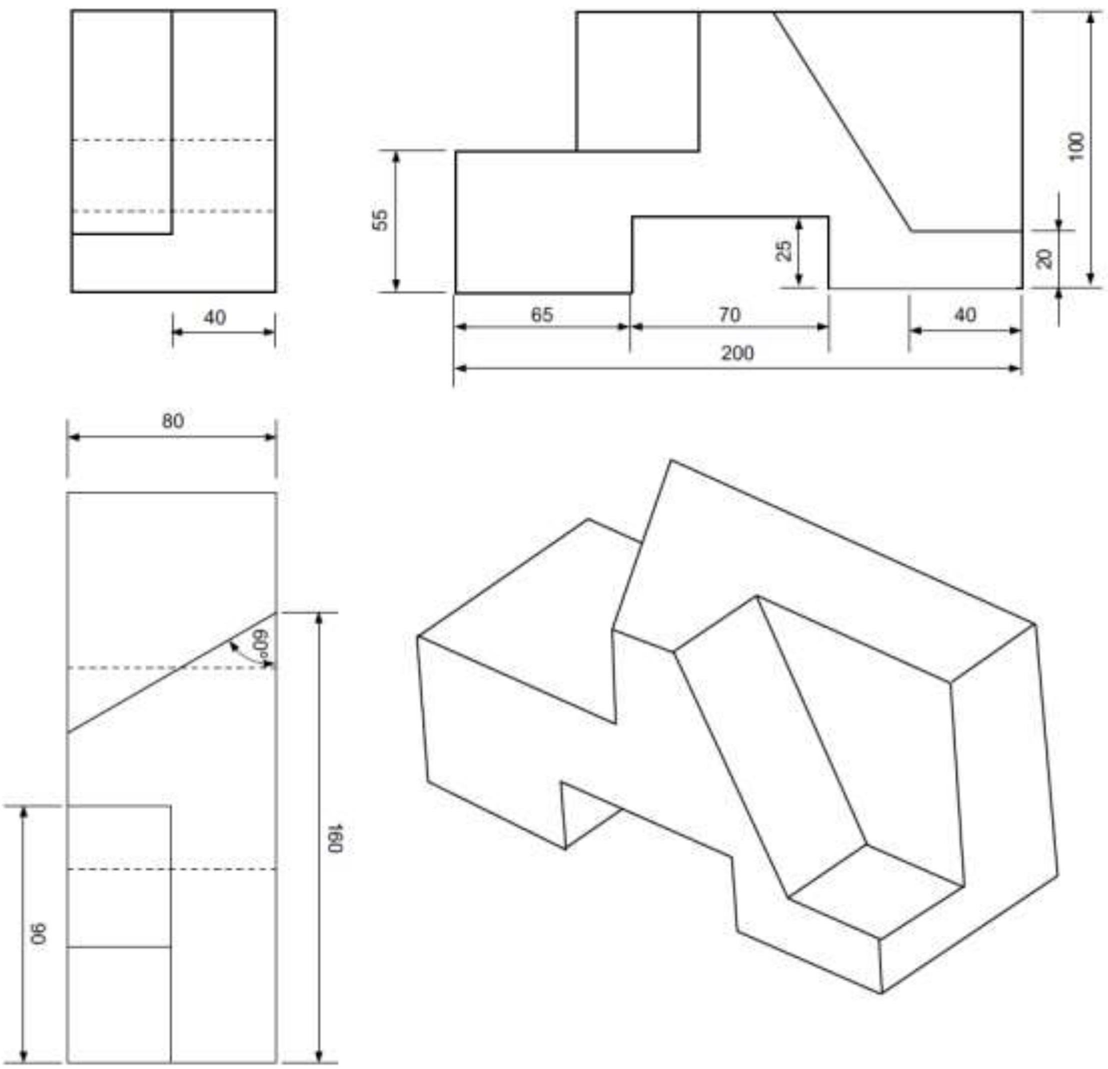
- **Solved example**

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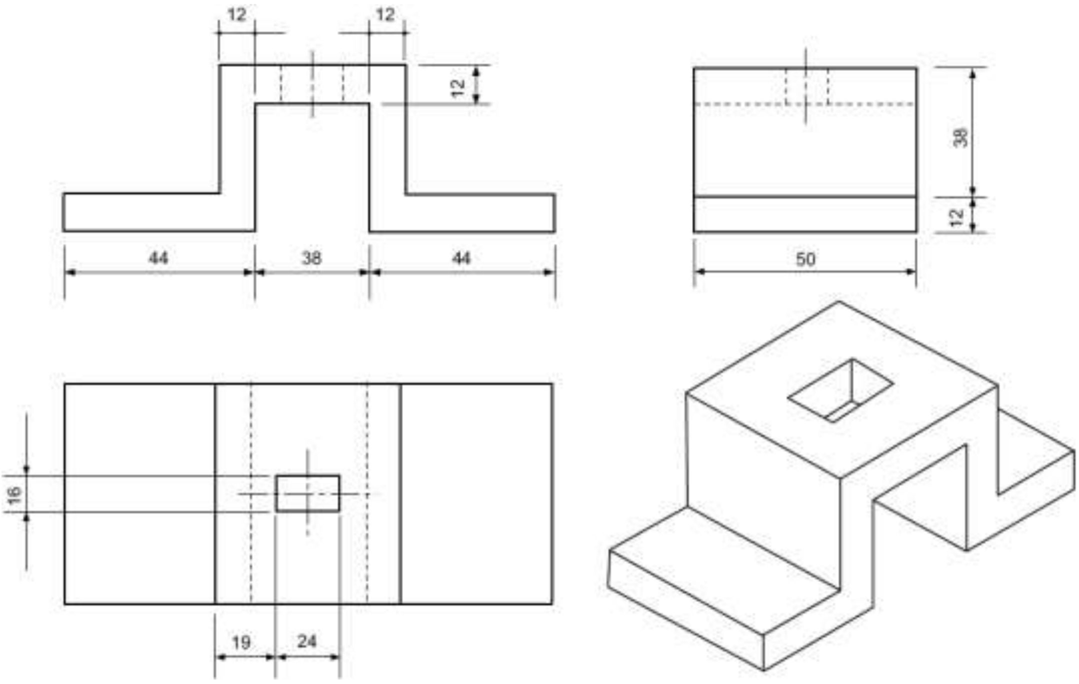
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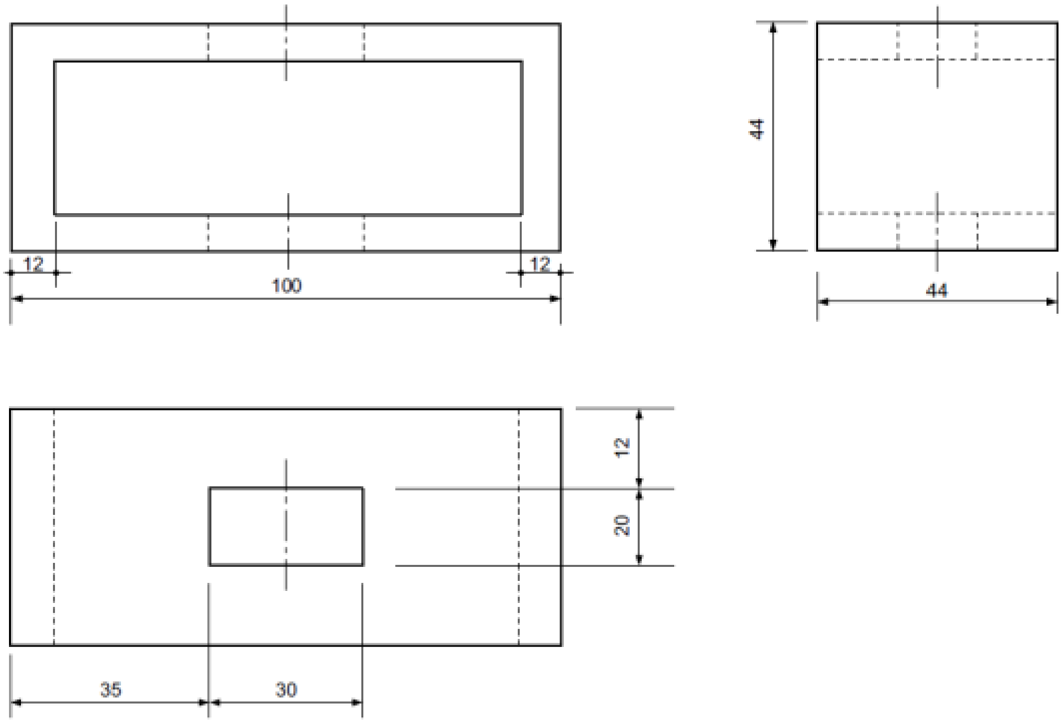
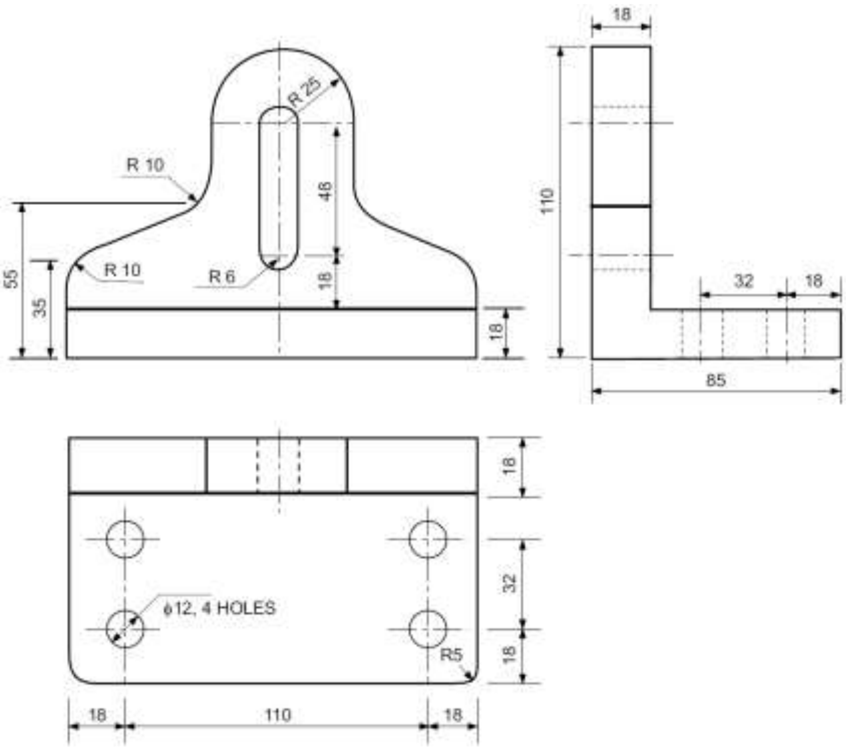
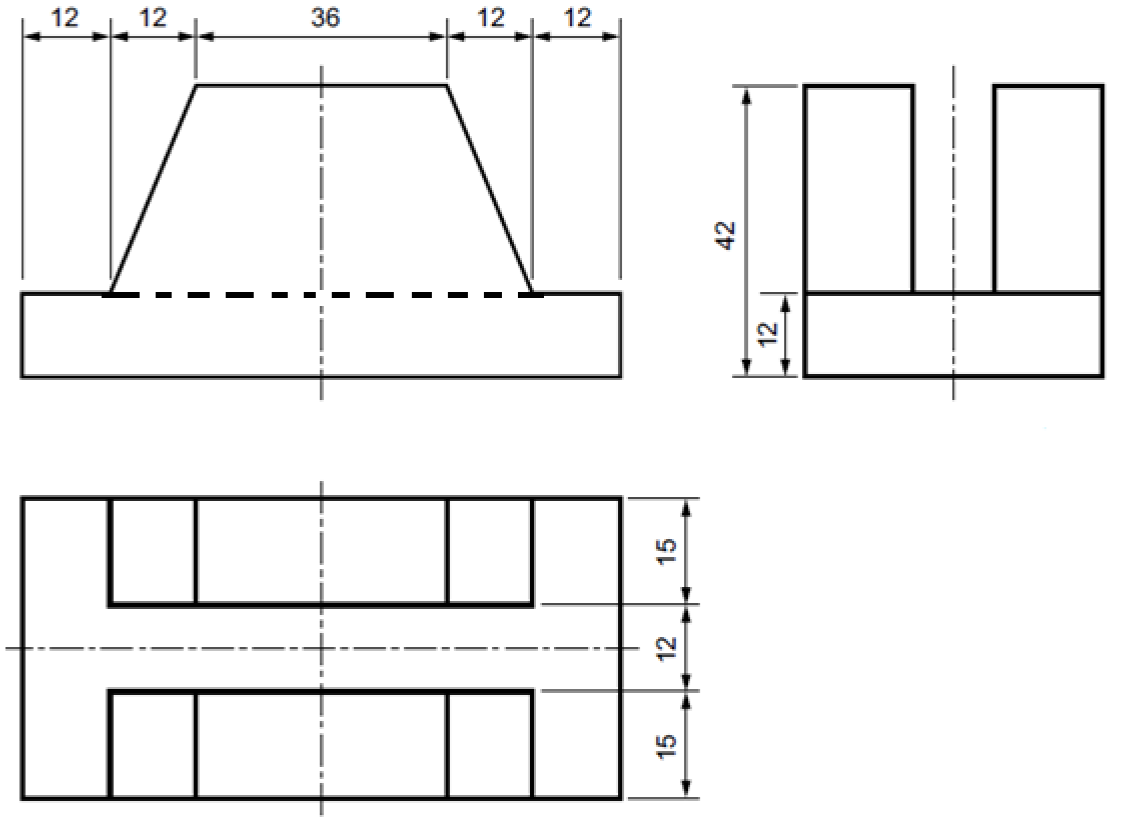
Exercises

The figures show the orthographic view of an object. Draw its isometric view.



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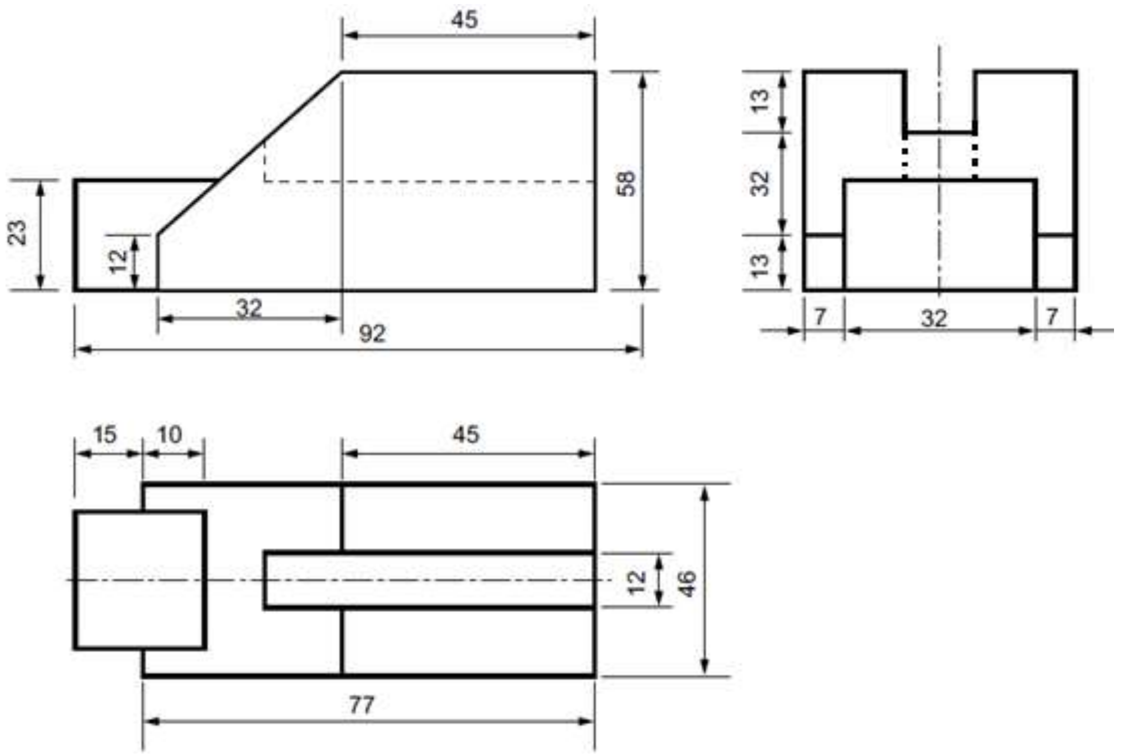
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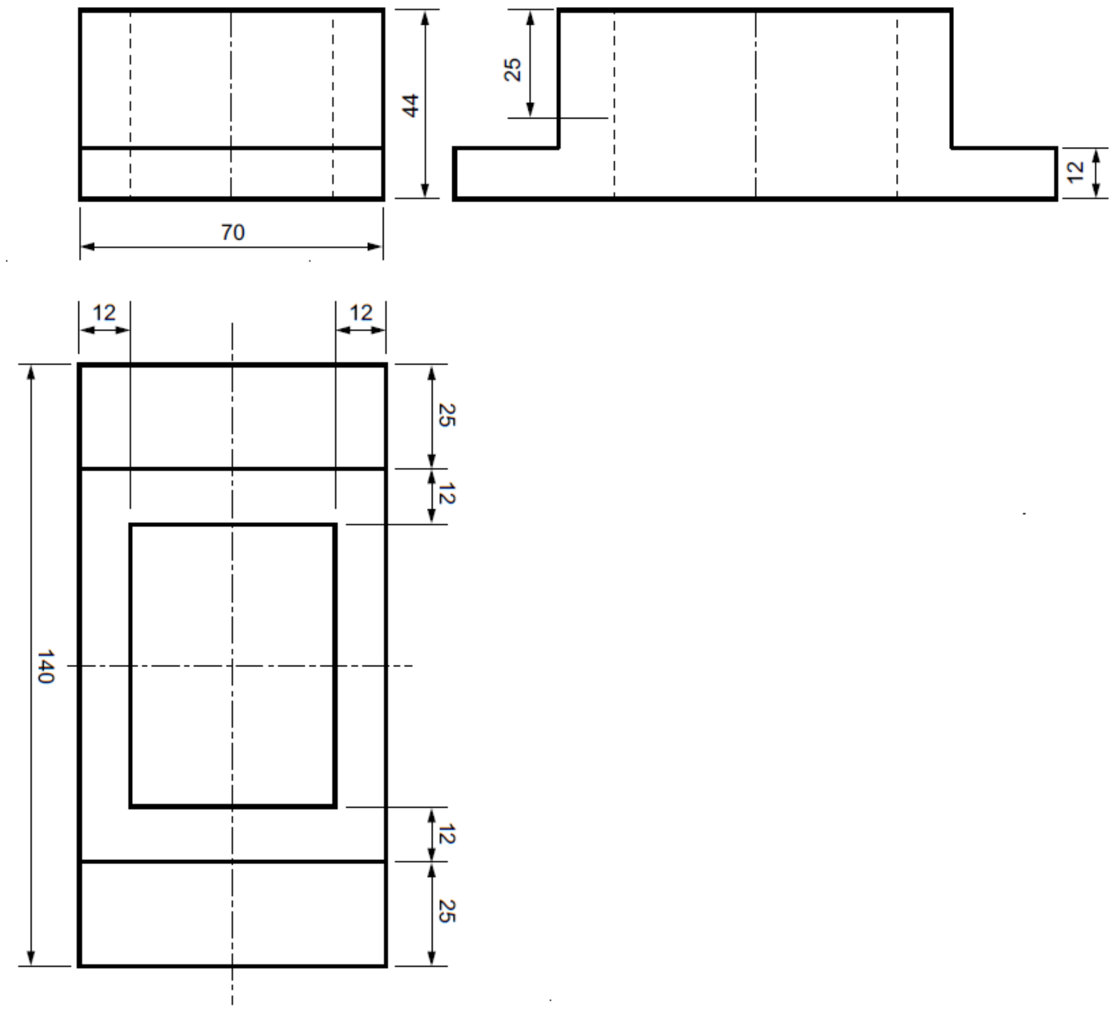
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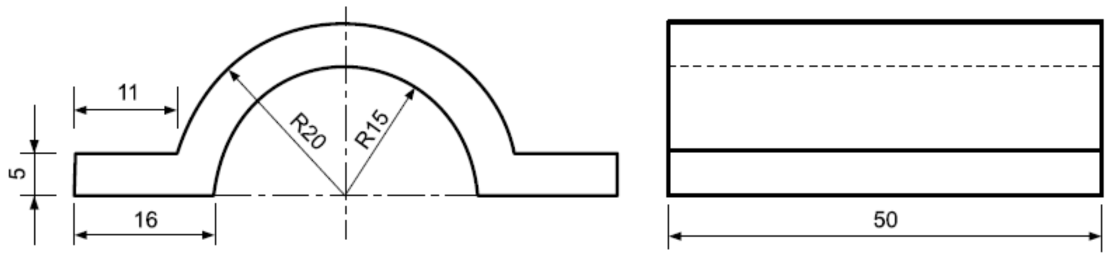
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R15
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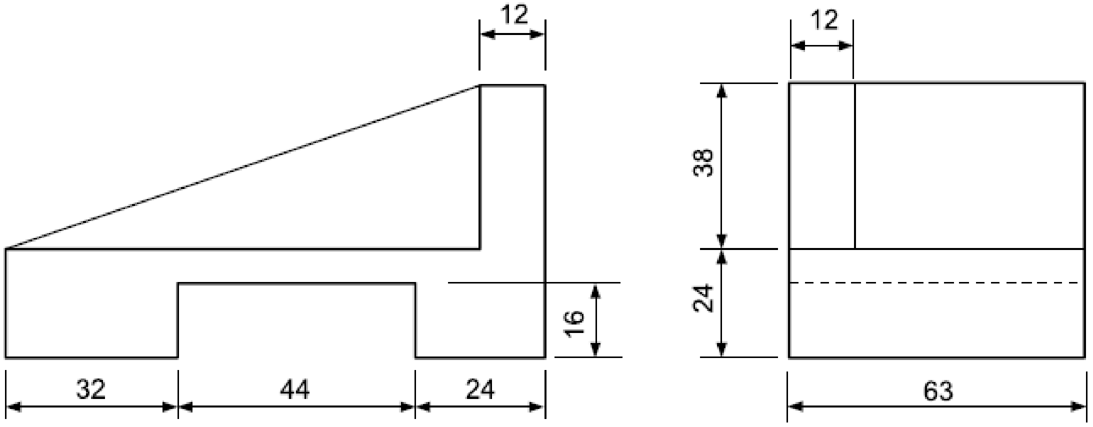
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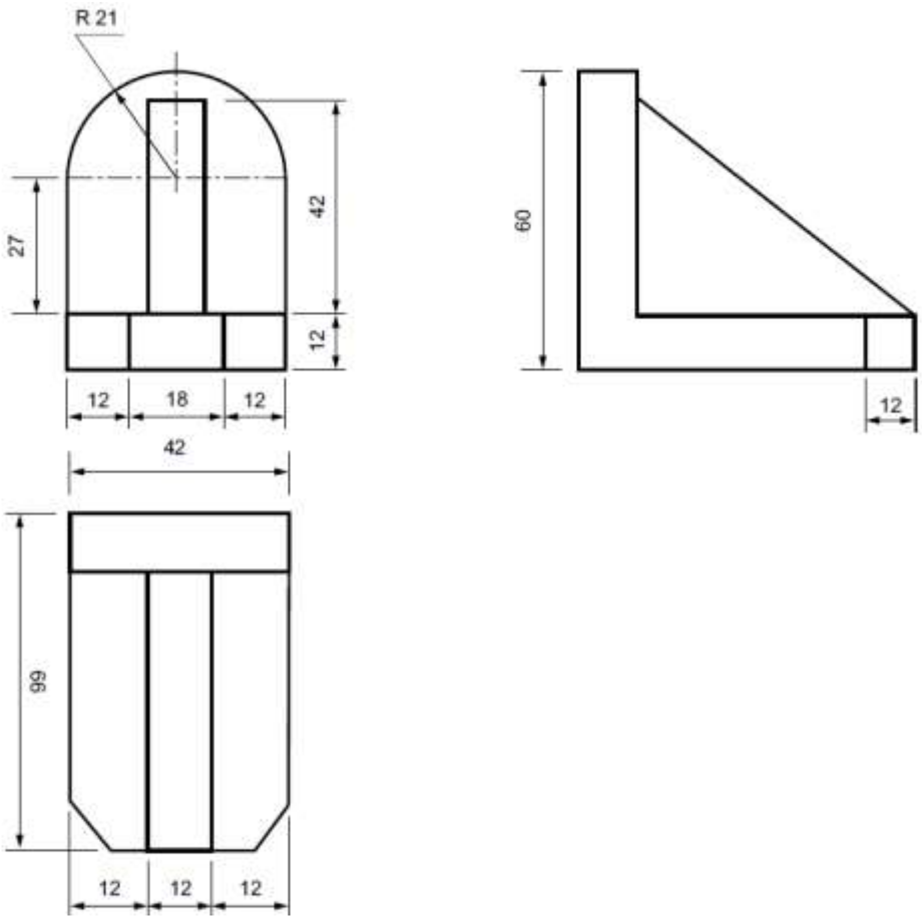
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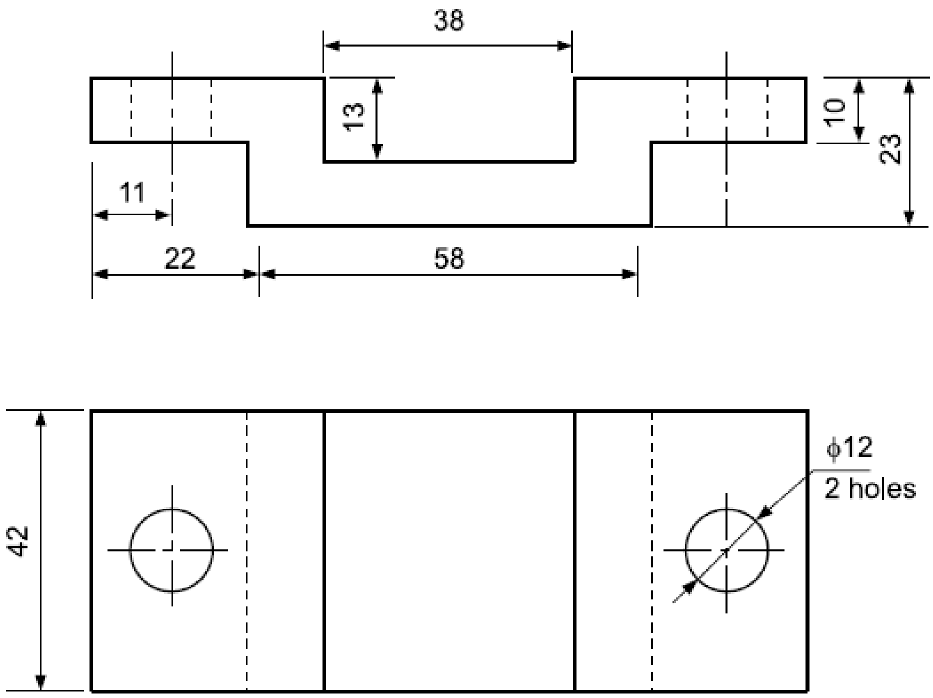
38
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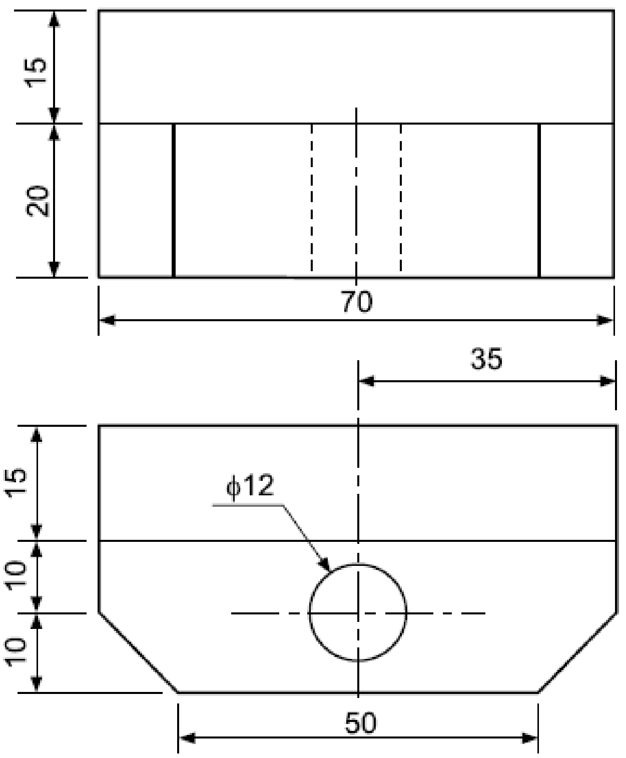
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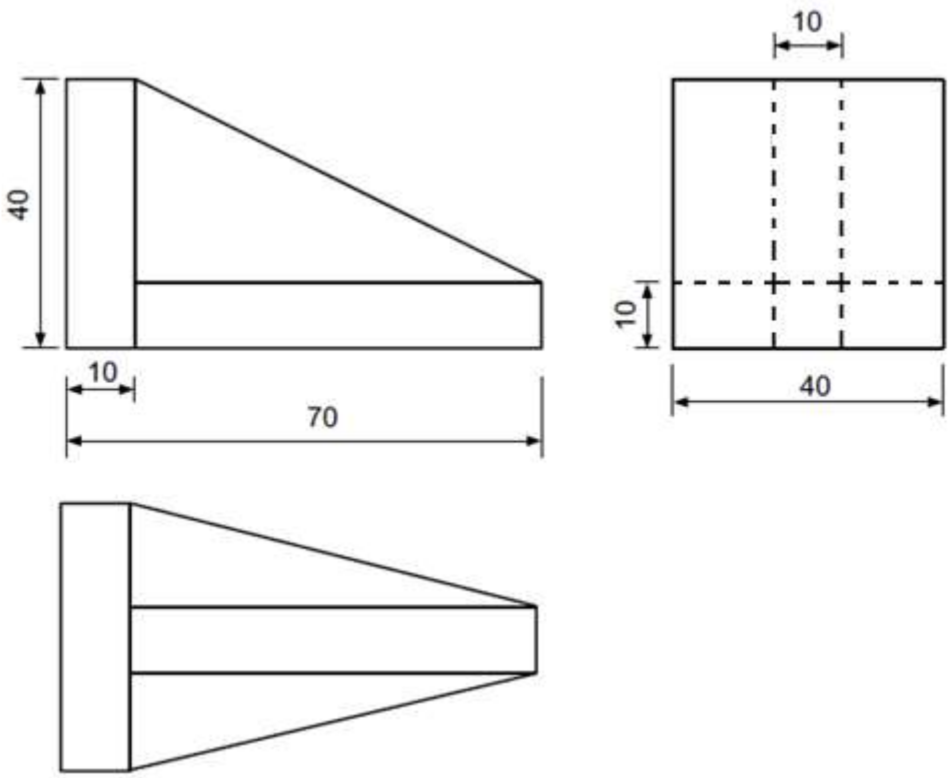
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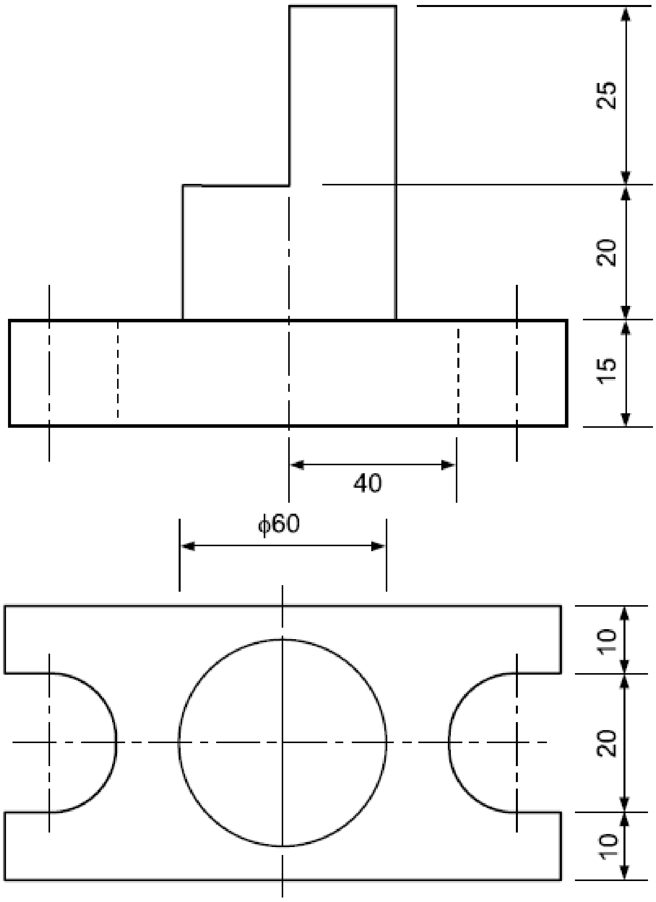
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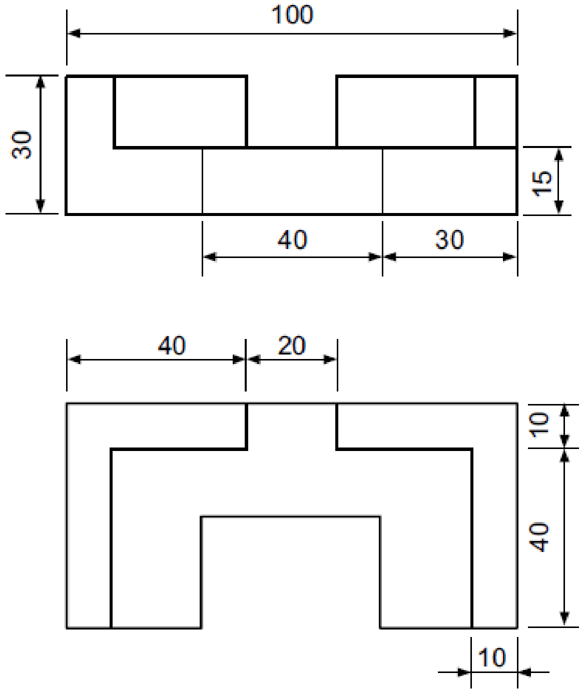
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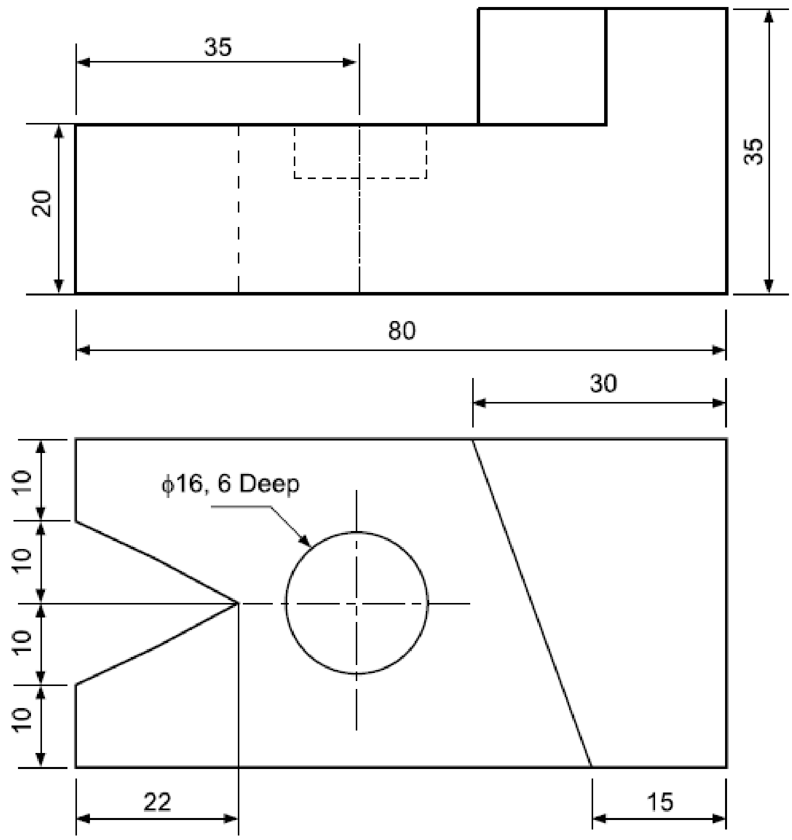
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ϕ16, 6 Deep
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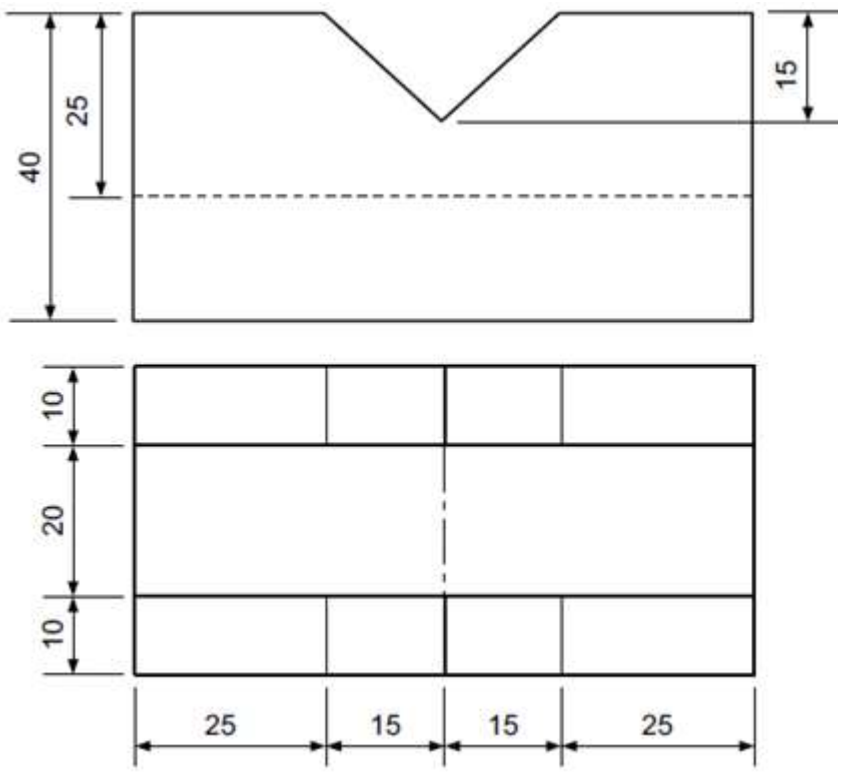
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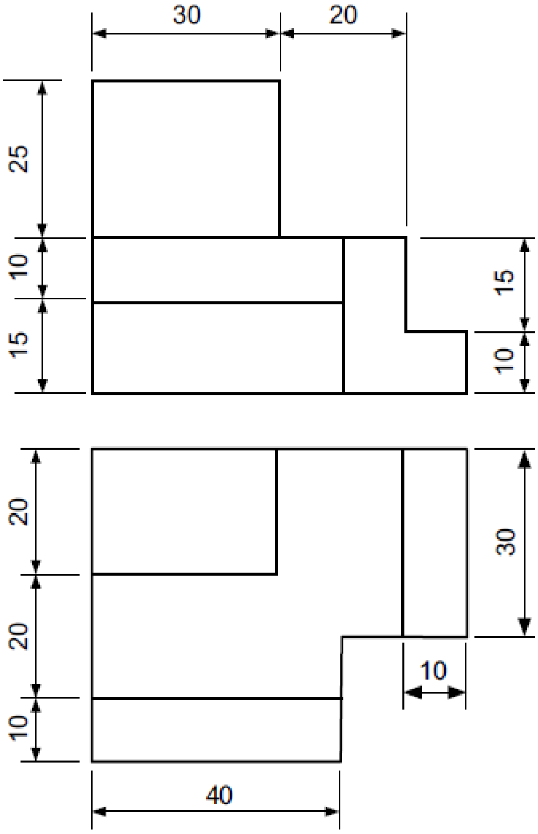
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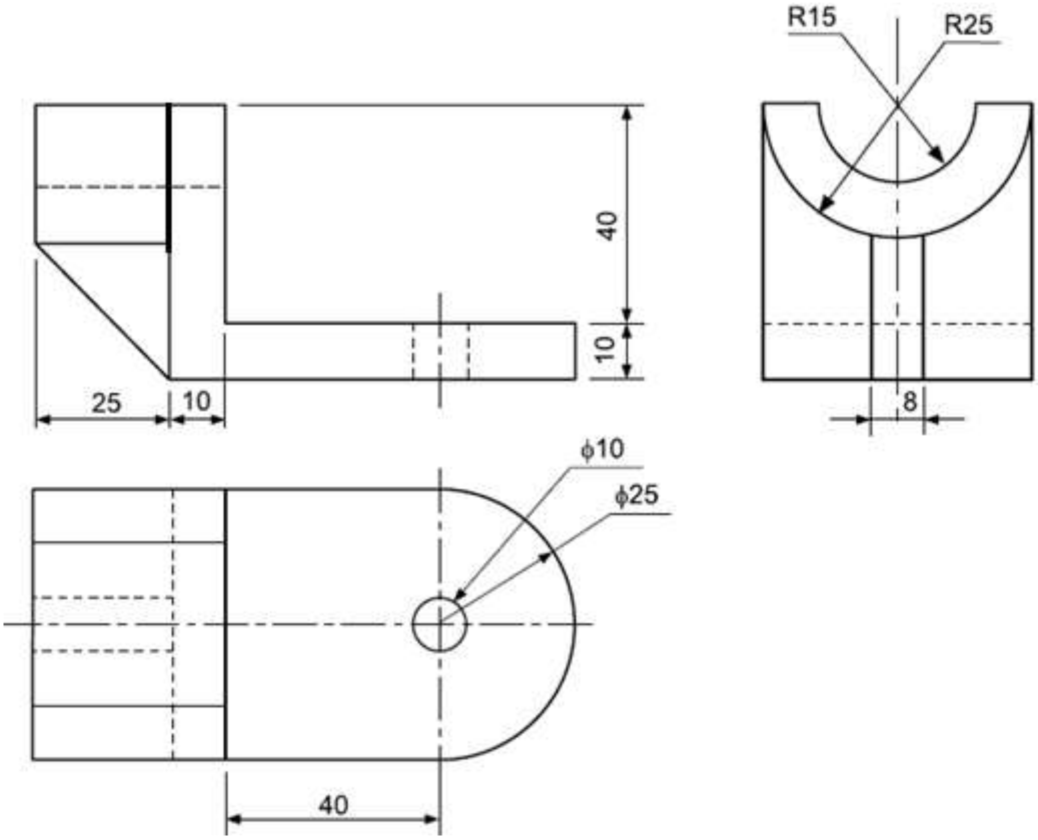
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R25
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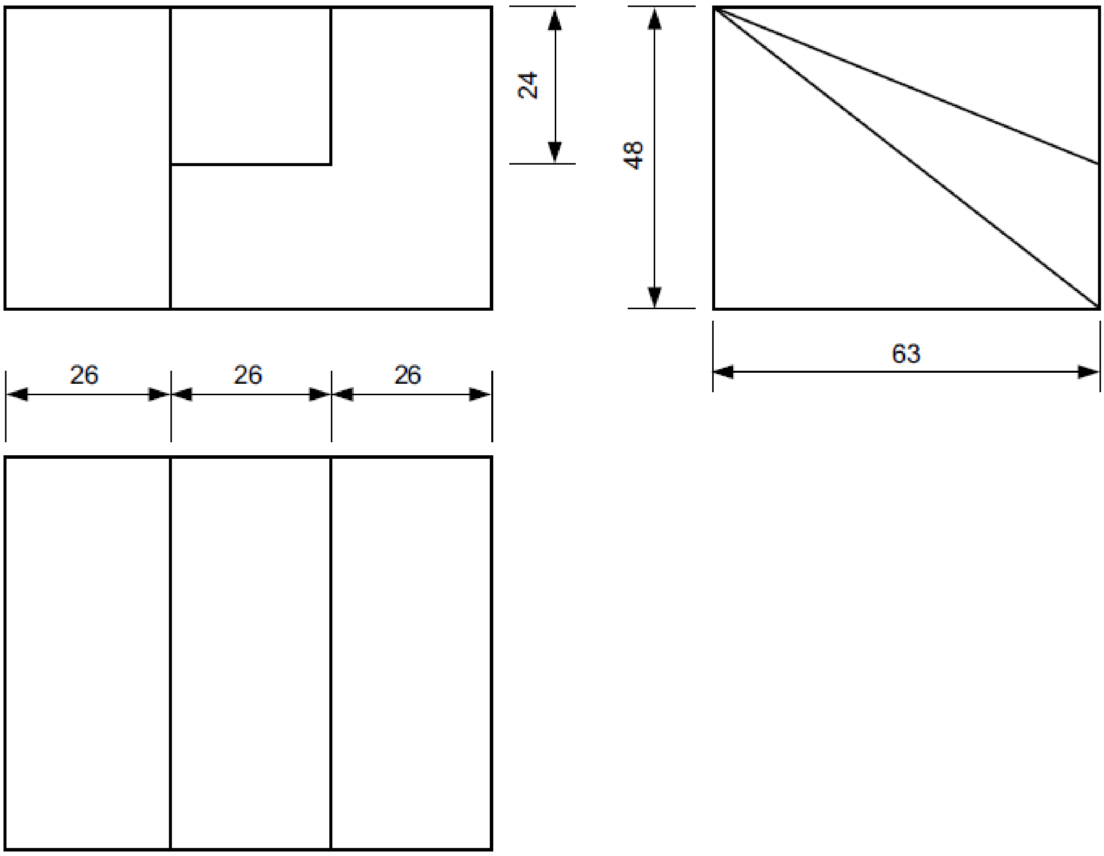
ϕ10
ϕ25
40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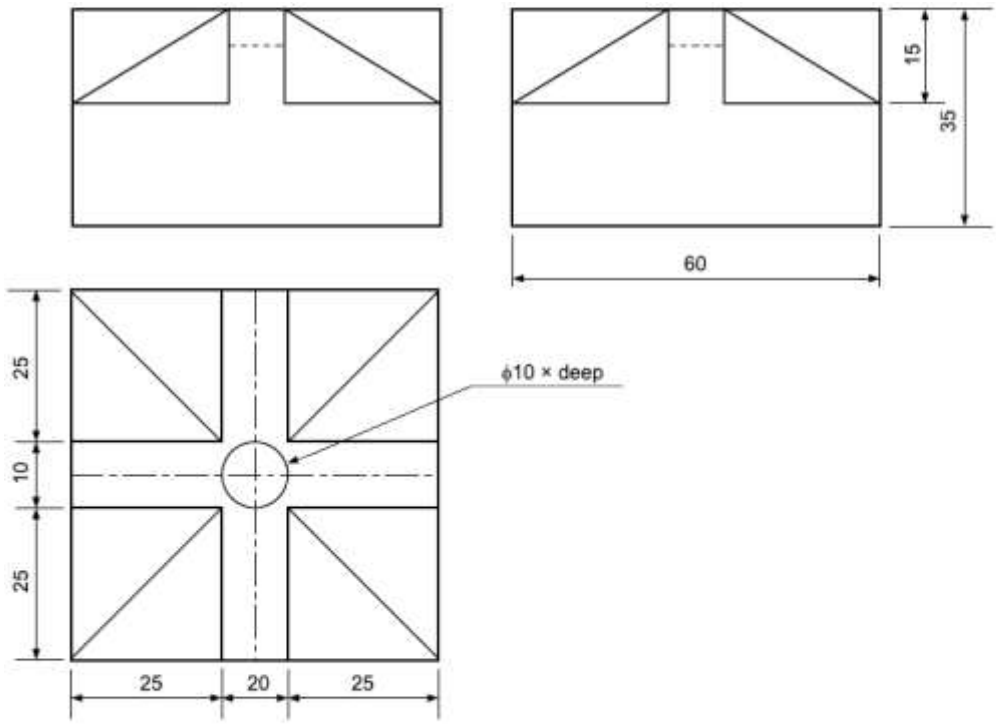
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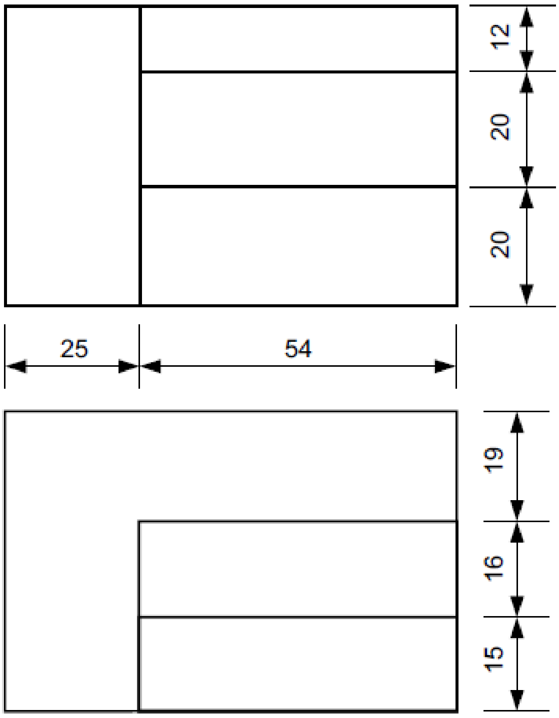
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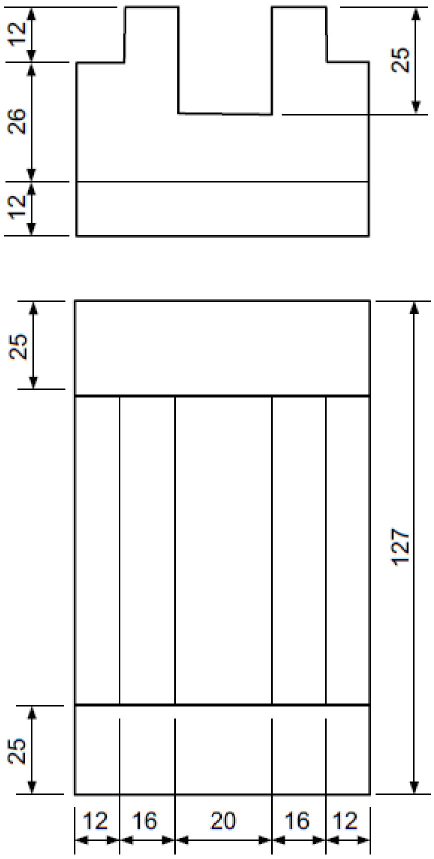
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Chapter 6

Missing Lines, Missing Views & Identification of Surfaces

Missing Lines, Missing Views& Identification of Surfaces

6.1 MISSING LINES

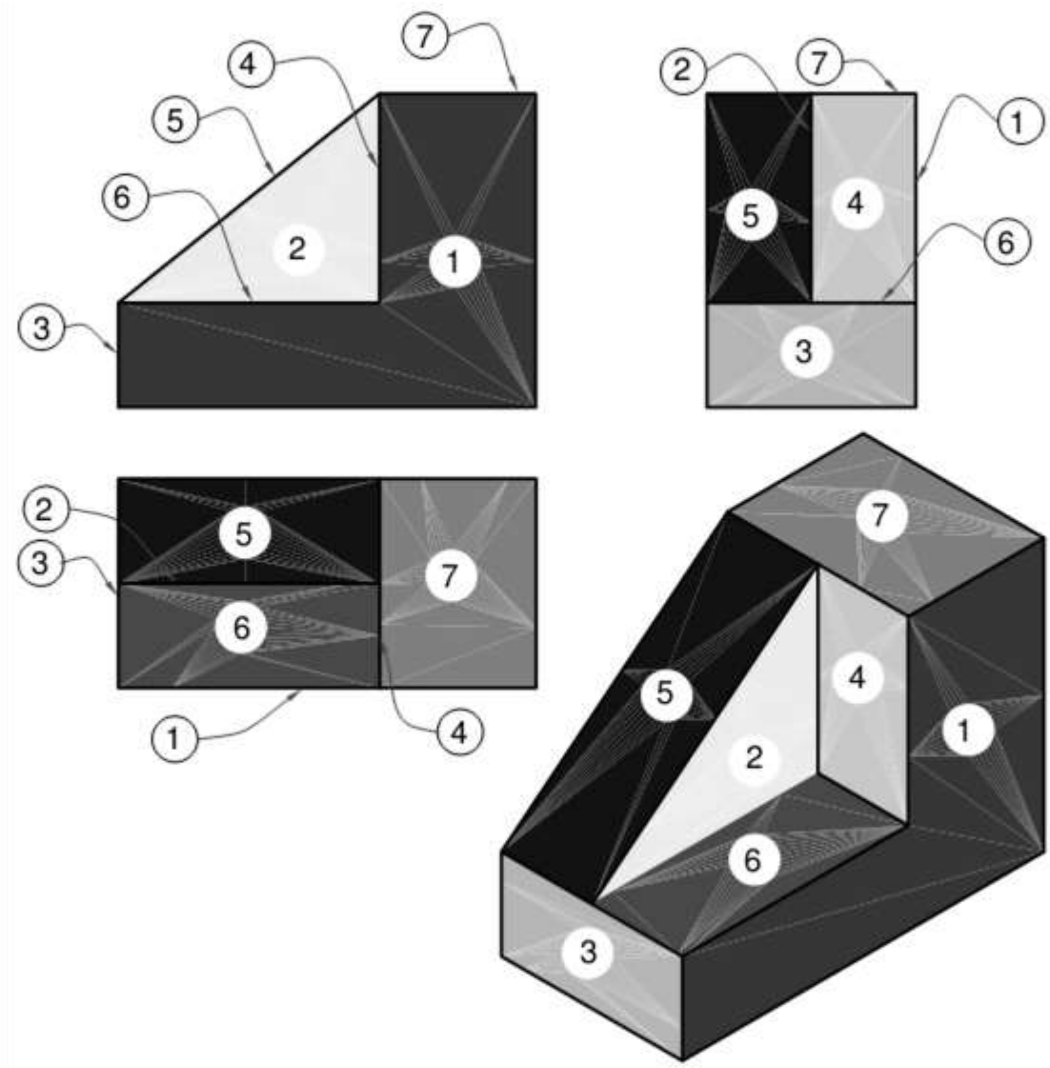
Sometimes the three views of an object are given. In one or more ways some projection lines may be found missing, in that case, these missing lines are drawn by comparing the projection of the other views. Therefore, it is defined as the lines which are added in the given orthographic view in order to complete the drawing of an object are known as missing lines. At this stage we can take the help of isometric sketch. Try to get a combined picture of the object in the mind and add missing lines on the drawing. Simple problems are easy visualize and there is no need of drawing isometric view. Following procedure can be adopted in order to identify missing lines of various object:

Step I: Draw the given orthographic views of the object with missing lines.

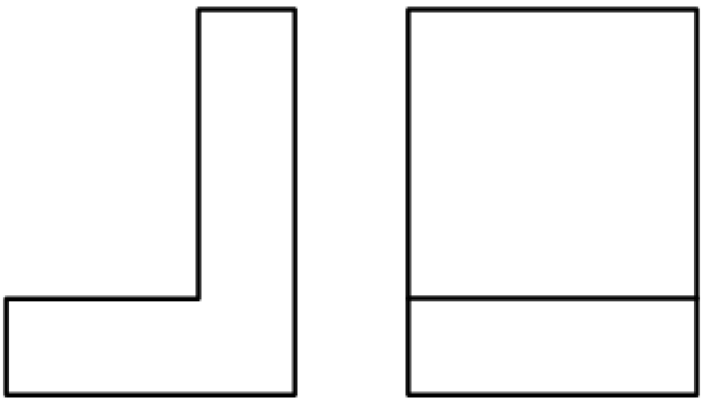
Step II: Draw the given orthographic views, firstly visualize an object and prepare rough pictorial view.

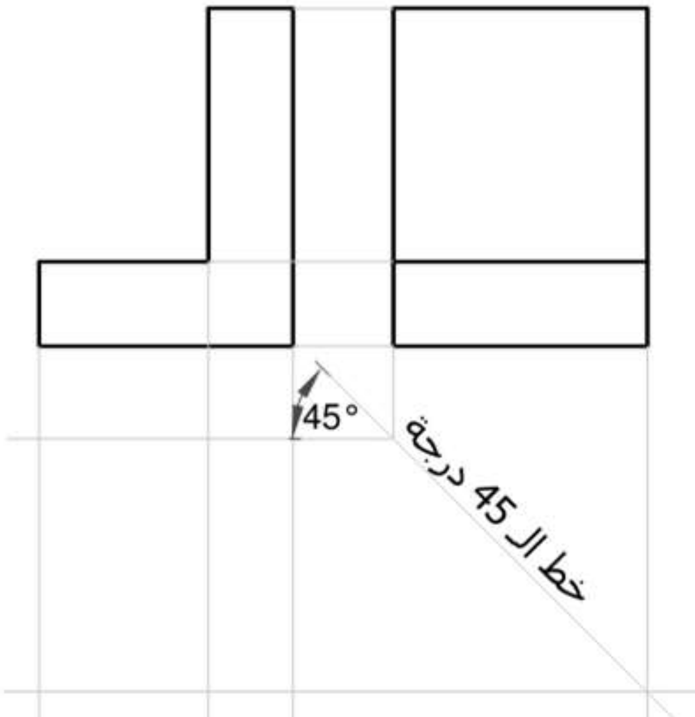
Step III: Now from this rough pictorial view draw the orthographic view and compare it with the given orthographic view.

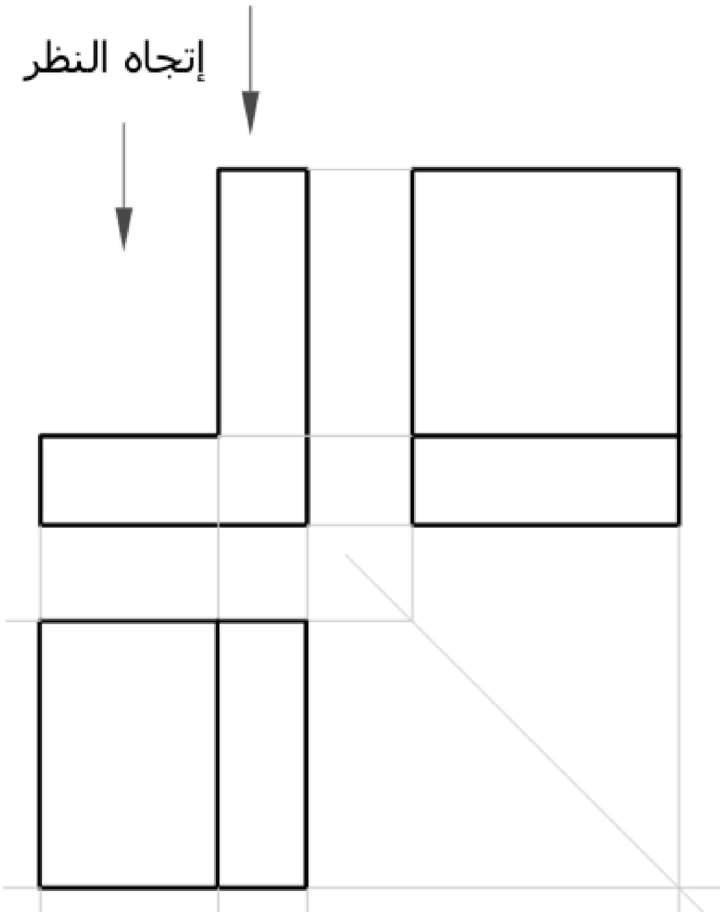
Step IV: Read carefully each line in each view and draw the required missing lines on the given orthographic view.



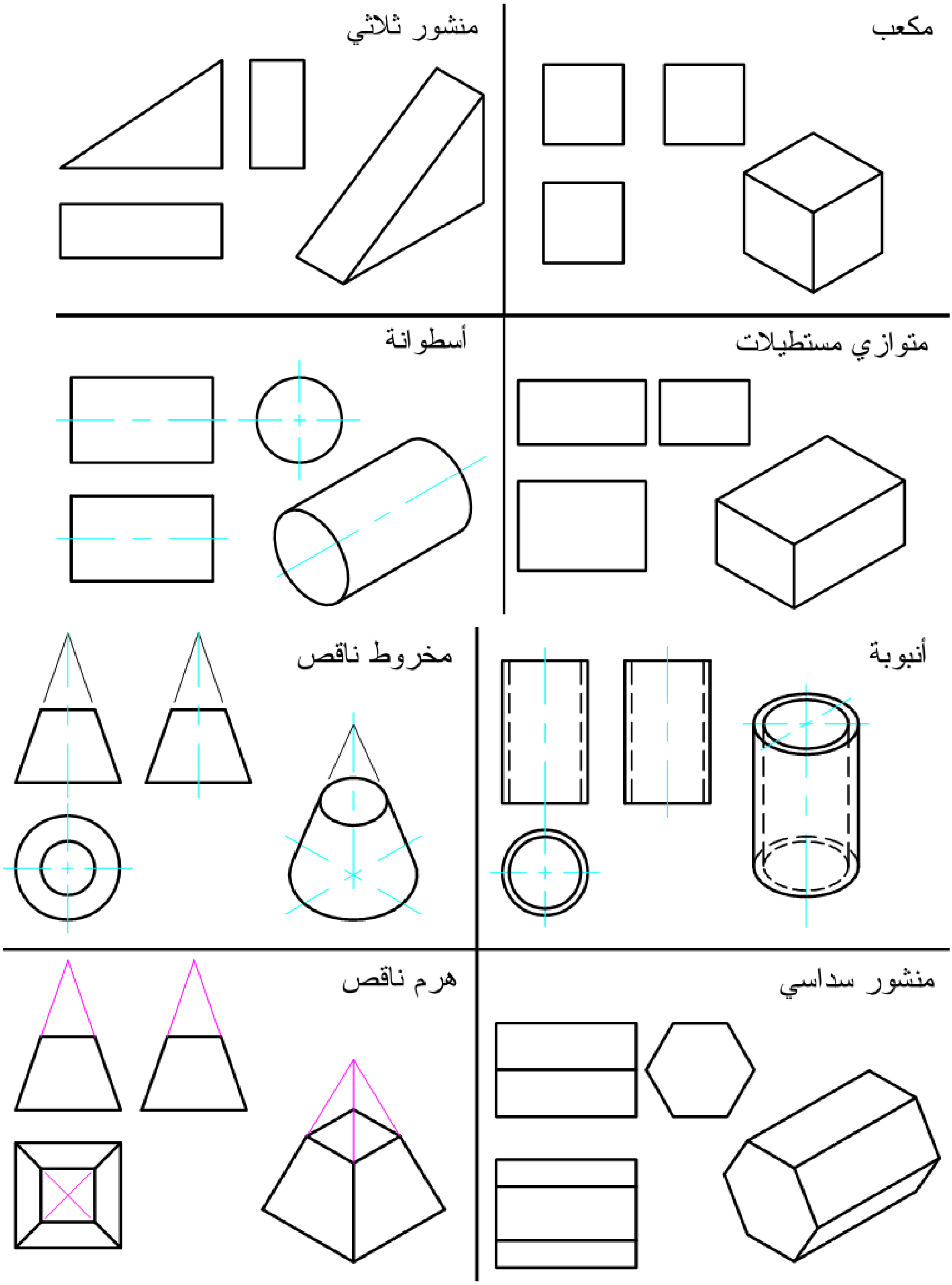
Example 6.1



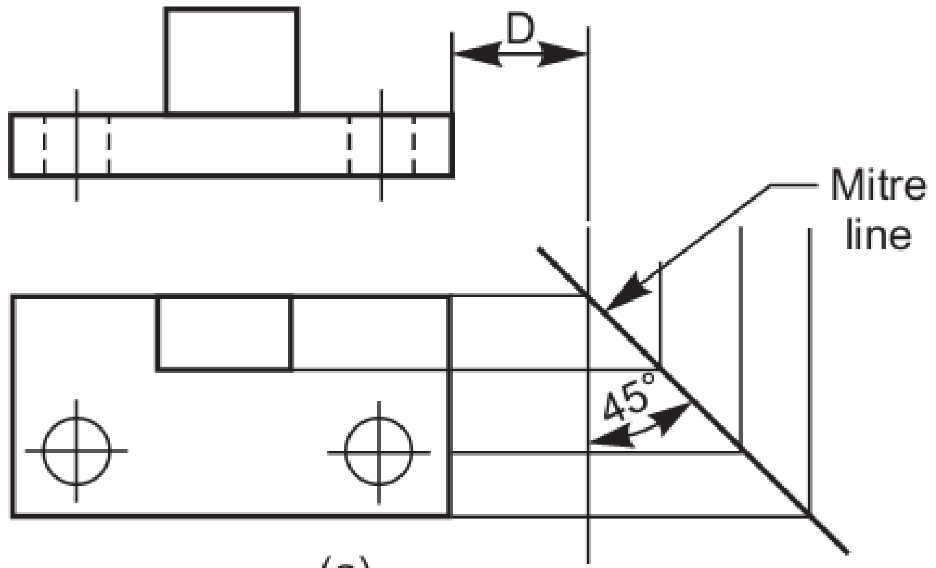

45°
خط الـ 45 درجة


إتجاه النظر

- Standared shapes:



Example 6.2



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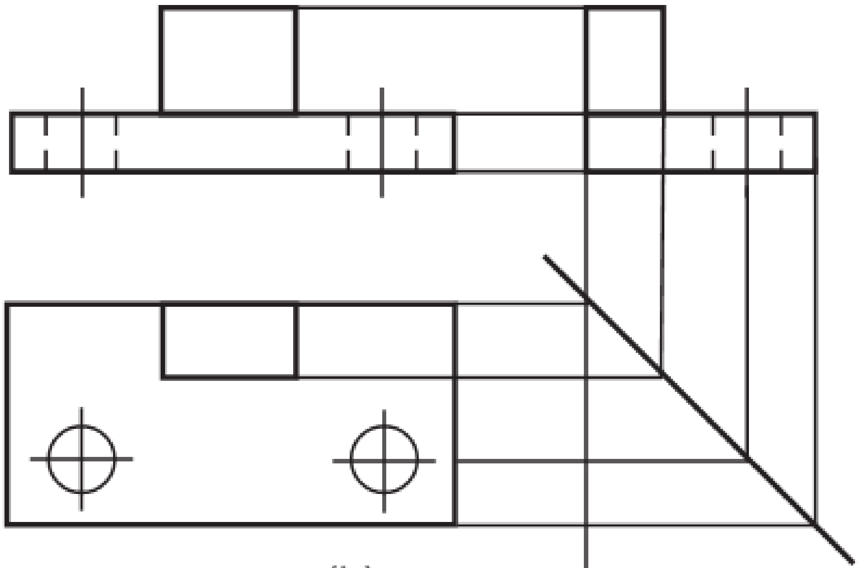
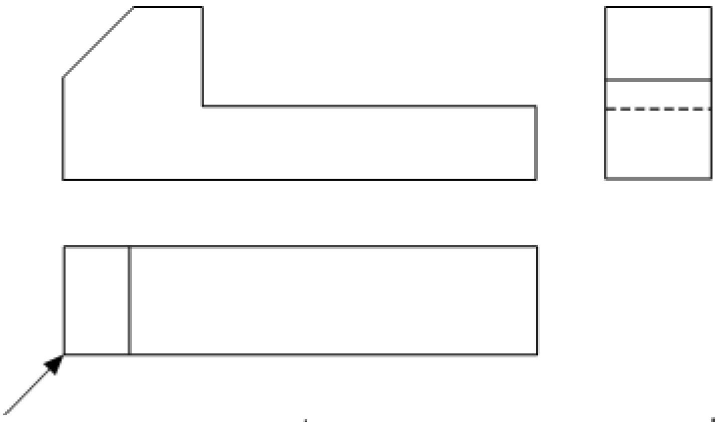
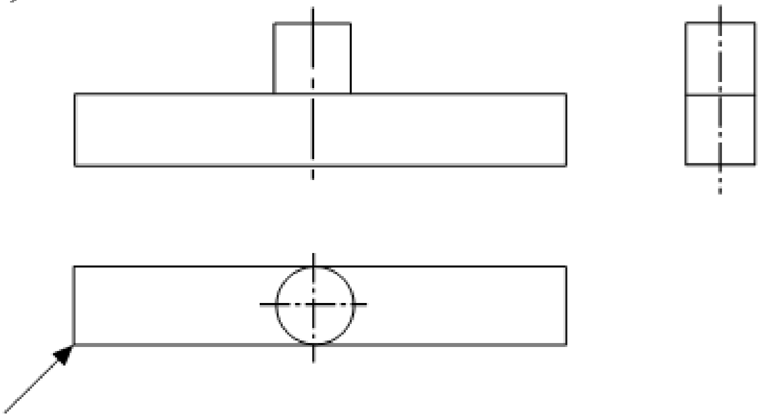


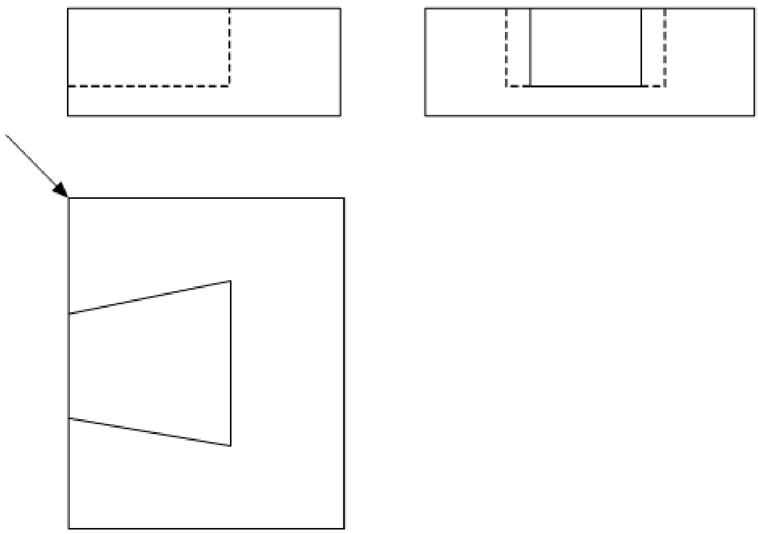
(b)

-

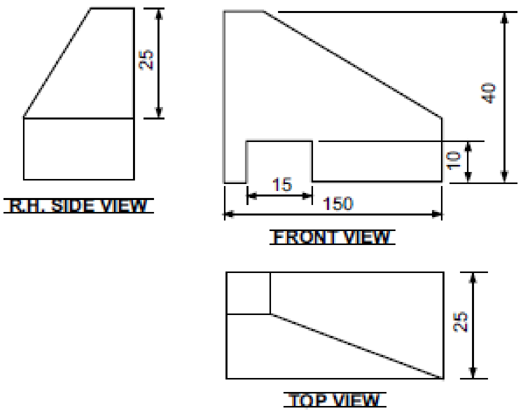
- The figure shows incomplete orthographic projections of an object. Draw the missing lines and complete the orthographic projection.







Problem 1. Figure shows an incomplete orthographic projection of an object. Draw the missing lines and complete the orthographic projection.



