H. T. I.

THE HIGHER TECHNOLOGICAL INSTITUTE – TENTH OF RAMADAN CITY

Lecture notes in

Engineering Drawing & Projection (1)

Course Code: ENG 003

Course Name: Engineering Drawing & Projection (1)

By

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Preface

Engineering drawing is the graphic language from which a trained person can visualize objects. It is a two dimensional representation of three dimensional objects. In general, it provides necessary information about the shape, size, surface quality, material, manufacturing process, etc., of the object. It is a language between designers and manufacturer to communicate.

Drawings prepared in one country may be utilized in any other country irrespective of the language spoken. Hence, engineering drawing is called the universal language of engineers. Any language to be communicative should follow certain rules so that it conveys the same meaning to everyone.

Similarly, drawing practice must follow certain rules, if it is to serve as a means of communication by graphical representation of objects and structures to be done using freehand, mechanical, or computer methods.

This work covers the basic aspects of engineering drawing practice required for first level in engineering. The tools needed for engineering drawing, with some basic exercises that help student in applying and using standard, are presented. Specific drawing exercise, for many geometric problems as geometrical construction which is necessary part of engineering design and analysis, are shown. Examples of two and three-dimensional geometries are provided to visualize shape and draw views with a high degree of fluency by learning theory of projection. Best wishes for all who studies this language, to success in constructing and presenting engineering drawing.

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مقدمة

الرسم الهندسي هو اللغة التوضيحية التي يستطيع من خلالها الشخص المتدرب تصور الأشياء. و تكون ثنائية الأبعاد لأشياء ثلاثية الأبعاد. بشكل عام، يوفر المعلومات الضرورية حول الشكل والحجم وجودة السطح والمواد وعملية التصنيع وما إلى ذلك. إنها لغة للتواصل بين المصممين والشركة المصنعة.

يمكن استخدام الرسومات المعدة في دولة واحدة لأي دولة أخرى بغض النظر عن اللغة المستخدمة. ومن هنا يطلق على الرسم الهندسي اللغة العالمية للمهندسين. أي لغة للتواصل يجب أن تتبع قواعد معينة حتى تنقل نفس المعنى للجميع.

وبالمثل، يجب أن تتبع ممارسة الرسم قواعد معينة، و هي وسيلة اتصال عن طريق التمثيل التوضيحي للأشياء والهياكل التي سيتم تنفيذها بإستخدام أساليب يدوية أو ميكانيكية أو حاسوبية.

يغطي هذا العمل الجوانب الأساسية لممارسة الرسم الهندسي المطلوبة للمستوى الأول في الهندسة. يتم عرض الأدوات اللازمة للرسم الهندسي، مع بعض التمارين الأساسية التي تساعد الطالب في تطبيق واستخدام المعيار. يتم عرض تمرين رسم محدد للعديد من المشكلات الهندسية مثل البناء الهندسي الذي يعد جزءًا ضروريًا من التصميم والتحليل الهندسي.

يتم تقديم أمثلة على الأشكال الهندسية ثنائية وثلاثية الأبعاد لتصور الشكل ورسم المناظير بدرجة عالية من الطلاقة من خلال تعلم نظرية الإسقاط.

أتمنى لكل من يدرس هذه اللغة التوفيق في بناء وتقديم الرسم الهندسي بأفضل صورة ممكنه.

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Chapter 1

Introduction

1.1. Drawing Sheets

Engineering drawings are prepared on standard size drawing sheets. The correct shape and size of the object can be visualized from the understanding of not only its views but also from the various types of lines used, dimensions, notes, scale etc.

Designations of sizes dimensions are given in Table 1 and their sizes are given in Fig. 1.1.

For class work use of A3 size drawing sheet is preferred.

Table 1. Drawing sheets dimensions:

Designation	Size (millimetres)	Area
A0	841 × 1189	1 m^2
A1	594 × 841	5000 cm^2
A2	420 × 594	2500 cm^2
A3	297 × 420	1250 cm^2
A4	210 × 297	625 cm^2

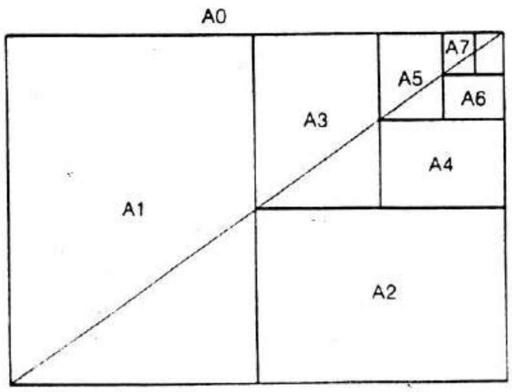


Fig.1.1 Drawing sheets sizes

Drawing sheets should be presented in one of the following formats:

(a) Landscape: presented to be viewed with the longest side of the sheet horizontal.

(b) Portrait: presented to be viewed with the longest side of the sheet vertical.

C. Compasses and Dividers

At least two compasses are needed: a small spring bow compass for small circles and a large bow for larger circles or arcs.

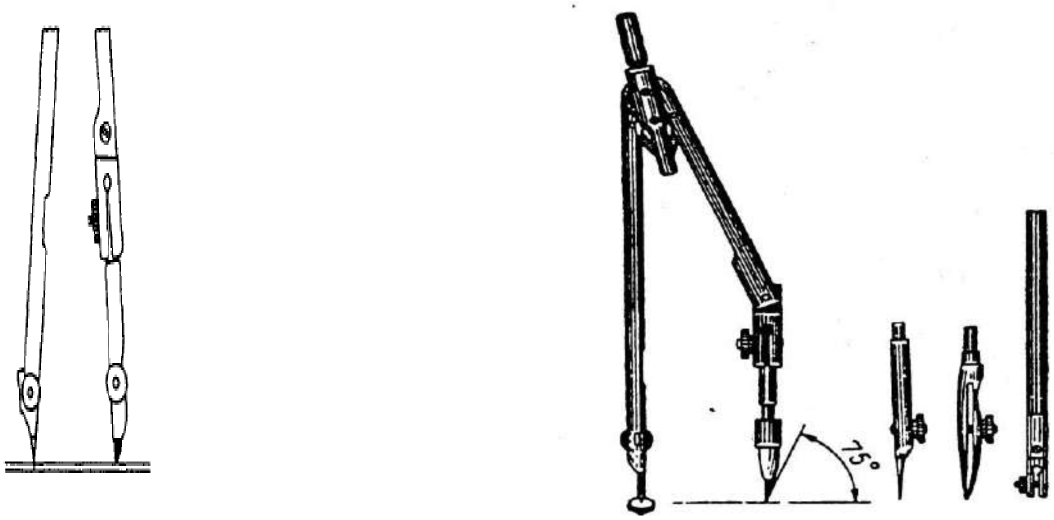


Fig.1.2 Positioning of the compass lead during drawing small and large circles

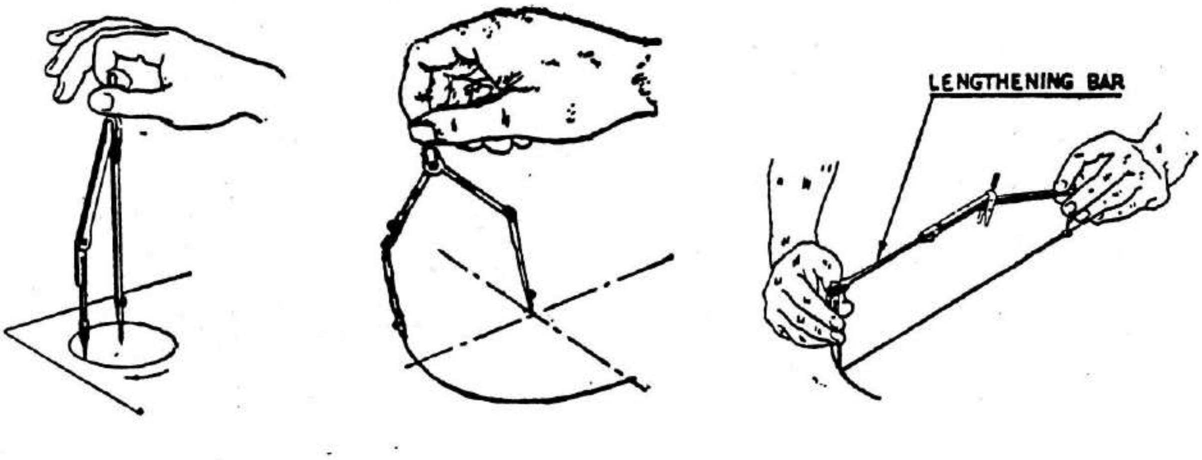


Fig.1.3 using the compass in drawing

The dividers are used in measurements transferring. Also, they could be used for lines and curves dividing into equal parts.

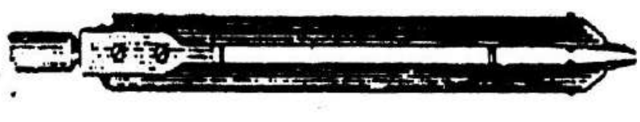


Fig.1.4 Plan divider

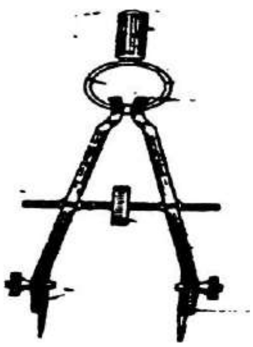


Fig.1.5 Bow divider

D. Set Squares

At least two set squares are needed: a 60 °- 30° and a 45 ° set squares. It will be also useful to have an adjustable set square, which will enable you to set the angle on the set square to anywhere between 0 ° and 90 °. If you have an adjustable set square you can manage without theother two.

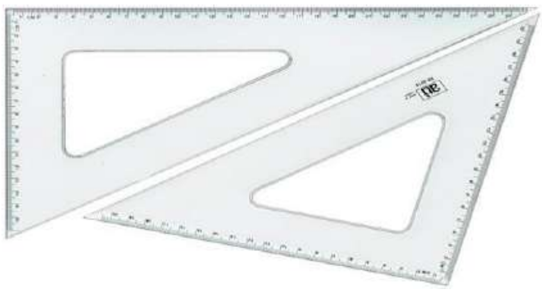


Fig. 1.6 Set Squares

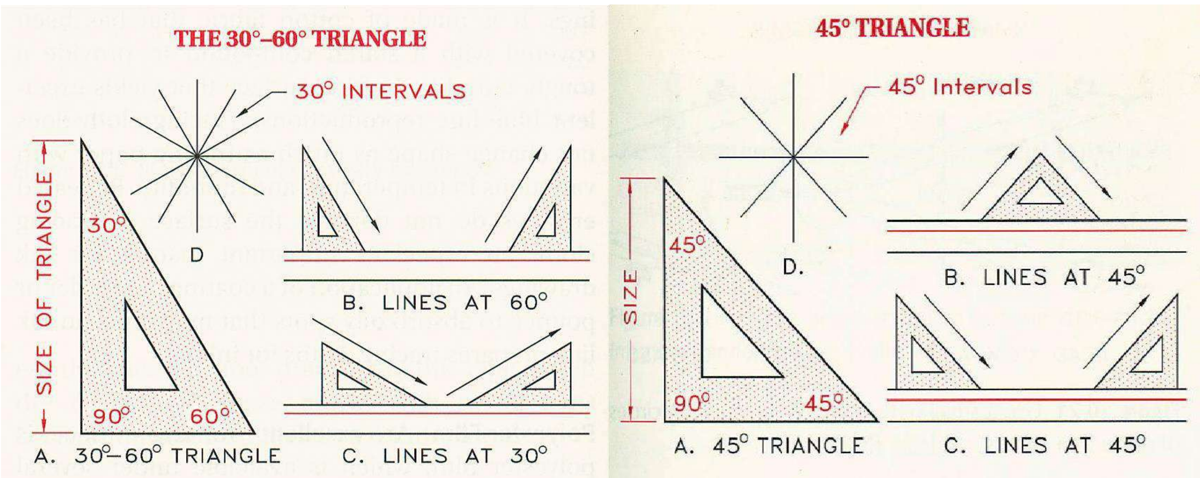


Fig. 1.7 Set square uses to draw lines with different angels

E. Pencils

A selection of mechanical pencils, 0.5 mm pencil, (6H) can be used for light lines, a softer pencil (2H) for the outlines and an even softer pencil (HB) for printing.

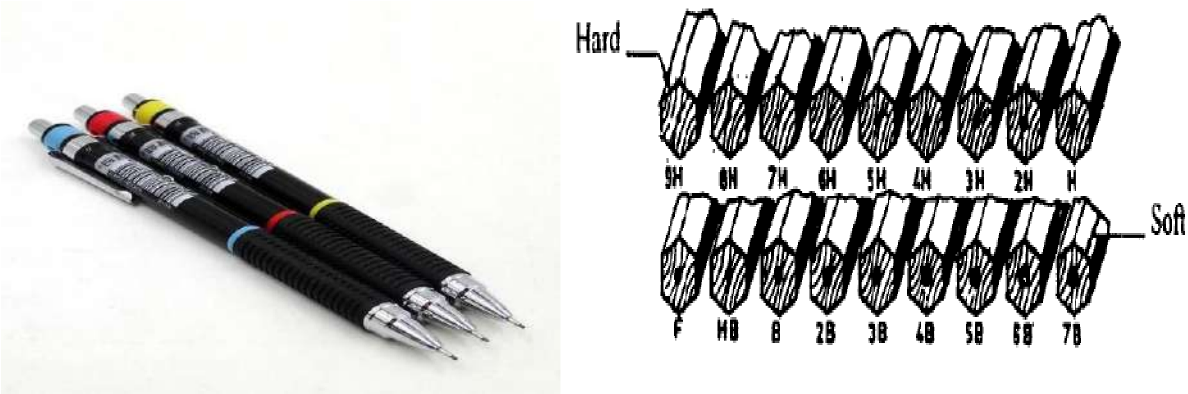


Fig.1.8 Mechanical Pencils, with Hardness and softness scale.

F. Ruler (Scale)

Scales are used to make drawing of the objects to a desired proportionate size. They are made of wood, steel or plastic. It is advisable to have a transparent ruler or scale.

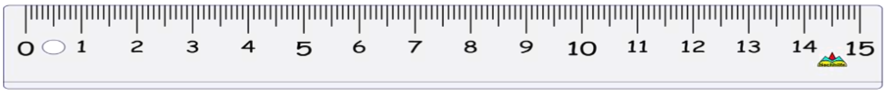


Fig.1.9 Scale

G. Rubber

Choose a good quality rubber, one that does not smudge.

Fig.1.10 Rubber

H. Clips or tape

The best tape to use, to hold paper on the drawing board, is masking tape but metal drawing board clips are easier to use.

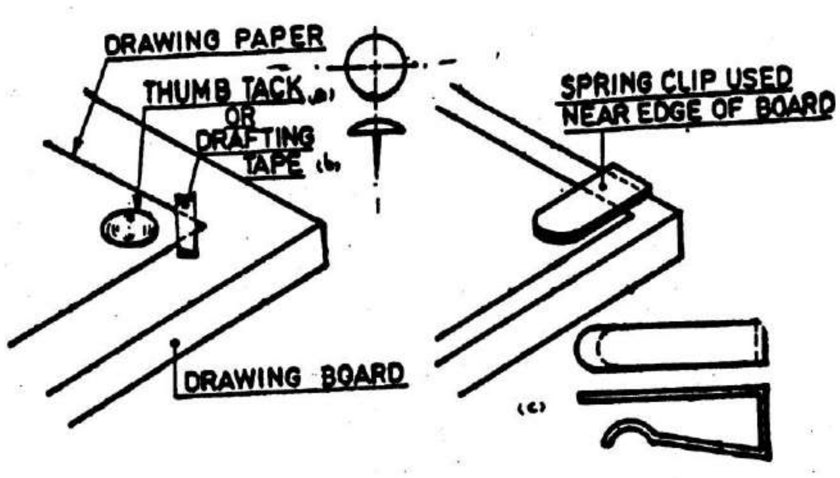


Fig. 1.11 Clips and tape

1.2 Lines Representations

In engineering drawing, the details of various objects are drawn by different types of lines. Each line has a definite meaning represent its applications. Lines should be black, bold, and consistent in thickness. The lines that are commonly represent the engineering drawing are illustrated in the following table.

Table 2: Drawing Lines Representations:

Example	Description & Representation	Application
A	Continuous wide line	Visible edges and outlines
B	Continuous narrow line	1 Dimension, extension and projection lines 2 Hatching lines for cross sections 3 Leader and reference lines 4 Outlines of revolved sections 5 Imaginary lines of intersection 6 Short centre lines 7 Diagonals indicating flat surfaces 8 Bending lines 9 Indication of repetitive features
C	Continuous narrow irregular line	Limits of partial views or sections provided the line is not an axis

D	Dashed narrow line	Hidden outlines and edges
E	Long dashed dotted narrow line	1 Centre lines. 2 Lines of symmetry 3 Pitch circle for gears 4 Pitch circle for holes
F	Long dashed dotted wide line	Surfaces which have to meet special requirements
G	Long dashed dotted narrow line with wide line at ends and at changes to indicate cutting planes	Note BS EN ISO 128-24 shows a long dashed dotted wide line for this application
H	Long dashed double dotted narrow line	1 Preformed outlines 2 Adjacent parts 3 Extreme positions of moveable parts 4 Initial outlines prior to forming 5 Outline of finished parts 6 Projected tolerance zones
J	Continuous straight narrow line with zig zags	Limits of partial or interrupted views; Suitable for CAD drawings provided the line is not an axis

1.3 Lettering and Dimensioning

A. Lettering

Lettering is defined as writing of titles, sub-titles, dimensions, etc., on a drawing.

Lettering in drawing should be in CAPITALS (i.e., Upper-case letters). Lower-case (small) letters are used for abbreviations like mm, cm, etc. Size of Letters is measured by the height h ofthe CAPITAL letters as well as numerals.

Standard heights for CAPITAL letters and numerals recommended by BIS are given below:

1.8, 2.5, 3.5, 5, 6, 10, 14 and 20 mm.

Note that, Size of the letters may be selected based upon the size of drawing.

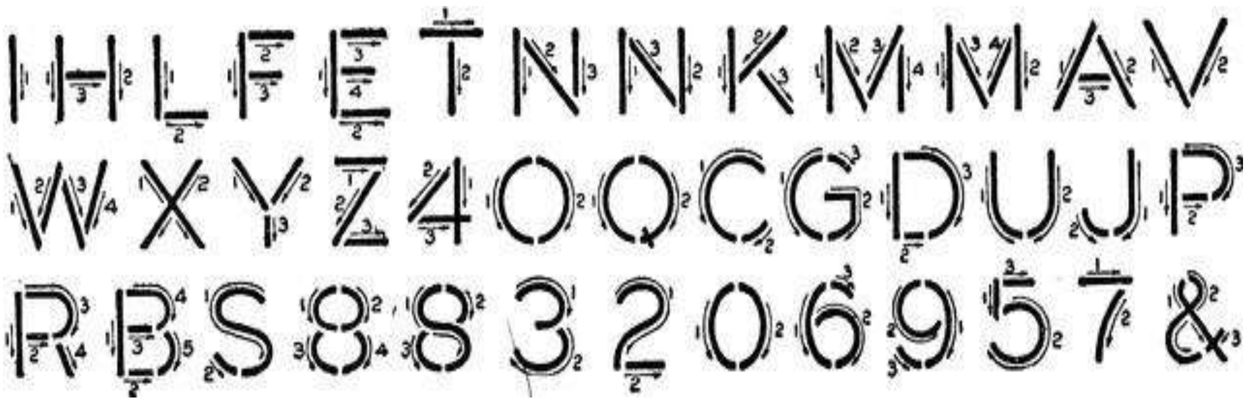


Fig.1.12. Letters and numbers writing technique

B. Dimensions

Dimensions are consists of a thin, solid line that shows the extent and direction of a dimension, fig.1.13. Dimensions lines are broken for insertion of the dimension numbers. It should be placed at least 10 mm away from the outline and all other parallel dimensions should be at least 6 mm apart, or more, if space permits fig. 1.13. It consist also arrows and dimensions. Arrows are 3 mm wide and should be 1/3rd as wide as they are long. These arrows are symbols placed at the end of dimension lines to show the limits of the dimension. Arrows are uniform in size and style, regardless of the size of the drawing. Various types of arrows used for dimensioning is shown in figure 1.14.

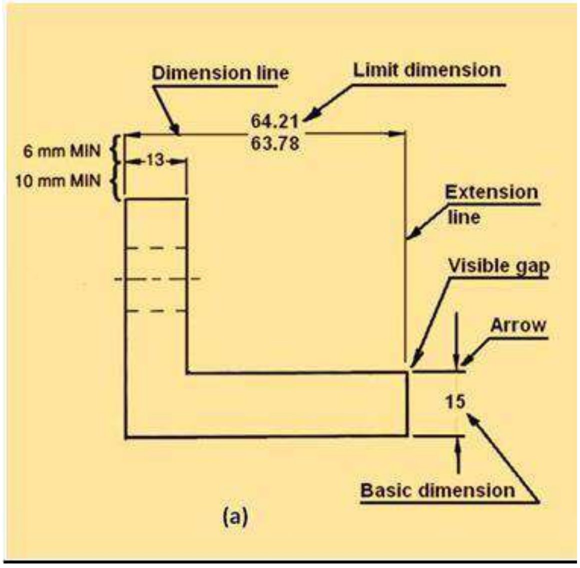


Fig.1.13 Specification of dimension lines.

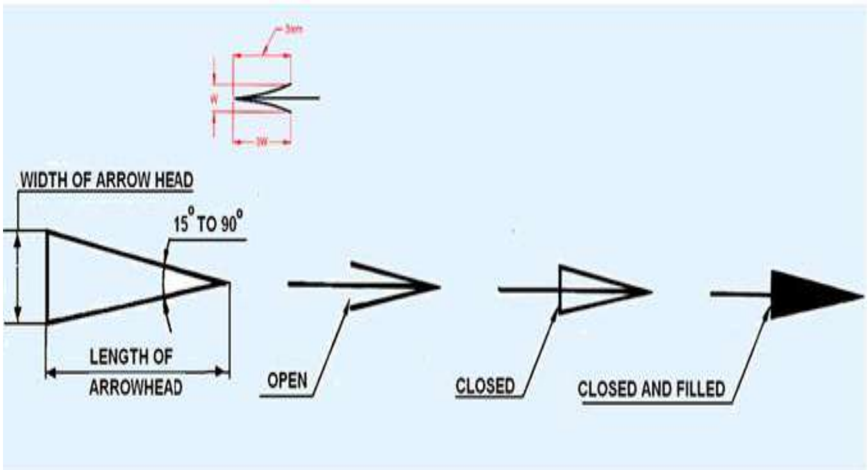


Fig. 1.14 Various types of arrows used for dimensioning.

Normally two types of dimensioning system exist, aligned system and the unidirectional system. These are shown in figure 1.15. In the aligned system the dimensions are placed perpendicular to the dimension line in such a way that it may be read from bottom edge or right hand edge of the drawing sheet. The horizontal and inclined dimension can be read from the bottom whereas all the vertical dimensions can be read from the right hand side of the drawing sheet. In the unidirectional system, the dimensions are so oriented such that they can be read from the bottom of the drawing. (see chapter 7)

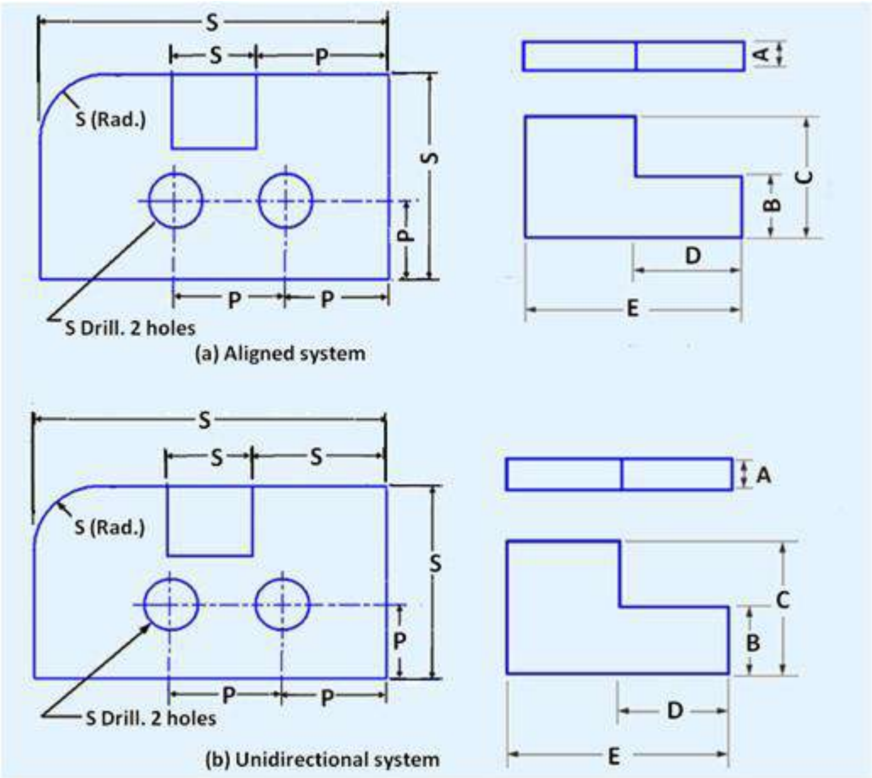


Fig. 1.15 The aligned system and unidirectional system of dimensioning.

1.4 Drawing Scales

It is not possible always to make drawings of an object to its actual size. If the actual linear dimensions of an object are shown in its drawing, the scale used is said to be a **full** size scale. Wherever possible, it is desirable to make drawings to full size. Objects which are very big in size cannot be represented in drawing to full size. In such cases the object is represented in reduced size by making use of reducing scales. Reducing scales are used to represent objects such as large machine parts, buildings, town plans etc. A reducing scale, say 1: 10 means that 10 units length on the object is represented by 1 unit length on the drawing. An enlarging scale, say 10: 1 means one unit length on the object is represented by 10 units on the drawing.

The designation of a scale consists of the word. SCALE, followed by the indication of its ratio as follows.

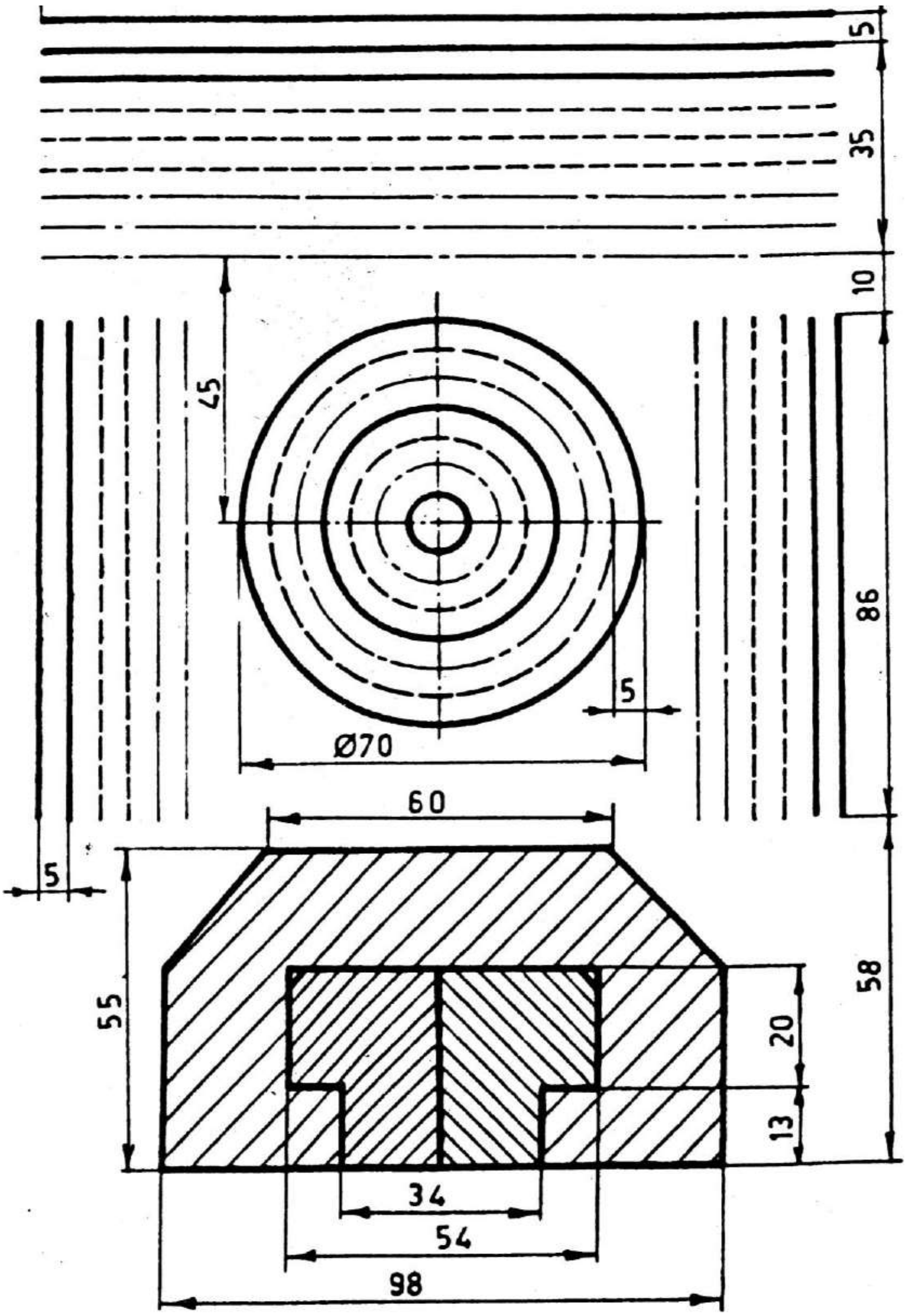
Scale 1: 1 for full size scale

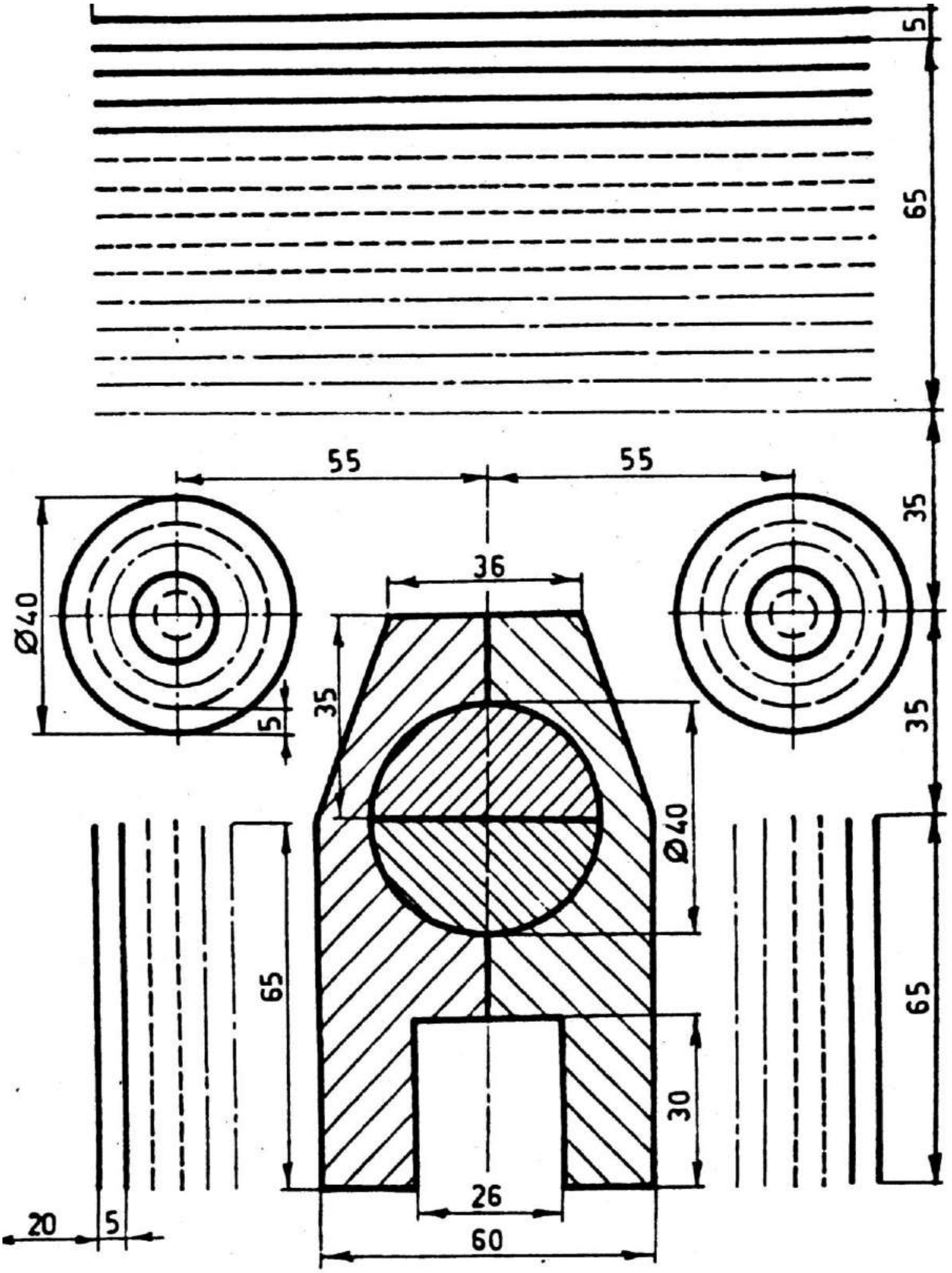
Scale 1: x for reducing scales (x = 10, 20........etc.)

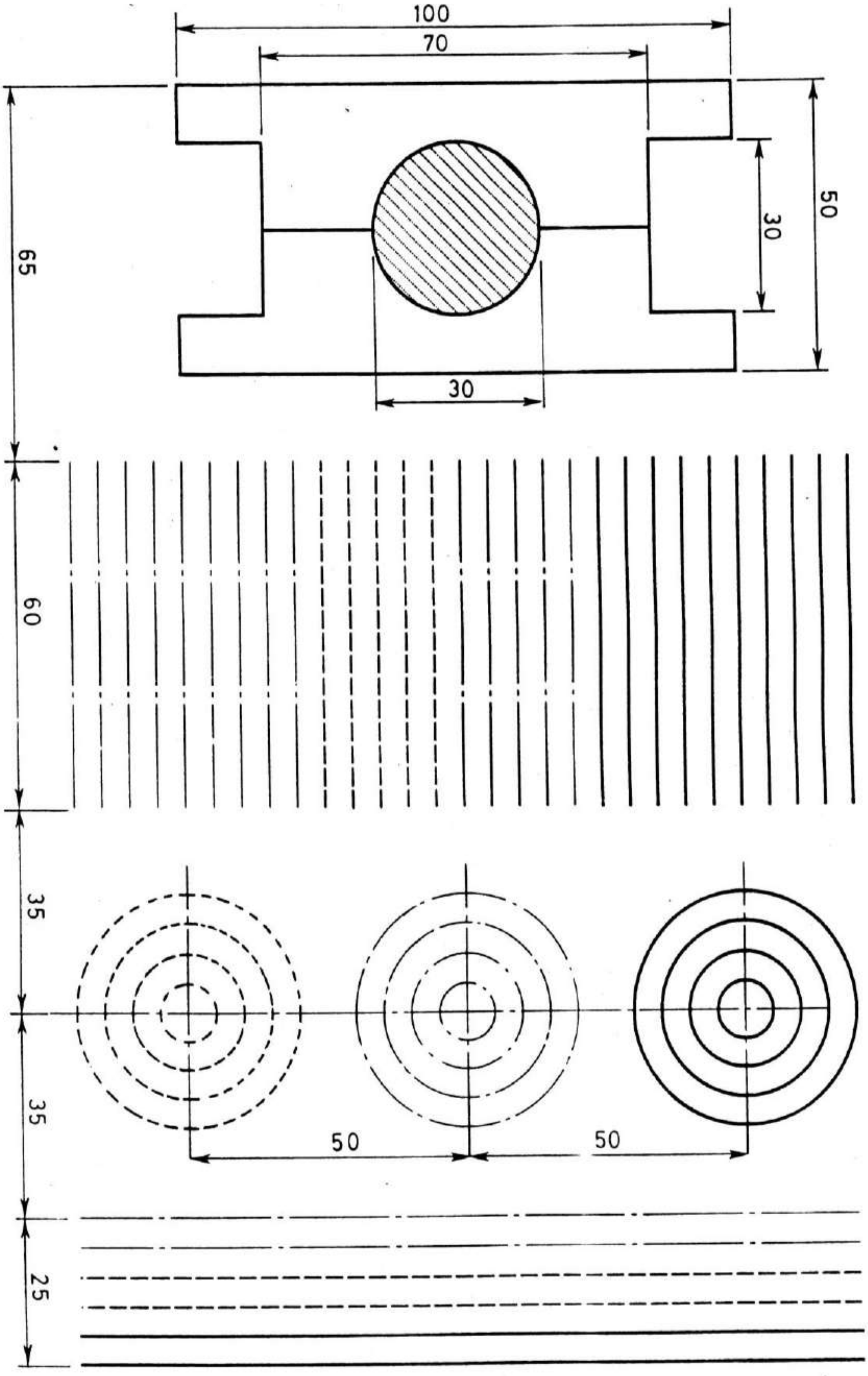
Scale x: 1 for enlarging scales.