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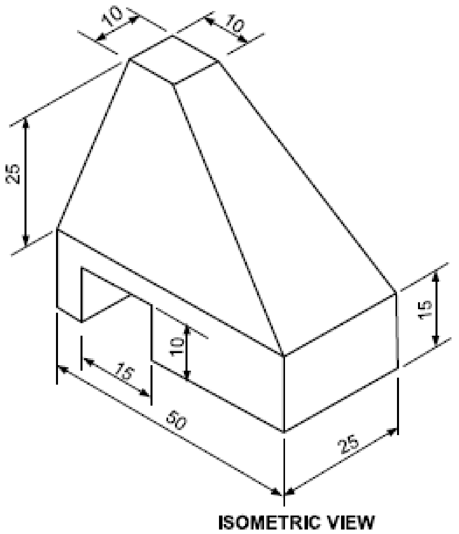

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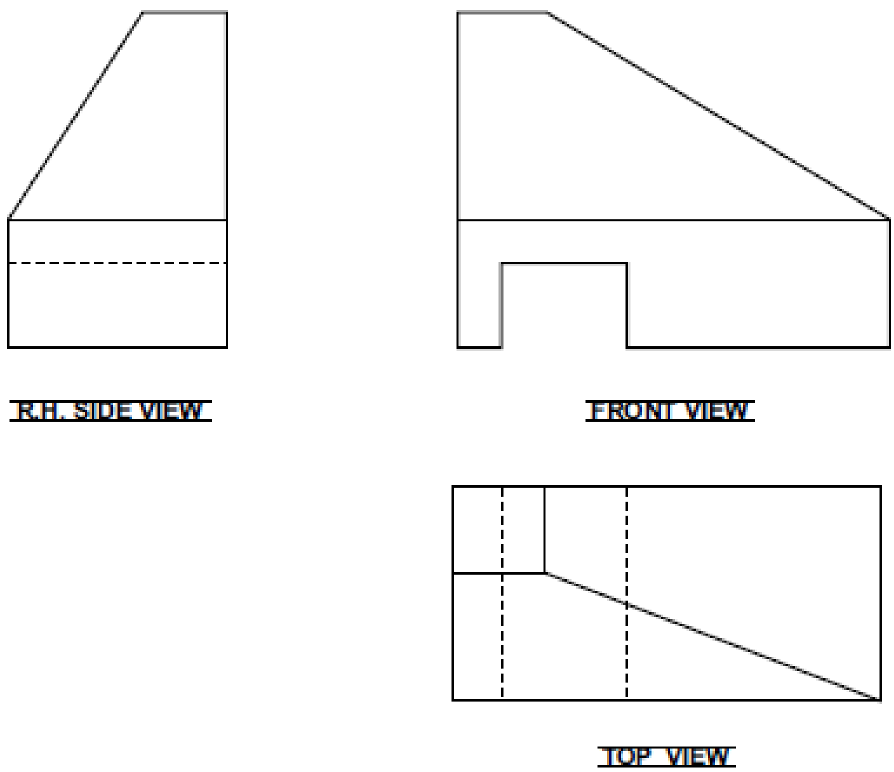
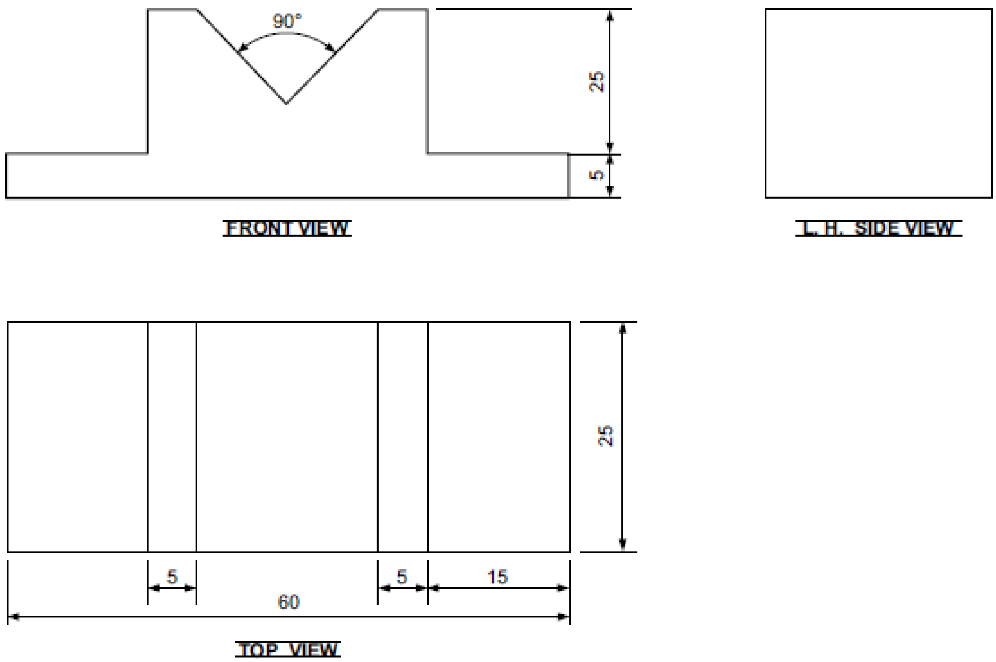
Solution:



Problem 2. Figure shows incomplete orthographic projections of an object. Draw the missing line and complete the orthographic projection.



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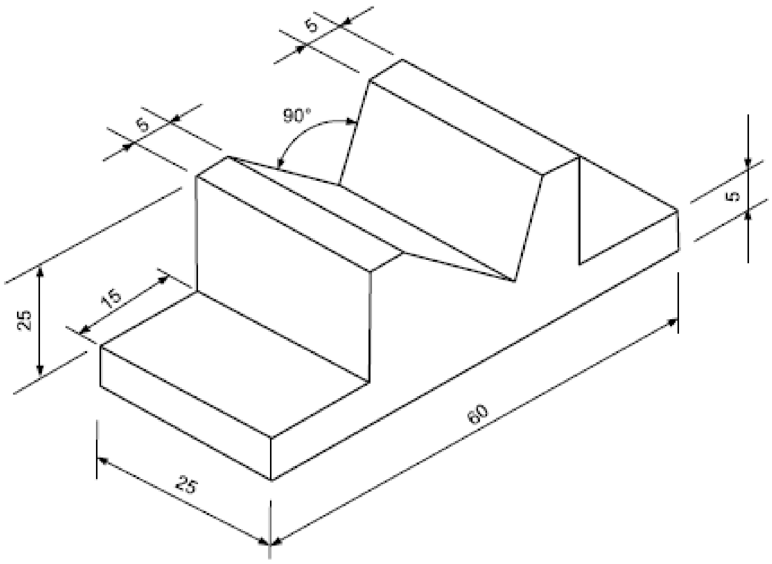
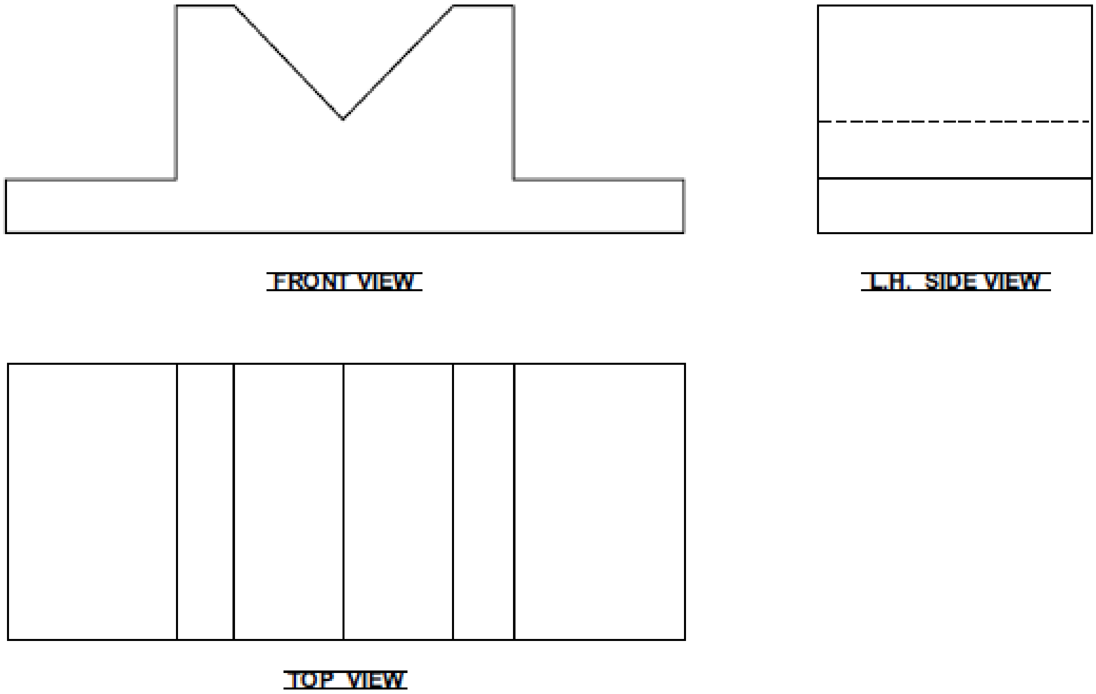
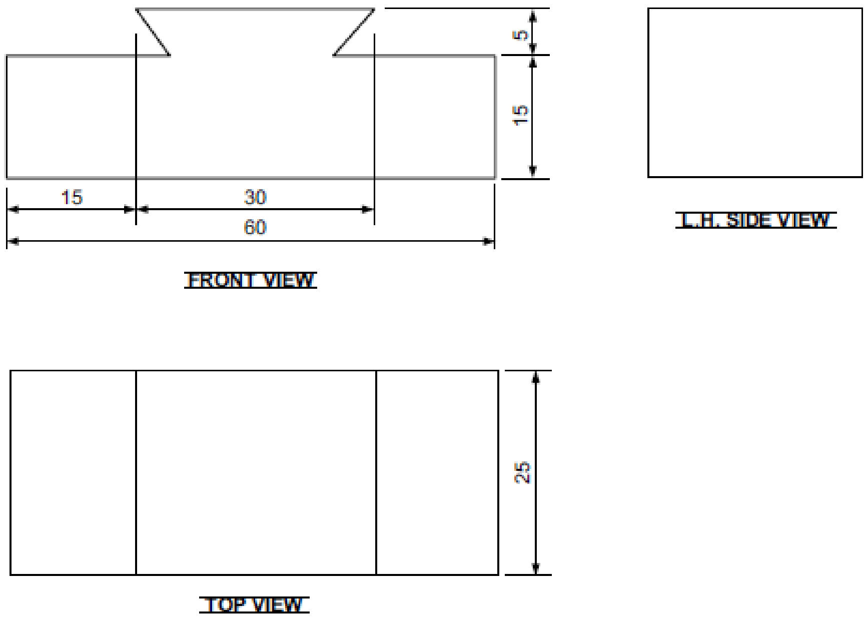


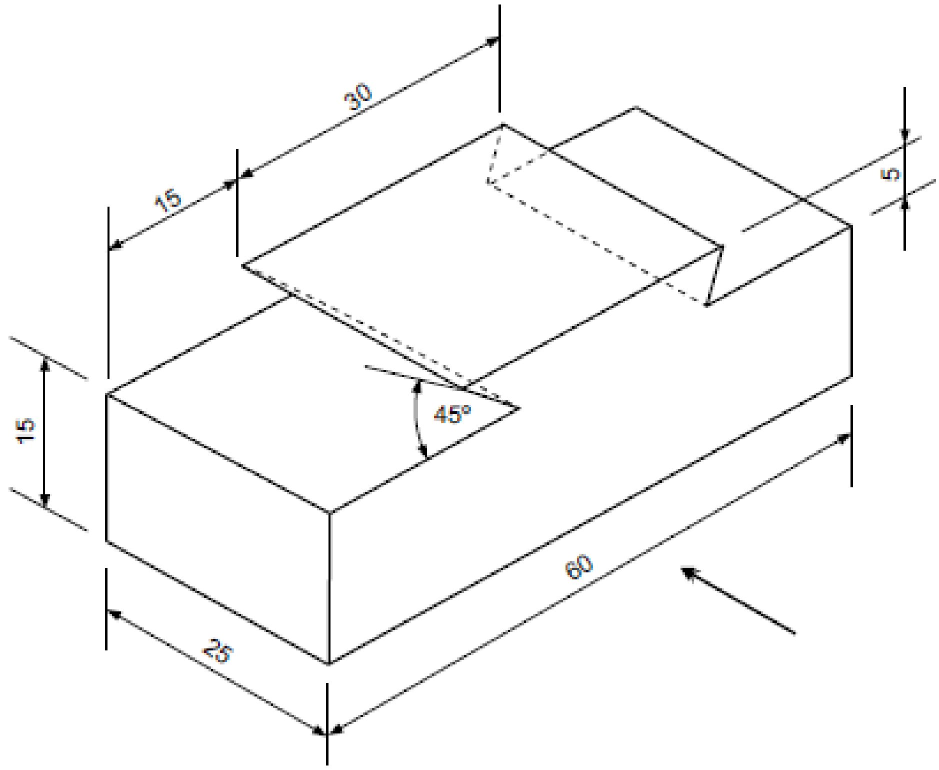
(ii) ISOMETRIC VIEW

Solution



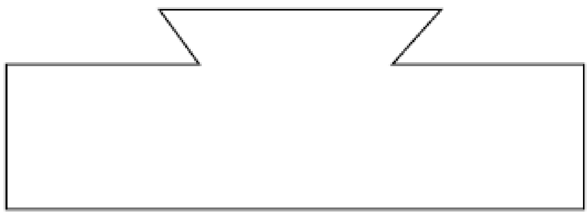
Problem 3. Figure shows incomplete orthographic projections of an object. Draw the missing lines and complete the orthographic projection.



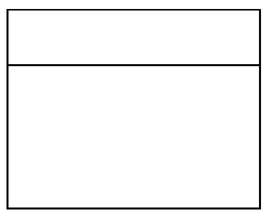


(ii) ISOMETRIC VIEW

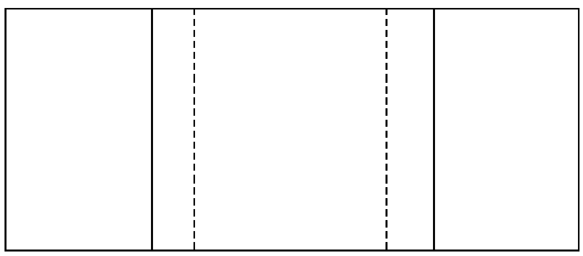
Solution:



FRONT VIEW



L.H. SIDE VIEW



TOP VIEW

6.2 MISSING VIEW

A missing view is defined as the view which is added in the given orthographic view in order to complete the drawing of an object. Following procedure can be adopted in order to identify missing views of various objects:

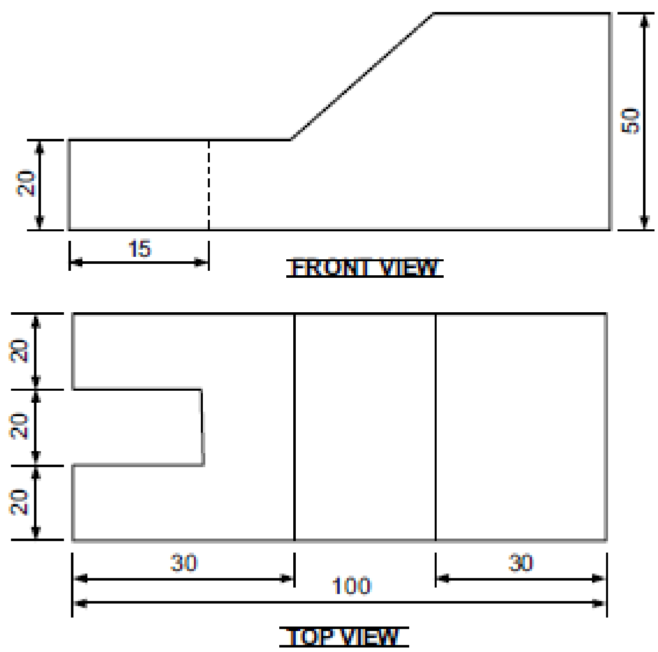
Step I. Draw the given orthographic views of the object with missing view.

Step II. For simple object, draw the missing view directly without drawing the rough pictorial view.

Step III. For complicated object, draw the rough pictorial view in order to understand the shape of the object.

Step IV. After completing the pictorial view, draw the required missing view of the object.

Problem 4. Figure shows the incomplete orthographic projection of an object. Draw the missing view.



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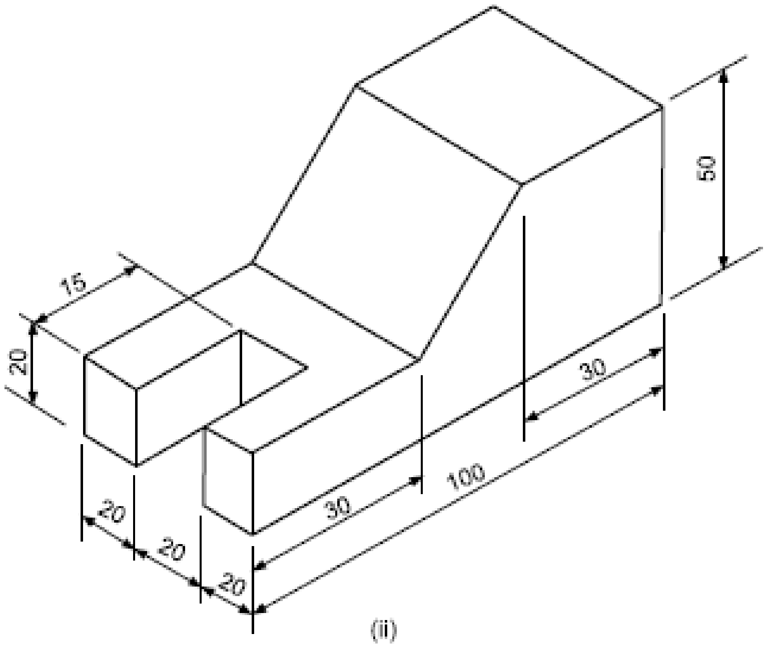
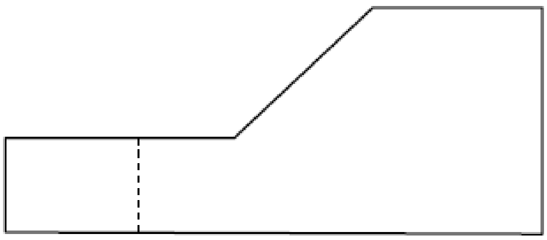
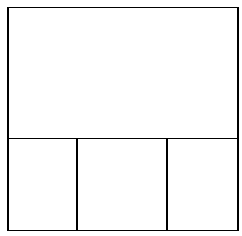


(ii)

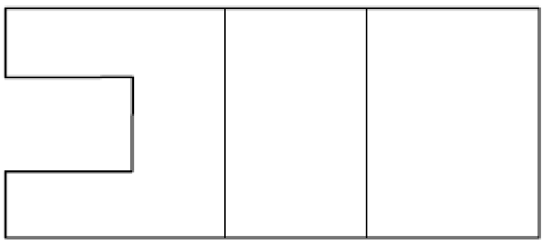
Solution:



FRONT VIEW

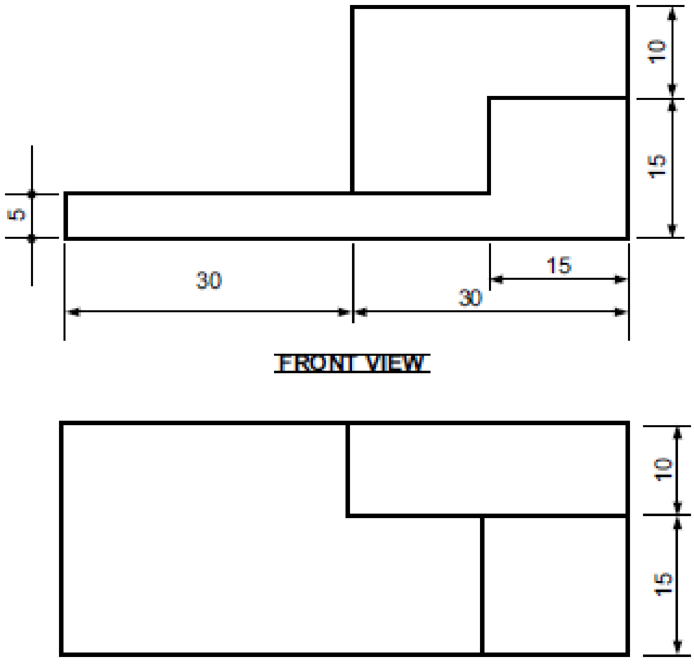


L.H. SIDE VIEW



TOP VIEW

Problem 5. Figure shows the incomplete orthographic projections of an object. Draw the missing view.



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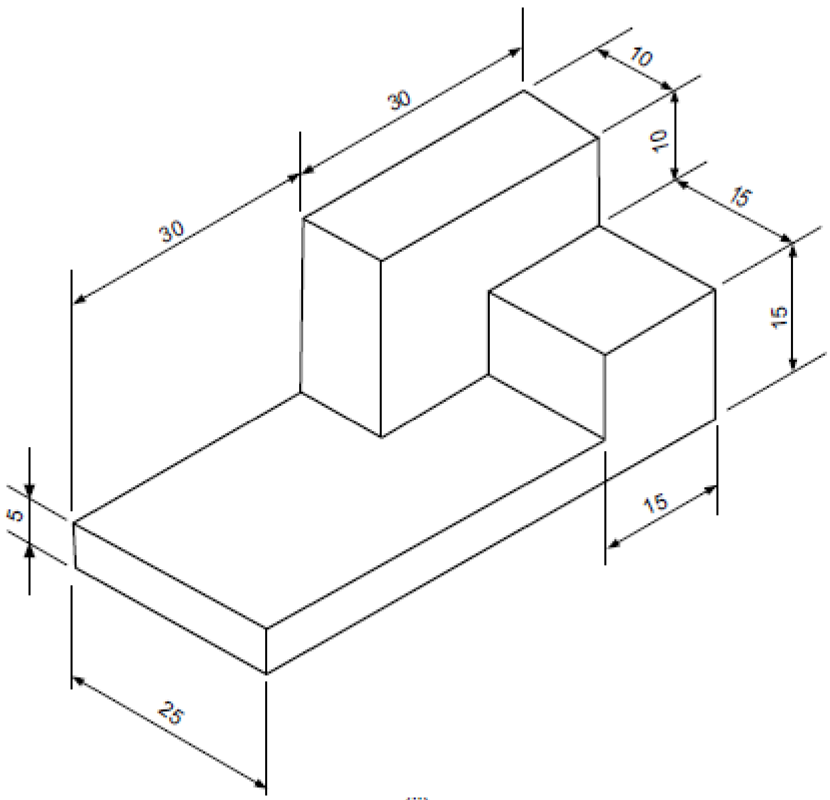
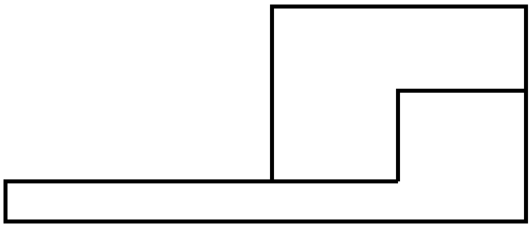
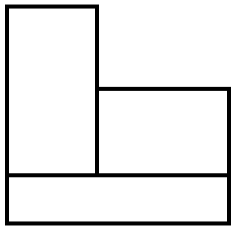


(ii)

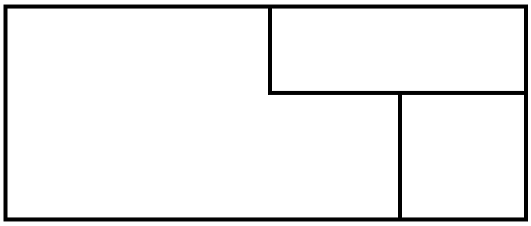
Solution:



FRONT VIEW



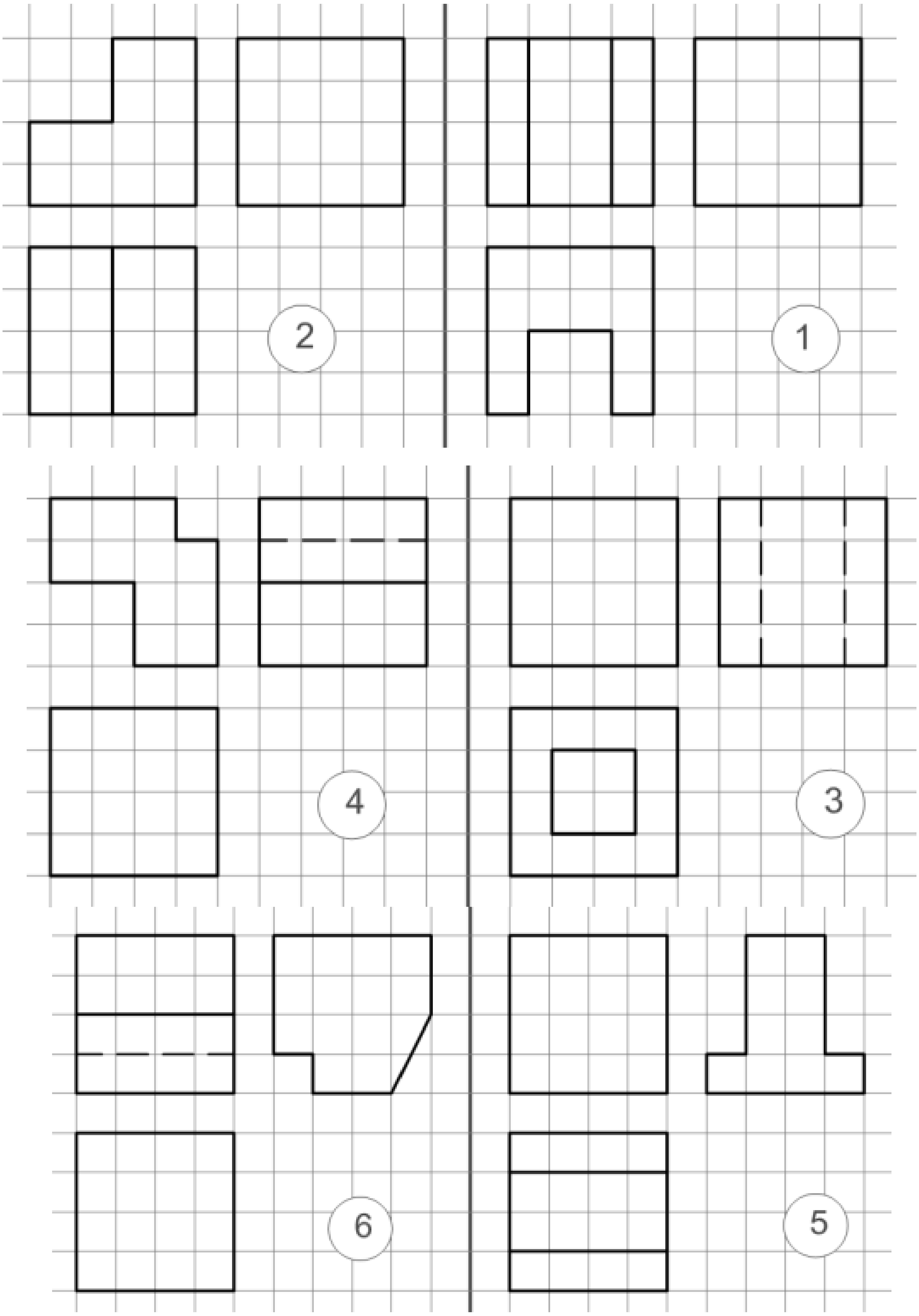
SIDE VIEW

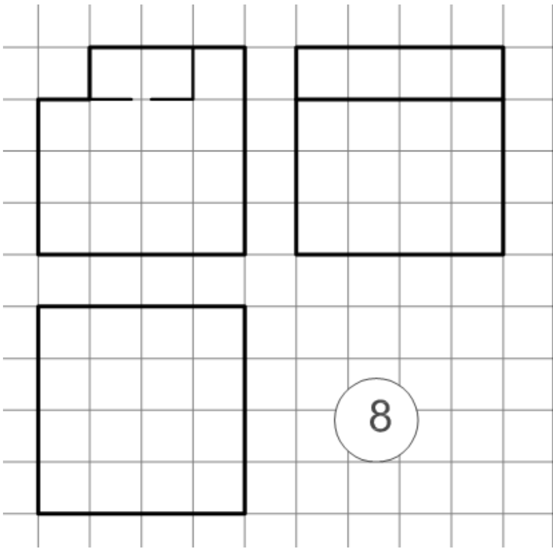


TOP VIEW

Exercises

The figure shows incomplete orthographic projections of an object. Draw the missing lines and complete the orthographic projection.





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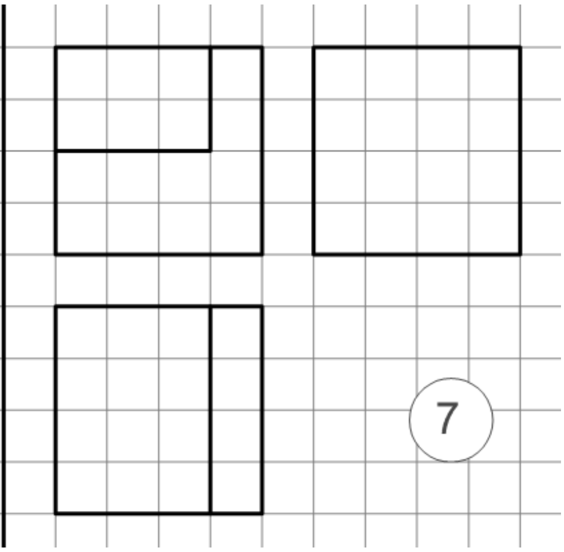

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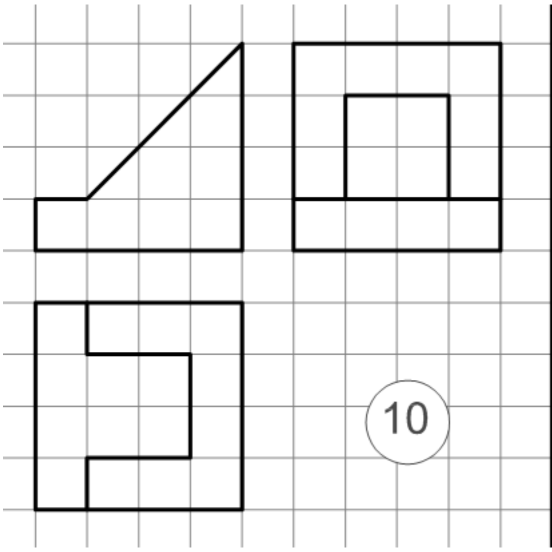

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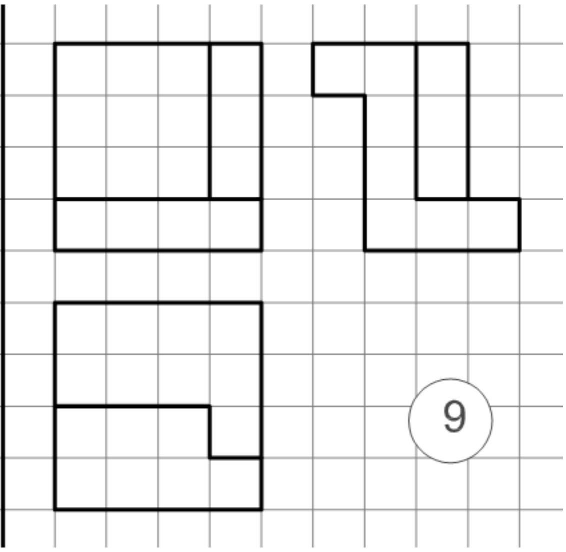

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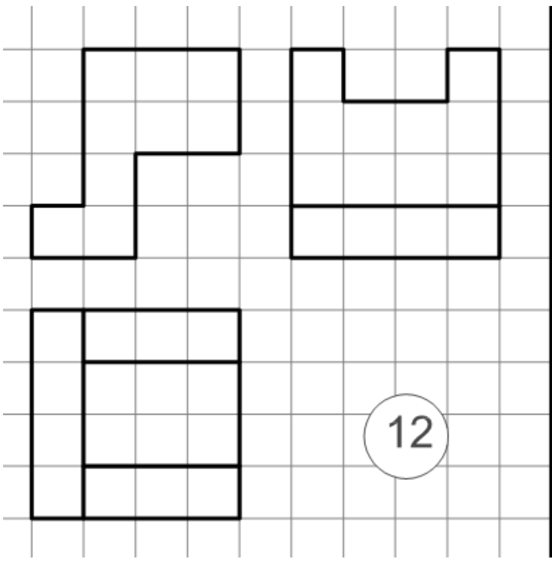

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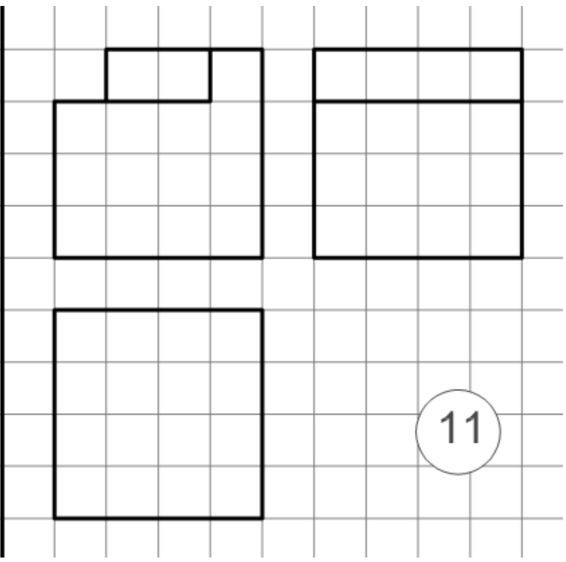

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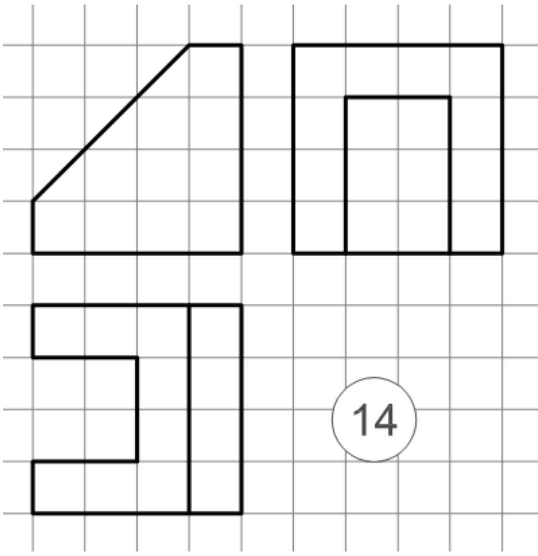
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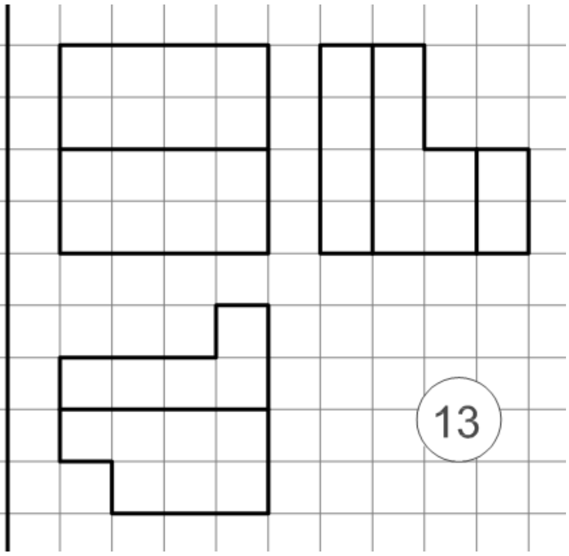
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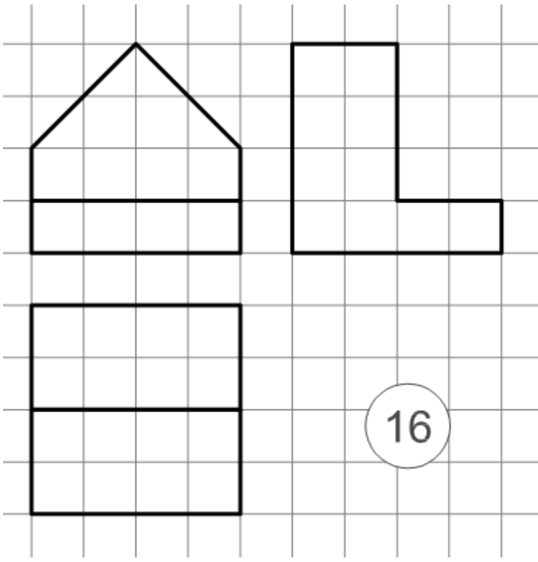
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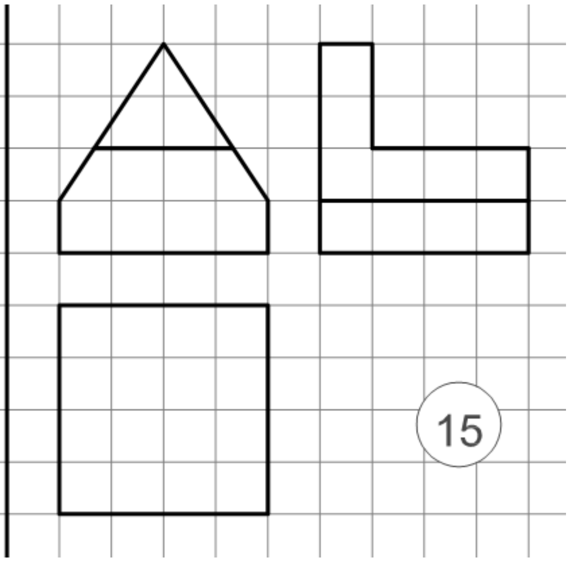
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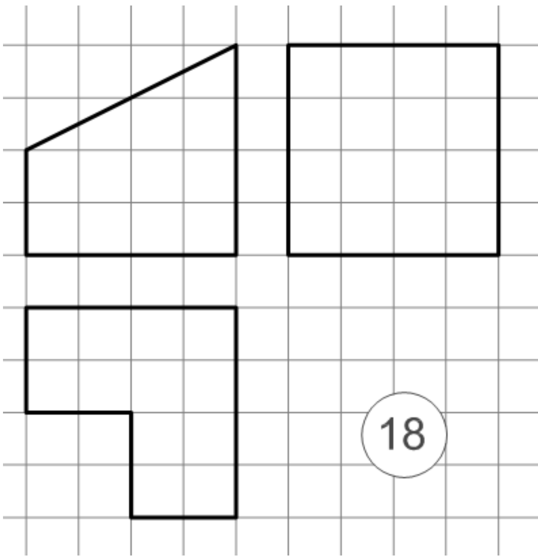
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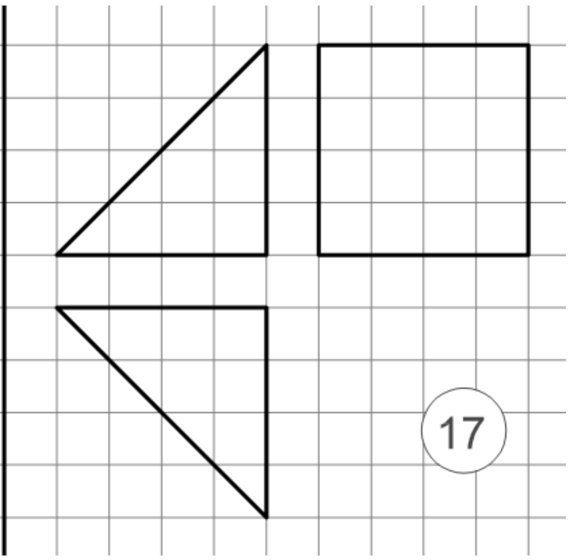
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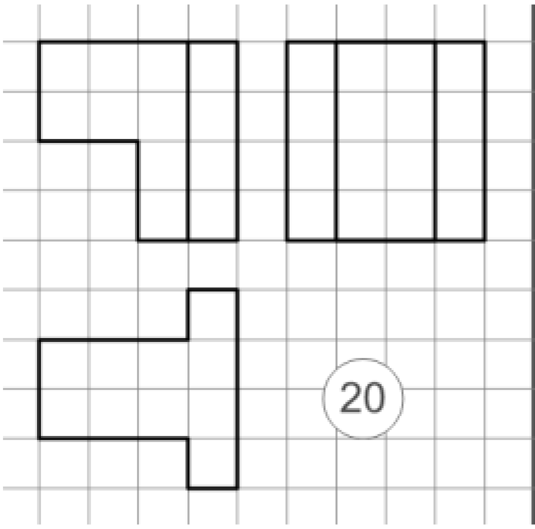
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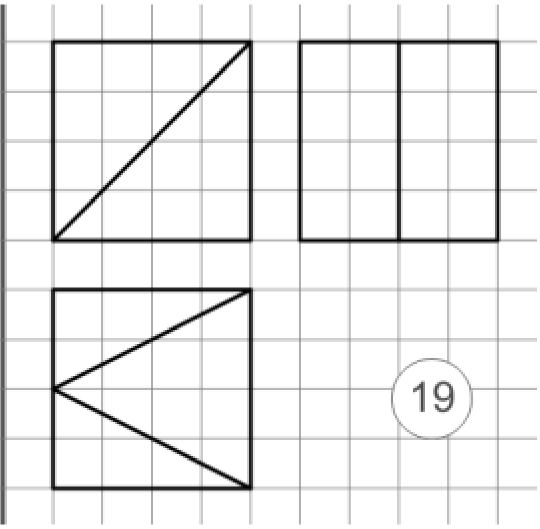
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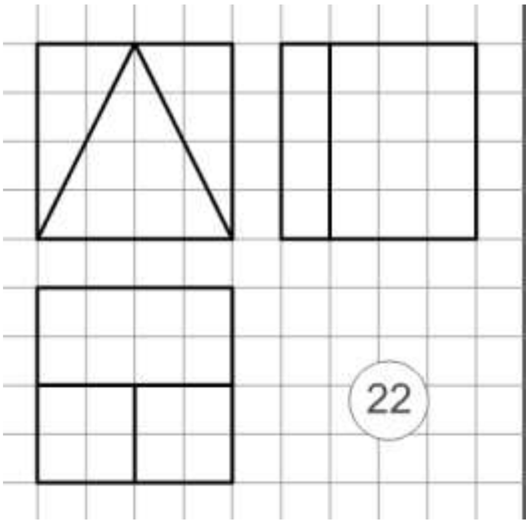
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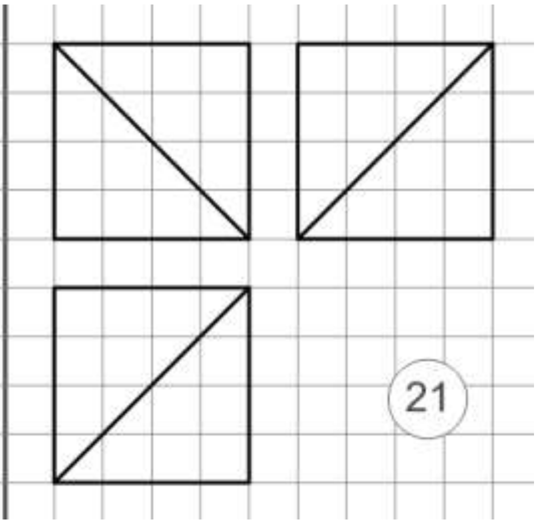
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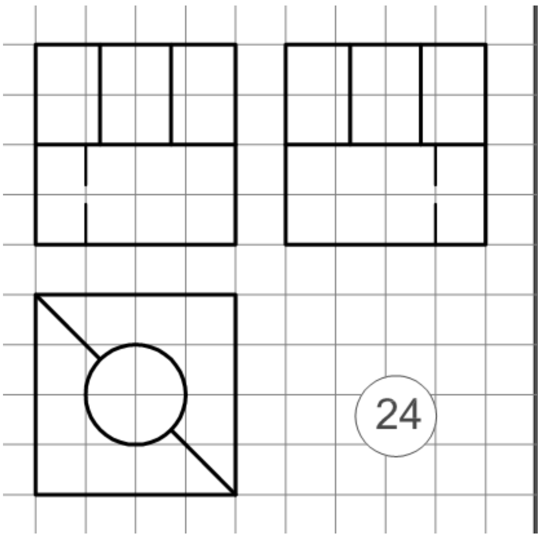
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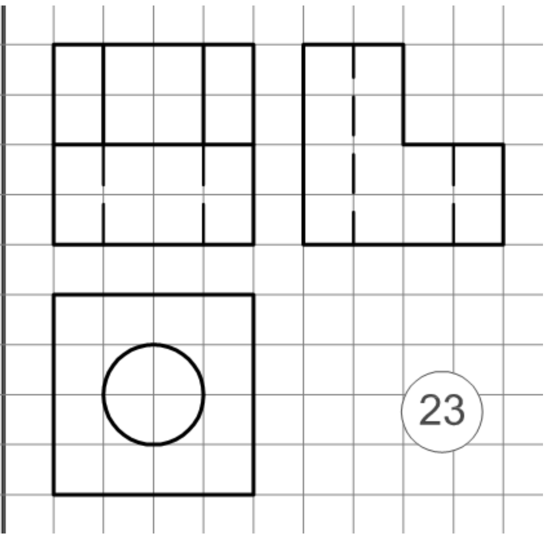
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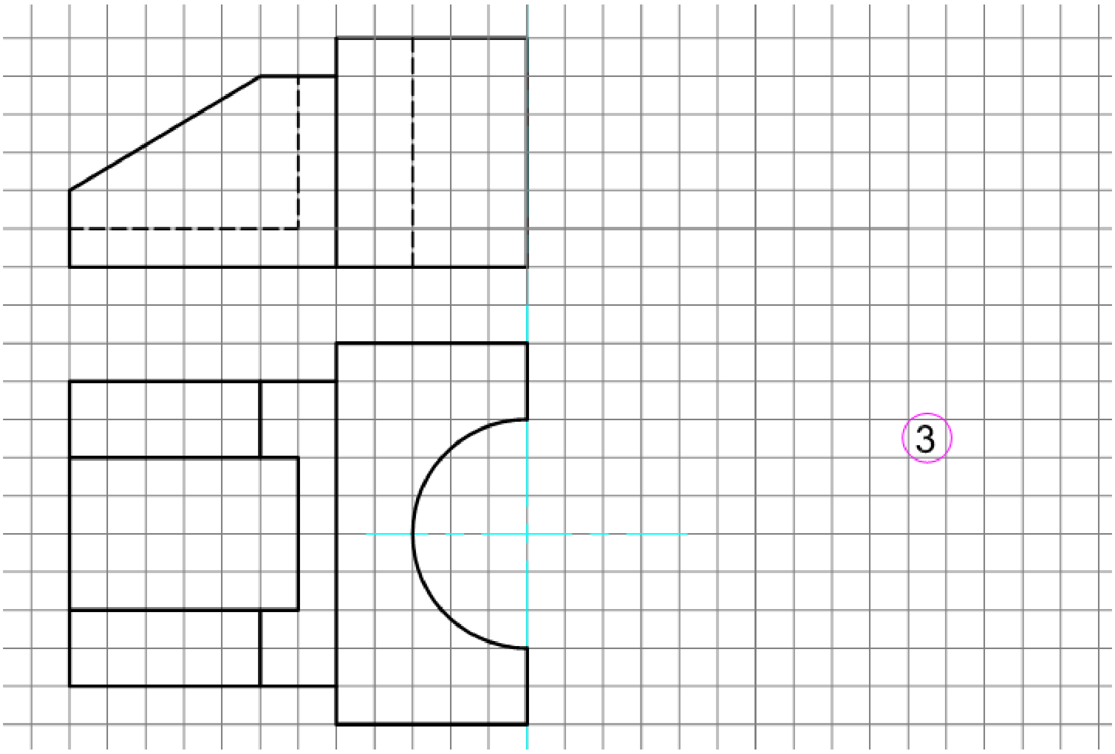
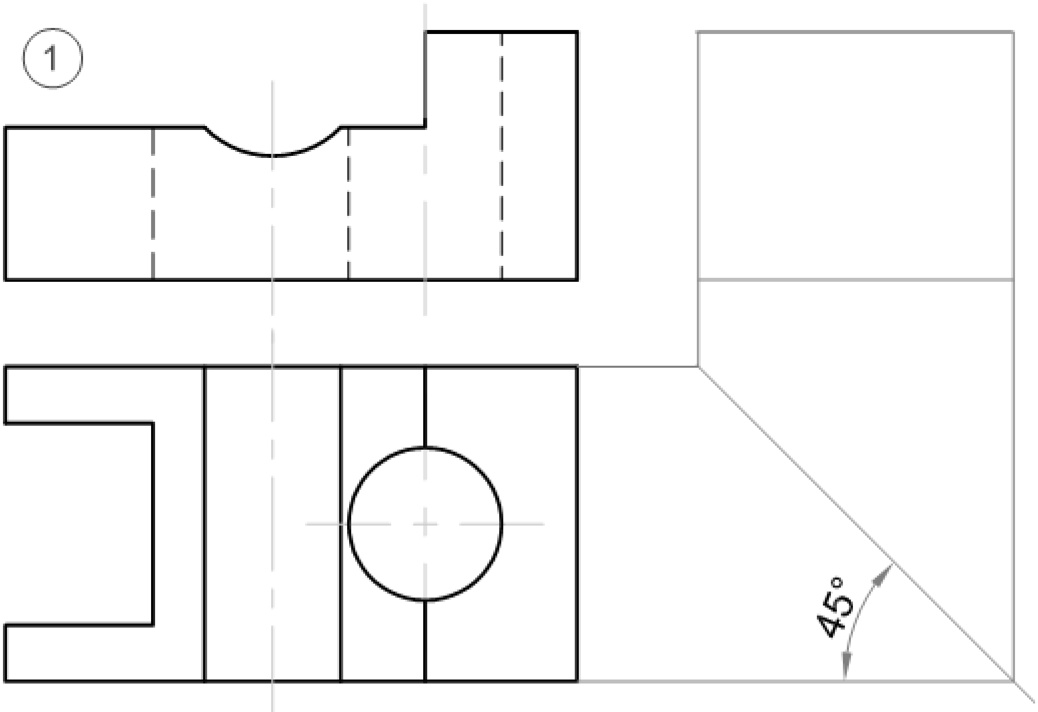
- The figure shows the incomplete orthographic projection of an object. Draw the missing view.





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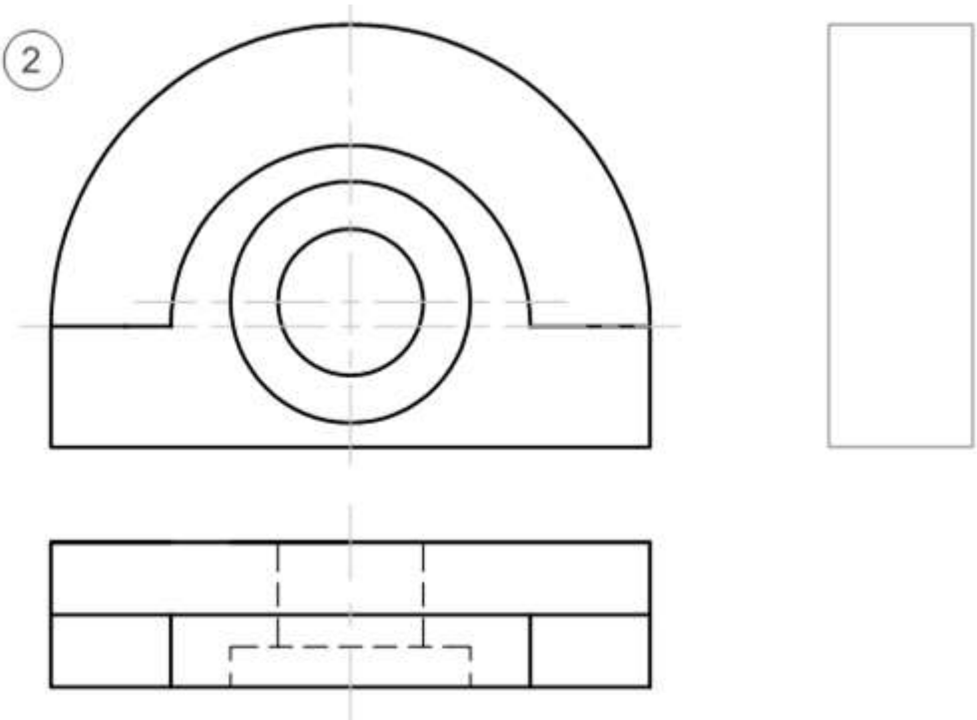

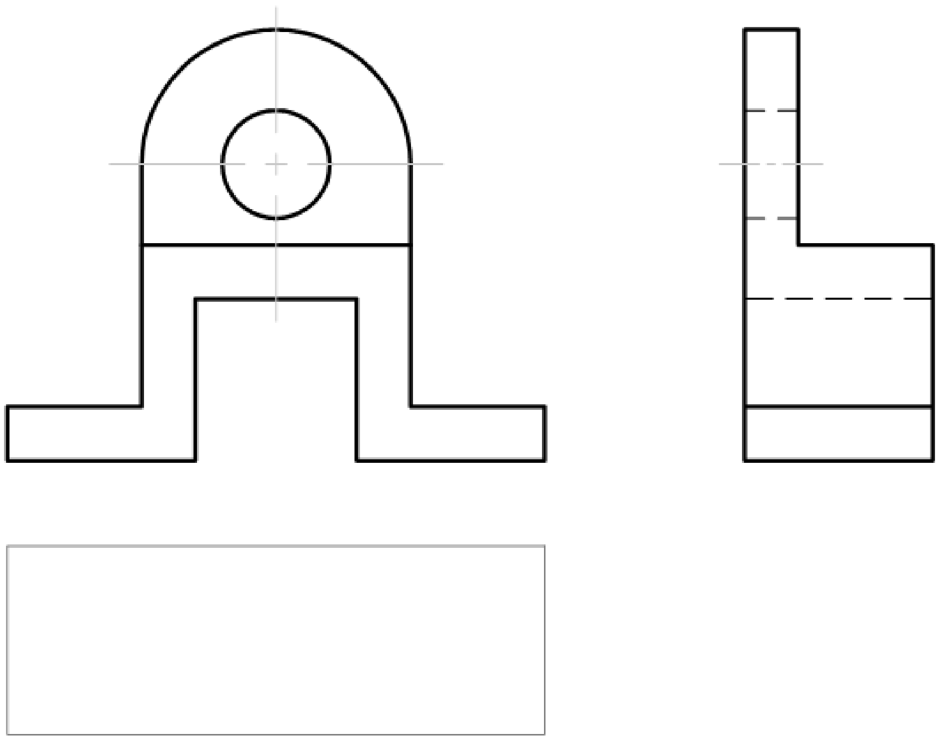

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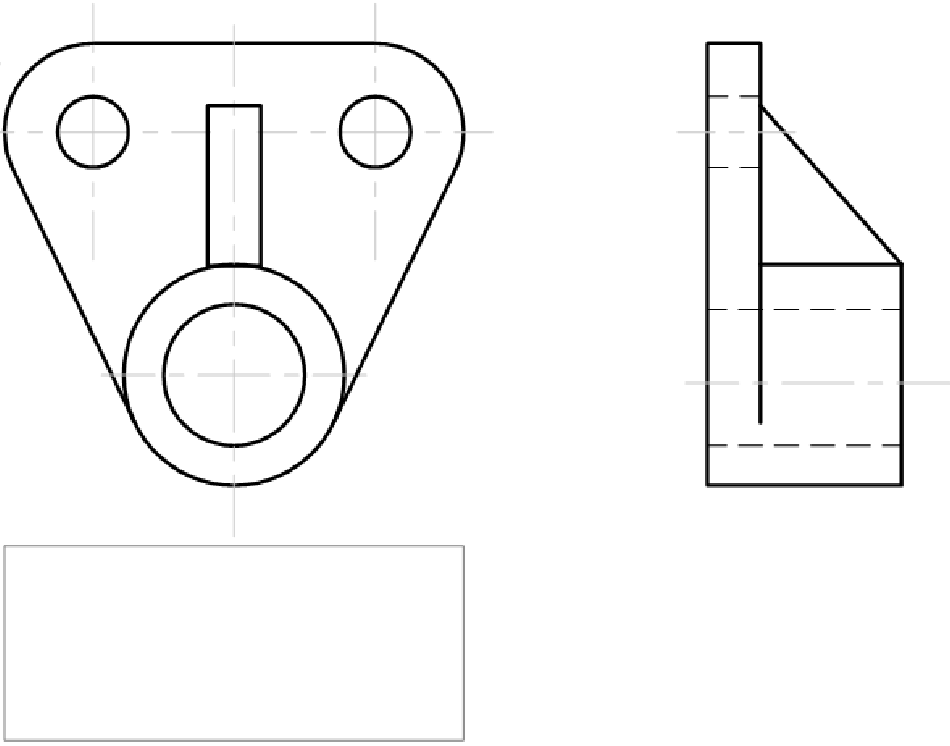

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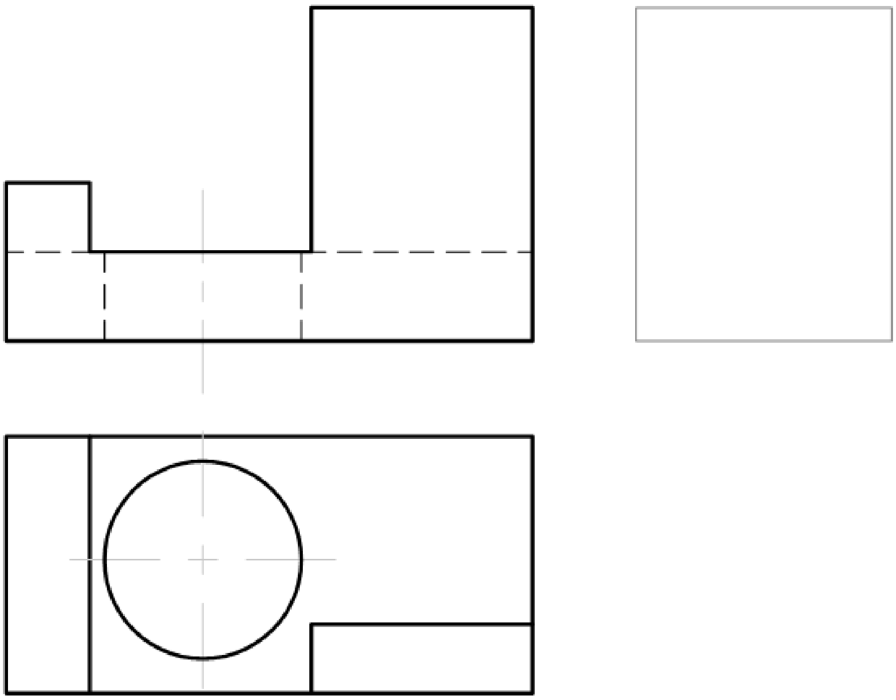

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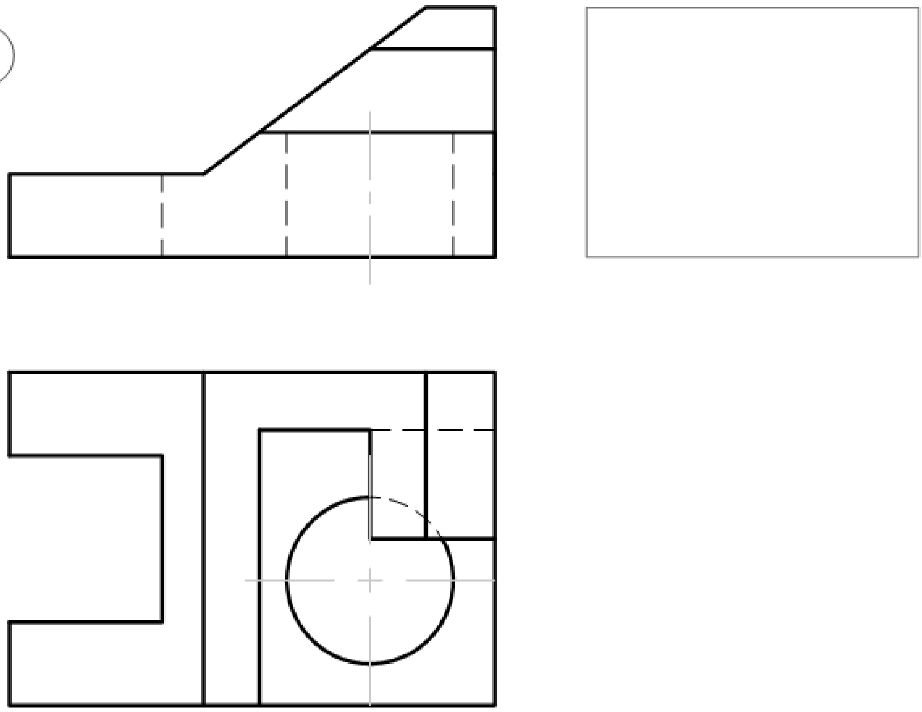

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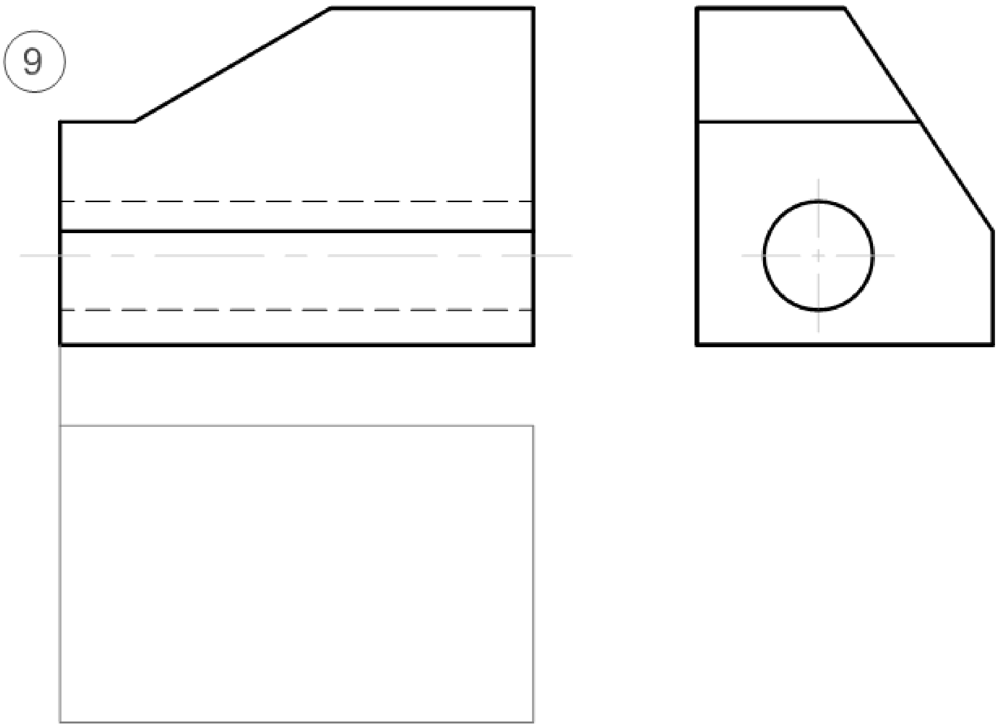

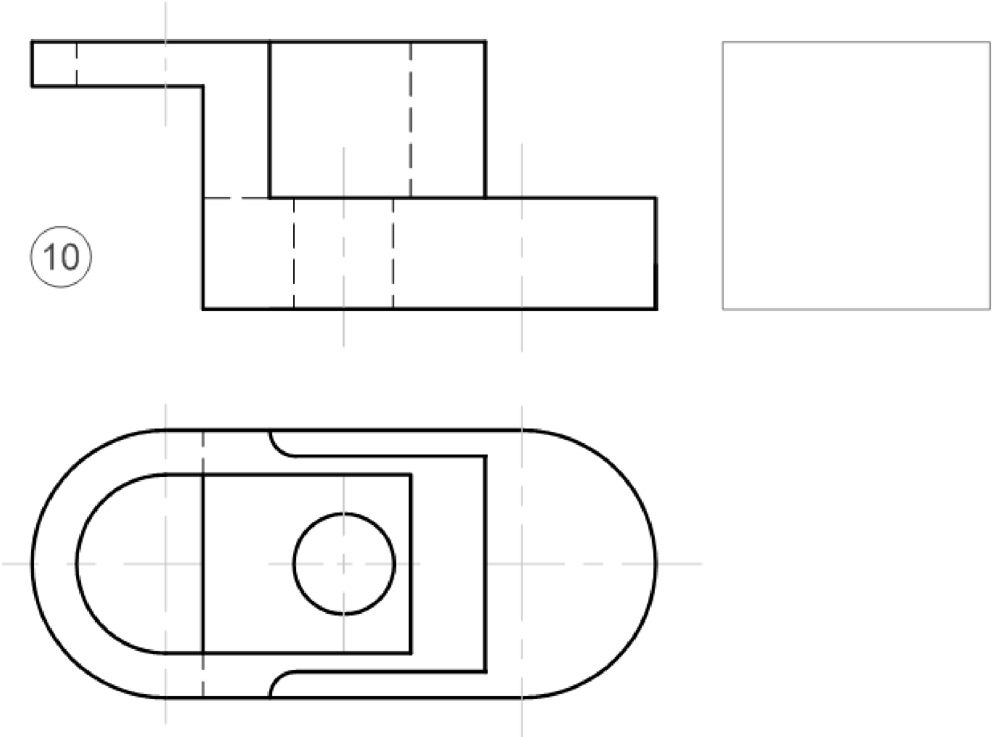
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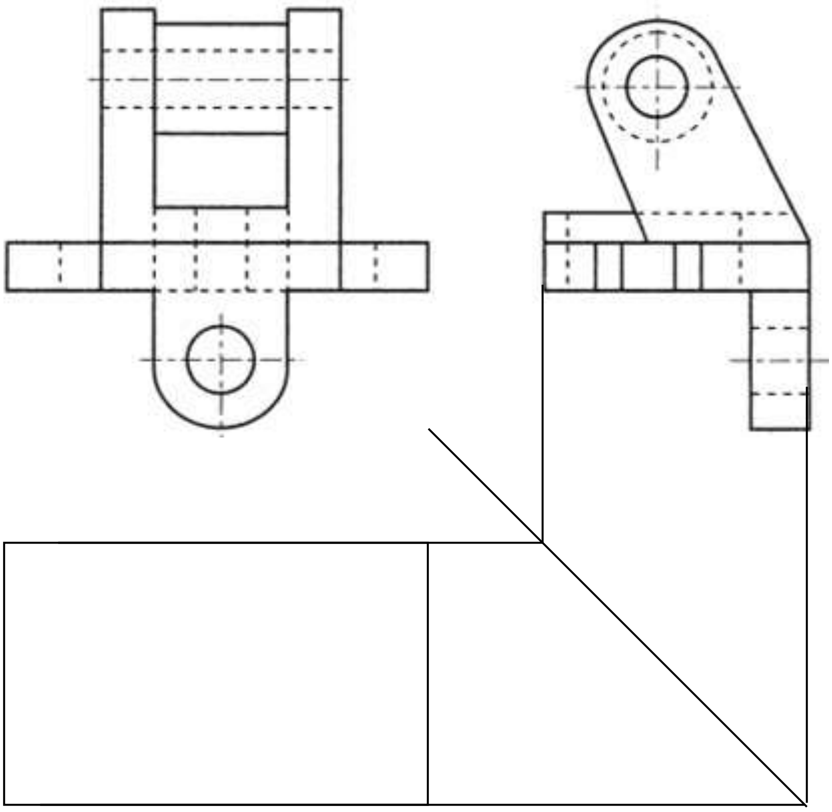
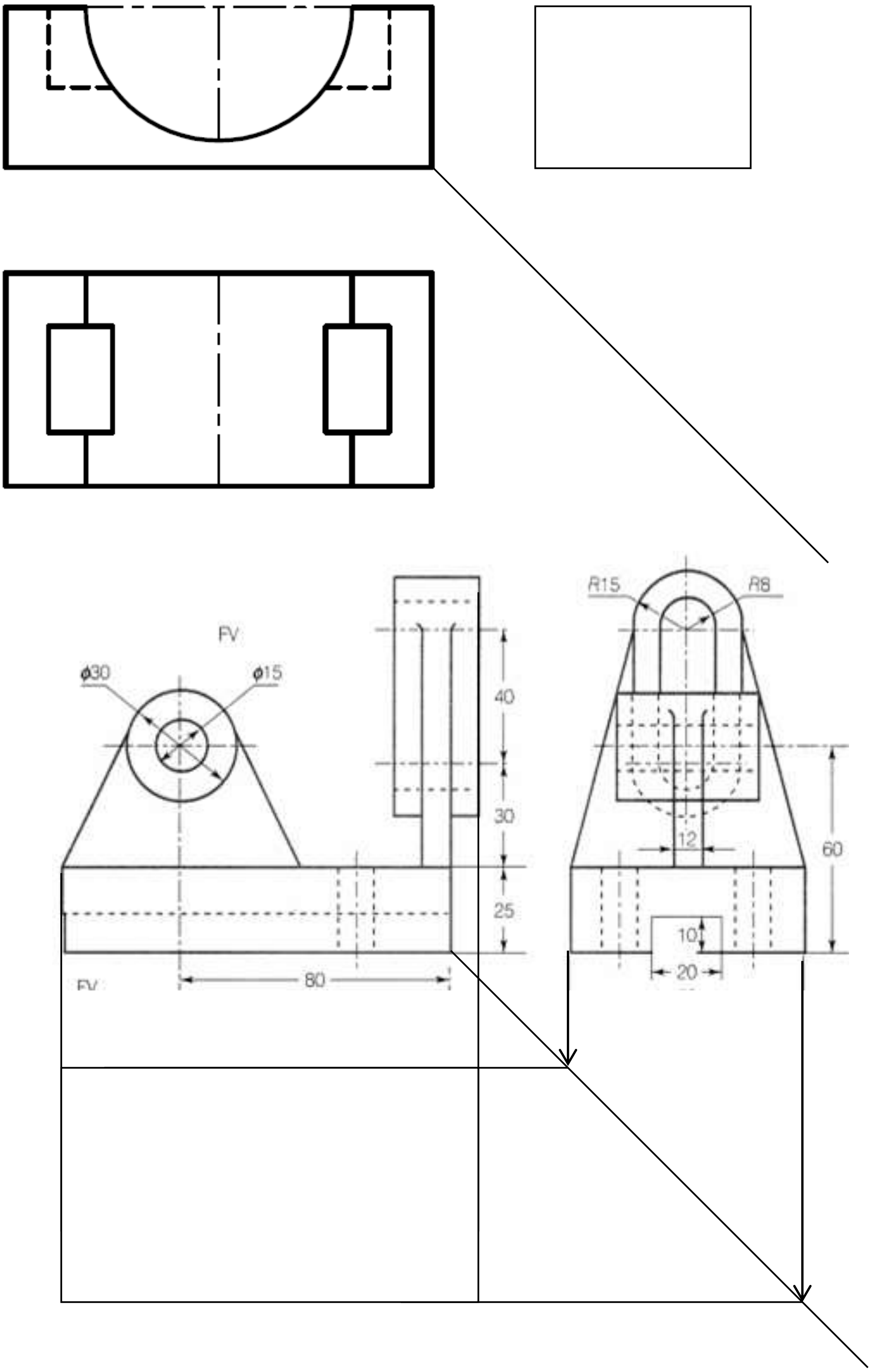
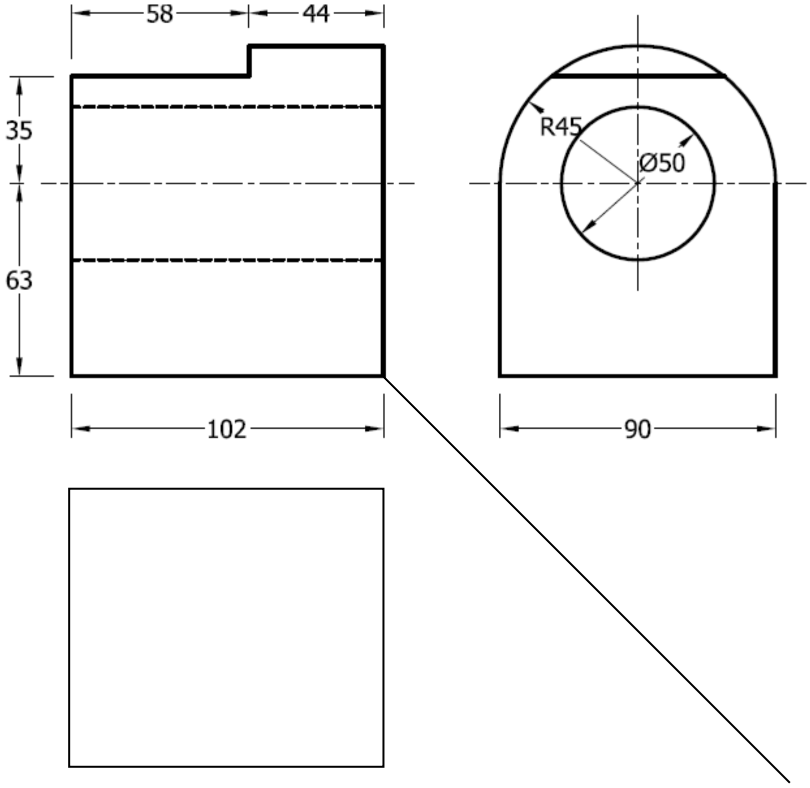
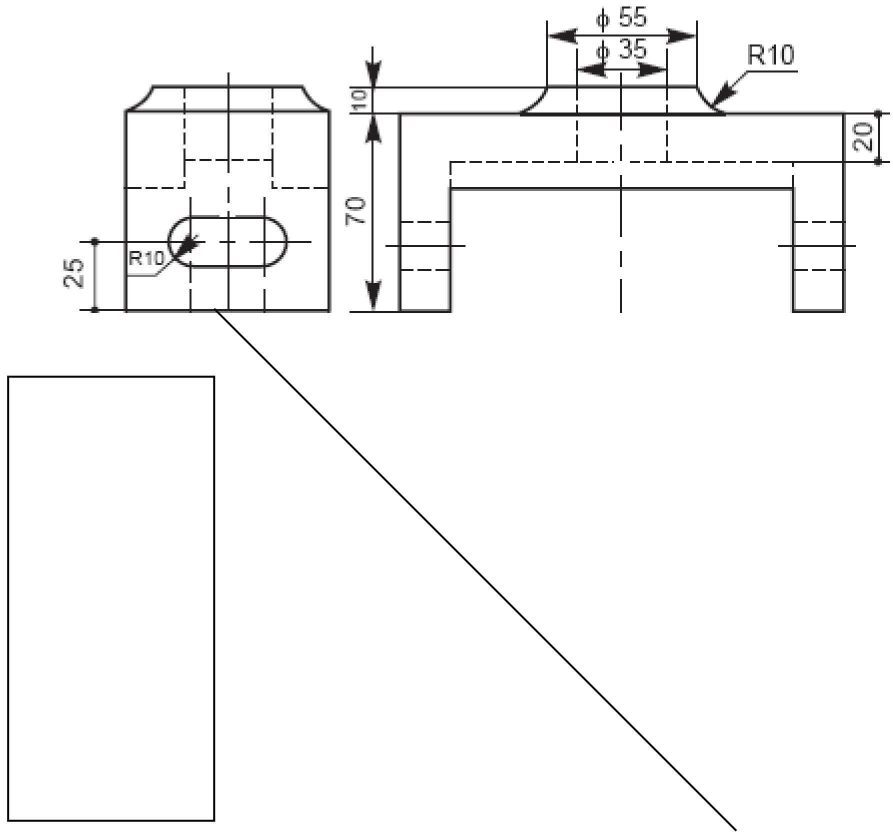
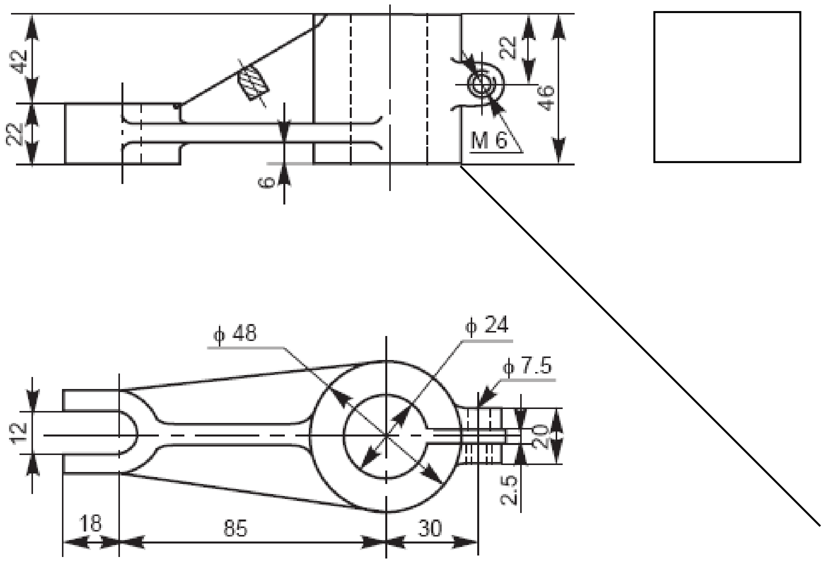
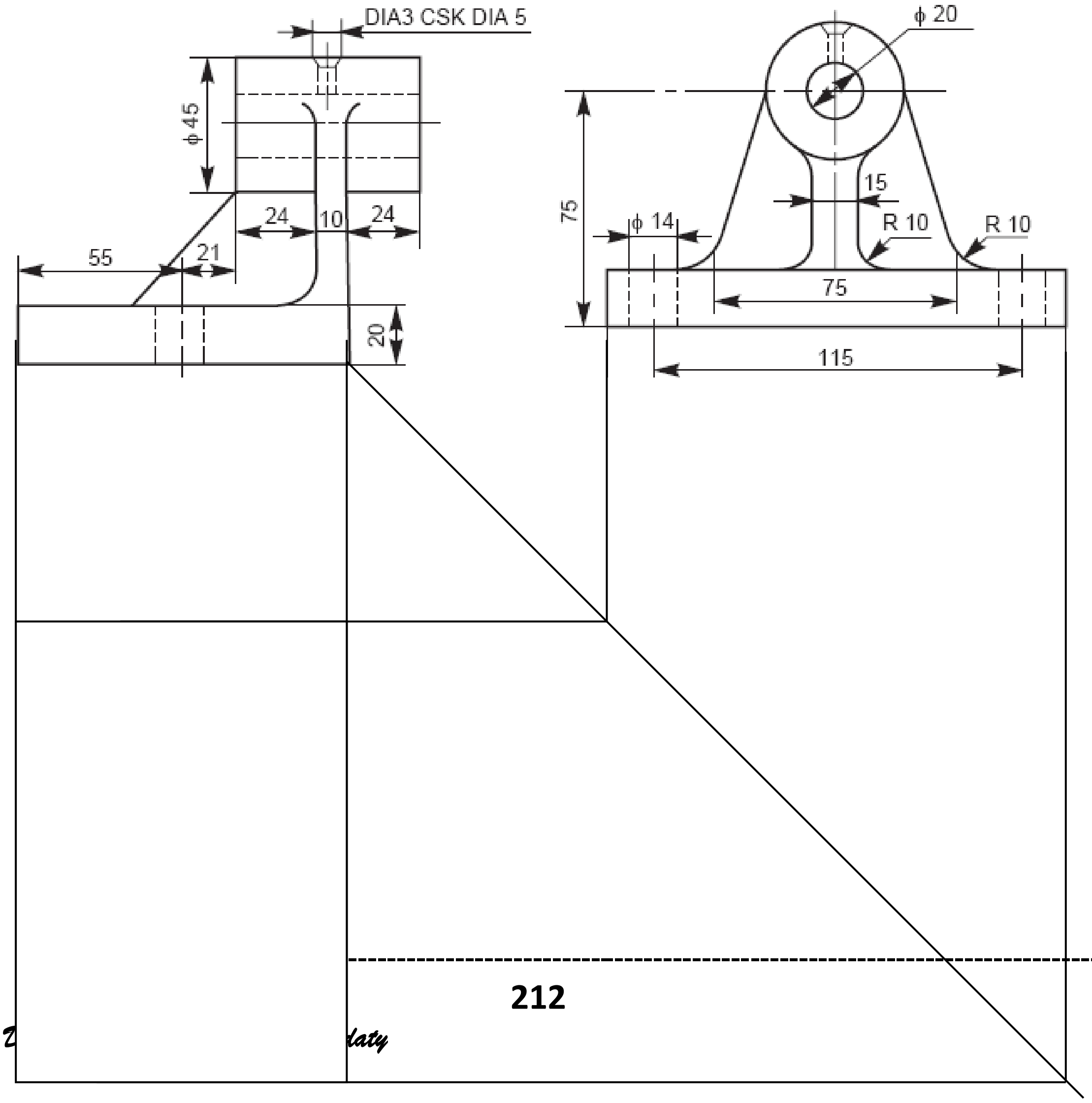


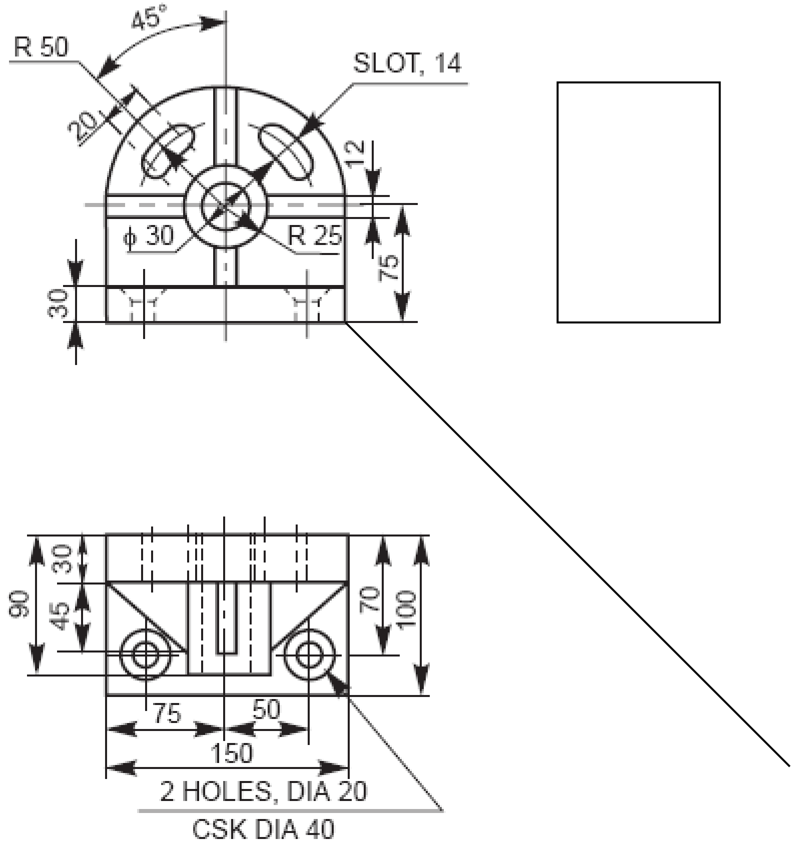

FV
ϕ30
ϕ15
40
30
25
80
FV
R15
R8
12
60
10
20

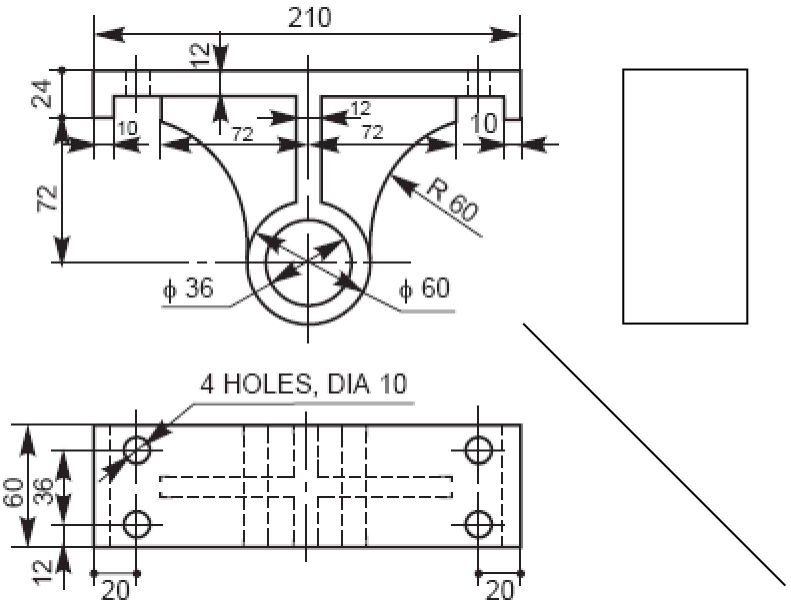

58
44
35
63
102
R45
Ø50
90

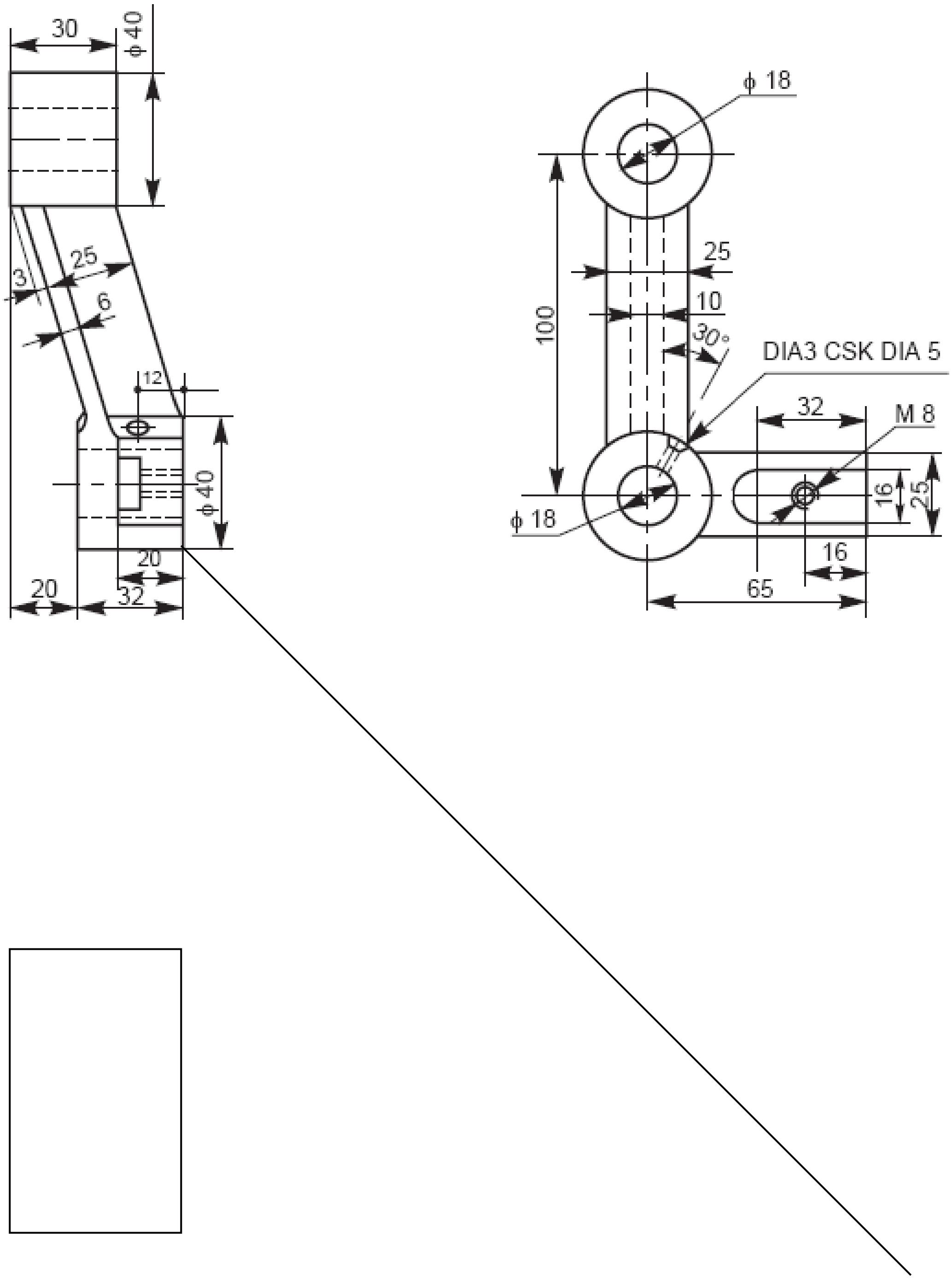

ϕ 55
ϕ 35
R10
10
20
70
25
R10


42
22
22
46
M 6
6
ϕ 48
ϕ 24
ϕ 7.5
12
20
2.5
18
85
30


DIA3 CSK DIA 5
ϕ 45
24
10
24
55
21
20
ϕ 20
75
ϕ 14
15
R 10
R 10
75
115


45°
R 50
SLOT, 14
20
12
ϕ 30
R 25
75
30
90
45
70
100
75
50
150
2 HOLES, DIA 20
CSK DIA 40


210
12
24
10
72
12
72
10
R 60
72
ϕ 36
ϕ 60
4 HOLES, DIA 10
60
36
12
20
20


30
ϕ 40
25
3
6
12
ϕ 40
20
20
32
ϕ 18
25
10
100
30°
DIA3 CSK DIA 5
32
M 8
16
25
ϕ 18
16
65

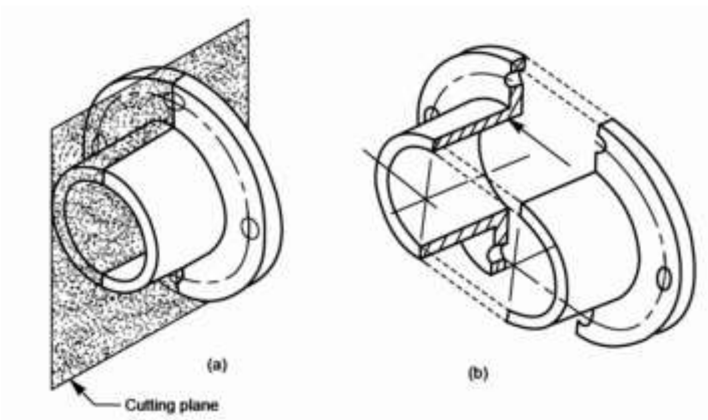
Chapter 7

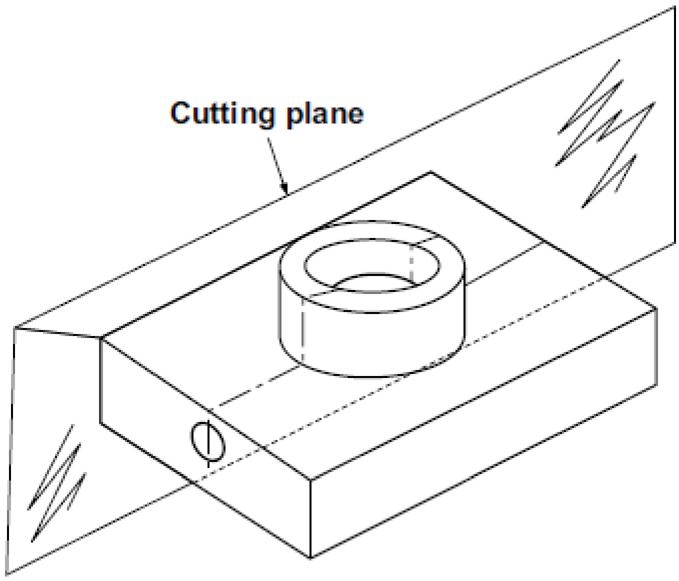
Sectional Views

SECTIONAL VIEWS

7.1 INTRODUCTION

In orthographic projection, the interior details of an object, which are not visible to observer from outside are shown by hidden lines. But some of the machine parts have the complicated interior details and they do not give the clear idea about the internal shape of an object. Too many hidden lines also confuse the observer. To avoid these confusions, the views are made in section and it is imagined that the object is being cut through or sectioned by a plane. The part of the object between the cutting plane and the observer is assumed to be removed and the remaining view of the object then obtained is called the sectional view. Therefore, section is defined as the view obtained after cutting the object in order to show the inner details by an imaginary cutting plane is called a sectional view. The imaginary plane is called a cutting plane or a section plane. The cutting plane is taken parallel to the plane on which the view is projected and the section view is drawn by removing the nearer portion of the object. The sectional view of the object is represented by the thin lines, and these lines are known as sectional lines or hatching lines. These hatching lines are drawn parallel to each other at an angle of 45º to the out lines of the object as shown in Fig. 7.1.





(i)

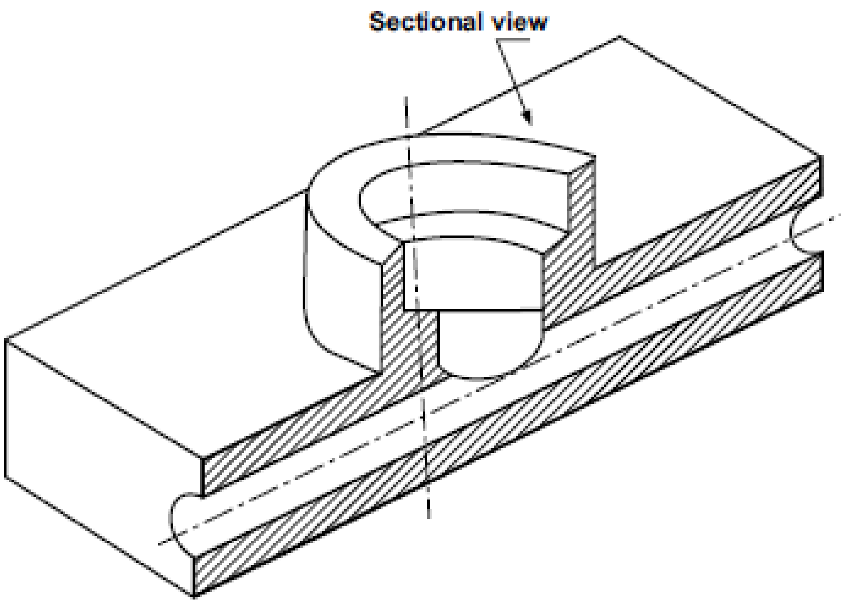
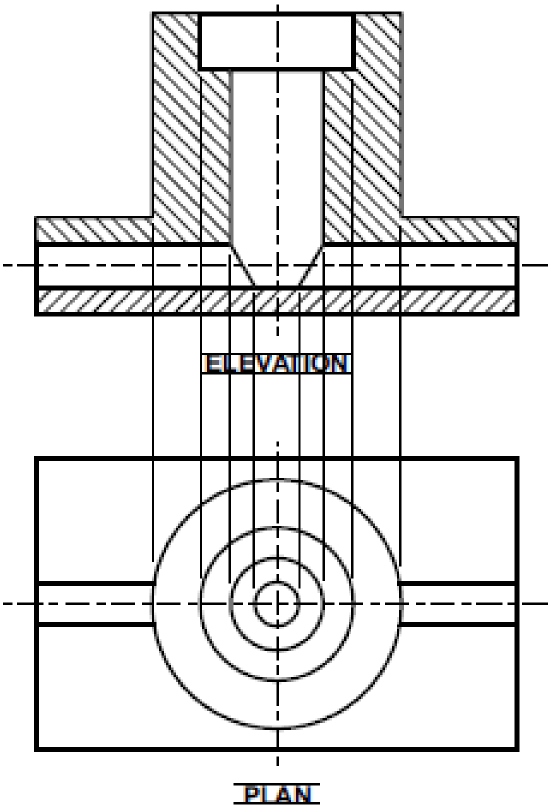


Fig.7.1



(ii)

The hidden portion of the object behind the section is generally omitted and not shown by hidden lines unless it is very essential for clarity. From the above discussion it is clear that orthographic views are used for simple objects and sectional views are used for complicated object. These sectional views play very important role in engineering drawing because they help in manufacturing and explaining the construction of complicated machine parts.

7.2 Cutting Plane Lines

Cutting plane lines are used to indicate the location of sectional planes for sectional views and viewing position for removed partial views. It is represented by long thick dashes and a short dashes alternatively spaced as shown in Figure below (7.2) The arrows are used to indicate the direction in which the cut surface is viewed and the lines show the edge of cutting plane.

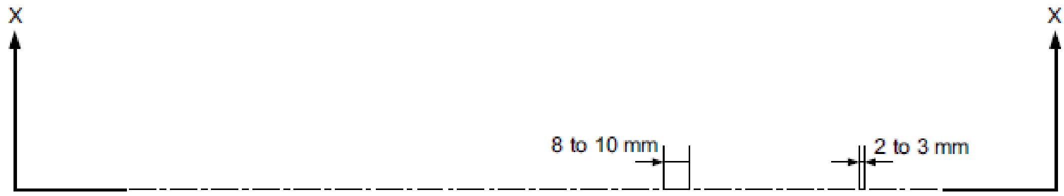


Fig.7.2

7.3 RULES OF SECTIONING

The following are some of the important rules of sectioning:

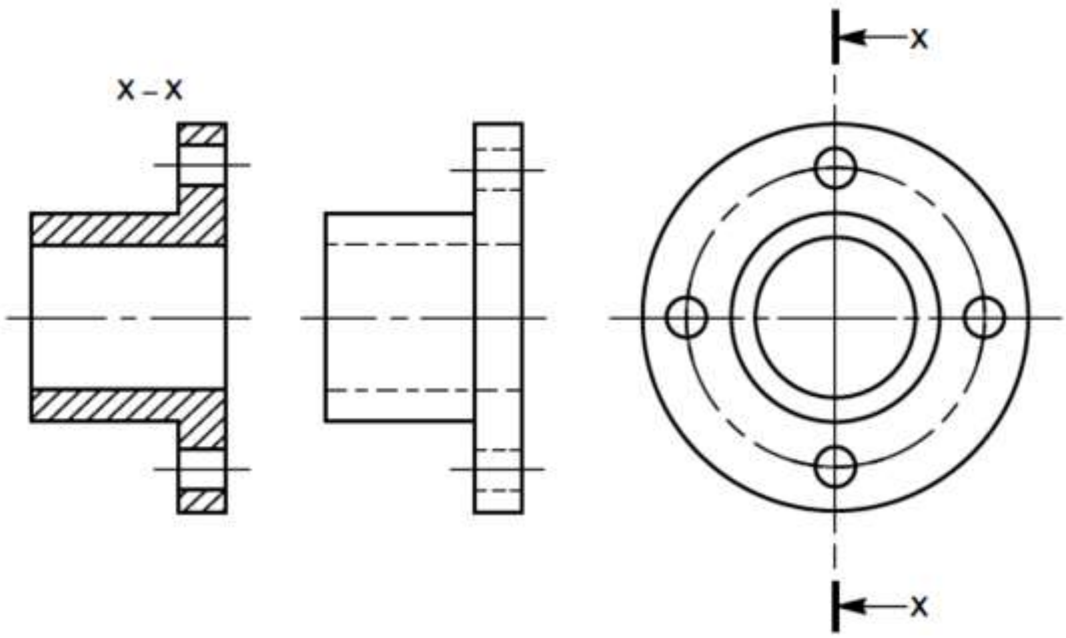
1. Nuts, bolts, screws, keys, cotter, rods, ribs, spokes and handles **are not sectioned.**
2. Section lines are drawn, at an angle of **45º** to the major outline of the object.
3. All the lines must be uniformly spaced and the distance between two section lines normally varies from **2 mm to 3 mm**.
4. The parts which are actually cut by cutting plane are hatched.
5. Hatching of different components is done on opposite direction.
6. Hidden lines are not used in sectioned area unless they are needed for clarity.
7. The arrows at each end of the cutting plane lines indicate the direction of viewing.
8. The position of the cutting plane are shown on final drawing.
9. Section lines should be drawn by **H or 2H** pencil.

7.4 Types of Sectional Views

There are different types of sectional views which are commonly used in engineering drawing. The following are the important types of sectional views:

7.4.1 Full Sectional View

Imagine when the sectional view obtained after removing the front half part of the object through its centre line by an imaginary cutting plane, the remaining object is said to bein full section. The projected view of the full sectioned surface along with the remaining half part to show clearly interior shape of the object is known as full sectional view as shown in Fig. 7.3. This type of section is used for both detail and assembly drawings. In the full sections, the observer views the object in the direction of arrow and the resulting full sectional front view is obtained. It is not necessary to draw all the views of an object in sectional views.



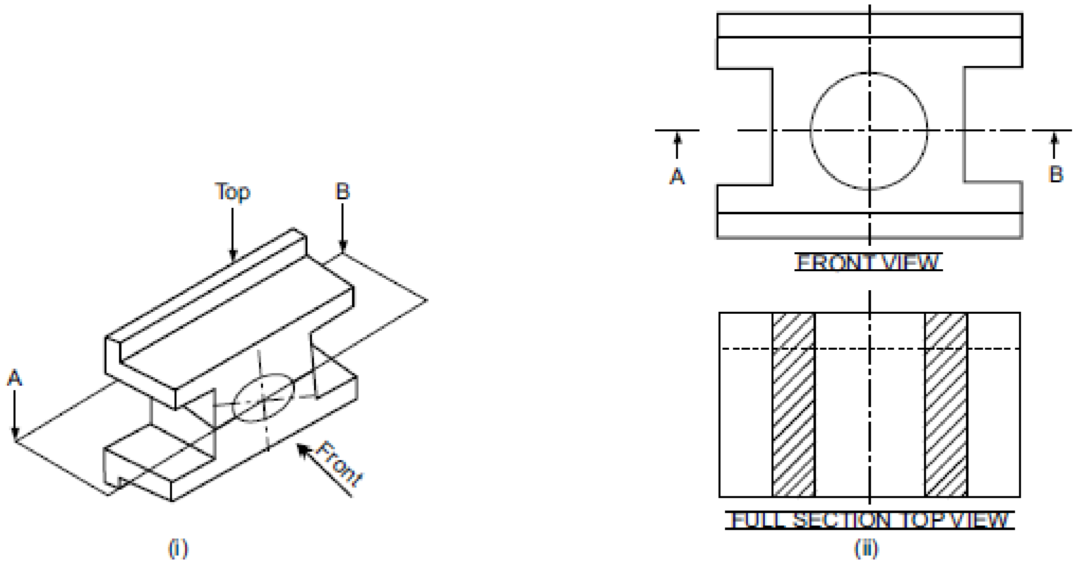


Fig.7.3

- **Correct and incorrect in full sectional shows in Fig.7.4.**

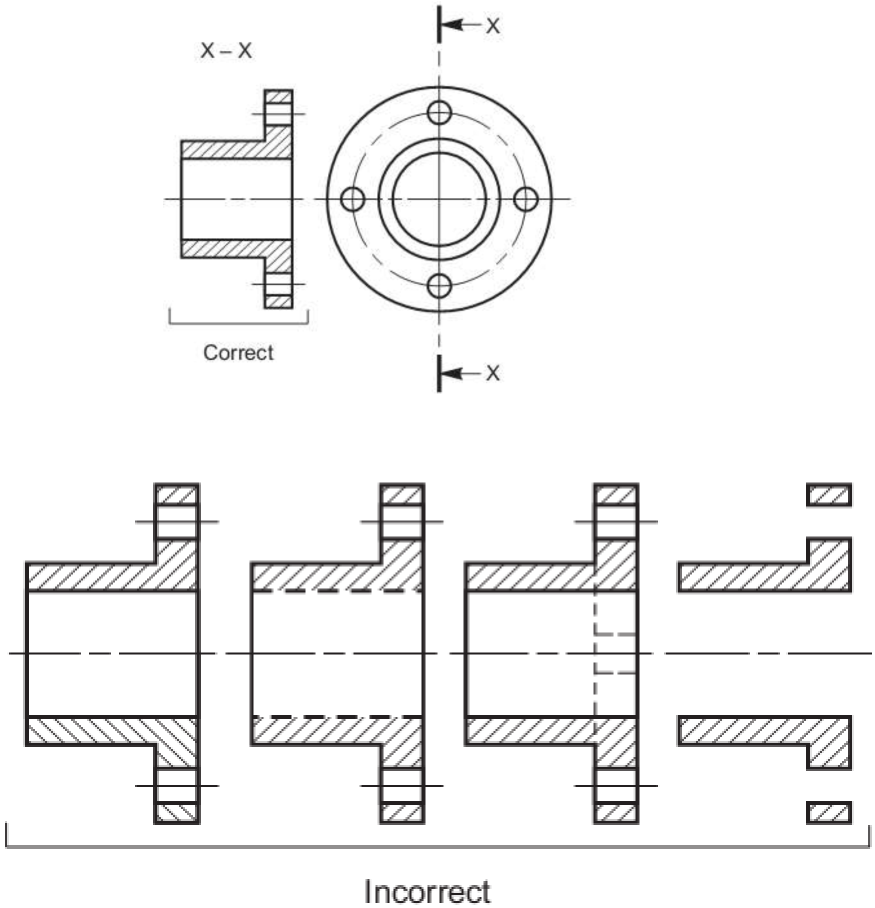


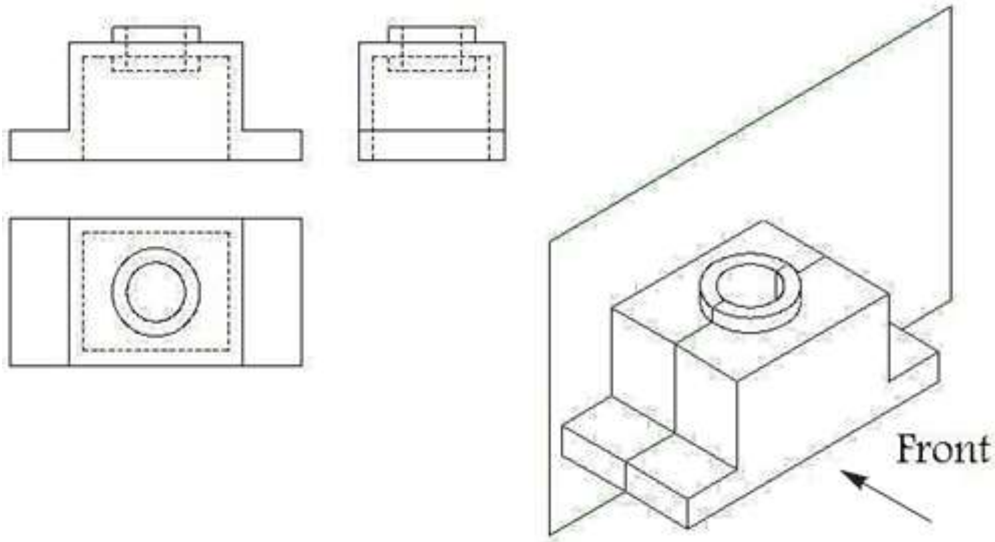
Fig.7.4

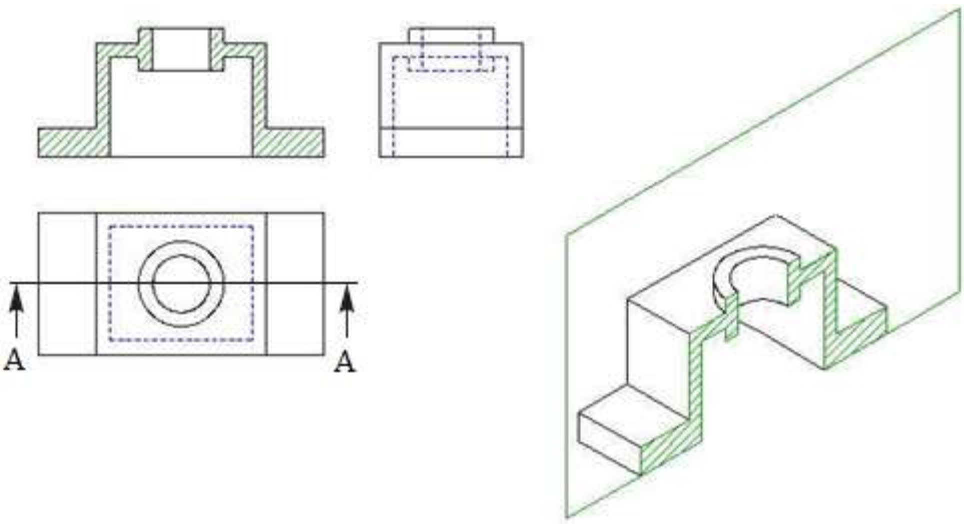
- **Example**

1- Section Elevation at AA

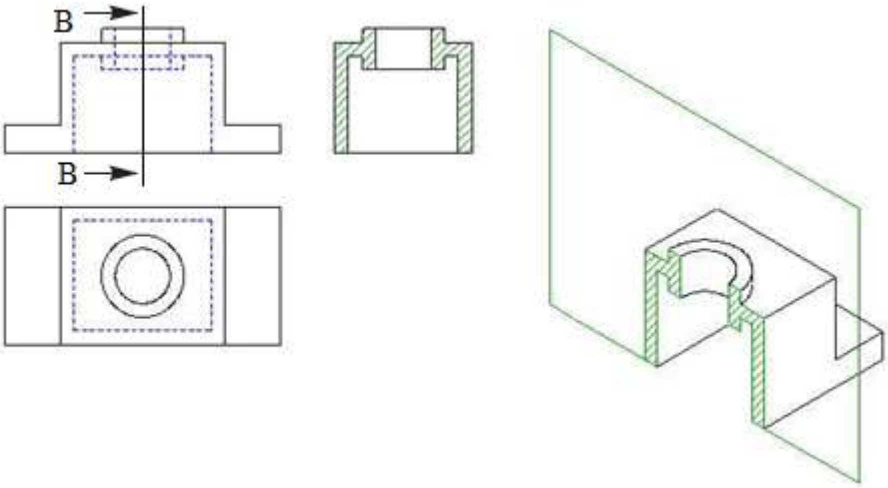
2- Section Side View at BB

3- Section Plan at CC

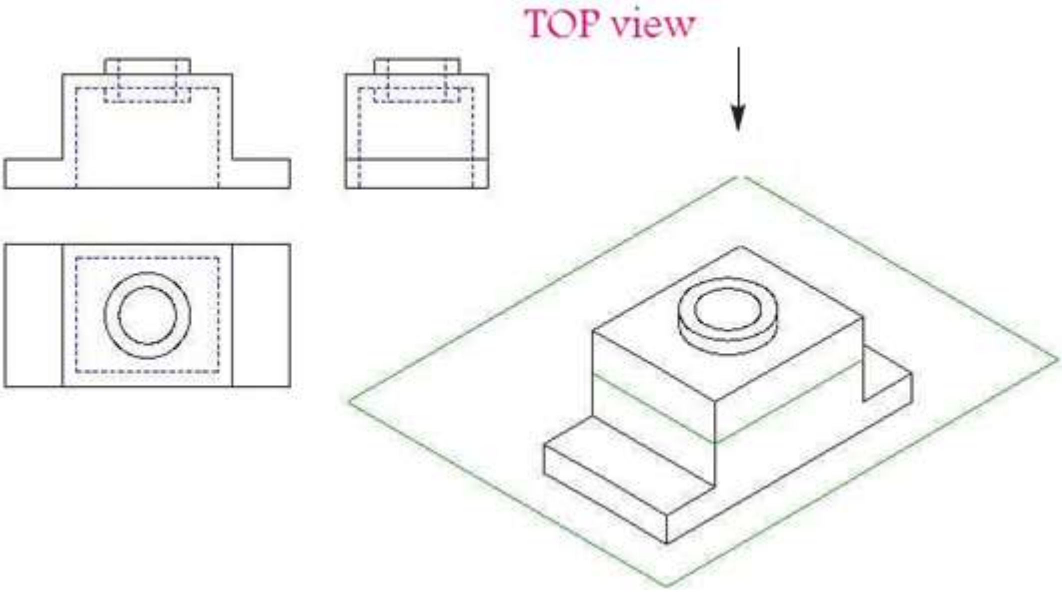


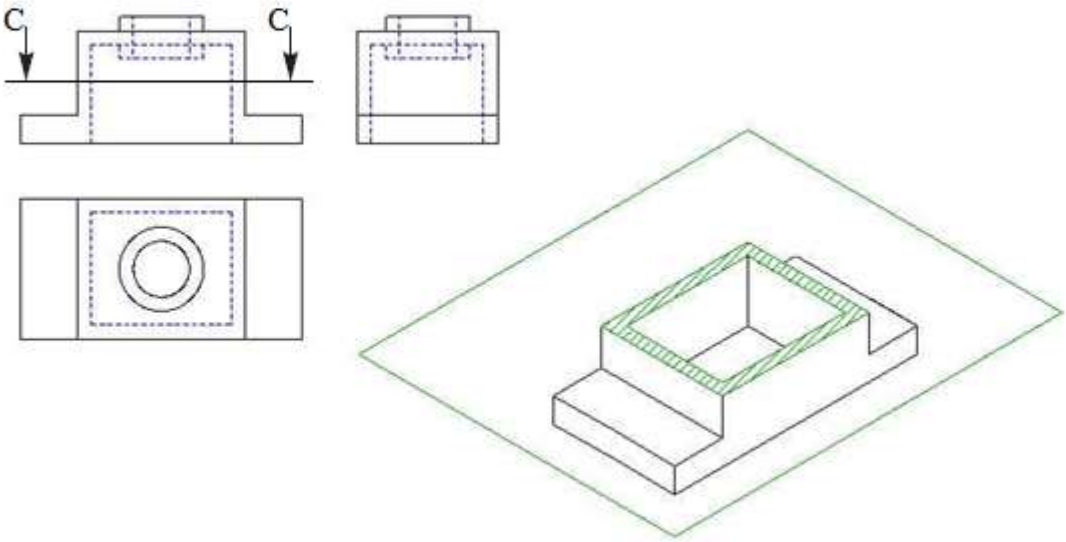


Section Elevation at AA

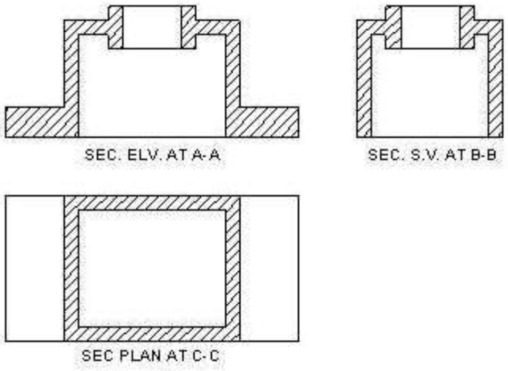


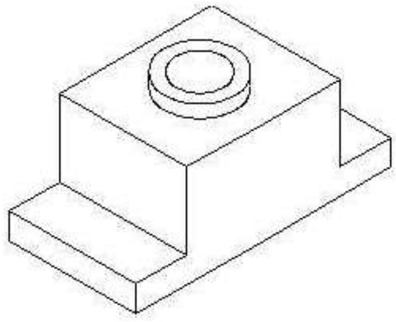
Section Side View at BB





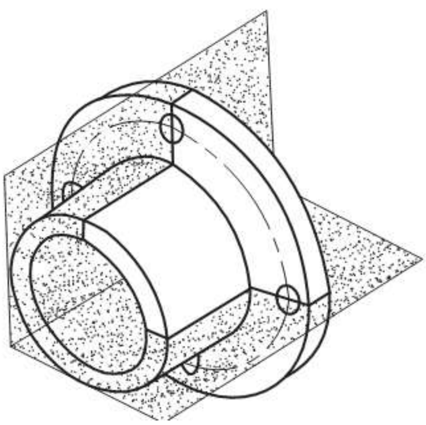
Section Plan View at CC

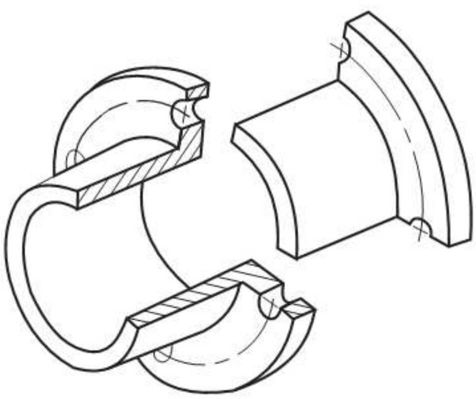


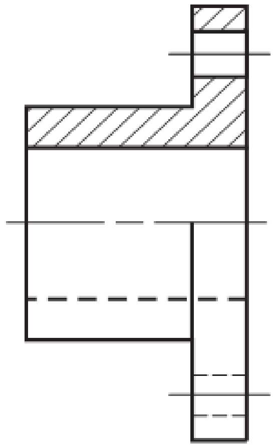


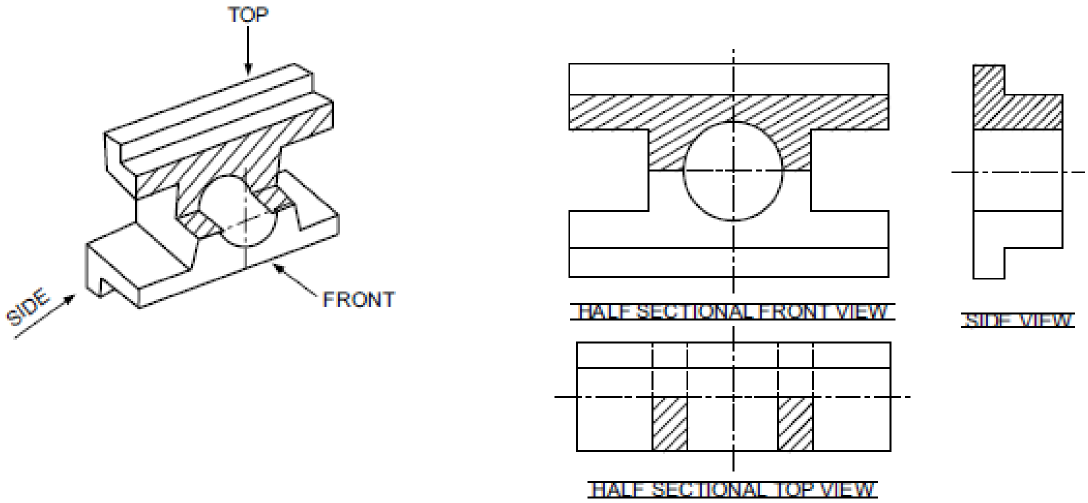
7.4.2 Half Sectional View

Imagine when the sectional view obtained after removing the front quarter part of the object by two imaginary cutting planes at right angles to each other the remaining object is said to be in half section. The front quarter part is cut and removed and then the projected view of sectioned surface along with the remaining half outside part is known as half sectional view as shown in Fig. 7.5. These type of sectional views are best suited for assembly drawings where both internal and external construction are to be shown in one view and where only overall and centre to centre dimensions are depicted.








TOP
SIDE
FRONT
HALF SECTIONAL FRONT VIEW
SIDE VIEW
HALF SECTIONAL TOP VIEW

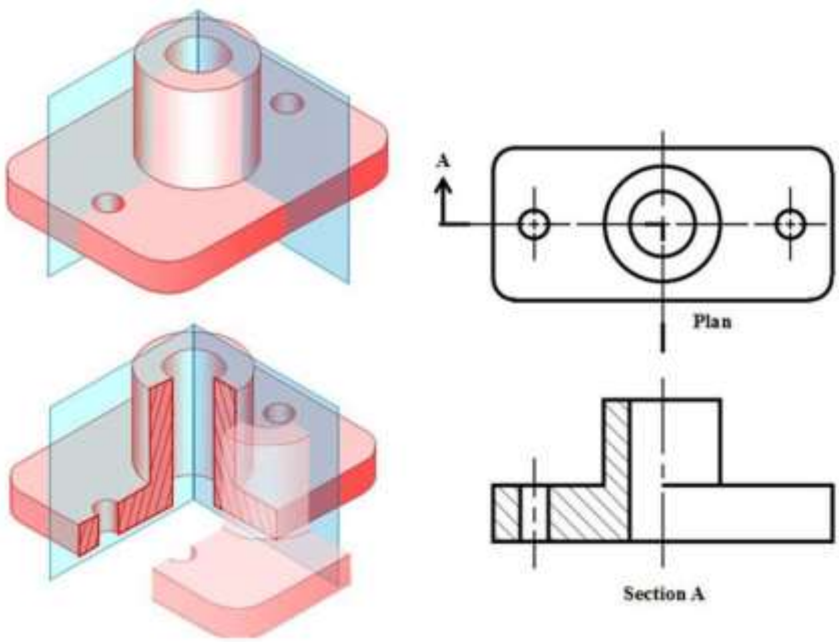
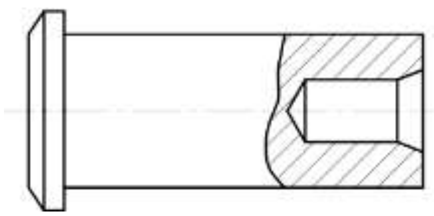
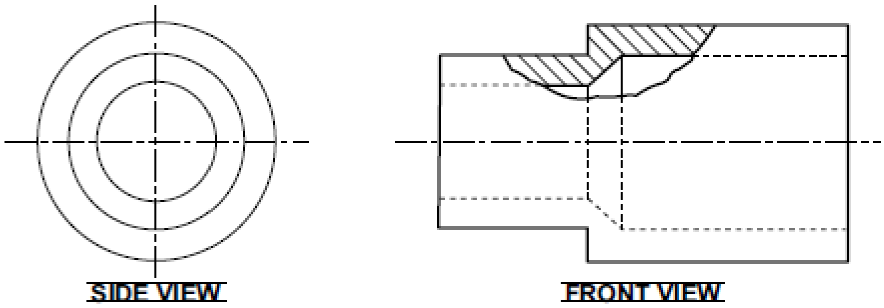

A
Plan
Section A

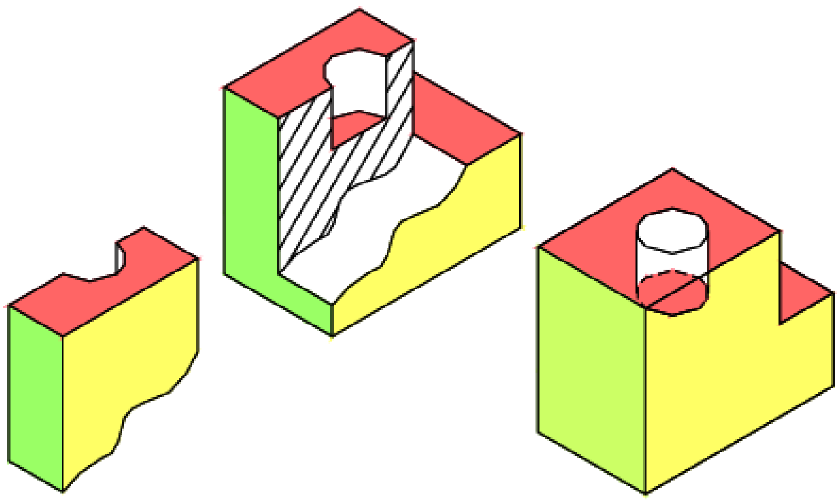
Fig.7.5

7.4.3 Partial or Broken out Section

A partial section is used where a particular hidden detail of the object is required to show, only the partial section of that object. Such a partial section is drawn by free hand short break line as shown in Fig. 7.6.







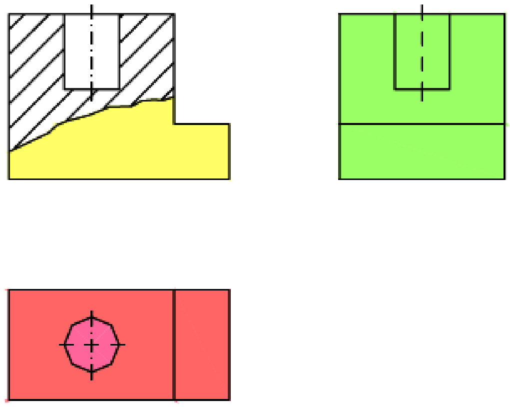
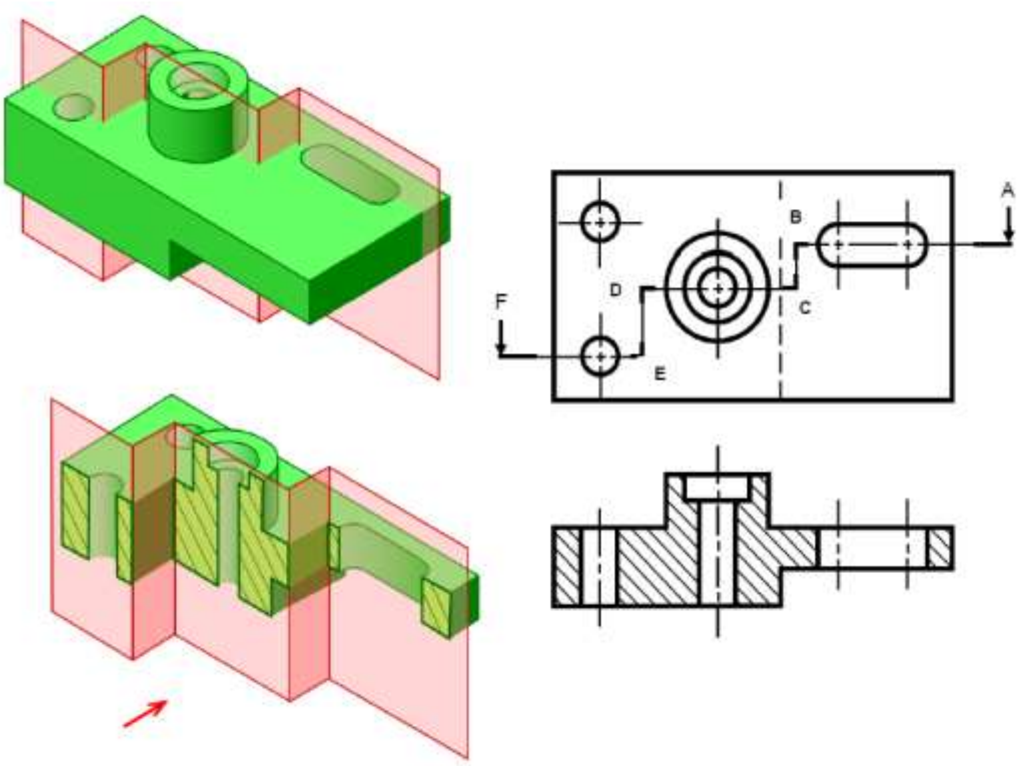


Fig.7.6

7.4.4 Offset Section

An offset section is used when the irregular object is cut by an offset plane (two or more planes) to show the maximum details, as shown in Fig. 7.7.



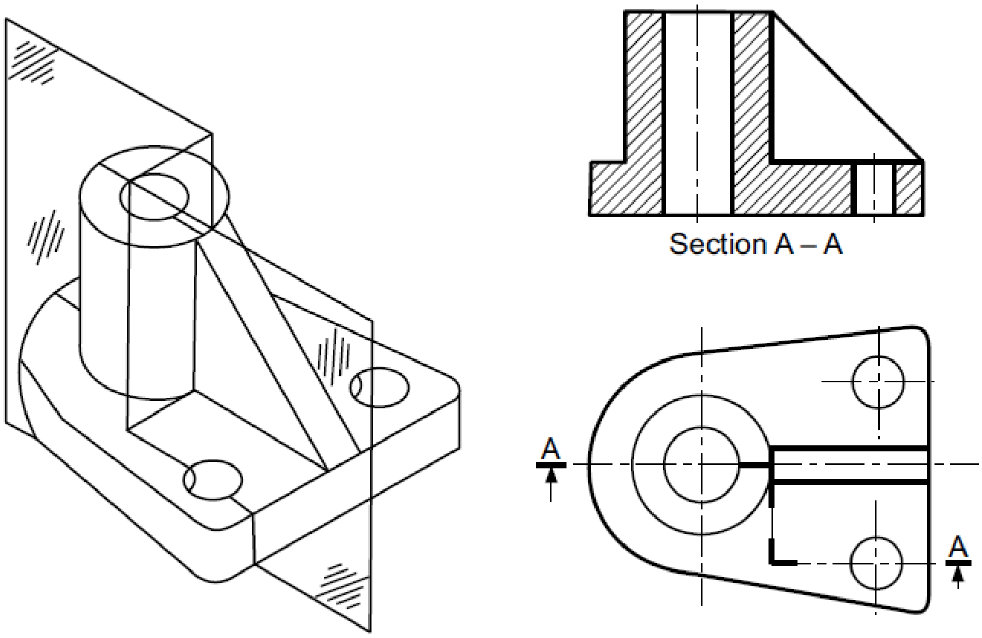


Fig.7.7

7.4.5 Revolved Section

It is obtained by passing a cutting plane through some part of structure machine at right angles to the axis of the object. It is used to show the cross-sectional shape of the object such as arms, spokes, structural section etc. as shown in Fig.7.8.

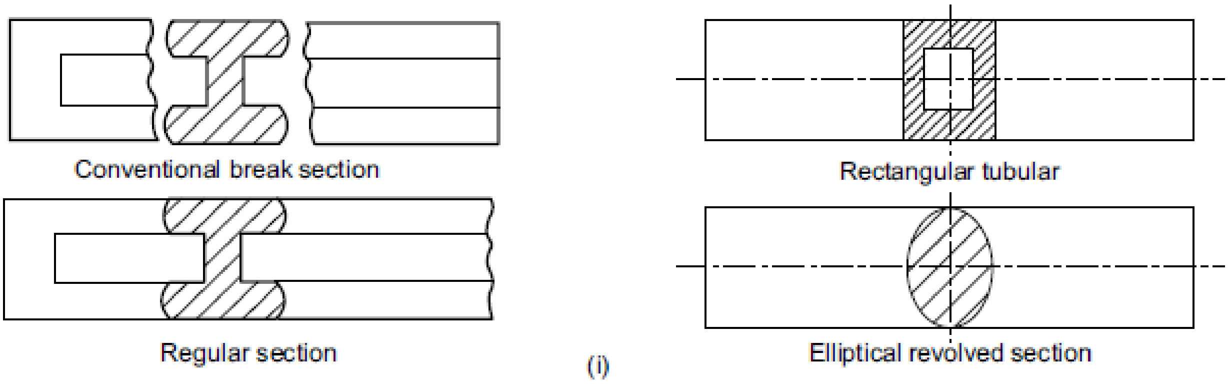


Fig.7.8

7.4.6 Removed Section

Removed section is obtained by passing a cutting plane through some part of structure machine **at right angles** to the axis of the object. The removed section is drawn separately outside the view, generally around the extension of the cutting plane as shown in Fig.7.9. It is obtained in the same manner as the revolved section. Frequently the removed section is drawn to an enlarged scale, for clarification and easier dimensioning.

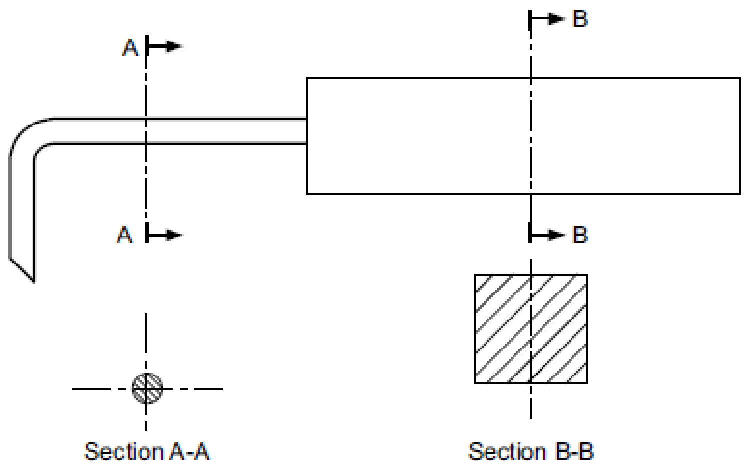


Fig.7.9

7.4.7Rolled Section

A rolled section is obtained by cutting the cross-section of **a bar, channel and angles etc**. To obtain section, an imaginary cutting plane is made to pass at right angles to the axis of the object as shown in Fig.7.10.

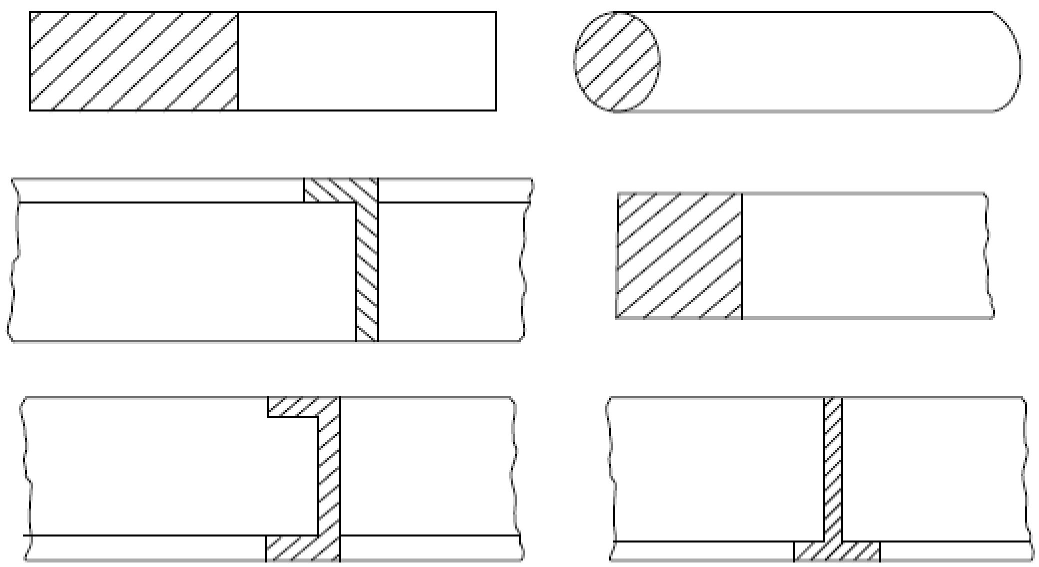


Fig.7.10

Problem 1. Fig. 7.11 shows cut view of an object. Draw the following.

(i) Full sectional front view

(ii) Half sectional side view

(iii) Top view. (Used I st angle projection)

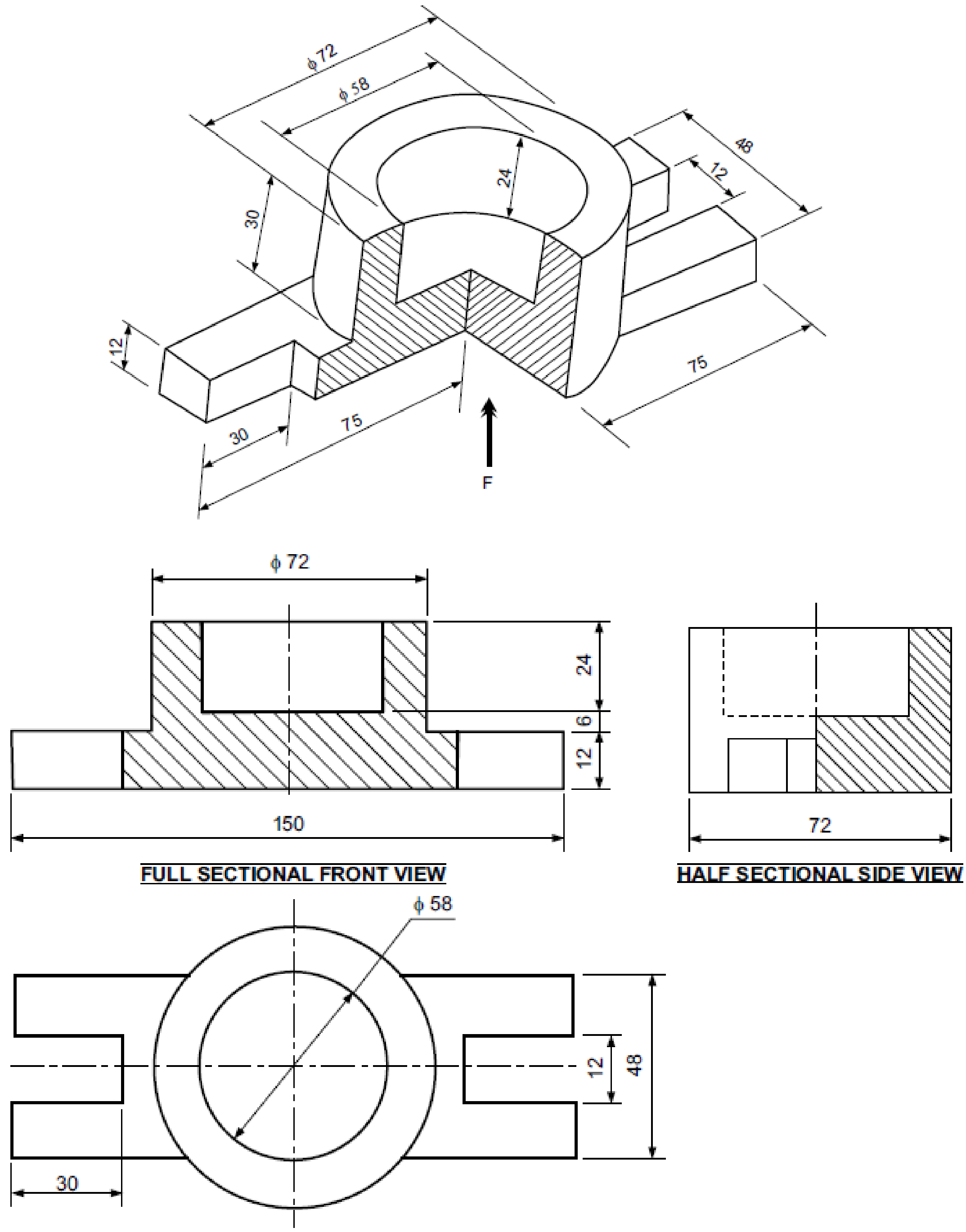


Fig.7.11

Problem 2. Fig. 7.12 shows cut view of an object. Draw the following.

(a) Full sectional front view.

(b) Side view.