Exercises

1. Draw the following by using instruments




5
65
55
55
35
36
Ø40
35
5
35
Ø40
65
65
30
26
60
20
5


100
70
50
30
65
30
60
35
35
50
50
25

2. Write Letters and numbers between the given two lines

ABCDEFGHIJKLMNOPQRST

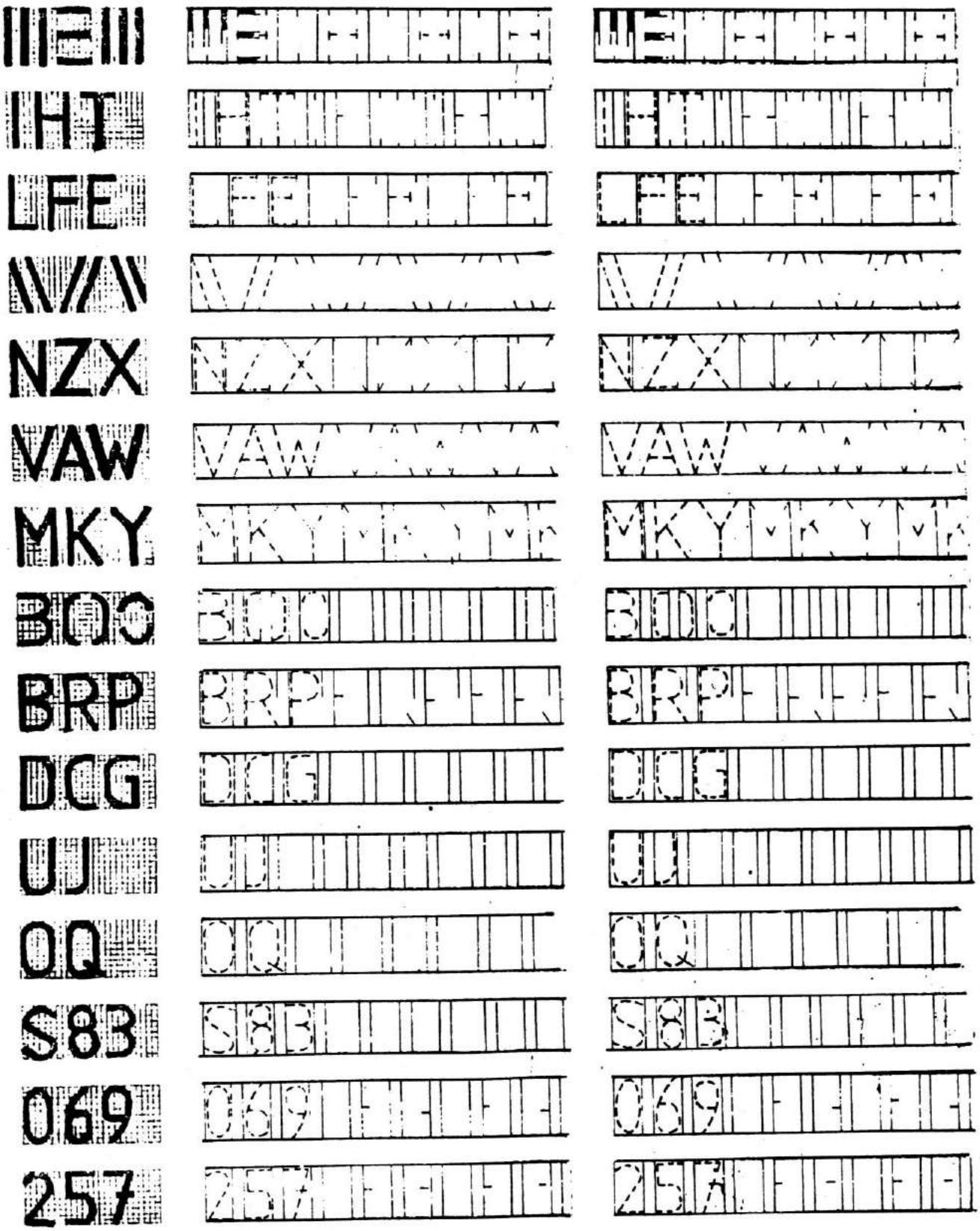
VWXYZABCDEFGHIJKLMNO

RSTUVWXYZBCDGOPRSUW

abcdefghijklmnopqrstuvwx

12345678901234567890123

1/2 3/4 5/6 7/8 8/9 2/3 0,1 2,3 4,5 6,7

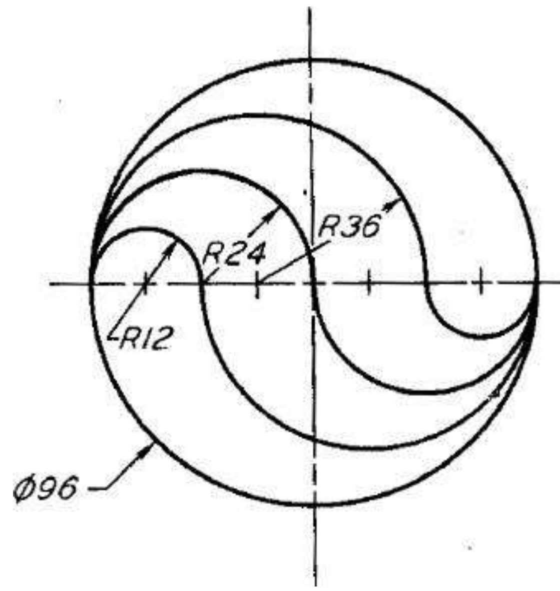

IHT
LFE
NZX
VAW
MKY
BRP
DCG
UJ
OQ
S83
069
257

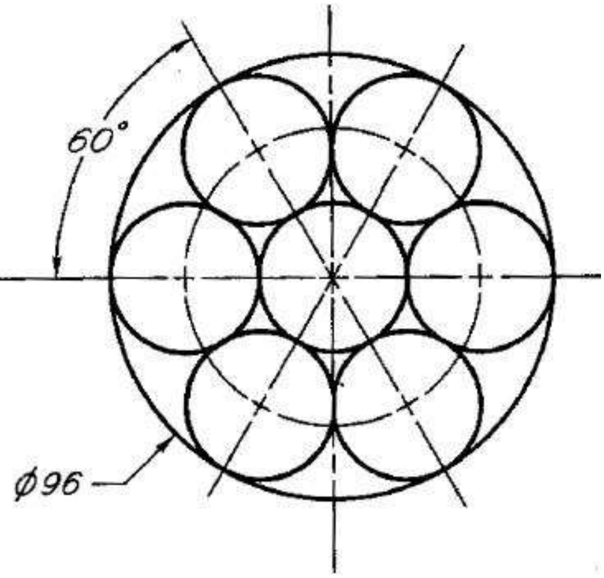
THE TECHNICAL LETTERS ARE STANDARDIZED.

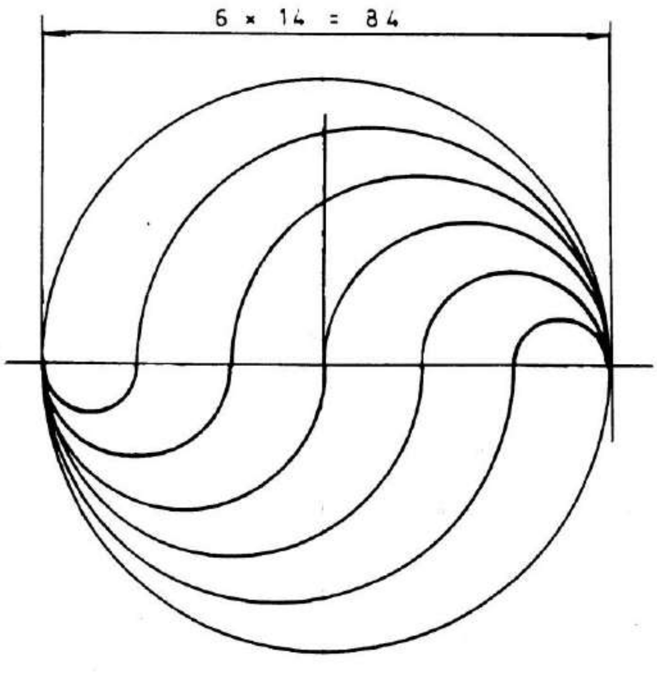
THE HEIGHTS OF LETTERS „h" ARE FROM 1mm UP TO 800mm. FOR LETTERING THE ENGINEERING DRAWING THE FOLLOWING HEIGHTS ARE MOSTLY USED h=2,5 ; 3,5 ; 5 , 7 ; 10 ; 14 ; 20mm. THE THICKNESS OF LINES OF LETTERS IS USUALLY th=1/10h. THE WIDTH OF CAPITAL LETTERS IS USUALLY s=7/10h, EXCEPT THAT OF LETTERS A,E,F,I,J,L,M,W (SEE INTO SHEET OF ALPHABETS). THE GAPS BETWEEN WORDS ARE m'=7/10h. THE GAPS BETWEEN LETTERS ARE m=2/10h. THE DISTANCE BETWEEN LINES (OF ROW) IS USUALLY r=6/10h.

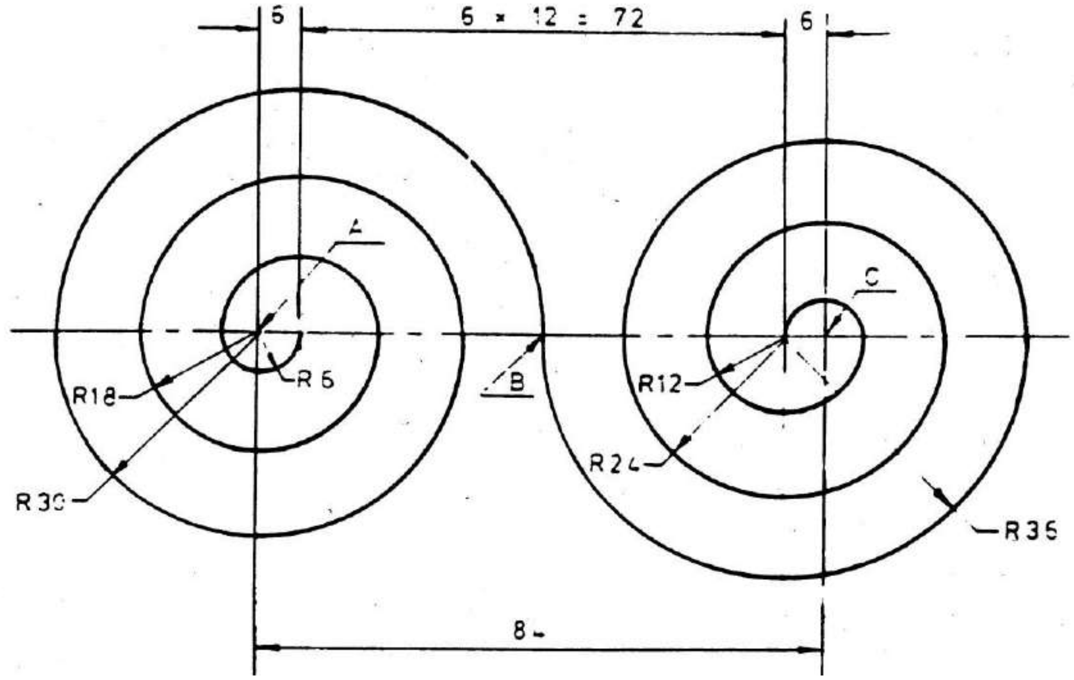
LETTERING OF ENGINEERING DRAWINGS MUST BE ELABORATED VERY CAREFULLY AND LEGIBLY.

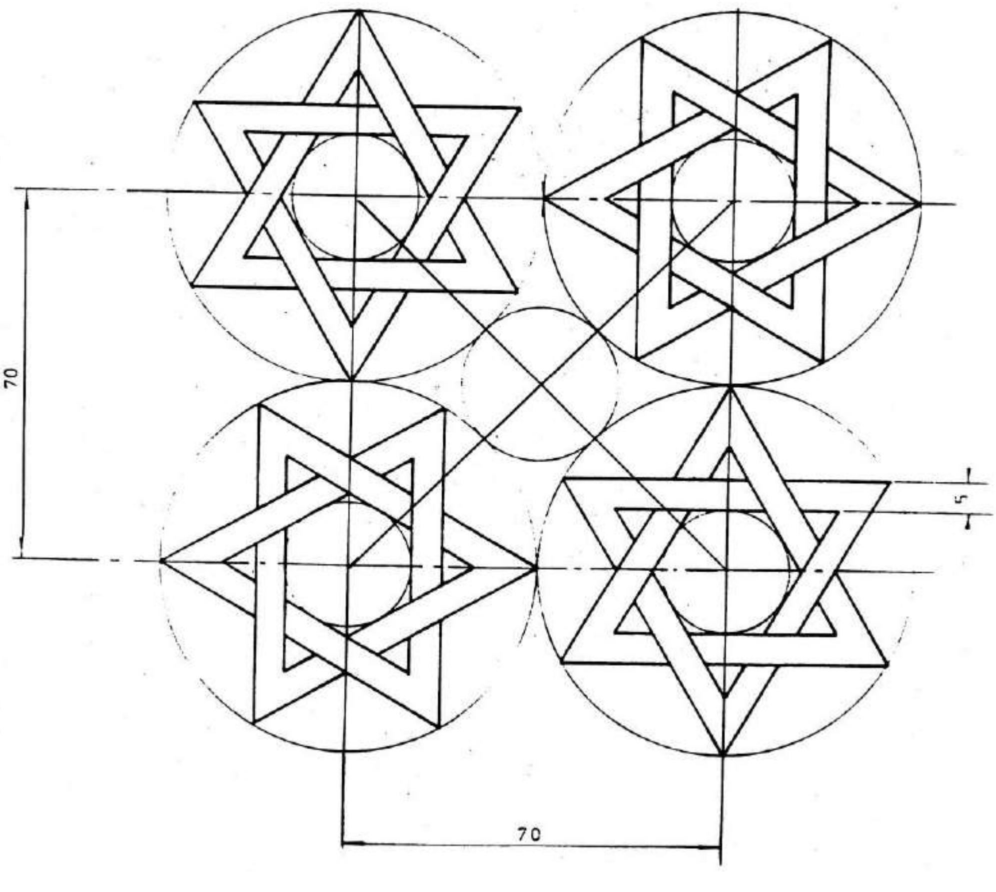
3. Draw the following Curves and circles








6
6 × 12 = 72
6
A
C
R18
R 6
B
R12
R24
R30
R35
84


70
5
70

Chapter 2

Geometrical Constructions

Geometrical constructions in Engineering drawing is an important technique to complete plane drawing in two dimensions by using point, line, arc, and circle as elements. Number of geometrical constructions is presented in this chapter to assess in finishing complex drawings.

2.1 Basic construction

A. Dividing a straight line into a given number of equal parts

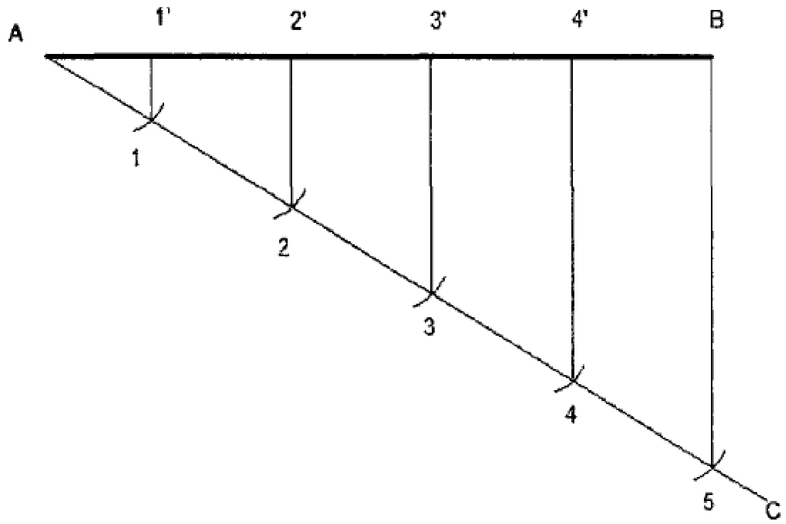


Fig. 2.1 Dividing of a line

Draw AC at any angle e to AB. Construct the required number of equal parts of convenient length on AC like 1, 2, and 3. Join the last point 5 to B. Through 4, 3, 2, and 1 draw lines parallel to 5 B to intersect AB at 4', 3', 2' and 1'.

B. Draw a line parallel to another at certain distant

Let AB be the given line, and at a given distance. From any two points well apart on AB, draw two arcs of radius equal to the distant. Draw a line tangential to the arcs, to give the required line.

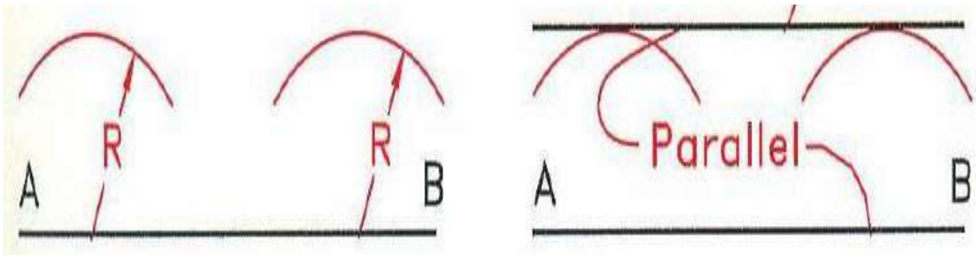


Fig. 2.2 Draw a line parallel to another at certain distant

C. Bisecting of an angle

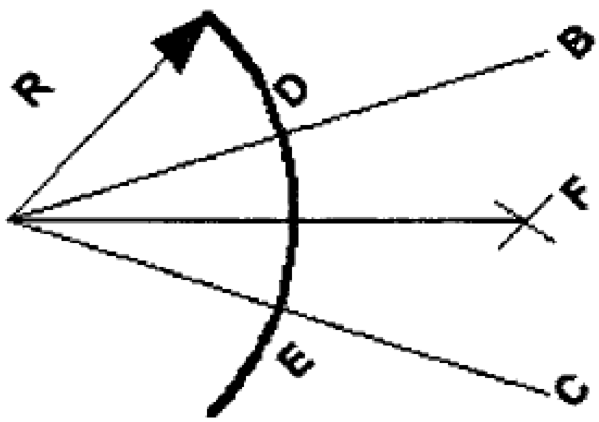


Fig. 2.3 Bisecting of an angle

Draw a line AB and AC making the given angle. With center A and any convenient radius R draw an arc intersecting the sides at D and E, respectively. With centers D and E and radius larger than half the chord length DE, draw arcs intersecting at F. Join AF, <BAF = <FAC

D. Inscribing a regular polygon of any number of sides in a given circle

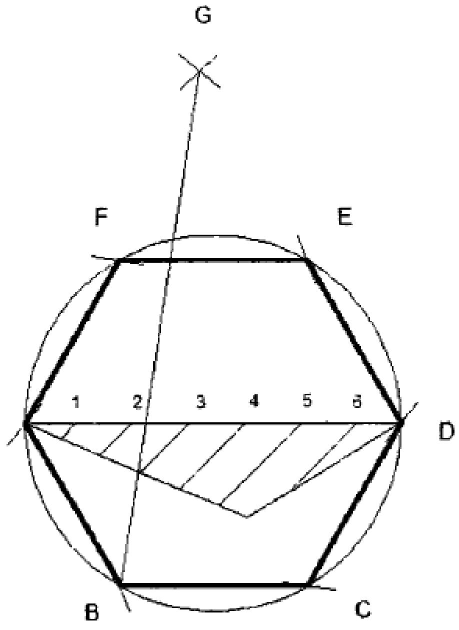


Fig. 2.4 Inscribing a regular polygon

Draw the given circle with AD as diameter. Divide the diameter AD into N equal parts say 6. With AD as radius and A and D as centers, draw arcs intersecting each other at G. Join G-2 and extend to intersect the circle at B. Join A-B which is the length of the side of the required polygon. Set the compass to the length AB and starting from B mark off on the circumference of the circles, obtaining the points C, D, etc. The figure obtained by join the points A,B, C etc., is the required polygon.

E. Constructing of a hexagon in a given circle

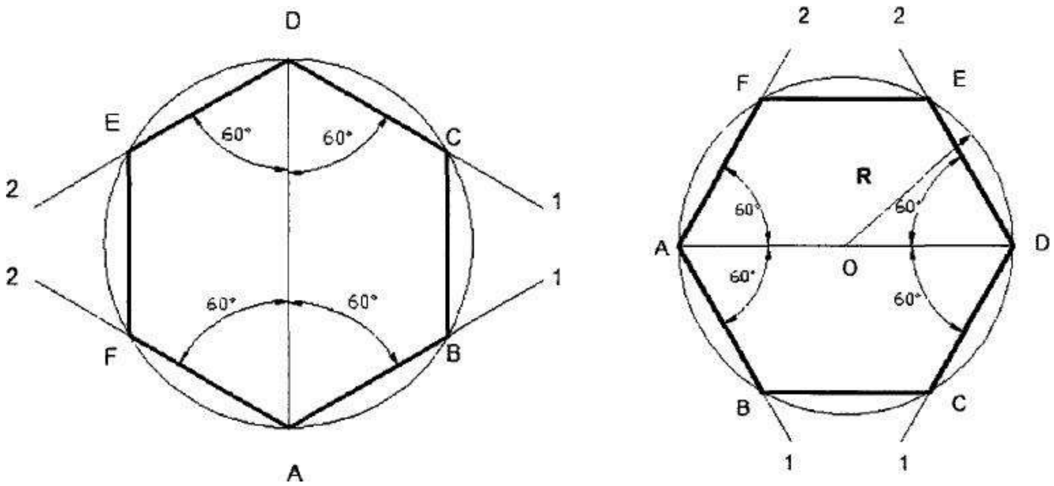


Fig. 2.5 Inscribing of a hexagon

With center 0 and radius R draw the given circle. Draw any diameter AD to the circle.
Using 30° - 60° set-square and through the point A draw lines AI, A2 at an angle 60° with AD, intersecting the circle at B and F respectively. Using 30° - 60° and through the point D draw lines Dl, D2 at an angle 60° with DA, intersecting the circle at C and E respectively.
By joining A, B, C, D, E, F, and A the required hexagon is obtained.

F. Constructing a regular hexagon given the diameter

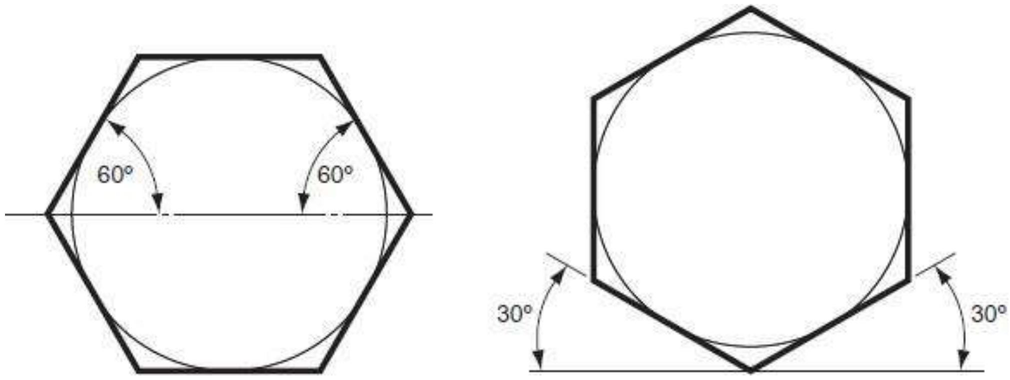


Fig. 2.6 Constructing a regular hexagon given the diameter:

This construction, using compasses and straight edge only, is quite feasible but is relatively unimportant. What is important is to recognize that a hexagon can be constructed, given the diameter or across-flats dimension, by drawing tangents to the circle with a 60 ° set square. This is very important when drawing hexagonal-headed nuts and bolts.

G. Constructing a hexagon, given the length of the side

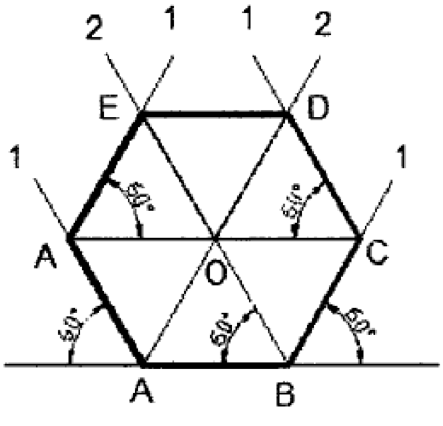


Fig. 2.7 constructing a hexagon

Draw a line AB equal to the side of the hexagon. Using 30° - 60° set-square draw lines A1, A2, and B1, B2. Through 0, the point of intersection between the lines A2 at D and B2 at E. Join D, E. ABC D E F is the required hexagon.

2.2 Tangents

A. Tangents between lines

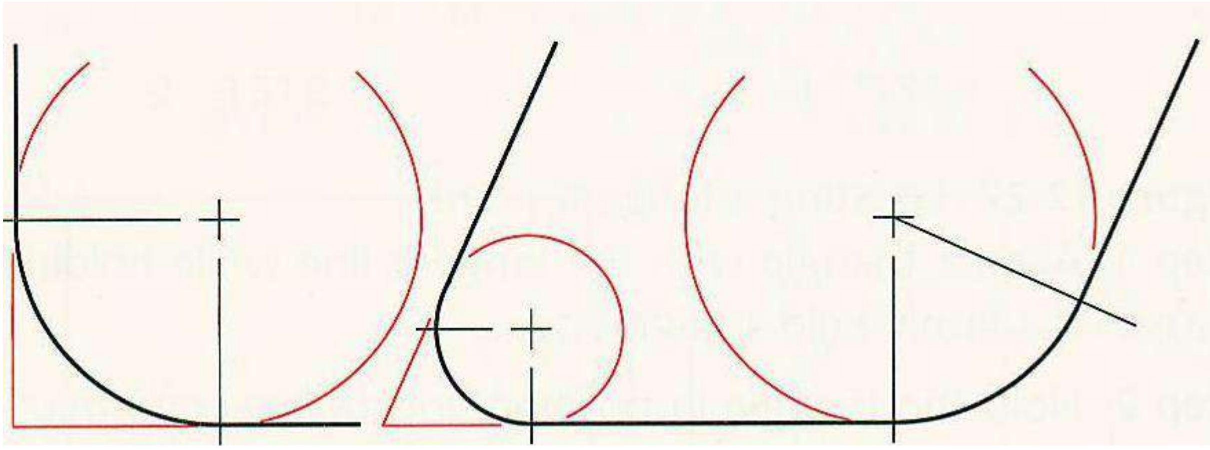


Fig. 2.8 Tangent whole circle view.

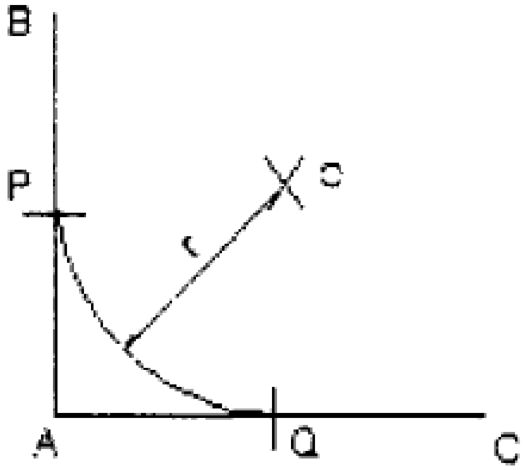


Fig. 2.9 Tangent between two right angle lines.

To draw an arc of given radius touching two straight lines at right angles to each other.
Let r be the given radius and AB and AC the given straight lines, with A as center.
Draw arc with radius r cutting AB at P and Q. With P and Q as centers draw arcs to meet at O.
With O as center and radius r draw the required arc.

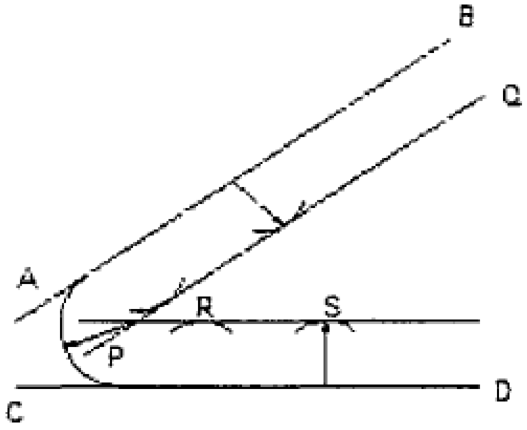


Fig. 2.10 Tangent between two acute angle lines.

Let AB and CD be the two straight lines and r, the radius. Draw a line PQ parallel to AB at a distance r from AB. Similarly, draw a line RS parallel to CD. Extend them to meet at O. With O as center and radius r draw the arc to the two given lines.

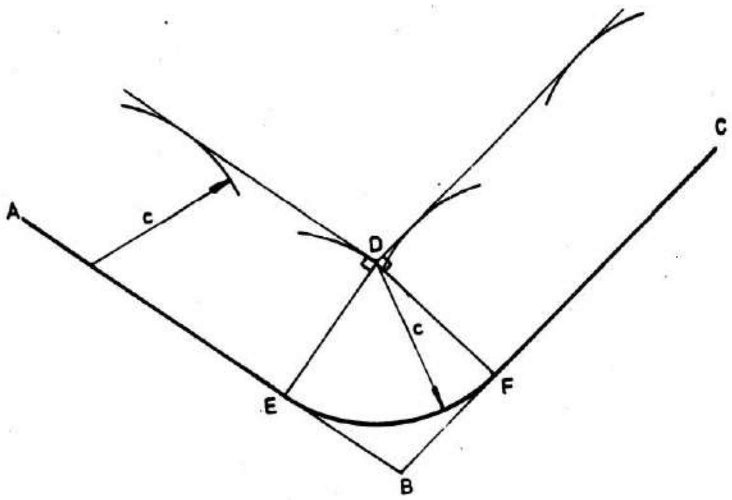


Fig. 2.11 Tangent between two obtuse angle lines.

Let AB and CB be the two given lines, while c is the radius of the required arc. Draw two lines parallel to the two given lines, and apart from each of them by distance c. The new parallel lines will intersect at point D. From point D and with radius c, draw an arc, which will be tangential to both given lines. Erect perpendiculars at D to intersect AB and BC at E and F respectively and these arc the points of tangency of the lines with the arc.

B. Tangents between line and curve

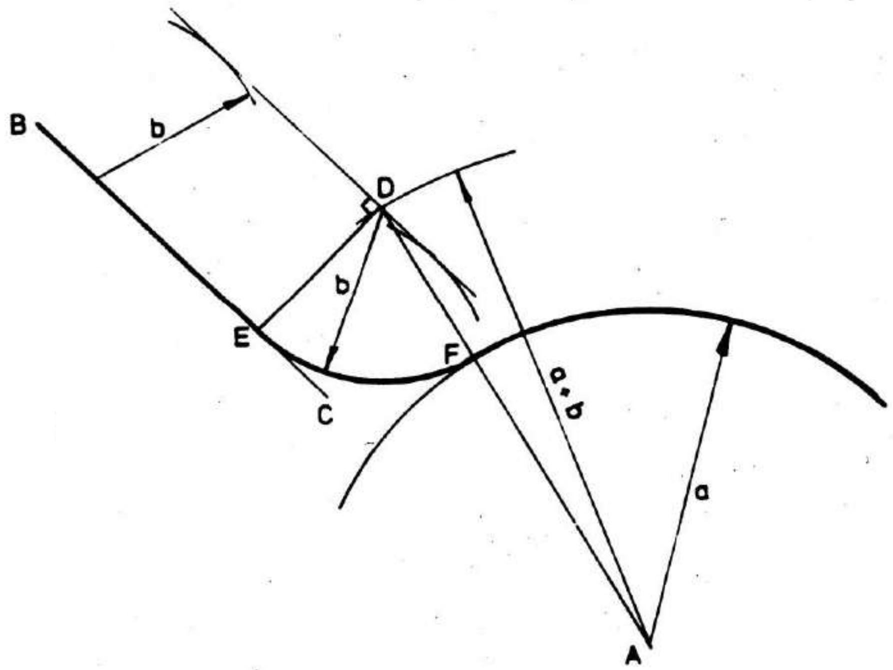


Fig. 2.12 Tangent between line and curve.

Let A be the center of a given arc having radius a, while BC is the given line, and b the radius of the required arc. From A, describe an arc with radius " a + b". Draw a line parallel to BC, spaced by distance b from it. The line will intersect with the arc "a + b" at D. From D describe an arc of radius b which will be tangential to both the given line BC and the given arc a.

C. Tangents between curves

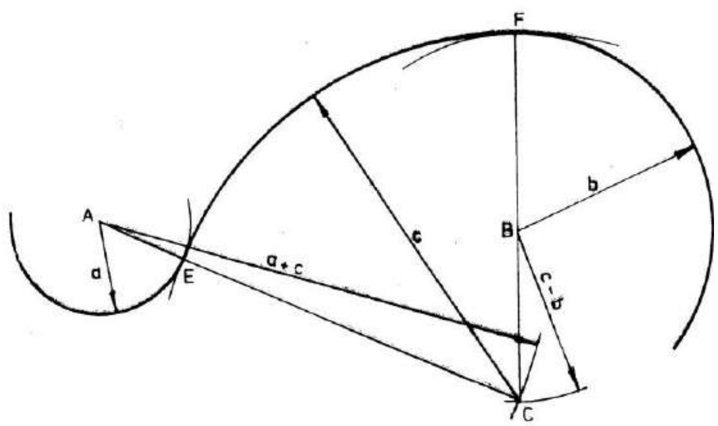


Fig. 2.13 Tangent between two arcs enclosing one of them

Let A and B be two centers of a two given arcs, having radii 'a' and 'b' respectively, and C is the radius of the required arc. From the two given centers (named A and B) describe two arcs having radii of a + c and c - b respectively to intersect at C. From the generated center C, describe an arc of radius c. Join AC, and BC to intersect the new curve at points E and F respectively, which are the points of tangency of the three arcs.

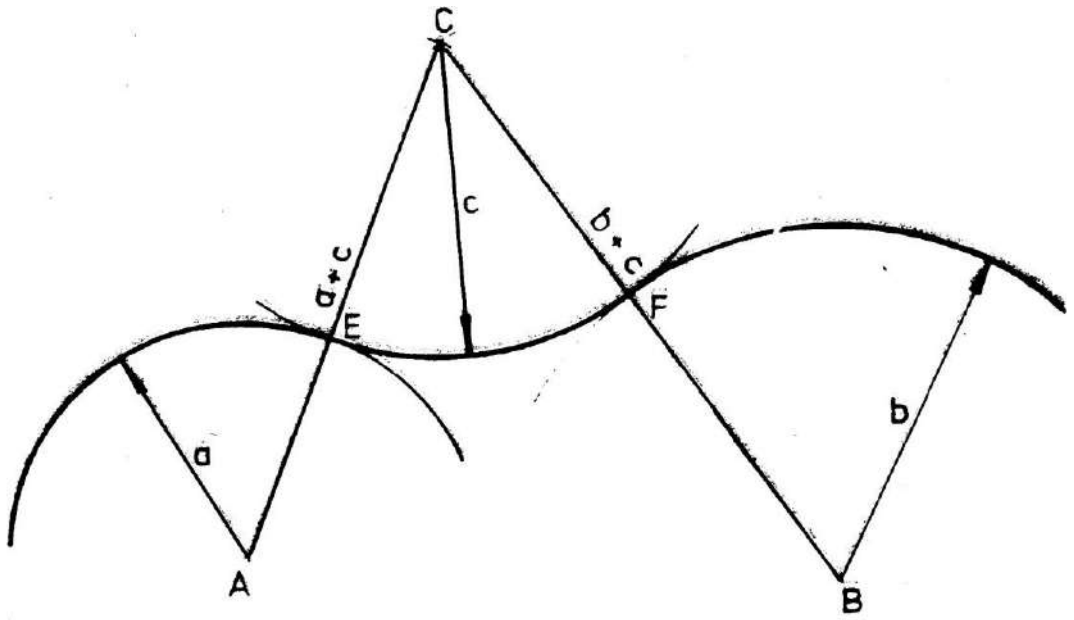


Fig. 2.14 Tangent between two curves externally.

Let A and B be two centers of a two given arcs, having radii 'a' and 'b' respectively, while c is the radius of the external arc. From points A and B, describe two Arcs having radii of "a + c" and "b + c" respectively, which will intersect at C. From the new center 'C' and with radius c. describe an arc which will be tangential to the given two arcs. Connect AC and BC to get points "E and F" which are the points of tangency of the three arcs.

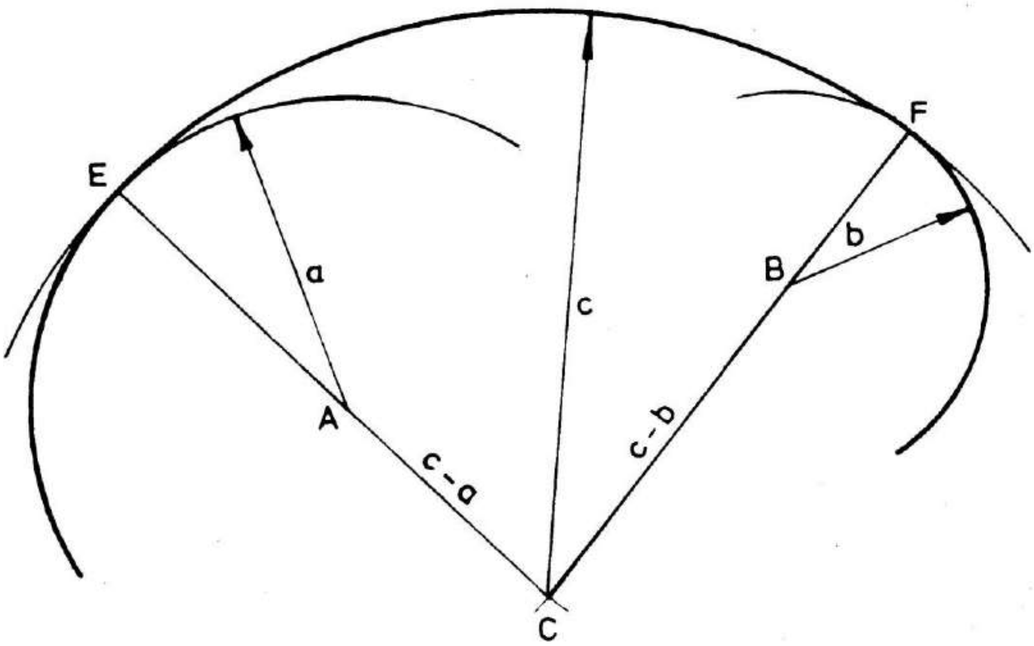


Fig. 2.15 Tangent between two curves internally.

Let 'A' and 'B' be two centers for two given arcs having radii 'a' and 'b' Respectively, while c is the radius of the required tangential arc. From points 'A' and 'B' describe two arcs having radii "c-a" and "c-b", respectively. The generated two arcs will intersect at C. From the new 'center C and with radius c describe an arc which will be tangential to the given two arcs. Produce CA and CB to intersect the curve at E and F respectively, which are the points of tangency of the arcs.

2.3 Ellipse

The ellipse is a closed symmetrical curve with a changing diameter. The ellipse diameter varies between minimum and maximum values, which are known as the minor and major axis of the ellipse. The ellipse may be geometrically defined as: the curve traced out by a point (P) which moves so that the sum of its distances from two fixed points (F and 'F) is constant and equal to the major axis. Some Methods of Drawing Ellipse are presented below.

A. The four centers method

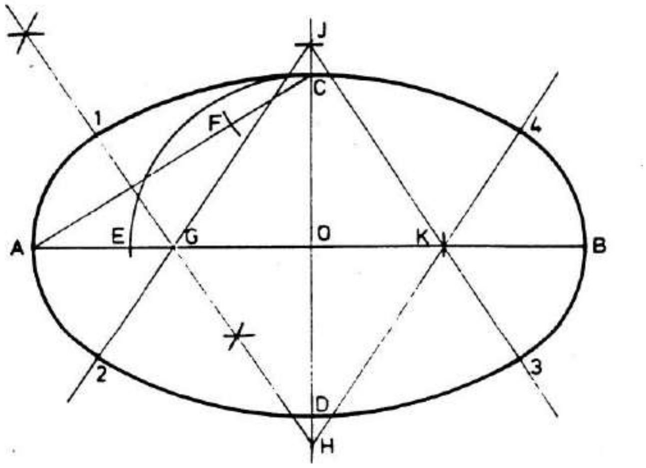


Fig. 2.16 The four centers ellipse

Let the major and minor axes are AB and CD. Draw the major and minor axes AB, and CD, to intersect at point O. From point O describe an arc with radius OC to intersect AO at point F. Join AC. From point C describe an arc having radius AE to intersect AC at point F. Divide AF into two equal parts, and eject bisector. The produced bisector will intersect AO at G and CD at H. From point G, draw an arc of radius GA (from point I to point 2). From point H. draw an arc of radius HC (from point I to point 4). Now, half of the ellipse is constructed, and the other half could be constructed by repeating the previous steps.

B. The isometric ellipse method

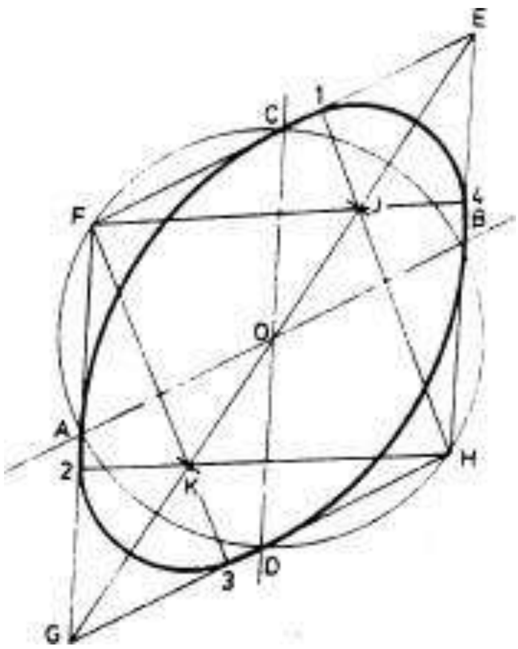


Fig. 2.17 The Isometric Ellipse

Through isometrics, circles are represented by ellipses. If O and AO, are the center and radius of the circle to be drawn. Draw the isometric axes (any two lines having an angle of 60" in between) to intersect at O. From the center O describe a circle having the radius AO to intersect the isometric axes at A, B, C, and D. Through these four points draw lines parallel to the isometric axes to intersect at E. F. G and H. Draw the long diagonal GE to pass through, O. From points E and G, mark off distances EJ and GK equal to AO along that diagonal.
Join FK, and Hk, and extend each of them to meet FG, and GH at points 2, and 3 respectively. From points H, and K, draw two arcs having radii HF, and KD respectively. Now, half of the ellipse is drawn, the other half could be constructed by repeating step 'g'.

C. The concentric circles method

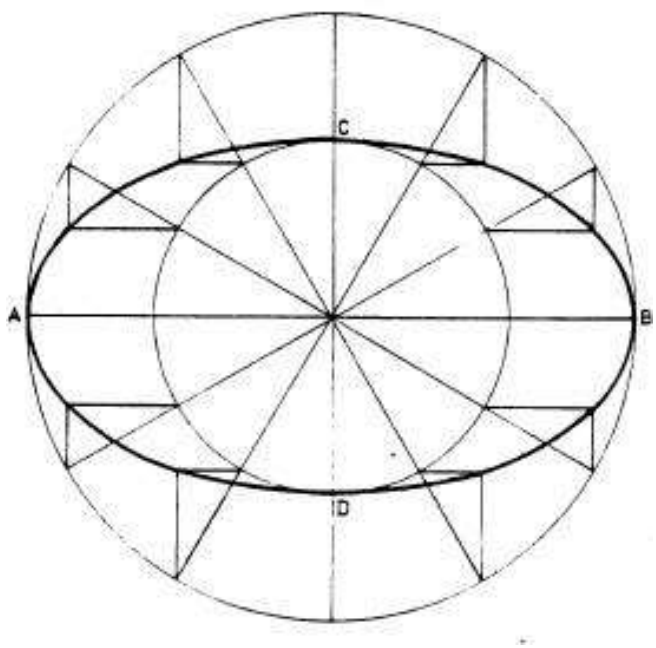
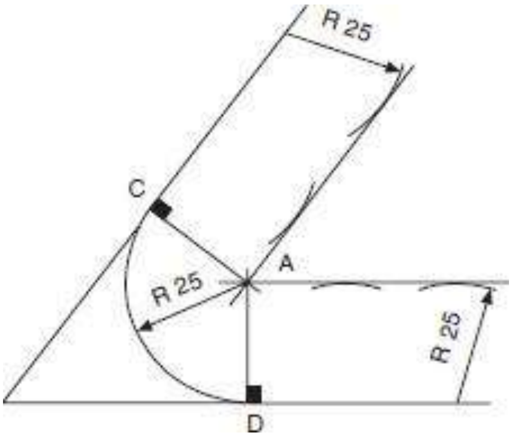


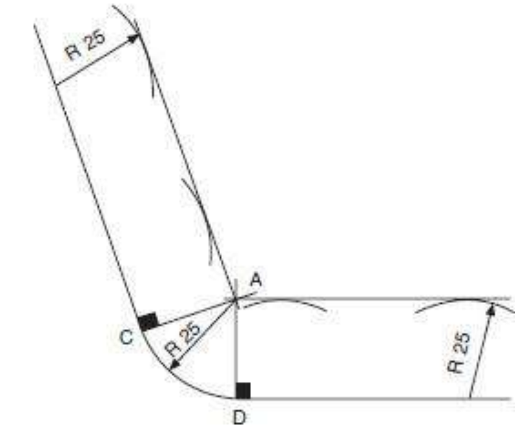
Fig. 2.18 The concentric circles Ellipse

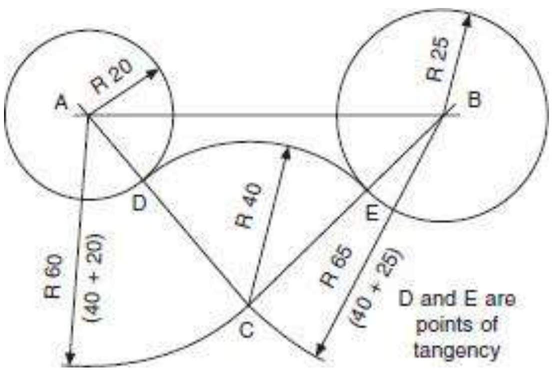
Draw two concentric circles having diameters equal to the major and minor axis of the ellipse, AB and CD respectively. Divide the circles into twelve equal parts. Draw perpendicular lines from points of division on the outside circle to intersect horizontal lines drawn from the corresponding points of division on the inside circle. Draw a smooth curve through these points of intersection to get the required ellipse.

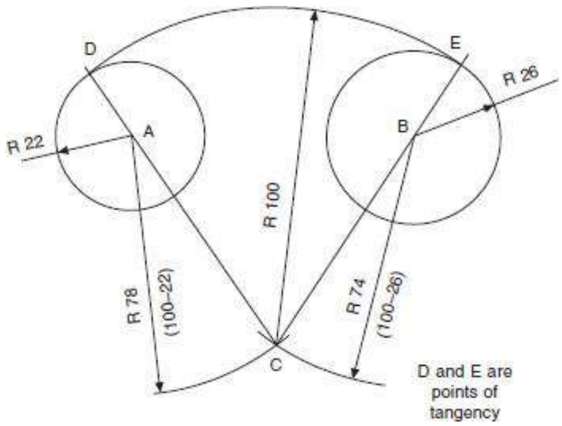
Exercise

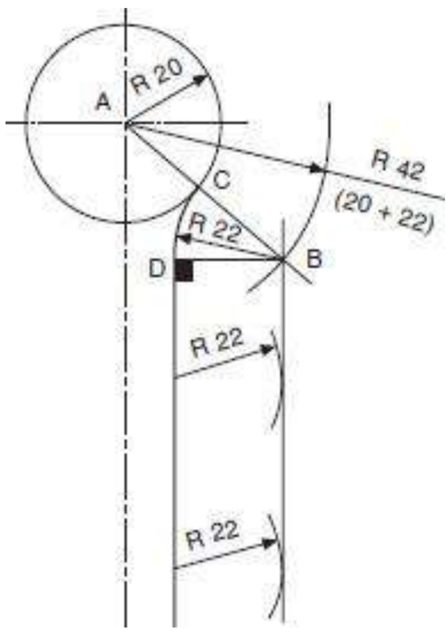
1. Practicing tangency



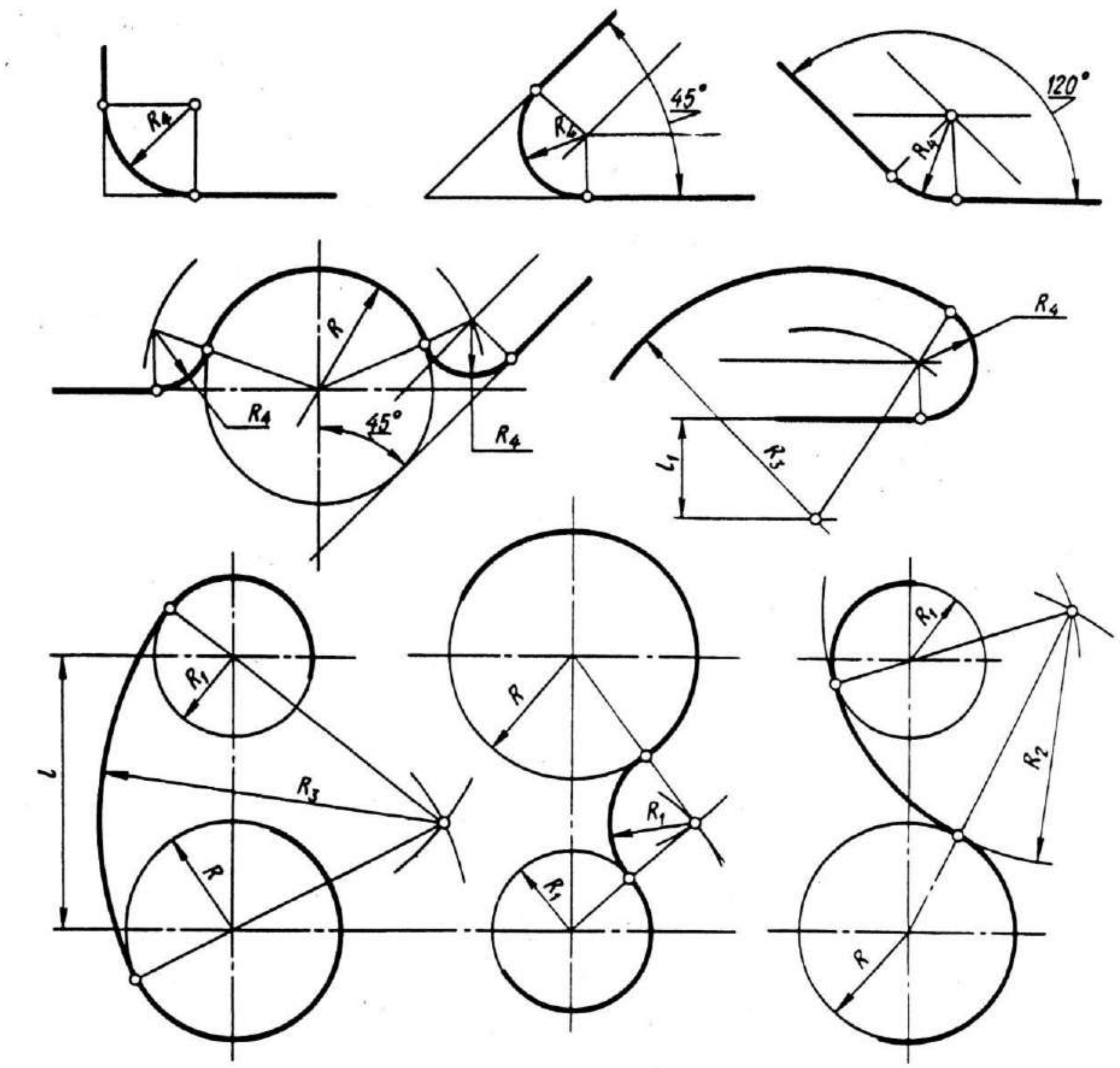






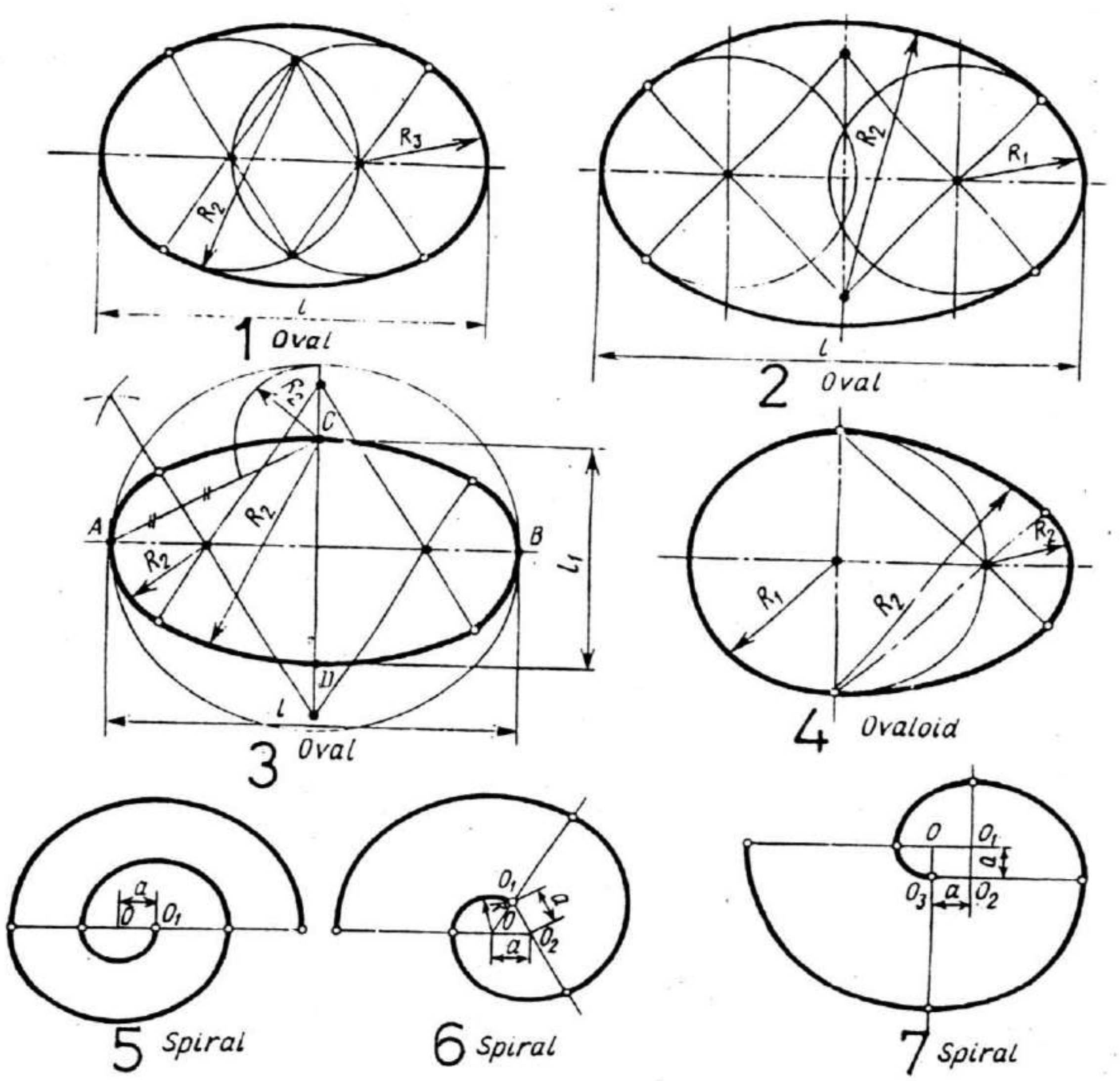


2. Draw using rules of tangency



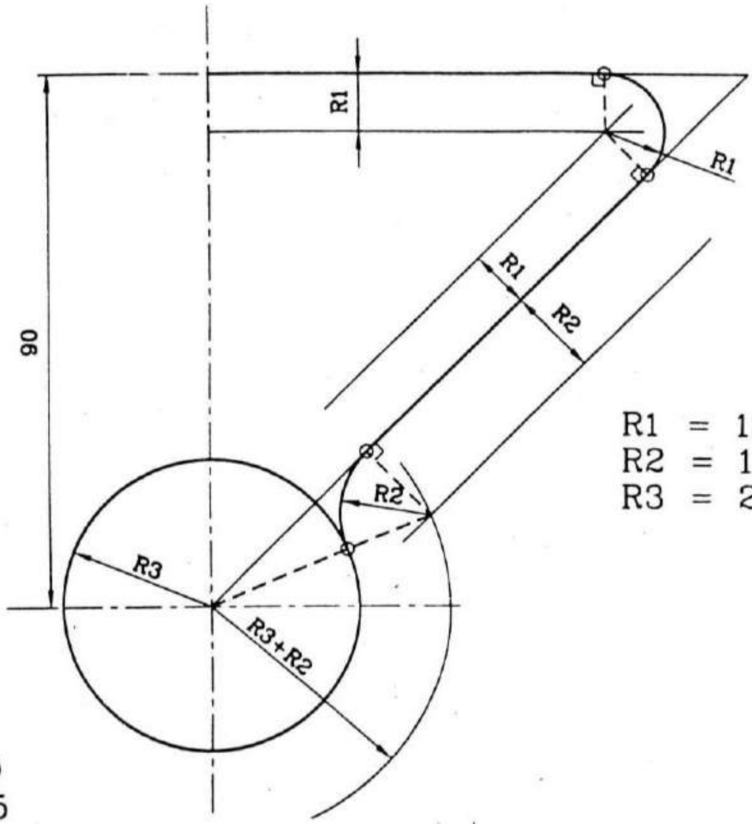
Tangency's Constructions

	Exercise No.															
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
R	70	60	64	60	66	60	68	68	62	32	50	50	56	66	68	58
R_1	50	40	36	40	40	50	50	50	52	52	30	32	36	48	52	36
R_2	100	80	84	100	110	120	100	110	120	120	96	94	80	110	104	96
R_3	140	130	120	170	136	140	140	140	138	138	136	136	138	140	140	138
R_4	30	28	24	26	28	26	28	32	28	28	24	24	26	36	32	32
l	140	130	120	140	140	150	138	150	148	148	120	120	148	142	138	144
l_1	50	44	36	42	50	44	50	20	42	42	40	40	38	38	50	46



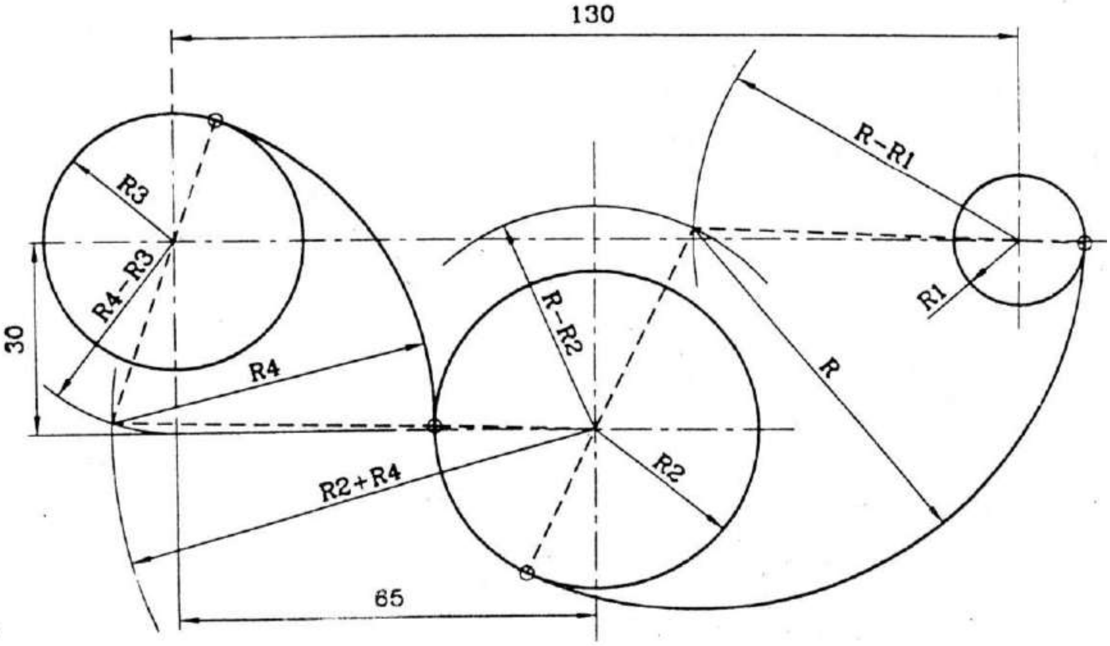
I mm	I_1 mm	R_1 mm	R_3 mm	a mm
100	72	24	1/3 L	10
96	66	25	1/3 L	10

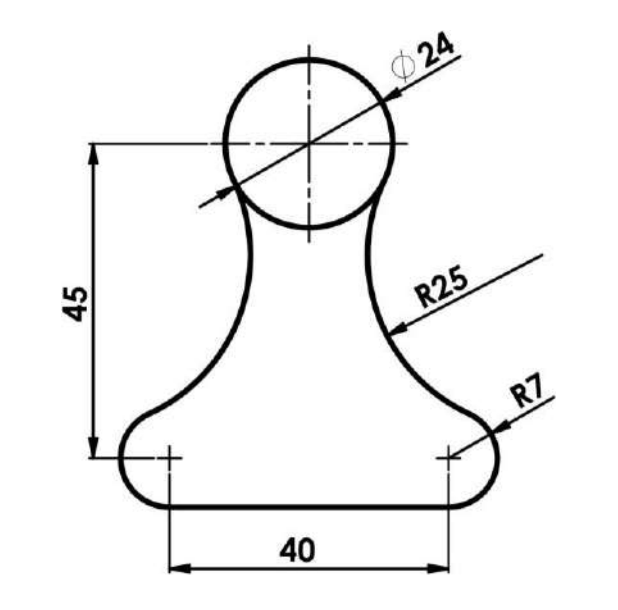
Note: $\mathbf{R_2}$ obtained from construction