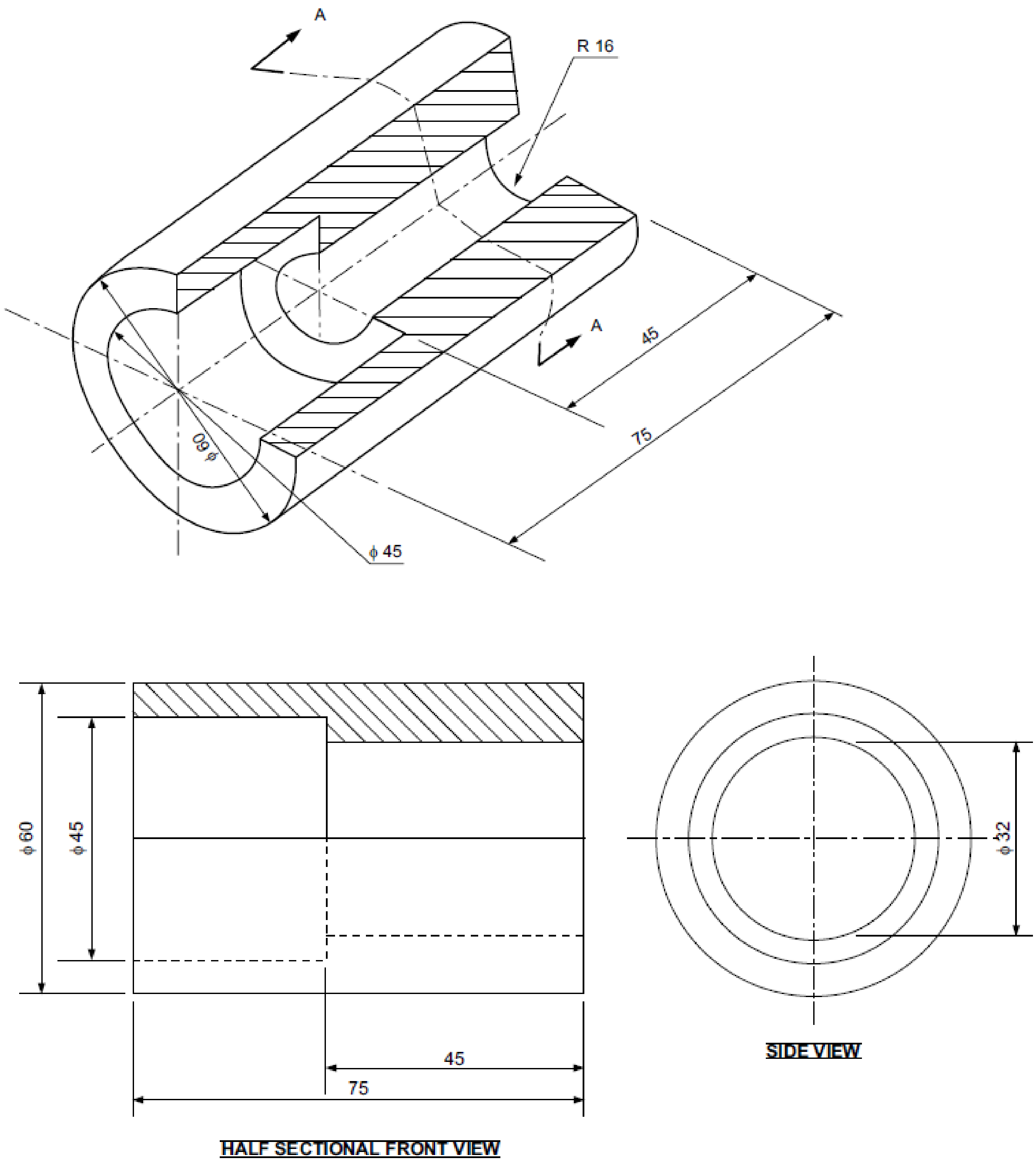
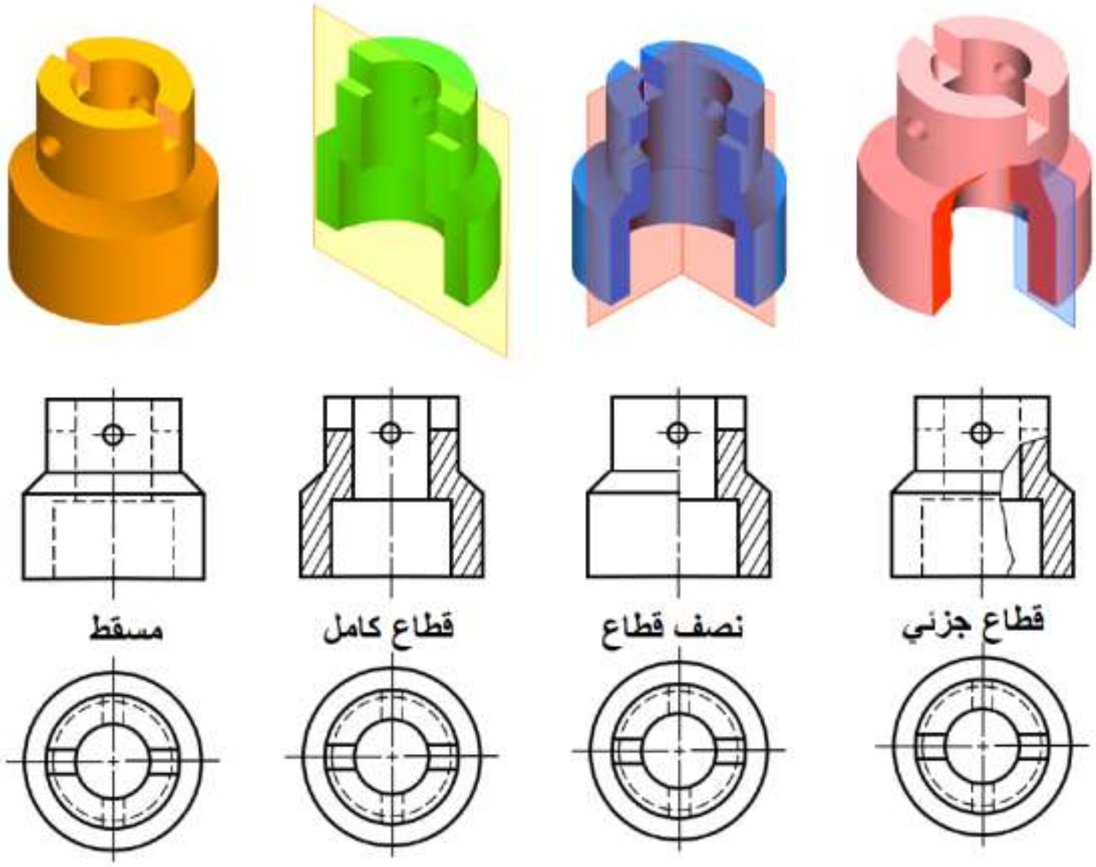
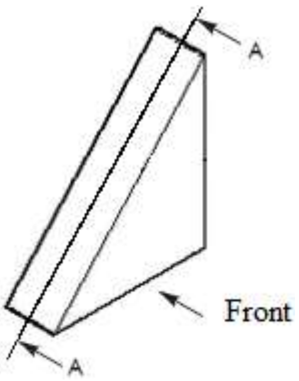
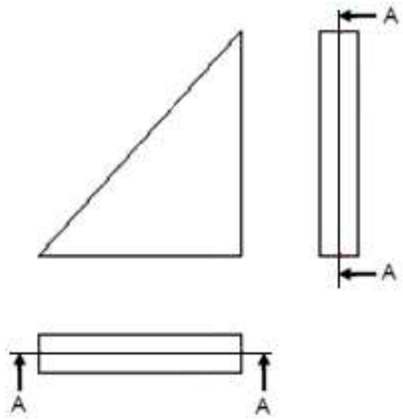


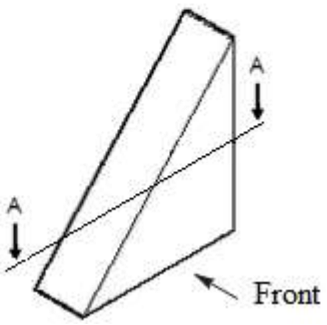
Fig.7.12

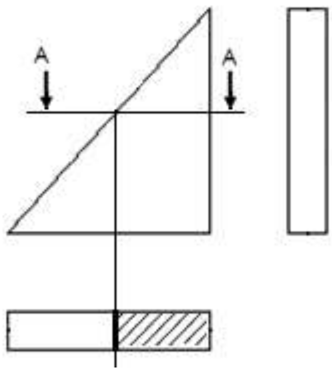

مسقط
قطاع كامل
نصف قطاع
قطاع جزئي

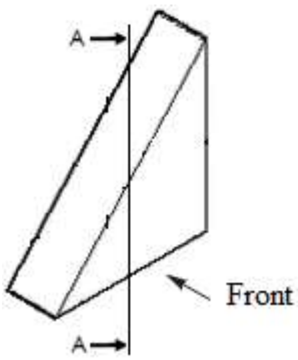
- Webs in sectional

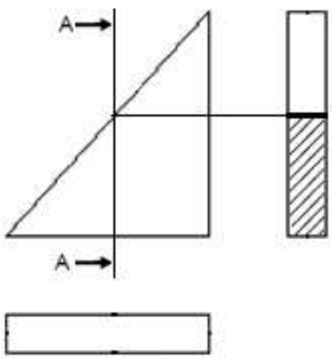












Problem 3. Fig. 7.14 shows an isometric object. Draw

1. Full sectional front view
2. Top view and
3. Side view.

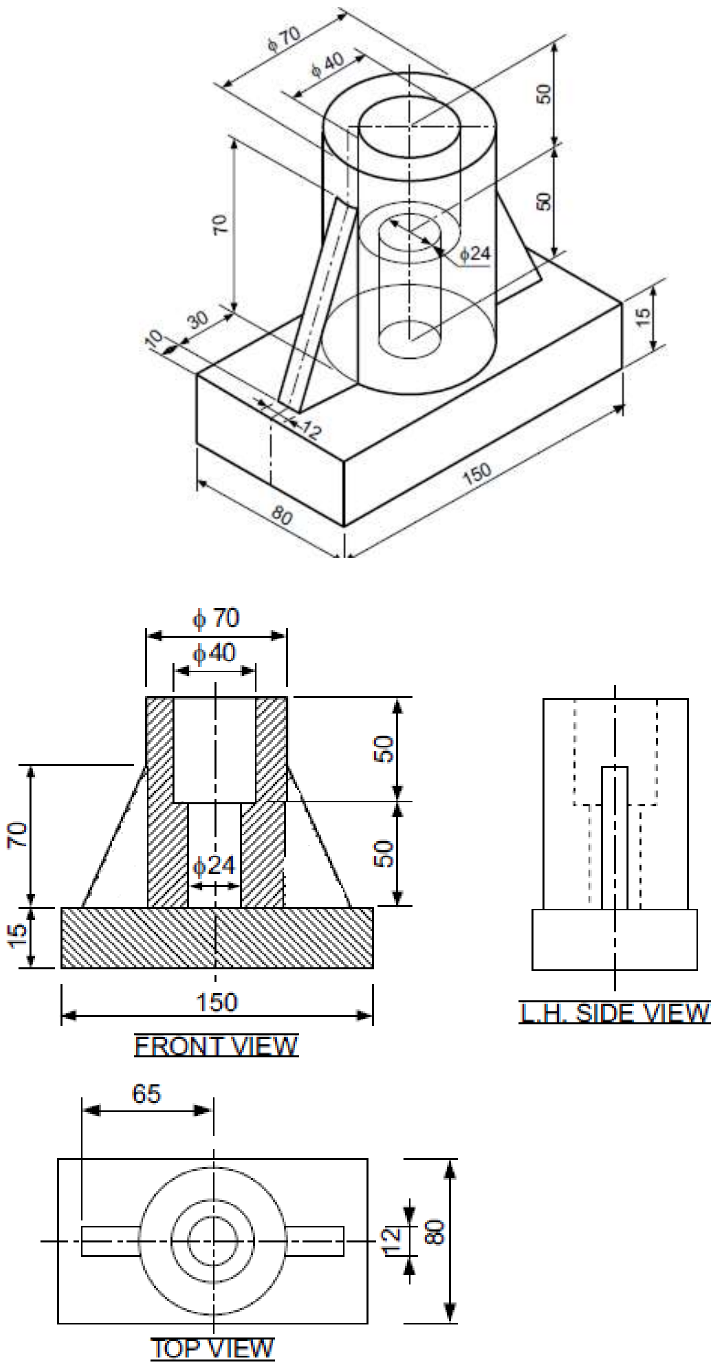


Fig. 7.14

Problem 4. Fig. 7.15 shows an isometric object. Draw

1. Full sectional front view
2. Top view and
3. Side view.

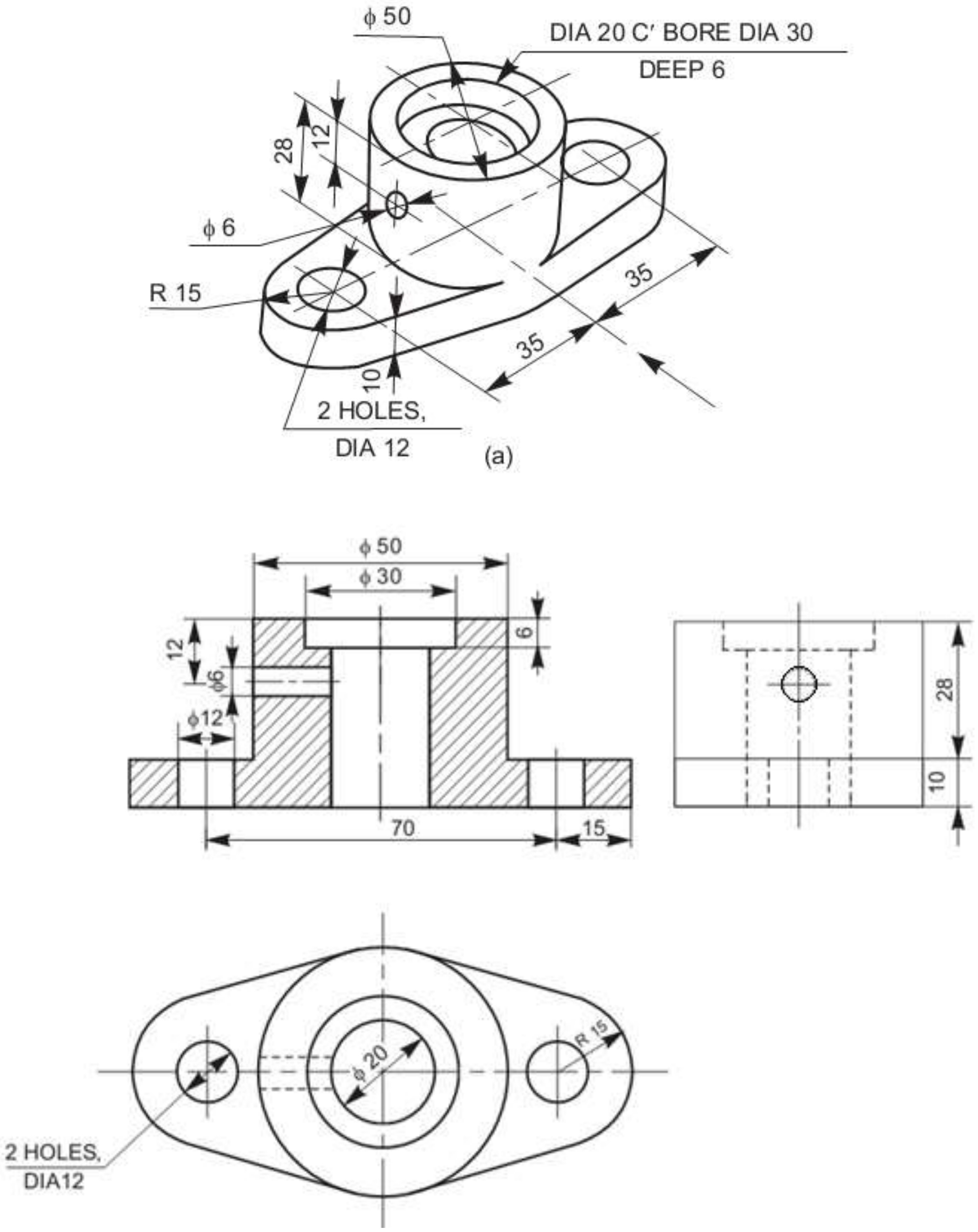


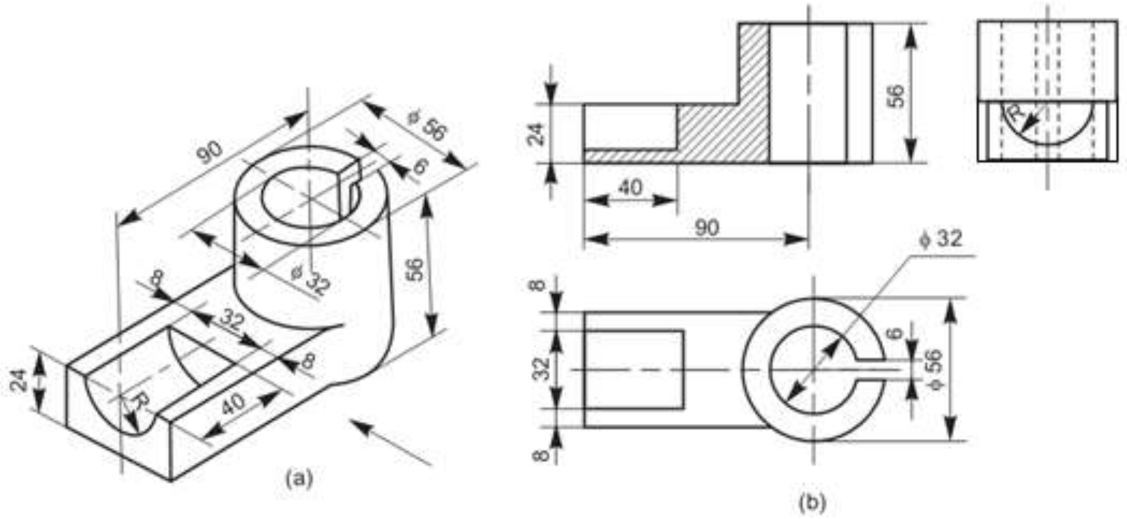
Fig. 7.15

Problem 5. Fig. 7.16 shows an isometric object. Draw

1. Full sectional front view

2. Top view and

3. Side view.



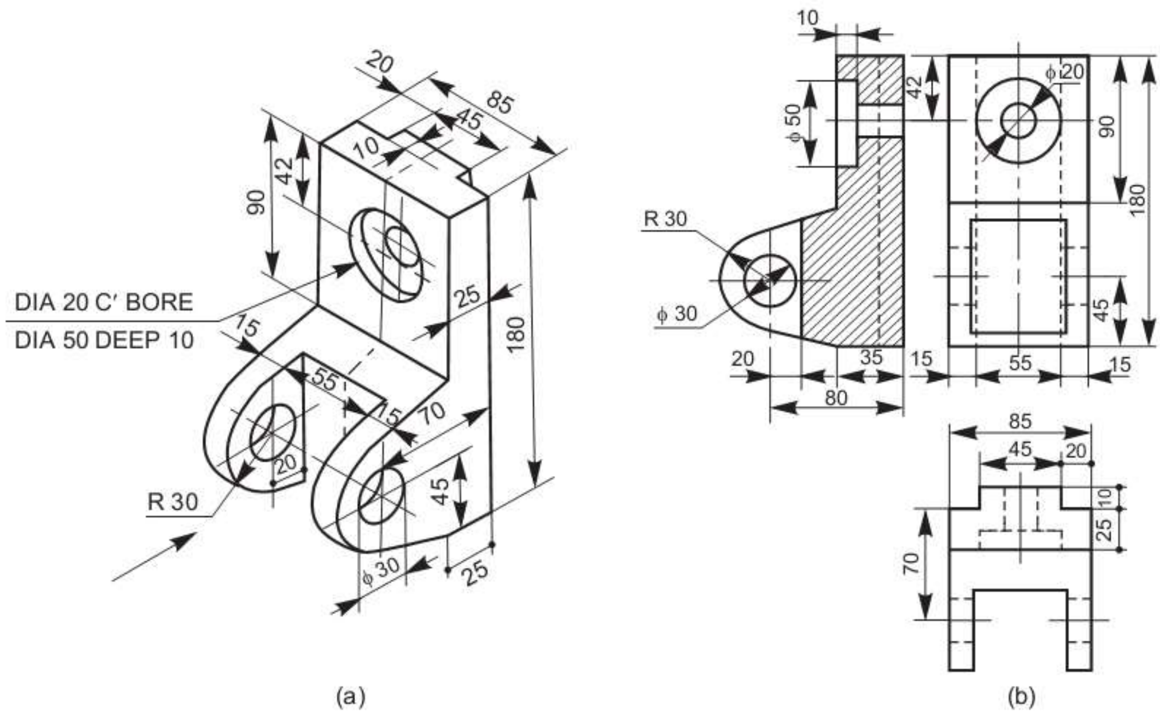


Fig. 7.16

Problem 6. Fig. 7.17 shows an isometric object. Draw

1. Full sectional front view
2. Top view and
3. Full sectional Side view.

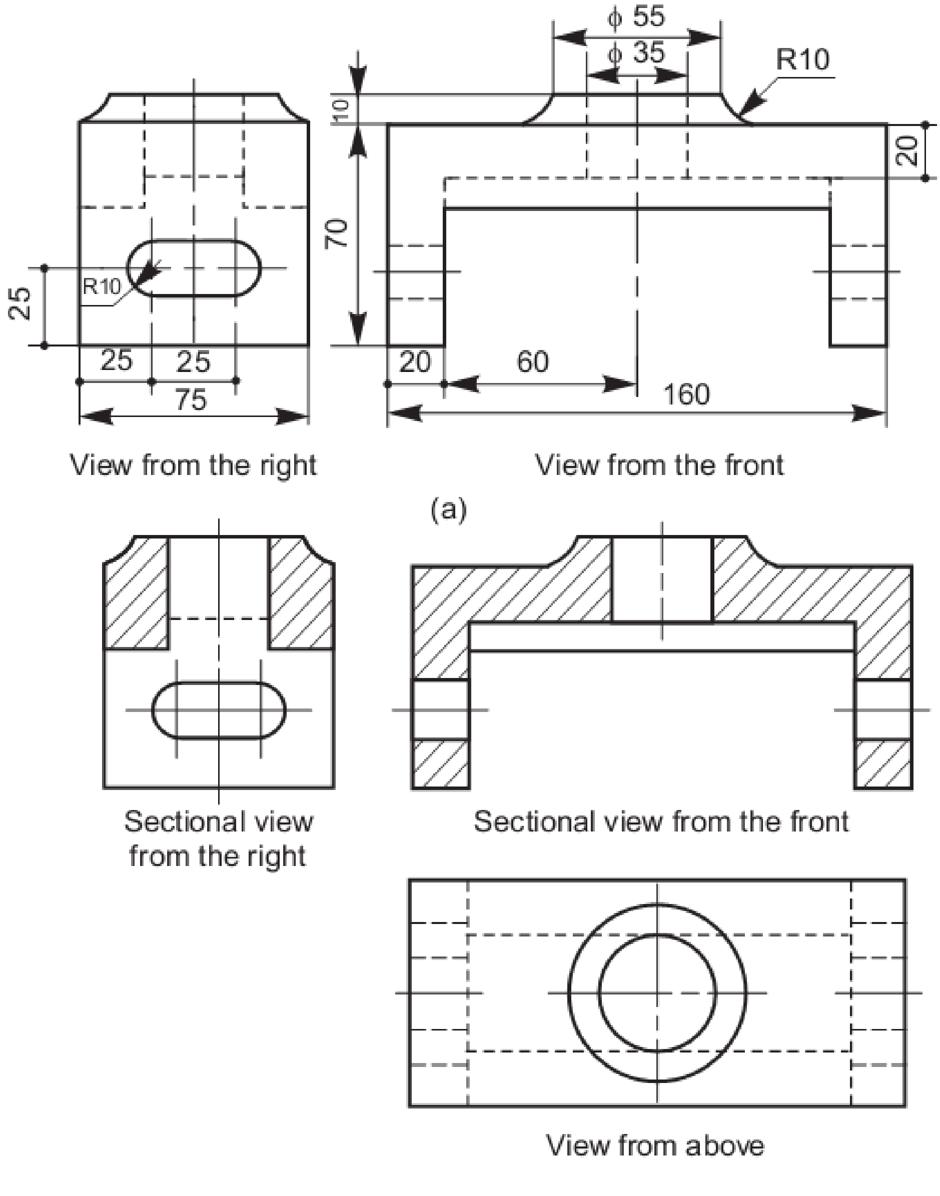


Fig. 7.17

Problem 6. Fig. 7.18 shows an isometric object. Draw

1. Full sectional front view
2. Top view and
3. Side view.

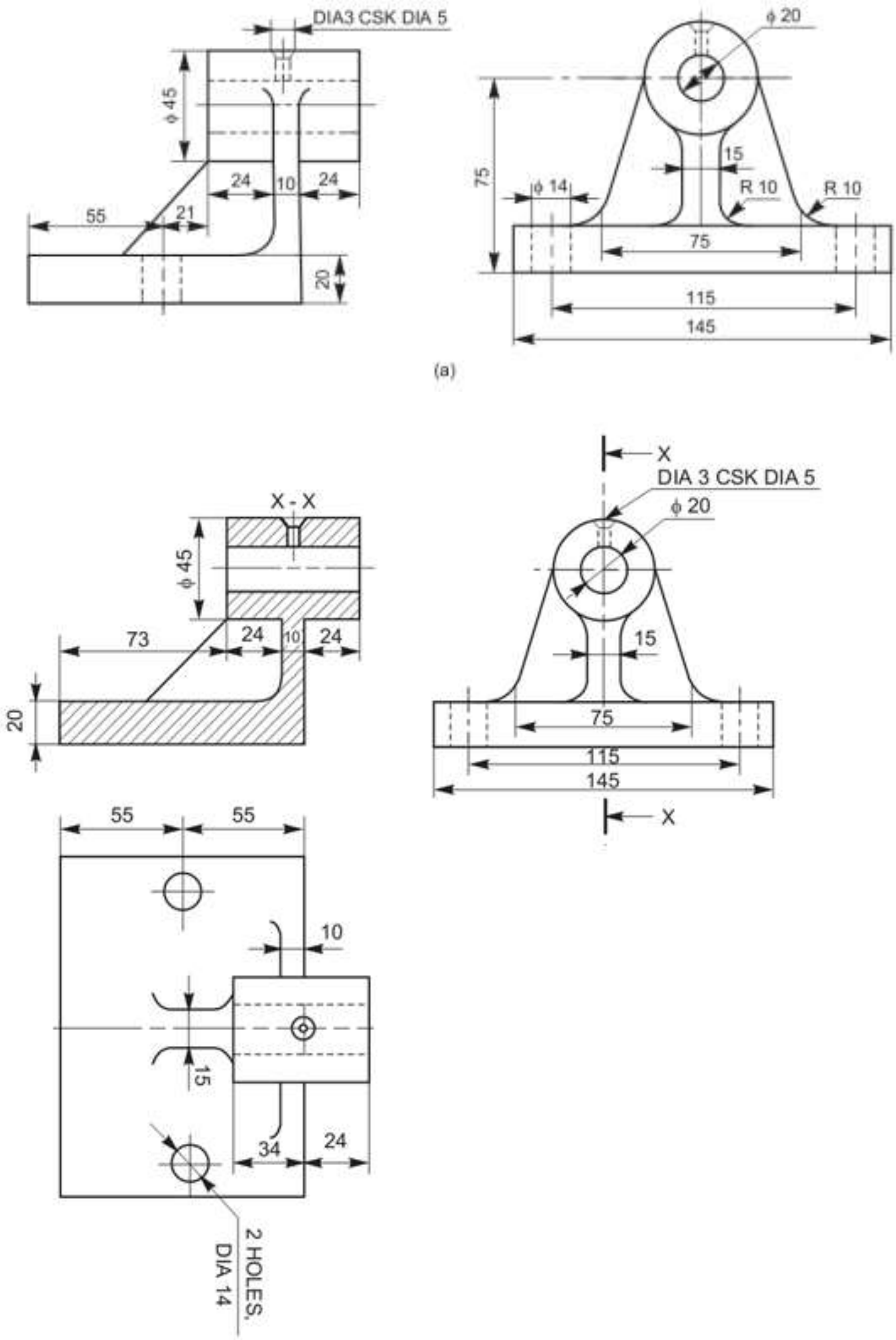
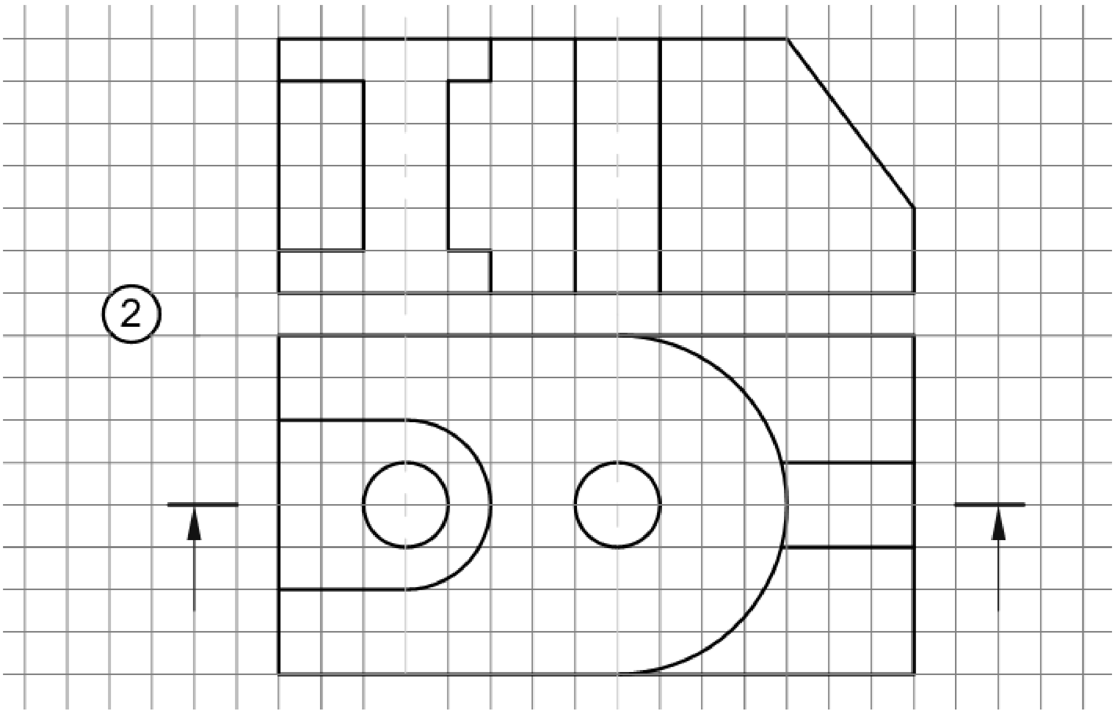
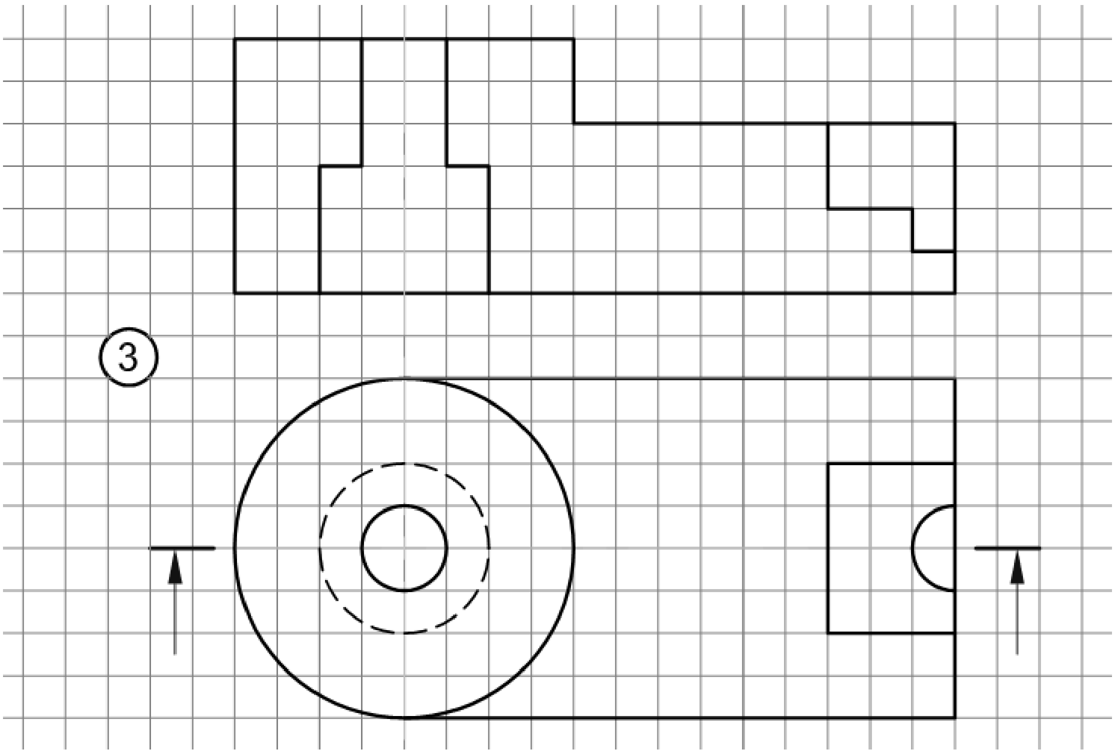


Fig. 7.18

Exercises

1. **Draw the missing lines and sectonal view.**





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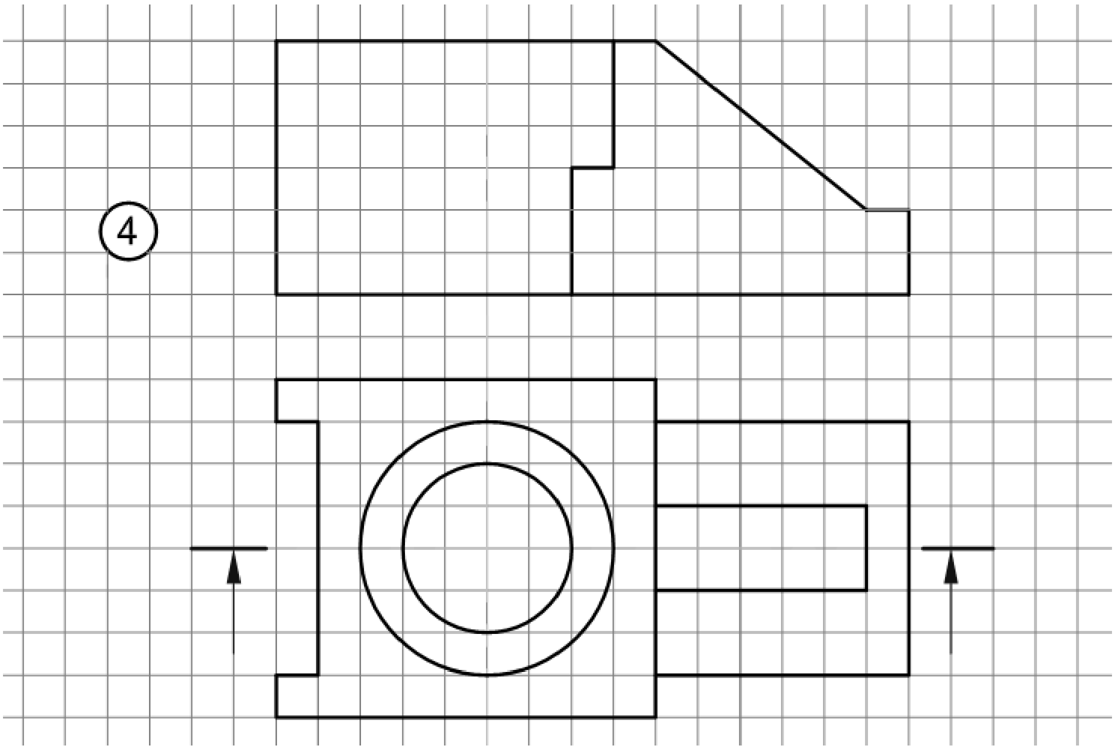

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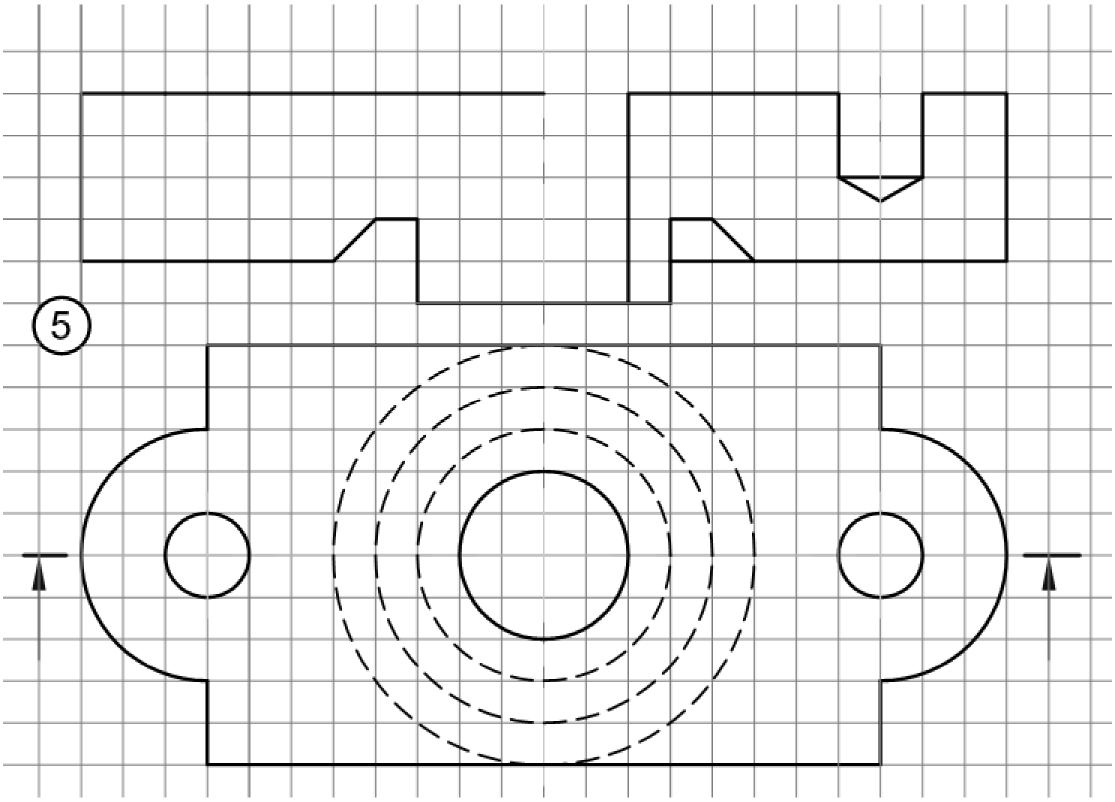
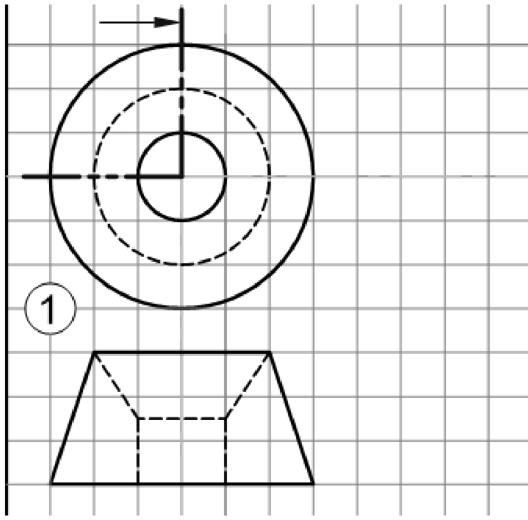
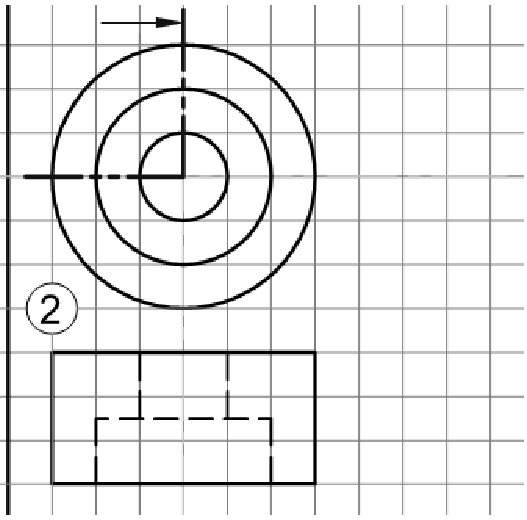
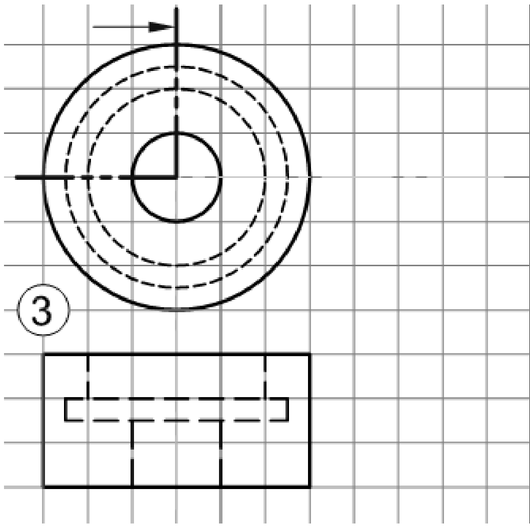
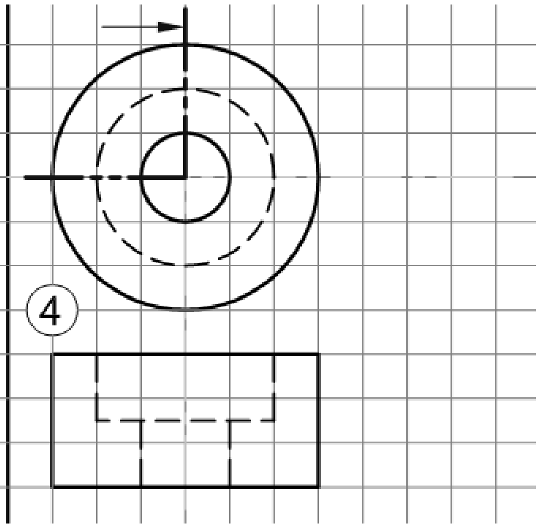


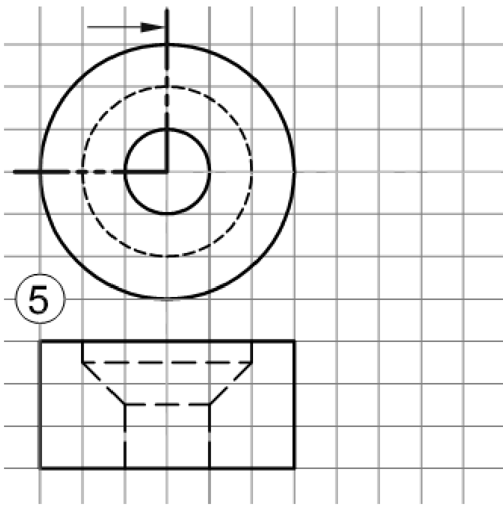
2. Draw Half Sectional Side View

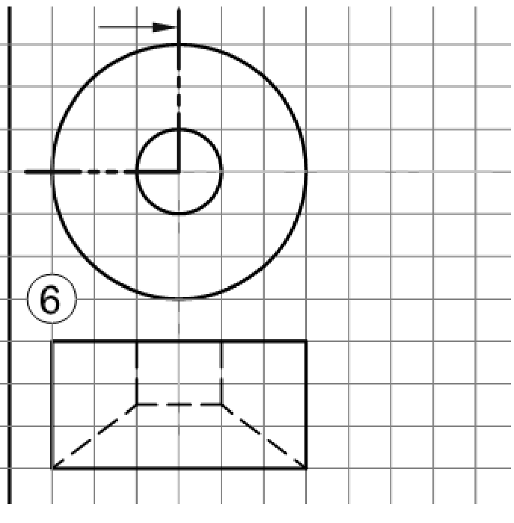




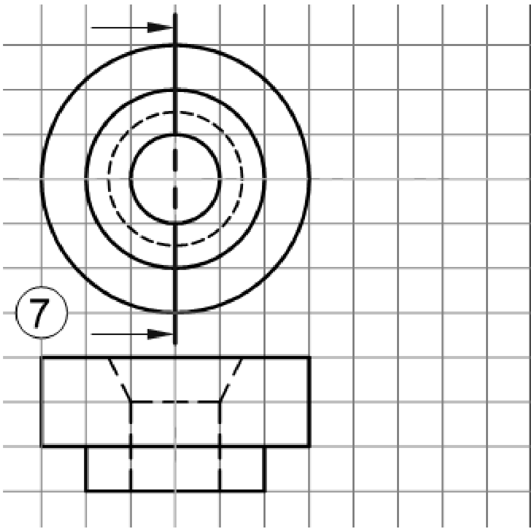


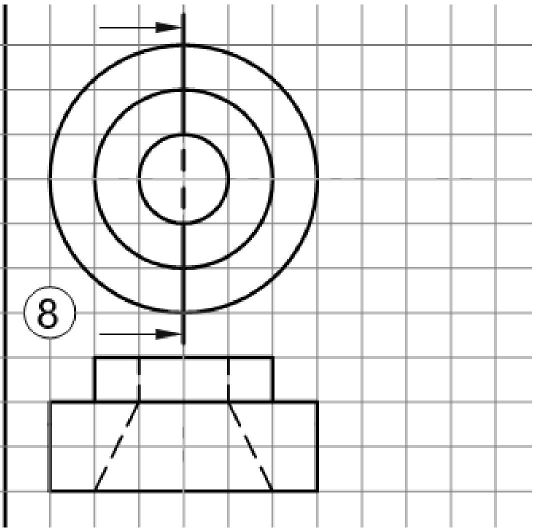


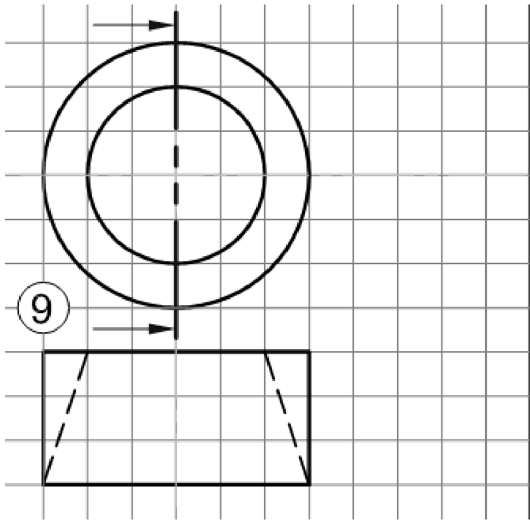


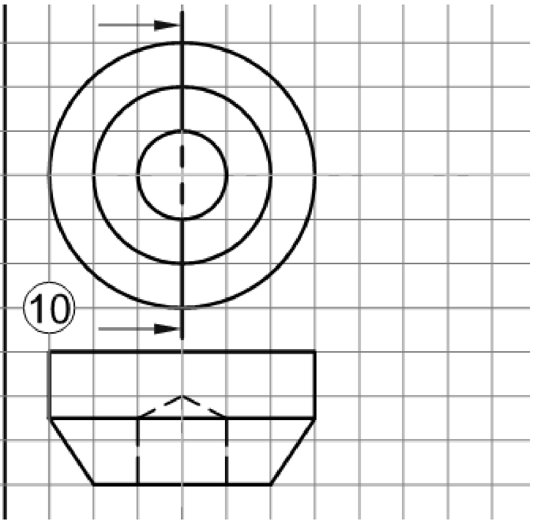


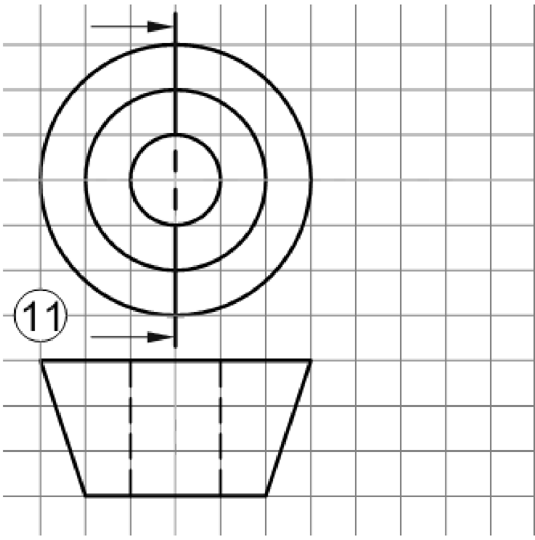
3. Draw Full Section Side View

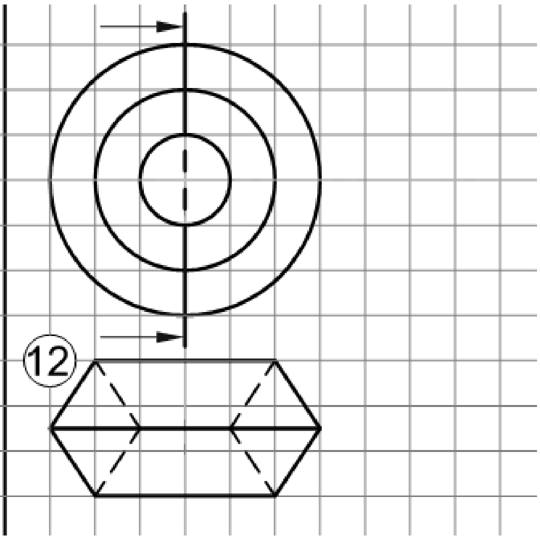




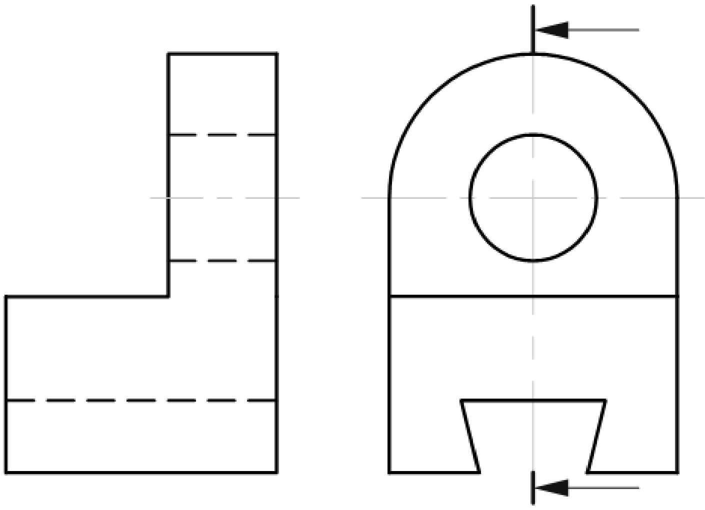








4. Draw Full Section Side View



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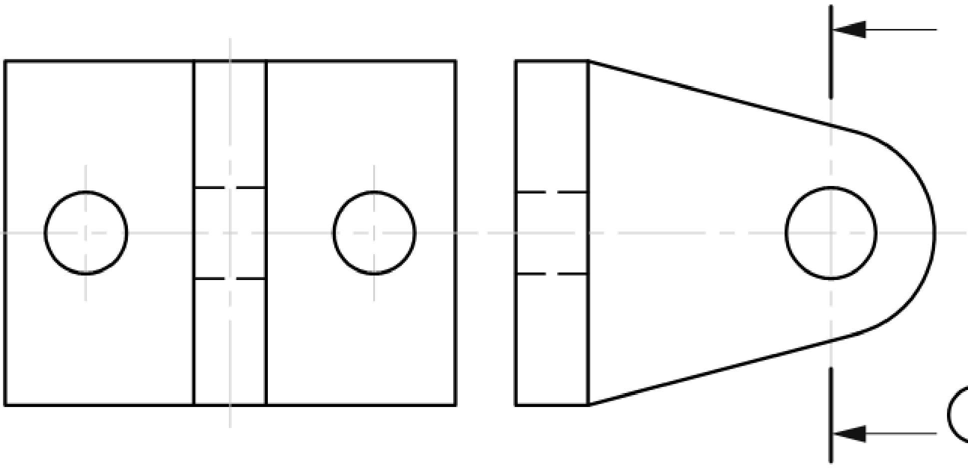

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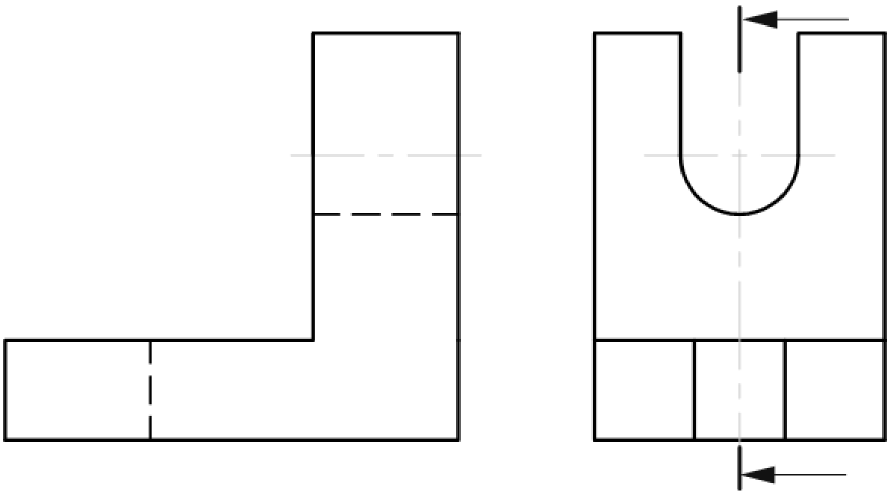

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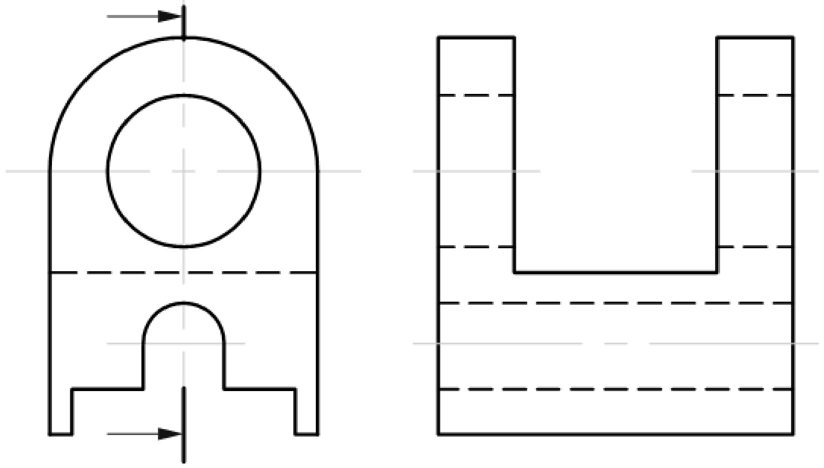

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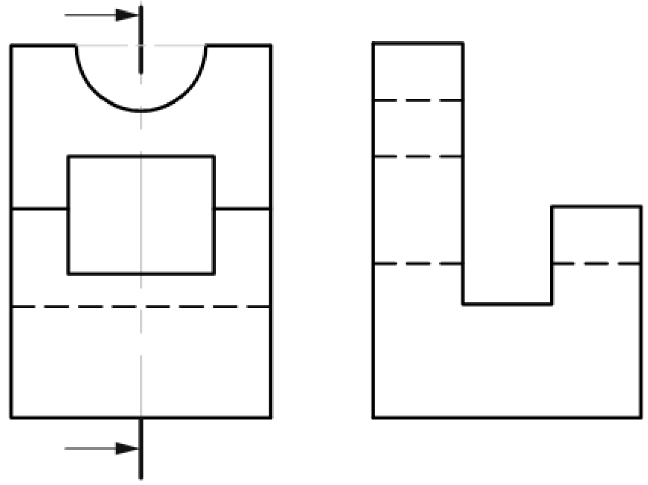

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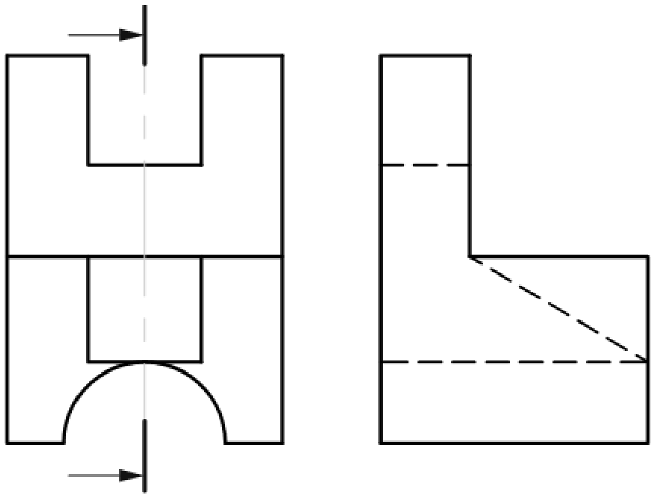

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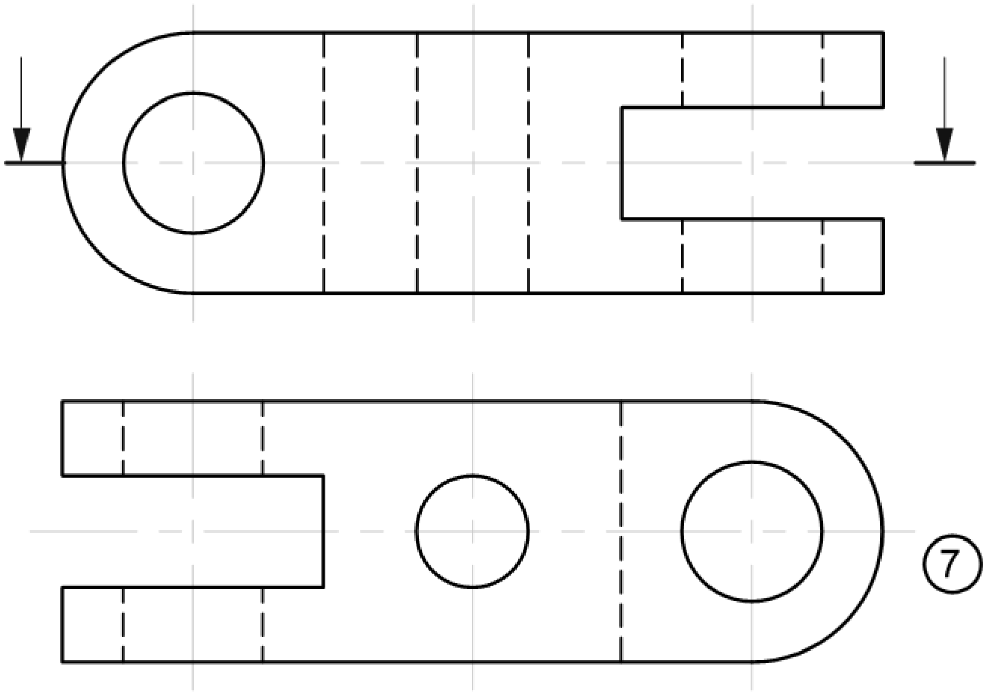
5. Draw Full Section plan.



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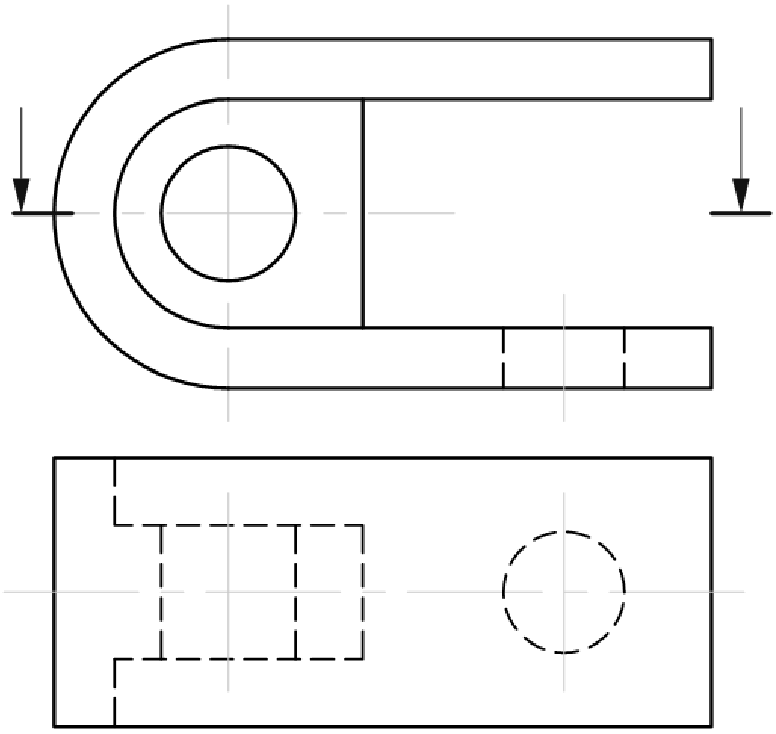
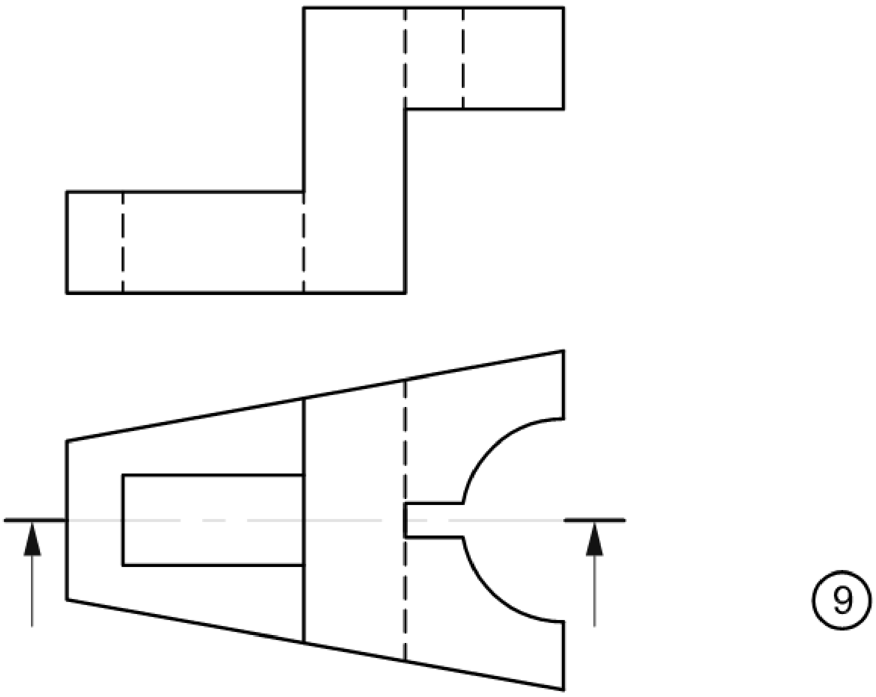
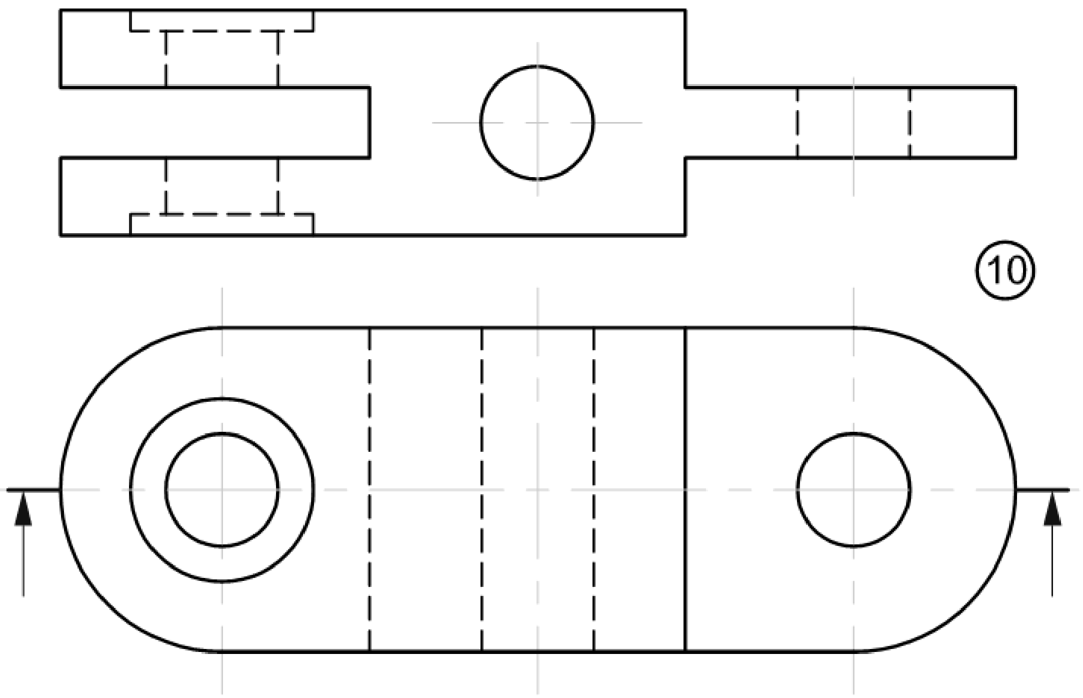
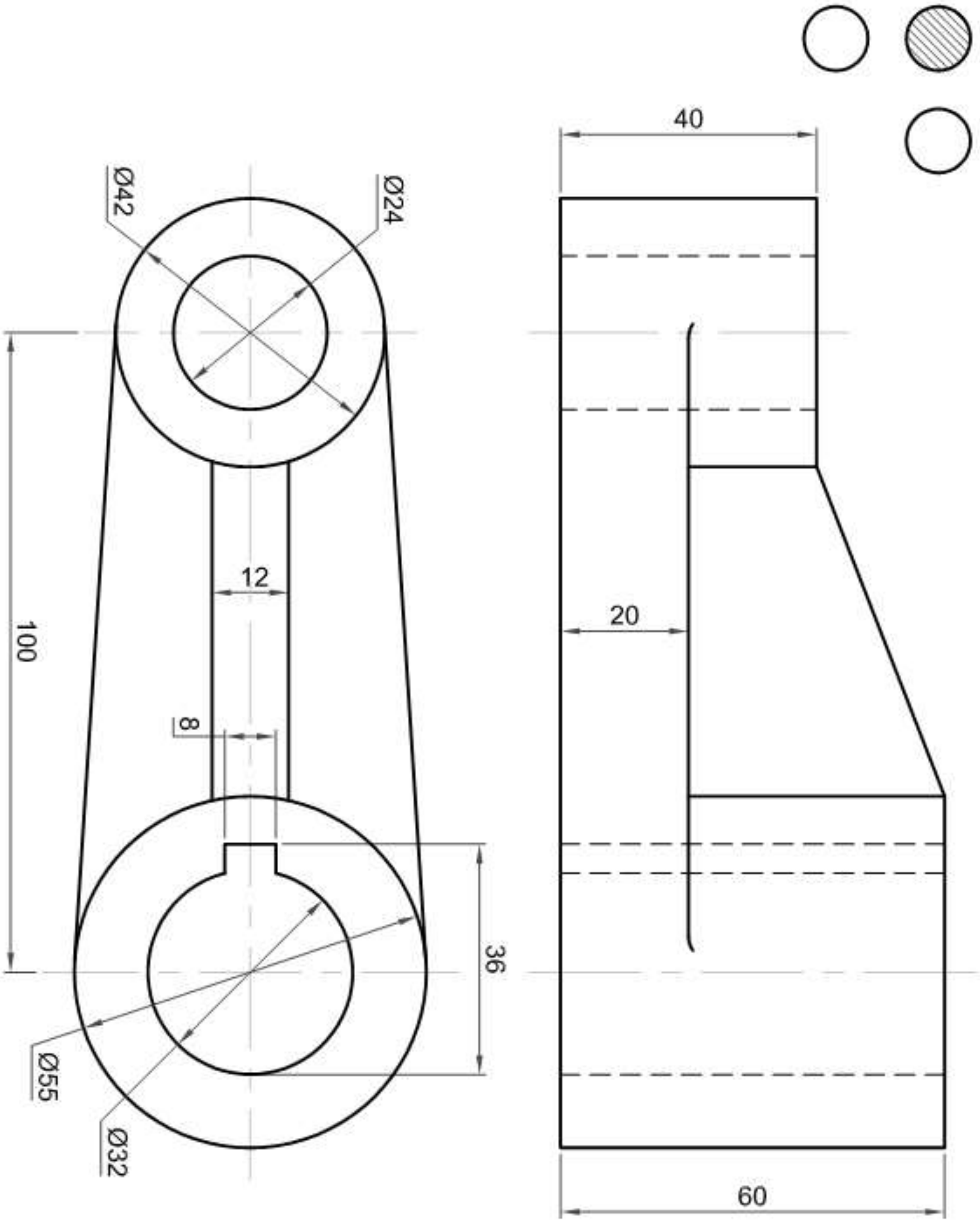