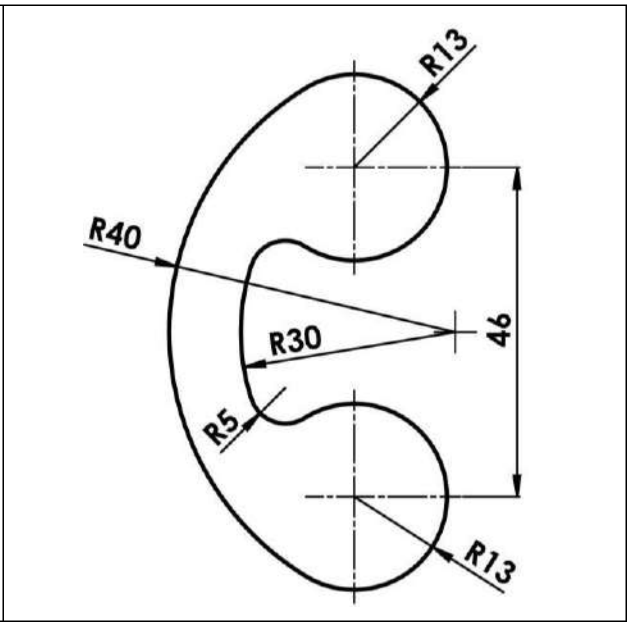


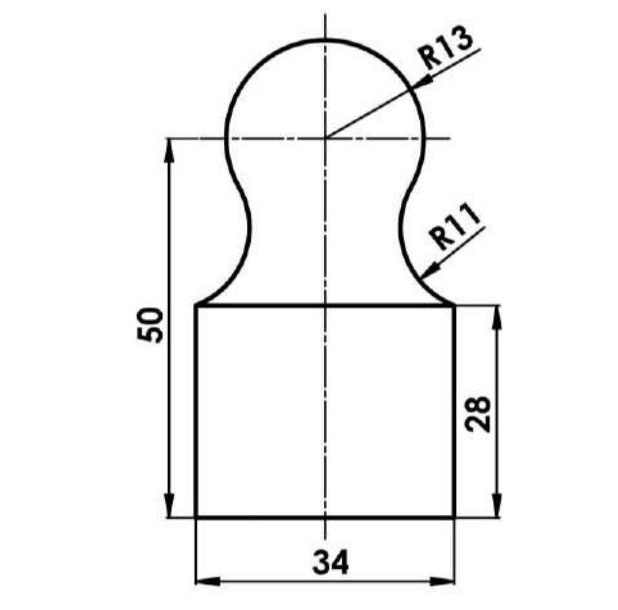
R1 = 10
R2 = 15
R3 = 25

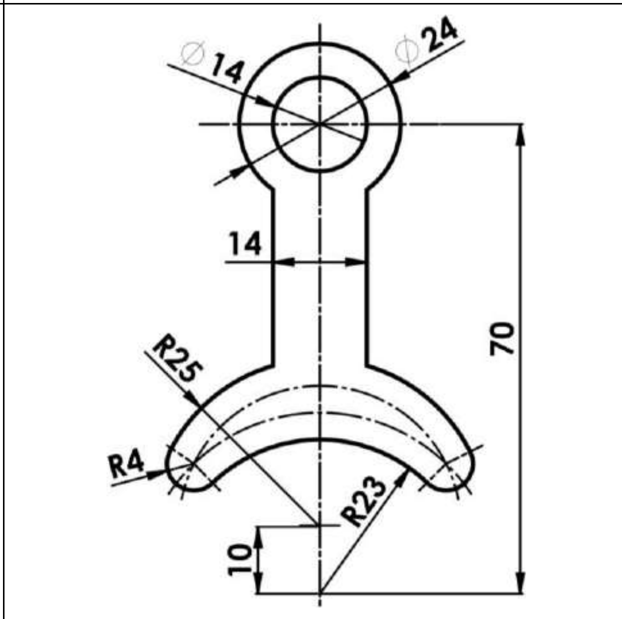
R1 = 10
R2 = 25
R3 = 20
R = 60
R4 = 50

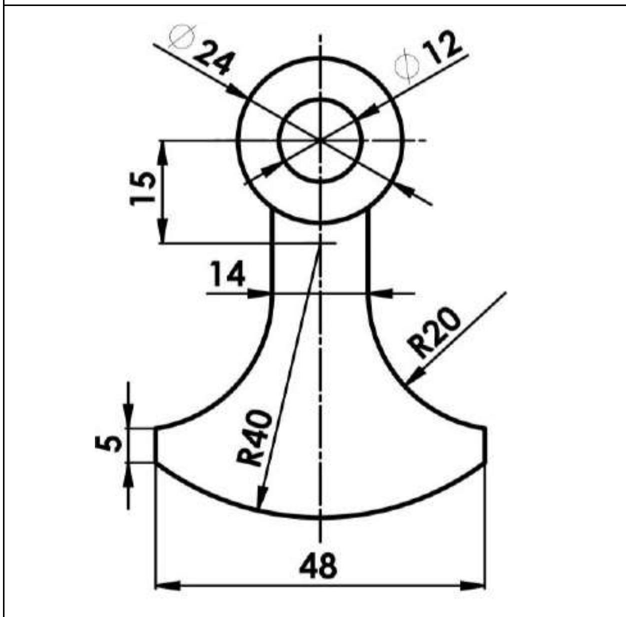


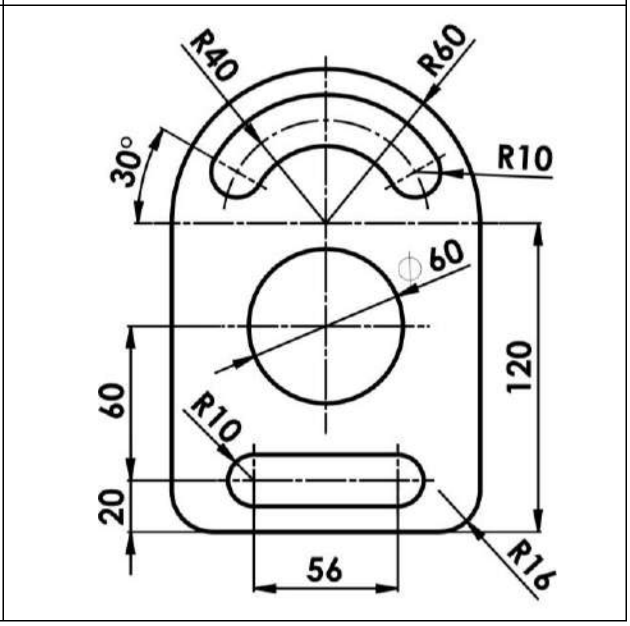

⌀24
45
R25
R7
40

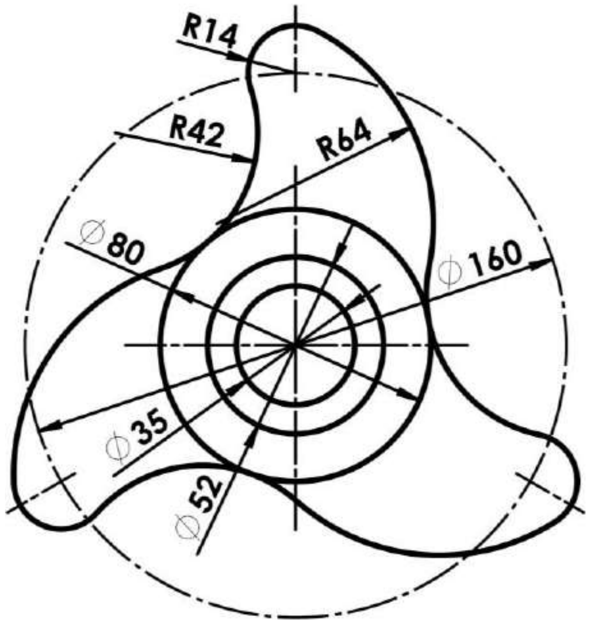

R13
R40
R30
46
R5
R13

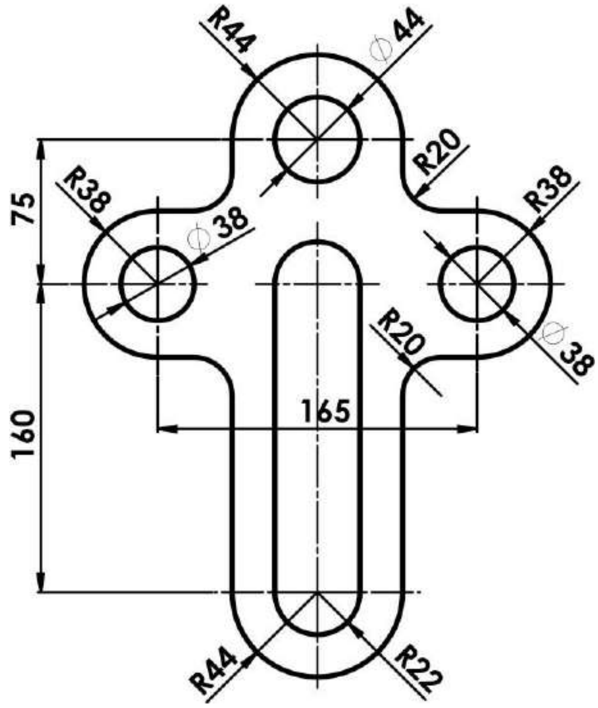

R13
R11
50
28
34

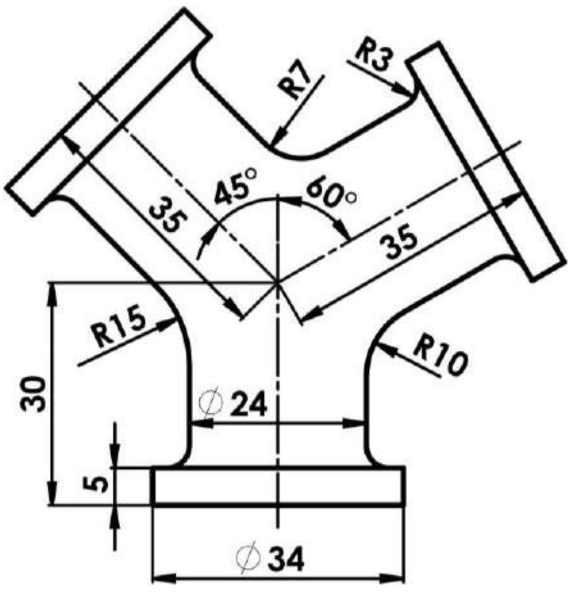

⌀14
⌀24
14
70
R25
R4
R23
10

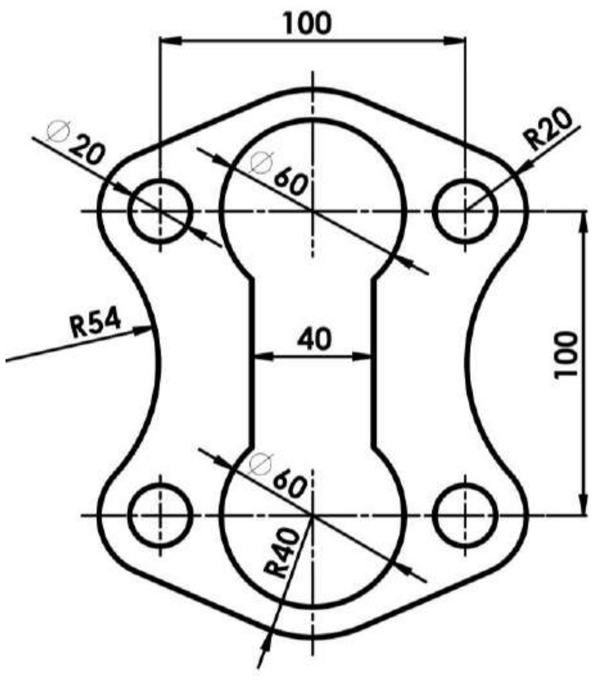

⌀24
⌀12
15
14
R20
5
R40
48

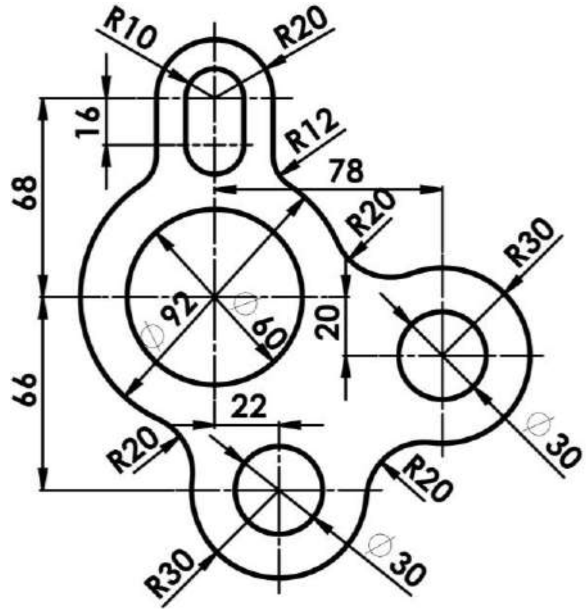

R40
R60
30°
R10
⌀60
120
60
R10
20
56
R16

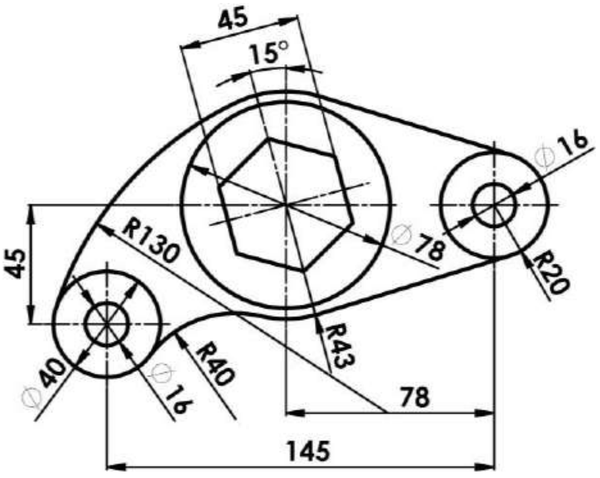

R14
R42
R64
∅80
∅160
∅35
∅52

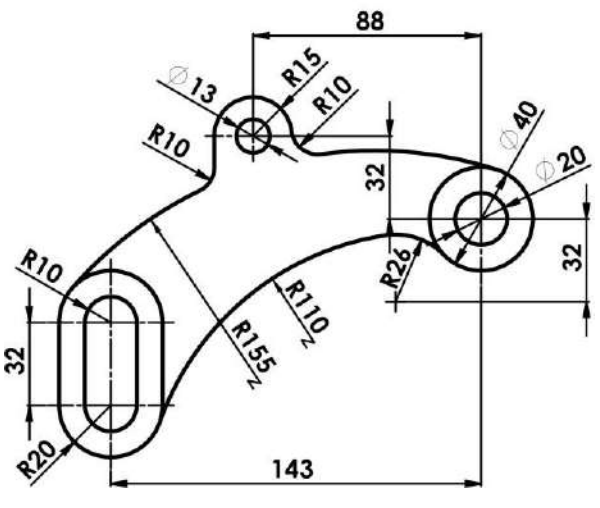

R44
∅44
R20
75
R38
∅38
R38
R20
∅38
165
160
R44
R22

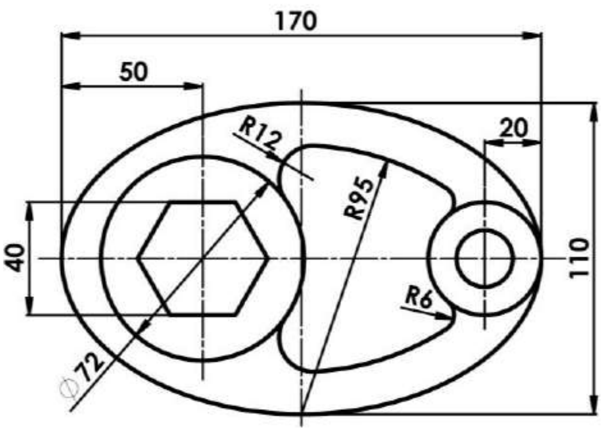

R3
R7
45°
60°
35
35
R15
R10
30
∅24
5
∅34

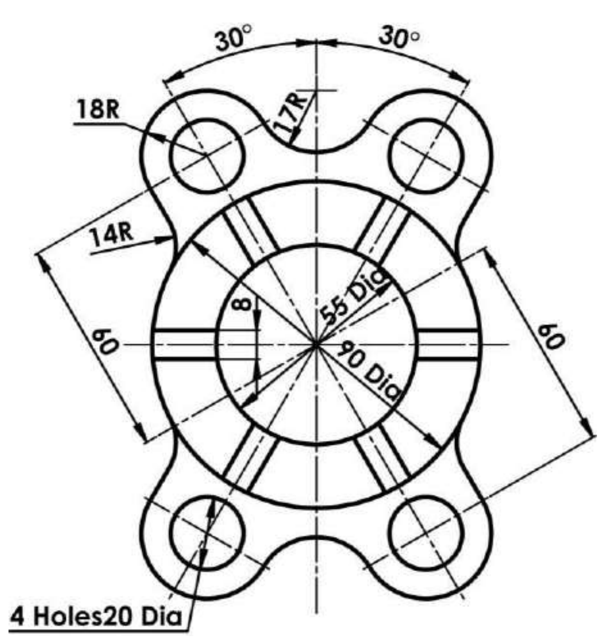

100
∅20
∅60
R20
R54
40
100
∅60
R40


R10
R20
16
68
R12
78
R20
R30
92
60
20
66
22
R20
R20
30
R30
30

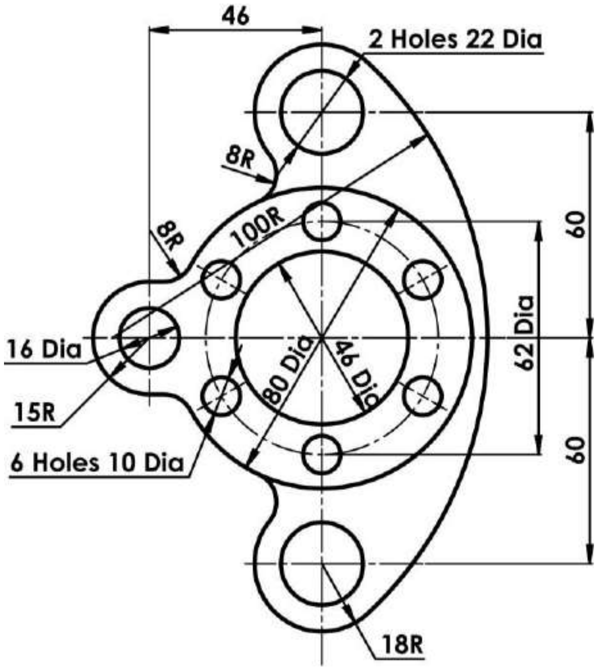

45
15°
16
R130
78
45
R20
R40
R43
40
16
78
145


88
13
R15
R10
R10
40
20
32
R10
32
R26
R110
R155
32
R20
143

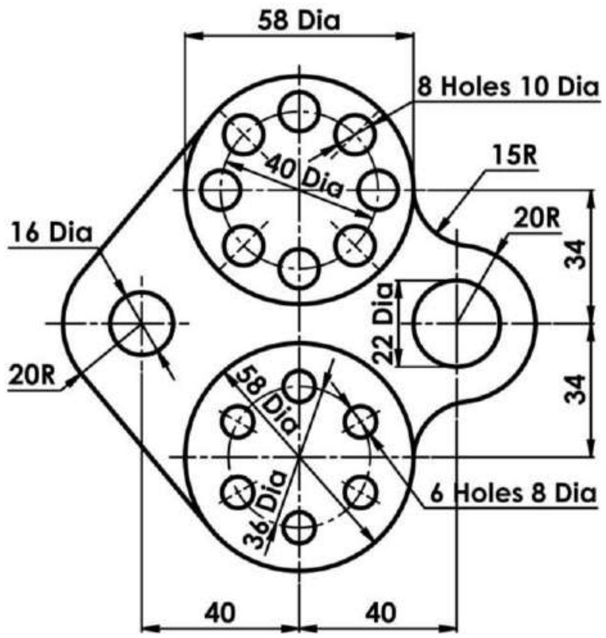

170
50
R12
20
R95
40
110
R6
72



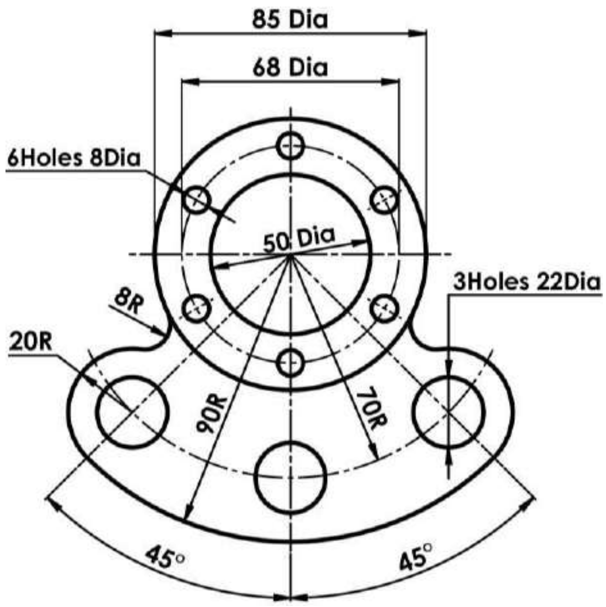
30°
30°
18R
17R
14R
60
8
55 Dia
90 Dia
60
4 Holes20 Dia



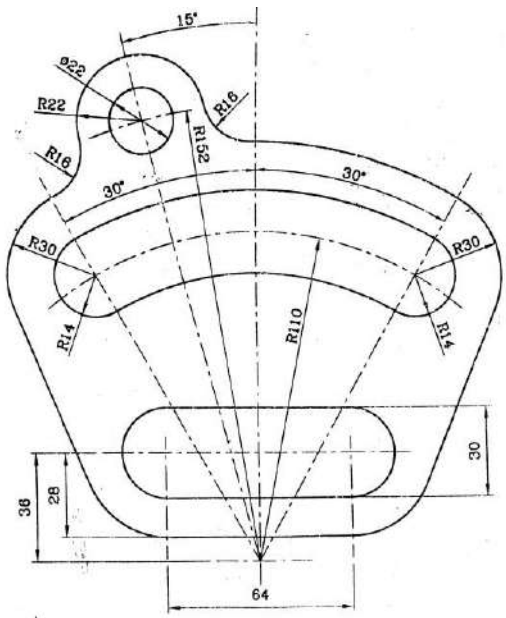
46
2 Holes 22 Dia
8R
8R
100R
60
62 Dia
16 Dia
80 Dia
46 Dia
15R
6 Holes 10 Dia
60
18R

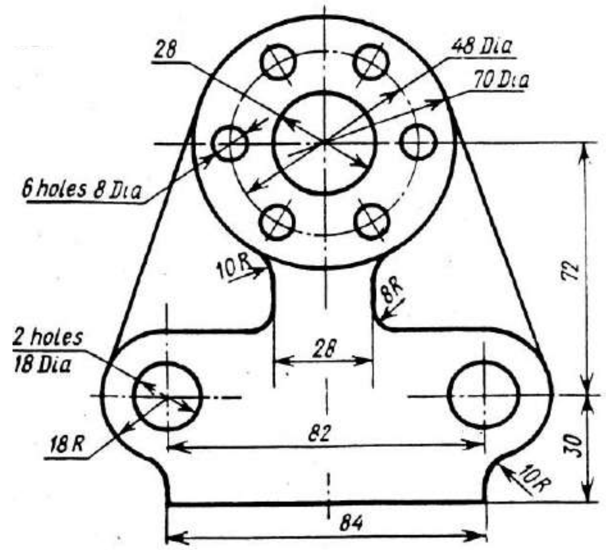


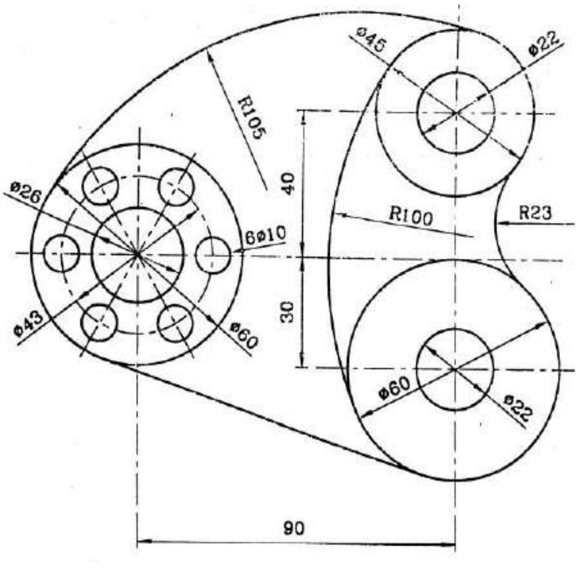
58 Dia
8 Holes 10 Dia
40 Dia
15R
20R
16 Dia
34
22 Dia
20R
58 Dia
34
36 Dia
6 Holes 8 Dia
40
40

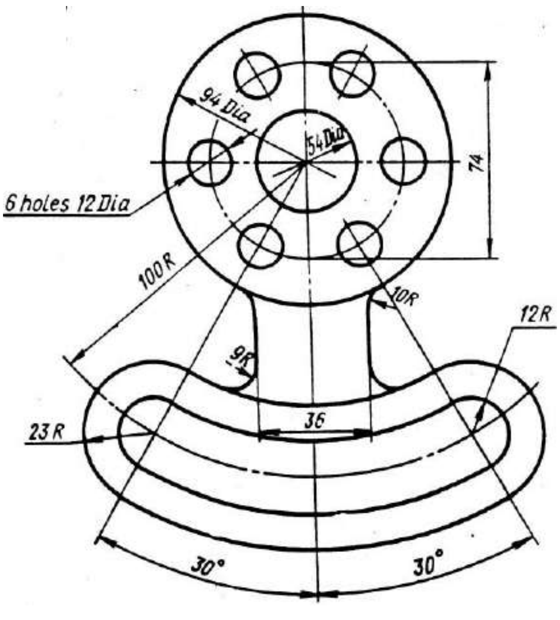


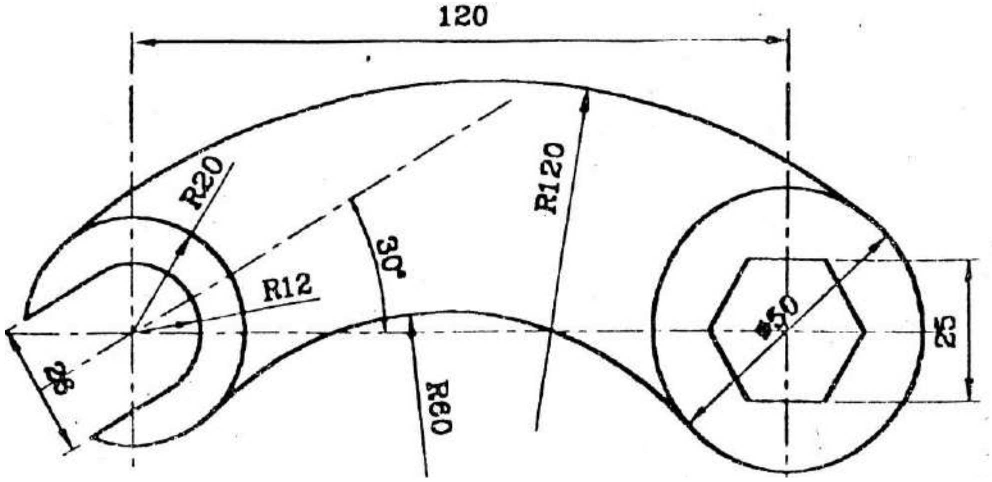
85 Dia
68 Dia
6Holes 8Dia
50 Dia
3Holes 22Dia
8R
20R
90R
70R
45°
45°

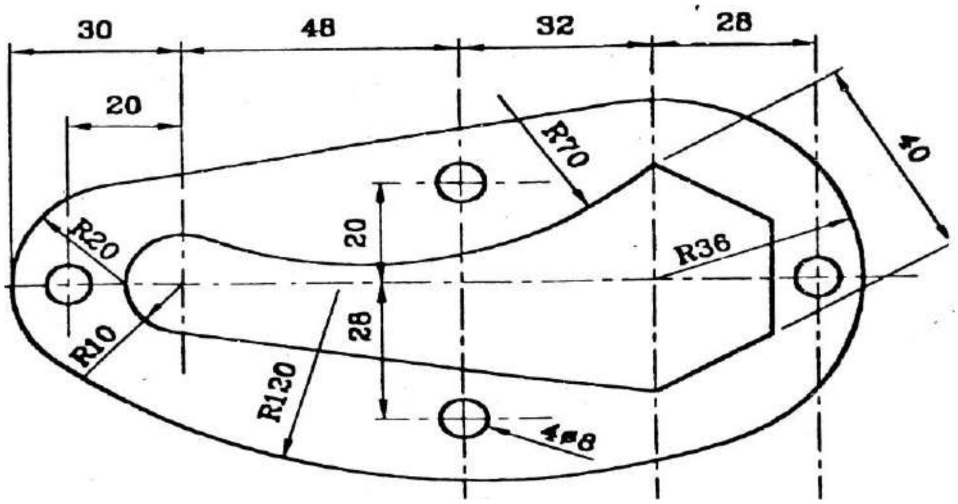

15°
ø22
R22
R16
R16
R152
30°
30°
R30
R30
R14
R14
R110
30
36
28
64

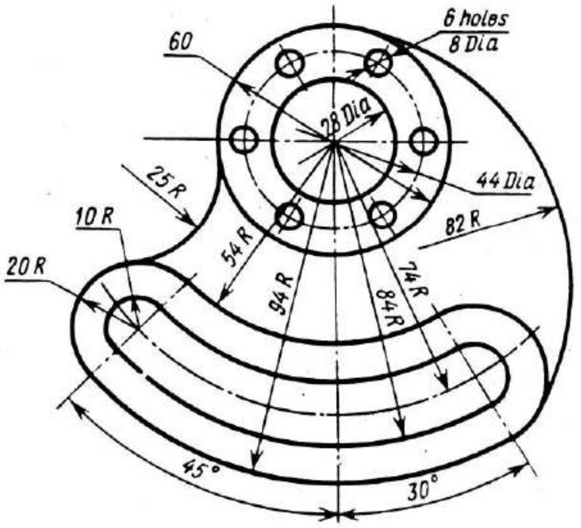

28
48 Dia
70 Dia
6 holes 8 Dia
10 R
8R
72
2 holes
18 Dia
28
18 R
82
30
10 R
84

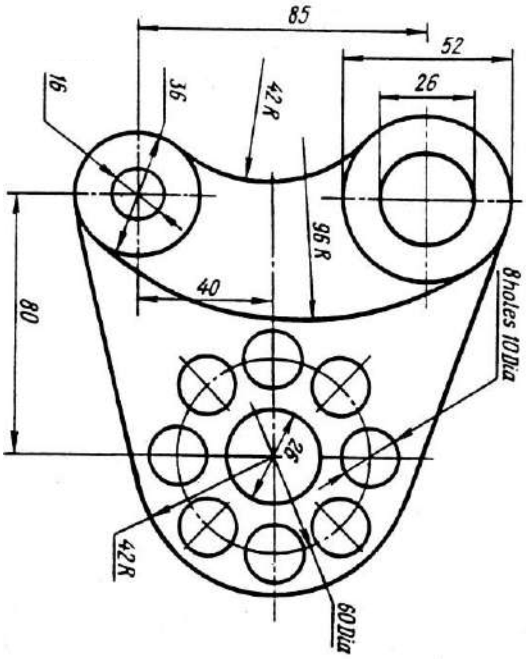

ø45
ø22
R105
40
ø26
R100
R23
6ø10
30
ø43
ø60
ø60
ø22
90

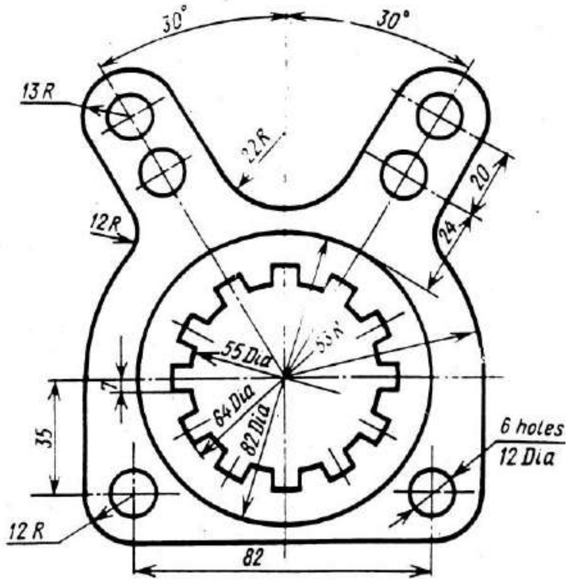

94 Dia
54 Dia
74
6 holes 12 Dia
100 R
10R
12R
9R
36
23 R
30°
30°

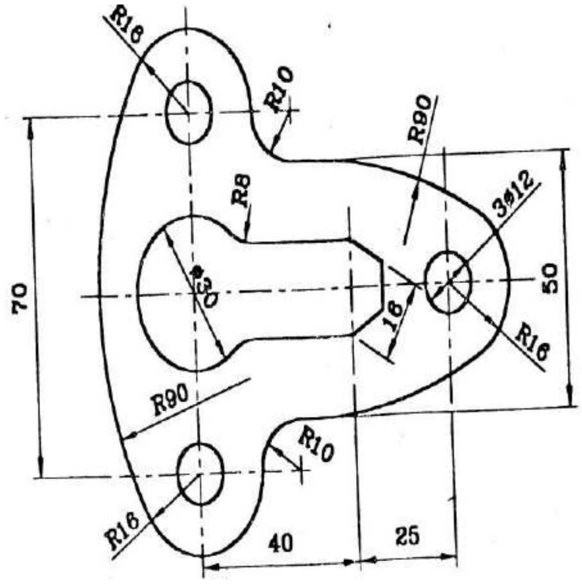

120
R20
R12
30°
R120
R60
26
ø50
25


30
48
32
28
20
R70
40
20
R20
R36
R10
28
R120
4ø8


6 holes
8 Dia
60
28 Dia
44 Dia
25 R
82 R
10 R
54 R
20 R
94 R
84 R
74 R
45°
30°


85
52
26
16
36
42 R
96 R
40
80
8 holes 10 Dia
26
42 R
60 Dia


30°
30°
13 R
22 R
20
24
12 R
55 Dia
55 R
64 Dia
82 Dia
7
35
6 holes
12 Dia
12 R
82


R16
R10
R90
R8
70
3ø12
ø30
16
50
R16
R90
R10
R16
40
25

Chapter 3 Three- Dimensions Solid Geometry

Isometric illustration is the method of depiction ideas which formed in actual 3-D objects. This method can describe the object shape easily. This construction could be done based on the objects shape which viewed from certain directions. Isometric projection is produced using 3 axes: vertical axis, left axis, right axis. The left and right axes are produced at the angle of 30° to the horizontal lines.

Figure below shows the main axis of isometric developed with 30° set square.

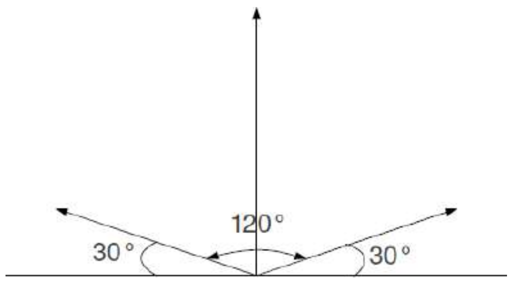


Fig. 3.1 The main axis of isometric

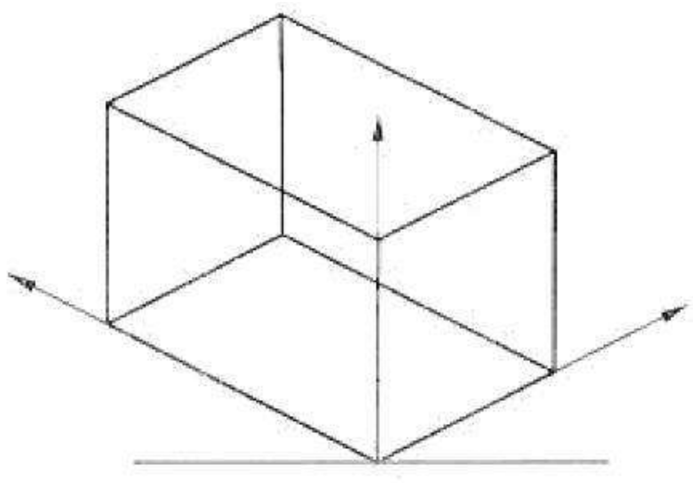


Fig. 3.2 A box for isometric drawing

3.1 Isometric drawing Steps

After positioning object, select an isometric axis. Sketch enclosing box (guide box). Add details i.e. Transfer the distance of object and draw the construction lines then darken visible lines.

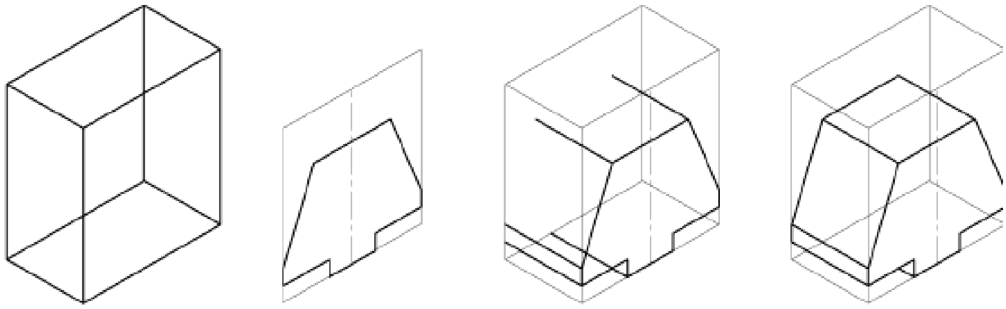


Fig. 3.3 Isometric drawing Steps:

3.2 Drawing of Circles and Arcs in isometric drawing

A circle appears as an ellipse and to draw it follow these steps.

Drawing steps:

1. Locate the center of an ellipse.
2. Construct an isometric square.
3. Construct a perpendicular bisector from each tangent point.
4. Locate the four centers.
5. Draw the arcs with these centers and tangent to isometric square.

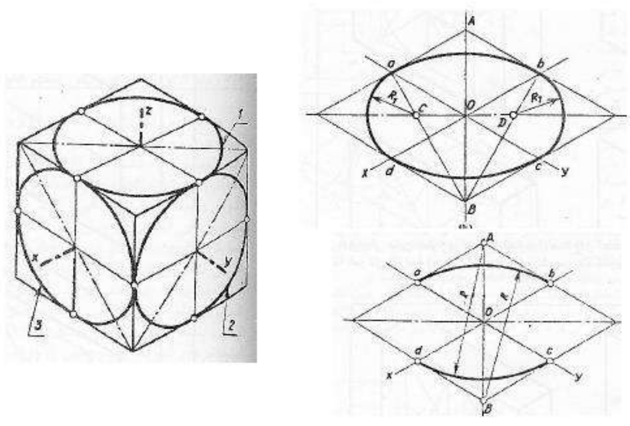
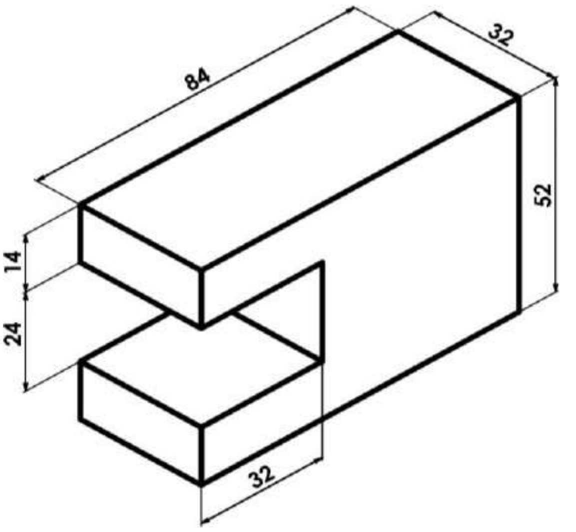
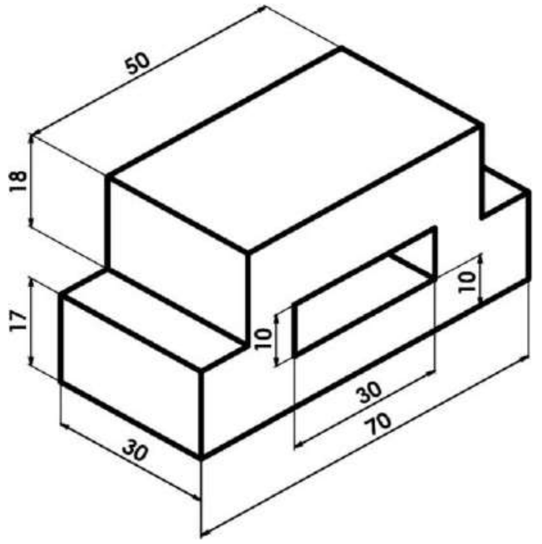


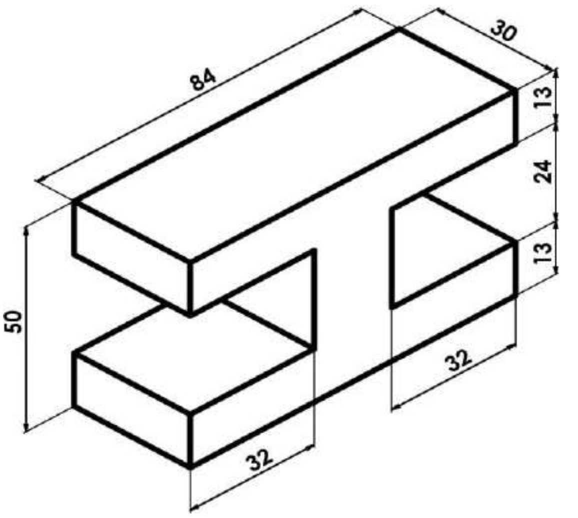
Fig. 3.4 Drawing of Circles and Arcs in isometric

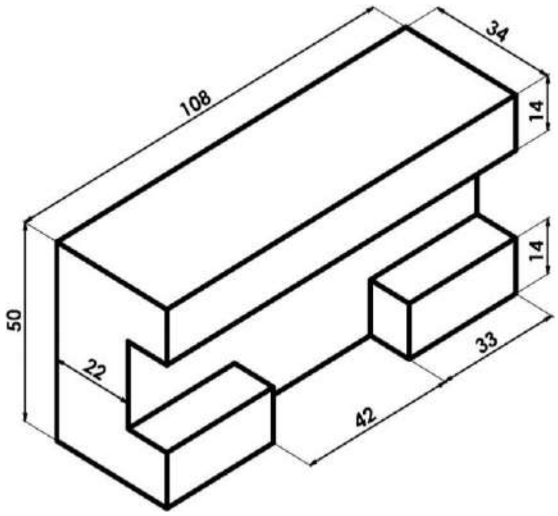
Exercise

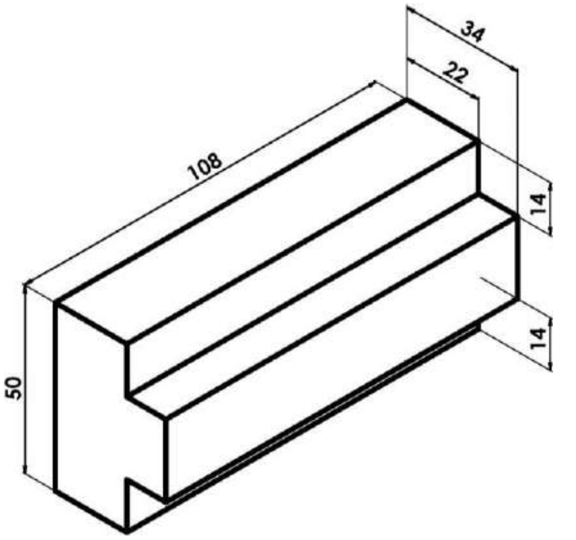
1. Draw the following perpendicular and inclined planes isometrics