8

6. **Draw Full Section Fornt view.**



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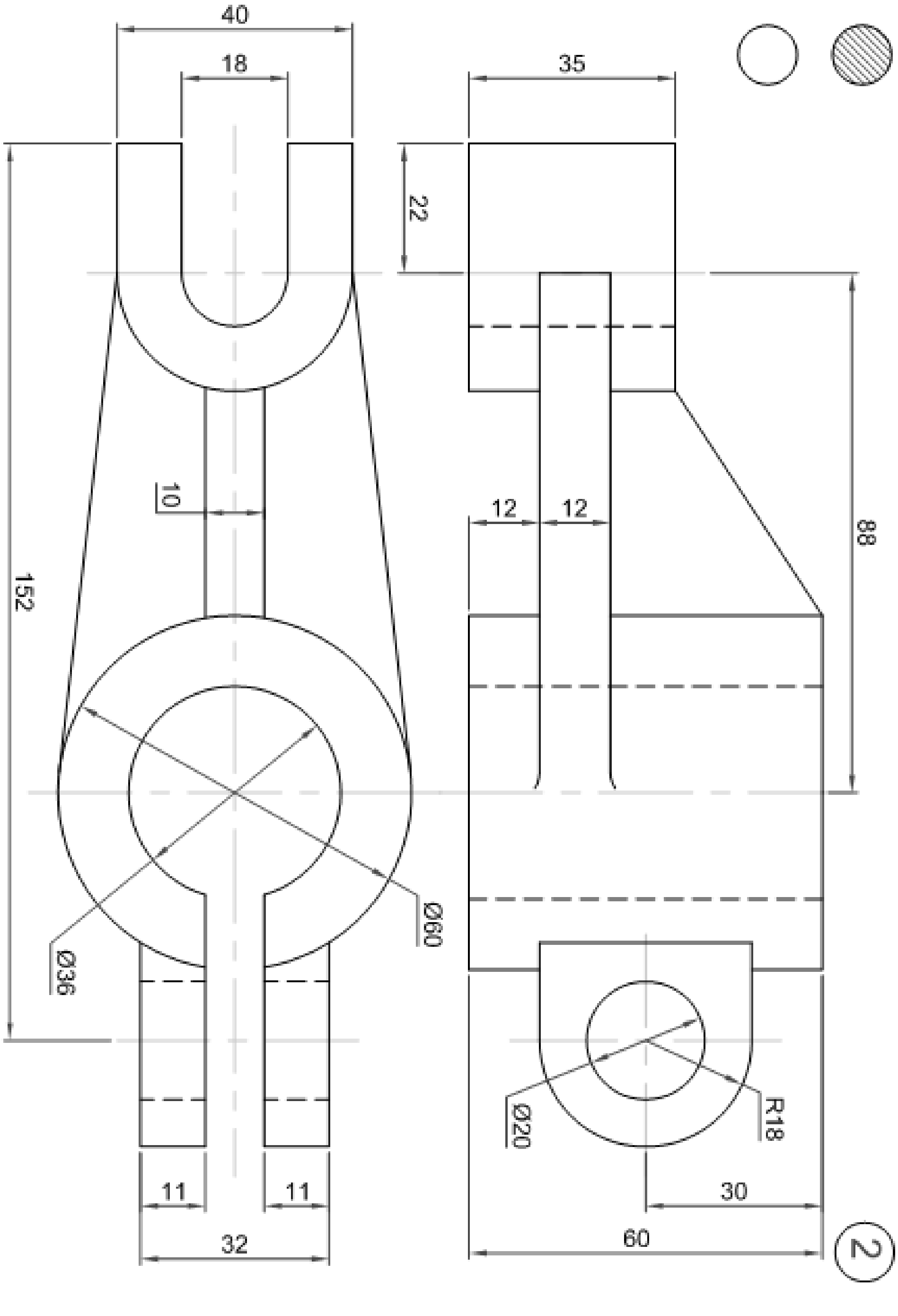
Ø42
Ø24
12
20
100
8
36
Ø55
Ø32
60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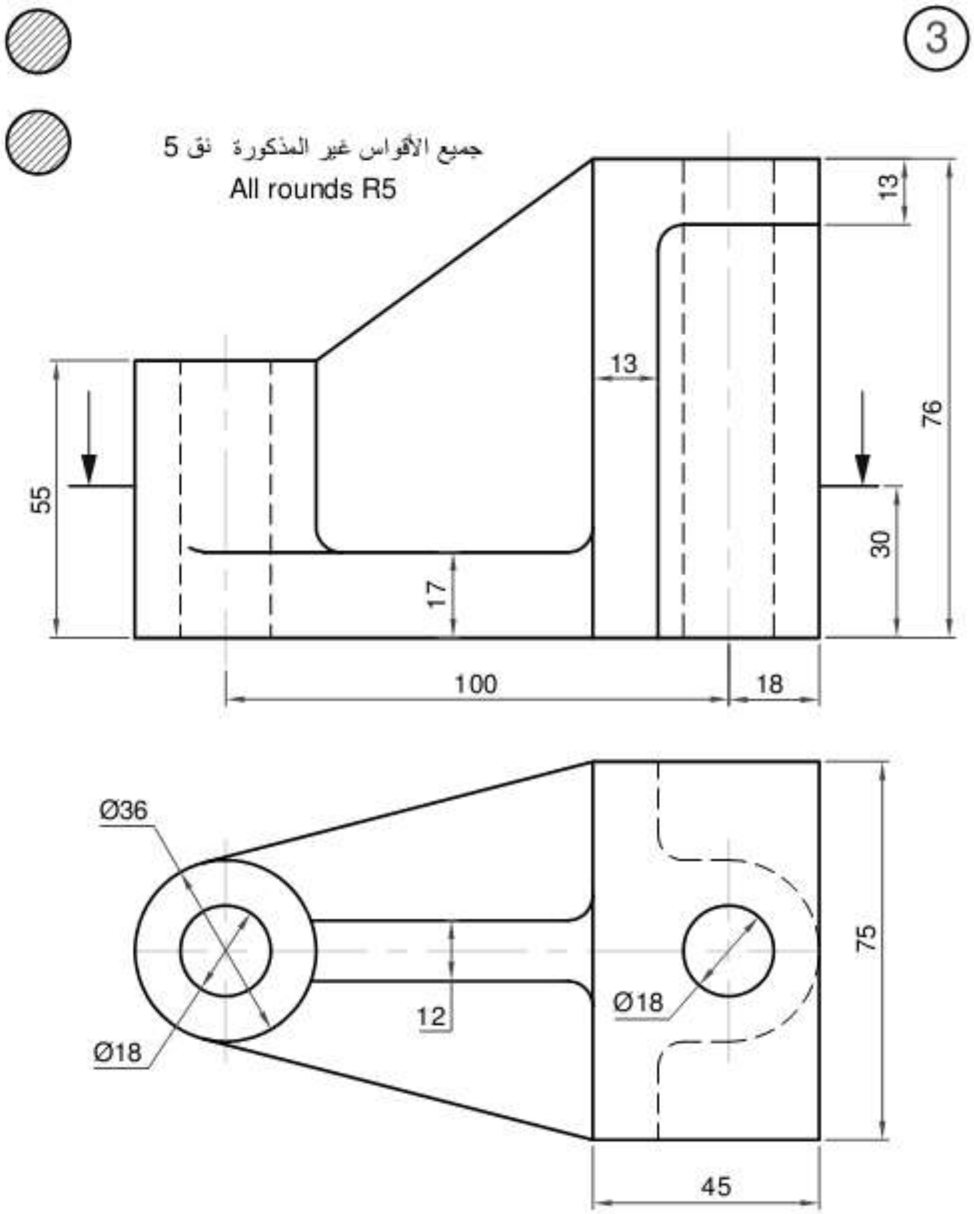
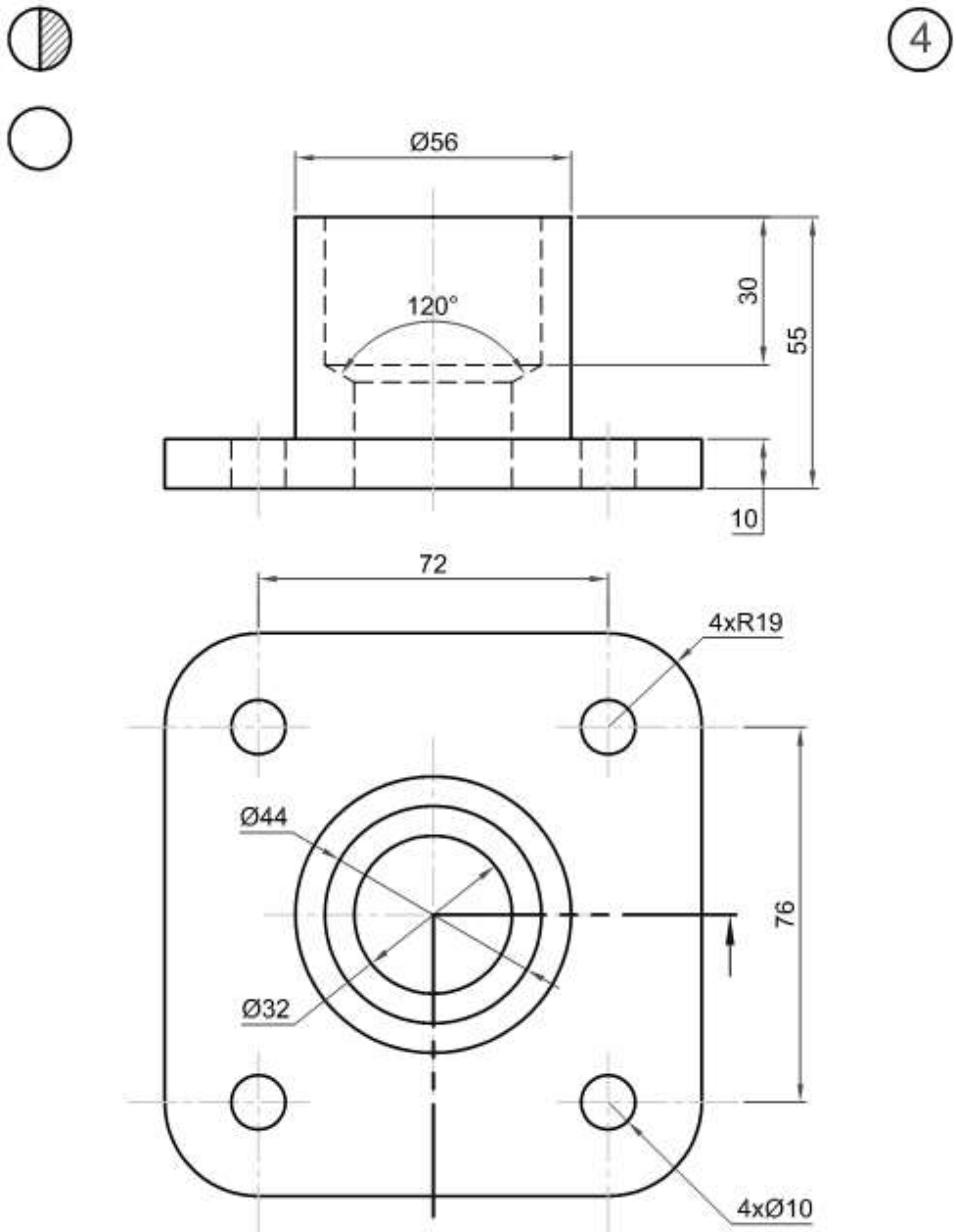
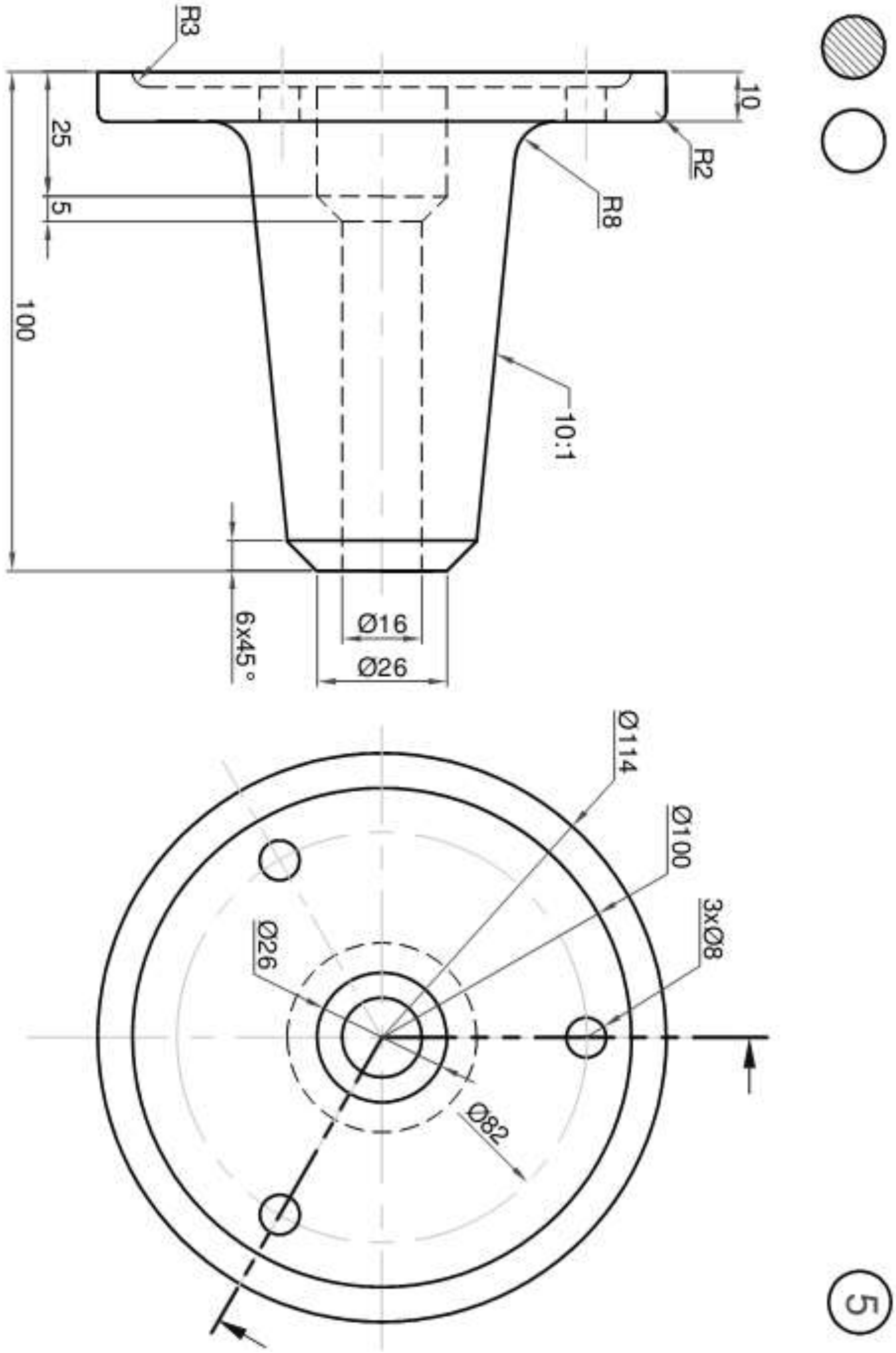
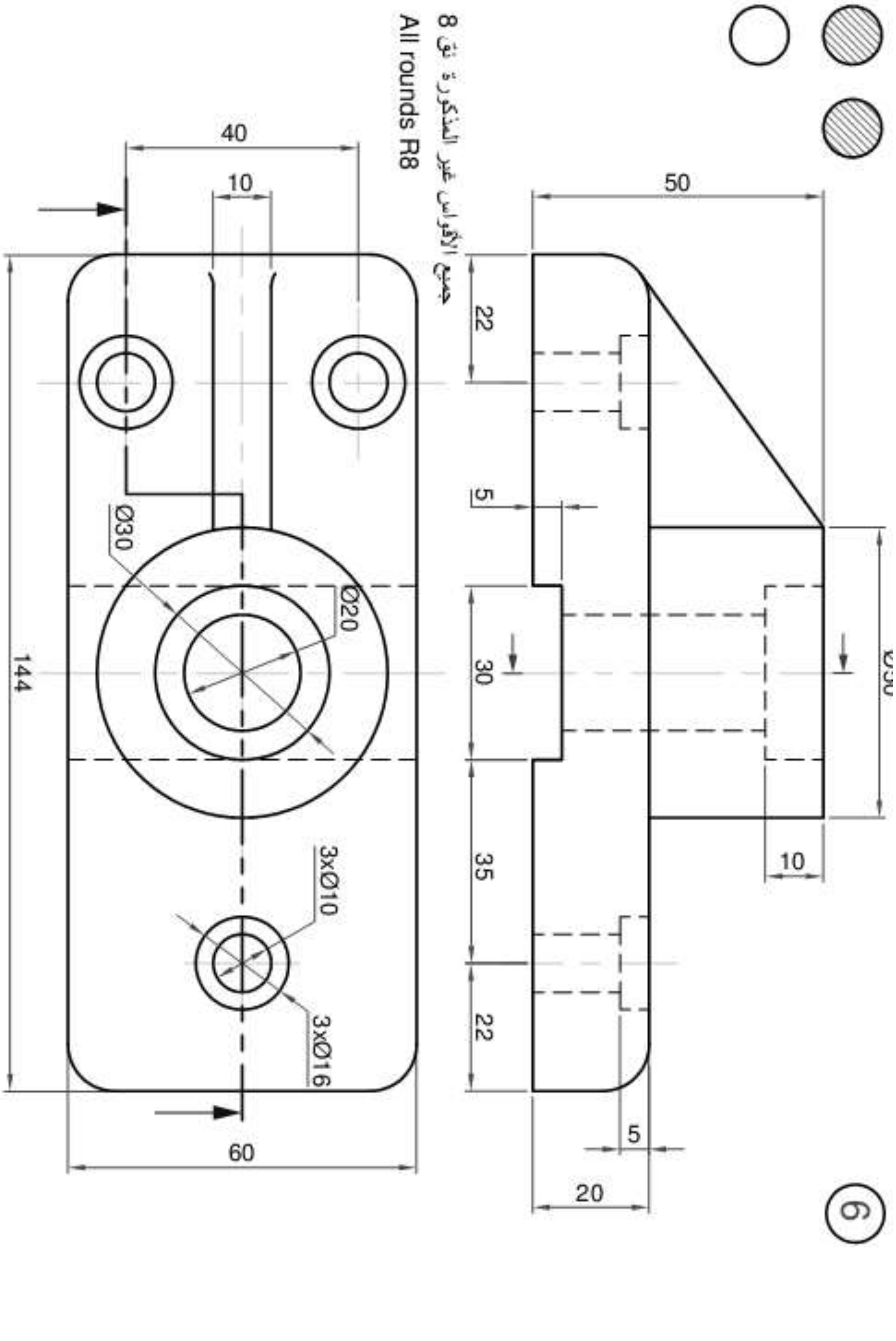
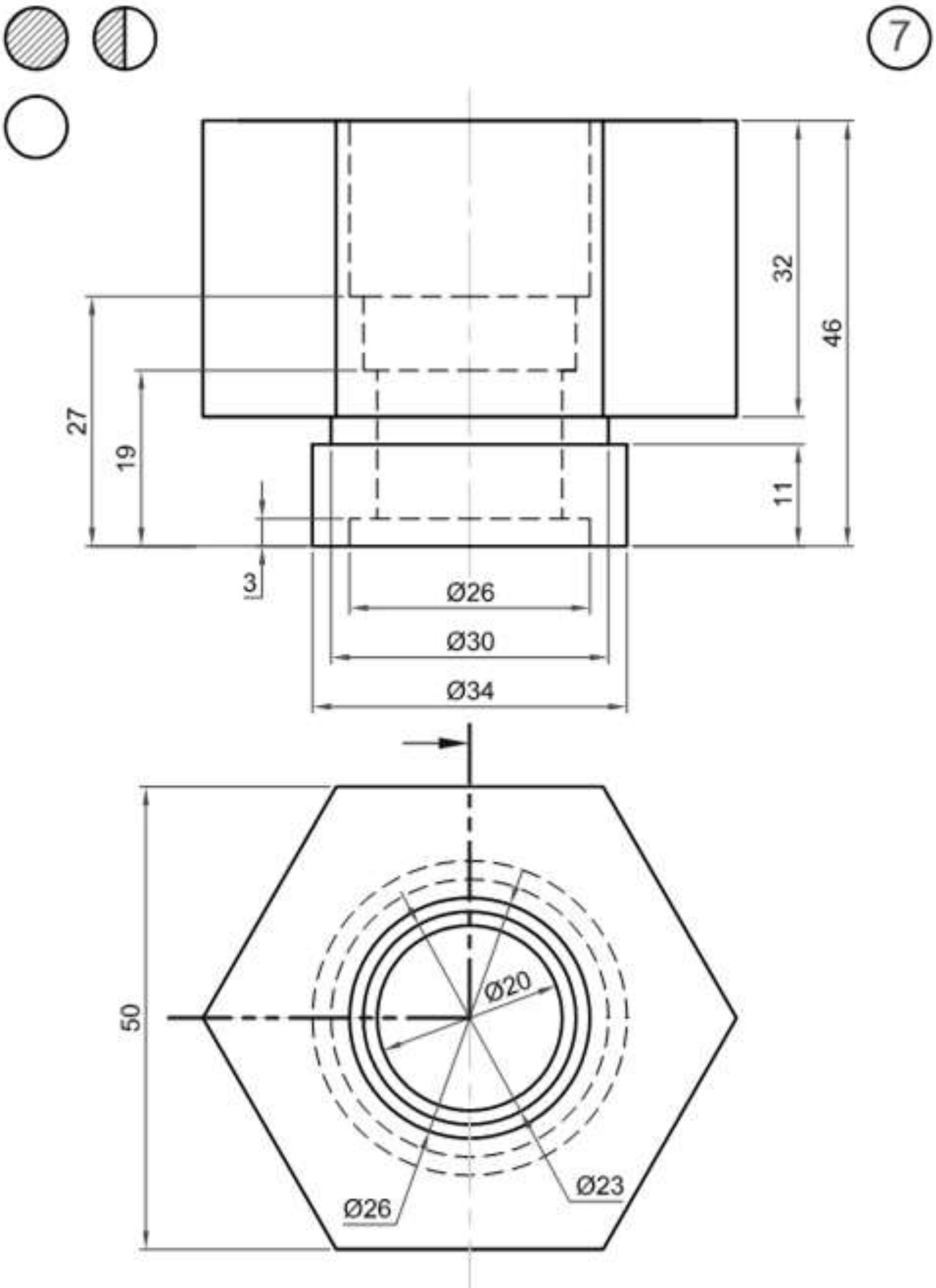
22
10
12
12
152
88
Ø60
Ø36
Ø20
R18
11
11
32
30
60
2

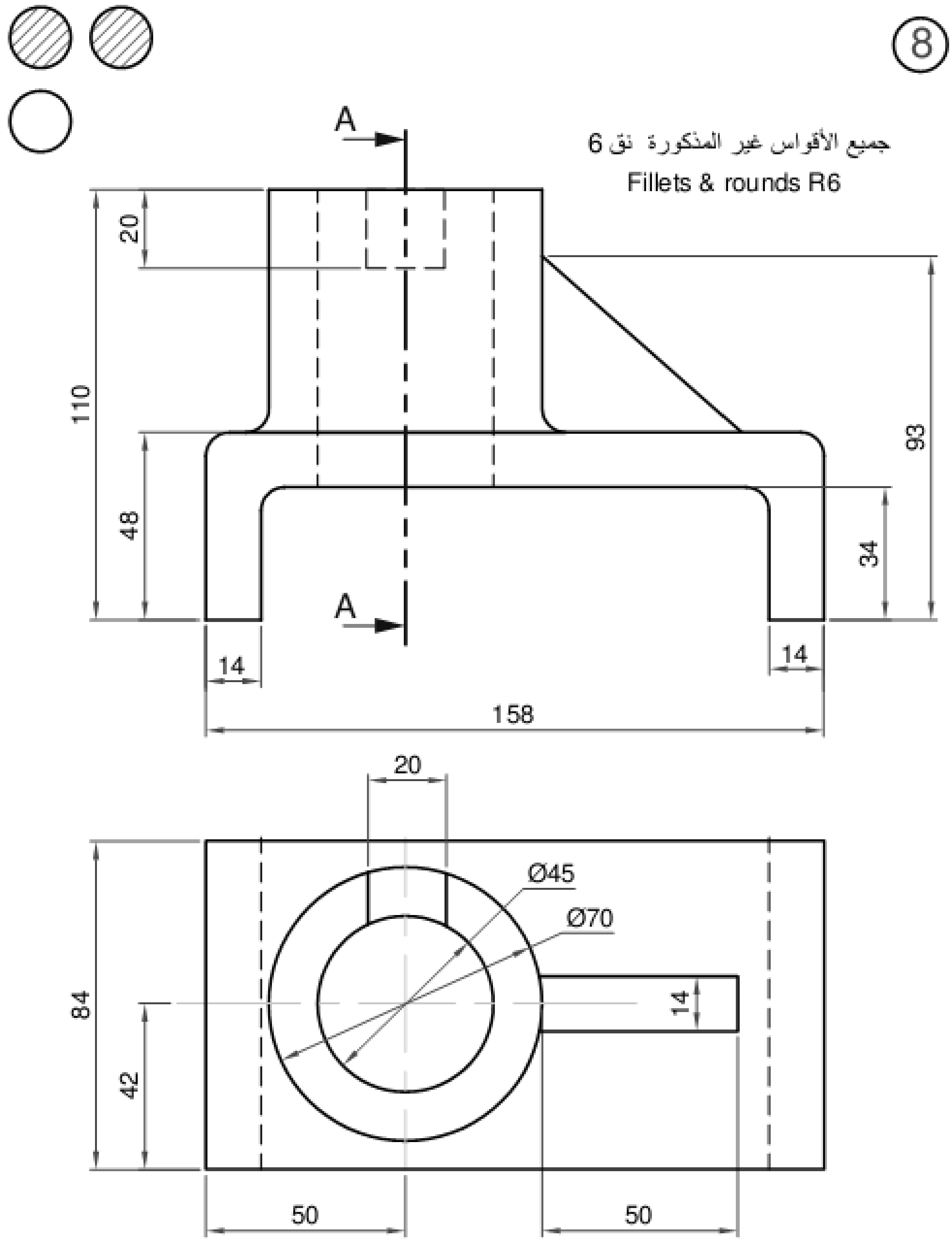

3
جميع الأقواس غير المذكورة نق 5
All rounds R5
13
13
76
55
30
17
100
18
Ø36
Ø18
12
Ø18
75
45

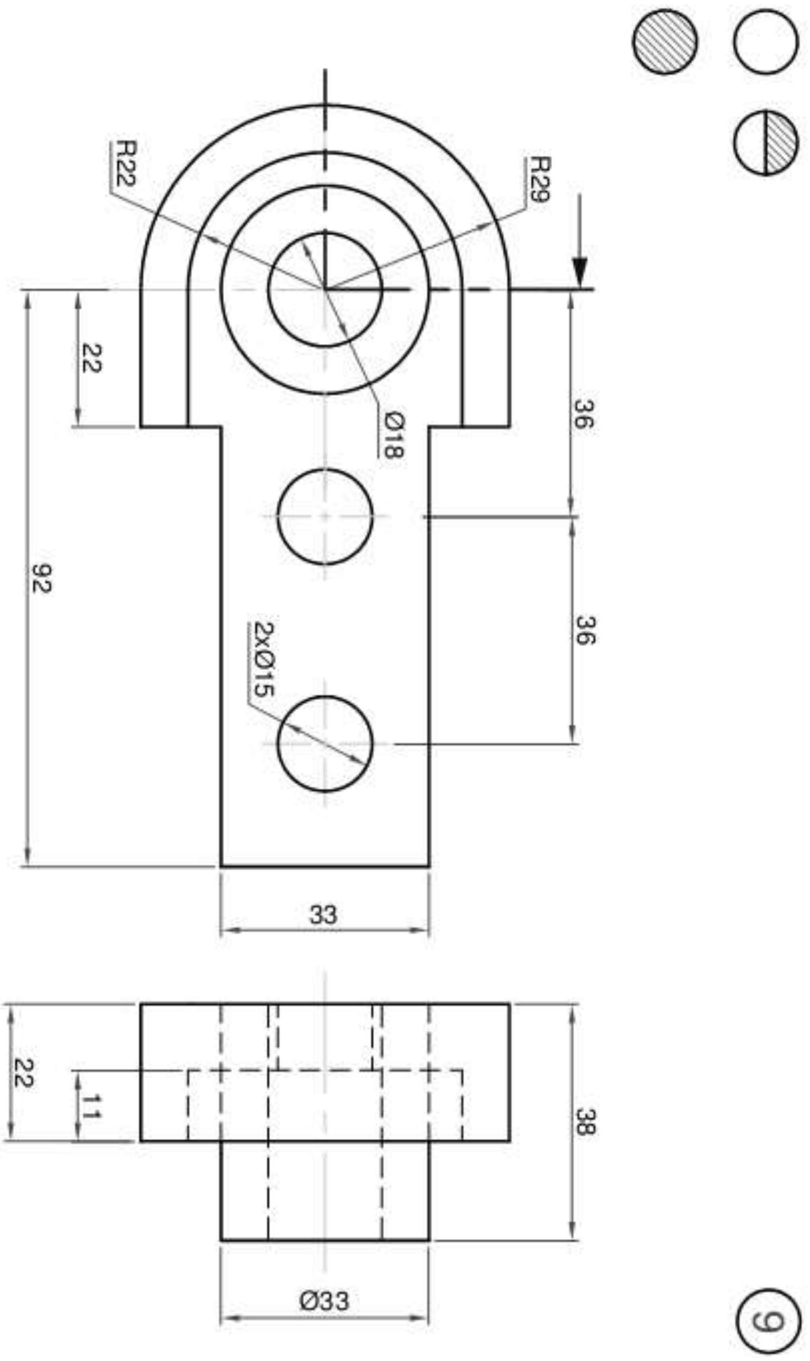

4
Ø56
120°
30
55
10
72
4xR19
Ø44
76
Ø32
4xØ10

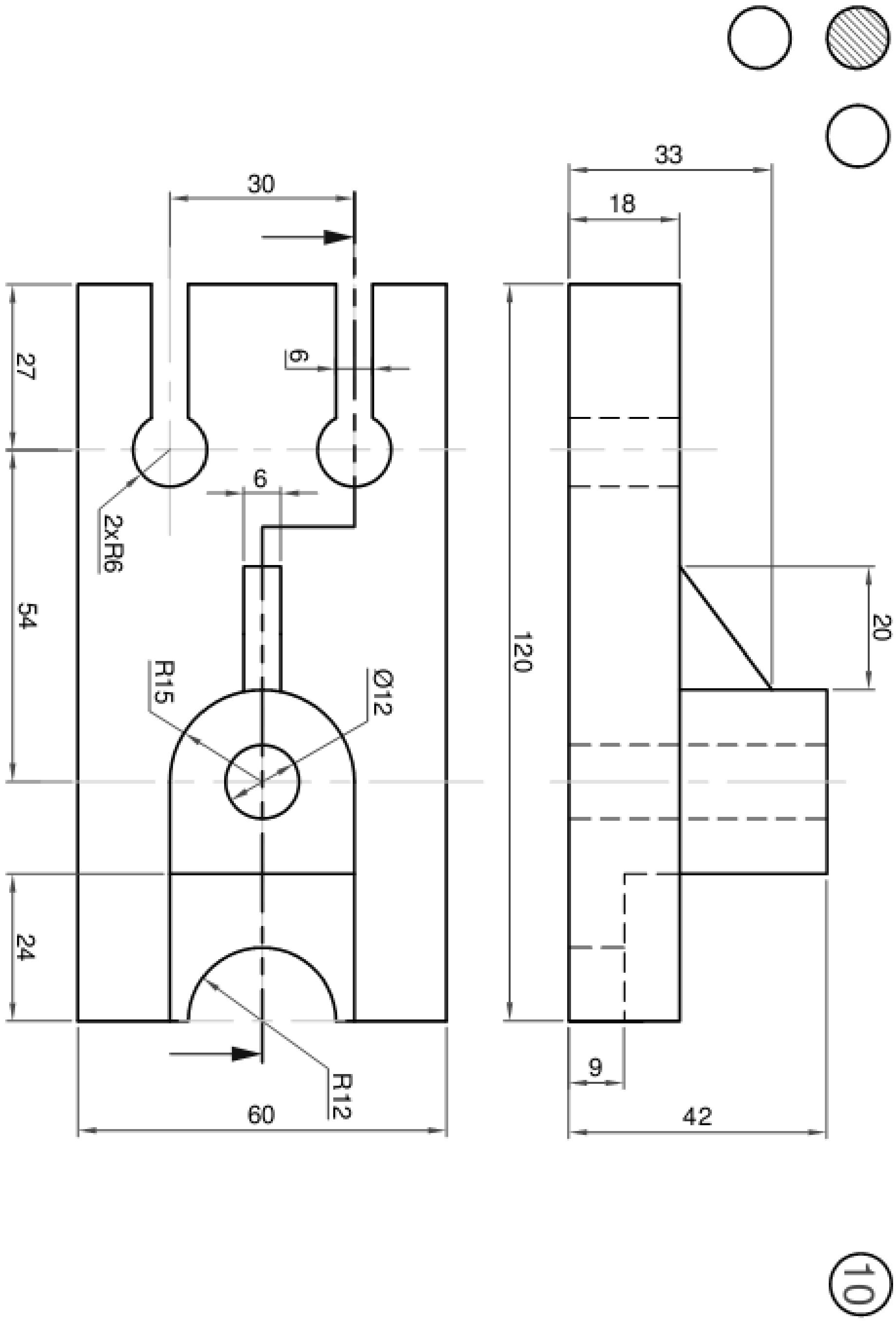

R3
10
25
R2
R8
5
100
10:1
6x45°
Ø16
Ø26
Ø114
Ø100
3xØ8
Ø26
Ø82
5

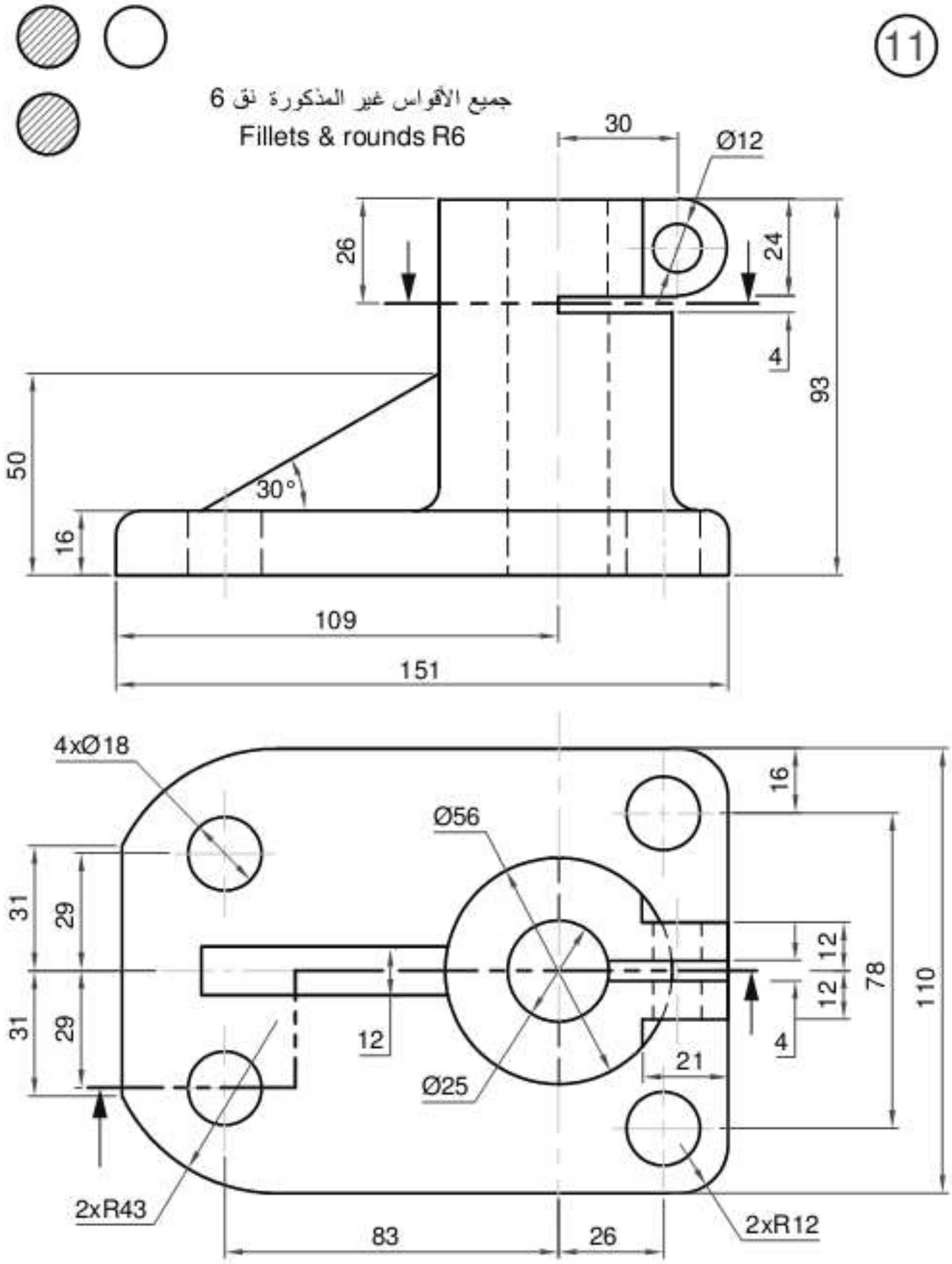

All rounds R8
جميع الأقواس غير المذكورة نق 8
40
10
50
144
22
5
30
35
22
Ø30
Ø20
3xØ10
3xØ16
60
20
5
10
Ø50
9

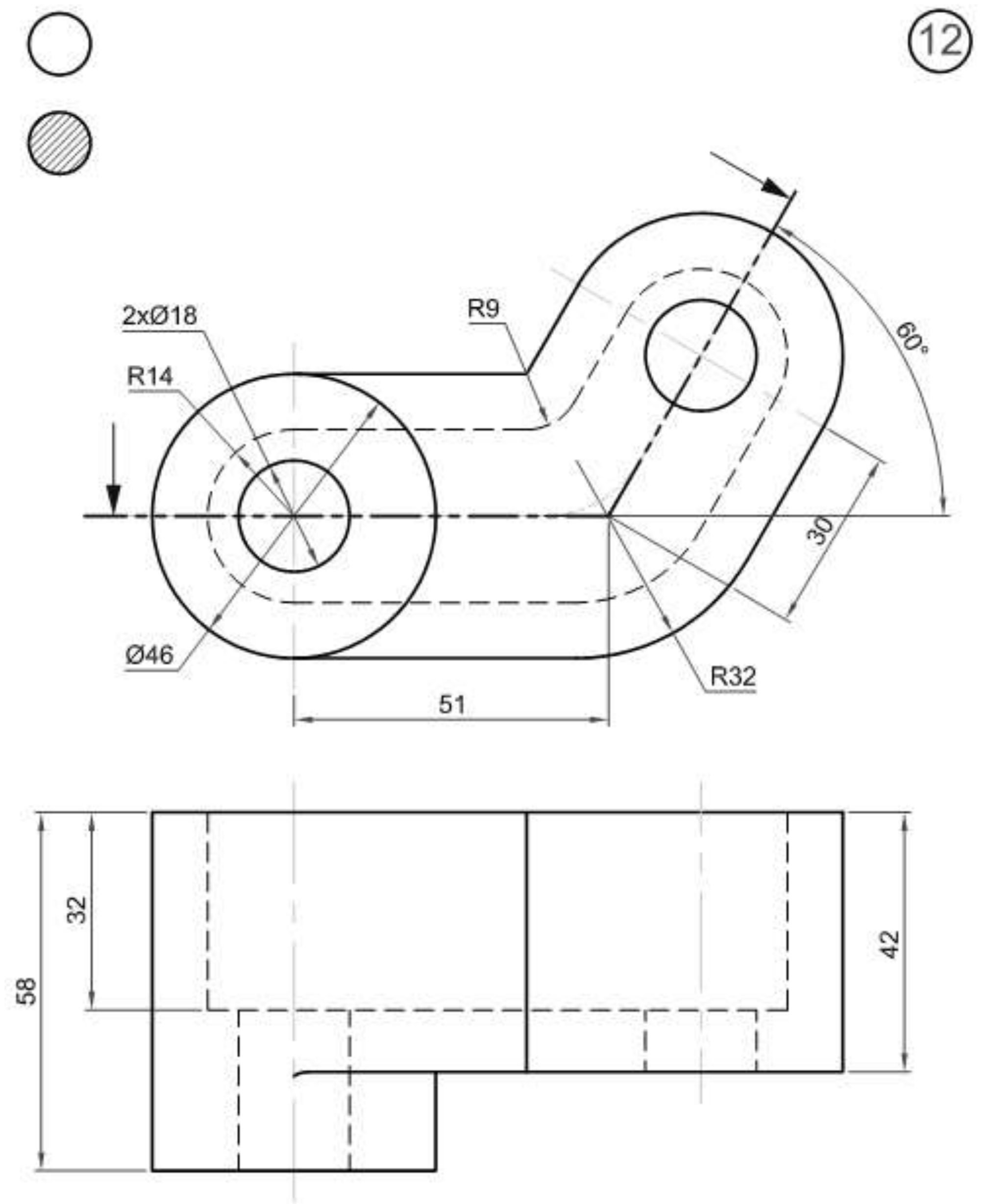

7
32
46
27
19
11
3
Ø26
Ø30
Ø34
50
Ø20
Ø23
Ø26

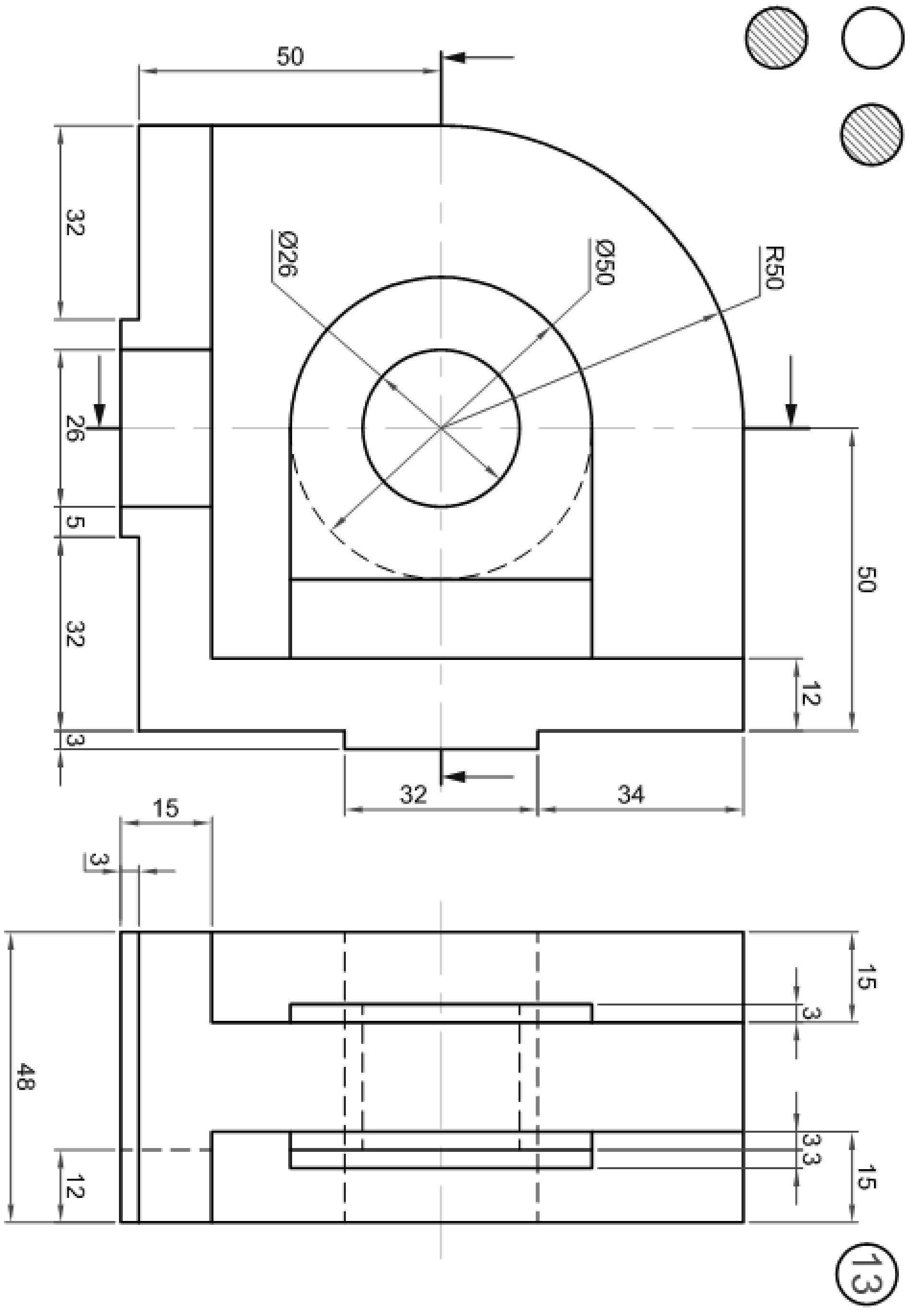

8
A
A
جميع الأقواس غير المذكورة نق 6
Fillets & rounds R6
20
110
48
93
34
14
14
158
20
Ø45
Ø70
84
42
14
50
50


R22
R29
22
36
Ø18
92
36
2xØ15
33
22
11
38
Ø33
9


30
6
27
6
2xR6
54
R15
Ø12
24
R12
60
120
33
18
20
9
42
10


11
جميع الأقواس غير المذكورة نق 6
Fillets & rounds R6
30
Ø12
26
24
4
93
50
30°
16
109
151
4xØ18
Ø56
16
31
29
31
29
12
Ø25
12
12
78
110
4
21
2xR43
83
26
2xR12


12
2xØ18
R9
R14
60°
30
Ø46
R32
51
32
58
42


50
32
Ø26
Ø50
R50
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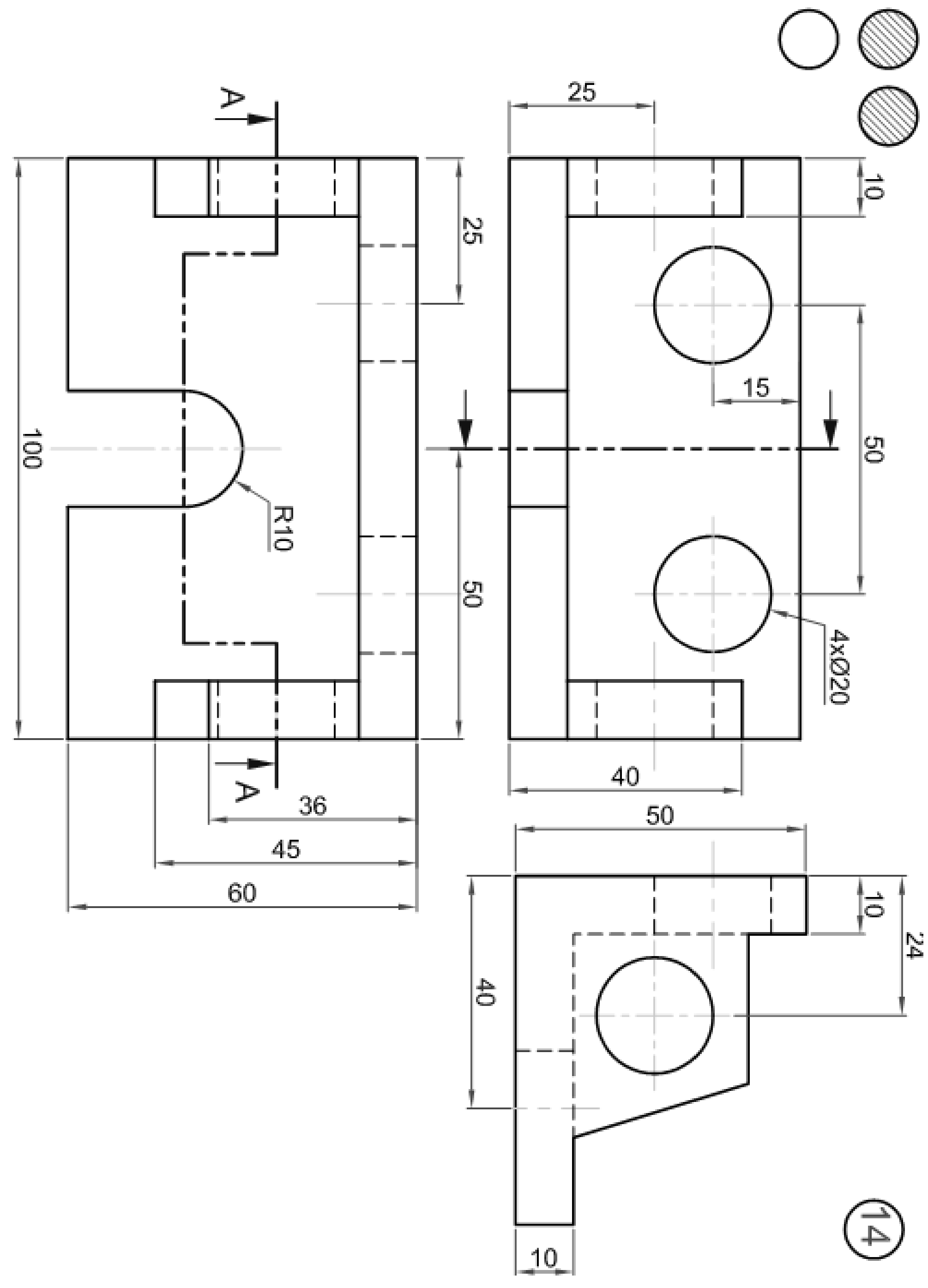
5
32
3
50
12
32
34
15
3
48
12
15
3
3 3
15
13


A
25
10
25
100
R10
50
15
50
4xØ20
A
40
36
45
60
50
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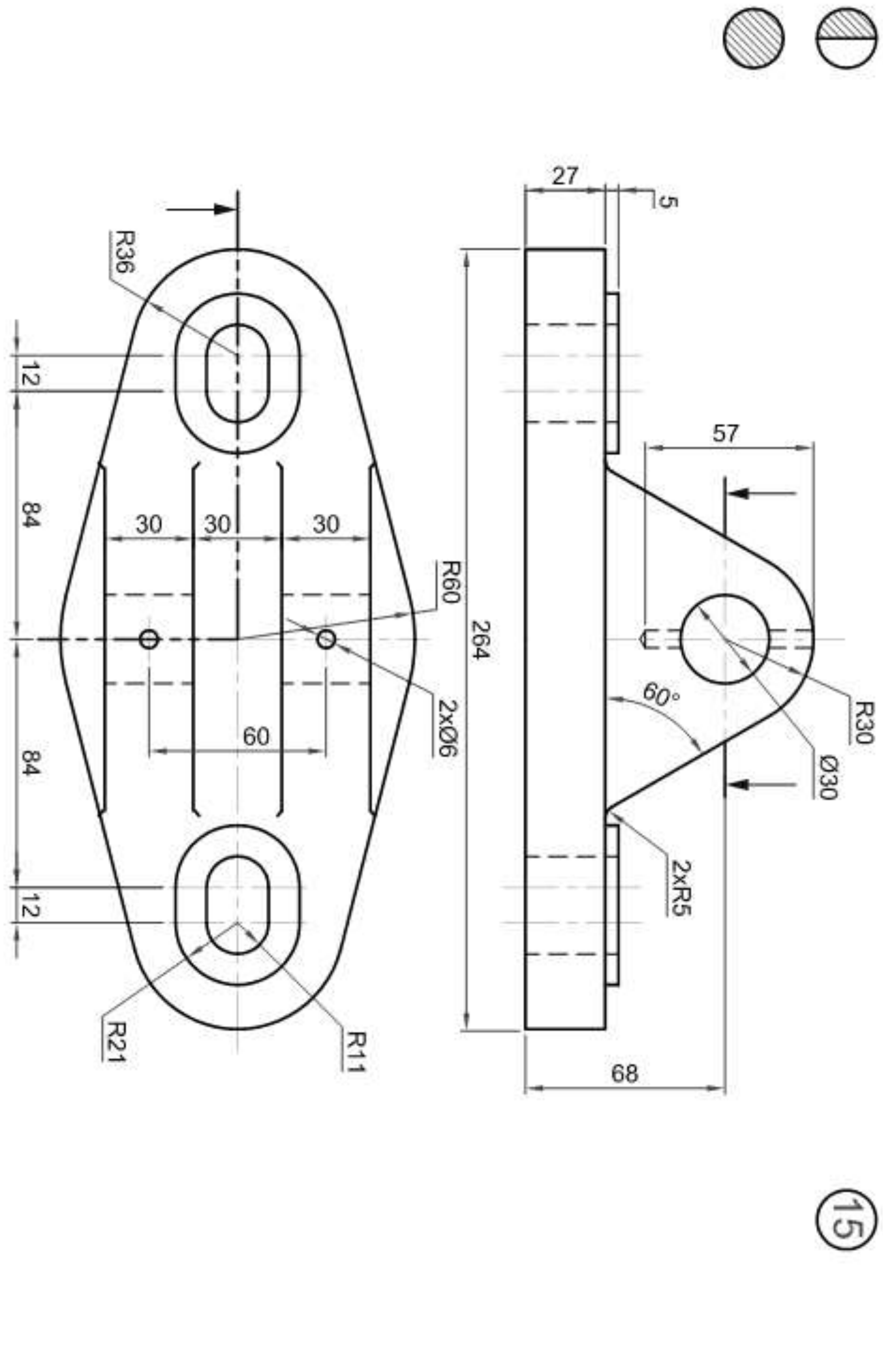
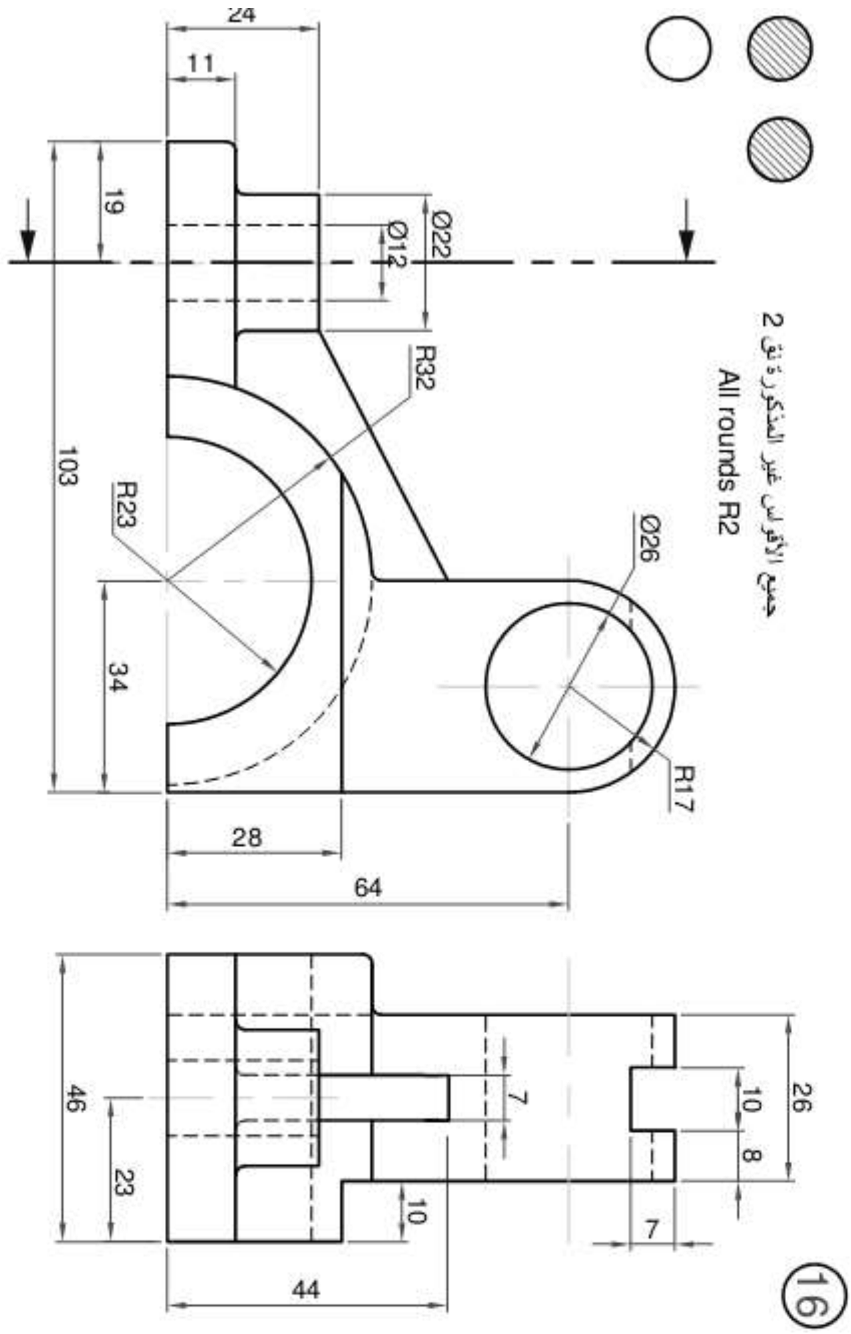
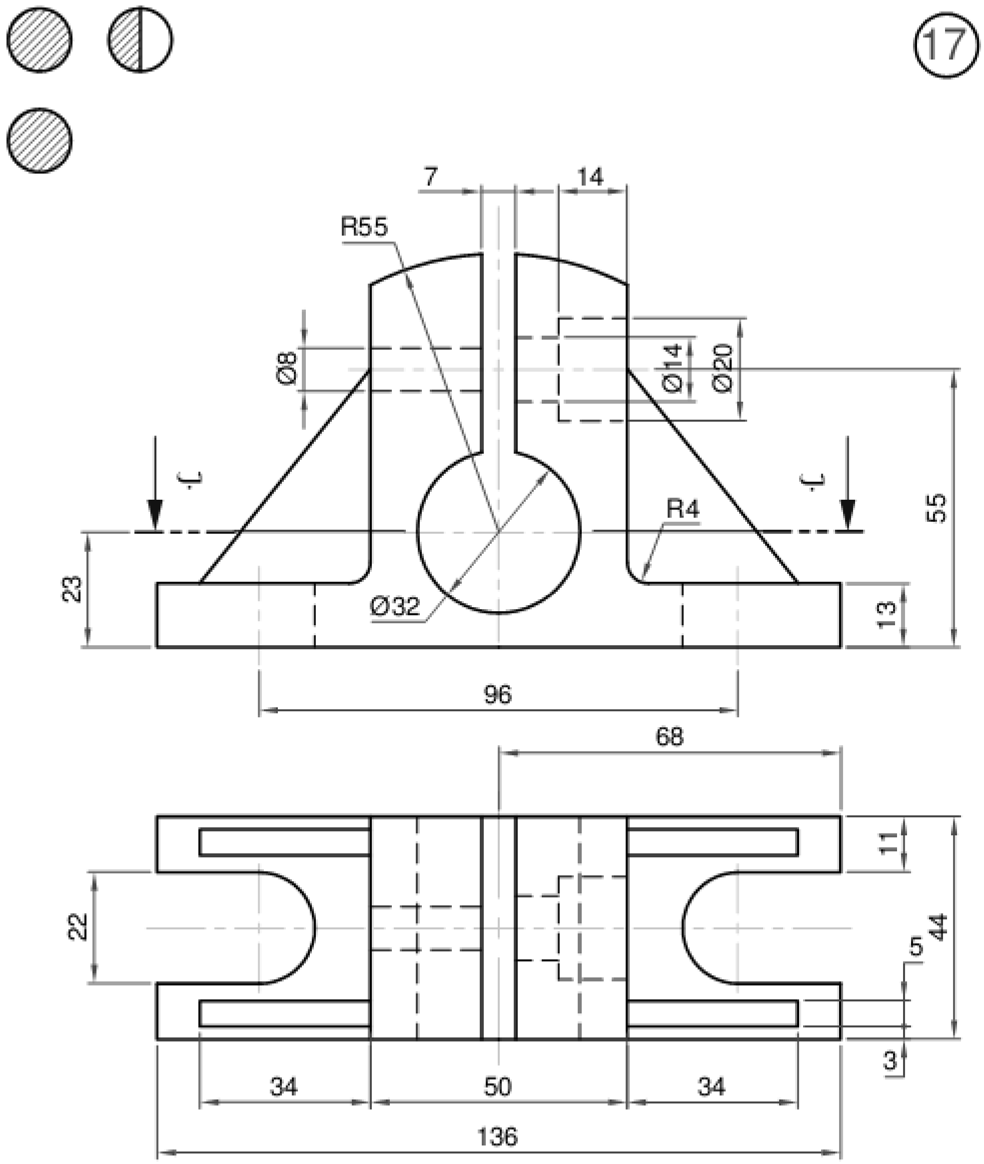
10
14


27
5
R36
12
84
30
30
30
R60
264
57
2xØ6
60
84
12
R21
R11
60°
R30
Ø30
2xR5
68
15


24
11
19
Ø22
Ø12
R32
103
R23
34
Ø26
R17
28
64
46
23
44
10
7
26
10
8
7
All rounds R2
جميع الأقواس غير المذكورة نق 2
16


17
7
14
R55
Ø8
Ø14
Ø20
55
R4
23
Ø32
13
96
68
11
22
44
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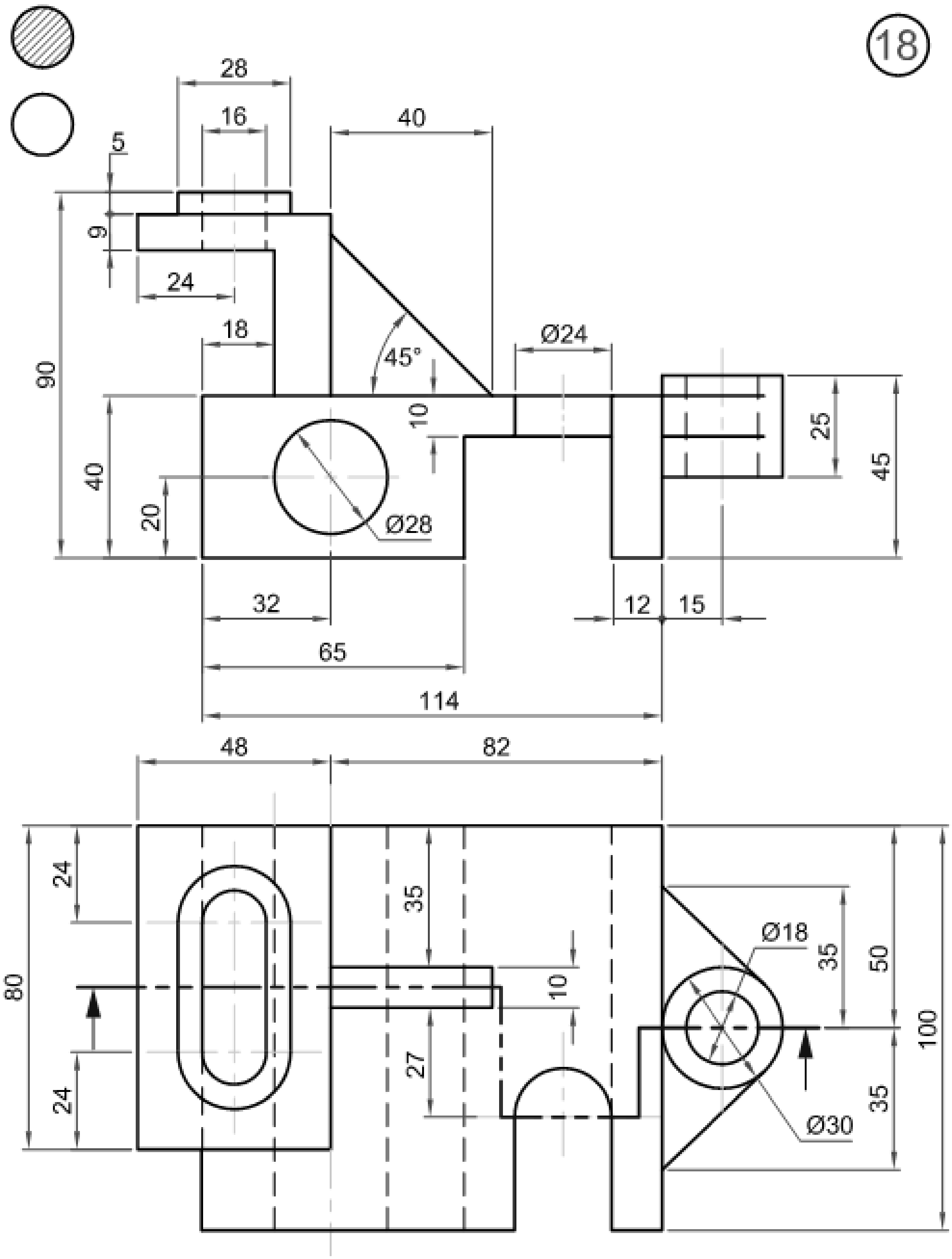
3
34
50
34
136


18
28
16
40
5
9
24
18
90
45°
Ø24
10
25
45
40
20
Ø28
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114
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48
82
24
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Ø18
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100
27
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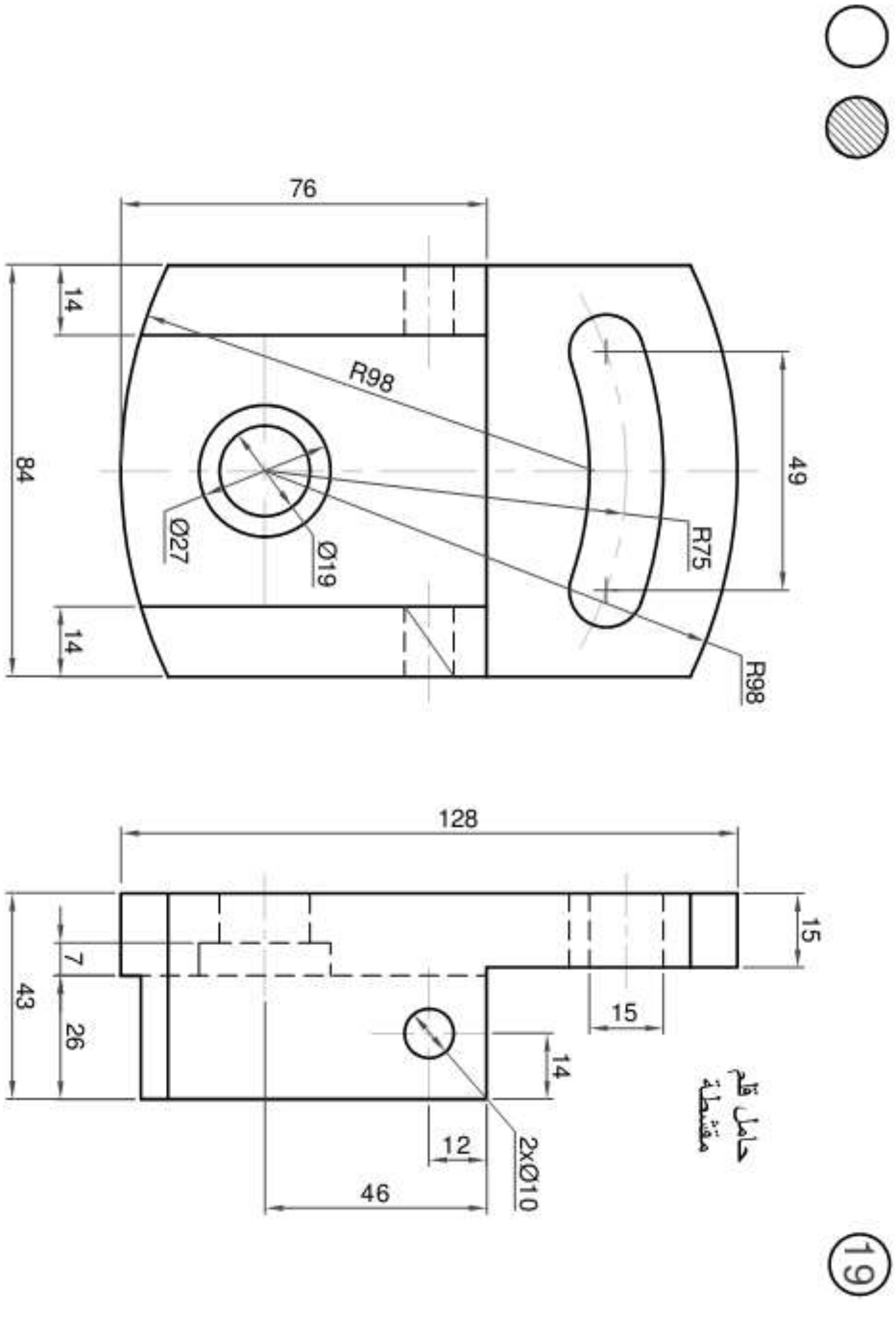
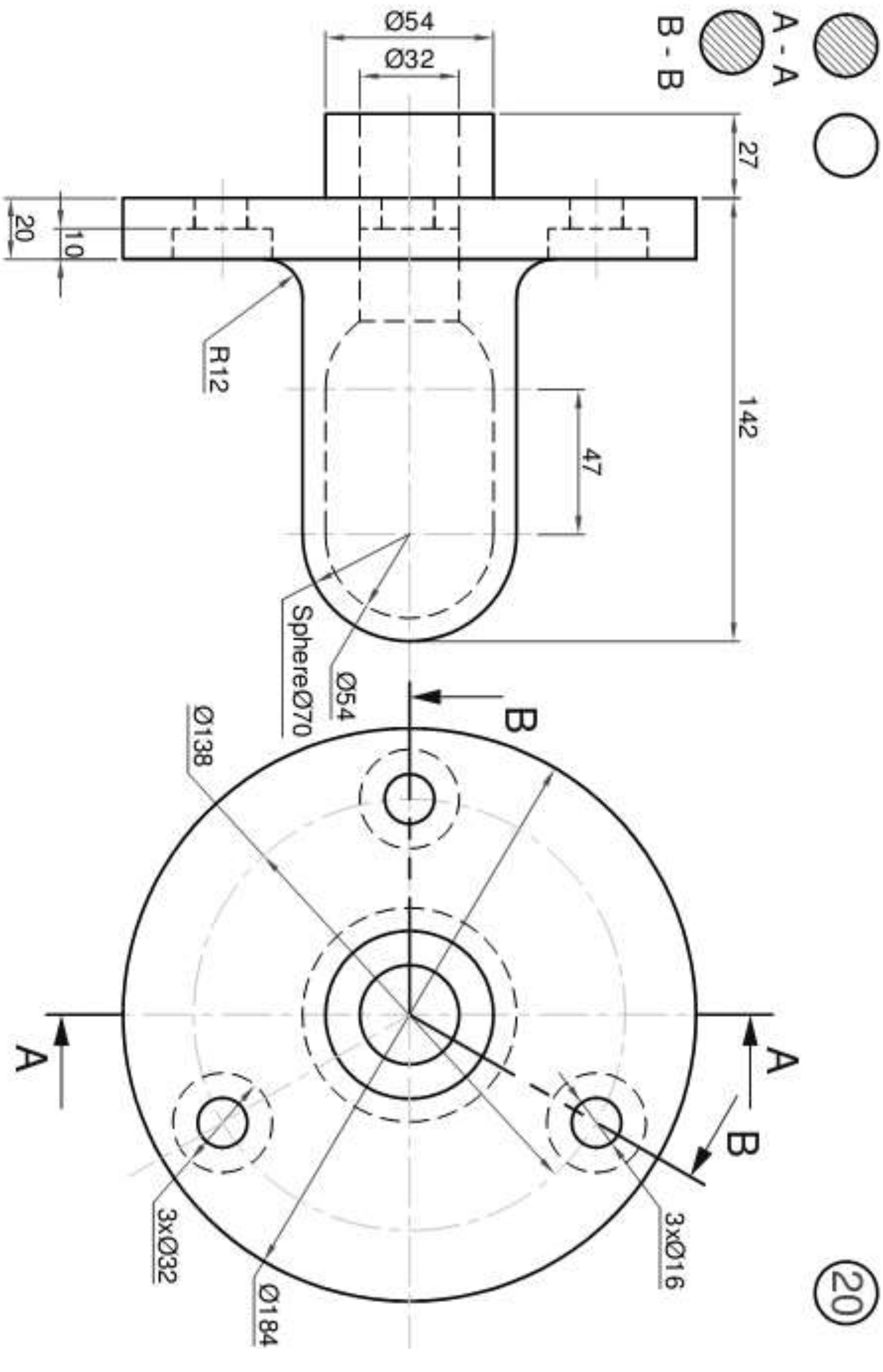
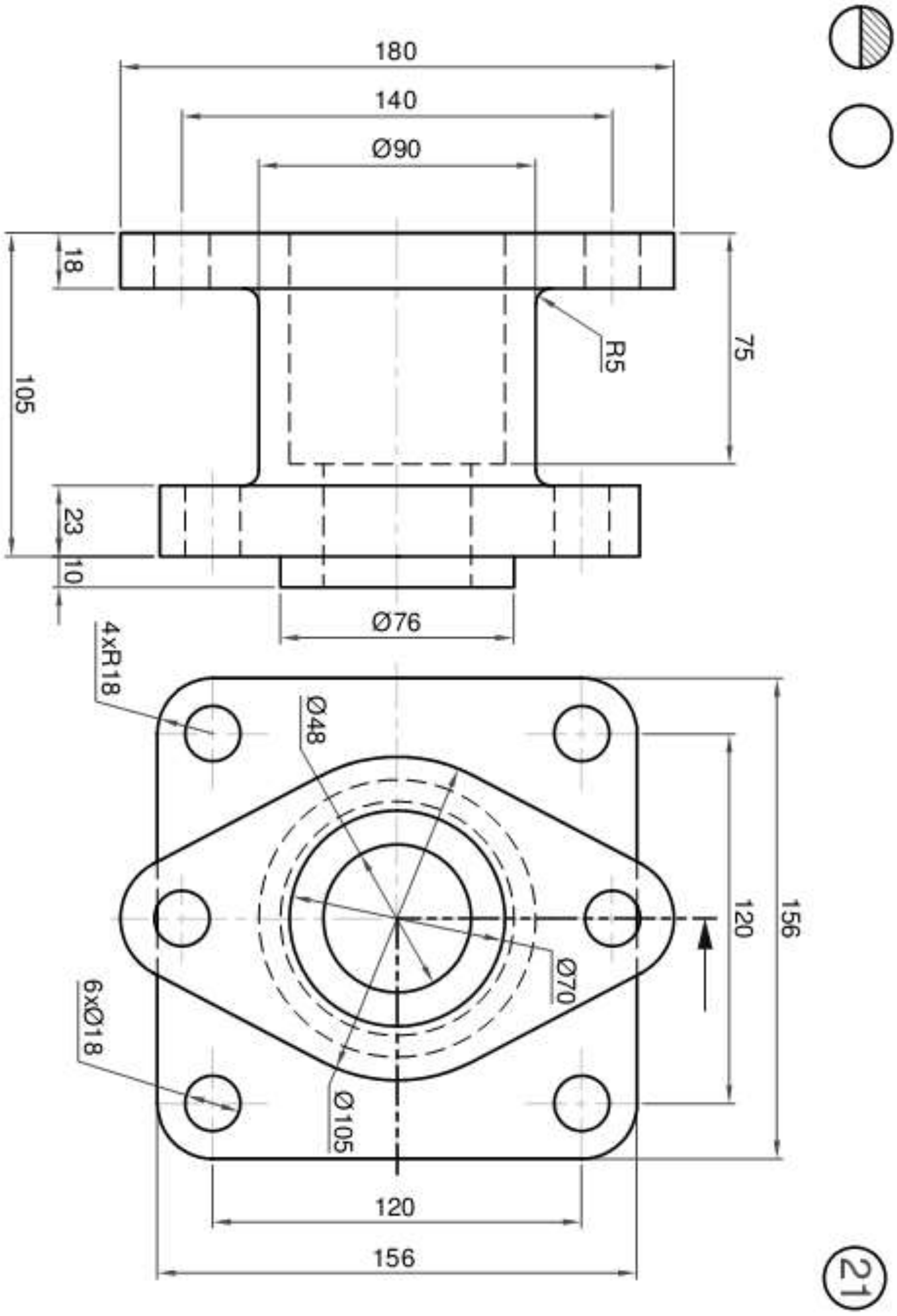
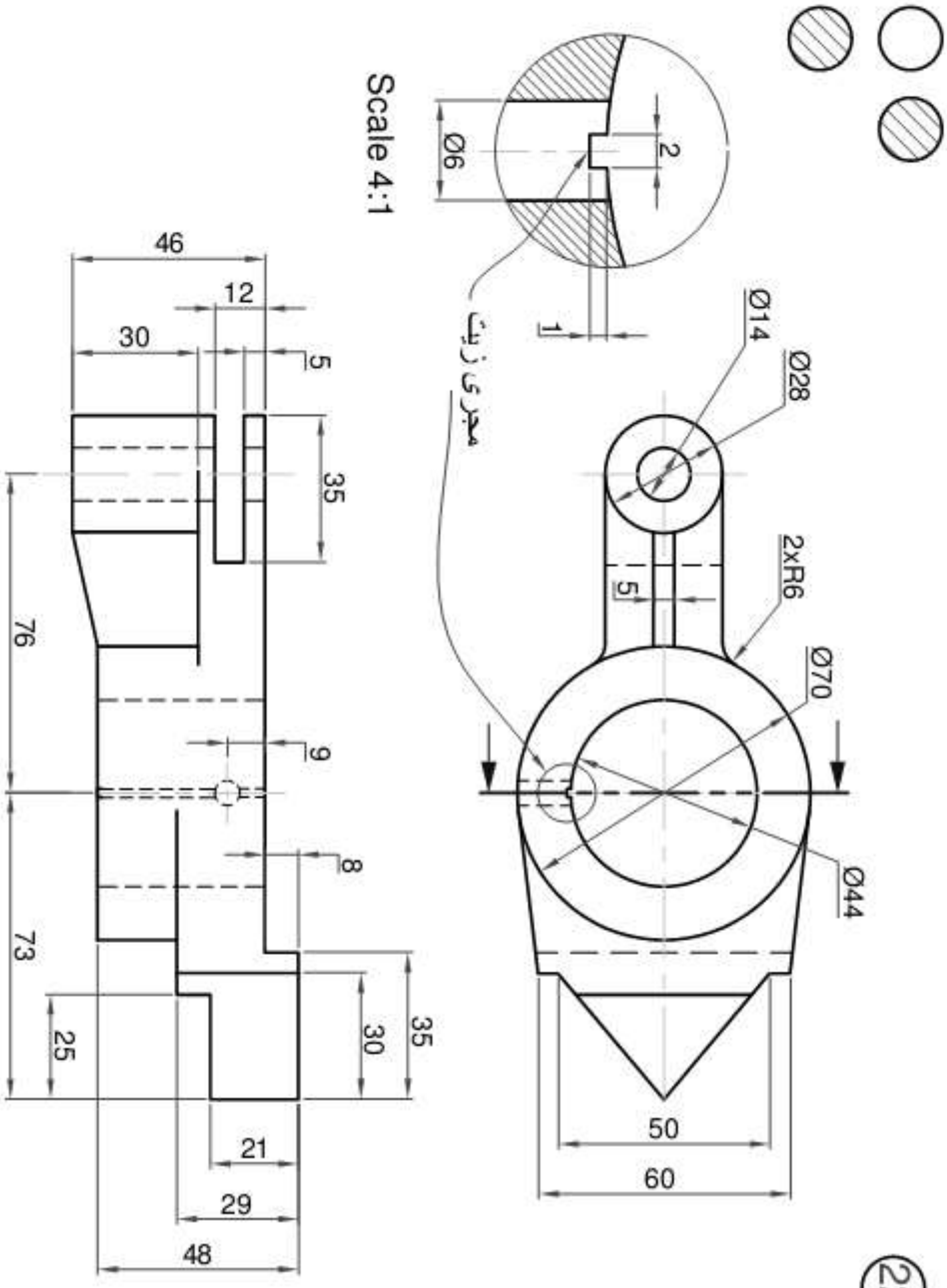
35
Ø30


76
14
84
R98
Ø27
Ø19
R75
49
14
R98
128
15
7
43
26
15
14
12
46
2xØ10
19

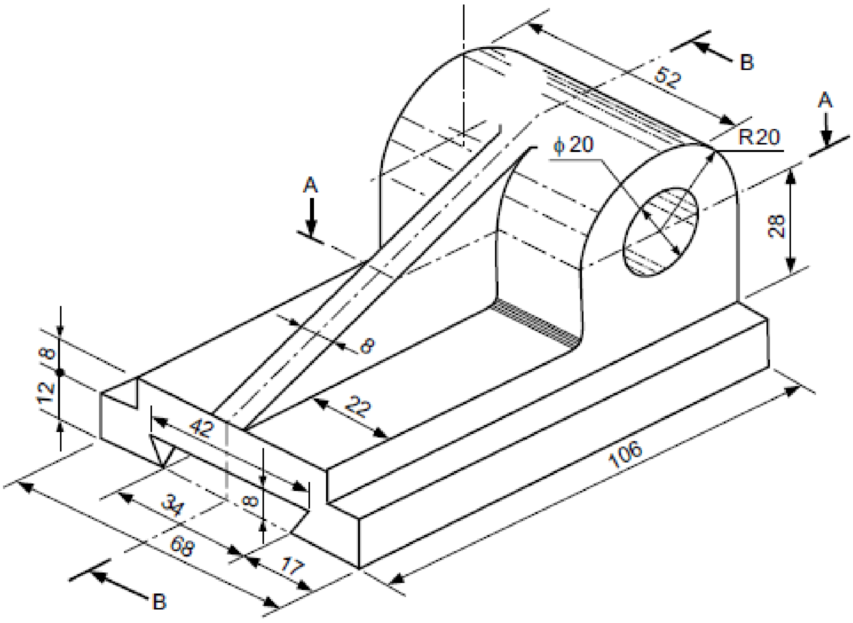

Ø54
Ø32
A - A
B - B
27
20
10
R12
142
47
SphereØ70
Ø54
Ø138
B
A
A
B
3xØ32
Ø184
3xØ16
20


180
140
Ø90
18
105
R5
75
23
10
Ø76
4xR18
Ø48
Ø70
120
156
6xØ18
Ø105
120
156
21

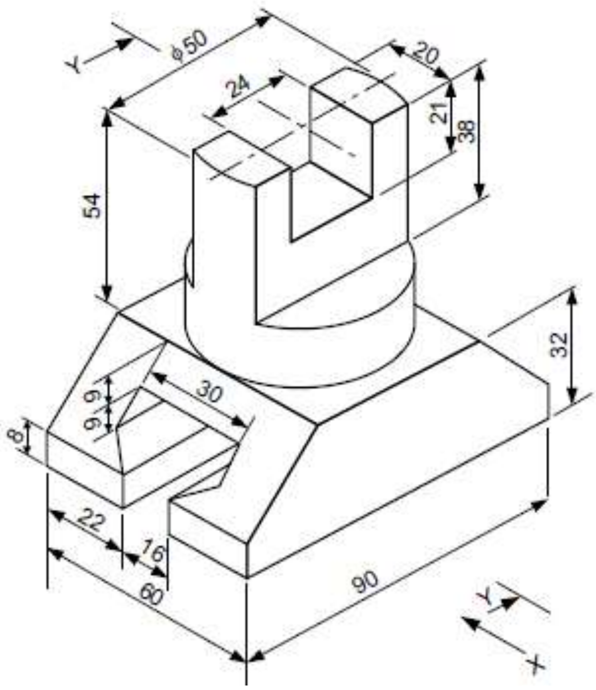

Scale 4:1
Ø6
2
1
مجرى زيت
Ø14
Ø28
2xR6
5
Ø70
Ø44
50
60
46
12
30
5
35
76
9
8
73
25
30
35
21
29
48
22

Additional problems

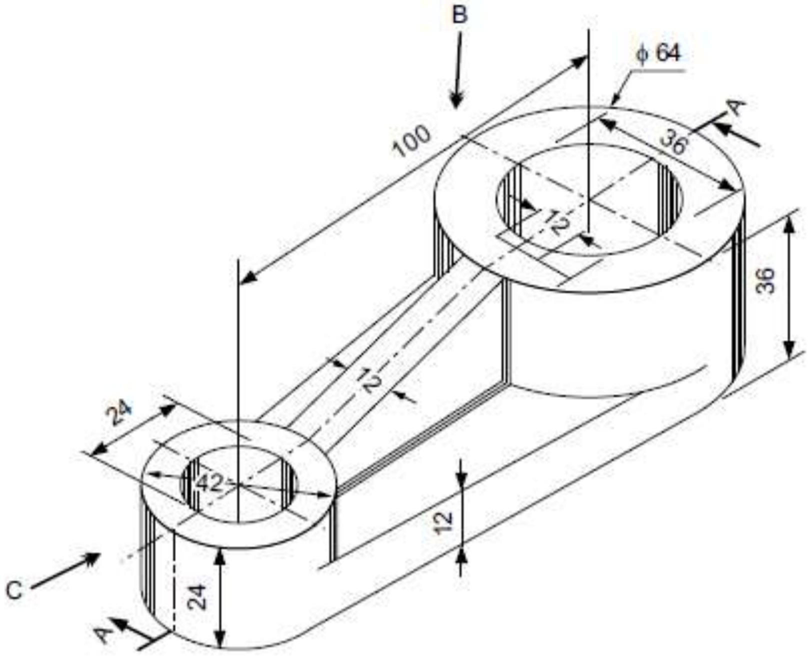
1-The Fig. shows the pictorial view of a machine block. Draw the following views to a suitable scale. (i) Full Sectional front view at B – B(ii) Top view (iii) Left hand side view.