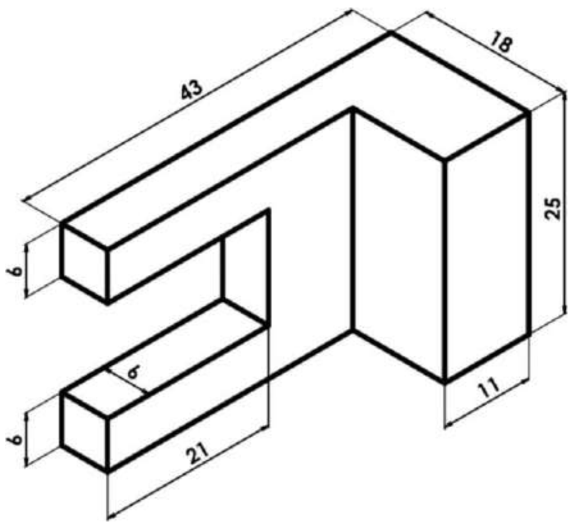


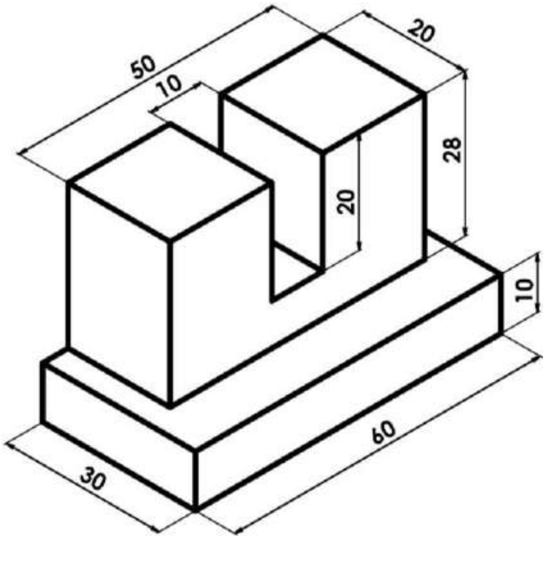








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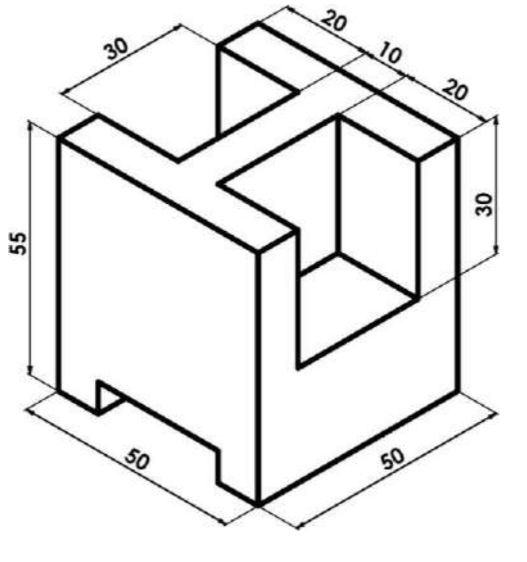
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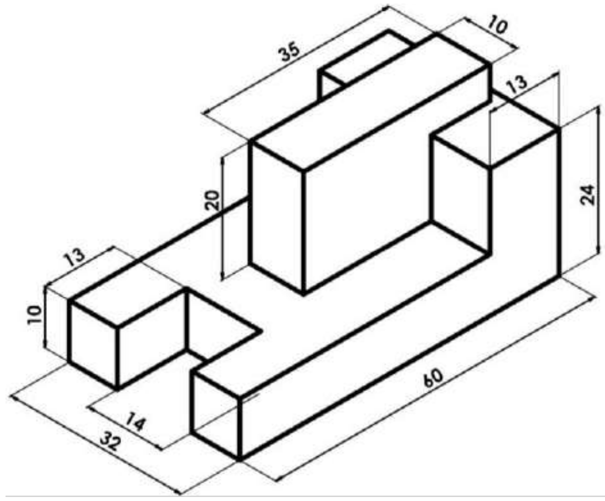
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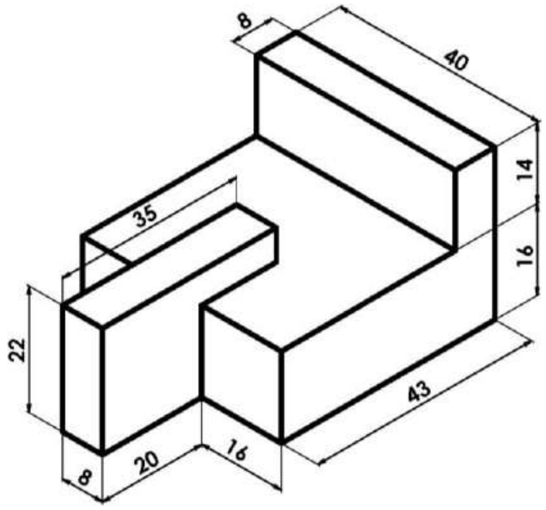
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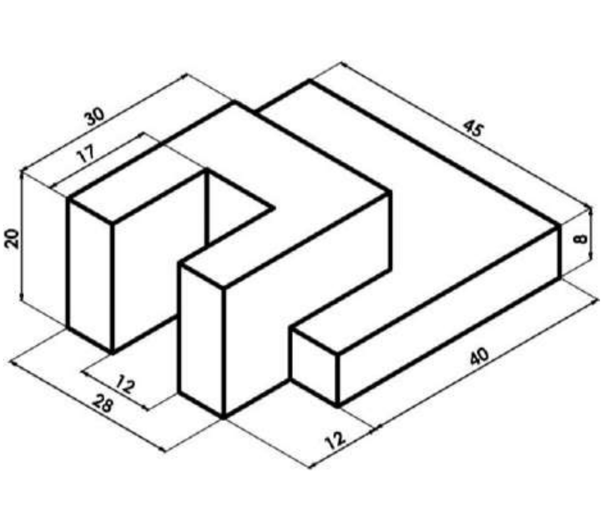
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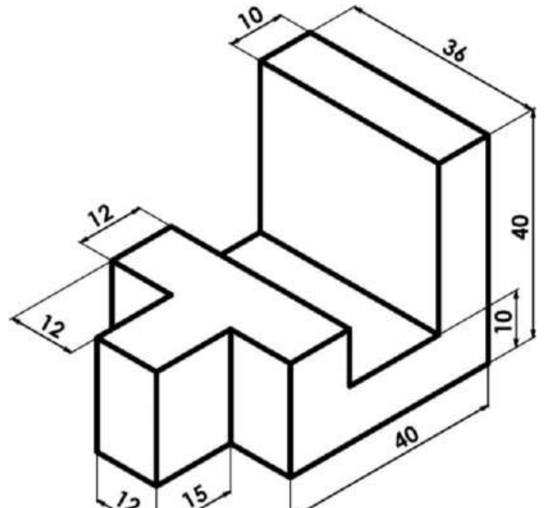
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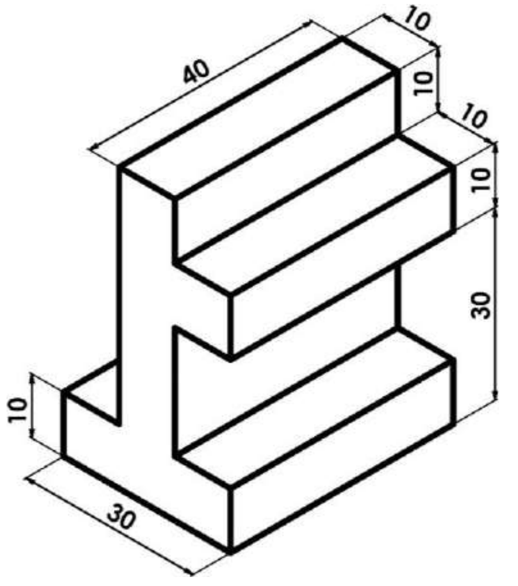
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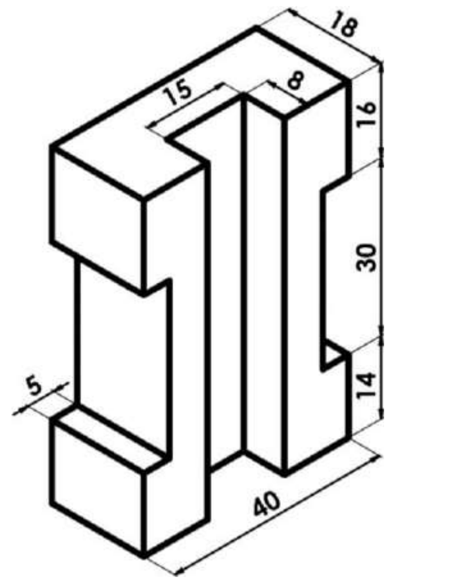
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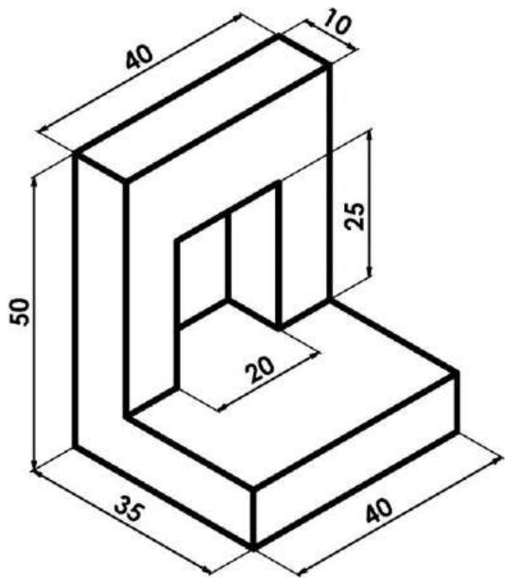
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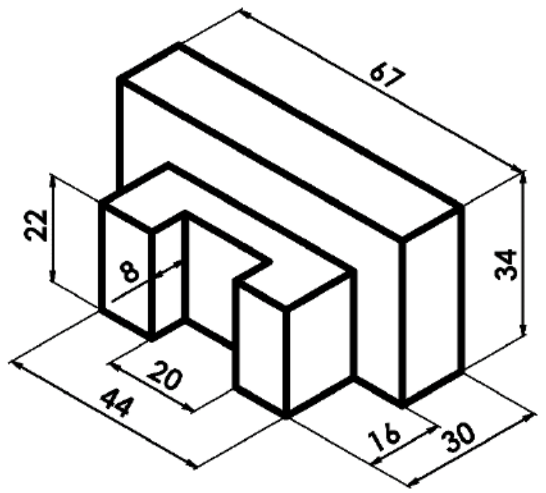
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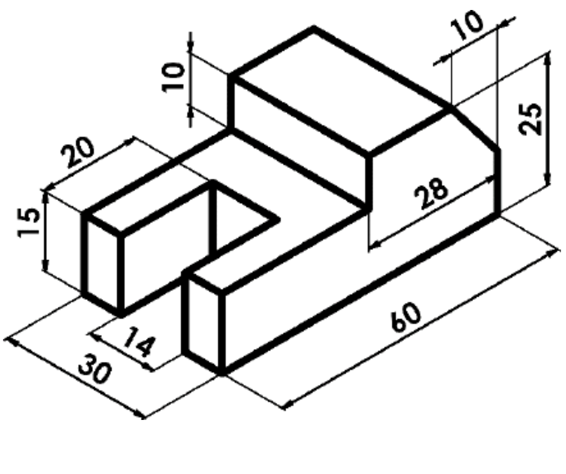
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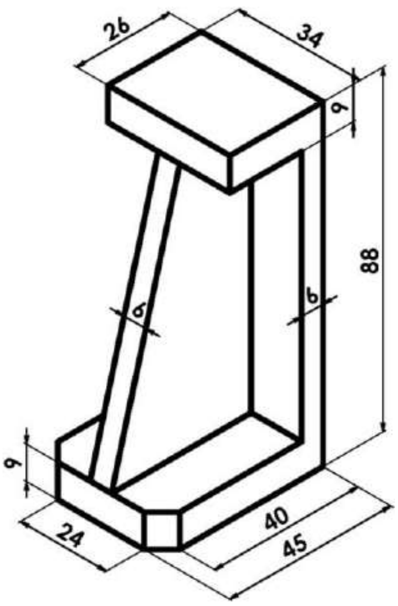
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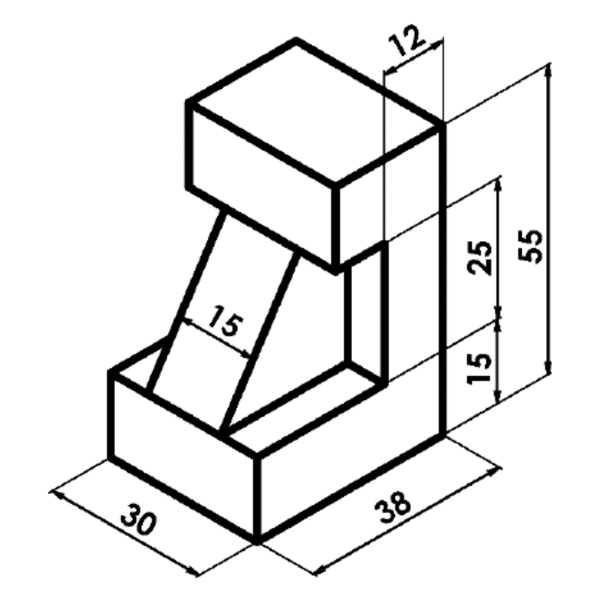
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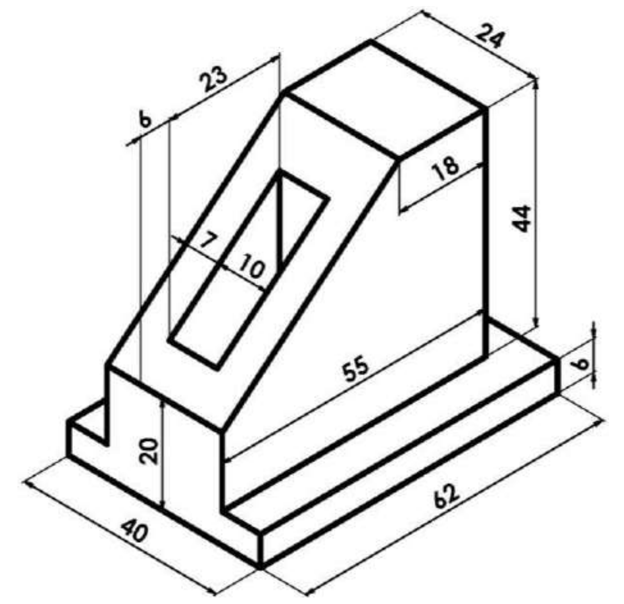
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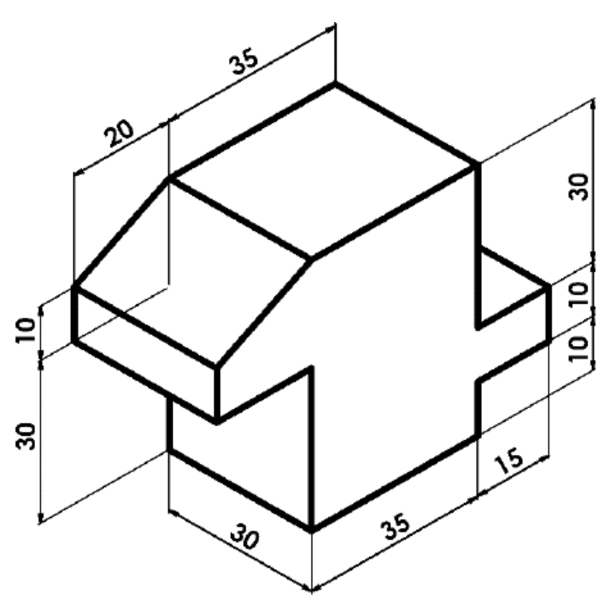
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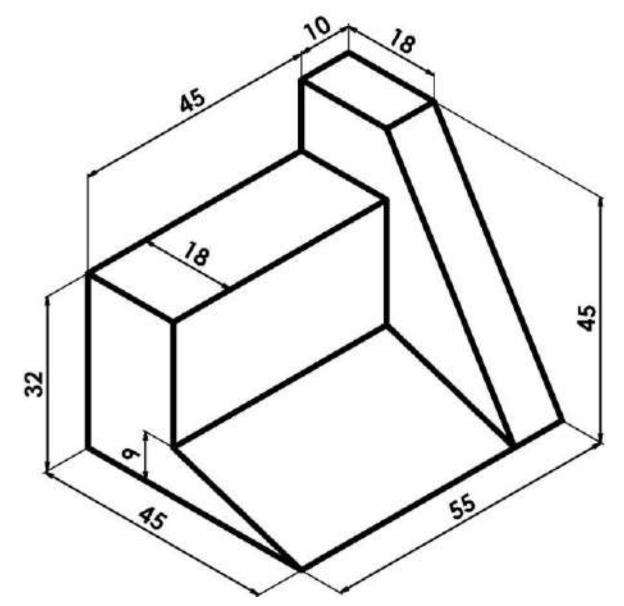
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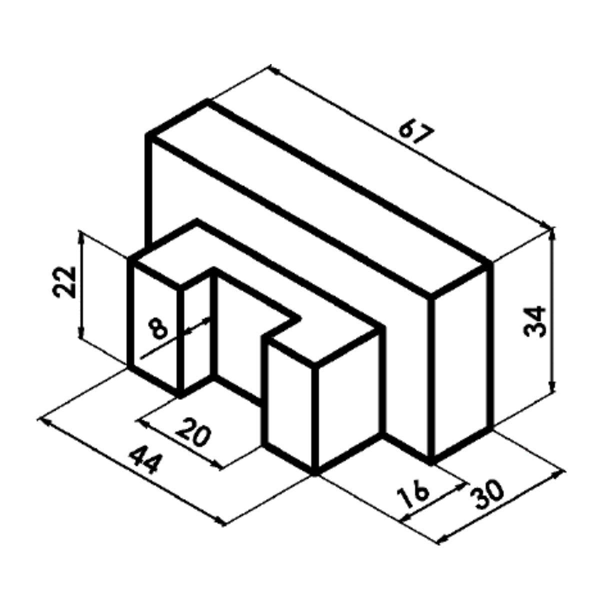
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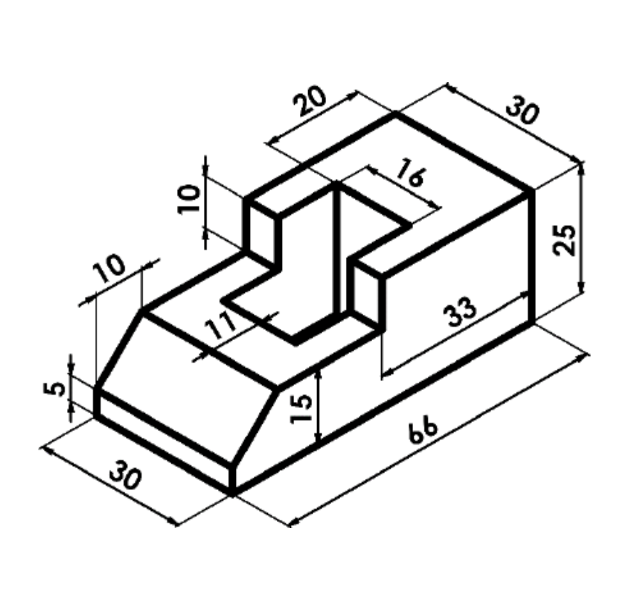
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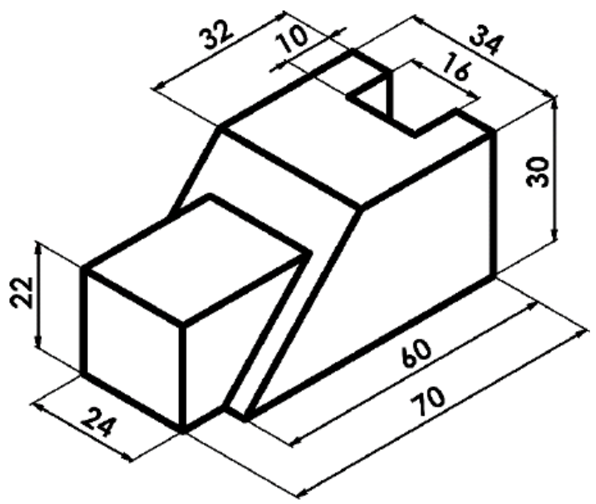
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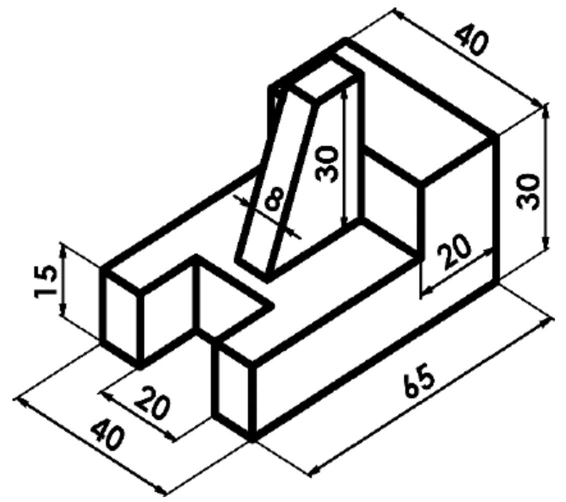
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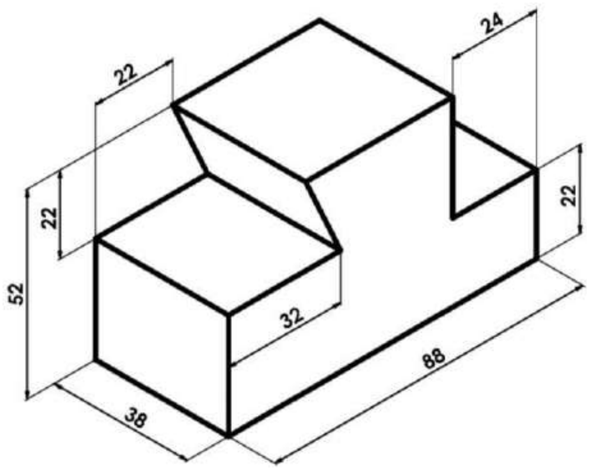
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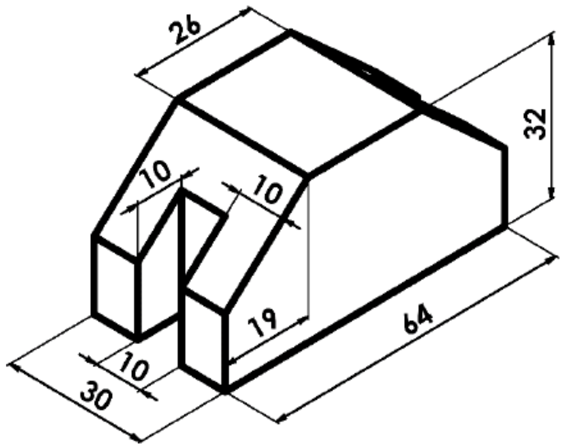
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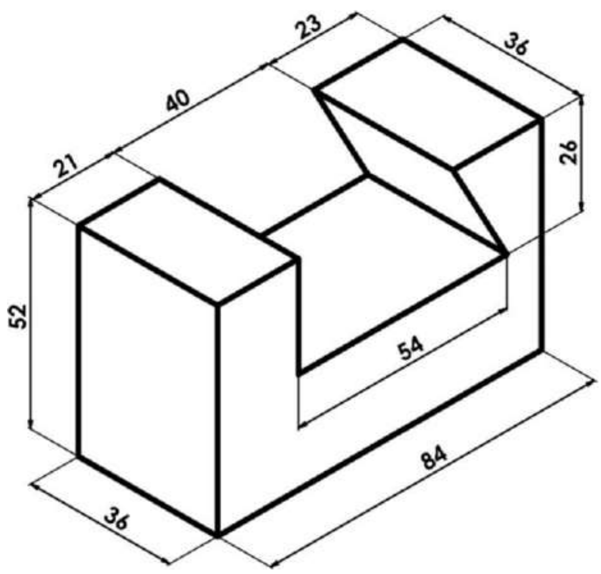
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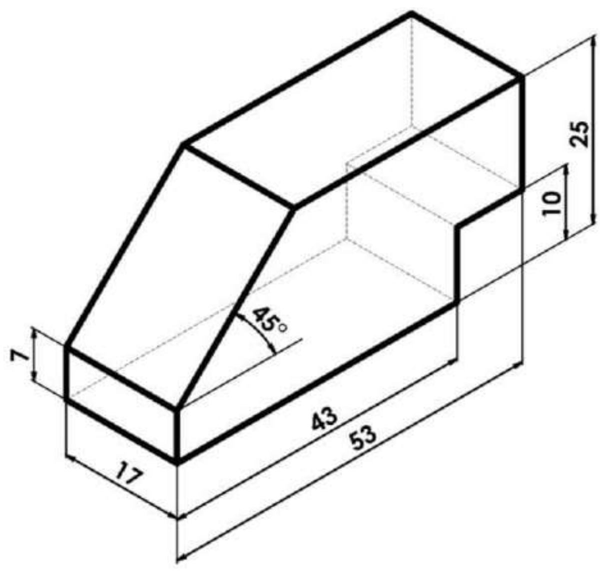
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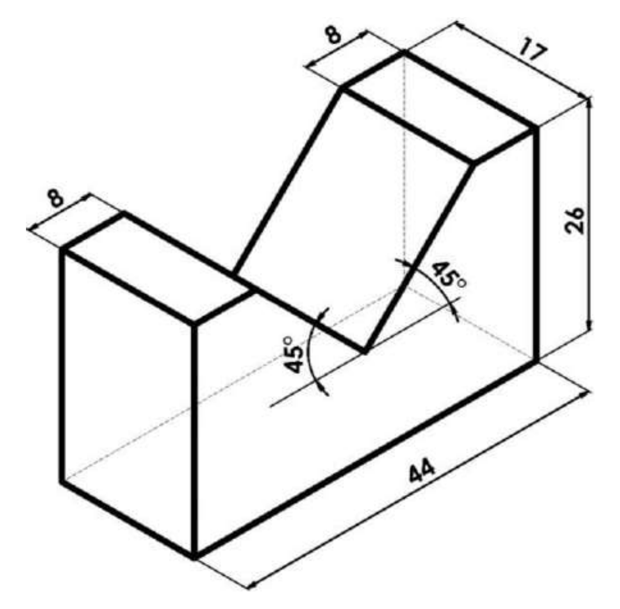
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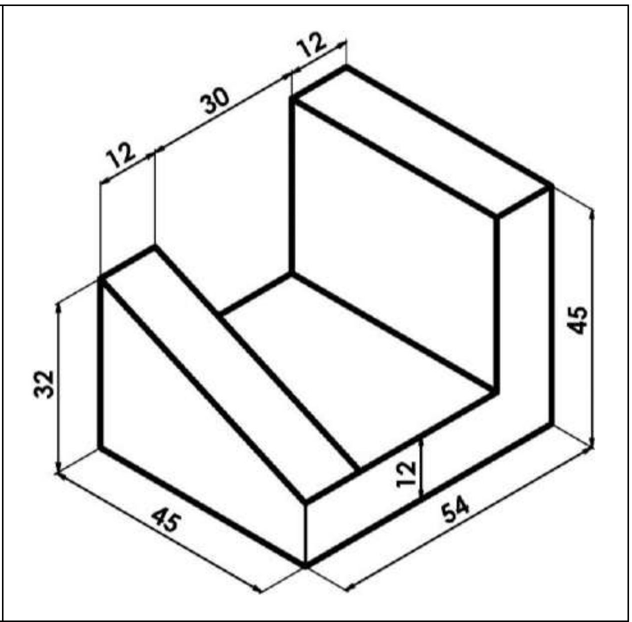
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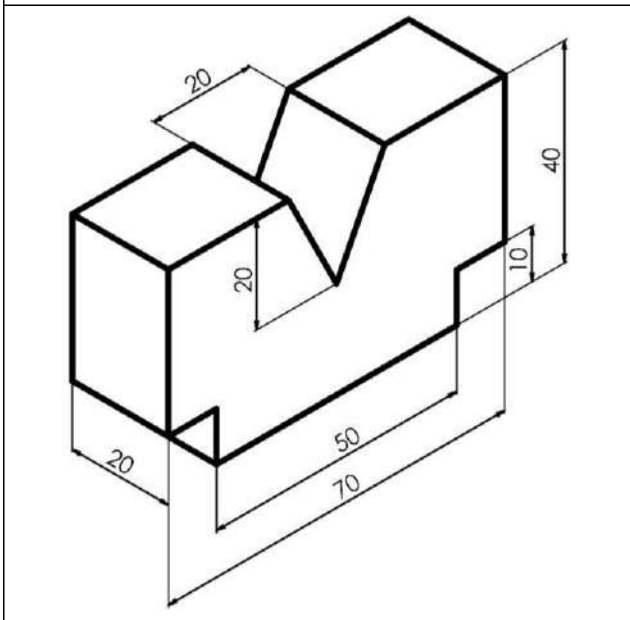
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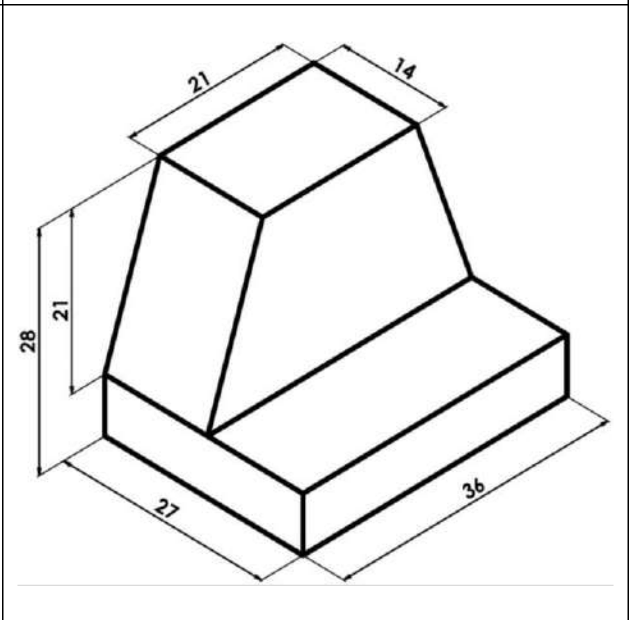
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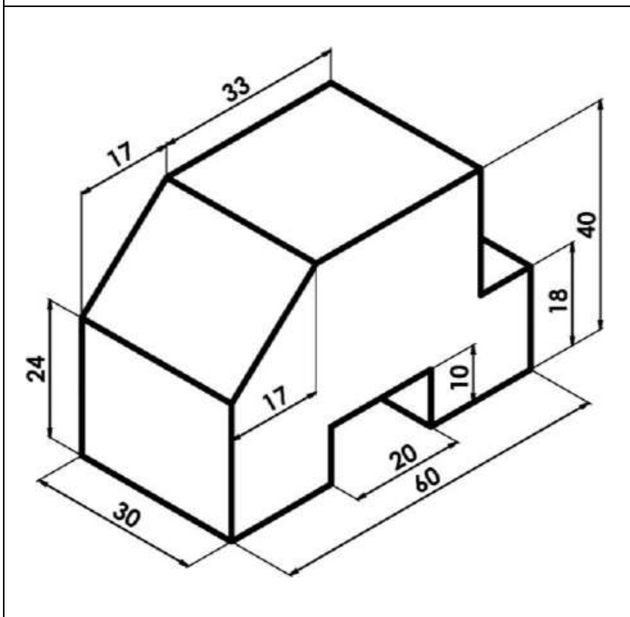
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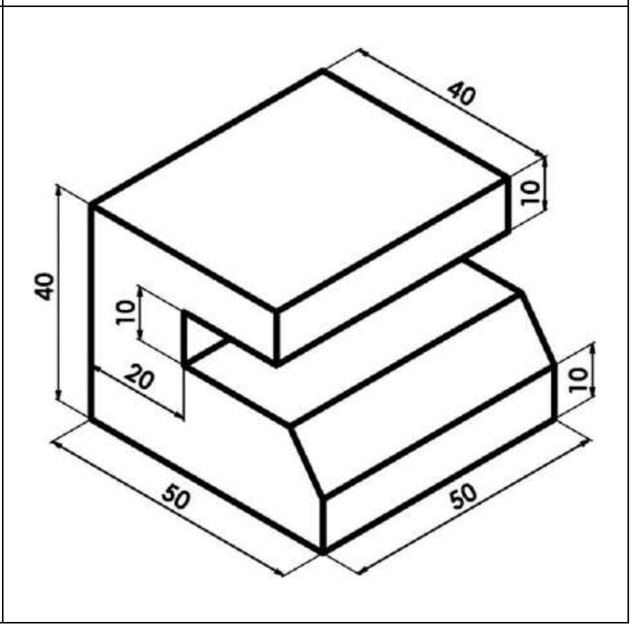
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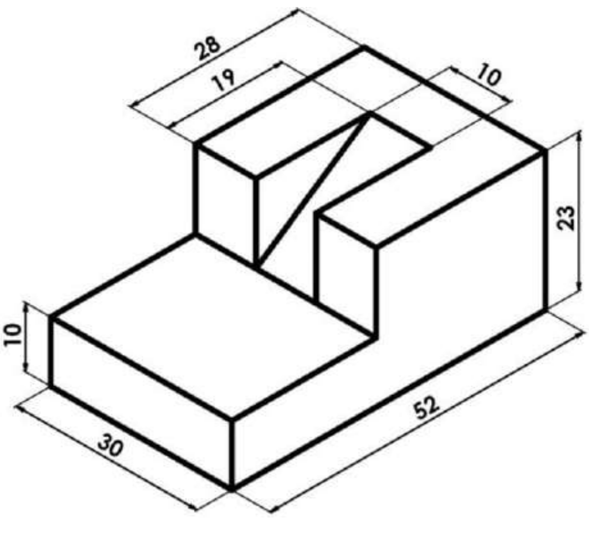
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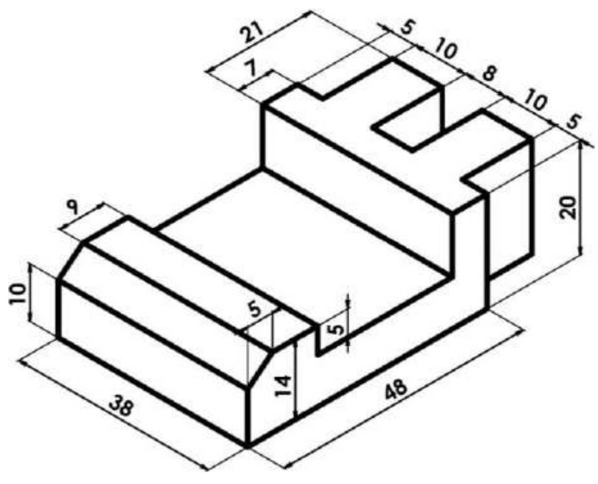
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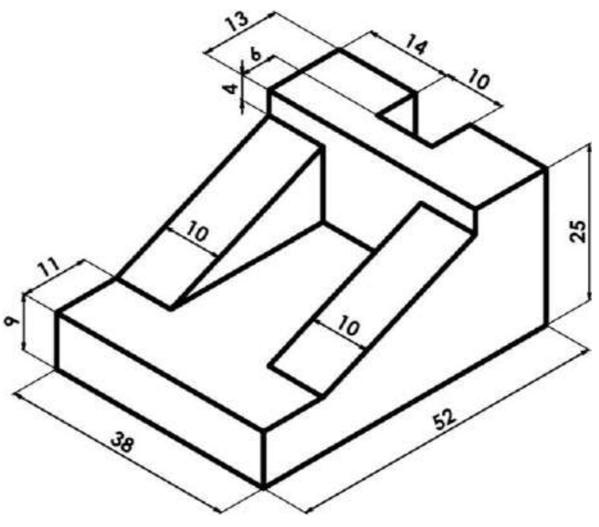
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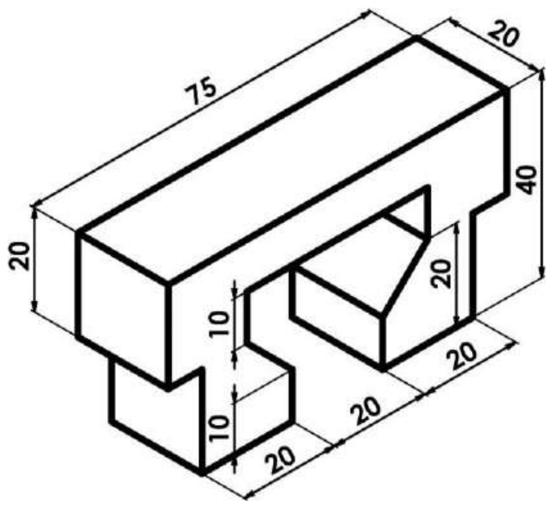
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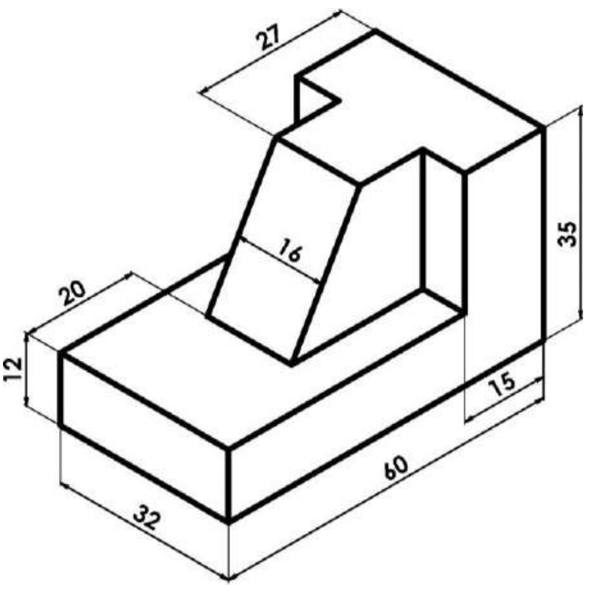
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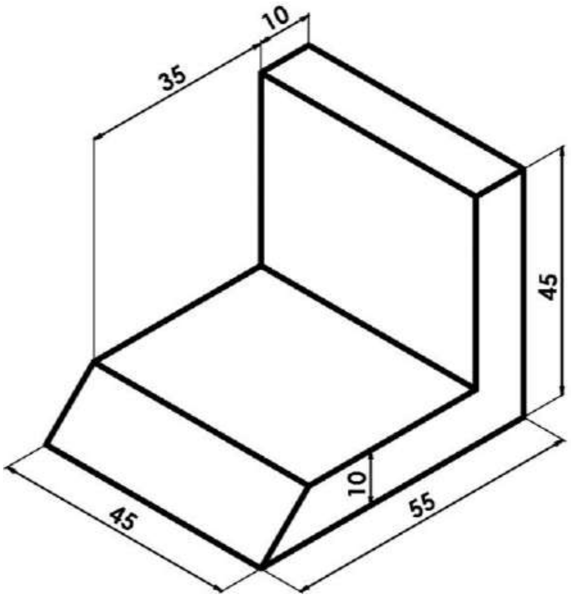
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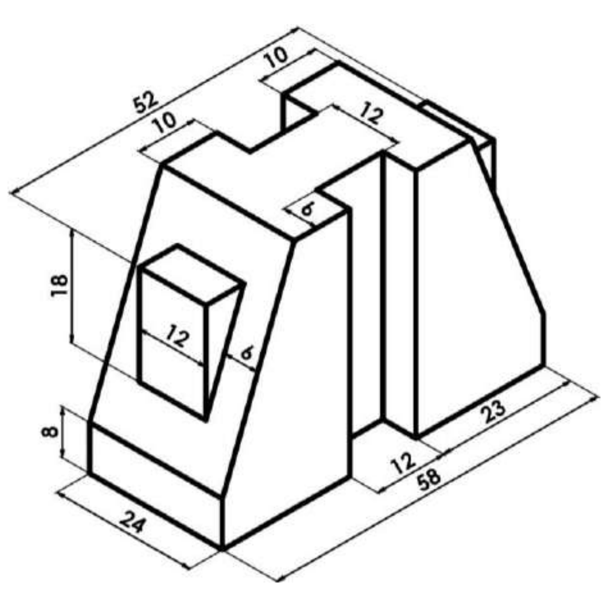
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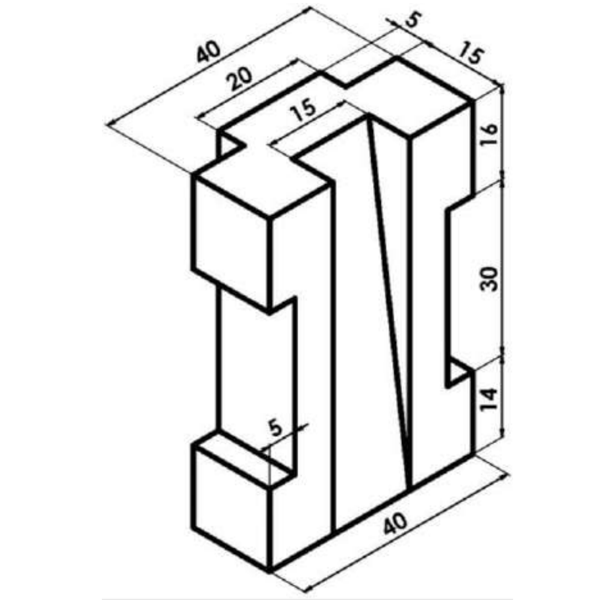
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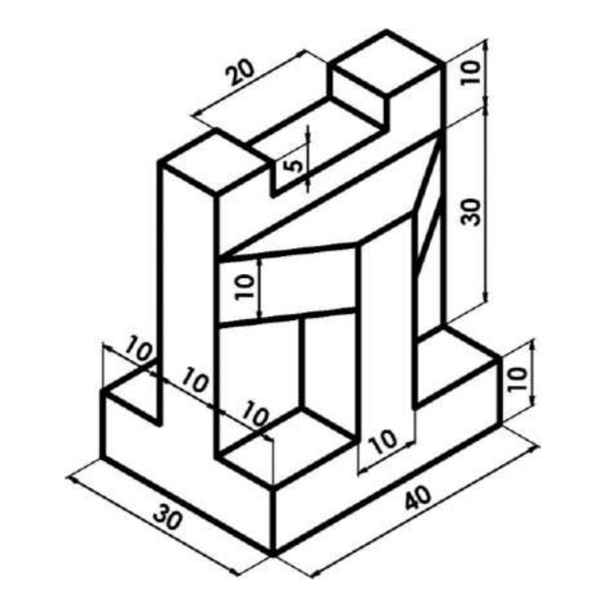
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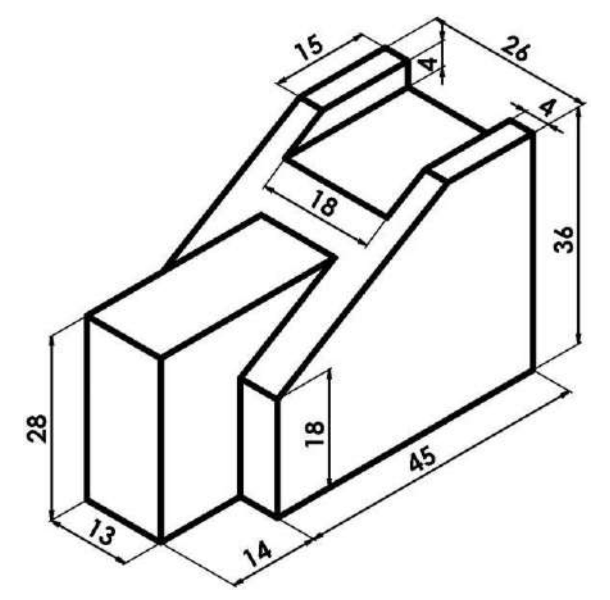
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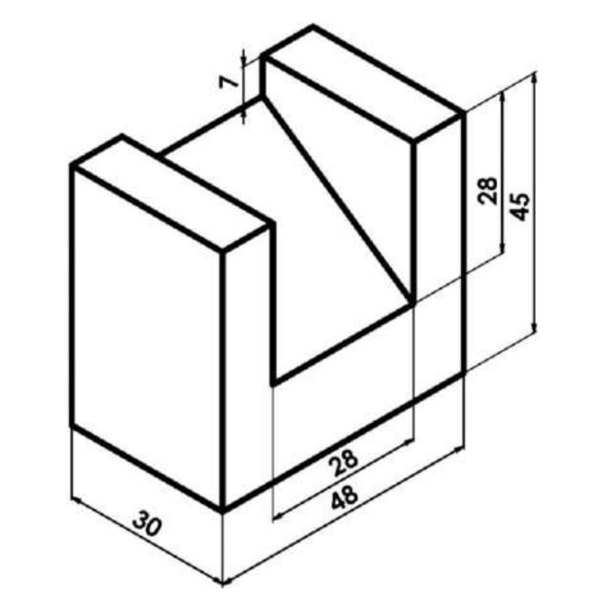
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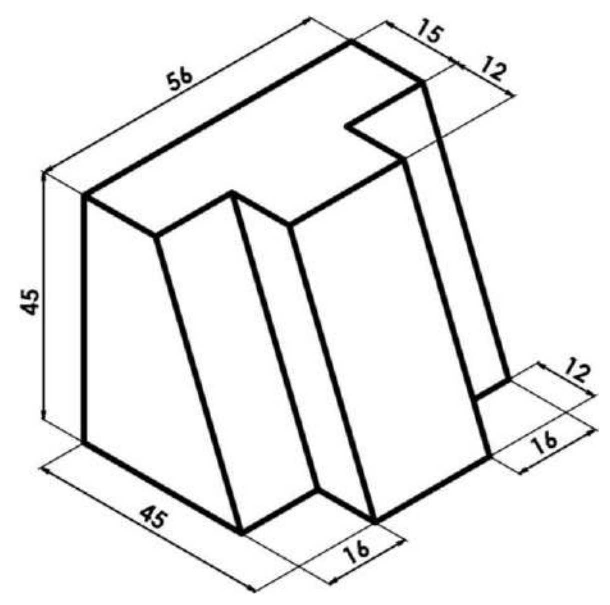
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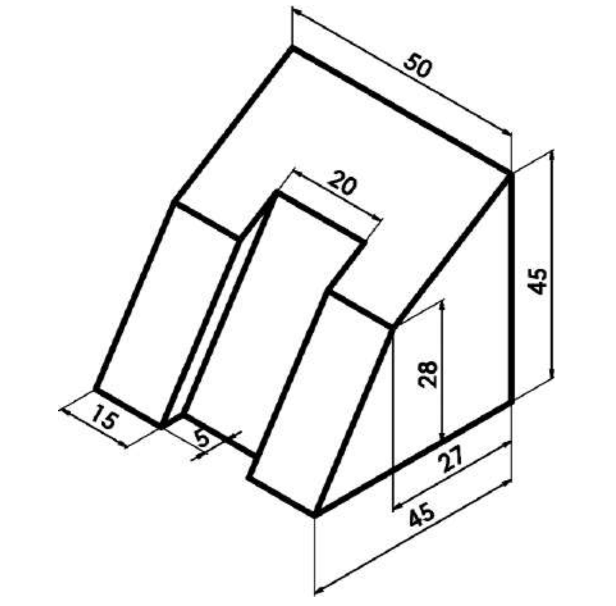
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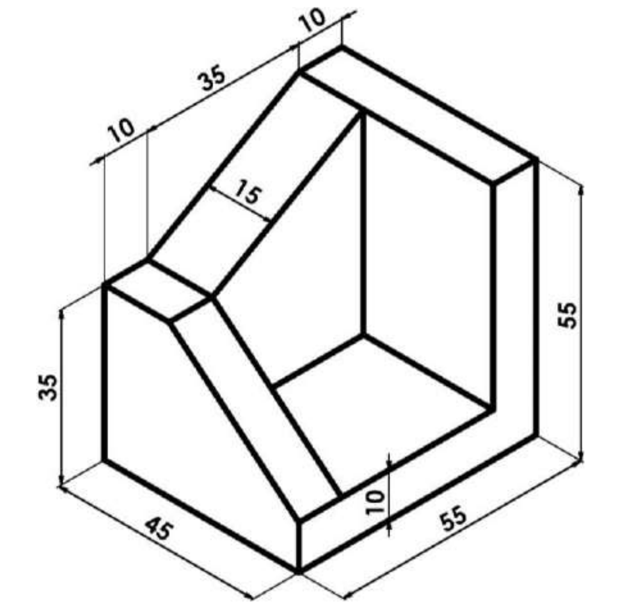
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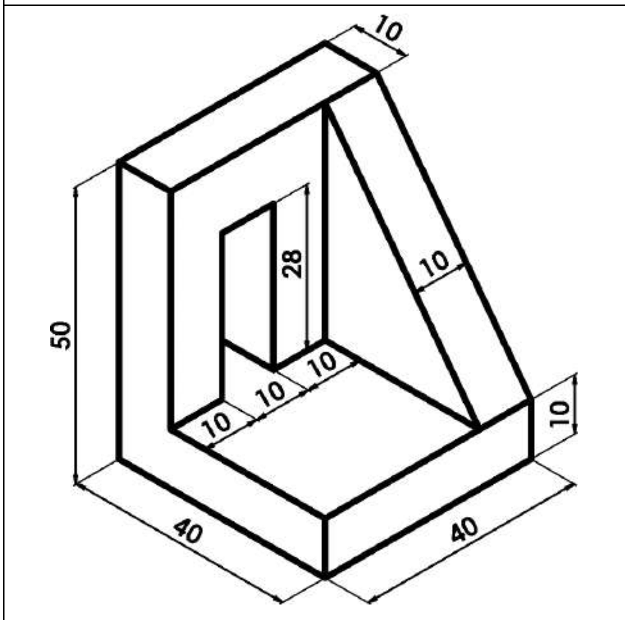
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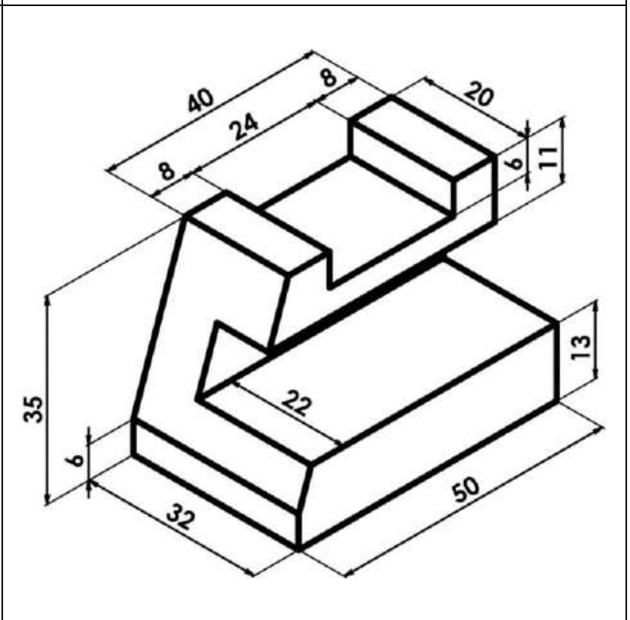
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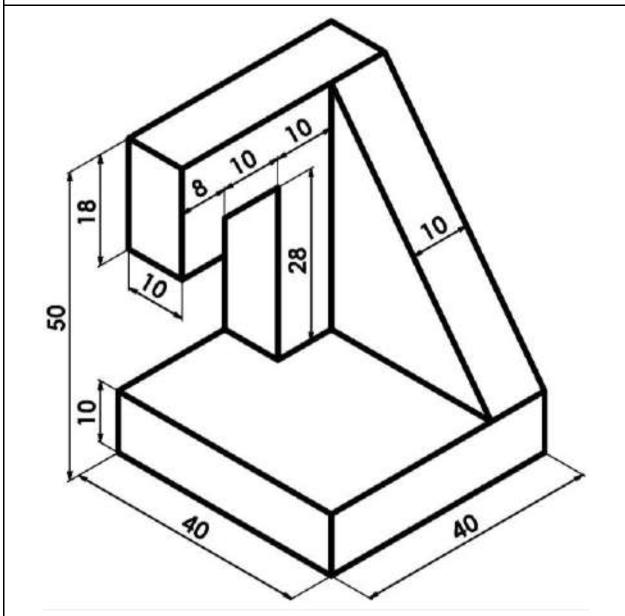
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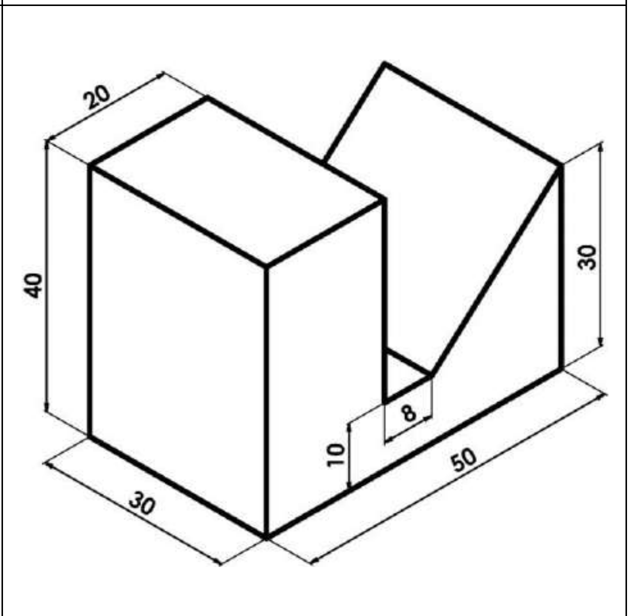
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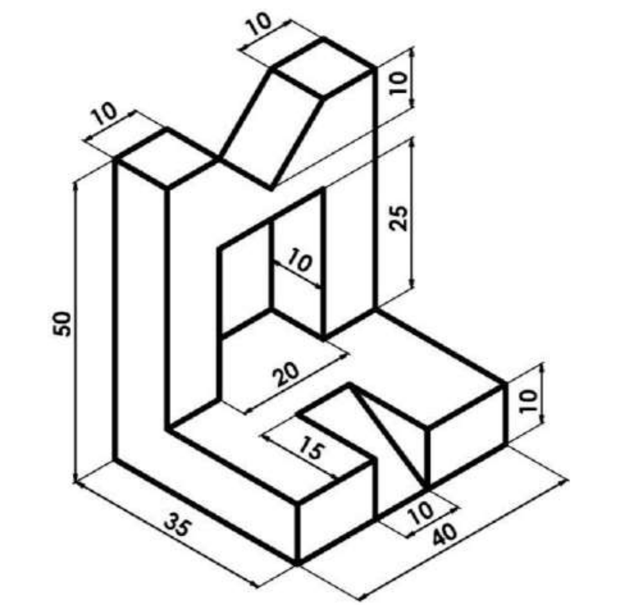
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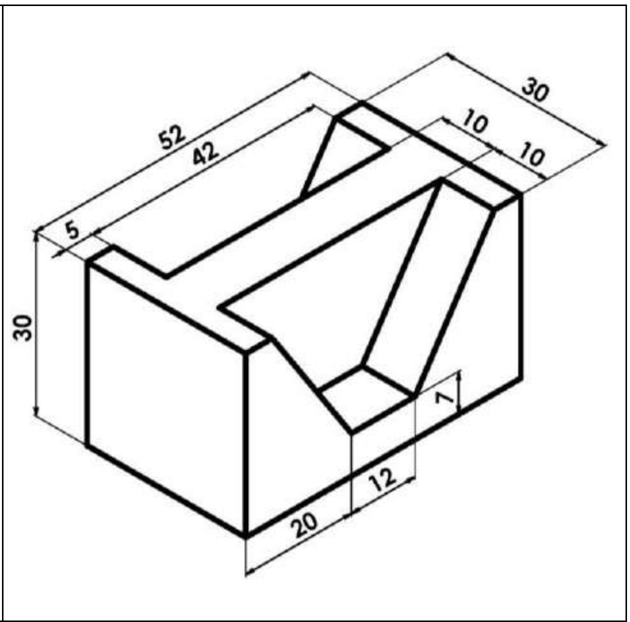
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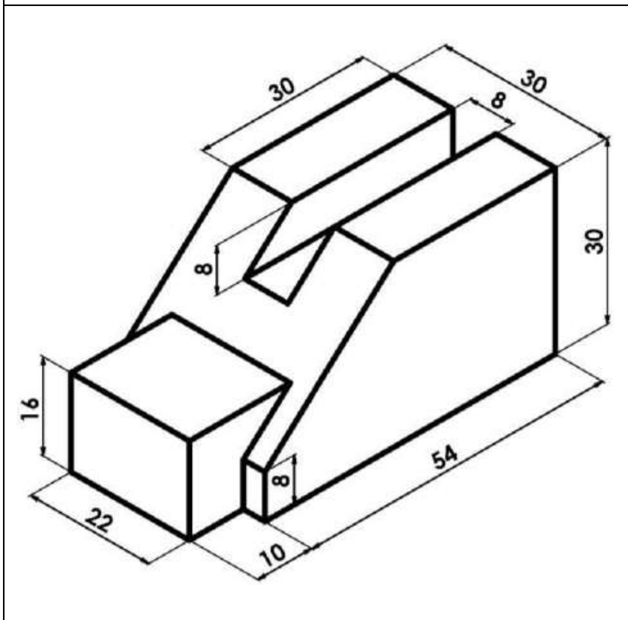
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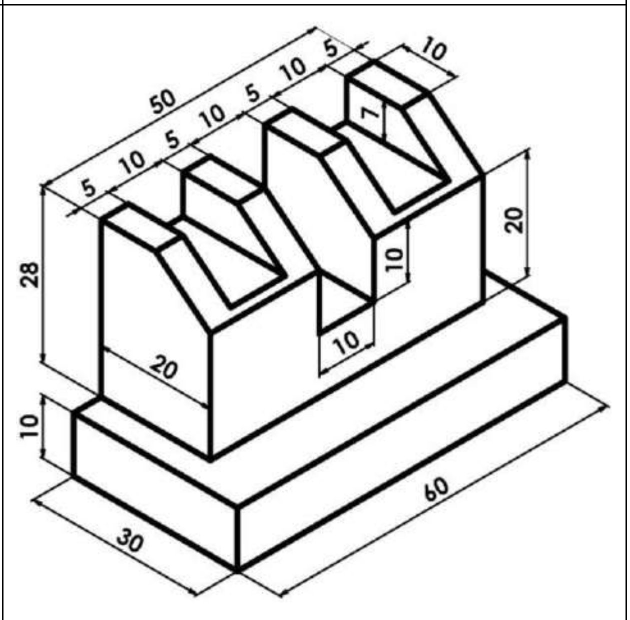
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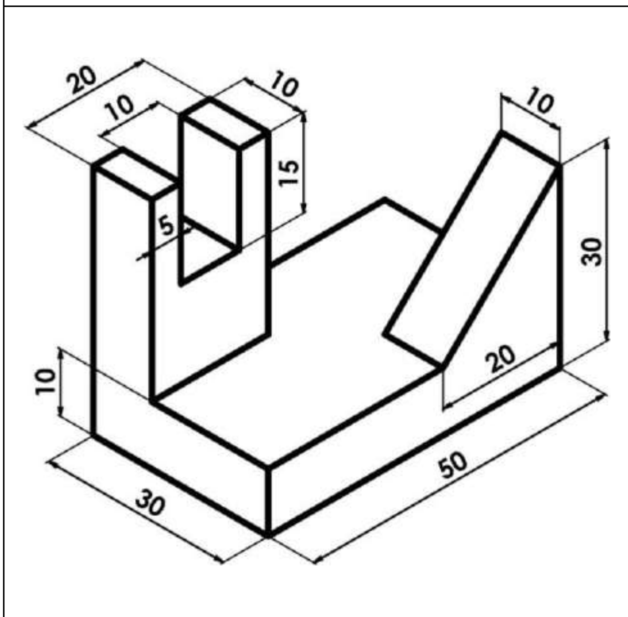
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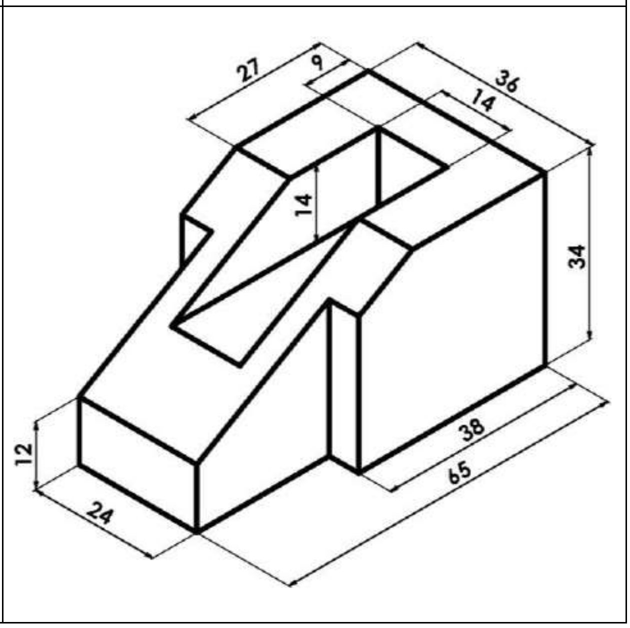
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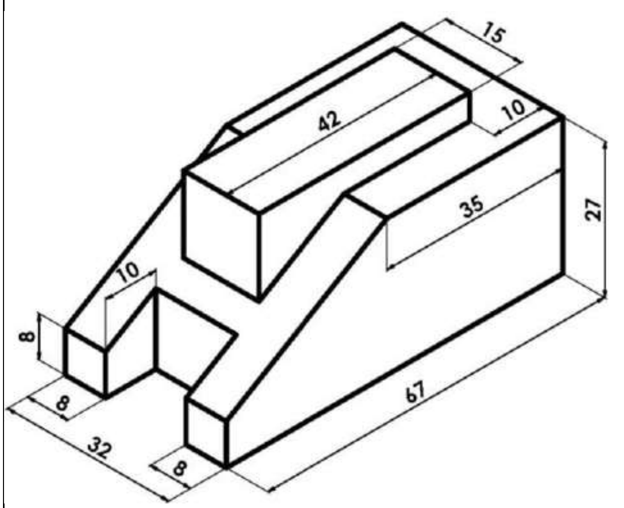
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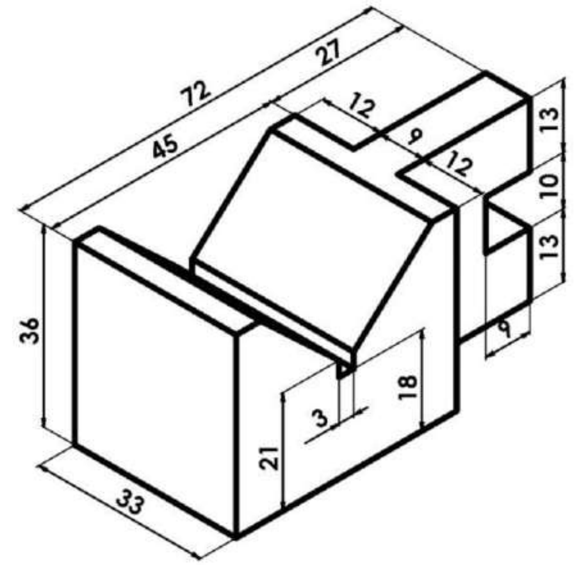
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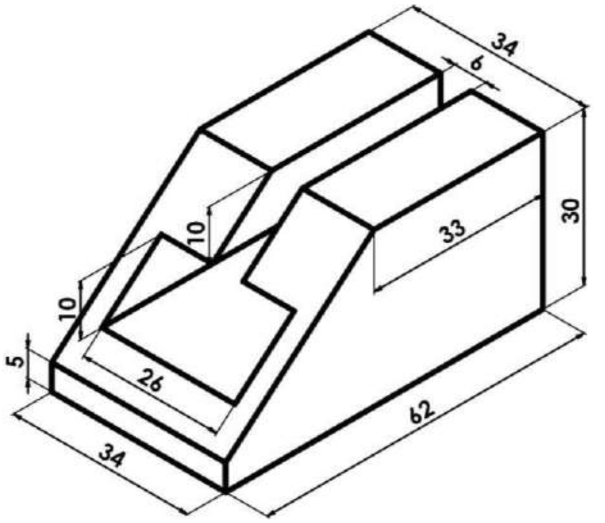
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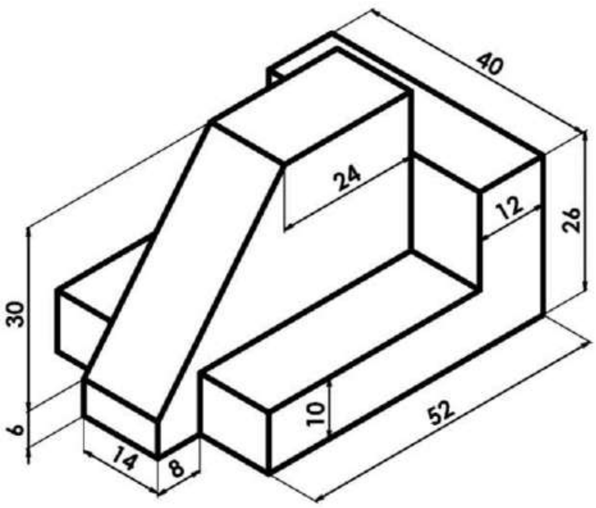
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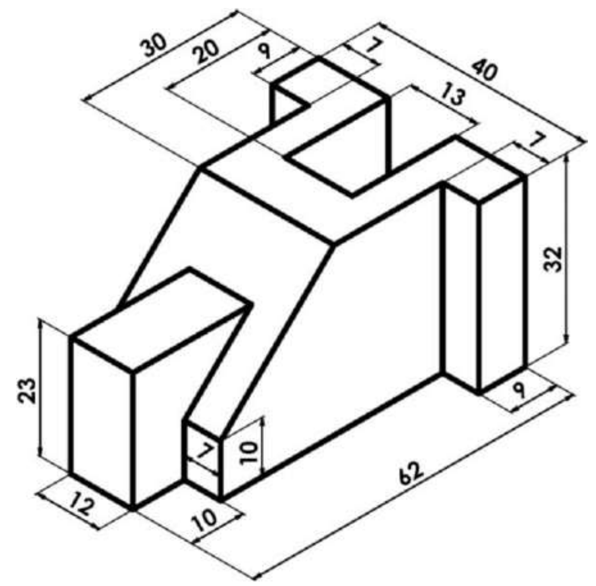
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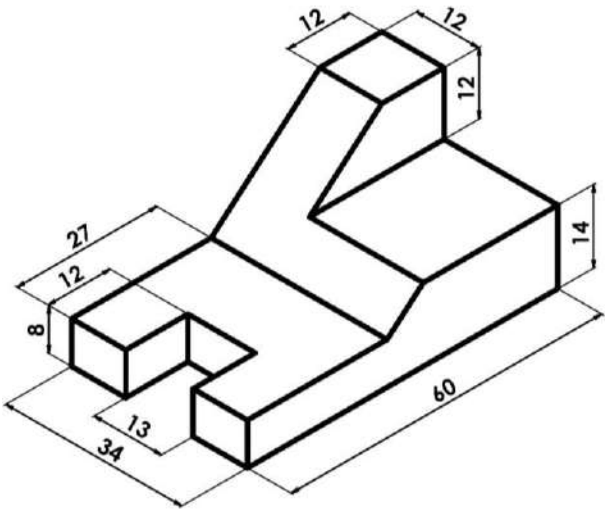
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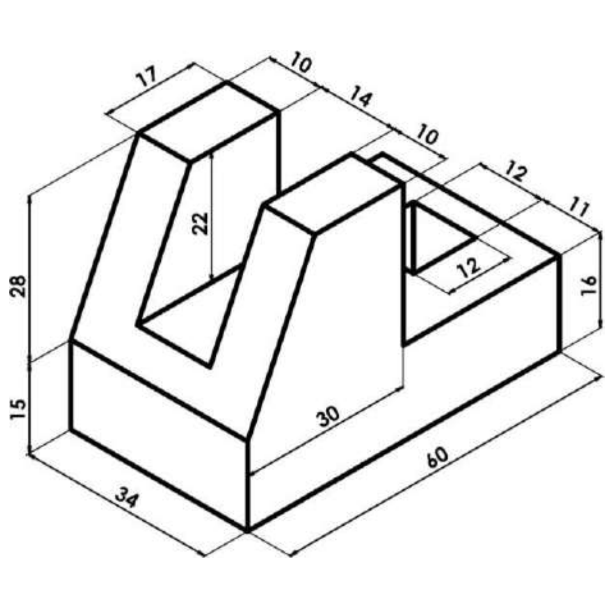
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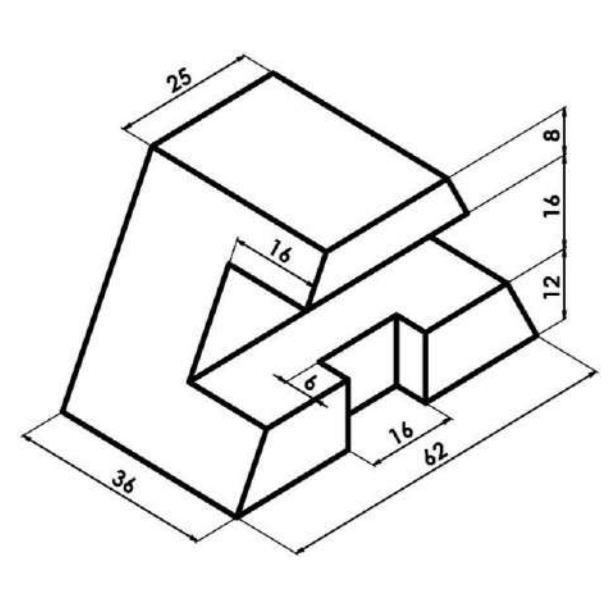
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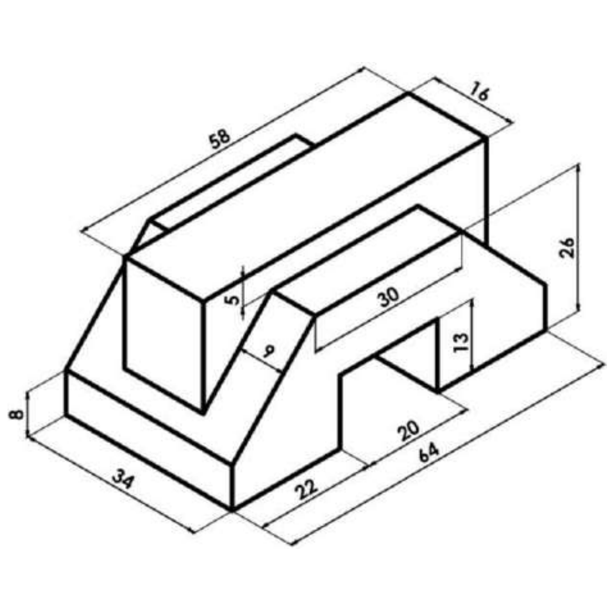
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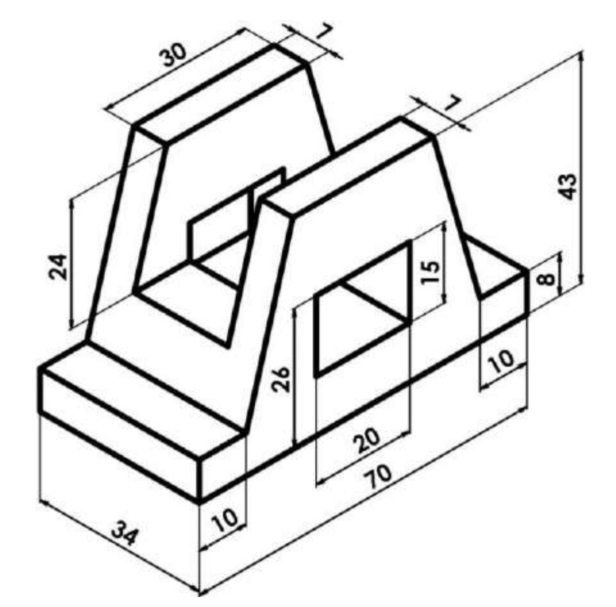
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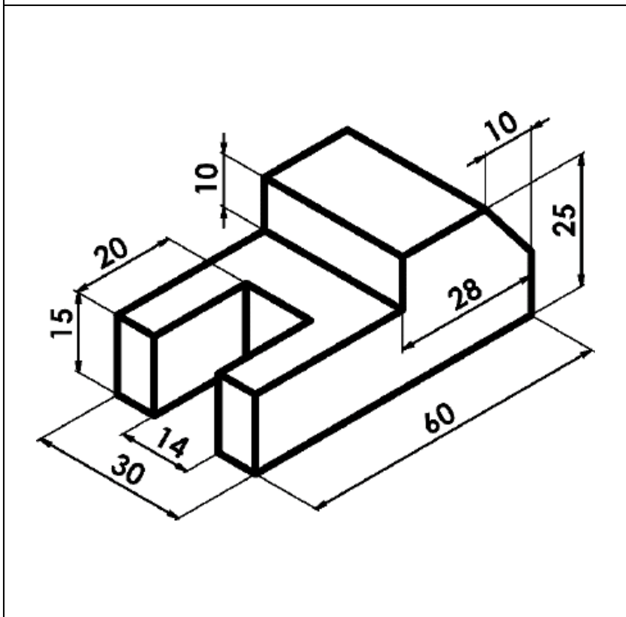
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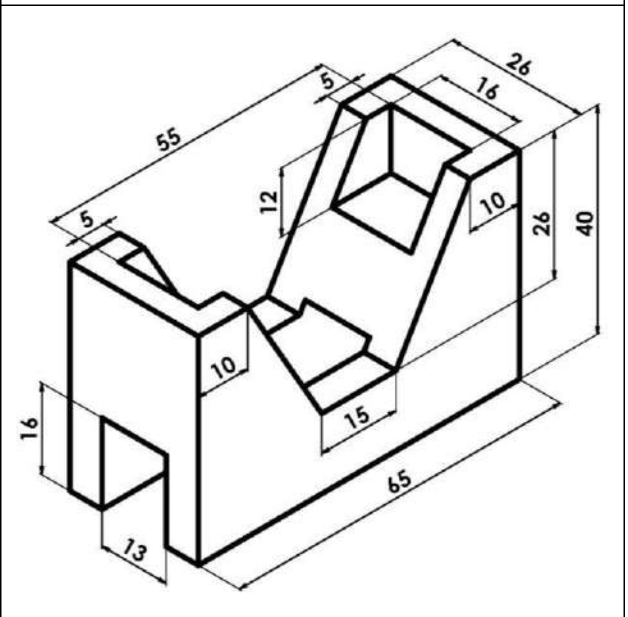
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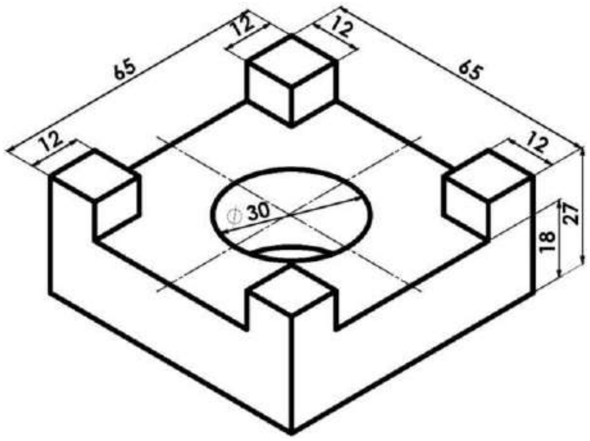
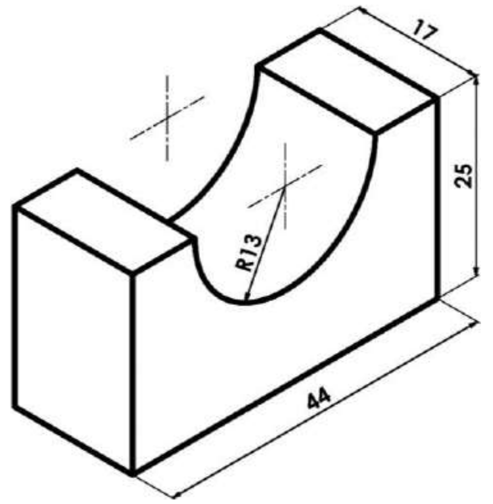
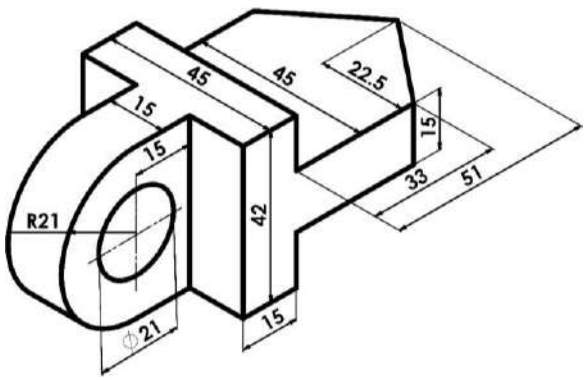
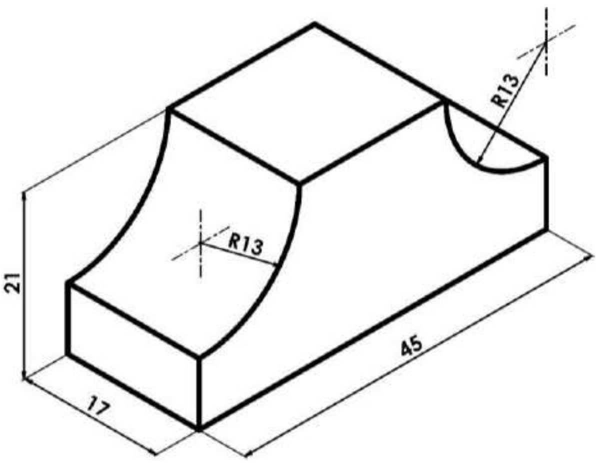
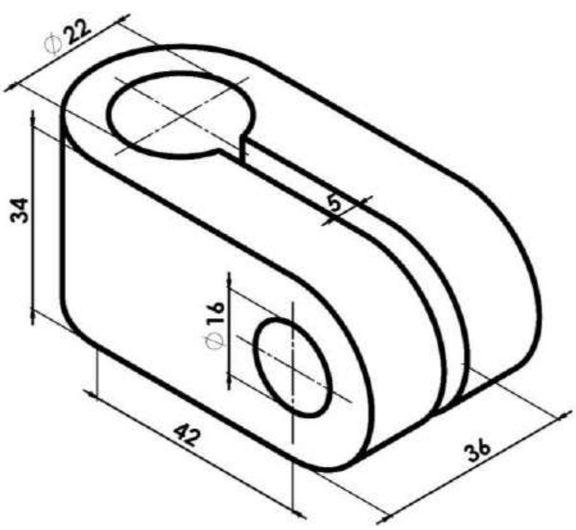
2. Draw the following curved geometry isometrics