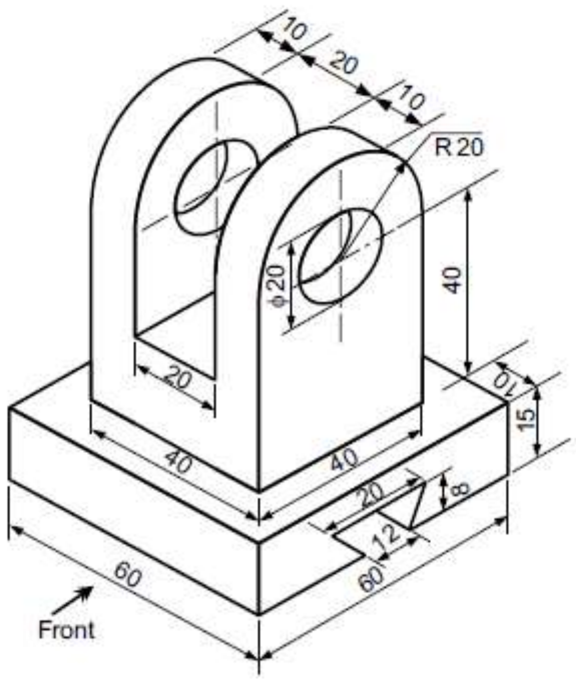
2- The Fig. shows the isometric view of a model. Draw to scale full size, the following views: (i) Front view looking in the direction of arrow – X
(ii) Sectional side view at – YY (iii) Top view.



3- The Fig.shows pictorial view of an object. Draw the following views in scale full size: (a) Front view through–A.A (b) Top view through–B (c) Left side view through–C.



4- The Fig. shows an isometric block of an object. Draw 1. Front view (Full Section) 2. Top view 3. Side view.



Chapter 8

Steel Structures

STEEL STRUCTURES

8.1 INTRODUCTION

When the need for a new structure arises, an individual or agency has to arrange the funds required for its construction. The individual or agency henceforth referred to as the owner then approaches an architect. The architect plans the layout so as to satisfy the functional requirements and also ensures that the structure is aesthetically pleasing and economically feasible. In this process, the architect often decides the material and type of construction as well. The plan is then given to a structural engineer who is expected to do locate the structural elements so as to cause least interference to the function and aesthetics of the structure. He then makes the strength calculations to ensure safety and serviceability of the structure. This process is known as structural design.

Finally, the structural elements are fabricated and erected by the contractor. If all the people work as a team then a safe, useful, aesthetic and economical structure is conceived. However in practice, many structures fulfill the requirements only partially because of inadequate coordination between the people involved and their lack of knowledge of the capabilities and limitations of their own and that of others. Since a structural engineer is central to this team, it is necessary for him to have adequate knowledge of the architects and contractors work. It is his responsibility to advise both the architect and the contractor about the possibilities of achieving good structures with economy.

Ever since steel began to be used in the construction of structures, it has made possible some of the grandest structures both in the past and also in the present day (The Hooghly cable stayed bridge, Jogighopa Road-cum-rail bridge across the river Brahmaputra). In the following paragraph, some of the aspects of steel structures, which every structural engineer should know, are briefly discussed.

Steel is by far the most useful material for building structures with strength of approximately ten times that of concrete, steel is the ideal material for modern construction. Due to its large strength to weight ratio, steel structures tend to be more economical than concrete structures for tall buildings and large span buildings and bridges. Steel structures can be constructed very fast and this enables the structure to be used early thereby leading to overall economy. Steel structures are ductile and robust and can withstand severe loadings such as earthquakes.

Steel structures can be easily repaired and retrofitted to carry higher loads. Steel is also a very eco-friendly material and steel structures can be easily dismantled and sold as scrap. Thus the lifecycle cost of steel structures, which includes the cost of construction, maintenance, repair and dismantling, can be less than that for concrete structures. Since steel is produced in the factory under better quality control, steel structures have higher reliability and safety. To get the most benefit out of steel, steel structures should be designed and protected to resist corrosion and fire. They should be designed and detailed for easy fabrication and erection. Good quality control is essential to ensure proper fitting of the various structural elements. The effects of temperature should be considered in design. To prevent development of cracks under fatigue and earthquake loads the connections and in particular the welds should be designed and detailed properly. Special steels and protective measures for corrosion and fire are available and the designer should be familiar with the options available.

8.2 NOTES ON STEEL MATERIAL

Steel is a term given to alloys containing a high proportion of iron with some carbon. Other alloying elements may also be present in varying proportions. The properties of steel are highly dependent on the proportions of alloying elements, so that their levels are closely controlled during its manufacture. The properties of steel also depend on the heat treatment of the metal.

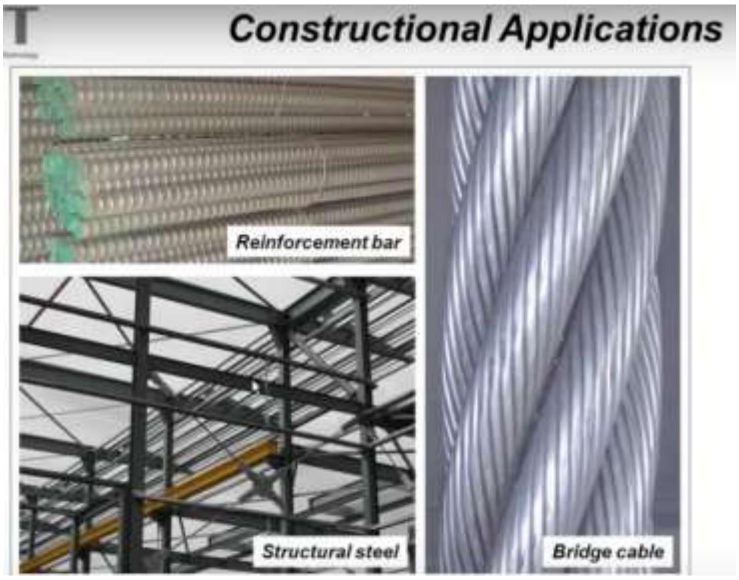
Steel is by far the most important metal, in tonnage terms, in the modern world, with the annual global production of over 700 million tonnes dwarfing the approximately 17 million tonnes of the next most prolific, aluminium. The low price and high strength of steel means that it is used structurally in many buildings and as sheet steel it is the major component of motor vehicles and domestic appliances. The major disadvantage of steel is that it will oxidize under moist conditions to form rust. Typical steel would have a density of about 7.7 g cm^{-3} and a melting point of about 1650^{oC}.

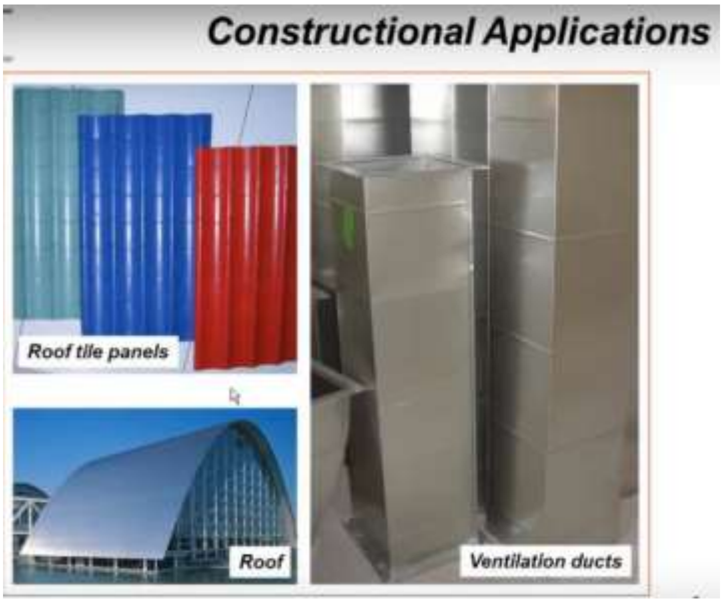
8.3 ADVANTAGES OF STEEL DESIGN

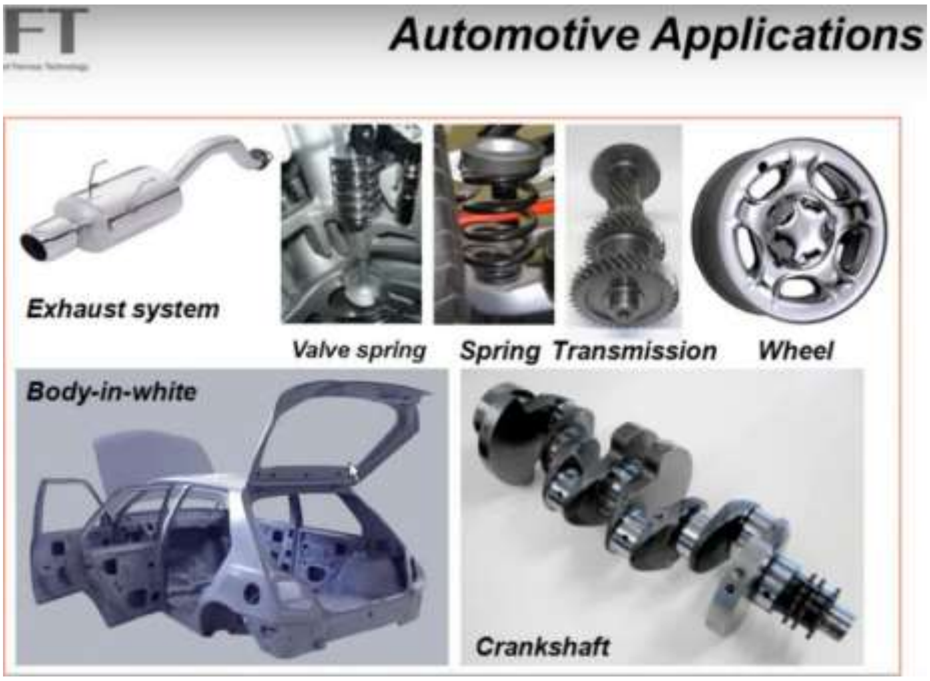
1. Better quality control
2. Lighter
3. Faster to erect
4. Reduced site time – Fast track construction
5. Large column free space and amenable for alteration
6. Less material handling at site
7. Less percentage of floor area occupied by structural elements
8. Has better ductility and hence superior lateral load behavior, better earthquake performance

8.4 DISADVANTAGES OF STEEL DESIGN

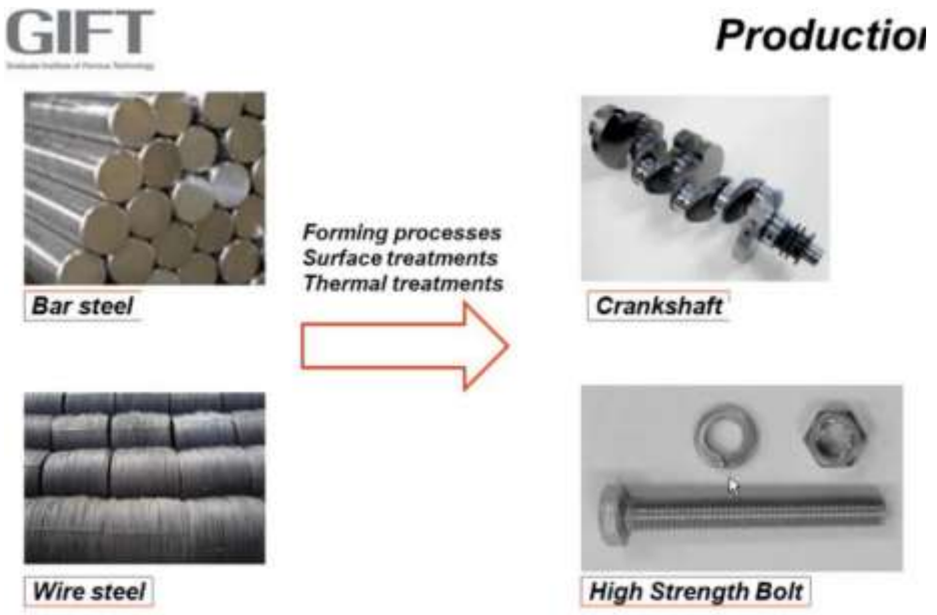
1. Skilled labor is required
2. Higher cost of construction
3. Maintenance cost is high (Due to corrosion)
4. Poor fire proofing as at 1000oF (538^{oC}) 65% and at 1600oF ($871\ ^{oC}$) 15% of strength remains
5. Electricity may be required (to hold joints, etc.)

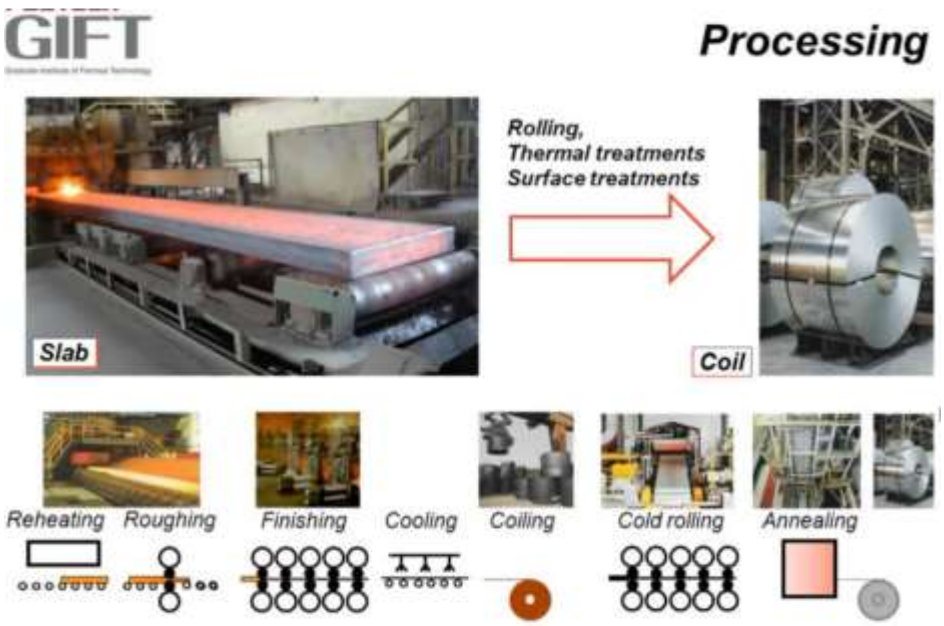





Automotive Applications
Exhaust system
Valve spring
Spring
Transmission
Wheel
Body-in-white
Crankshaft


Oil Industry, Shipbuilding
Ship building
Offshore construction





8.5 ANATOMY OF A STEEL STRUCTURE

Q. What is the anatomy of a steel structure?

Ans.

Beams

Columns

Floors

Bracing systems-- which is very important for higher rise cases

Foundation

Connections

So these are the anatomy of a steel building. (Anatomy means usually the study or an examination of what something is like, the way it works or why it works)

8.6 TYPES OF STRUCTURAL STEEL

Now let us discuss some rolled steel sections

8.6.1 ROLLED STEEL SECTIONS

The steel sections manufactured in rolling mills and used as structural members are known as rolled structural steel sections. The steel sections are named according to their cross sectional shapes. The shapes of sections selected depend on the types of members which are fabricated and to some extent on the process of erection. Many steel sections are readily available in the market and have frequent demand. Such steel sections are known as regular steel sections. Some steel sections are rarely used. Such sections are produced on special requisition and are known as special sections. „SP 6-1 (1964) ISI Handbook for Structural Engineers -Part- 1 Structural Steel Sections gives nominal dimensions, weight and geometrical properties of various rolled structural steel sections.

For Example:

TYPES OF ROLLED STRUCTURAL STEEL SECTIONS

The various types of rolled structural steel sections manufactured and used as structural members are as follows:

1. Rolled Steel I-sections (Beam sections).

2. Rolled Steel Channel Sections.

3. Rolled Steel Tee Sections.

4. Rolled Steel Angles Sections.

5. Rolled Steel Bars.

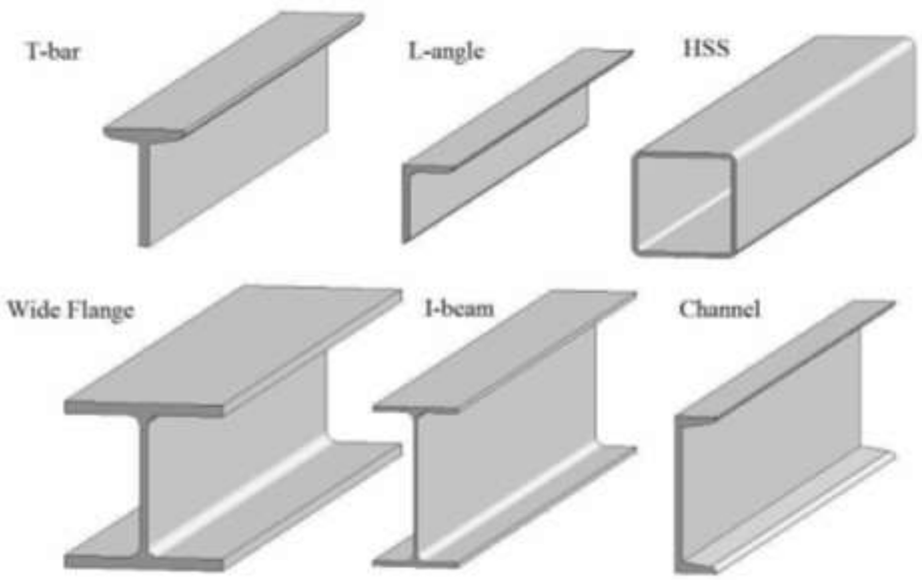
6. Rolled Steel Tubes.

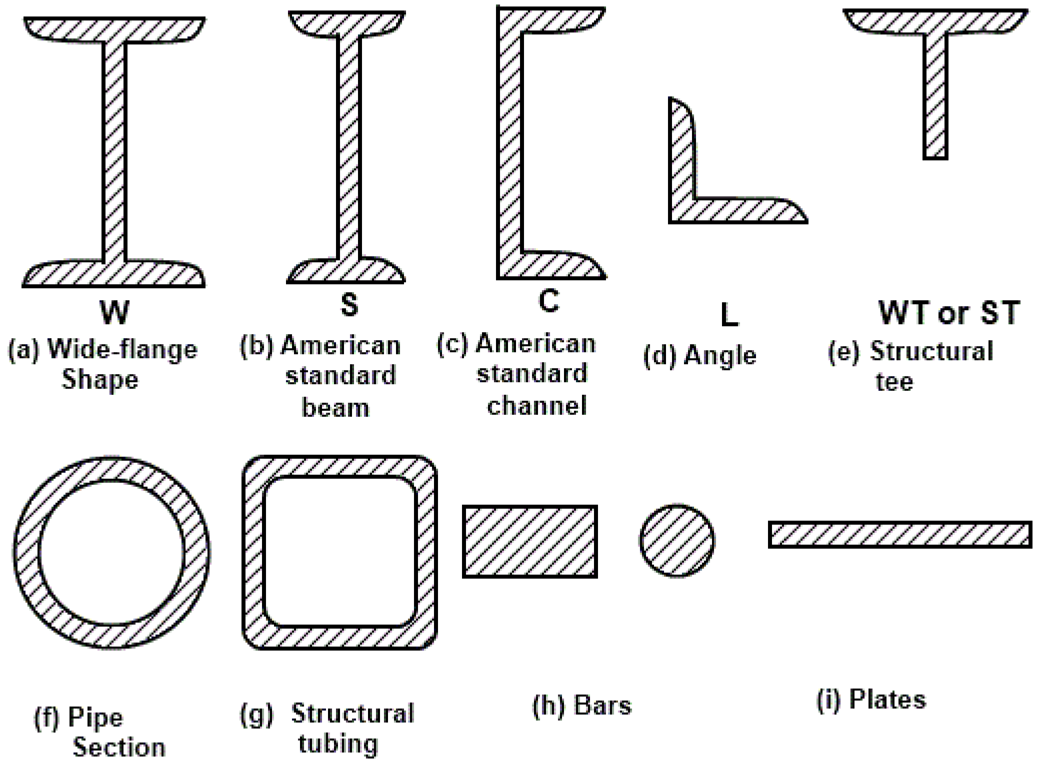
7. Rolled Steel Flats.

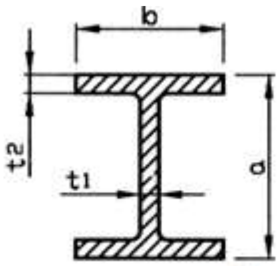
8. Rolled Steel Sheets and Strips.

9. Rolled Steel Plates.

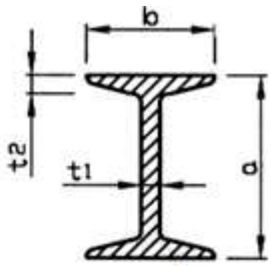
.



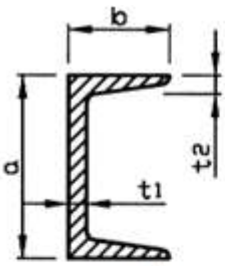




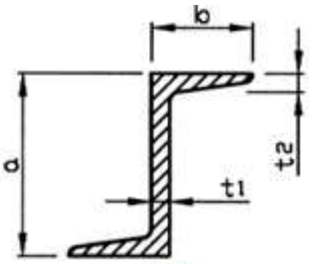
B.F.I.(WF) $\frac{a \times b}{t_1 \times t_2}$



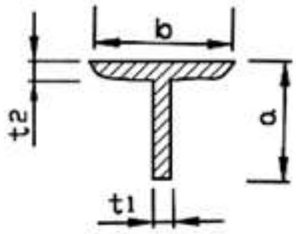
S.I. $\frac{a \times b}{t_1 \times t_2}$



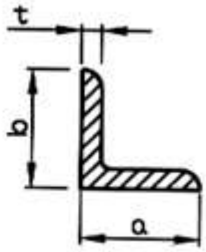
C $\frac{a \times b}{t_1 \times t_2}$



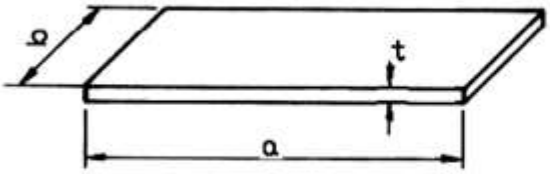
Z $\frac{a \times b}{t_1 \times t_2}$



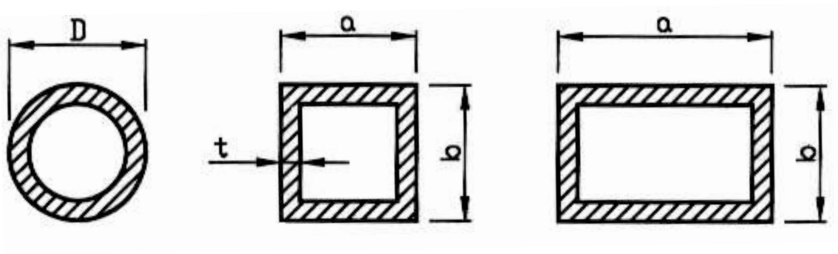
$$T\,\frac{a \times b}{t_1 \times t_2}$$

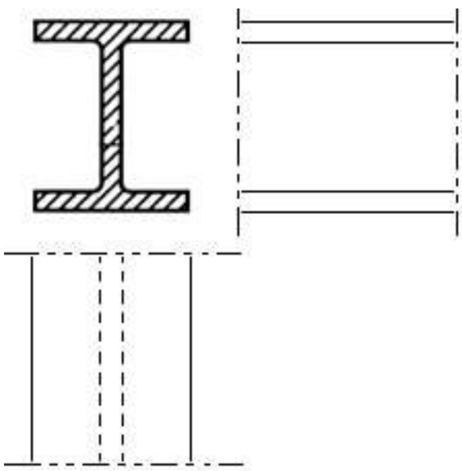


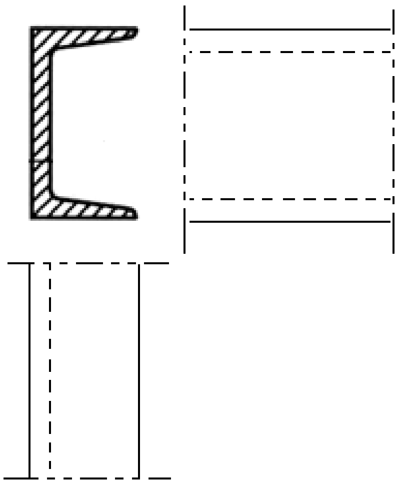
$$L\,\frac{a \times b}{t_1 \times t_2}$$

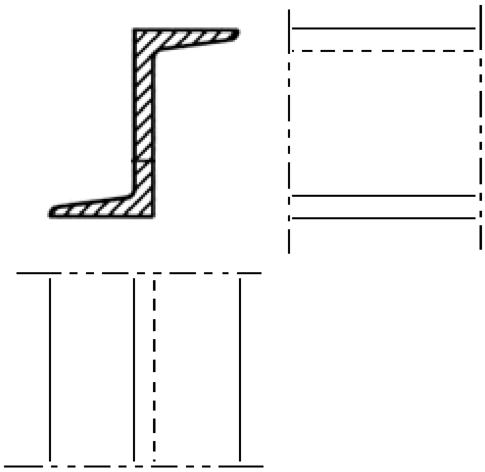


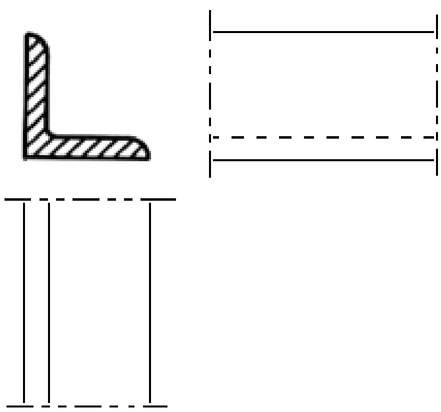
$$PL\ a \times b \times t$$

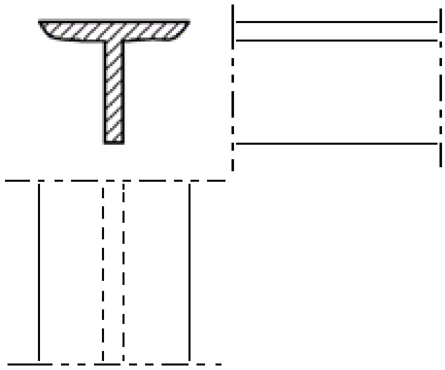


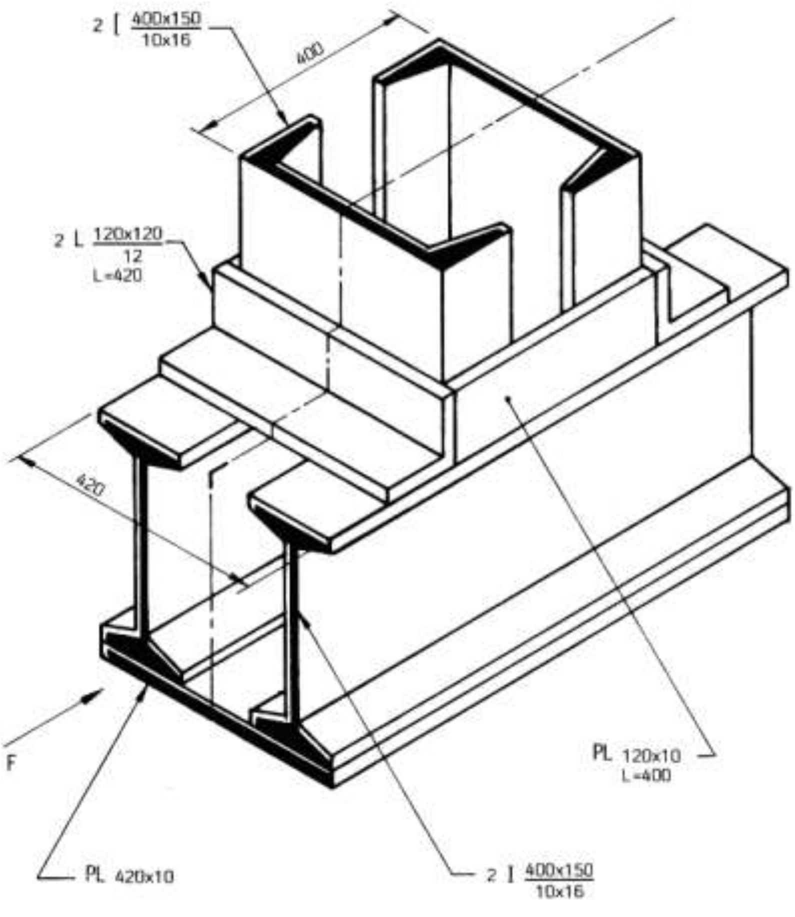


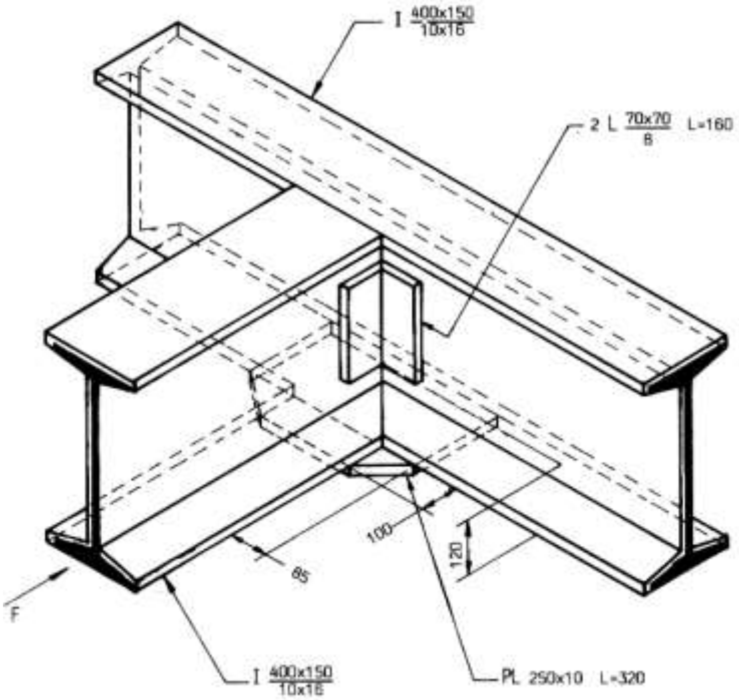




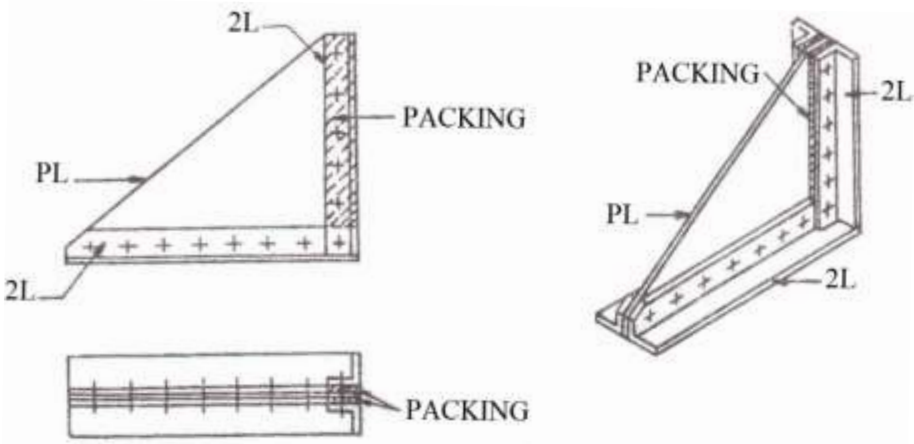


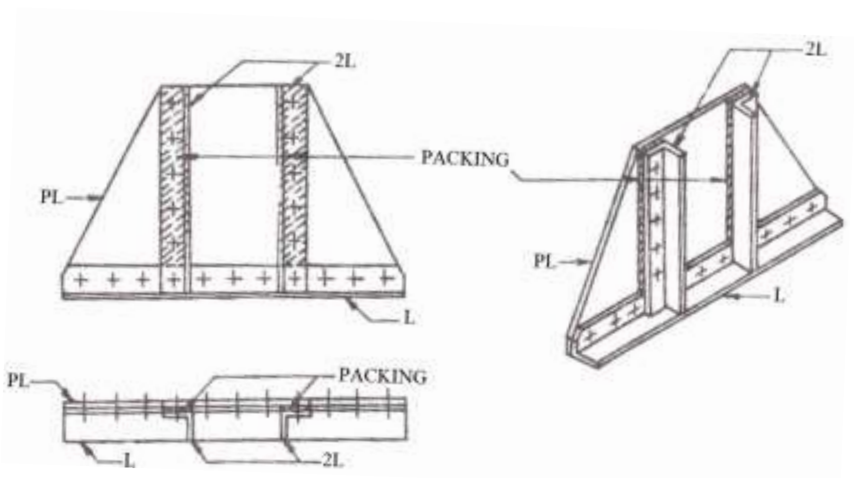



2 [400x150 / 10x16
400
2 L 120x120 / 12 / L=420
420
F
PL 120x10 / L=400
PL 420x10
2 I 400x150 / 10x16


I 400x150/10x16
2 L 70x70/8 L=160
100
120
85
F
I 400x150/10x16
PL 250x10 L=320

Packing


2L
PACKING
PL
2L
PACKING
PACKING
2L
PL
2L


2L
PACKING
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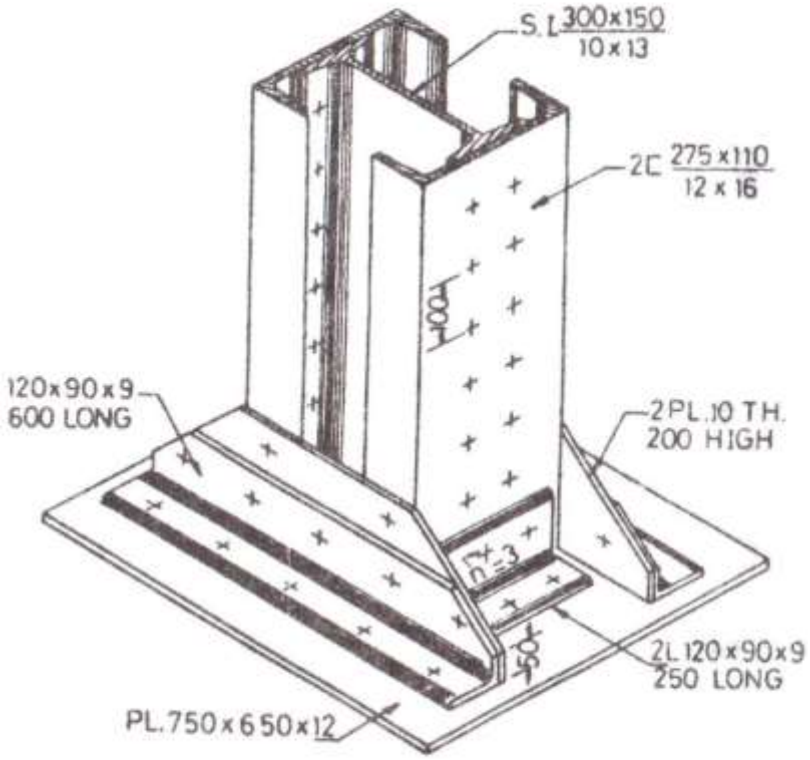
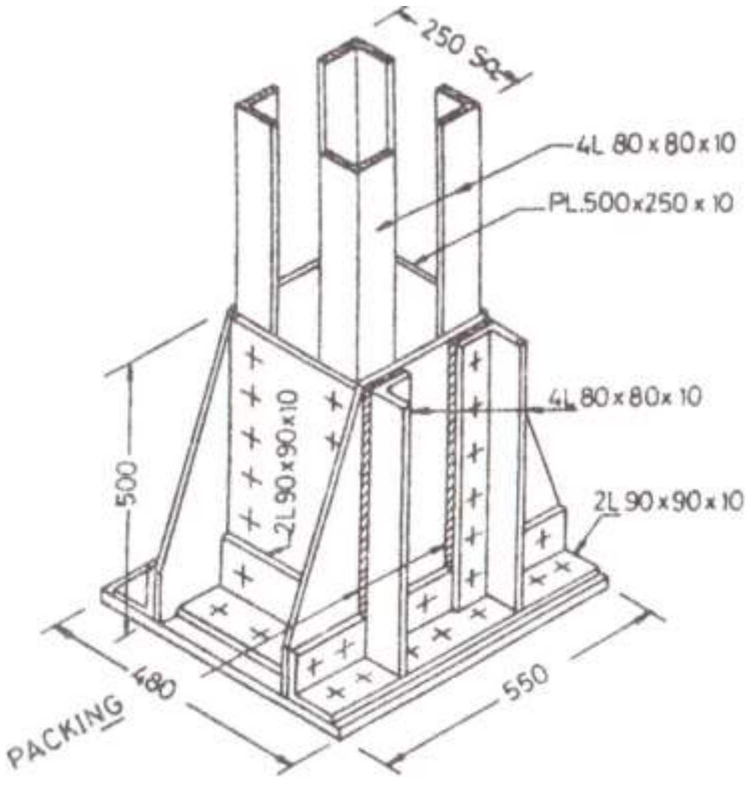
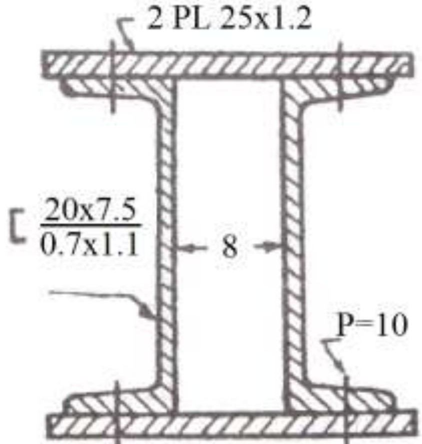
L
PL
PACKING
L
2L
2L
PACKING
PL
L

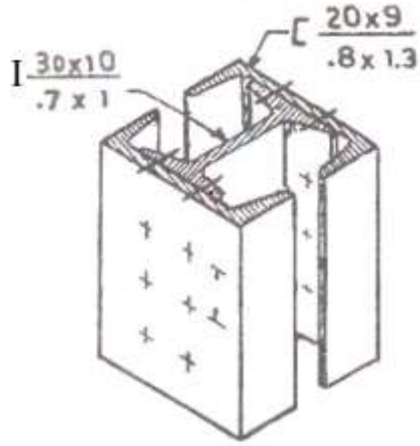

S.L 300x150 / 10x13
2C 275x110 / 12x16
100
120x90x9
600 LONG
2PL.10 TH.
200 HIGH
n=3
50
2L 120x90x9
250 LONG
PL.750x650x12

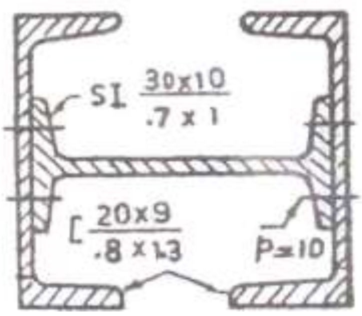

250 Sq.
4L 80x80x10
PL.500x250x10
4L 80x80x10
2L 90x90x10
500
2L 90x90x10
480
550
PACKING

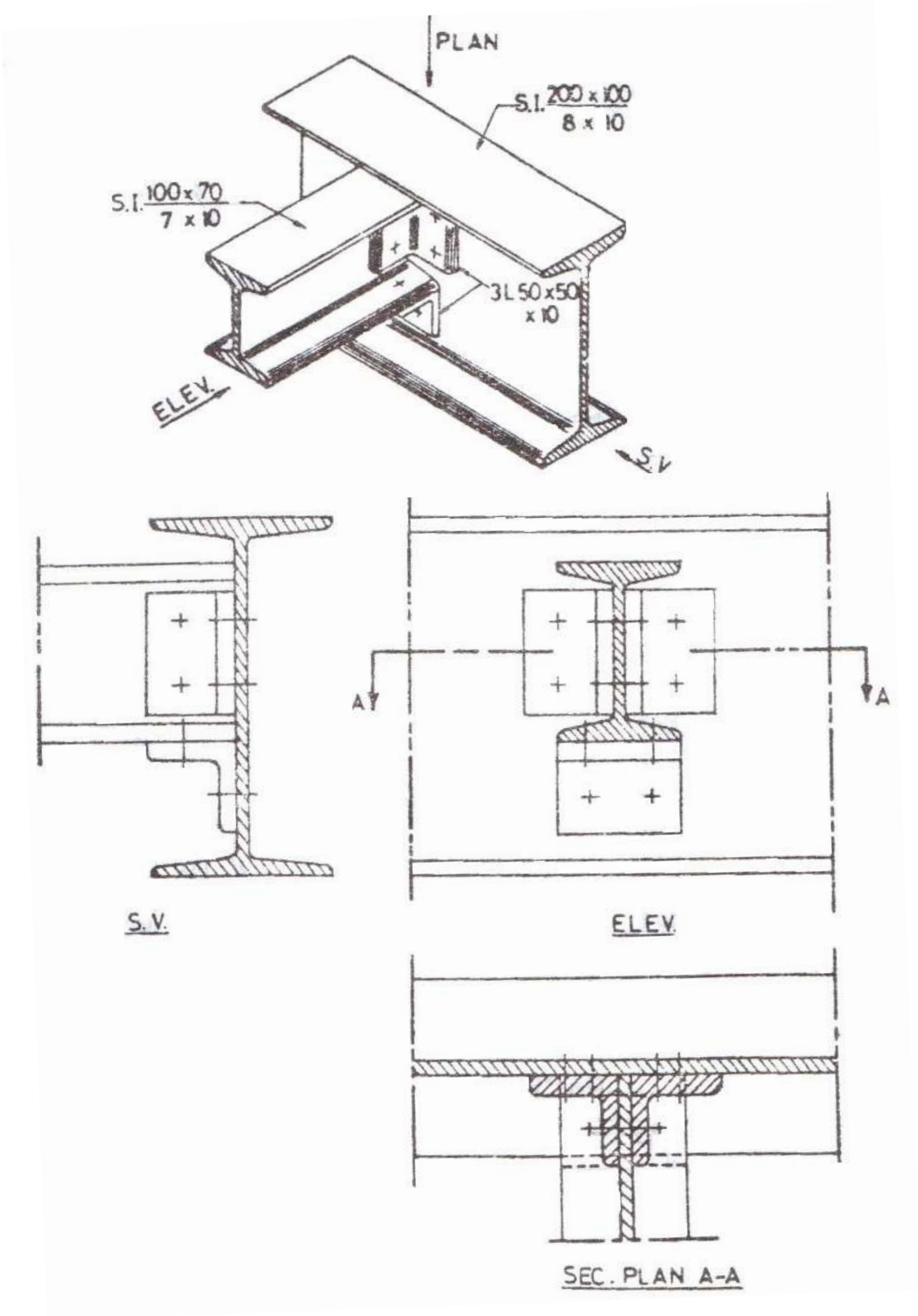

PL 25x1.2


2 PL 25x1.2
[20x7.5 / 0.7x1.1
8
P=10

Side view


I 30x10 / .7 x 1
[20x9 / .8 x 1.3


SI 30x10 / .7 x 1
[20x9 / .8 x 1.3
P=10


PLAN
S.I. 200 x 100
8 x 10
S.I. 100 x 70
7 x 10
3L 50 x 50
x 10
ELEV.
S.V
A
A
S.V.
ELEV.
SEC. PLAN A-A

Chapter 9

An Introduction to AutoCAD 2D

Dr. Huda Mohamed Al-Baz

9.1 Introduction to AutoCAD

Welcome to AutoCAD's tutorial. With this suite of tools, you will be able to produce high quality designs in less time, via the significant improvements in precision and flexibility while working in both 2D sketches and 3D modeling.

Fig. (1)

9.1.1 AutoCAD vs. AutoCAD LT

AutoCAD and its lighter, more streamlined version, AutoCAD LT, are both leading design and engineering software programs. Both offer 2D drafting and documentation along with dozens of design, connectivity and customization features. The primary difference between the two versions is that AutoCAD capabilities include 3D modeling and collaboration tools, along with several other special features.

For those unsure which product is right for them, a complete **AutoCAD vs AutoCAD LT** comparison is available on the Autodesk website. Or, you can try AutoCAD and **AutoCAD LT** for free, if you'd like to get hands-on information about which product will work best for your projects.

9.2 User Interface Tour

The AutoCAD user interface was designed to be intuitive and user-friendly. Even if you are at the beginner's level for AutoCAD, you should still be able to easily navigate through the software. As shown Fig (2) .

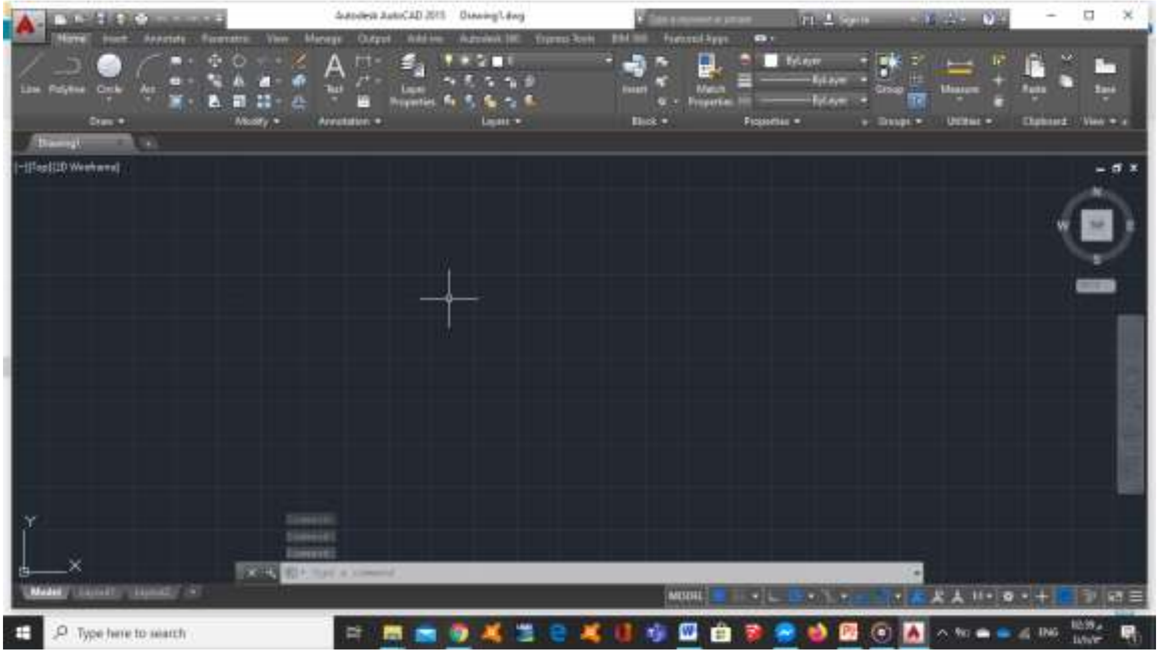


Fig. (2)

Upon first opening the program, you can either start drawing from a template of your choice, or open a recent document. If there is any product update information (like the opportunity to get the current version of software), you'll see it available in the notifications button. Lastly, you can also log into A360 Connect from this screen.

The drawing area is where you can create and modify objects. This area allows for lots of flexibility with view cubes, navigation bars, and much more. From this view, you have many other options to aid in the creative process.

Viewport controls allow you to adjust both visual styles and views directly on the viewport canvas.

Through the navigation bar, you'll find frequently used tools such as steering wheels, view cube, and show motion, as well as pan, zoom, and orbit.

Planar segments can now be detected in point clouds using dynamic UCS tools, allowing you to draw geometry directly on a point cloud.

The AutoCAD ribbon gallery offers a fast and intuitive workflow while the quick access toolbar lets you easily reach the tools you most commonly use.

Command preview helps you reduce the number of Undo commands you make by letting you evaluate the potential changes of commands such as Offset, Fillet, and Trim. When you are ready to see your creation in 3D, use the workspace control buttons to get to a 3D workspace.

Customization of the user interface is done by modifying the XML-based customization (CUIx) file with the Customize User Interface (CUI) Editor. After a command is defined, it can be used to create a button or a response to an action in the drawing area. **Default shortcut keys** are available for your reference on the Autodesk Knowledge Network.

You can edit customization (CUIx) files to make the following changes:

Add, delete, or change user interface elements, such as ribbon tabs and panels, and Quick Access toolbars:

- Create or change workspaces
- Create or change macros and tooltips for commands
- Create or change double-click actions
- Create or change keyboard shortcuts and temporary overrides
- Control the properties displayed when using rollover tooltips and the Quick Properties palette
- Create and load new CUIx files

9.3 Guide to AutoCAD Basics

In this section, you'll find a comprehensive technical AutoCAD tutorial which includes all the basic commands you will need when creating 2D drawings with AutoCAD or AutoCAD LT.

If you have just completed your AutoCAD training, or if you are comfortable with AutoCAD basics but would like a refresher, this is a great place to start. The included commands are grouped together according to types of activity, and are arranged to follow a general workflow. The following sections are covered:

plotter, or a file. Save and restore the printer settings for each layout.

For questions, the **product discussion** group is a great resource, as is the **AutoCAD blog**.

Basics: This section reviews the basic AutoCAD controls.

Viewing: Pan and zoom in a drawing, and control the order of overlapping objects.

Geometry: Create basic geometric objects such as lines, circles, and hatched areas.

Precision: Ensure the precision required for your models.

Layers: Organize your drawing by assigning objects to layers.

Properties: You can assign properties such as color and linetype to individual objects, or as default properties assigned to layers.

Modifying: Perform editing operations such as erase, move, and trim on the objects in a drawing.

Blocks: Insert symbols and details into your drawings from commercial online sources or from your own designs.

Layouts: Display one or more scaled views of your design on a standard-size drawing sheet called a layout.

Notes and Labels: Create notes, labels, bubbles, and callouts. Save and restore style settings by name.

Dimensions: Create several types of dimensions and save dimension settings by name. Printing:

Output a drawing layout to a printer, a plotter, or a file. Save and restore the printer settings for each layout.

For questions, the **product discussion** group is a great resource, as is the **AutoCAD blog**.

Basics

Review the basic AutoCAD controls.

After you launch AutoCAD, click the Start Drawing button to begin a new drawing. As shown Fig (3) .

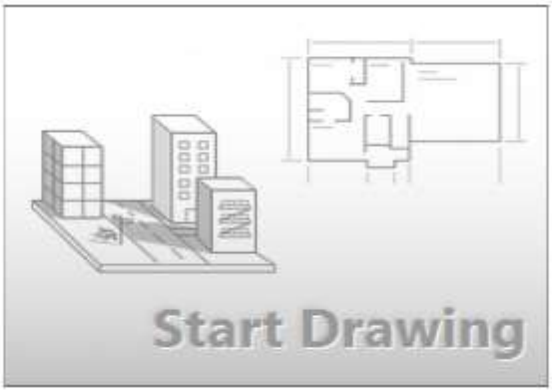


Fig. (3)

AutoCAD includes a standard tabbed ribbon across the top of the drawing area. You can access nearly all the commands presented in this guide from the **Home** tab. In addition, the Quick Access toolbar shown below includes familiar commands such as New, Open, Save, Print, Undo, and so on. As shown Fig (4)

.

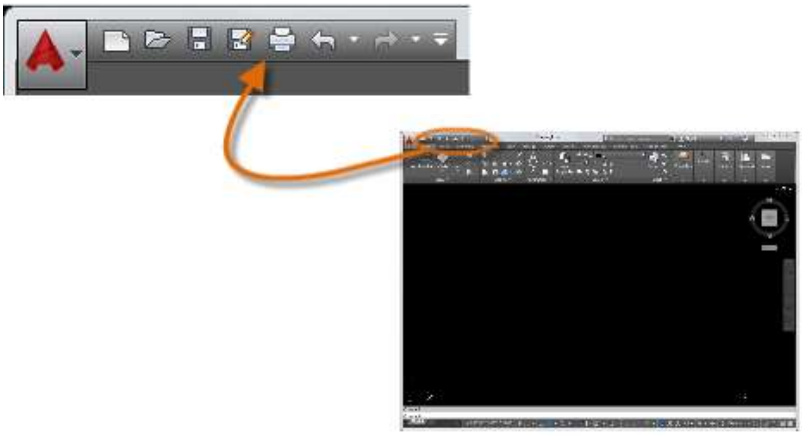


Fig. (4)

NOTE: *If the Home tab is not the current tab, go ahead and click it.*

The Command Window

At the heart of AutoCAD is the Command window, which is normally docked at the bottom of the application window. The Command window displays prompts, options, and messages. As shown Fig (5) .

Fig. (5)

You can enter commands directly in the Command window instead of using the ribbon, toolbars, and menus. Many long-time AutoCAD users prefer this method.

Notice that as you start to type a command, an autocomplete menu appears. When several options are available, such as in the example below, make your choice by clicking the correct option or using the arrow keys and then pressing Enter or the Spacebar to confirm your selection. As shown Fig (6) **.**

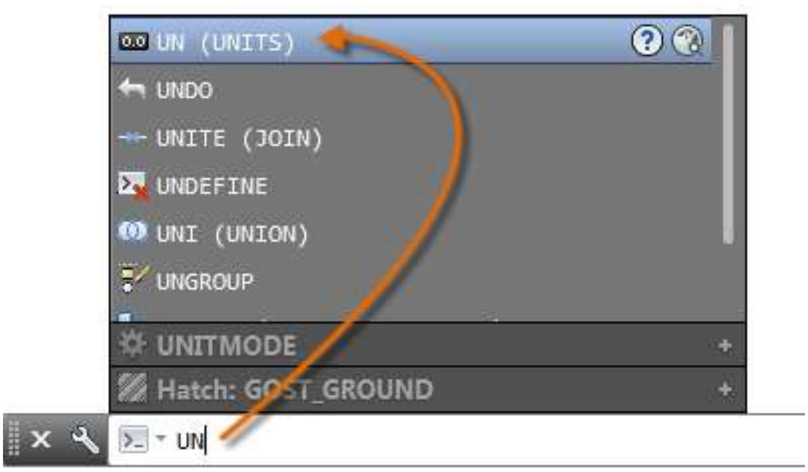


Fig.(6)

The Mouse

Most people use a mouse as their pointing device, but other devices have equivalent controls. As shown Fig (7) .

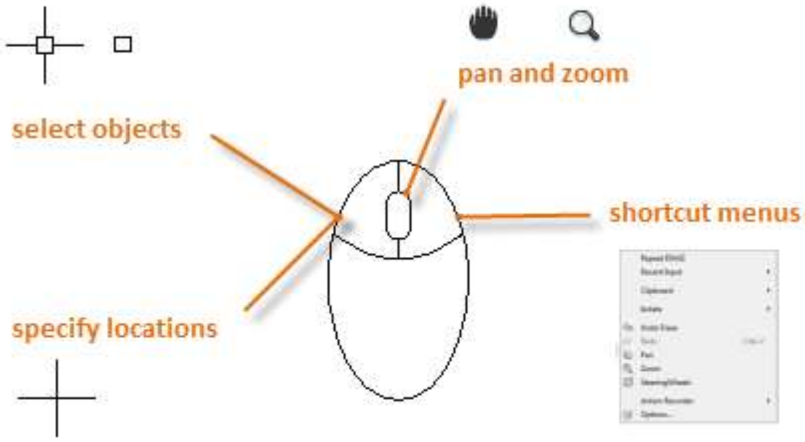


Fig. (7)

Here's a Tip:

When looking for a command or option, try right-clicking. Depending on where your cursor is located, different menus will display relevant commands and options.

New Drawings

You can easily conform to industry or company standards by specifying settings for text, dimensions, linetypes, and several other features. For example, this backyard deck design displays two different dimension styles. As shown Fig (8) .

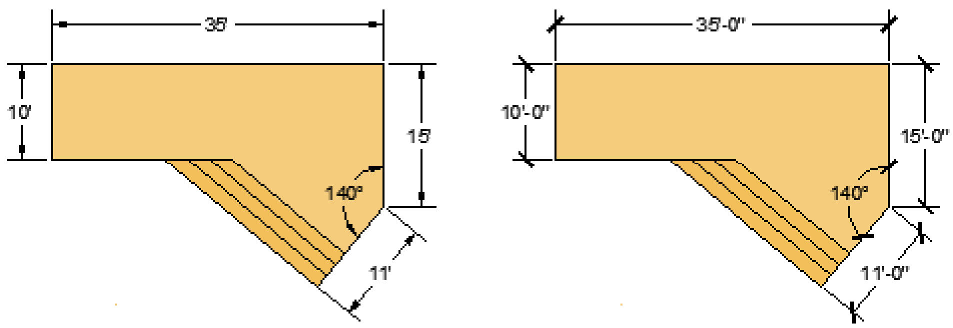


Fig. (8)

All these settings can be saved in a *drawing template* file. Click New to choose from several drawing template files: As shown Fig (9) .

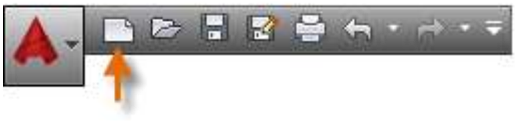


Fig. (9)

For imperial drawings that assume your units are inches, use *acad.dwt* or *acadlt.dwt*.

For metric units that assume your units are millimeters, use *acadiso.dwt* or *acadltiso.dwt*. As shown in Fig (10) .

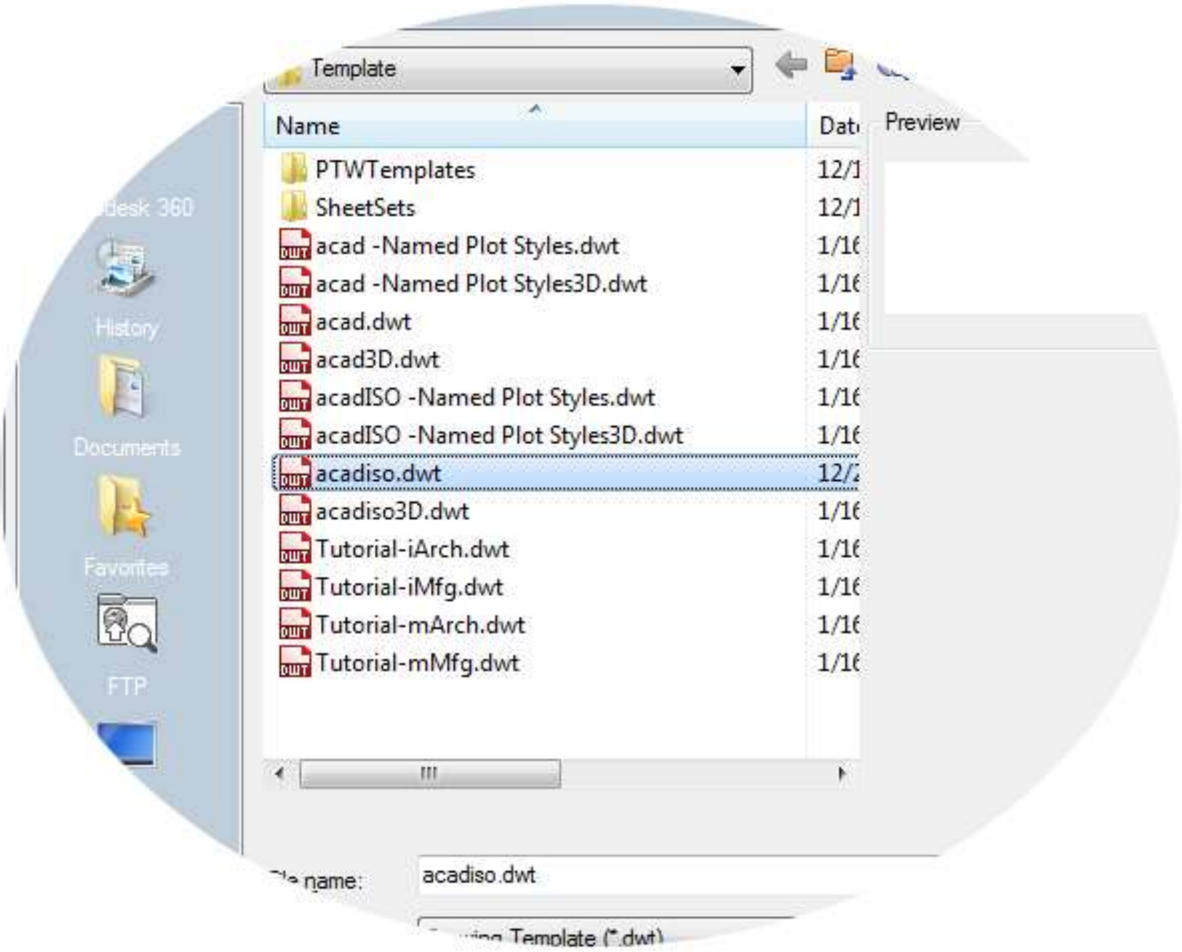


Fig. (10)

The "Tutorial" template files in the list are examples of the architectural or mechanical design templates using both imperial (i) and metric (m) measurements. You might want to experiment with them.

Most companies use drawing template files that conform to company standards, and they will often use different drawing template files depending on the project or client.

9.4 Create Your Own Drawing Template File

You can save any drawing (.*dwg*) file as a drawing template (.*dwt*) file. You can also open any existing drawing template file, modify it, and then save it again with a different filename if needed.

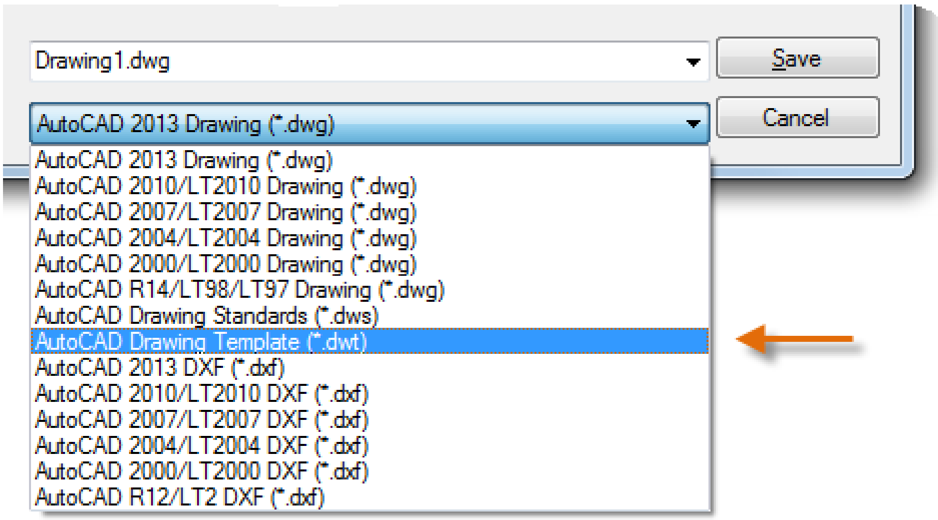


Fig. (10)

If you work independently, you can develop your drawing template files to suit your working preferences, adding settings for additional features as you become familiar with them.

To modify an existing drawing template file, click Open, specify Drawing Template (*.dwt) in the Select File dialog box, and choose the template file. As shown in Fig. (11) .

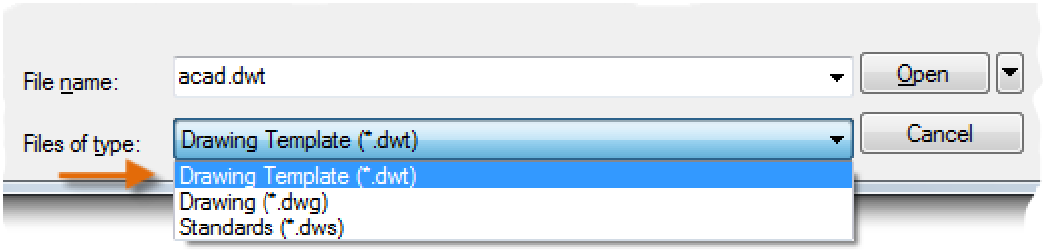


Fig. (11)

Important:

If your company has already established a set of drawing template files, check with your CAD manager before modifying any of them.

Units

When you first start a drawing, you need to decide what the length of one unit represents—an inch, a foot, a centimeter, a kilometer, or some other unit of length. For example, the objects below could represent two buildings that are each 125 feet long, or they could represent a section from a mechanical part that is measured in millimeters. As shown in Fig. (12) .