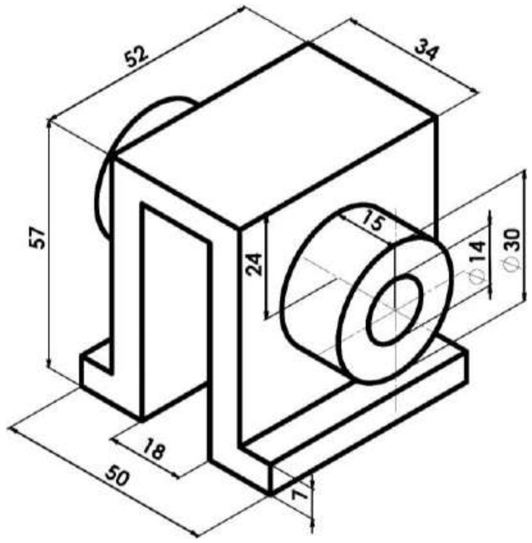


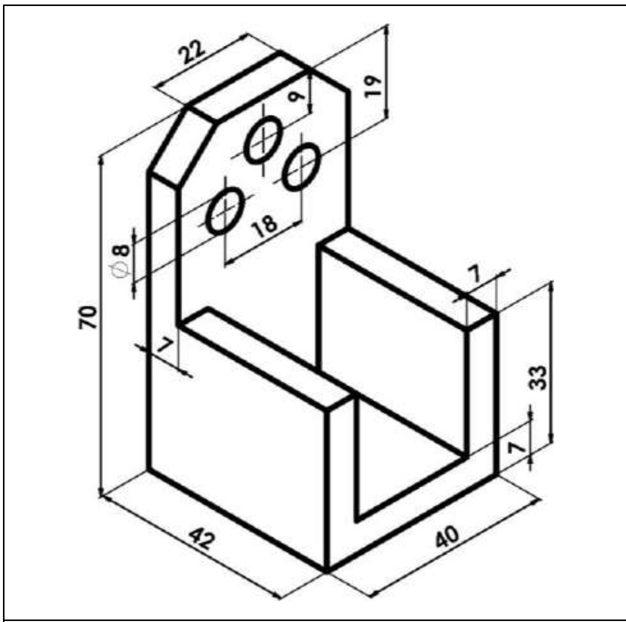








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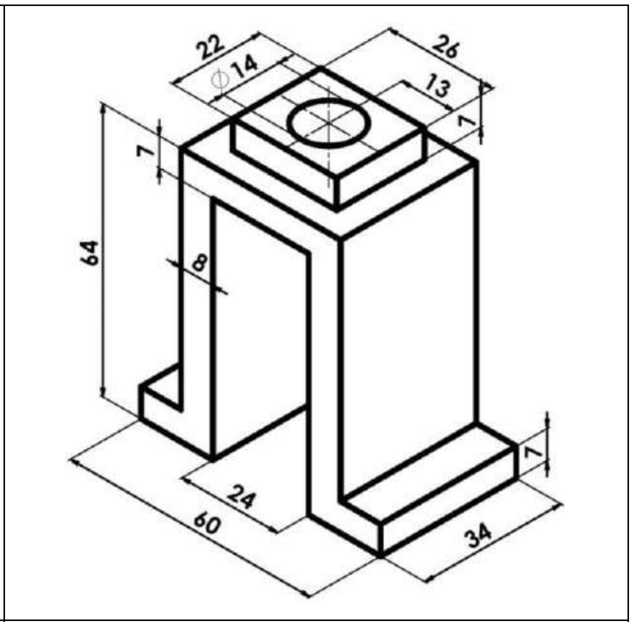
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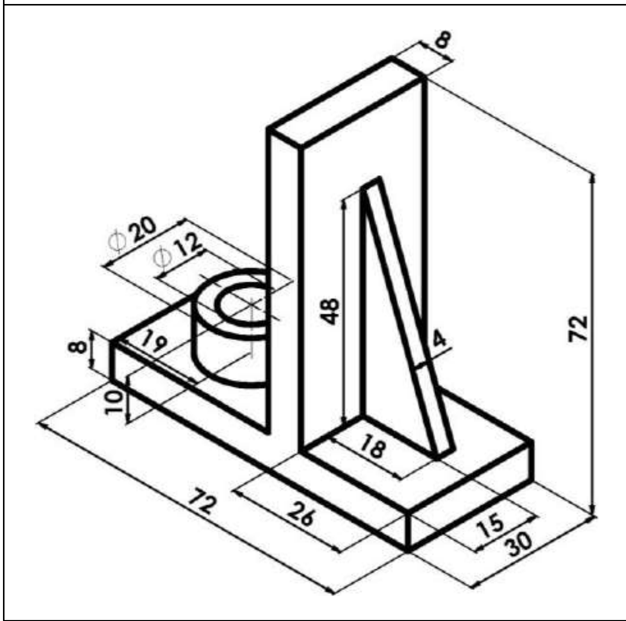
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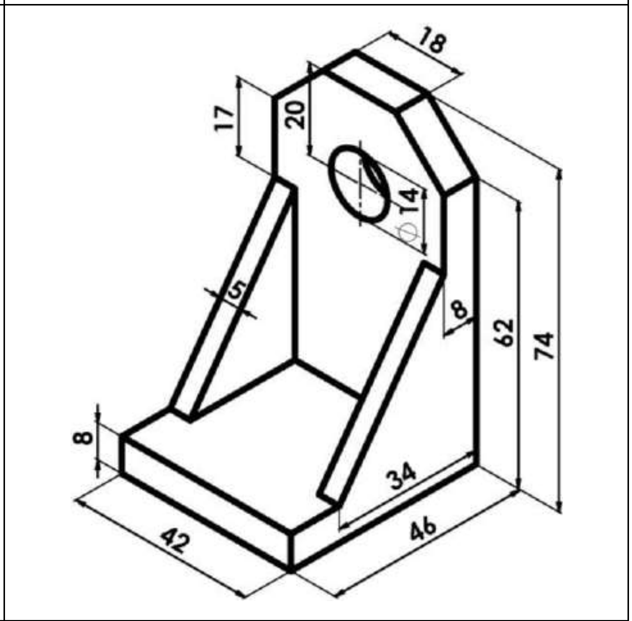
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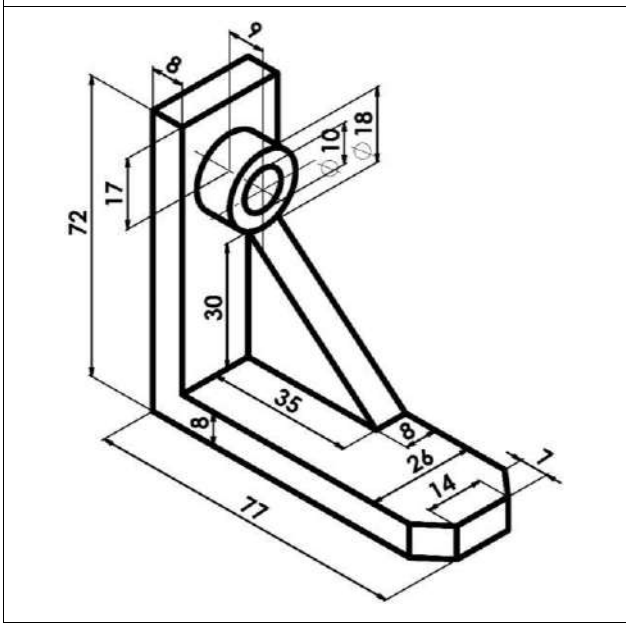
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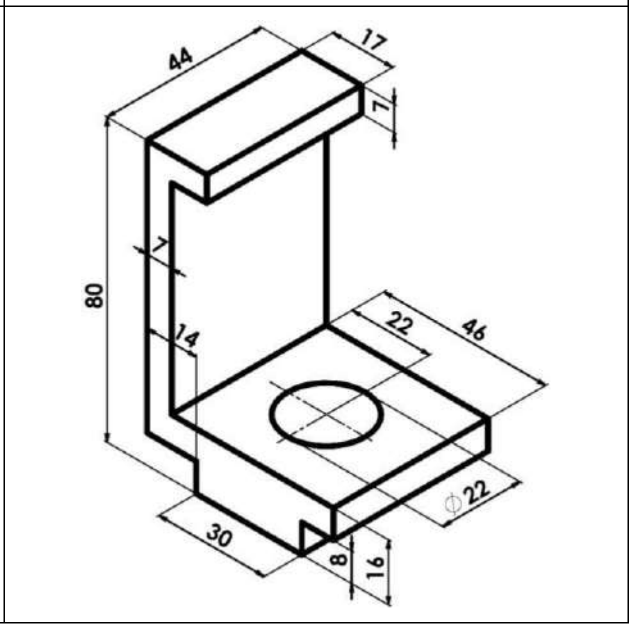
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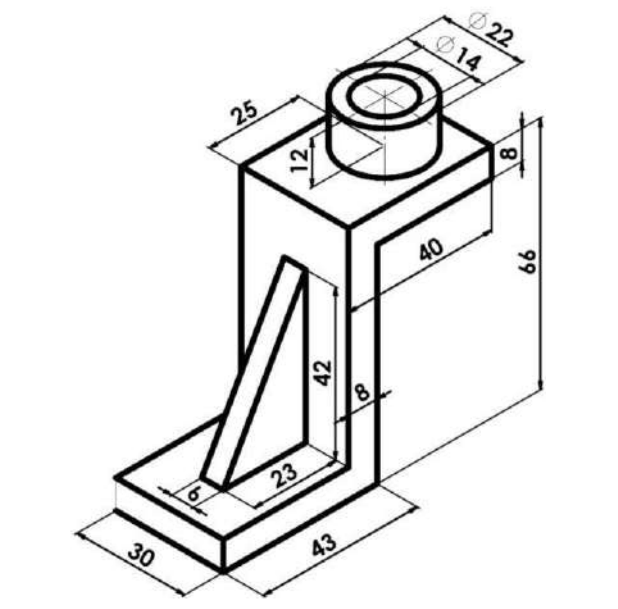
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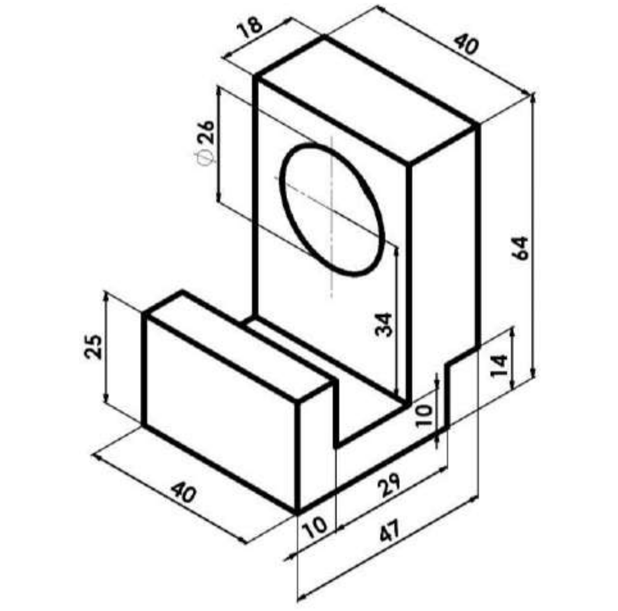
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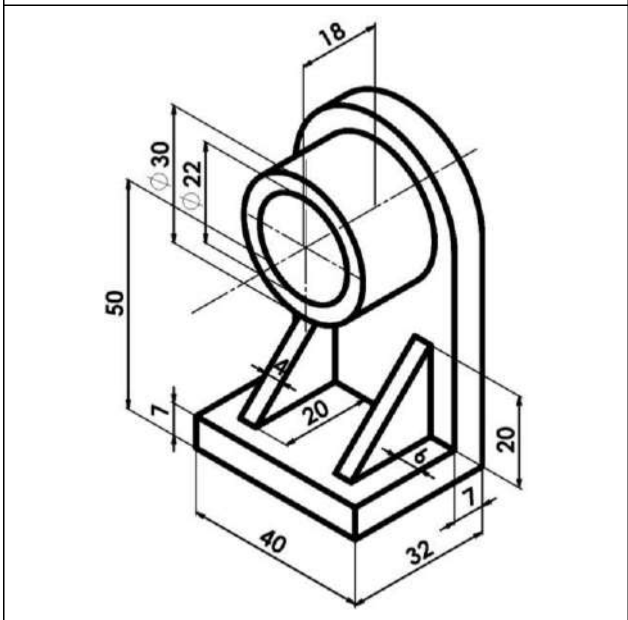
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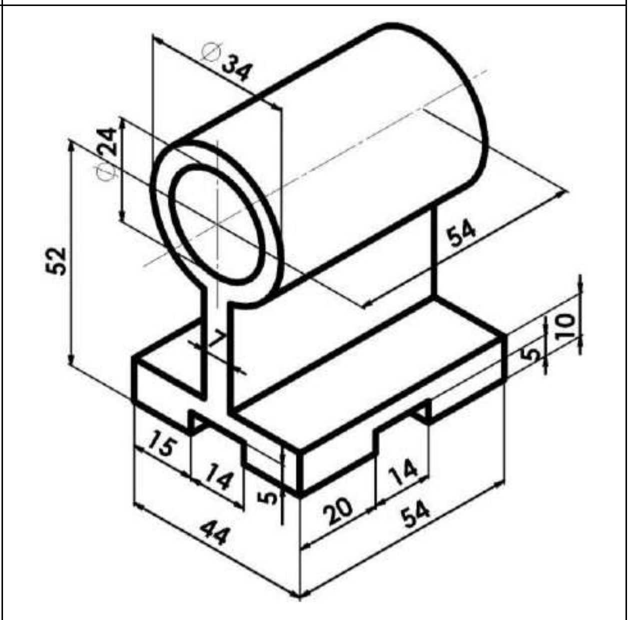
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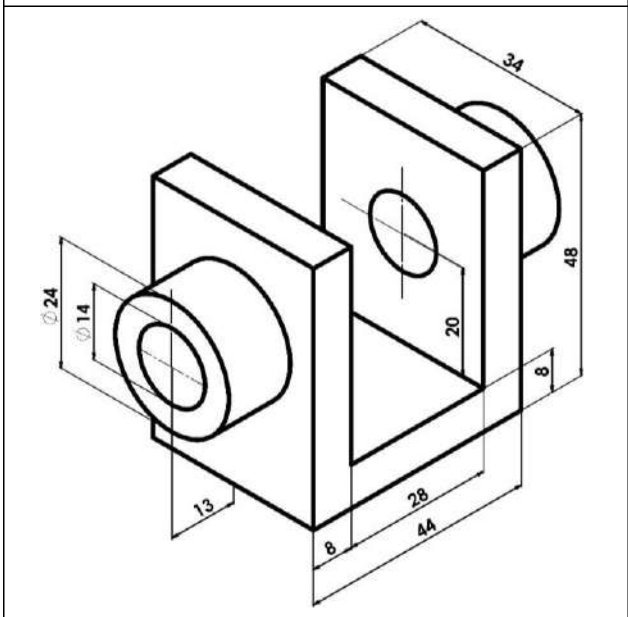
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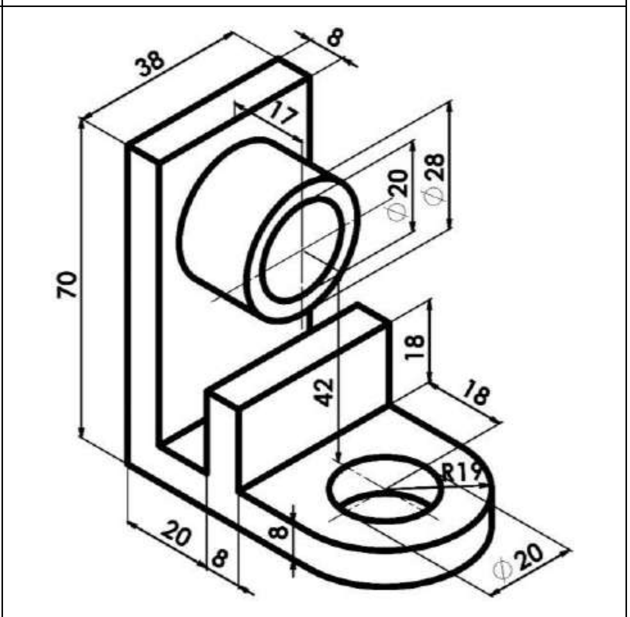
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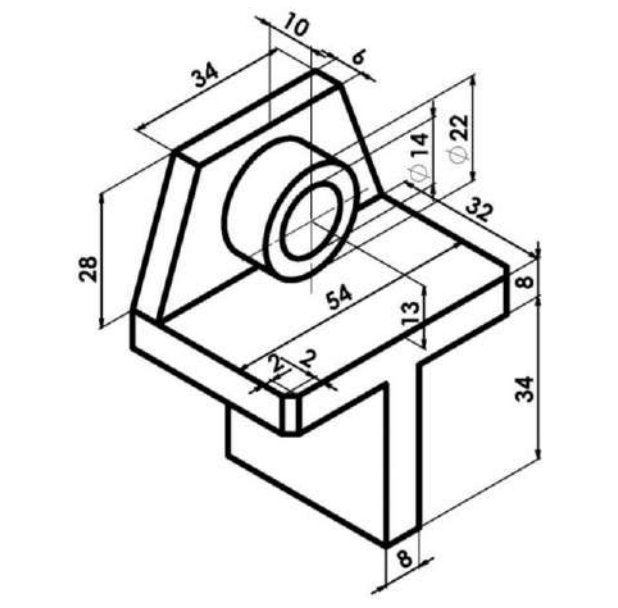
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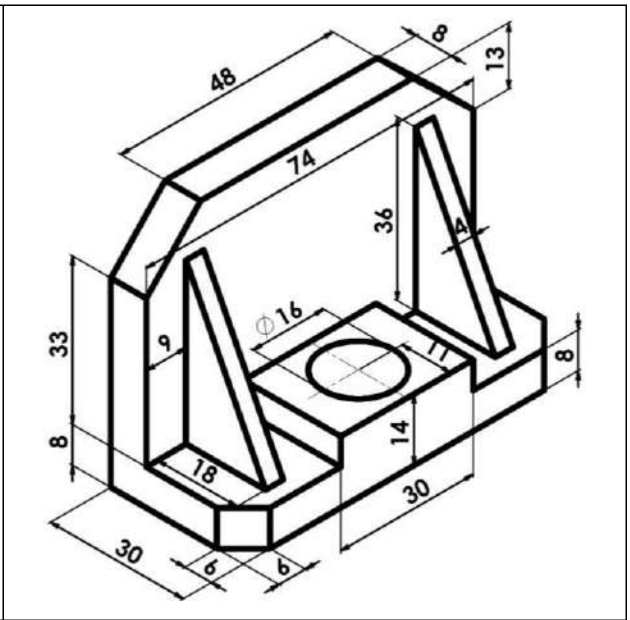
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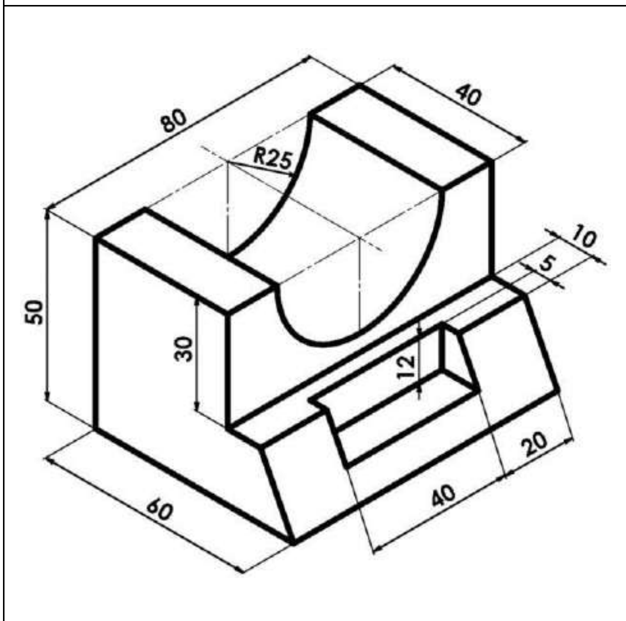
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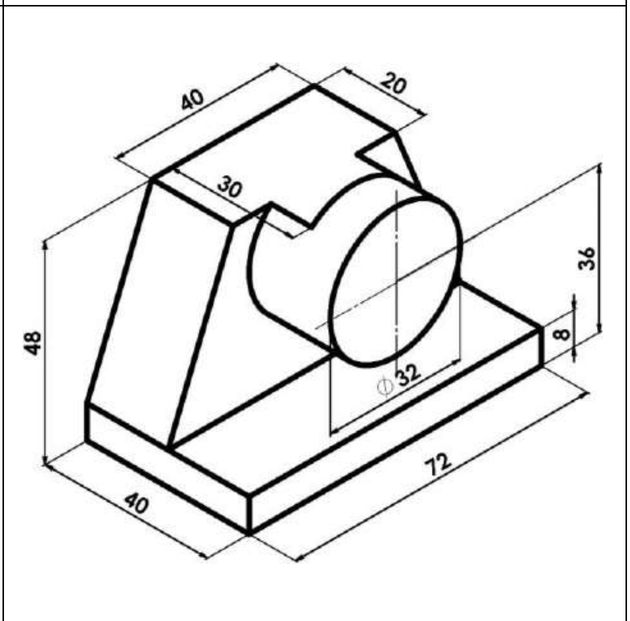
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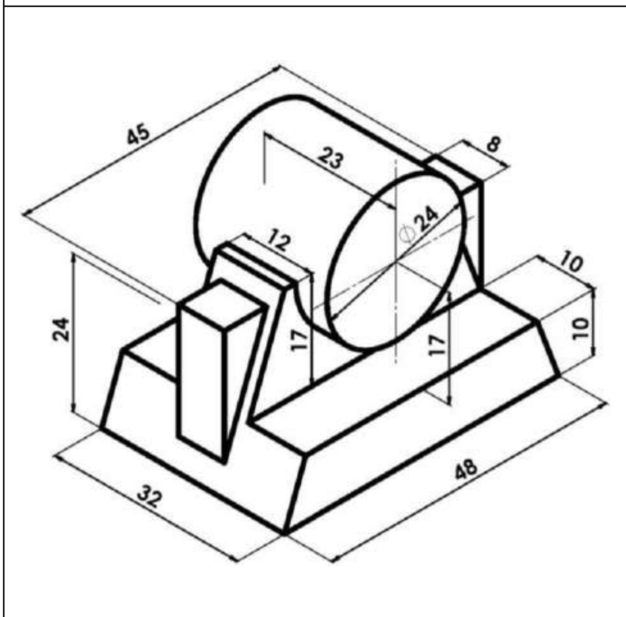
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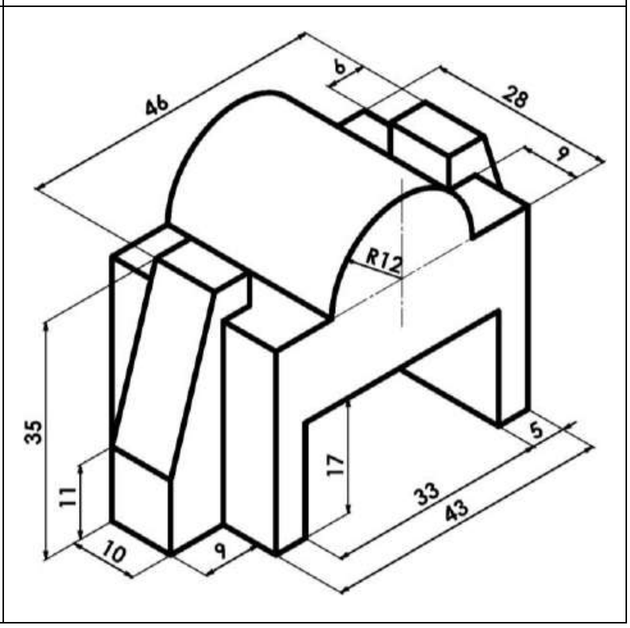
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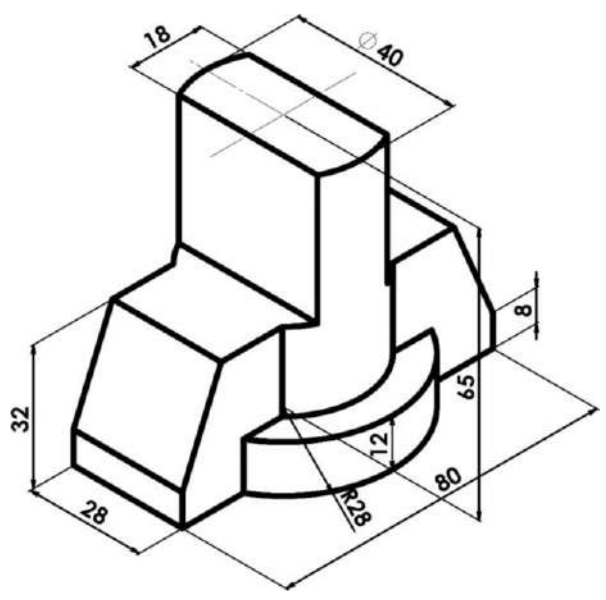
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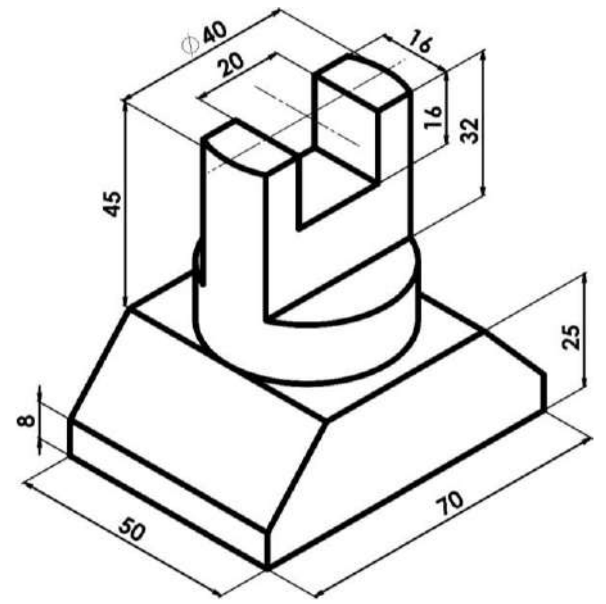
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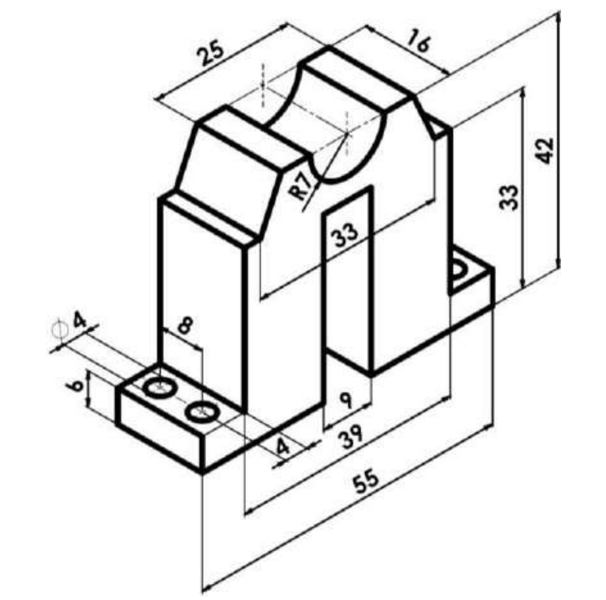
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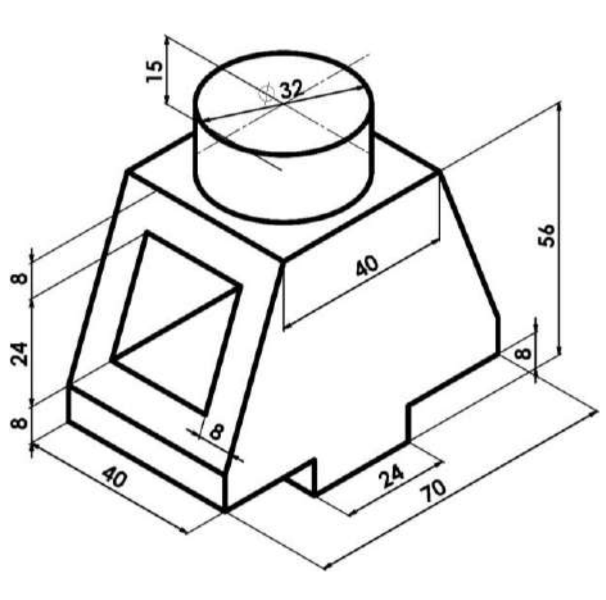
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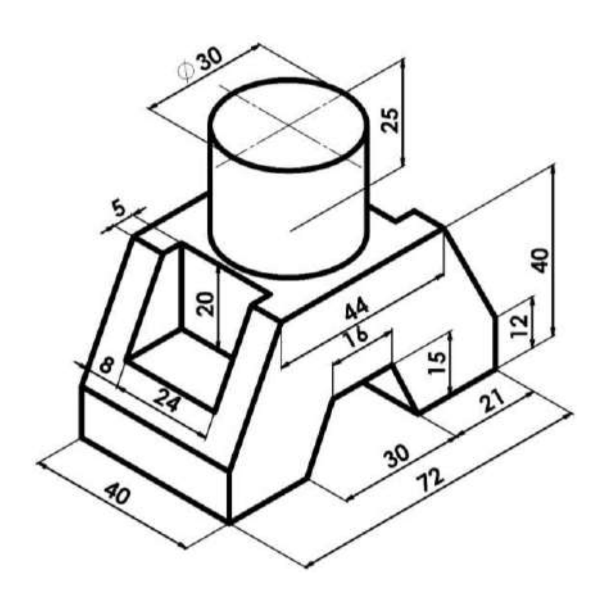
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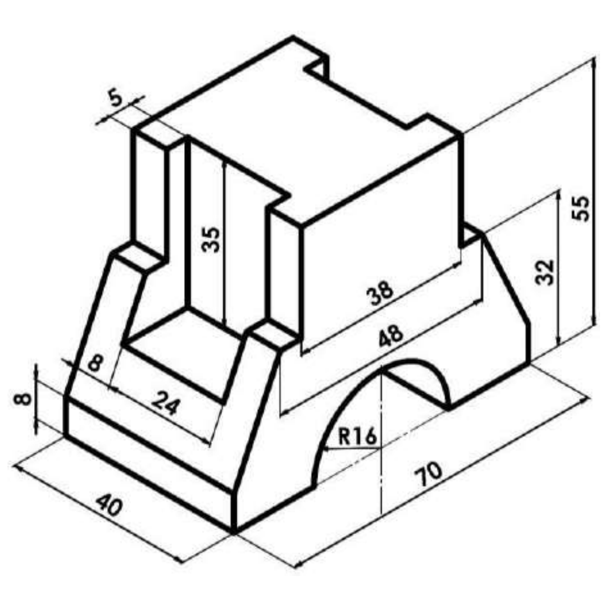
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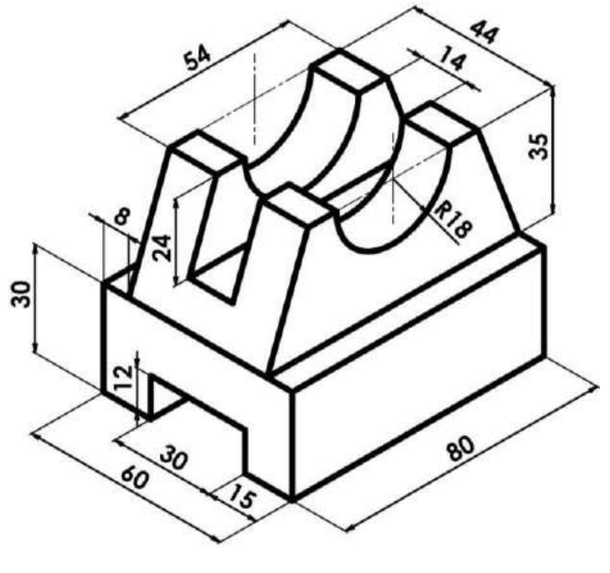
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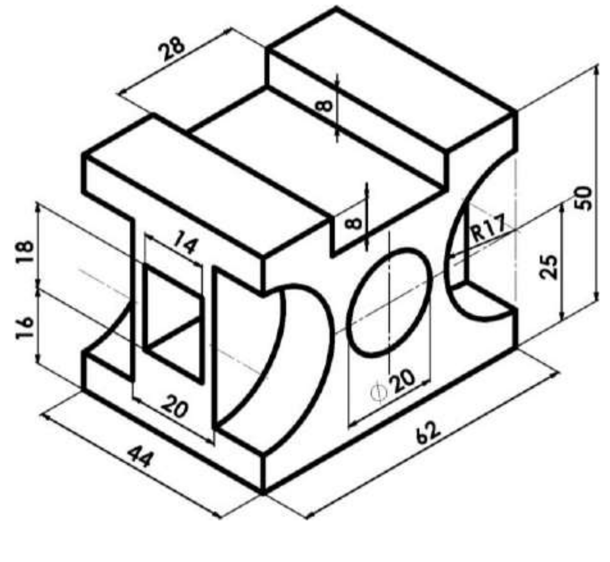
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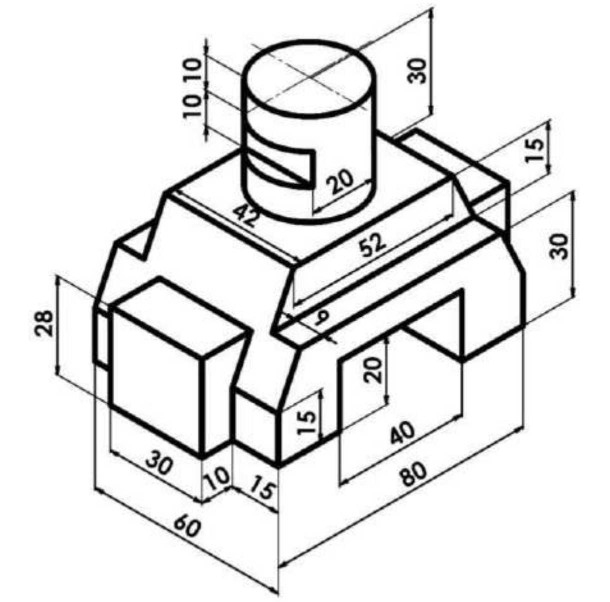
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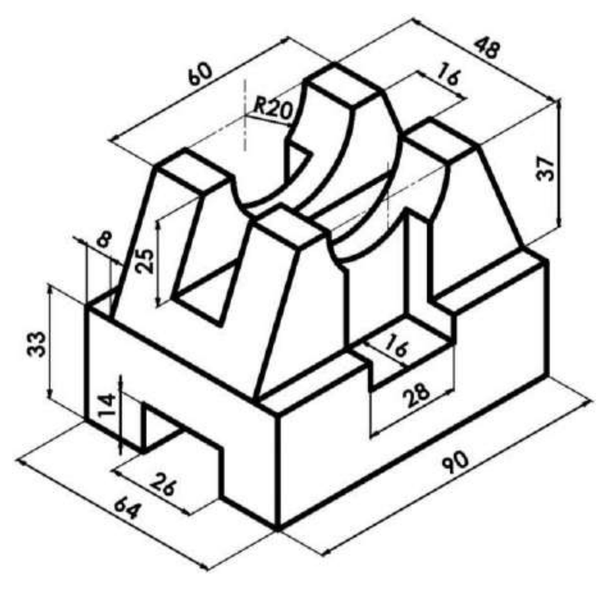
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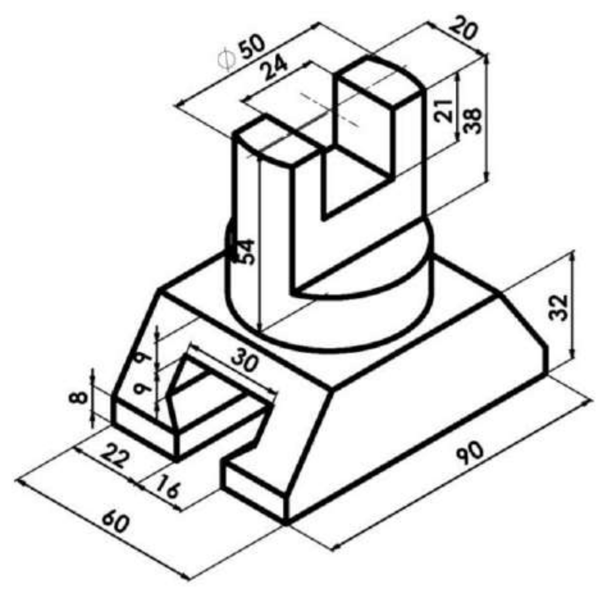
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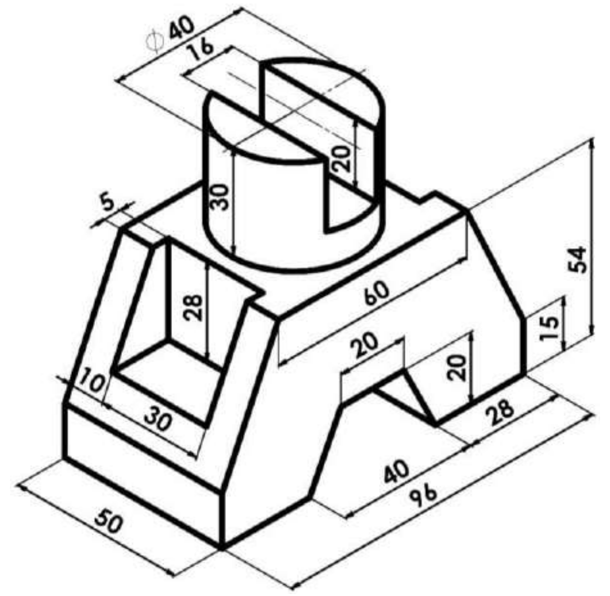
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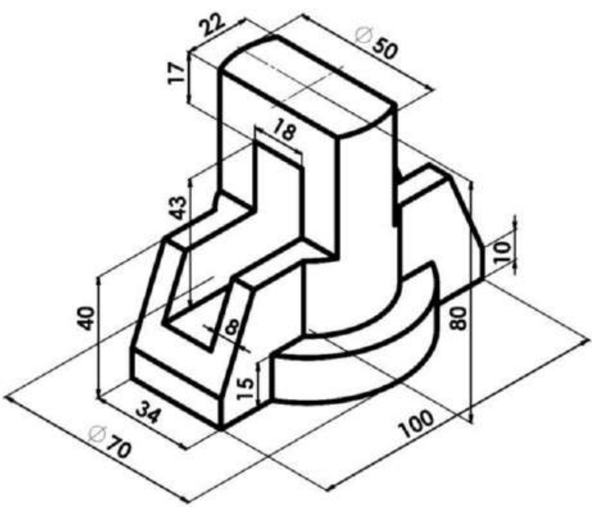
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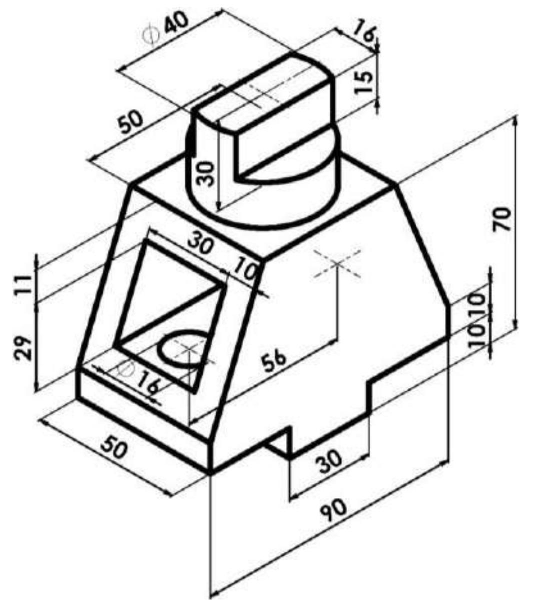
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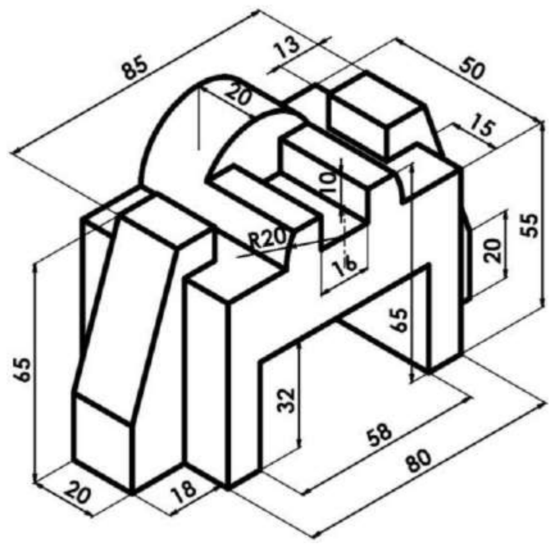
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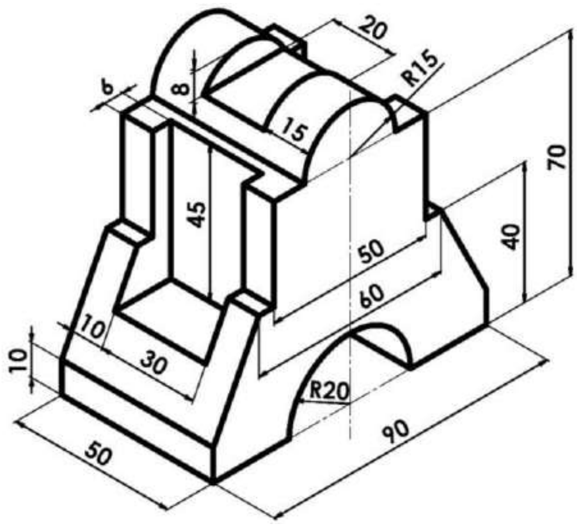
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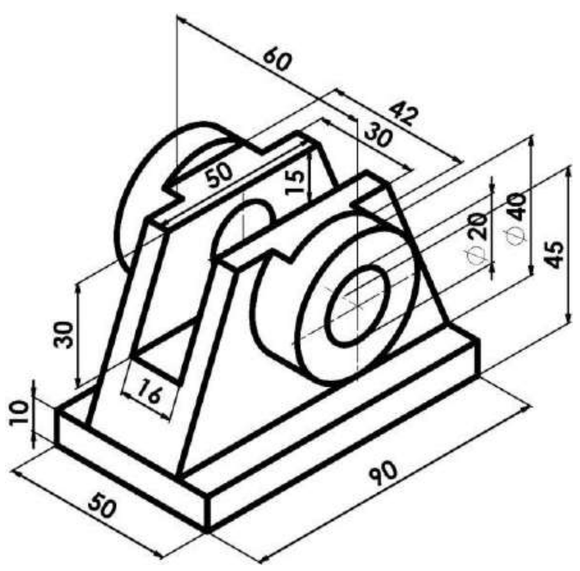
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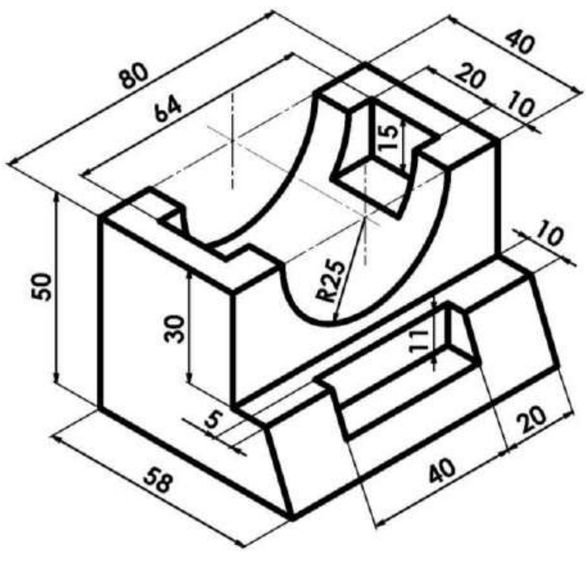
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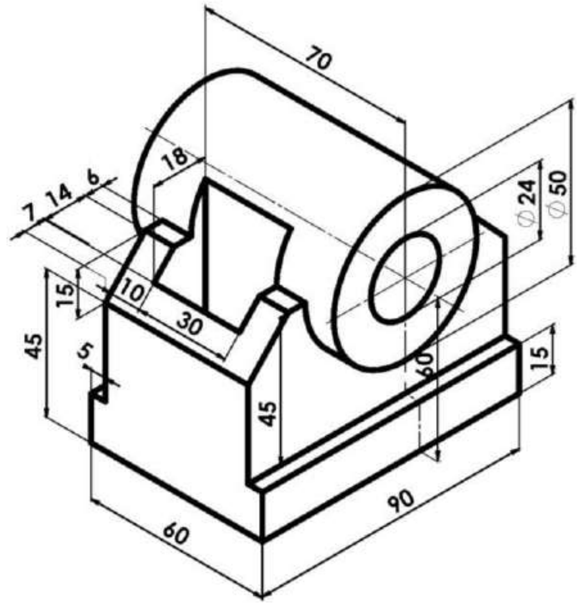
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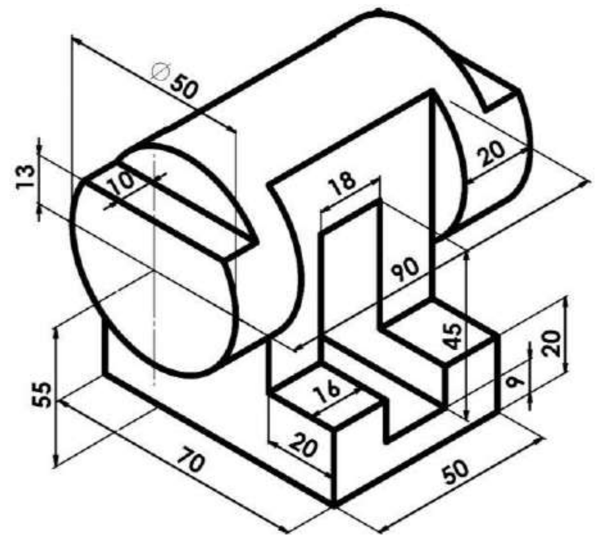
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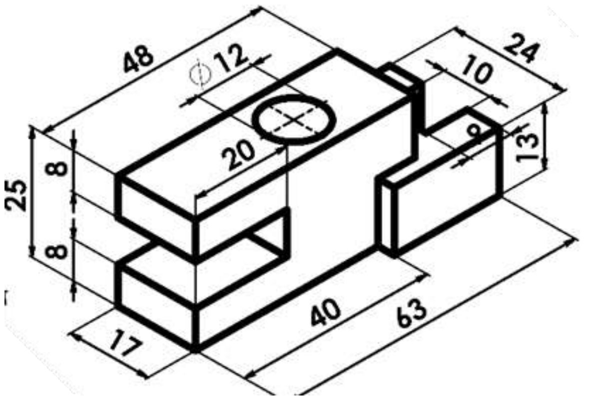
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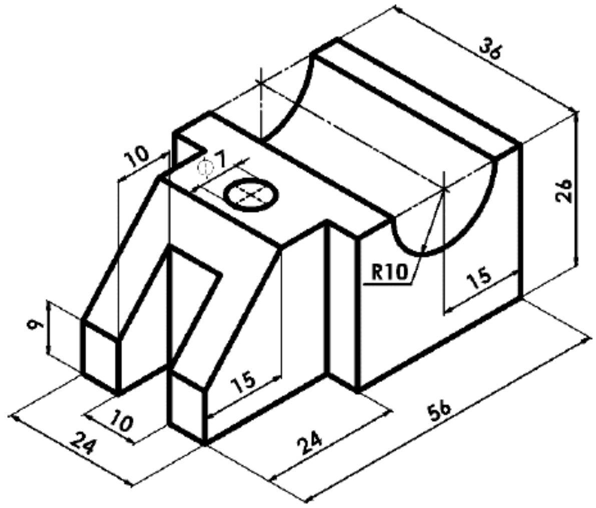
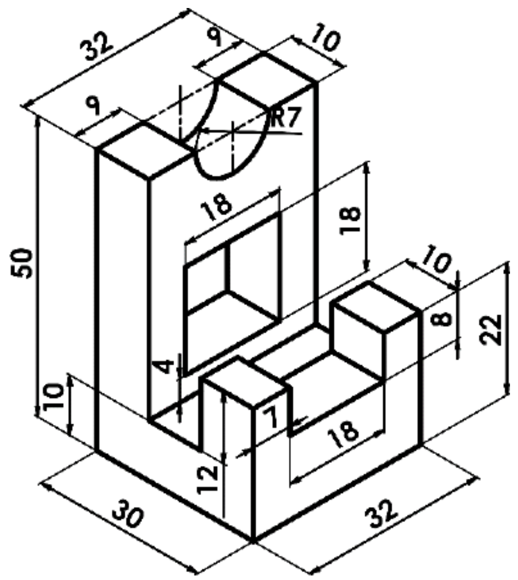