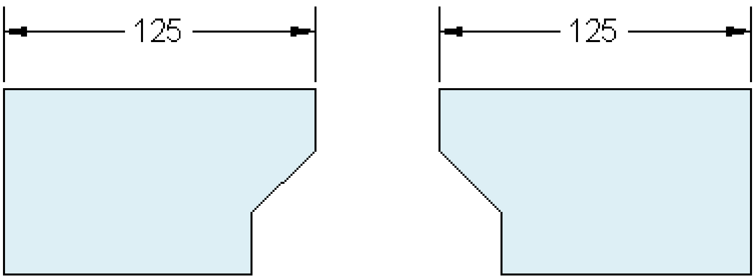


Fig. (12)

Unit Display Settings

After you decide what unit of length that you want to use, the **UNITS** command lets you control several unit display settings including the following:

Format (or Type): For example, a decimal length of 6.5 can be set to display as a fractional length of 6-1/2 instead.

Precision: For example, a decimal length of 6.5 can be set to display as 6.50, 6.500, or 6.5000.

If you plan to work in feet and inches, use the **UNITS** command to set the unit type to Architectural, and then when you create objects, specify their lengths in inches. If you plan to use metric units, leave the unit type set to Decimal. Changing the unit format and precision does not affect the internal precision of your drawing, it affects only how lengths, angles, and coordinates are displayed in the user interface.

Here's a Tip:

If you need to change the **UNITS** settings, make sure that you save the drawing as a drawing template file (.dwt). Otherwise, you will need to change the **UNITS** settings for each new drawing.

Model Scale

Always create your models at full size (1:1 scale). The term *model* refers to the geometry of your design. A *drawing* includes the model geometry along with the views, notes, dimensions, callouts, tables, and the title block displayed in the *layout*.

You can specify the scaling that is necessary to print a drawing on a standard-sized sheet later, when you create the layout.

Recommendations

- To open **Help** for information about the command in progress, press F1.
- To **repeat** the previous command, press Enter or the Spacebar.
- To see various **options**, select an object and right-click or right-click a user interface element.
- To **cancel** a command in progress or if you ever feel stuck, press Esc. For example, if you click in the drawing area before entering a command, you will see something like the following: As shown in Fig (13) .

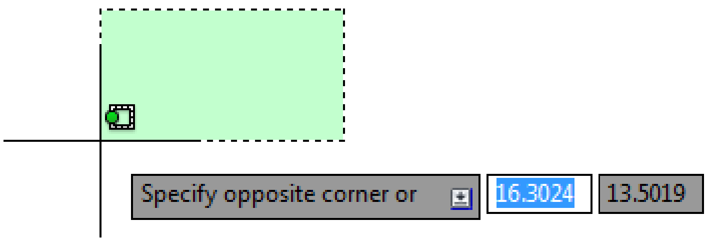


Fig. (13)

Here's a Tip:

Press Esc to cancel this preselection operation.

Viewing

Zoom in on a drawing to better control the order of overlapping objects.

The easiest way to change your view is by using the mouse wheel.

Zoom in or out by rolling the wheel.

Pan a view in any direction by holding the wheel down while moving your mouse.

Zoom in on a specific area for greater detail holding your mouse over the area and clicking the wheel twice. As shown in Fig (14) .

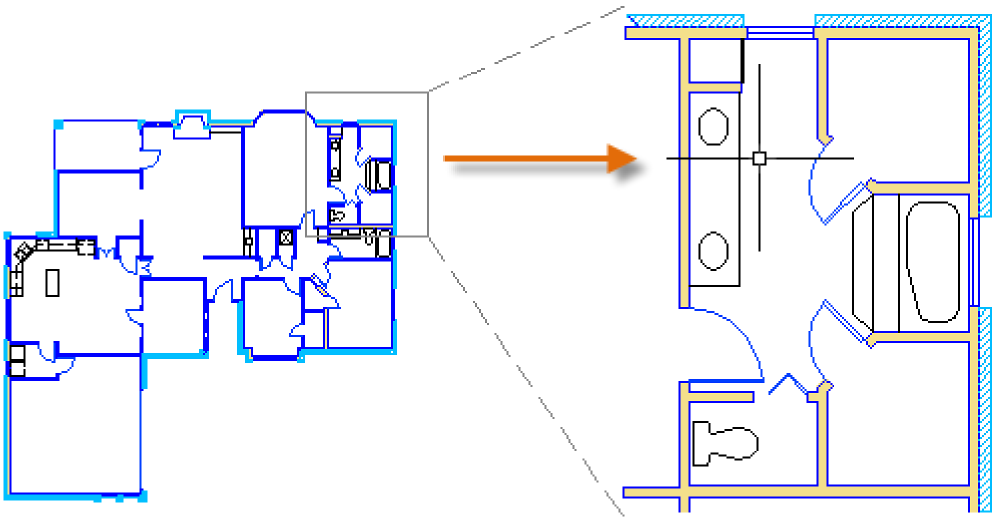


Fig. (14)

9.5 GUIDE TO AUTOCAD BASICS: VIEWING

Here's a Tip:

When you zoom in or out, the location of the cursor is important. Think of your cursor as a magnifying glass. For example, if you position the cursor in the upper-right area of the floor plan as shown below, zooming in magnifies the dressing room without shifting the view.

NOTE: *If you cannot zoom or pan any more, type* ***REGEN*** *in the Command window and press Enter. This command regenerates the drawing display and resets the extents available for panning and zooming.*

Overlapping Objects

If you create objects that overlap, you might need to change which objects are displayed on top or in front of the others. For example, if you want the yellow highway to cross the blue river rather than the other way around, use the **DRAWORDER** command to reorder the objects. As shown in Fig (15) .

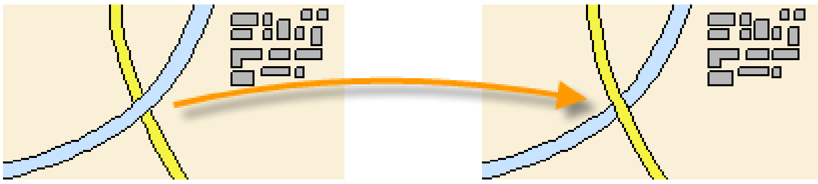


Fig. (15) .

You can access several draw order options from the Modify panel on the ribbon. Click to expand the Modify panel, and then click the down-arrow as shown below. As shown in Fig (16) .

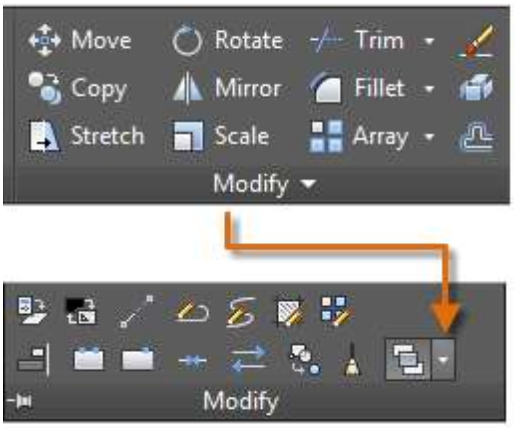


Fig. (16)

The draw order options that are listed include sending all hatches to the back, all text to the front, and so on. As shown in Fig (17) .

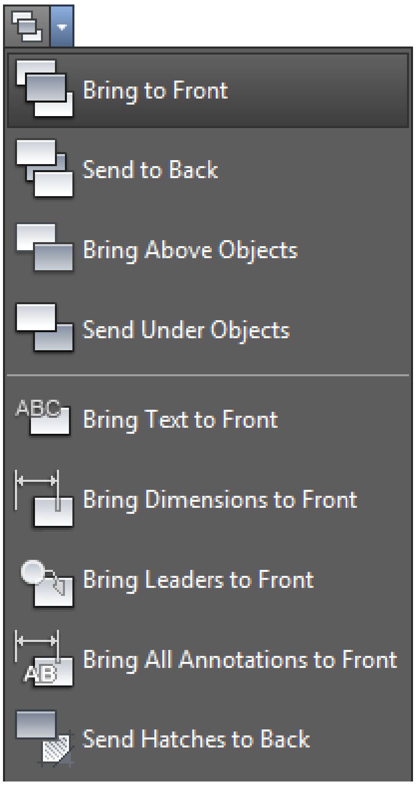


Fig. (17)

Geometry

Create basic geometric objects such as lines, circles, and hatched areas.

You can create many different types of geometric objects in AutoCAD, but you only need to know a few of them for most 2D drawings.

NOTE: *If you want to simplify the display while creating geometric objects, press F12 to turn off dynamic input.*

Lines

The line is the most basic and common object in AutoCAD drawings. To draw a line, click the Line tool. As shown in Fig (18) .

Fig. (18)

Alternatively, you can type **LINE** or just **L** in the Command window, and then press Enter or the Spacebar.

Notice the prompt in the Command window for a point location. As shown in Fig (10). As shown in Fig (19) .

Fig. (19)

To specify the starting point for this line, you would type in the coordinates 0,0. It is a good idea to locate one corner of your model at 0,0, which is called the origin point. To locate additional points, you could specify additional X,Y coordinate locations in the drawing area, however more efficient methods for specifying points are available, and will be presented in the Precision topic. As shown in Fig (20) .

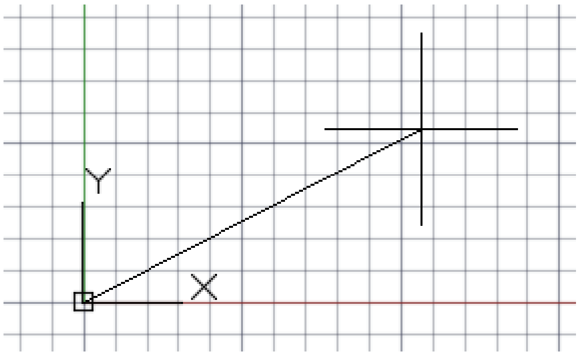


Fig. (20)

After you specify the next point, the **LINE** command automatically repeats itself, and it keeps prompting you for additional points. Press Enter or the Spacebar to end the sequence.

The User Coordinate System

The user coordinate system (UCS) icon indicates the direction of the positive X and Y axis for any coordinates that you enter, and it also defines the horizontal and vertical directions in a drawing. In some 2D drawings, it can be convenient to click, drag, and rotate the UCS to change the origin point, and the horizontal and vertical directions. As shown in Fig (21) .

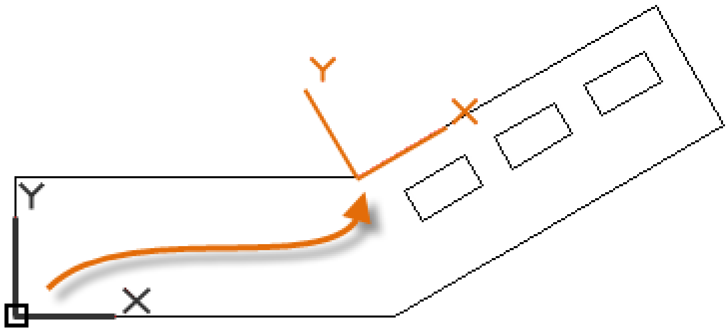


Fig. (21)

Grid Display

Some people like working with grid lines as a reference, while others prefer working in a blank area. To turn off the grid display, press F7. Even with the grid turned off, you can force your cursor to snap to grid increments by pressing F9.

Lines as Construction Aids

Lines can serve as reference and construction geometry such as:

- Property line setbacks
- The mirror line of a symmetrical mechanical part
- Clearance lines to avoid interferences
- Traversal path lines

Circles

The default option of the **CIRCLE** command requires you to specify a center point and a radius . As shown in Fig (22) .

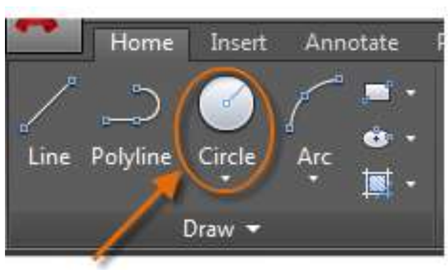


Fig. (22)

The other circle options are available from *the drop-down:* As shown in Fig (23) .

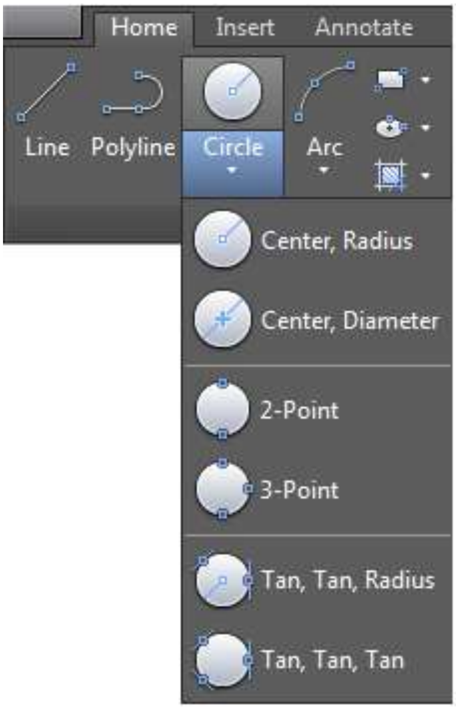


Fig. (23)

Alternatively, you can also enter **CIRCLE** or just **C** in the Command window and click to choose an option. If you do, you can specify a center point, or you can click one of the highlighted command options as shown below. As shown in Fig (24) .

Fig. (24)

Circles can be useful as reference geometry. For example, you can see that the two doors in the illustration can interfere with each other. As shown in Fig (25) .

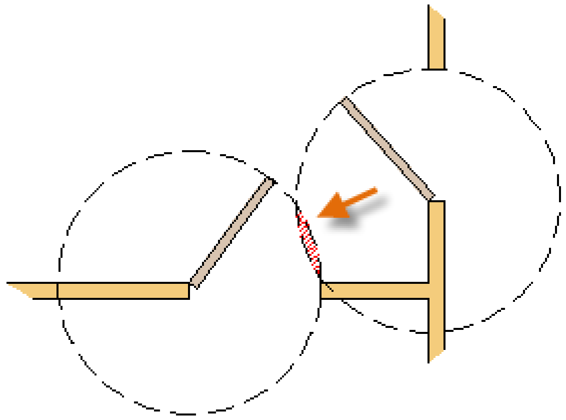


Fig. (25)

Polylines and Rectangles

A polyline is a connected sequence of line or arc segments that is created as a single object. As shown in Fig (26) .

Fig. (26)

Use the **PLINE** command to create open or closed polylines for:

- Geometry that requires fixed-width segments.
- Continuous paths for which you need to know the total length.
- Contour lines for topographic maps and isobaric data.
- Wiring diagrams and traces on printed circuit boards.
- Process and piping diagrams.

Polylines can have a constant width or they can have different starting and ending widths. After you specify the first point of the polyline, you can use the Width option to specify the width of all subsequently created segments. You can change the width value at any time, even as you create new segments. As shown in Fig (27) .

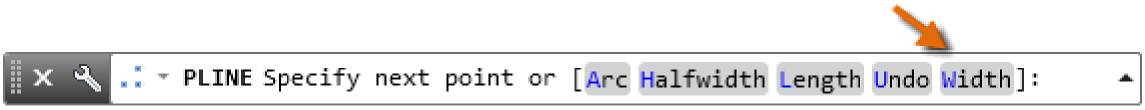


Fig. (27)

Here is an example of a printed circuit board in which the traces were created with wide polylines. The landing pads were created with the **DONUT** command. As shown in Fig (28) .

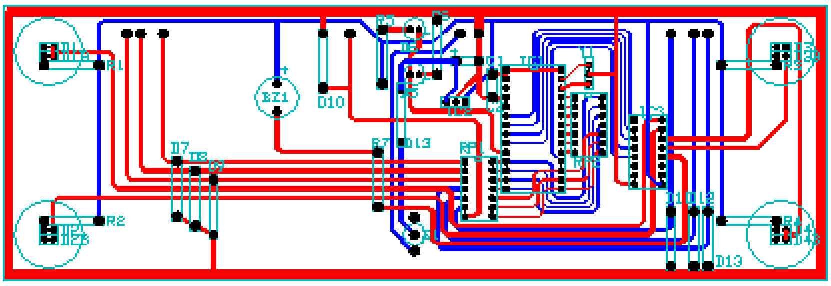


Fig. (28)

Polylines can have different starting and ending widths for each segment as shown here: As shown in Fig (29) .

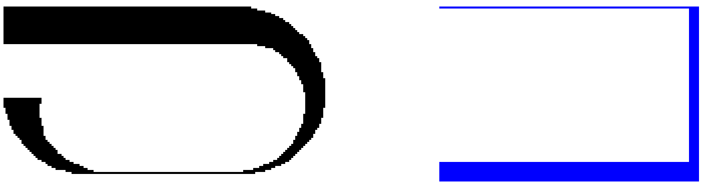


Fig. (29)

A fast way to create closed rectangular polylines is to use the **RECTANG** command (enter **REC** in the Command window). As shown in Fig (30) .

Fig. (30)

Simply click two diagonal points for the rectangle as illustrated. If you use this method, turn on grid snap (F9) for precision. As shown in Fig (31) .

Fig. (31)

Hatches and Fills

In AutoCAD, a hatch is a single, compound object that covers a specified area with a pattern of lines, dots, shapes, a solid fill color, or a gradient fill. As shown in Fig (32).

Fig. (32)

When you start the **HATCH** command, the ribbon temporarily displays the Hatch Creation tab. On this tab, you can choose from over 70 industry-standard imperial and ISO hatch patterns along with many specialized options.

The simplest procedure is to choose a hatch pattern and scale from the ribbon, and click within any area that is completely enclosed by objects. You must specify the scale factor for the hatch in order to control its size and spacing.
After you create a hatch, you can move the bounding objects to adjust the hatch area, or you can delete one or more of the bounding objects to create partially bounded hatches: As shown in Fig (33).

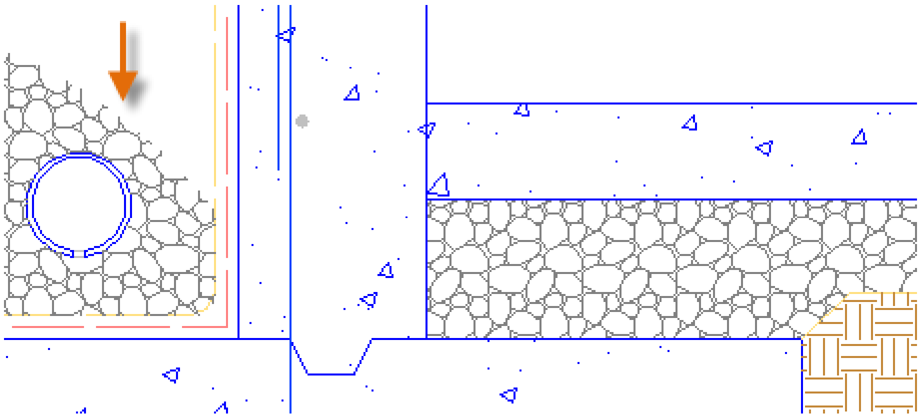


Fig. (33)

Here's a Tip:
If you set a solid or gradient fill hatch pattern, also consider setting a transparency level on the Hatch Creation tab for interesting overlap effects.

Here are some examples of how you can use solid-fill hatches:

As shown in Fig (34).

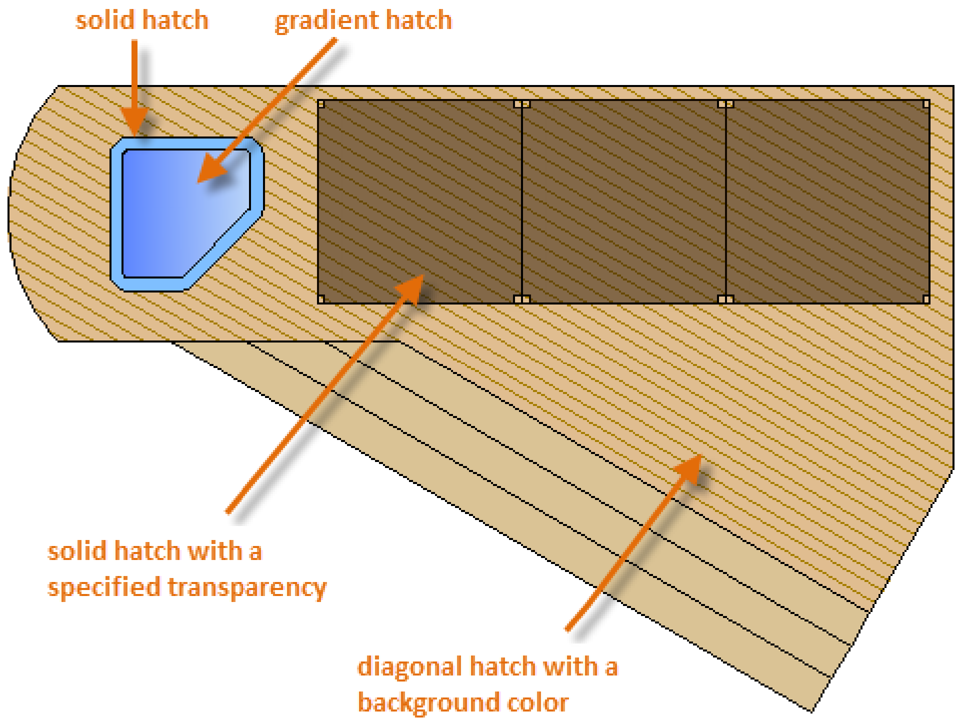


Fig. (34)

Here's a Tip:

If you need to align the pattern in a hatch, which might be the case with the decking boards above, use the Set Origin option to specify an alignment point. As shown in Fig (35).

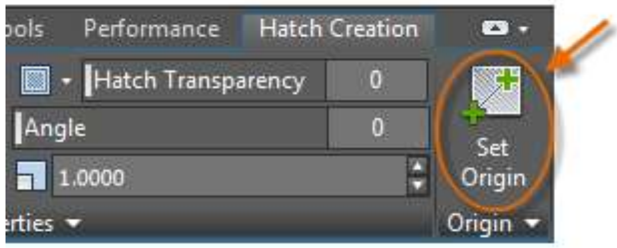


Fig. (35)

NOTE: *If an area is not completely enclosed, red circles appear to indicate potential gaps. Enter* ***REDRAW*** *in the Command window to dismiss the red circles.*

Precision

Ensure the precision required for your models.

There are several precision features available, *including:*

- **Polar tracking:** Snap to the closest preset angle and specify a distance along that angle.
- **Locking angles:** Lock to a single, specified angle and specify a distance along that angle.
- **Object snaps:** Snap to precise locations on existing objects, such as an endpoint of a polyline, the midpoint of a line, or the center point of a circle.
- **Grid snaps:** Snap to increments on a rectangular grid.
- **Coordinate entry:** Specify a location by its Cartesian or polar coordinates, either absolute or relative.

The three most commonly used features are polar tracking, locking angles, and object snaps.

Polar Tracking

When you need to specify a point, such as when you create a line, you can use polar tracking to guide the movement of your cursor in certain directions.

For example, after you specify the first point of the line below, move your cursor to the right, and then enter a distance in the Command window to specify a precise horizontal length for the line. As shown in Fig (36).

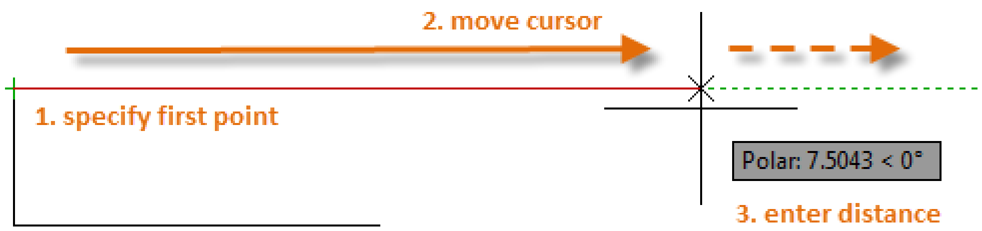


Fig. (36)

By default, polar tracking is turned on and guides your cursor in a horizontal or vertical direction (0 or 90 degrees).

Locking Angles

If you need to draw a line at a specified angle, you can lock the angle for the next point. For example, if the second point of a line needs to be created at a 45 degree angle, you would enter ‘<45’ in the Command window. As shown in Fig (37).

Fig. (37)

After you move your cursor in the desired direction along the 45-degree angle, you can enter the length of the line.

Object Snaps

By far, the most important way for you to specify precise locations on objects is to use object snaps. In the following illustration, several different kinds of object snaps are represented by markers. As shown in Fig (38).

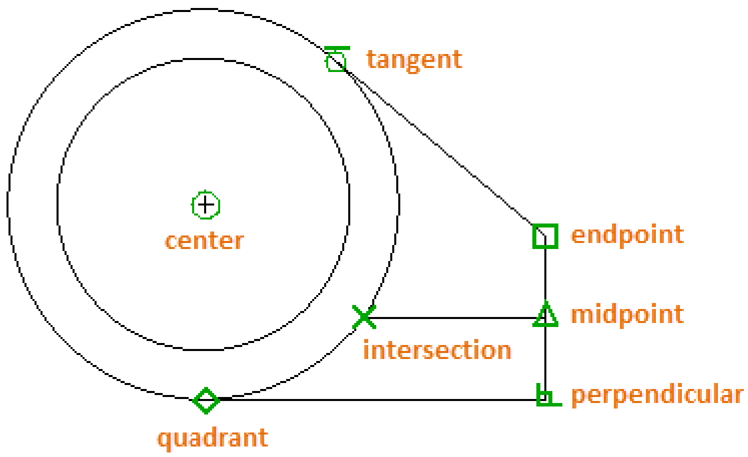


Fig. (38)

Object snaps become available during a command whenever AutoCAD prompts you to specify a point. For example, if you start a new line and move your cursor near the endpoint of an existing line, the cursor will automatically snap to it. As shown in Fig (39).

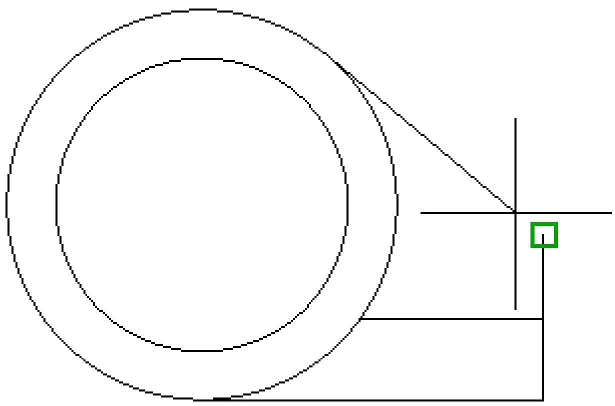


Fig. (39)

Set Default Object Snaps

Enter the **OSNAP** command to set the default object snaps, which are also called "running" object snaps. For example, you might find it useful to turn on the Midpoint object snap by default. As shown in Fig (40).

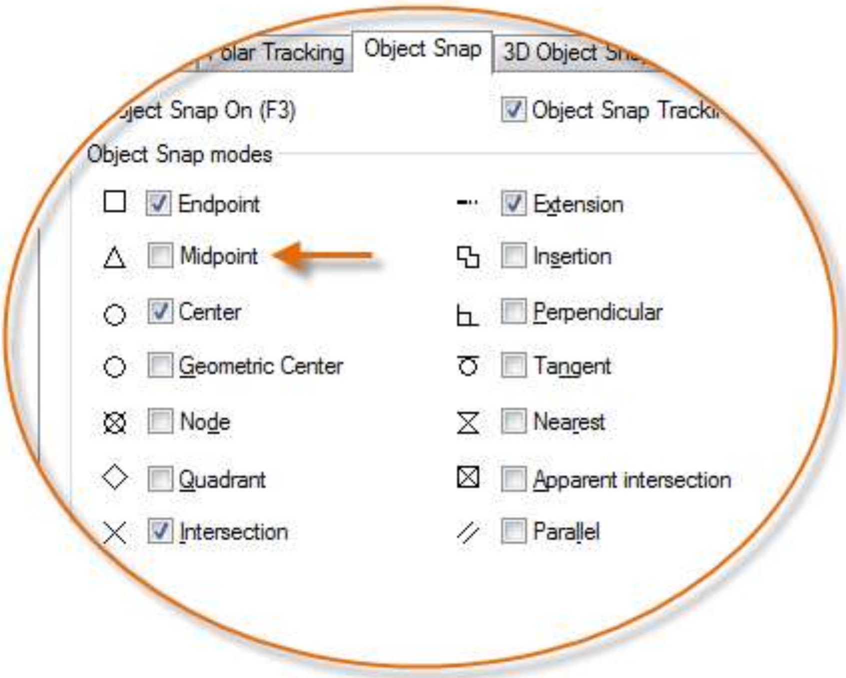


Fig. (40)

Recommendations

At any prompt for a point, you can specify a single object snap that overrides all other object snap settings. Hold down Shift, right-click in the drawing area, and choose an object snap from the Object Snap menu. Then move the cursor to select a location on an object.
Make sure that you zoom in close enough to avoid mistakes. In a densely populated model, snapping to the wrong object will result in an error that can propagate throughout your model.

Object Snap Tracking

During a command, you can align points both horizontally and vertically from object snap locations.

In the following illustration, you first hover over endpoint 1 and then hover over endpoint.

2. When you move your cursor near location

3, the cursor locks into the horizontal and vertical location shown. As shown in Fig (41).

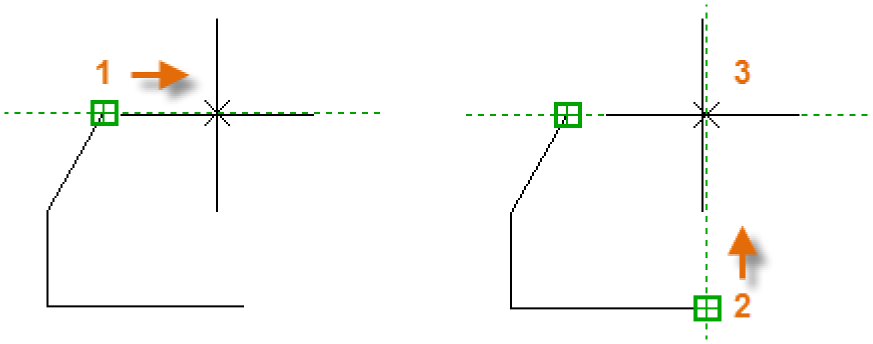


Fig. (41)

You can now finish creating the line, circle, or other object that you were creating from that location.

Verify Your Work

Recheck your geometry to catch mistakes early. Enter the **DIST** command (or just **DI**) to measure the distance between any two points in your model.

For example, you might need to find the clearance between two points shown, which might represent the corner of a wall and a small table, or perhaps a 2D section of a plastic part and a wire.

After you enter **DIST**, click the endpoint on the corner (1). Next, hold down Shift as you right-click, and then choose Perpendicular from the object snap menu. Finally, click the circle (2). As shown in Fig (42).

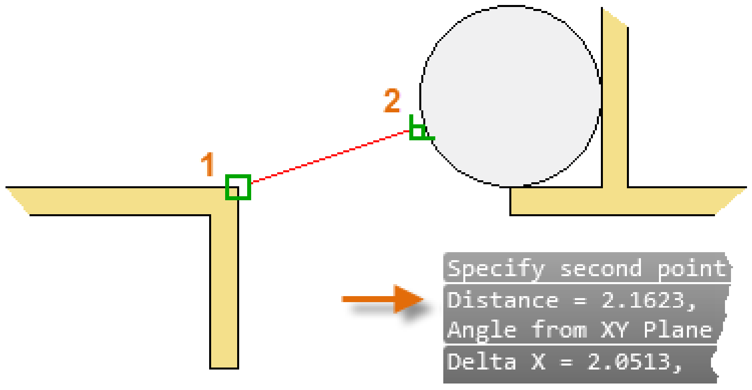


Fig. (42)

The number of decimal places and unit style displayed in the result is controlled by the **UNITS** command.

Handy Function Key Reference

All keyboard function keys have assignments in AutoCAD. The ones that are most commonly turned on and off are indicated with a key. As shown in table (1).

Table (1)

Key	Feature	Description
F1	Help	Displays Help for the active tooltip, command palette, or dialog box.
F2	Expanded History	Displays expanded command history in the Command Window.
F3	Object Snap	Turns object snap on and off.
F4	3D Object Snap	Turns on additional object snaps for 3D elements.
F5	Isoplane	Cycles through 2-1/2D isoplane settings.
F6	Dynamic UCS	Turns on UCS alignment with planar surfaces.

F7	Grid Display	Turns the grid display on and off.
F8	Ortho	Locks cursor movement to horizontal or vertical.
F9	Grid Snap	Restricts cursor movement to specified grid intervals.
F10	Polar Tracking	Guides cursor movement to specified angles.
F11	Object Snap Tracking	Tracks the cursor horizontally or vertically from object snap locations.
F12	Dynamic Input	Displays distances and angles near the cursor and accepts input as you use Tab between the fields.

Layers

Organize your drawing by assigning objects to layers.

When a drawing becomes visually complex, you can hide objects that you currently do not need to see. As shown in Fig (43).

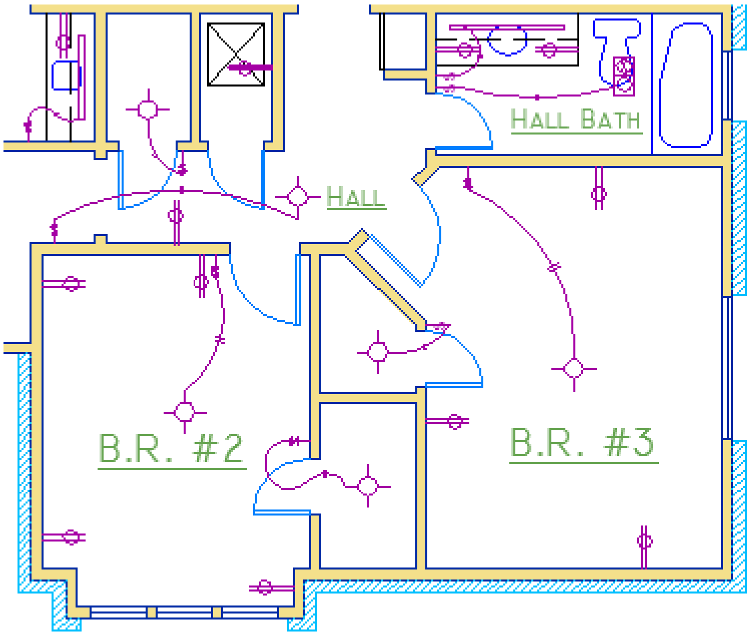


Fig. (43)

In the drawing below, the doors and electrical wiring were temporarily hidden by hiding their layers. As shown in Fig (44).

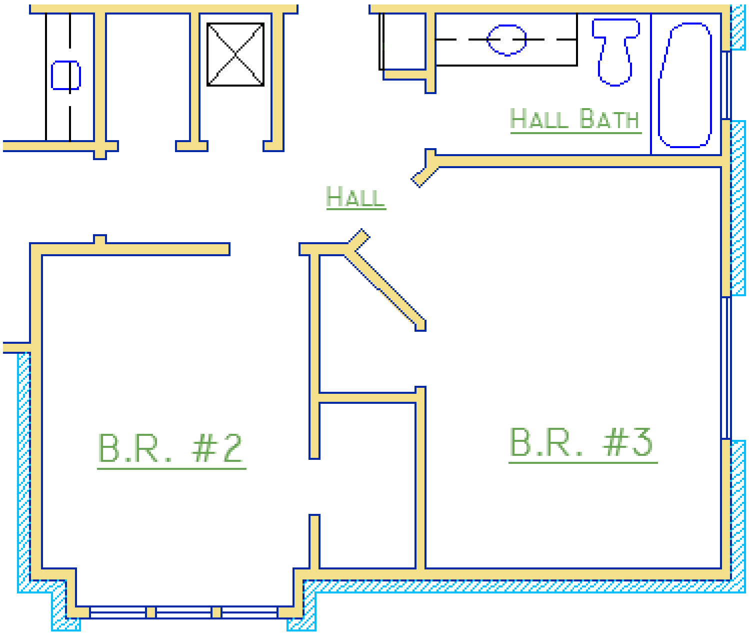


Fig. (44)

You gain this level of control by organizing the objects in your drawing on layers that are associated with a specific function or purpose. It might be helpful to think of layers as clear plastic sheets: As shown in Fig (45).

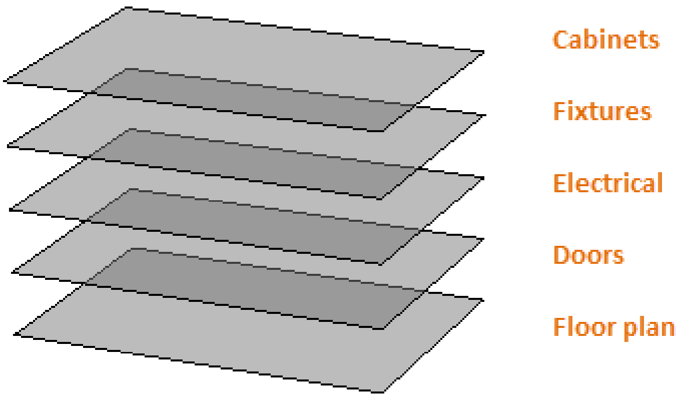


Fig. (45)

With layers, you can:

- Associate objects by their function or location
- Display or hide all objects related to a single operation
- Enforce linetype, color, and other property standards for each layer

Important:

Resist the temptation to create everything on one layer. Layers are the most important organizing feature available in AutoCAD drawings.

Layer Controls

To see how a drawing is organized, use the **LAYER** command to open the Layer Properties Manager. You can either enter **LAYER** or **LA** in the Command window, or you can click the Layer Properties tool on the ribbon. As shown in Fig (46).

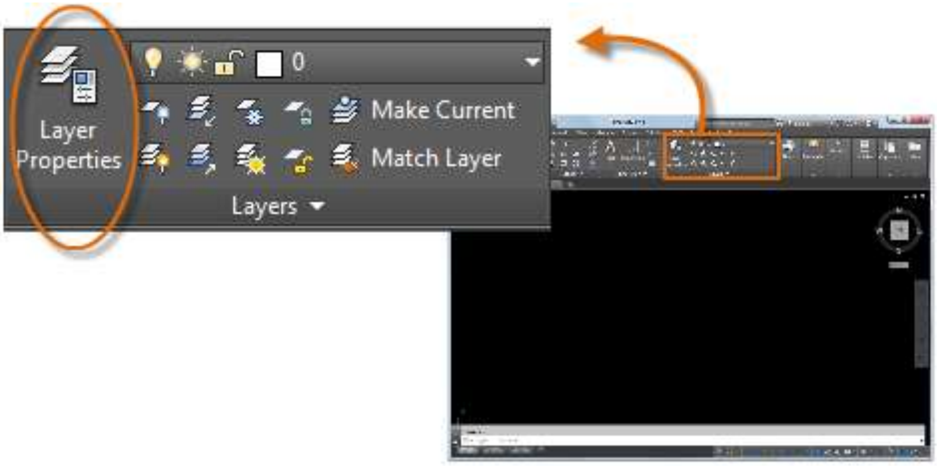


Fig. (46)

Here's what the Layer Properties Manager displays for this drawing. As shown in Fig. (47).

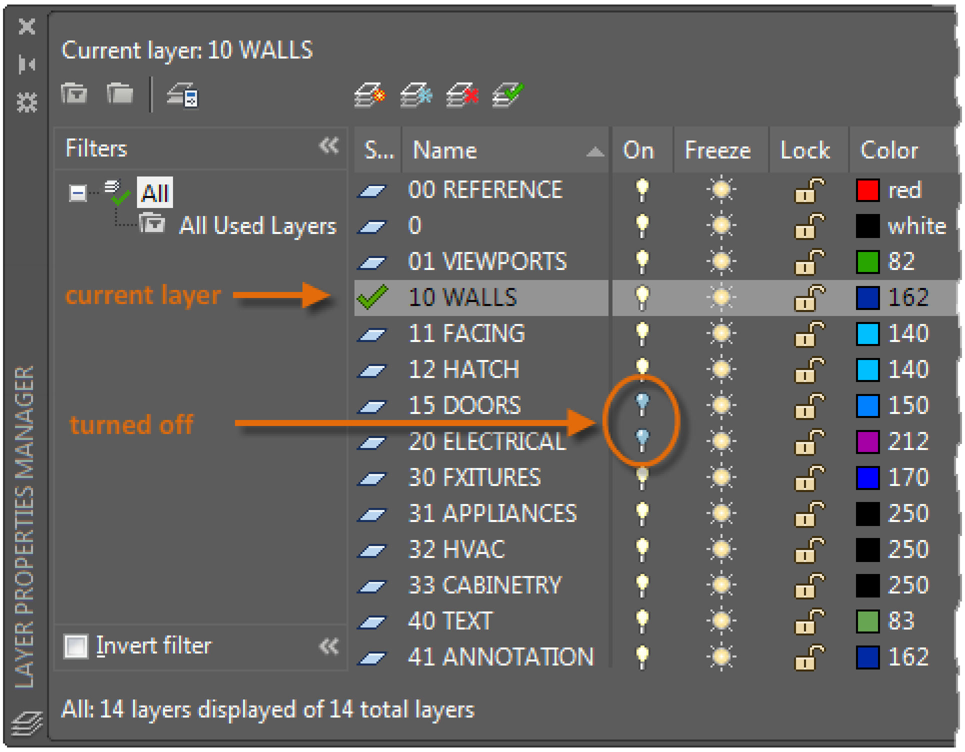


Fig. (47)

As indicated, layer 10 WALLS is the *current layer*. All new objects are automatically placed on that layer. In the list of layers, the green check next to layer 10 WALLS indicates that it is the current layer.

In the column labeled On, notice that the light bulb icons for two layers are dark. This indicates that these layers were turned off to hide the doors and electrical wiring in the floor plan.

Notice that each layer name starts with a two-digit number. This convention makes it easy to control the order of the layers because their order does not depend on the alphabet.

Here's a Tip:

For complex drawings, you might want to consider a more elaborate layer naming standard. For example, layer names could begin with 3 digits followed by a naming code that accommodates multiple floors in a building, project numbers, sets of survey and property data, and so on.

9.7 Practical Recommendations

- Layer 0 is the default layer that exists in all drawings and has some esoteric properties. Instead of using this layer, it’s best to create your own layers with meaningful names.
- Any drawing that contains at least one dimension object automatically includes a reserved layer named Defpoints.
- Create a layer for behind-the-scenes construction geometry, reference geometry, and notes that you usually do not need to see or print.
- Create a layer for layout viewports. Information about layout viewports is covered in the Layouts topic.
- Create a layer for all hatches and fills. This lets you to turn them all on or off in one action.

Layer Settings

The following are the most commonly used layer settings in the Layer Properties Manager. Click the icon to turn the setting on and off.

COMMONLY USER LAYER SETTINGS

As shown in table (2).

Table (2)

Turn Off Layers: This will help reduce the visual complexity of your drawing while you work. 
Freeze Layers: Freeze layers that you do not need to access for a while. Freezing layers is similar to turning them off, but improves performance in very large drawings. 

Lock Layers: Locking layers prevents accidental changes to the objects on those layers. Also, the objects on locked layers appear faded, which helps reduce the visual complexity of your drawing. 
Set Default Properties: You can set the default properties for each layer, including color, linetype, line weight, and transparency. New objects that you create will use these properties unless you override them. Overriding layer properties is explained later in this topic.

Controls in the Layer Properties Manager

To create a new layer, click the button shown and enter the name of the new layer. To make a different layer the current one, click the layer and then click the indicated button. As shown in Fig. (48).

Fig. (48)

Quick Access to Layer Settings

The Layer Properties Manager takes up a lot of space, and you may not always need to access all the options. For quick access to the most common layer controls, use the controls on the ribbon. When no objects are selected, the Layers panel on the Home tab displays the name of the current layer as shown here. As shown in Fig. (49).

Fig. (49)

Occasionally, check to make sure that the objects you create will be on the correct layer. It's easy to forget to do this, but it's also easy to set. Click the drop-down arrow to display a list of layers, and then click a layer on the list to make it the current layer. You can also click on any layer setting icon in the list to change its setting. As shown in Fig. (50).

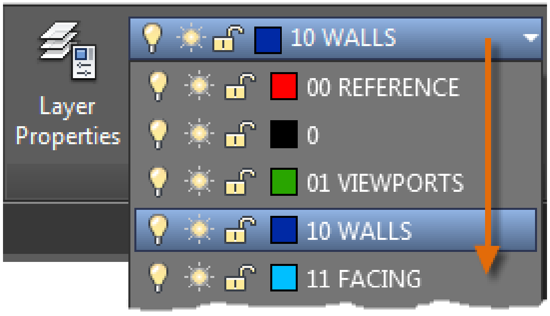


Fig. (50)

Maintain Your Standards

It's important to either establish or conform to a company-wide layer standard. With a layer standard, drawing organization will be more logical, consistent, compatible, and maintainable over time and across departments. Layer standards are essential for team projects.

If you create a standard set of layers and save them in a drawing template file, those layers will be available when you start a new drawing, and you can start working immediately. Additional information about drawing template files is presented in the Basics topic.

NOTE: *Some experienced AutoCAD users set properties only with layers, while others set properties independently of layers or in combination with layers. Assigning properties to objects is covered in the Properties topic.*

Summary

Layers organize your drawing, enabling you to temporarily hide unneeded graphical data. You can also assign default properties such as color and linetype to each layer.

Properties

You can assign properties such as color and linetype to individual objects, or as default properties assigned to layers.

In the following drawing, the walls, exterior stone facing, doors, fixtures, cabinetry, HVAC, electrical, and text were created using different colors to help differentiate them. As shown in Fig. (51).

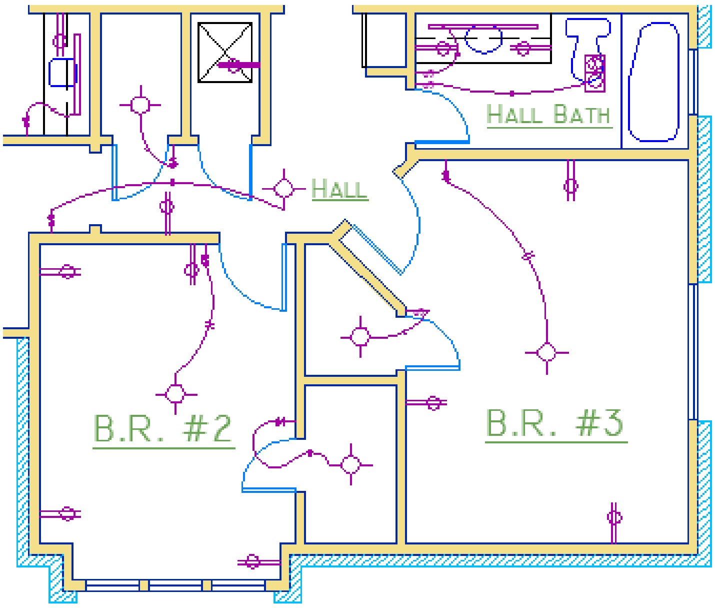


Fig. (52)

The Properties Palette

The Properties palette is an essential tool. You can open it with the **PROPERTIES** command (enter **PR** in the Command window), you can press Ctrl + 1, or you can click the tiny arrow in the Properties panel on the Home tab—whichever you prefer. As shown in Fig. (53).

Fig. (53)

The Properties palette displays a list of all the important property settings. You can click any of the available fields to change the current settings. In the following example, if no objects are selected, the current color will be changed from ByLayer to Red and the UCS icon will be turned off. As shown in Fig. (54).

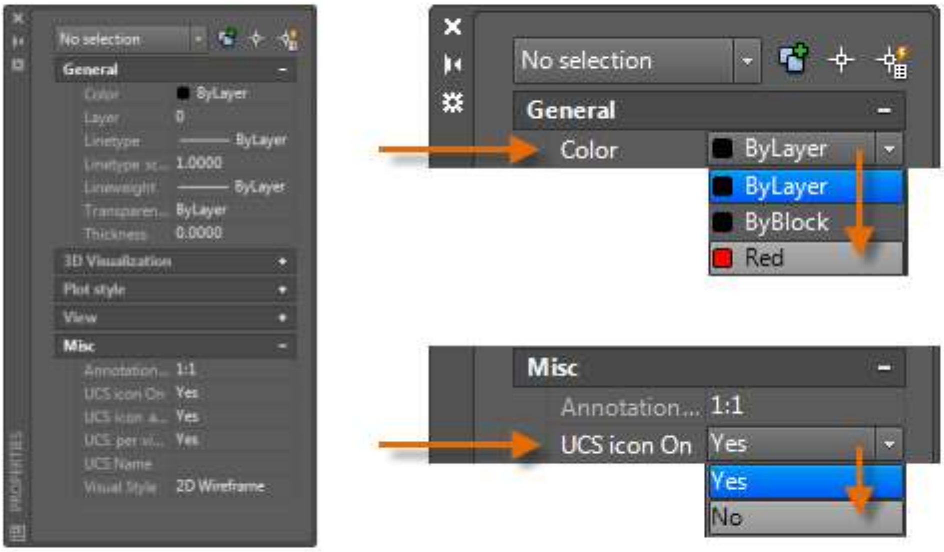


Fig. (54)

Verify and Change Object Properties

You can use the Properties palette to verify and change property settings for selected objects. If you click an object in your drawing to select it, here is what you might see in the Properties palette. As shown in Fig. (55).

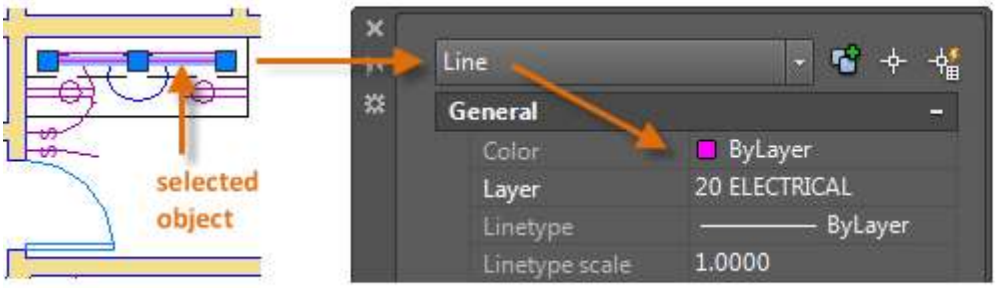


Fig. (55)

Notice that the current properties for the selected object are displayed in the palette. You can change any of these properties by clicking and changing the setting.

A property that is set to "By Layer" inherits its setting from the layer. In the previous example, the objects that were created on the 20 ELECTRICAL layer are purple because that is the default color of the objects on that layer.

If you select several objects, only their common properties are listed in the Properties palette. If you change one of these properties, all the selected objects will change in one operation. Selecting objects is covered in more detail in the Modifying topic.

NOTE: *To clear the current selection, press Esc.*

Quick Access to Property Settings

The Properties palette can take up a lot of space. For quick access to the most common properties, use the Properties panel. As you can see in this example, the listed properties will all be determined by the current layer. As shown in Fig. (56).

Fig. (56)

The Properties panel works the same way as the Properties palette. When you select an object, the current property settings are replaced by the properties assigned to the selected object, and you can use this panel to easily change the properties of one or more selected objects.

Match the Properties of Objects

For a fast way to copy the properties of a selected object to other objects, use the Match Properties tool, or enter **MATCHPROP** or **MA** in the Command window. As shown in Fig. (57).

Fig. (57)

Select the source object, and then select all of the objects that you want to modify.

Linetypes

- Dashed and other non-continuous line types are assigned from the Properties panel. You must first load a line type before you can assign it.
- In the Line type drop-down list, click Other. As shown in Fig. (58).

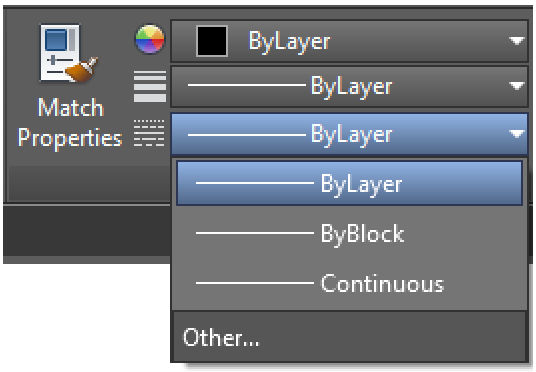


Fig. (58)

This action displays the Linetype Manager dialog box.

Perform the following steps in order:

1. Click Load. Choose one or more linetypes that you want to use. Notice that dashed (non-continuous) linetypes come in several preset sizes.
2. Click Show/Hide details to display additional settings.
3. Specify a different "global scale factor" for all linetypes—the larger the value, the longer the dashes and spaces. Click OK.
4. Once you've loaded the linetypes that you plan to use, you can select any object and specify a linetype from the Properties panel or the Properties palette. Alternatively, you can specify a default linetype for any layer in the Layer Properties Manager. As shown in Fig. (59).

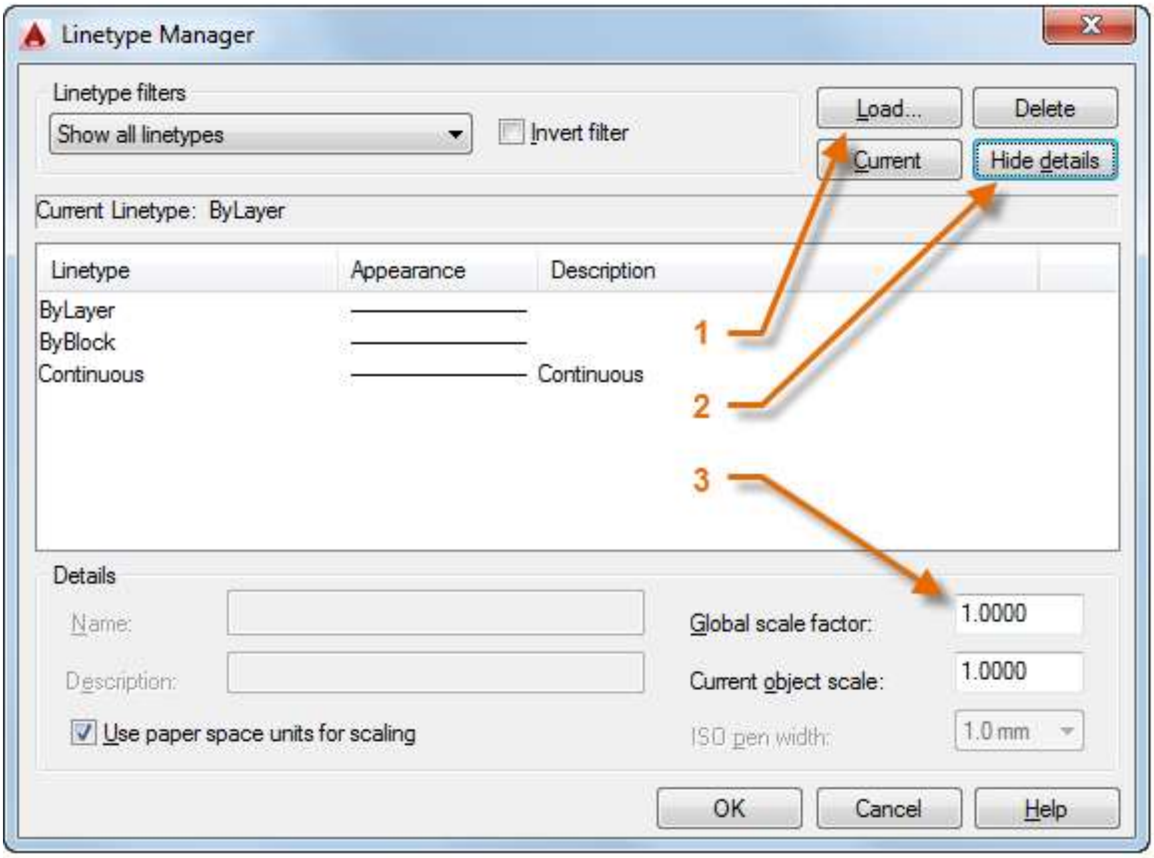


Fig. (59)

Lineweights

The Line weight property provides a way to display different thicknesses for selected objects. The thickness of the lines remains constant regardless of the scale of the view. In a layout, line weights are always displayed and printed in real-world units. Line weights can also be assigned from the Properties panel. As shown in Fig. (60).

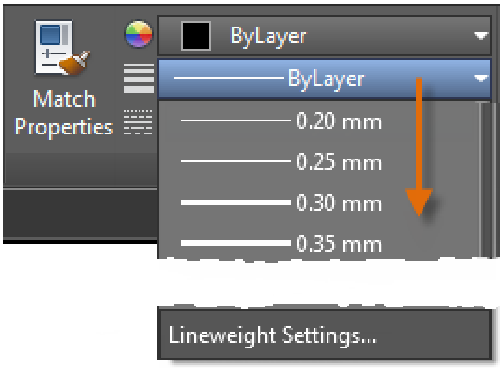


Fig. (60)

You can leave the line weight set to By Layer, or you can specify a value that overrides the layer's line weight. In some cases, the line weight previews look the same because they are displayed in approximated pixel widths on a monitor. However, they will print at the correct thickness.

To control the display of line weights, click the Line weight Settings button at the bottom of the line weight list. In the Line weight Settings dialog box, you can choose whether you want to display or hide line weights. As shown in Fig. (61).

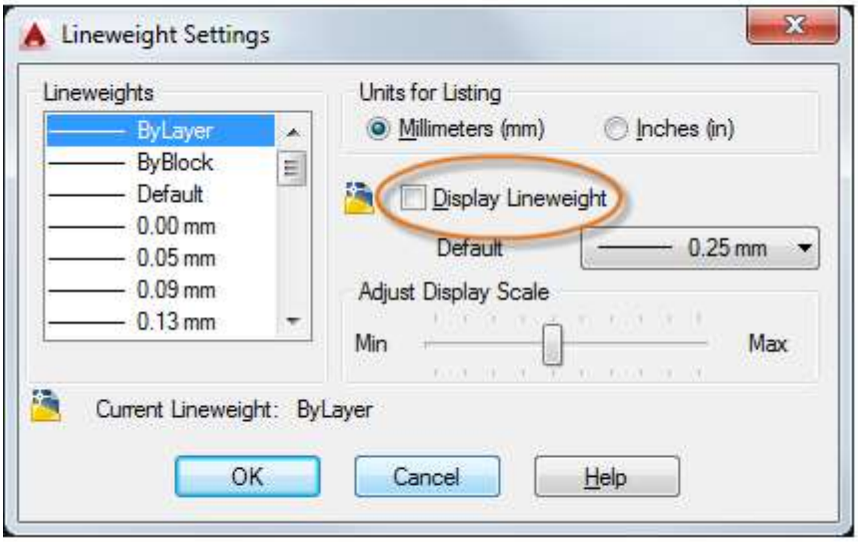


Fig. (61)

Regardless of the display setting, lineweights will always be printed at the correct scale.

Here's a Tip:

It's usually best to leave lineweights turned off while you work. Heavy lineweights can obscure nearby objects when you use object snaps. You might want to turn them on for checking purposes just before you print.

Modifying

Perform editing operations such as erase, move, and trim on the objects in a drawing.

The most common modifying tools are located on the Modify panel of the Home tab. Take a minute to look through them. As shown in Fig. (62).

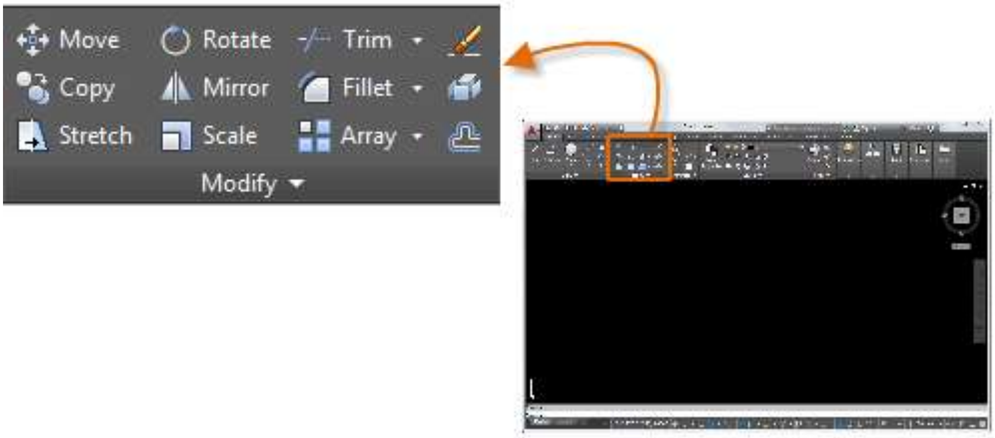


Fig. (62)

Erase

To erase an object, use the **ERASE** command. You can enter **E** in the Command window, or click the Erase tool. When you see the cursor change to a square pickbox, click each object that you want to erase, and then press Enter or the Spacebar.

NOTE: *Alternatively, before you enter any command, you can select several objects and then press the Delete key. Experienced users often use this method as well.*

Select Multiple Objects

Sometimes you may need to select a large number of objects. Instead of selecting each object individually, you can select the objects in an area by clicking an empty location (1), moving your cursor right or left, and then clicking a second time (2). As shown in Fig. (63).

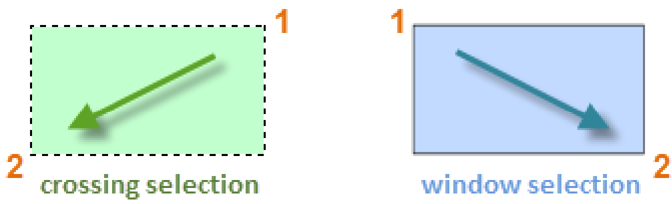


Fig. (63)

With a *crossing selection*, any objects within or touching the green area are selected.

• With a *window selection*, only the objects completely contained within the blue area are selected.

The result is called the *selection set*, which is the set of objects that will be processed by a command.

Here's a Tip:

You can easily remove objects from the selection set. For example, if you select 42 objects, and two of them should not have been selected, hold down Shift and then select the two that you want to remove. Then, press Enter or the Spacebar, or right click to end the selection process.

NOTE: *Clicking and dragging results in a different selection method called lasso selection.*

Move and Copy

Here's how you would use the **COPY** command to lay out a row of decorative tiles. Starting with a polyline that represents its shape, you need to make copies that are 1/8" apart. As shown in Fig. (64).

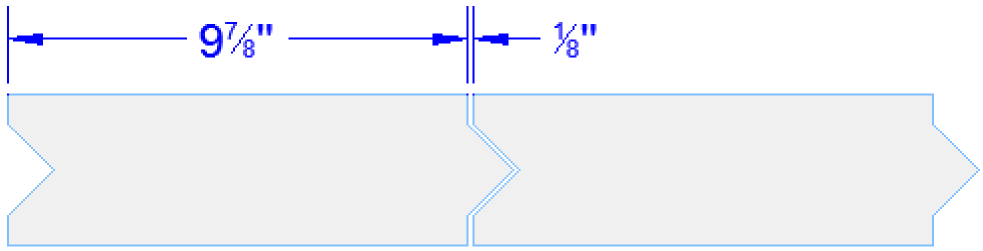


Fig. (64)

Click the Copy tool or enter **CP** in the Command window to start the command. From here, you can choose between two methods, depending on which is more convenient. You will use these two methods frequently.

The Two Points Method

Another method, one that you will often use when you don't want to add numbers together, requires two steps. Enter the **COPY** command and select the tile as before, but this time click the two endpoints as shown. These two points also define a distance and direction. As shown in Fig. (65)

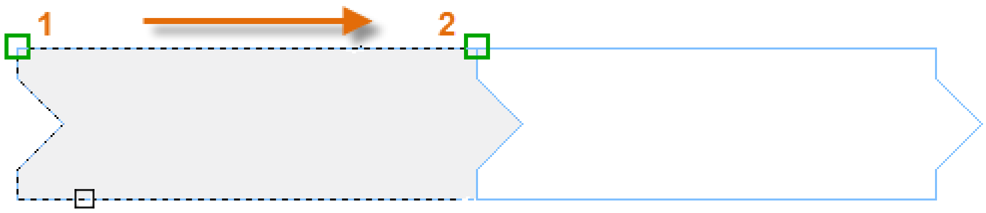


Fig. (65)

Next, to add the 1/8" space between the tiles, click the Move tool or enter **M** in the Command window. The **MOVE** command is similar to the **COPY** command. Select the newly copied tile, and press Enter or the Spacebar. As before, click anywhere in the drawing area and move your cursor to the right. Enter 1/8 or .125 for the distance.

Here's a Tip:

The two points that define the distance and direction don't need to be located on the object that you want to copy. You can use two points specified anywhere in your model.

The Distance Method

The second tile needs to be a total of 9-7/8" + 1/8" = 10" to the right of the original tile. Start by selecting the tile, pressing Enter or the Spacebar to end your selection, and clicking anywhere in the drawing area (1). This point does not have to be located on the tile. As shown in Fig. (66).

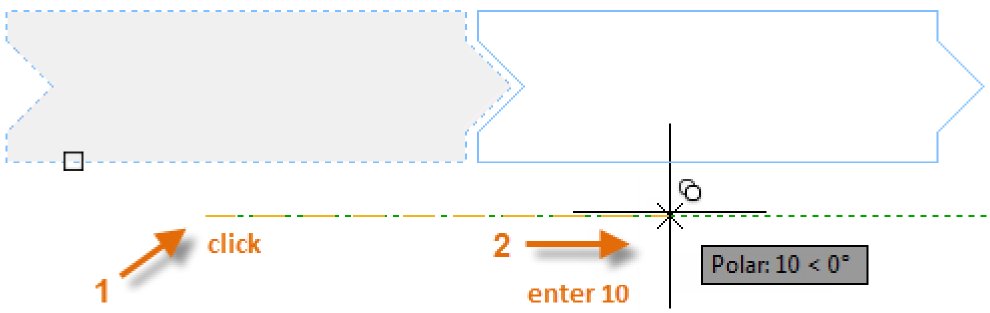


Fig. (66)

Next, move your cursor to the right, relying on the polar tracking angle to keep the direction horizontal, and then enter 10 for the distance. Press Enter or the Spacebar a second time to end the command.

The specified distance and a direction from a point (1) is applied to the tile that you selected.

Create Multiple Copies

You can use the two-point method as a repeating sequence. Let's say that you want to make more copies of the circle at the same horizontal distance. Enter the **COPY** command and select the circle as shown. As shown in Fig. (67).

Fig. (67)

Then, using the Center object snap, click the center of circle 1, followed by the center of circle 2, and so on. As shown in Fig. (68).

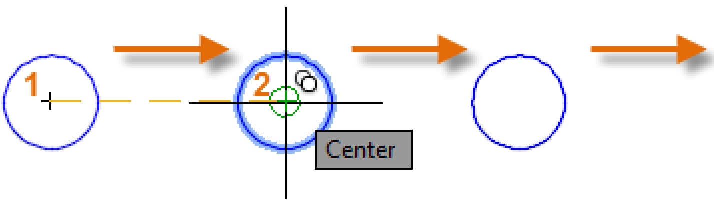


Fig. (68)

For larger numbers of copies, try experimenting with the Array option of the **COPY** command. For example, here's a linear arrangement of deep foundation piles. From a base point, you specify number of copies and the center-to-center distance. As shown in Fig. (69).

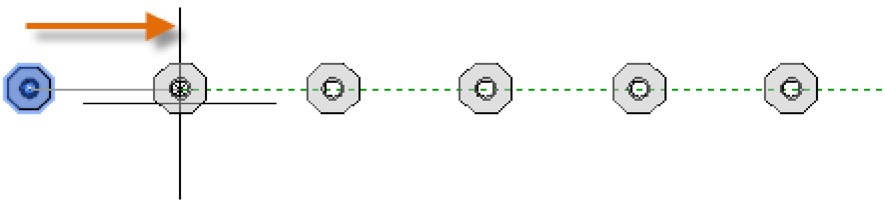


Fig. (69)

Offset

Most models include a lot of parallel lines and curves. Creating them is easy and efficient with the **OFFSET** command. Click the **OFFSET** tool or enter **O** in the Command window. As shown in Fig. (70).

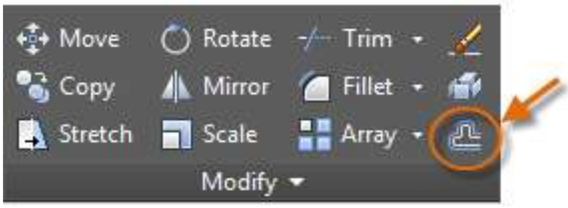


Fig. (70)

Select the object (1), specify the offset distance, and click to indicate on which side of the original that you want the result (2). Here is an example of offsetting a polyline. As shown in Fig. (71).

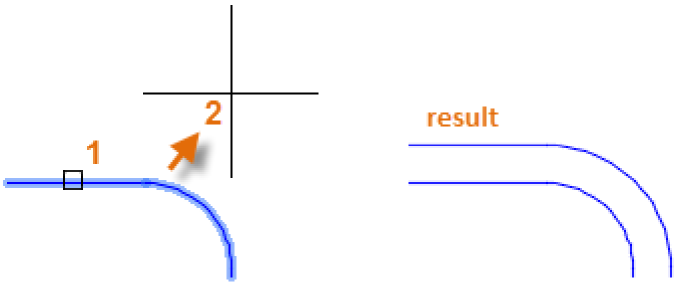


Fig. (71)

Here's a Tip:

A fast way to create concentric circles is to offset them.

Trim and Extend

A popular technique is to use the **OFFSET** command in combination with the **TRIM** and **EXTEND** commands. In the Command window, you can enter **TR** for **TRIM** or **EX** for **EXTEND**. Trimming and extending are some of the most commonly used operations. As shown in Fig. (72).

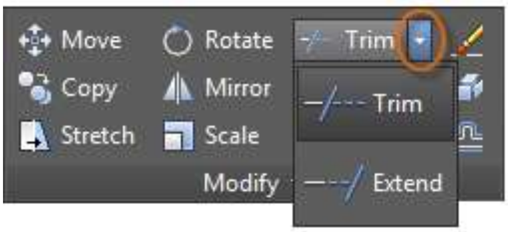


Fig. (72)

In the following illustration, you want to extend the lines that represent the steps for this deck. Enter the **EXTEND** command, select the boundary, and then press Enter or the Spacebar. As shown in Fig. (73).

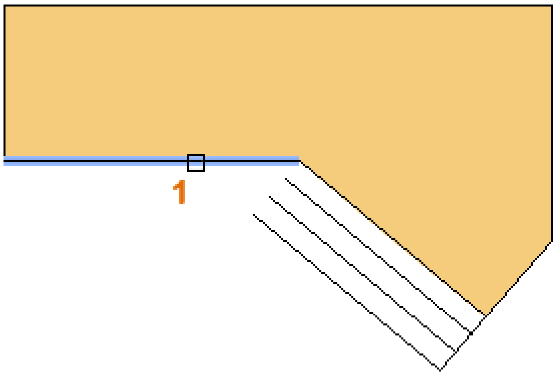


Fig. (73)

Pressing Enter or the Spacebar indicates that you've finished selecting the boundaries, and that you're now ready to select the objects to be extended.

Here's a Tip:

A faster method is to press Enter or the Spacebar right away instead of selecting any boundary objects. The result is that all objects are available as possible boundaries.

Next, select the objects to be extended (near the ends to be extended), and then press Enter or the Spacebar to end the command. As shown in Fig. (74).

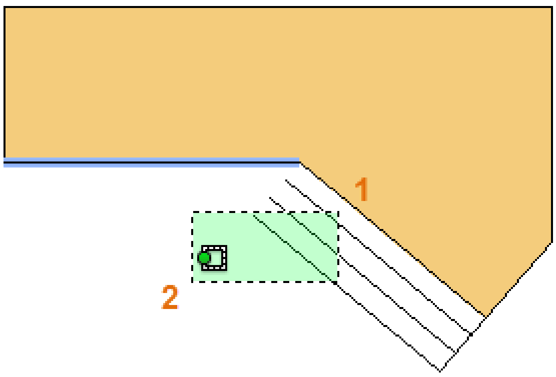


Fig. (74)

As you can see, the selected lines are extended to the boundary. As shown in Fig. (75).

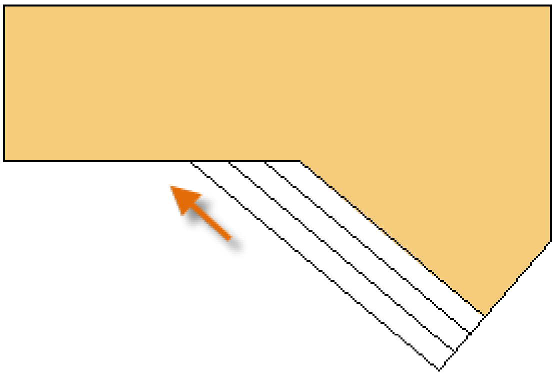


Fig. (75)

The **TRIM** command follows the same steps, except that when you select the objects to trim, you select the portions to trim away.

Mirror

The following illustration comes from a tile project. The walls in this residential bathroom are flattened out to be able to lay out the tile pattern and estimate the number of tiles needed. As shown in Fig. (76).

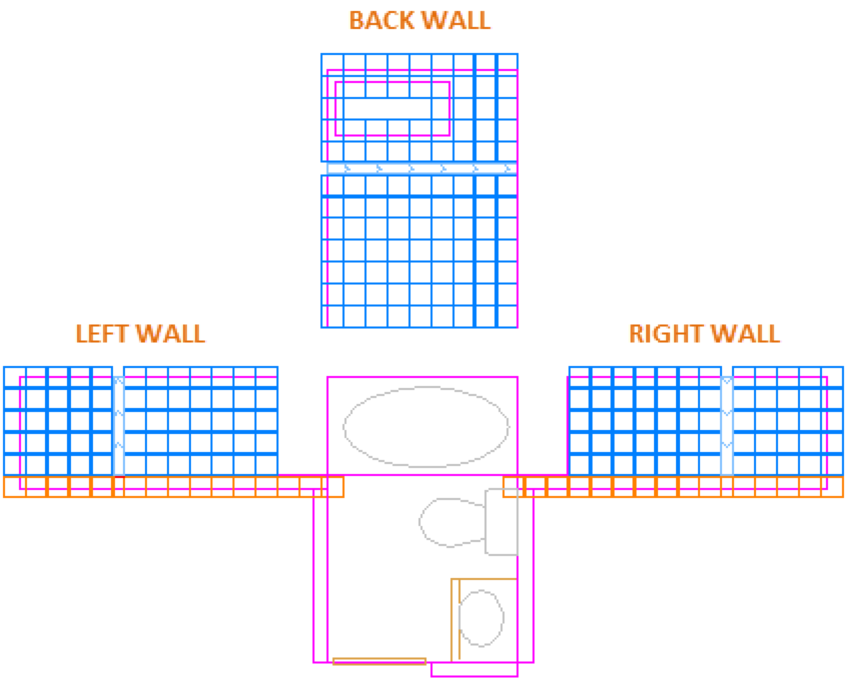


Fig. (76)

You can save a lot of work by taking advantage of the symmetry between the left and right walls. All you need to do is create the tiles on one wall and then mirror the wall across the center of the room.

In the example below, begin the **MIRROR** command (or enter **MI** in the Command window), use window selection (1 and 2) to select the geometry on the right wall, press Enter or the Spacebar, and then specify a mirror line (3 and 4) corresponding to the centerline of the bathroom. As shown in Fig. (77).

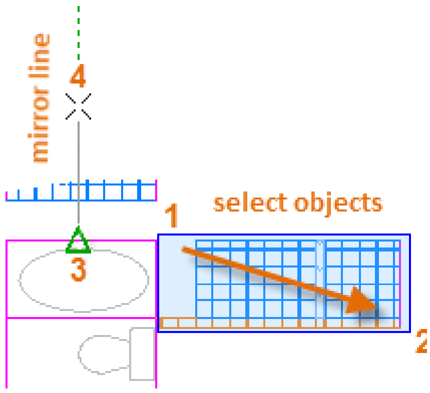


Fig. (77)

Finally, decline the option to "Erase source objects" by pressing Enter or the Spacebar. As shown in Fig. (78).

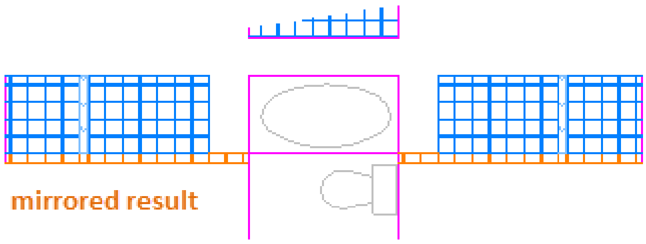


Fig. (78)

Here's a Tip:

Always look for symmetry to save yourself extra work, even if the symmetry is not 100% identical.

Stretch

You can stretch most geometric objects. This lets you lengthen and shorten parts of your model. For example, this model might be a gasket or the design for a public park. As shown in Fig. (79).

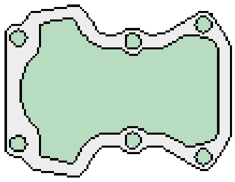


Fig. (79)

Use the **STRETCH** command (or enter **S** in the Command window) and select the objects with a crossing selection as shown below (1 and 2). The crossing selection is mandatory—only the geometry that is crossed by the crossing selection is stretched. Then click anywhere in the drawing area (3), move the cursor to the right, and enter 50 as the distance. This distance might represent millimeters or feet. As shown in Fig. (80).

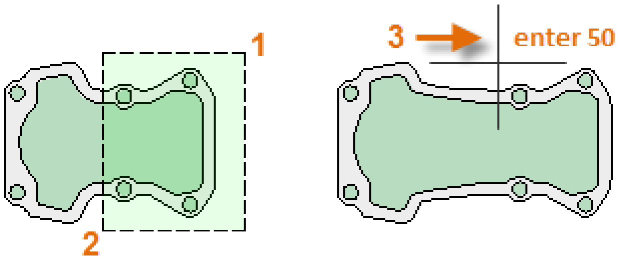


Fig. (80)

To shorten the model by a specified amount, you'd move your cursor to the left instead.

Fillet

The **FILLET** command (enter **F** in the Command window) creates a rounded corner by creating an arc that is tangent to two selected objects. Notice that the fillet is created relative to where you select the objects.

You can create a fillet between most types of geometric objects, including lines, arcs, and polyline segments. As shown in Fig. (81).

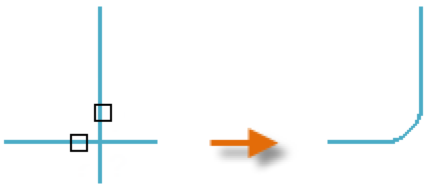


Fig. (81)

Here's a Tip:

If you specify 0 (zero) as the radius of the fillet (imagine a circle shrinking to a radius of 0), the result trims or extends the selected objects to a sharp corner.

Explode

The **EXPLODE** command (enter **X** in the Command window) disassociates a compound object into its component parts. You can explode objects such as polylines, hatches, and blocks (symbols).

After you explode a compound object, you can modify each resulting individual object.

Edit Polylines

You can choose from several useful options when you want to modify a polyline. The **PEDIT** command (enter **PE** in the Command window) is located on the drop-down list of the Modify panel. As shown in Fig. (82).

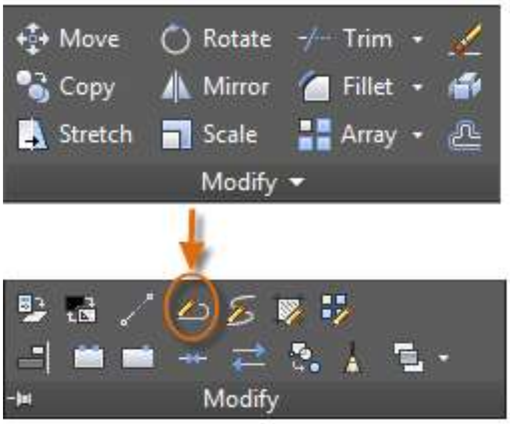


Fig. (82)

With this command, *you can:*

- Join two polylines into a single polyline if they share a common endpoint.
- Convert lines and arcs into a polyline—simply enter **PEDIT** and select the line or arc.
- Change the width of a polyline.

Here's a Tip:

In some cases, the easiest method to modify a polyline is to explode it, make the modifications, and then turn the objects back into a polyline using the Join option of the ***PEDIT*** *command.*

Grips

Grips are displayed when you select an object without starting a command. Grips are often handy for light editing. For example, the line below accidentally snapped to the wrong endpoint. You can select the misaligned line, click on a grip and then click to specify the correct location. As shown in Fig. (83).

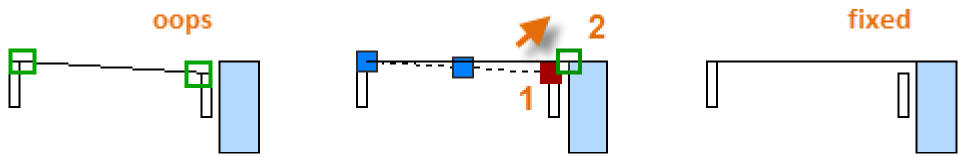


Fig. (83)

By default, when you click a grip, you automatically start in **STRETCH** mode as indicated in the Command window. If you want to explore other ways of editing objects with grips, press Enter or the Spacebar to cycle through several other editing modes. Some people perform most editing operations using grips.

Blocks

Insert symbols and details into your drawings from commercial online sources or from your own designs.

In AutoCAD, a *block* is a collection of objects that are combined into a single named object. The following are some sample blocks at various scales. As shown in Fig. (84).

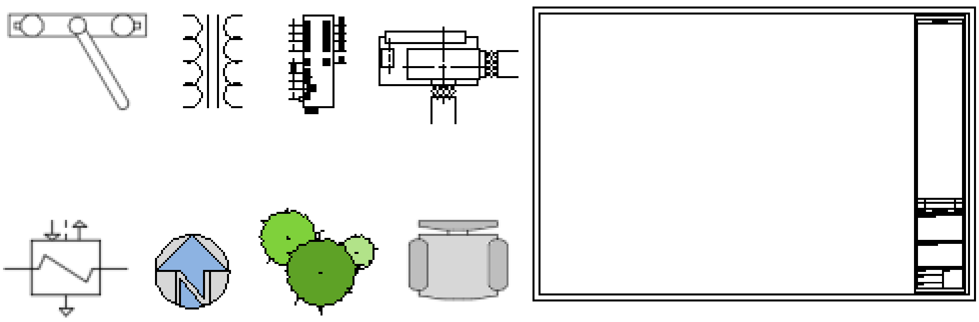


Fig. (84)

Some of these blocks are realistic representations of objects, some are symbols, and one of them is an architectural title block for a D-size drawing.

Insert a Block

Typically, each of these blocks is an individual drawing file, perhaps saved in a folder with similar drawing files. When you need to insert one into your current drawing file, use the **INSERT** command (or enter **I** in the Command window). As shown in Fig. (85).

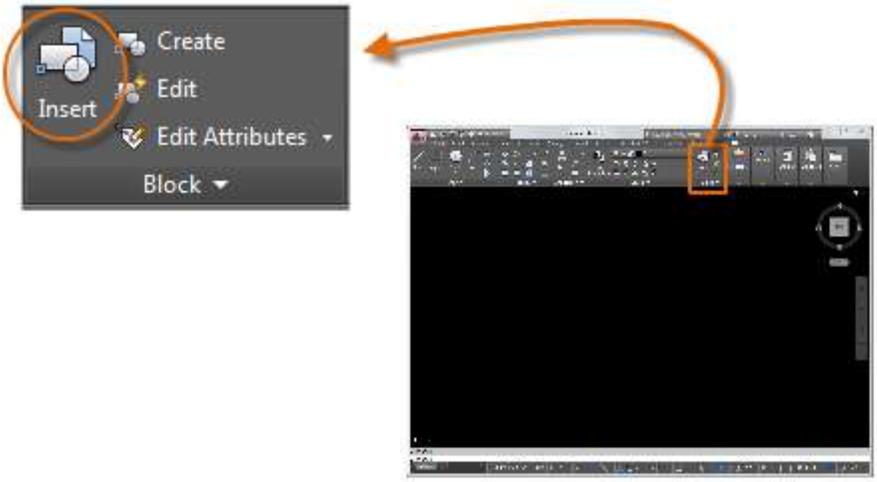


Fig. (85)

The first time you insert the drawing as a block, you must click Browse to locate the drawing file. Make sure you organize your blocks into easy-to-find folders. As shown in Fig. (86).

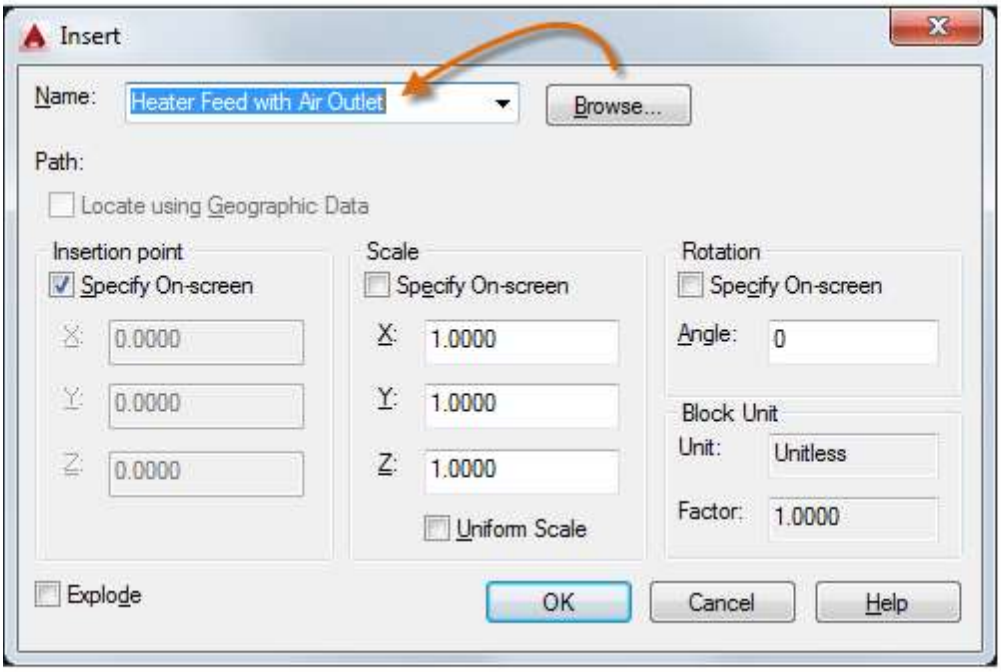


Fig. (86)

Once inserted, the *block definition* is stored in your current drawing. From then on, you can choose it from the Name drop-down list without needing to click the Browse button.

Here's a Tip:

The default settings in the Insert dialog box are usually acceptable. After you choose the block name, click OK, and then specify its location in your drawing. You can rotate it later, if necessary.

Notice that when you insert a block, it is attached to your cursor at the point indicated. This location is called the insertion point. By default, the insertion point is the origin point (0,0) of the original drawing. As shown in Fig. (87).

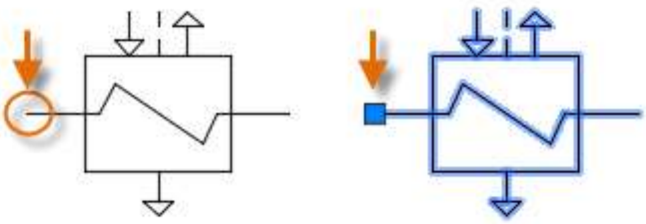


Fig. (87)

After inserting the block, you can select it and a grip appears. You can easily move and rotate this block using this grip. As shown in Fig. (88).

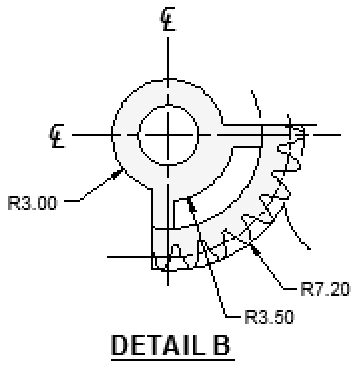


Fig. (88)

In the following example, a drawing file is inserted into the current drawing to provide a standard detail view.

NOTE: *Inserting a drawing file as a block provides a static reference to the specified drawing. For a reference that automatically updates, you can attach the drawing with the External References palette (**XREF** command) instead.*

Create a Block Definition

Instead of creating a drawing file to be inserted as a block, you might want to create a block definition directly in your current drawing. Use this method if you do not plan to insert the block into any other drawing. In that case, use the **BLOCK** command to create the block definition. As shown in Fig. (89).

Fig. (89)

For example, this is how you could create a module for a *cubicle design.*

1. Create the objects for the block.

2. Start the **BLOCK** command.

3. Enter a name for the block, in this case Quad-Cube.

4. Select the objects that you created for the block (click 1 and 2).

5. Specify the block insertion point. As shown in Fig. (90).

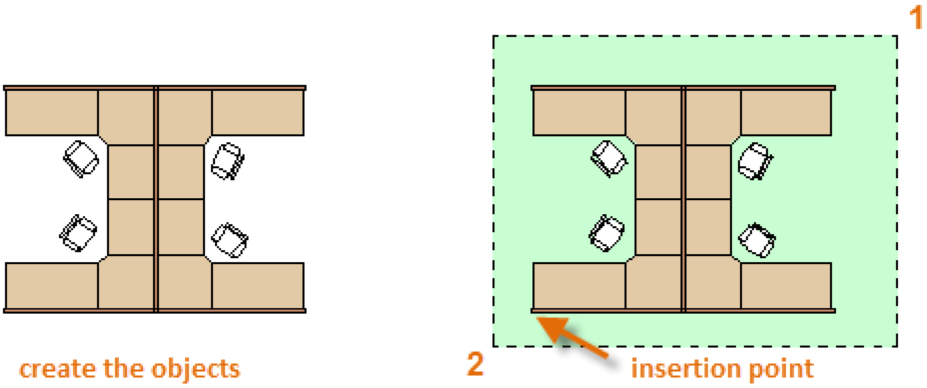


Fig. (90)

You can enter the information for steps 3, 4, and 5 into the Block Definition dialog box in any order. As shown in Fig. (91).

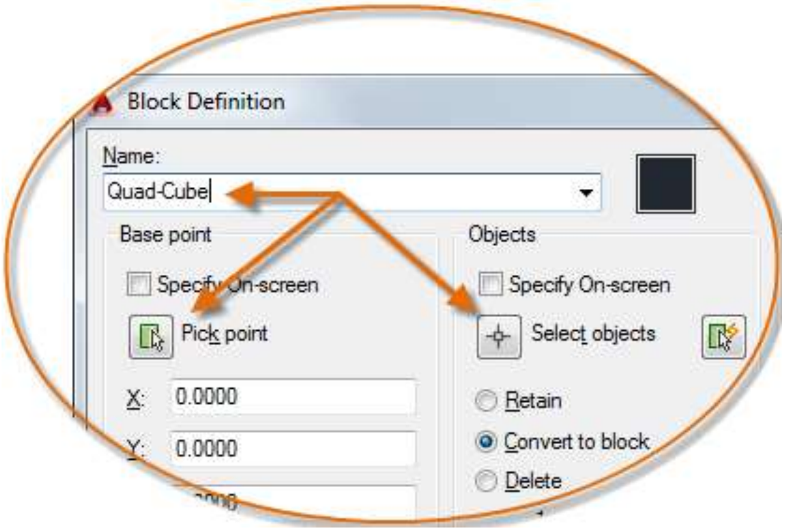


Fig. (91)

After creating the block definition, you can insert, copy, and rotate the block as needed. As shown in Fig. (92).

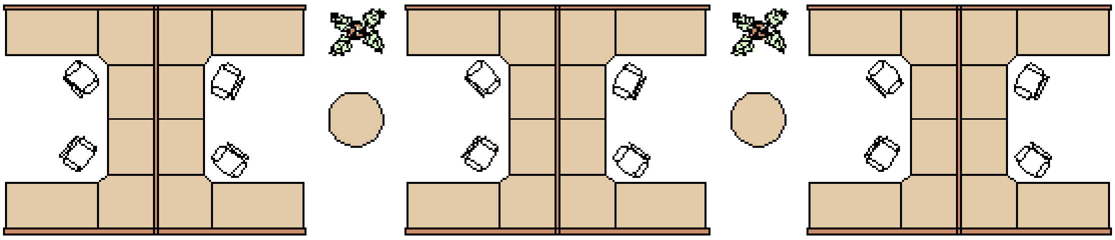


Fig. (92)

Using the **EXPLODE** command, you can explode a block into its component objects if you need to make changes. In the illustration below, the cubicle on the right was exploded and modified. As shown in Fig. (93).

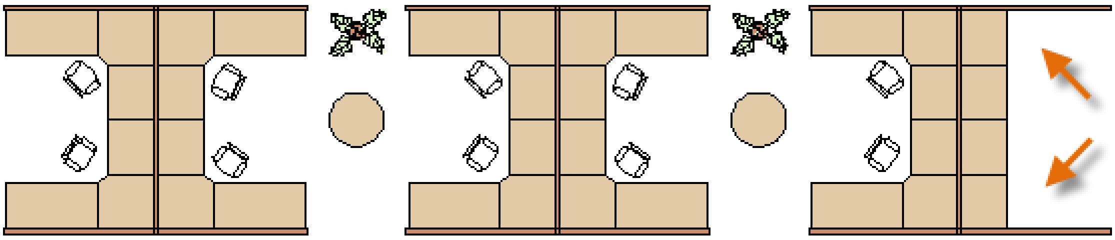


Fig. (93)

In this example, you would probably create a new block definition from the objects in the exploded block.

Recommendations

There are several different schemes for saving and retrieving block definitions.

You can create an individual drawing file for each block that you intend to use. You save these drawing files in folders, each of which would contain a family of related drawing files.

You can include the block definitions for title blocks and common symbols in your drawing template files to make them available immediately when starting a new drawing.

You can create several drawing files, which are sometimes called *block library drawings*. Each of these drawings contains a family of related block definitions. When you insert a block library drawing into your current drawing, all the blocks that are defined in that drawing become available.

Here's a Tip:

With online access, you can download AutoCAD drawing files from the web sites of commercial vendors and suppliers. This can save you time, but always check to make sure that they are drawn correctly and to scale. Autodesk Seek (**http://seek.autodesk.com/**) is a convenient way of accessing BIM (building information modeling) libraries.

NOTE: *You can create block definitions that include one or more attributes that store and display information. The command that you would use is* ***ATTDEF****. Typically, attributes include data such as part number, name, cost, and date. You can export block attribute information to a table or to an external file.*

Layouts

Display one or more scaled views of your design on a standard-size drawing sheet called a layout.

After you finish creating a model at full size, you can switch to a *paper space layout* to create scaled views of the model, and to add notes, labels, and dimensions. You can also specify different linetypes and line widths for display in paper space.

Model Space and Paper Space

As you know, you create the geometry of your model in *model space*. As shown in Fig. (94).

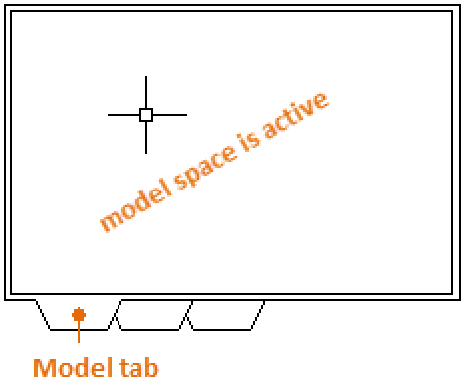


Fig. (94)

Originally, this was the only space available in AutoCAD. All notes, labels, dimensions, and the drawing border and title block were also created and scaled in the model space.

After *paper space* was introduced, you could click a layout tab to access a space designed specifically for layouts and scaling. In the following illustration, paper space is active. There are currently only two objects in paper space: a drawing border block, and a single *layout viewport*, which displays a view of model space. As shown in Fig. (95).

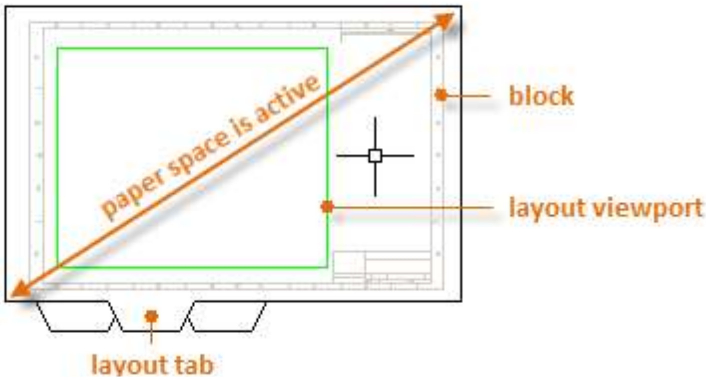


Fig. (95).

Working with layout viewports is described in more detail later in this topic.

Four Methods for Scaling

There are four different methods in AutoCAD that are used to scale views, notes, labels, and dimensions. Each method has its advantages depending on how the drawing will be used. Here’s a brief summary of each of the methods:

1. **The Original Method:** You create geometry, annotate, and print from model space. Dimensions, notes, and labels must all be scaled in reverse. You set the dimension scale to the inverse of the plot scale. With this method, scaling requires a little math. For example, a common scale used in architecture is 1/4” = 1’-0” which is 1:48 scale. If a note is to be printed 1/4” high, then it must be created 48 times as large, or 12” high in model space. The same scale factor also applies to dimensions, and an ARCH D drawing border at that scale is 144 feet long. When the drawing is printed as a D-size sheet, everything scales down to the correct size.
2. **The Layout Method:** You create geometry and annotate in model space, and print from the layout. Set the dimension scale to 0 and the dimensions will scale automatically.
3. **The Annotative Method:** You create geometry in model space, create *annotative* dimensions, notes, and labels (using a special annotative style) in model space from the layout, and you print from the layout. Annotative objects display only in layout viewports that share the same scale. The dimension scale is automatically set to 0 and all annotative objects scale automatically.

4. **The Trans-Spatial Method:** You create geometry in model space, create annotations in paper space on a layout with dimension scale set to 1, and you print from the layout. This is arguably the easiest, most direct method, and it is the method of choice for this guide.

Talk to other AutoCAD users in your discipline about these four methods and why they chose the method that they use.

NOTE: *Many AutoCAD drawings were created with this method, and many companies still use it. Once everything is set up, the method works well for 2D drawings with single views and inserted details.*

Specifying the Paper Size of a Layout

The first thing you should do when you access a layout tab (1) is right-click the tab (2) and rename it (3) to something more specific than Layout 1. For a D-size layout, ARCH D or ANSI D might be good choices. As shown in Fig. (96).

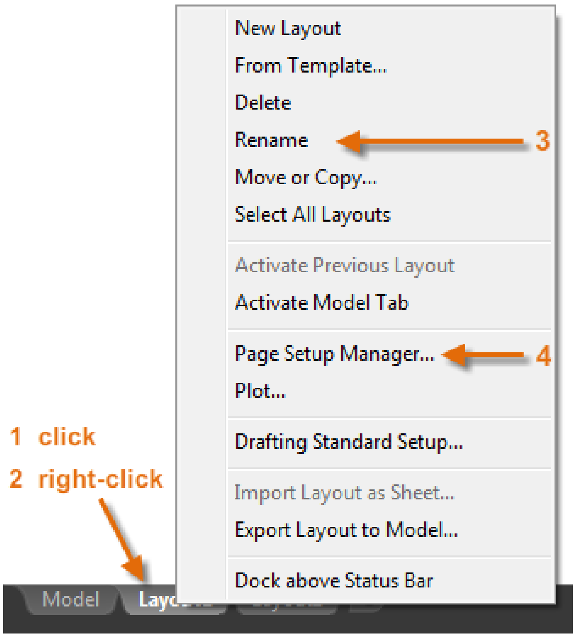


Fig. (96)

Next, open the Page Setup Manager (4) to change the paper size displayed in the layout tab. As shown in Fig. (97).

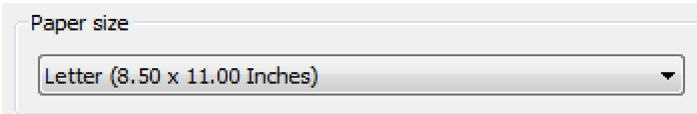


Fig. (97).

NOTE: You might be wondering why there are two entries in the list for every sheet size. This is because some printers and plotters do not recognize the drawing orientation setting. As shown in Fig. (98).

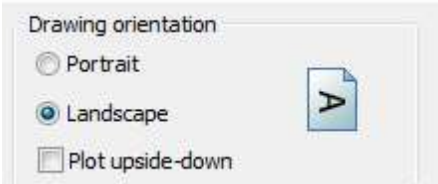


Fig. (98)

Layout Viewports

A layout viewport is an object that is created in paper space to display a scaled view of model space. You can think of it as a closed-circuit TV monitor that displays part of model space. In the illustration, model space is active and accessible from within the current layout viewport. As shown in Fig. (99).

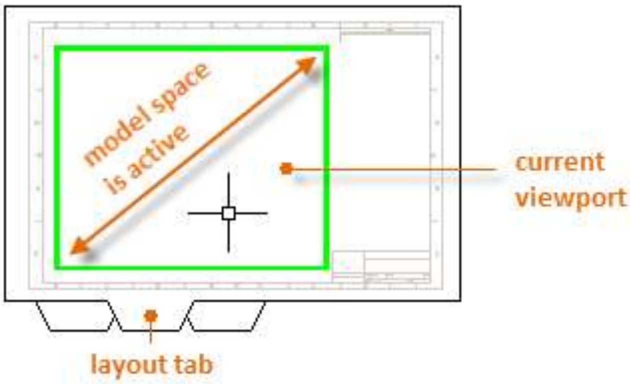


Fig. (99)

In a layout, when model space is active, you can pan and zoom, and anything else that you could do on the Model tab. For example, let's say that you created a backyard deck design in model space, and now you want to lay out and print your design from a layout tab. As shown in Fig. (100).

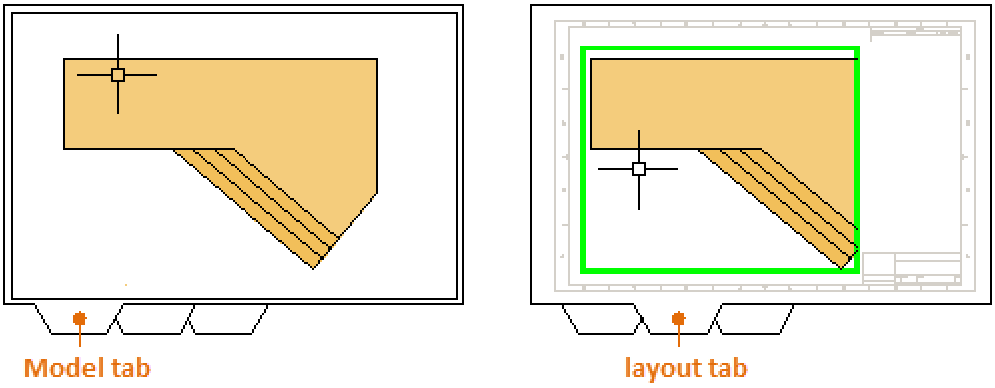


Fig. (100)

The view in the layout viewport is not yet set to the correct scale.

Important:

You can switch between paper space and model space by double-clicking inside or outside the layout viewport.

NOTE: *You can use the* ***MVIEW*** *(make view) command to create additional layout viewports in paper space. With several layout viewports, you can display several views of model space at the same or at different scales.*

Scaling Views and Trans-Spatial Annotation

Here are the steps to follow if you use the trans-spatial method of annotating your drawing:

1. Click the layout tab. If you started the drawing with your own custom drawing template file, several tasks might already have been completed: the layout might already be set to D-size, and the title block might already have been inserted in the layout.

2. By default, paper space is active, so double-click within the layout viewport to make the model space active. Notice that the edge of the layout viewport becomes thicker as a result of switching to model space.

3. Zoom out and center the model space view by panning. Note that the displayed view will not yet be set to the correct scale.

4. Double-click outside the layout viewport to make paper space active again.

5. Open the Properties palette and then click to select the edge of the layout viewport. As shown in Fig. (101).

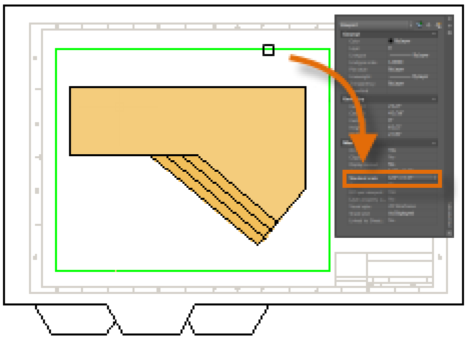


Fig. (101)

6. In the Properties palette, specify a standard scale of 1/4" = 1'-0" from the drop-down list. This action scales your view of model space precisely to the D-size drawing. You should also set the Display Locked property from No to Yes, which will prevent any unintentional display changes to the view. As shown in Fig. (102).

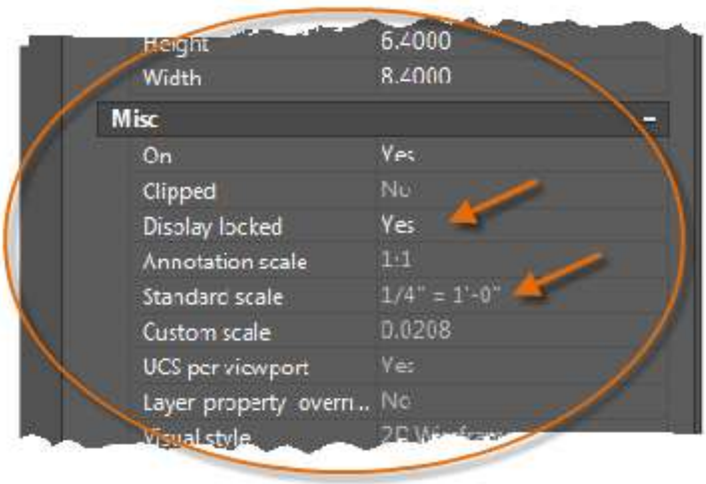


Fig. (102)

NOTE: By default, the dashes and spaces in a non-continuous linetype appear the same length regardless of the scale of the layout viewport.

7. Move the layout viewport as needed, and adjust its edges using grips.

8. Create notes, labels, and dimensions directly in paper space. They will automatically appear at the correct size.

9. Turn off the layer on which you created the layout viewport object. This hides the edges of the layout viewport as shown below. As shown in Fig. (103).

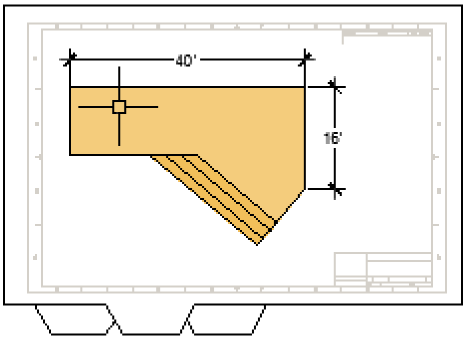


Fig. (103)

10. Print the drawing to paper or as a DWF or PDF file.

NOTE: *After you have finished dimensioning, you can use the EXPORTLAYOUT command to merge everything in model and paper space into the model space of a separate drawing file. This operation creates a drawing file that conforms to the original method of creating the model and all annotations in model space.*

Notes and Labels.

Create notes, labels, bubbles, and callouts. Save and restore style settings by name.

You can create general notes using the **MTEXT** command (or enter **MT** in the Command window), which stands for *multiline text*. The multiline text tool is available on the Annotation panel. As shown in Fig. (104).

Fig. (104)

After you start the **MTEXT** command, you are prompted to create a "text box" with two diagonal clicks. As shown in Fig. (105).

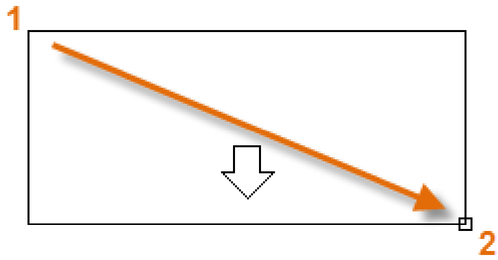


Fig. (105)

The exact size of the text box is not that important. After you specify the text box, the In-Place Editor is displayed and you can easily change the length and width of the note before, during, or after typing the text. As shown in Fig. (106).

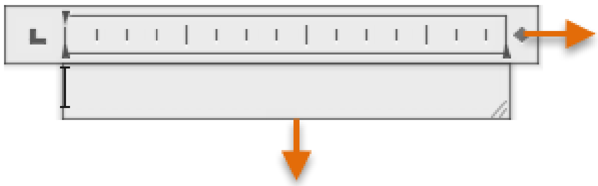


Fig. (106)

All the usual controls are available in the In-Place Editor, including tabs, indents, and columns. Also notice that when you start the **MTEXT** command, the ribbon temporarily changes, displaying many options such as text styles, columns, spell checking, and so on.

1. To exit the text editor after you finish entering the text, click anywhere outside it.
2. To edit a note, simply double-click it to open the text editor.

Here's a Tip:

You can use the Properties palette to control the text style used for one or more selected multiline text objects. For example, after selecting five notes that use different styles, click the Style column and choose a style from the list. As shown in Fig. (107).

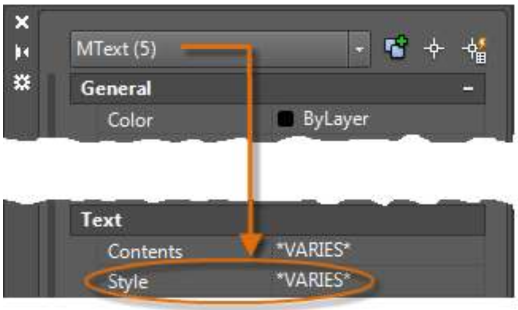


Fig. (107)

Create a Text Style

As with several other annotation features, multiline text provides a lot of settings. You can save these settings as a *text style* using the **STYLE** command, and then you can access the text styles you've saved by clicking the drop-down arrow on the Annotation panel. The current text style is displayed at the top of the drop-down list.

To create a new text style, click the Text Style control as shown. As shown in Fig. (108).

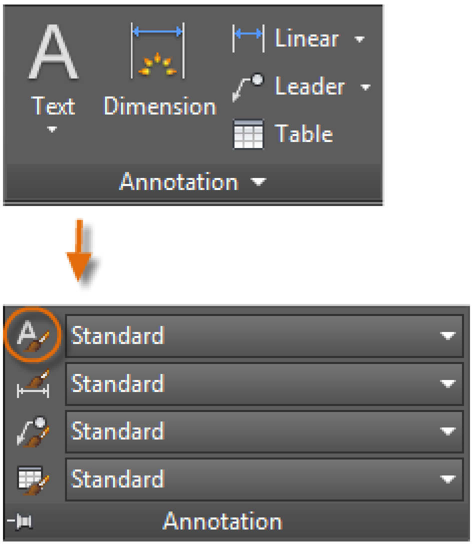


Fig. (108)

When you create a new text style, you will give it a name and then choose a font and a font style. The order in which you click the buttons is shown below: As shown in Fig. (109).

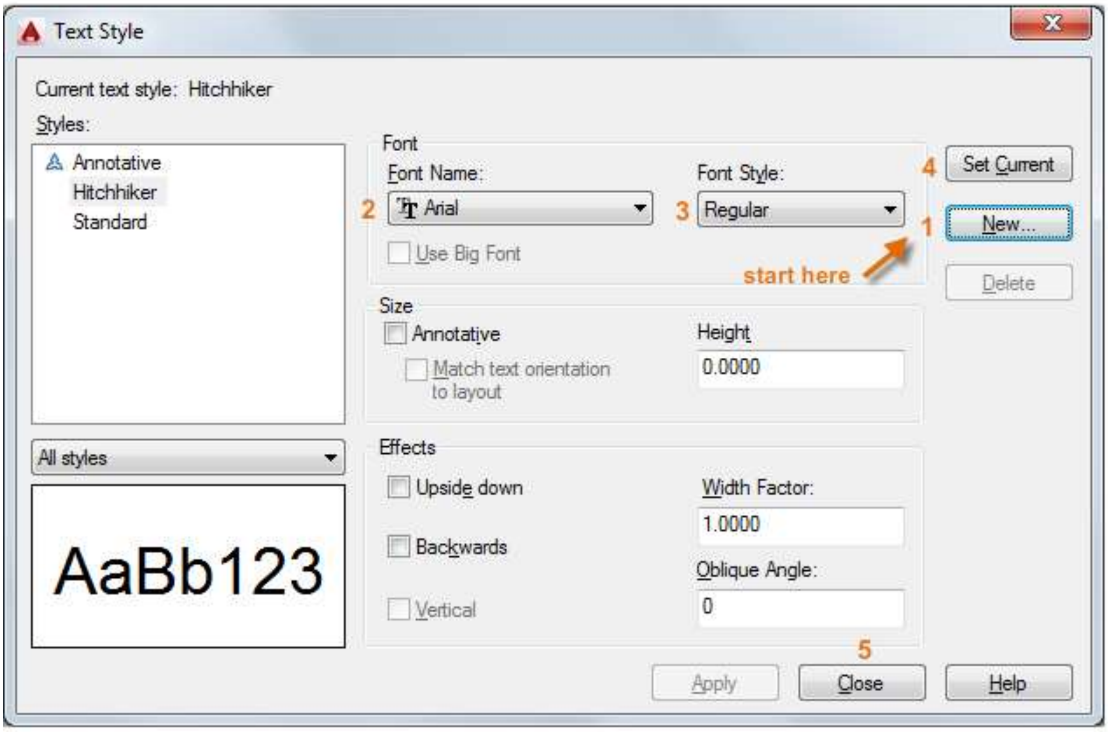


Fig. (109)

Here's a Tip:

Save any new or changed text styles in your drawing template files. This will save you time by making them available in all new drawings.

<u>Multileaders</u>

Multileader objects are used to create text with leader lines such as general labels, reference labels, bubbles, and callouts. As shown in Fig. (110).

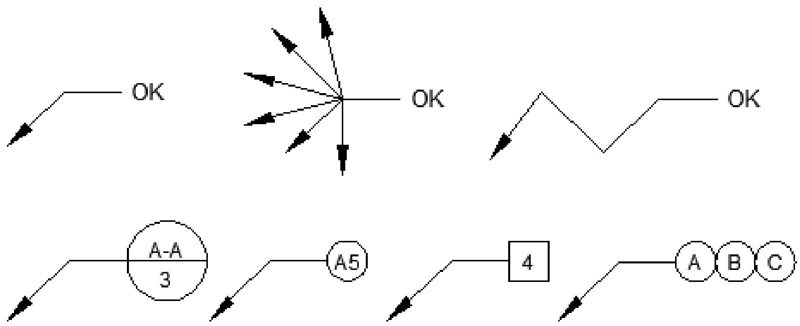


Fig. (110)

Create a Multileader

To create a multileader, use the **MLEADER** command. Click the Multileader tool in the Annotation panel or enter **MLD** in the Command window. Follow the prompts and options in the Command window. Feel free to experiment.
After you create a multileader, select it and then modify it by clicking and moving its grips. As shown in Fig. (111).

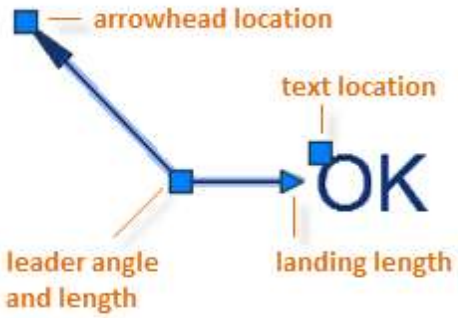


Fig. (111)

Grip menus appear when you hover over arrowhead and leader grips. From these menus, you can add leader segments or additional leaders. As shown in Fig. (112).

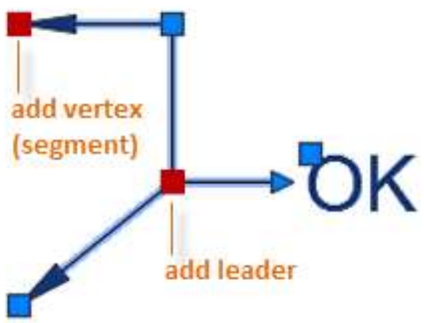


Fig. (112)

You can edit the text in a multileader by double-clicking it. As shown in Fig. (103).

Create a Multileader Style

You can create your own multileader styles from the drop-down list in the expanded Annotation panel, or by entering **MLEADERSTYLE** in the Command window. As shown in Fig. (113).

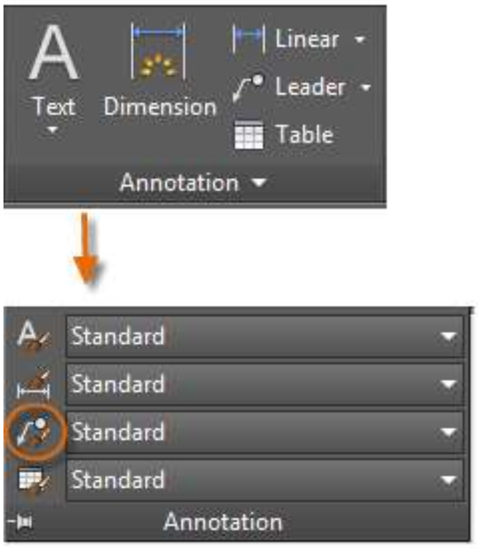


Fig. (113)

For example, to create a “detail callout” style, start the **MLEADERSTYLE** command. In the Multileader Styles Manager, click New and choose a descriptive name for the new multileader style. Click the Content tab, choose Block, and then Detail Callout as shown. As shown in Fig. (114).

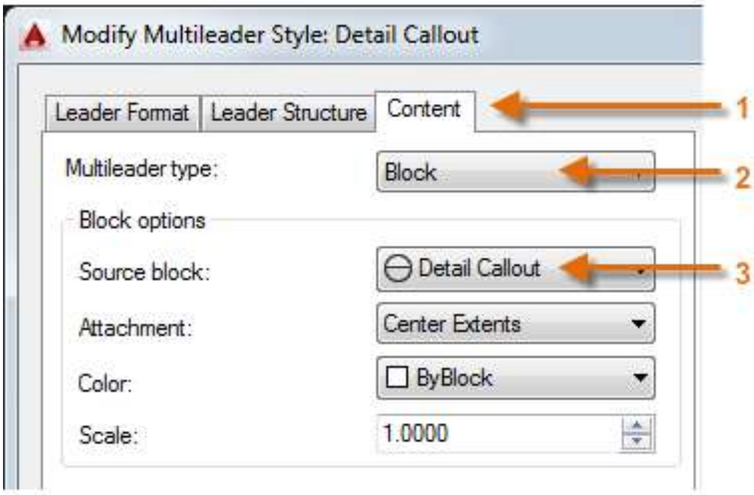


Fig. (114)

GUIDE TO AUTOCAD BASICS:

NOTES AND LABELS

NOTE: As with text styles, once you create one or more multileader styles, save them in your drawing template files.

Dimensions

- Create several types of dimensions and save dimension settings by name.
- Here is an example of several types of dimensions using an architectural dimension style with imperial units. As shown in Fig. (115).

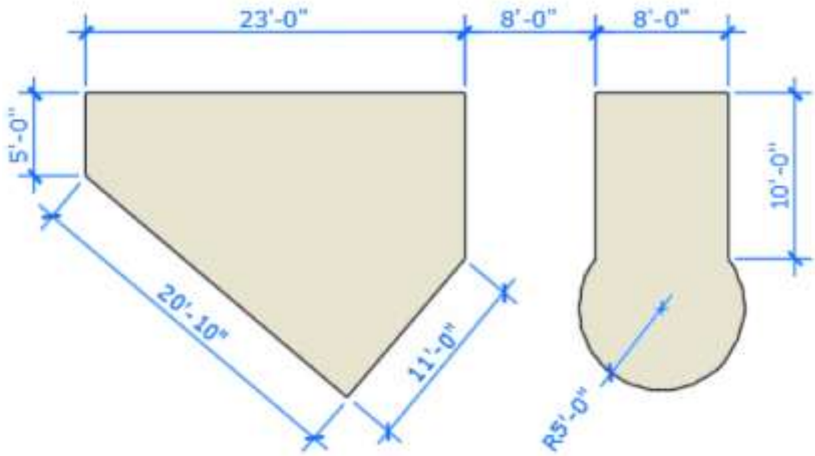


Fig. (115)

Linear Dimensions

You can create horizontal, vertical, aligned, and radial dimensions with the **DIM** command. The type of dimension depends on the object that you select and the direction that you drag the dimension line. As shown in Fig. (116).

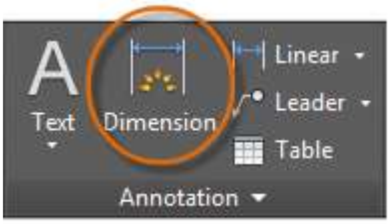


Fig. (116)

The following illustration demonstrates one method for using the **DIM** command. Once you start the command, press Enter or the Spacebar, select the line (1), and then click the location of the dimension line (2). As shown in Fig. (117)

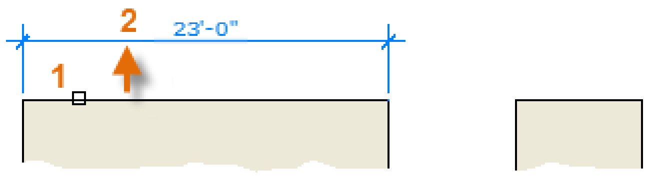


Fig. (117)

For the 8'-0" dimension below, you use another method. You start the DIM command, click two endpoints (1 and 2) and then the location of the dimension line (3). To line up the dimension lines point 3 was snapped to the endpoint of the previously created dimension line. As shown in Fig. (118).

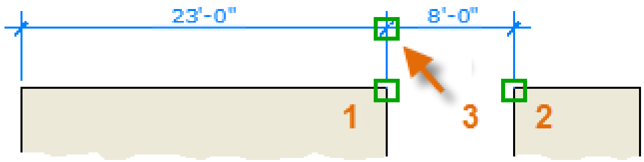


Fig. (118)

Use the **DIM** command to create dimensions that are parallel to an object by dragging the dimension line at an angle rather than horizontally or vertically. As shown in Fig. (119).

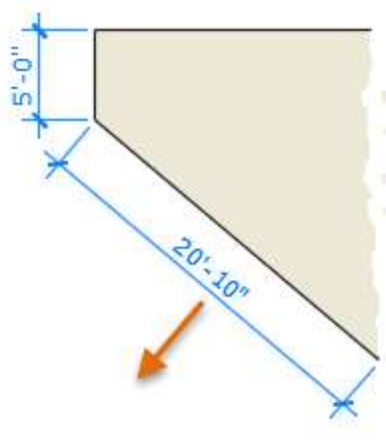


Fig. (119)

Here's a Tip:

*If points 1 and 2 are not on the same horizontal line, press Shift to force the dimension line to be horizontal. In addition, if the building or part being dimensioned is at an angle, enter **DIMROTATED** for that case.*

Another Tip:

Because it is easy to accidentally snap to the wrong feature or part of a dimension object, be sure to zoom in closely enough to avoid mistakes.

Modify Dimensions

For simple adjustments to dimensions, nothing is faster than using grips.

In this example, you select the dimension to display its grips. Next, click the grip on the dimension text and drag it to a new location, or click one of the grips at the end of the dimension line and drag the dimension line. As shown in Fig. (120).

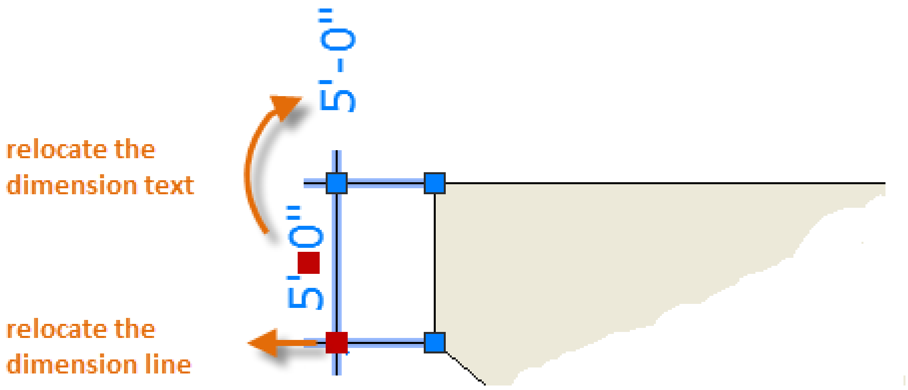


Fig. (120)

Here's a Tip:

If the changes are more complicated than this, it might be faster simply to delete and then recreate the dimension.

Dimension Styles

Dimension styles help establish and enforce drafting standards. There are many dimension variables that can be set with the **DIMSTYLE** command to control virtually every nuance of the appearance and behavior of dimensions. All these settings are stored in each dimension style.

The default dimension style is named either Standard (imperial) or ISO-25 (metric). It is assigned to all dimensions until another style is set as the current dimension style.

The current dimension style name, Hitchhiker in this case, is displayed in the drop-down list of the Annotation panel. As shown in Fig. (121).

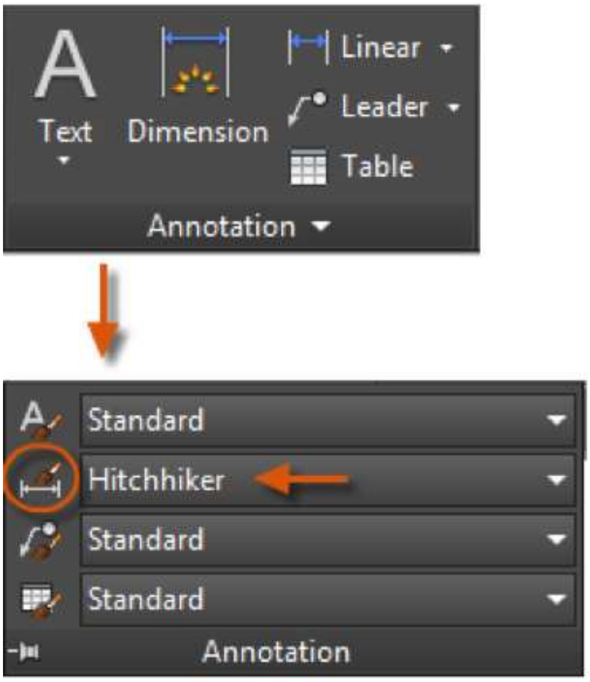


Fig. (121)

To open the Dimension Style Manager, click the indicated button. You can create dimension styles that match nearly any standard, but you will need to invest time into specifying them completely. For this reason, you should save any dimension styles that you create in one or more drawing template files. As shown in Fig. (122).

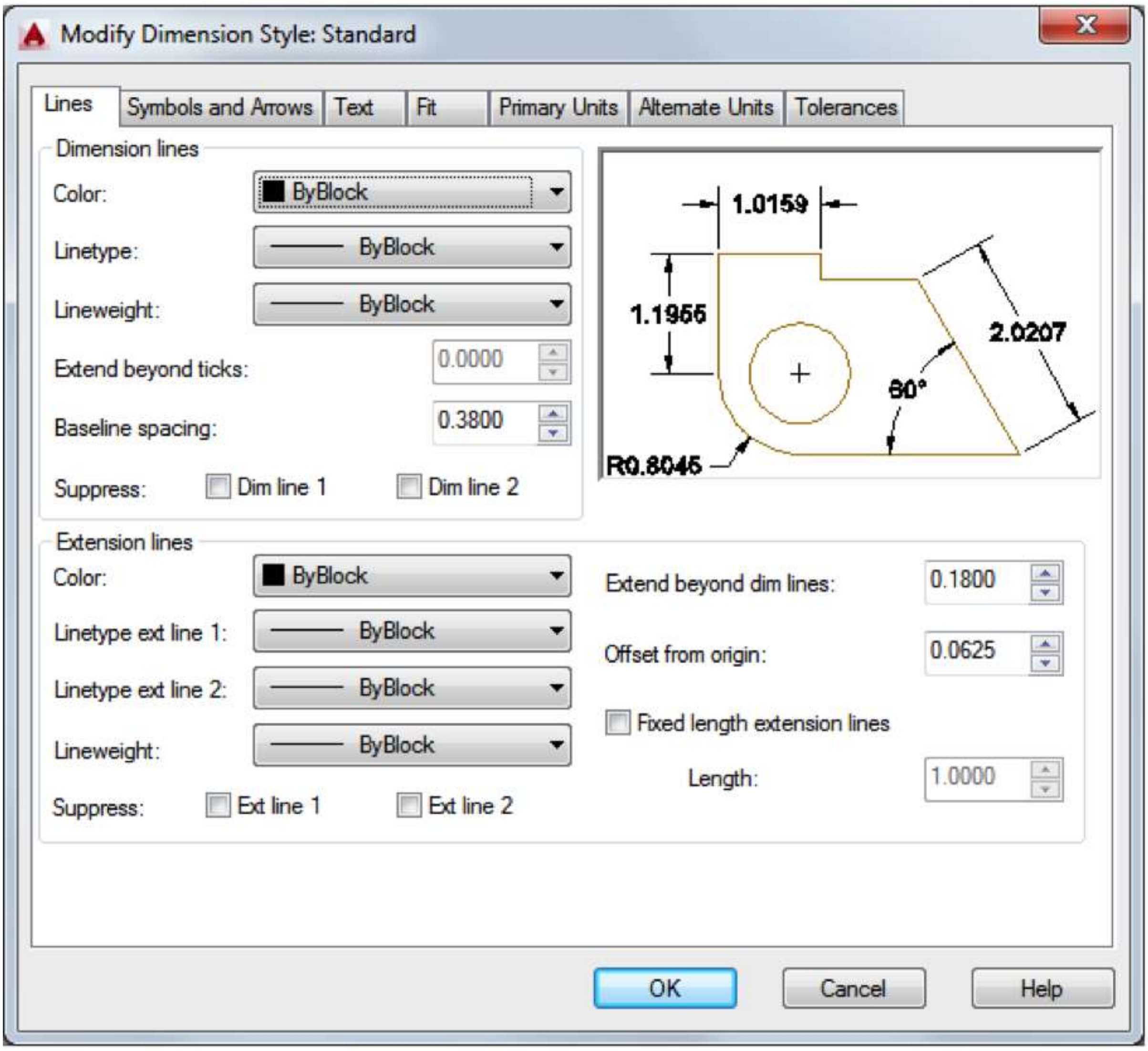


Fig. (122)

Recommendations

- When you save a dimension style, choose a descriptive name.
- If applicable, check with your CAD manager regarding existing dimension style standards and drawing template files.

Printing

Output a drawing layout to a printer, a plotter, or a file. Save and restore the printer settings for each layout. Originally, people *printed* text from printers and *plotted* drawings from plotters. Now, you can perform both with either device, so this guide will use the terms 'print' and 'plot' interchangeably.

The command to output a drawing is **PLOT** and you can access it from the Quick Access toolbar. As shown in Fig. (123).

Fig. (123)

To display all of the options in the Plot dialog box, click the More Options button. As shown in Fig. (124).

Fig. (124)

As you can see, there are many settings and options available. As shown in Fig. (125).

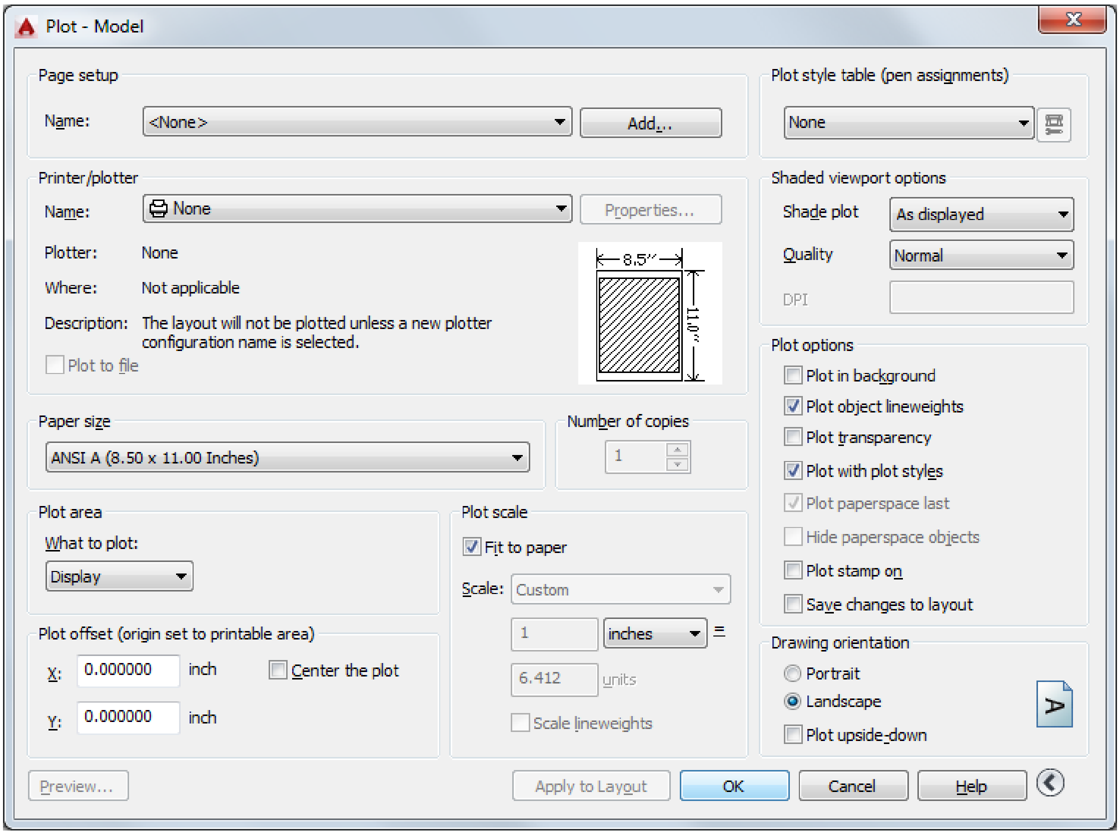


Fig. (125)

For the sake of convenience, you can save and restore collections of these settings by name. These are called *page setups*. With page setups you can store the settings that you need for different printers, printing in grayscale, creating a PDF file from your drawing, and so on.

Create a Page Setup

To open the Page Setup Manager, right-click on the Model tab or a layout tab and choose Page Setup Manager. The command is **PAGESETUP**.

Each layout tab in your drawing can have an associated page setup. This is convenient when you use more than one output device or format, or if you have several layouts with different sheet sizes in the same drawing. As shown in Fig. (126).

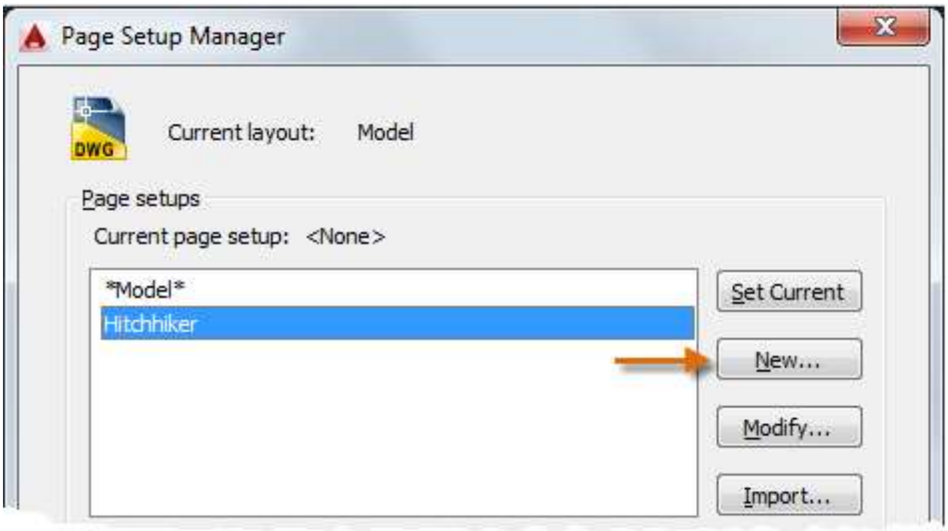


Fig. (126)

To create a new page setup, click New and enter the name of the new page setup. The Page Setup dialog box that displays next looks like the Plot dialog box. Choose all the options and settings that you wish to save.

When you are ready to plot, simply specify the name of the page setup in the Plot dialog box, and all your plot settings will be restored. In the following illustration, the Plot dialog box is set to use the Hitchhiker page setup, which will output a DWF (Design Web Format) file rather than print to a plotter. As shown in Fig. (127).

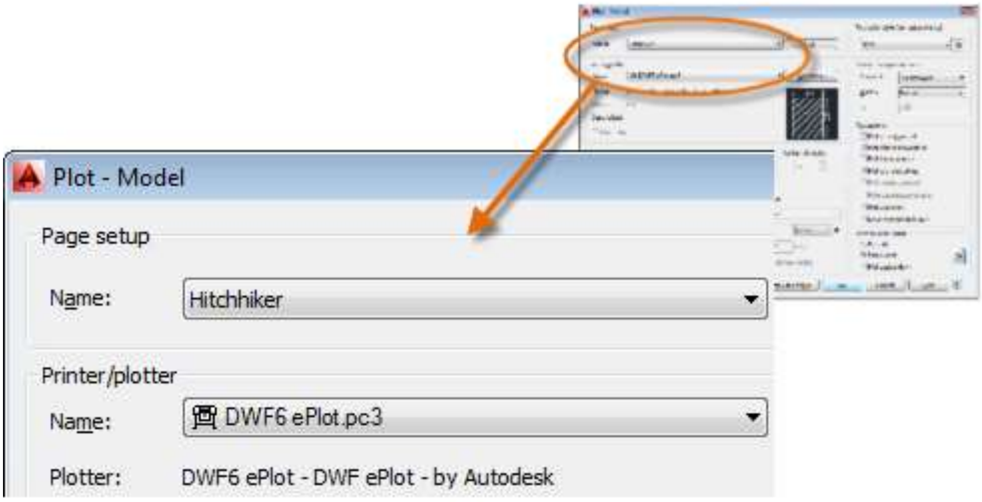


Fig. (127)

Here's a Tip:

You can save page setups in your drawing template files, or you can import them from other drawing files.

Output to a PDF File

The following example shows you how to create a page setup for creating PDF files.

From the Printer/plotter drop-down list, choose *AutoCAD PDF (General Documentation).pc3*: As shown in Fig. (128).

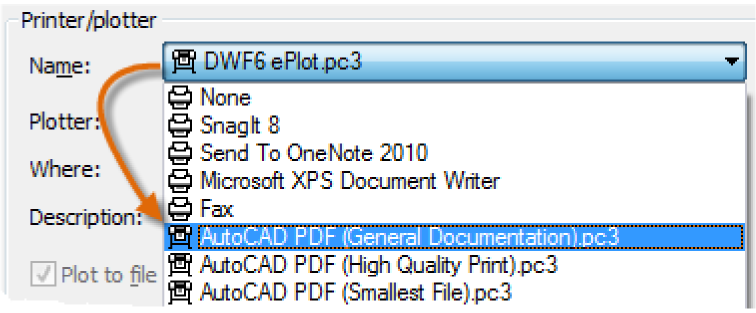


Fig. (128)

Next, choose the size and scale options that you want to use:

- **Paper Size:** The orientation (portrait or landscape) is built into the choices in the drop-down list.

- **Plot Area:** You can clip the area to be plotted with these options, but typically everything will be plotted.
- **Plot Offset:** This setting changes based on your printer, plotter, or other output. Try centering the plot or adjusting the origin, but remember that printers and plotters have a built-in margin around the edges.
- **Plot Scale:** Choose your plot scale from the drop-down list. A scale such as 1/4” = 1’-0” is meant for printing to scale from the Model tab. On a layout tab, you normally print at a 1:1 scale.

  The plot style table provides information about processing colors. Colors that look good on your monitor might not be suitable for a PDF file or for printing. For example, you might want to create a drawing in color, but create monochrome output. Here is how you specify monochrome output: As shown in Fig. (129).

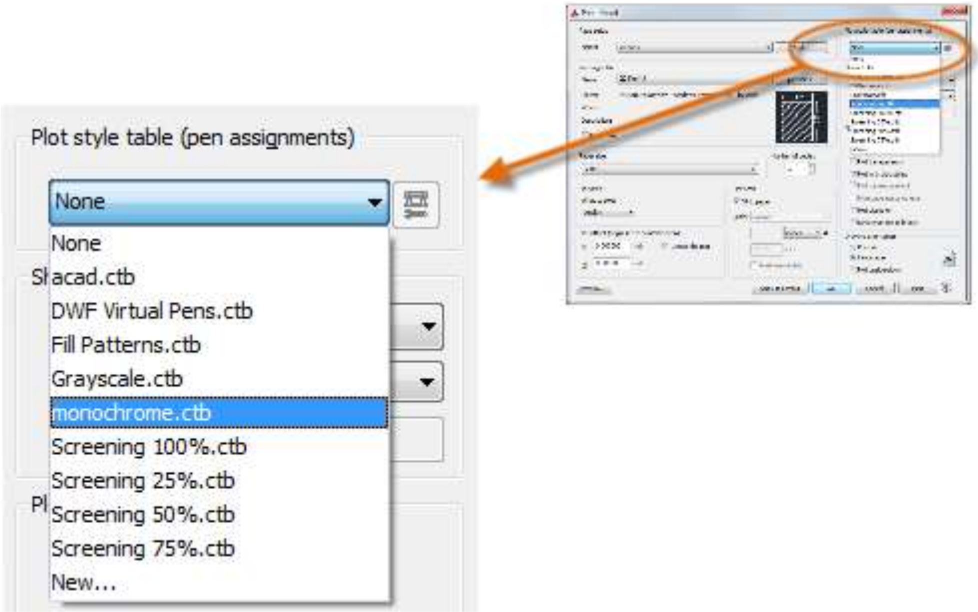


Fig. (129)

Here’s a Tip:

Always double-check your settings with the Preview option. As shown in Fig. (130).

Fig. (130)

The resulting Preview window includes a toolbar with several controls, including Plot and Exit.

After you are satisfied with your plot settings, save them to a page setup with a descriptive name such as “PDF-monochrome.” Then, whenever you want to output to a PDF file, all you need to do is click Print, choose the PDF-monochrome page setup, and click OK.

Recommendations

If you want to share a static image of your drawing, you can output a PDF file from a drawing file.

If you want to include additional data from your drawing, use DWF (Design Web Format) files instead.

If you want to review an AutoCAD drawing file with a person in a different location, consider using Autodesk A360 and the AutoCAD 360 web and mobile applications, which you can access from the Autodesk website.

Refrences

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[2] C. H. Simmons and D. E. Maguire, *Manual of Engineering Drawing*, 2nd ed. Elsevier, 2004.

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[5] Textbook of Engineering Drawing- prof.Dr. N.D.BHATT