36
10
∅7
26
R10
15
9
15
10
24
24
56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10
9
R7
18
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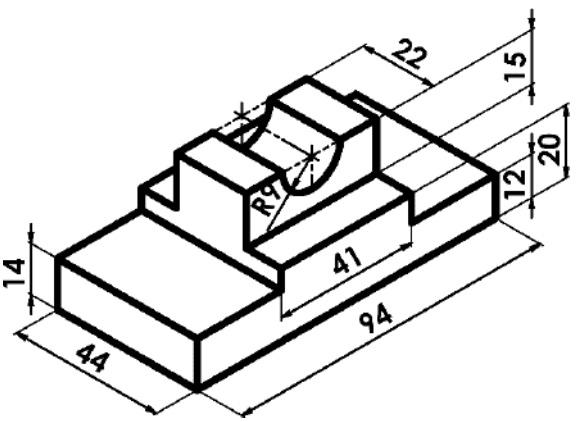
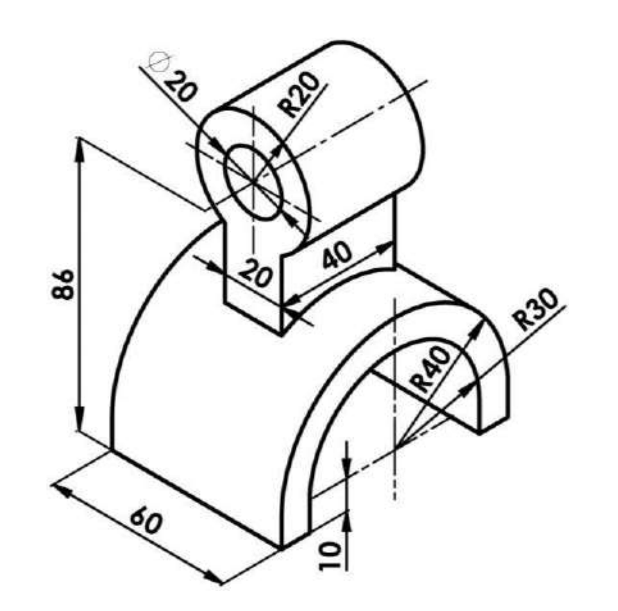
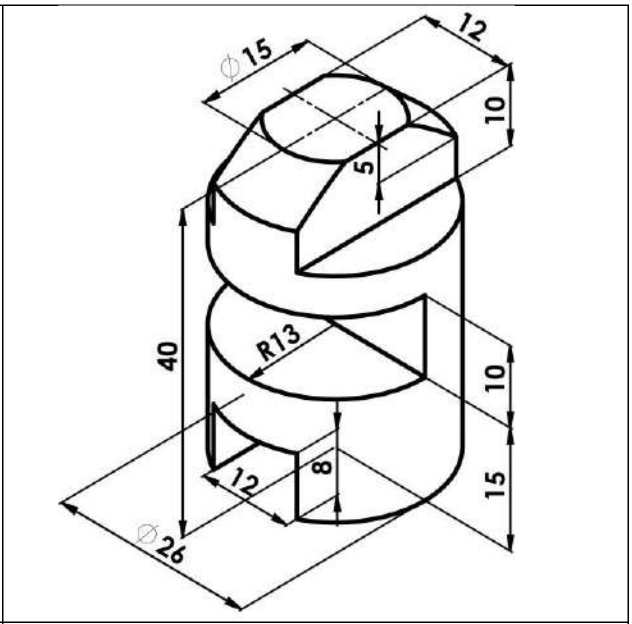
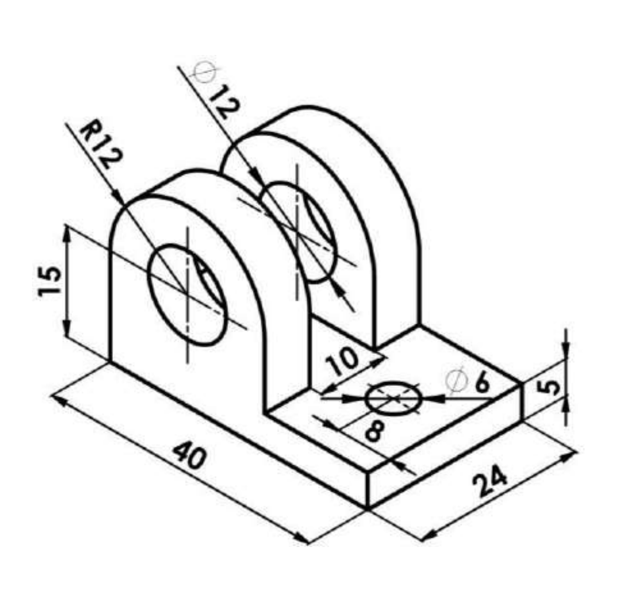
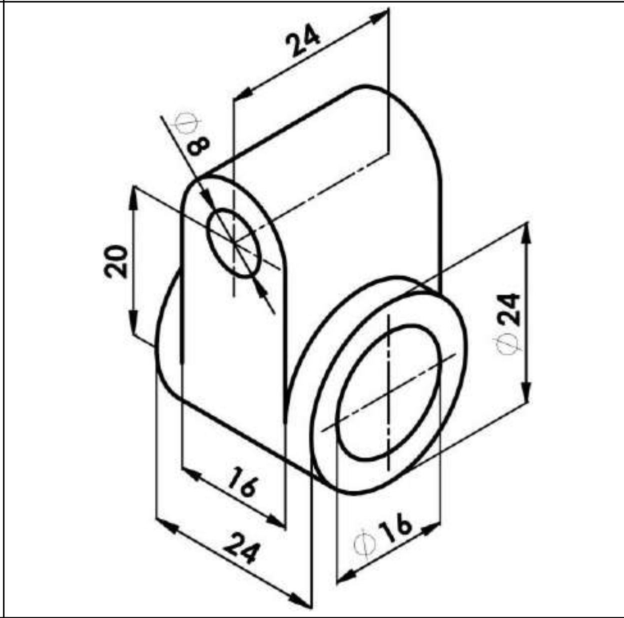
50
10
8
22
4
10
7
18
12
30
32


22
15
20
12
R9
14
41
94
44


∅20
R20
86
40
20
R30
R40
60
10


∅15
12
10
5
40
R13
10
8
12
15
∅26


R12
∅12
15
10
∅6
5
8
40
24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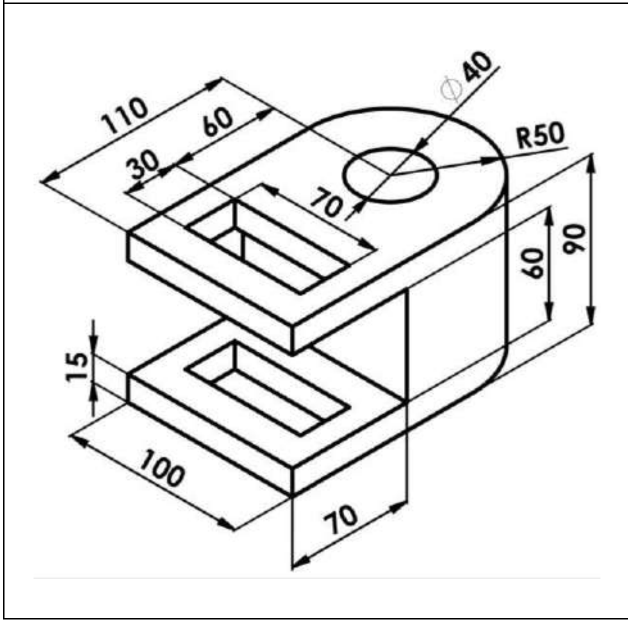
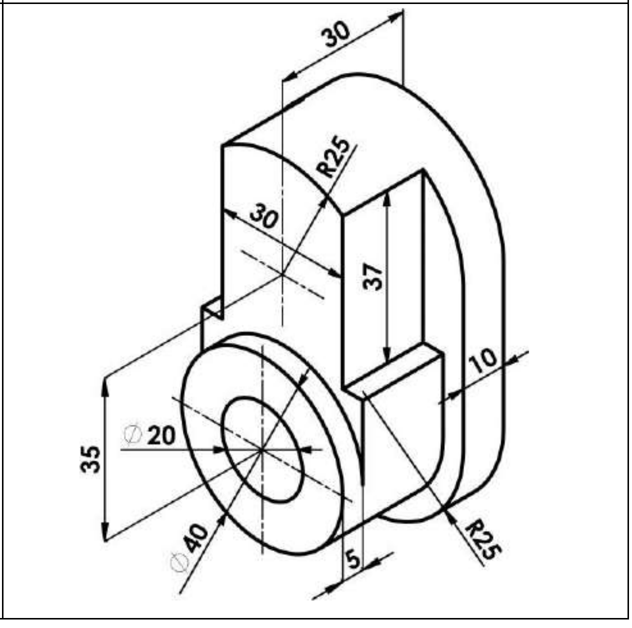
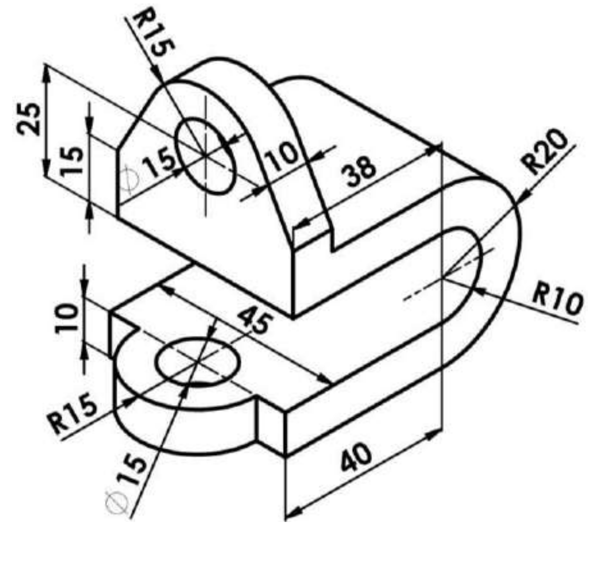
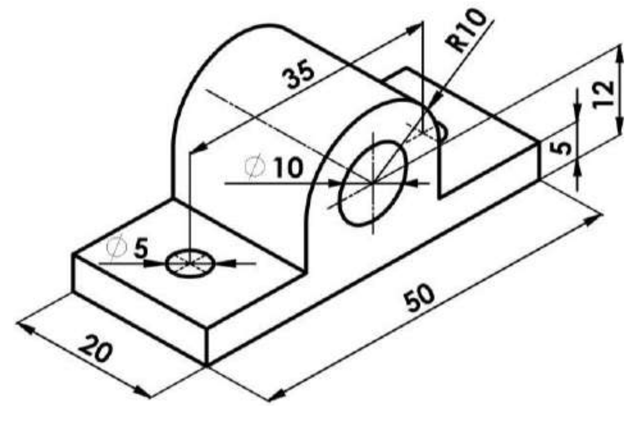
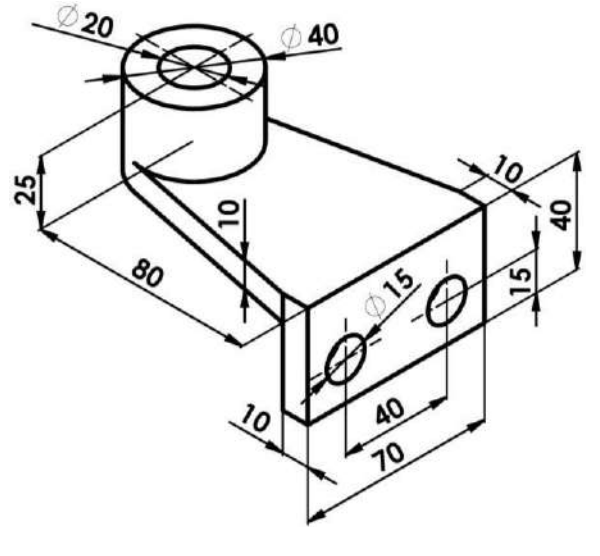
∅8
20
∅24
16
24
∅16

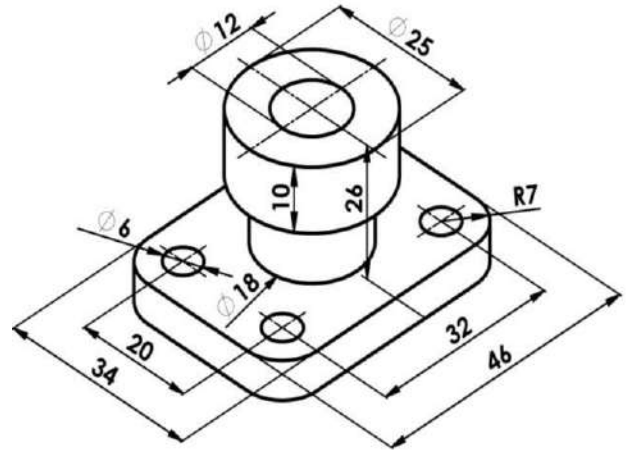

∅40
110
60
R50
30
70
90
60
15
100
70

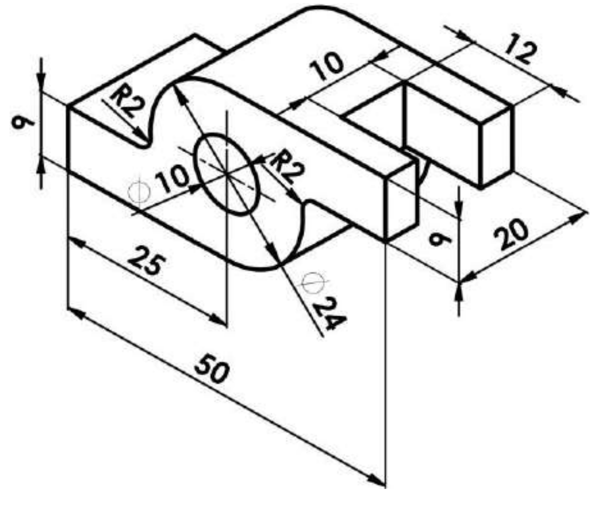

30
R25
30
37
10
∅20
35
∅40
5
R25

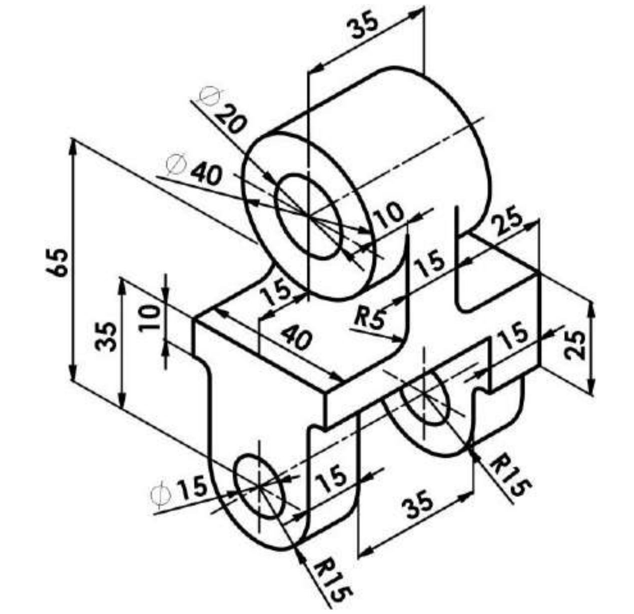

R15
25
15
⌀15
10
38
R20
R10
10
45
R15
⌀15
40

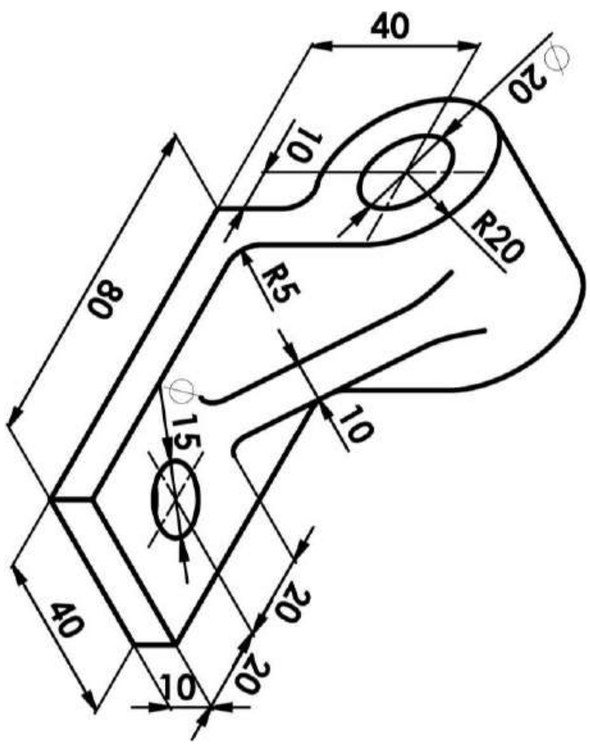

35
R10
12
5
⌀10
⌀5
50
20

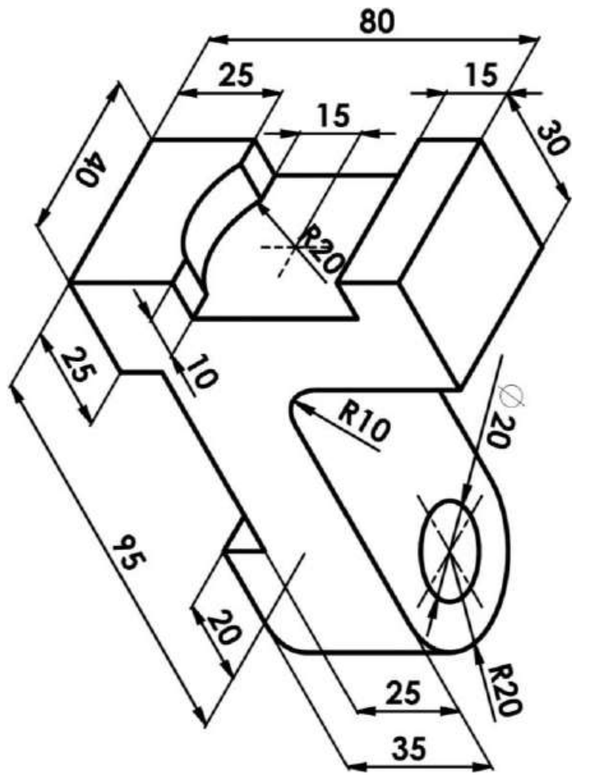

⌀20
⌀40
25
10
10
40
80
⌀15
15
10
40
70

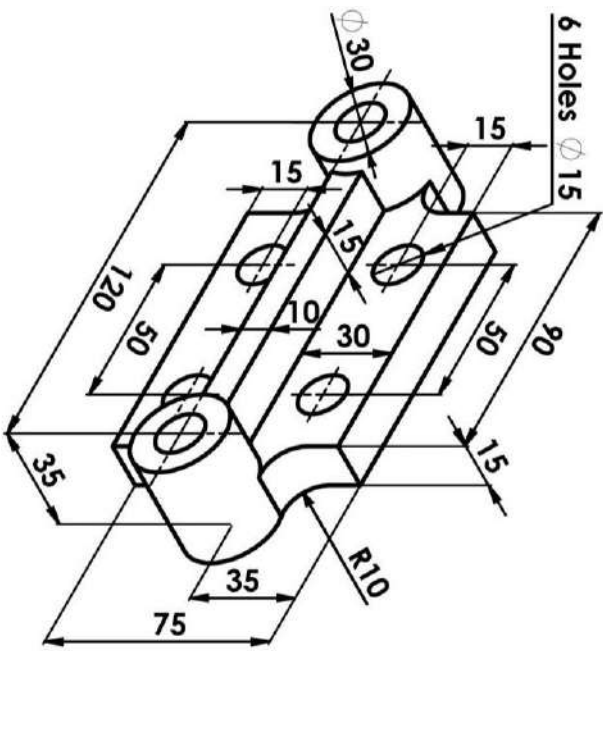

⌀12
⌀25
10
26
R7
⌀6
⌀18
20
34
32
46

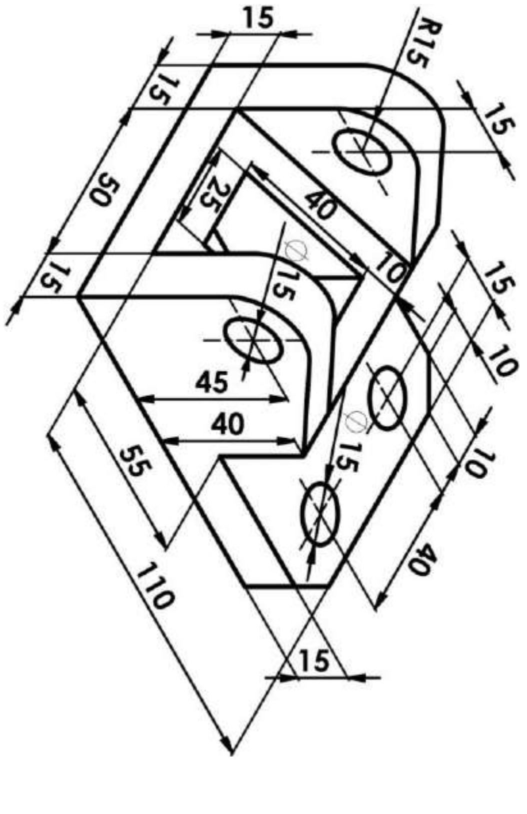

12
10
9
R2
⌀10
R2
9
20
25
⌀24
50

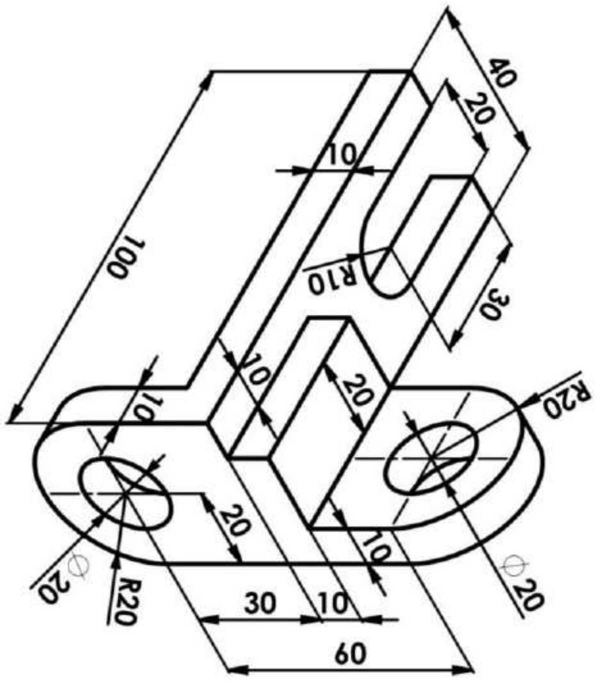

35
⌀20
⌀40
65
10
25
15
15
R5
40
10
35
15
25
⌀15
15
35
R15
R15


40
Ø 20
10
R20
80
R5
Ø 15
10
40
20
10
20

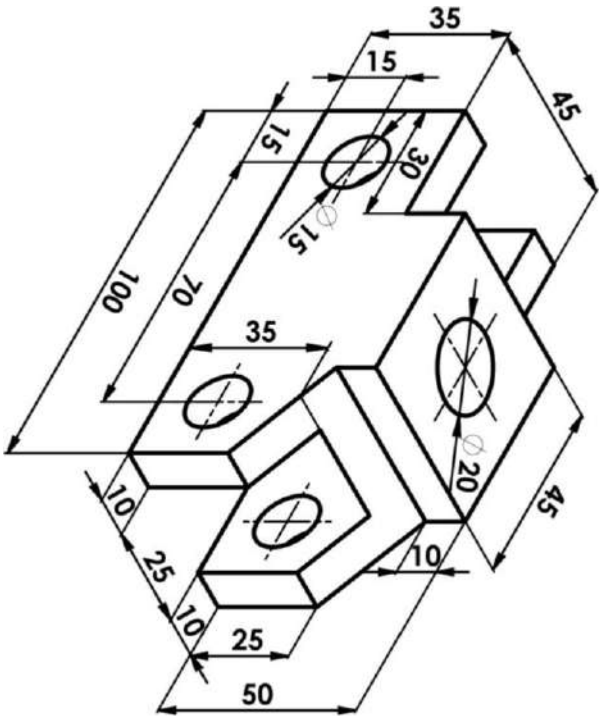

80
25
15
15
30
40
R20
25
10
R10
Ø 20
95
20
25
R20
35


Ø 30
6 Holes Ø 15
15
15
120
15
10
30
50
50
90
15
35
R10
35
75

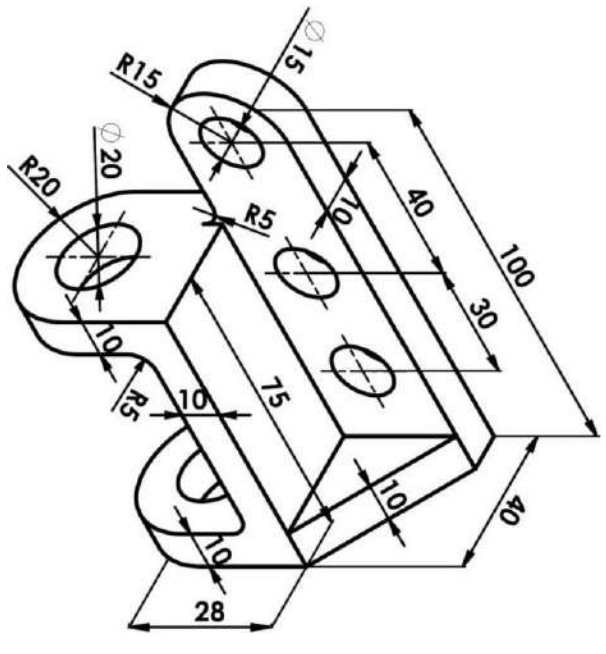

15
R15
15
15
50
25
40
15
10
Ø 15
15
15
45
40
10
Ø 15
55
10
40
110
15



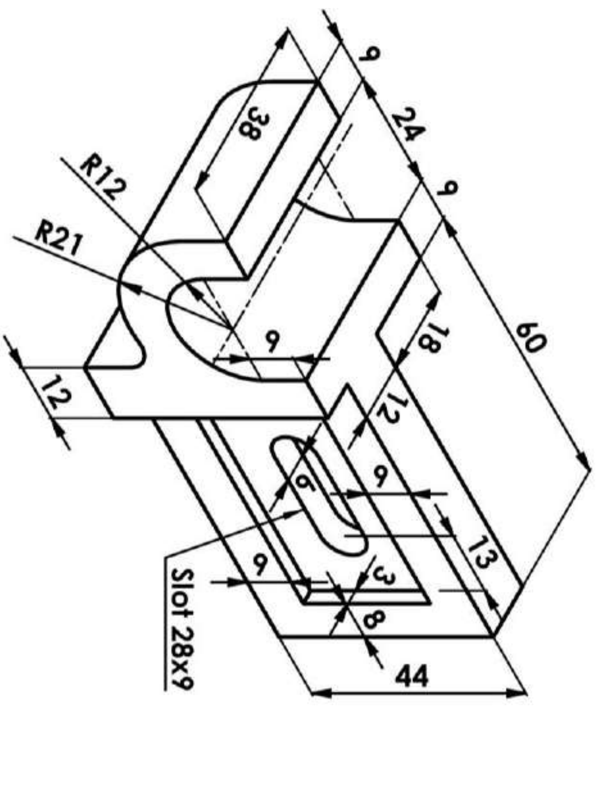
40
20
10
100
R10
30
10
20
R20
10
20
Ø20
R20
10
Ø20
30
10
60



35
15
45
15
30
Ø15
100
70
35
Ø20
45
10
25
10
10
25
50



R15
Ø15
R20
Ø20
R5
10
40
100
30
10
75
R5
10
10
10
40
28



9
38
24
R12
9
R21
60
18
9
12
12
9
9
13
9
3
8
Slot 28x9
44

Chapter 4

Orthographic Projection

Orthographic projection is the solution to the biggest problem to convert three-dimensional object on a two dimensional plane. The drawing must show quite clearly the detailed outlines of all the faces and these outlines must be fully dimensioned. This may be achieved with a freehand sketch. Circles and curves are difficult to draw in either system and neither shows more than three sides of an object in any one view. Orthographic projection, because of its flexibility in allowing any number of views of the same object, has none of these drawbacks.

Orthographic projection has two forms: first angle and third angle. British industry has used first angle while the United States of America and, more recently, the continental countries used the third angle system.

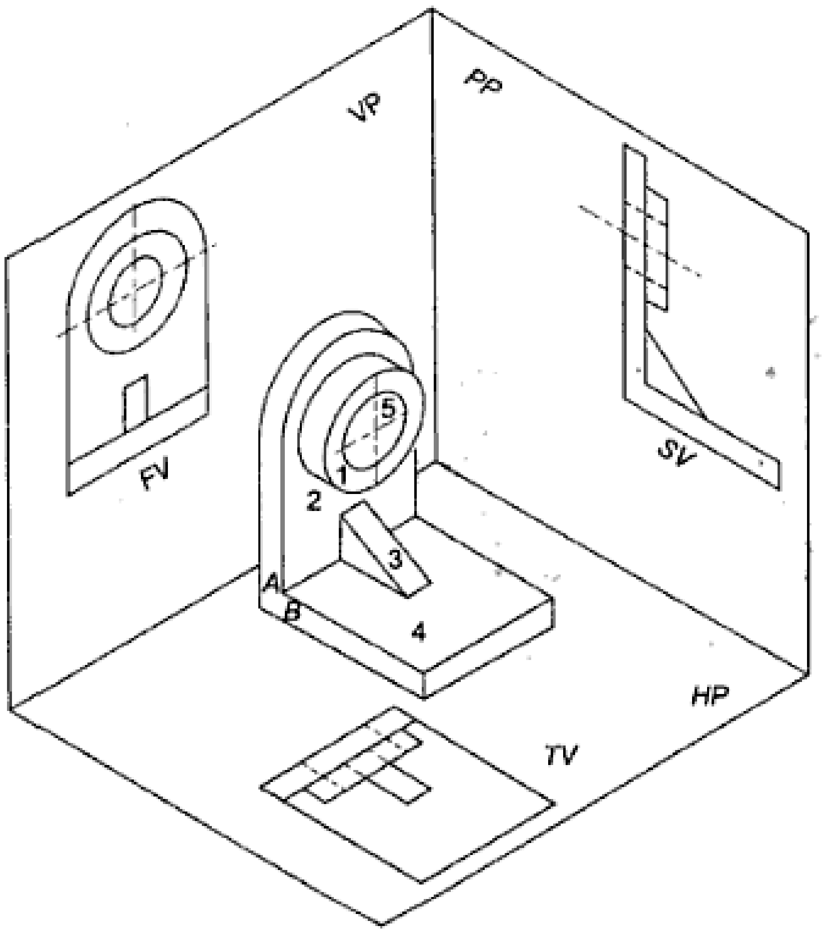


Fig. 4.1 Multi views orthographic projections

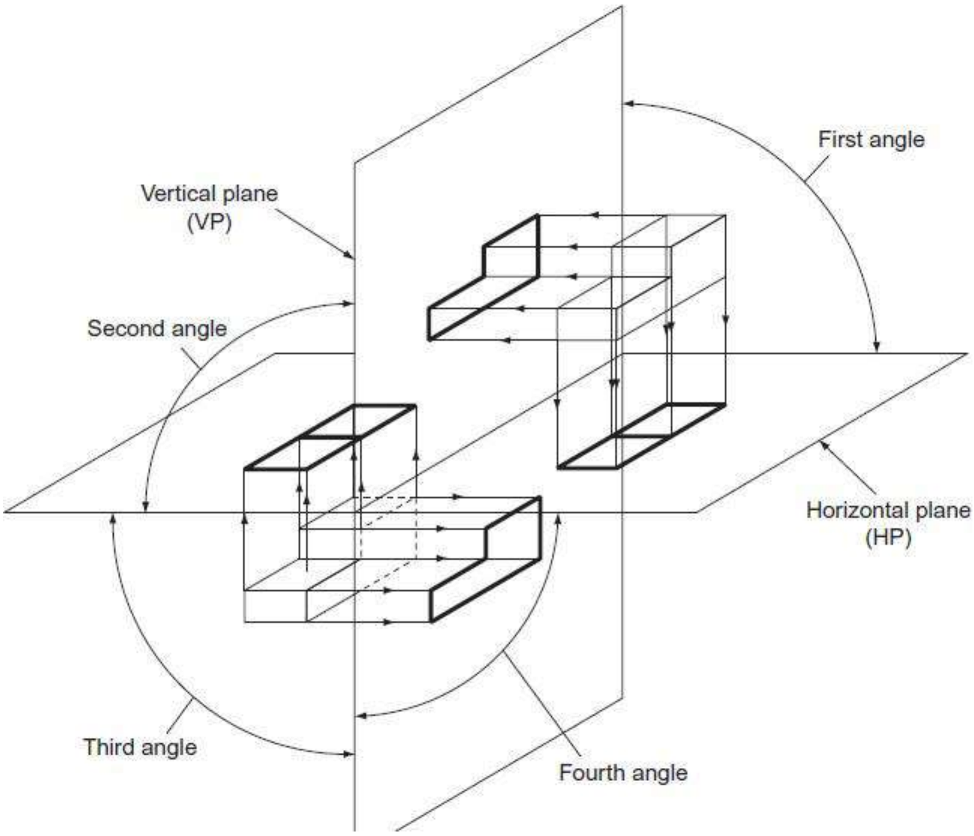


Fig. 4.2 Relative positions of first and third angle projections.

4.1 First angle projection

In order to describe the orthographic views we need to select a principal view and in this case we have chosen the view in direction of arrow A to be the view from the front, fig. 4.3.

In first angle projection the views in the directions of arrows B, C, D and E are arranged with reference to the front view as follows:

1- The view from B is placed on the right.
2- The view from C is placed on the left.
3- The view from D is placed underneath.
4- The view from E is placed above.

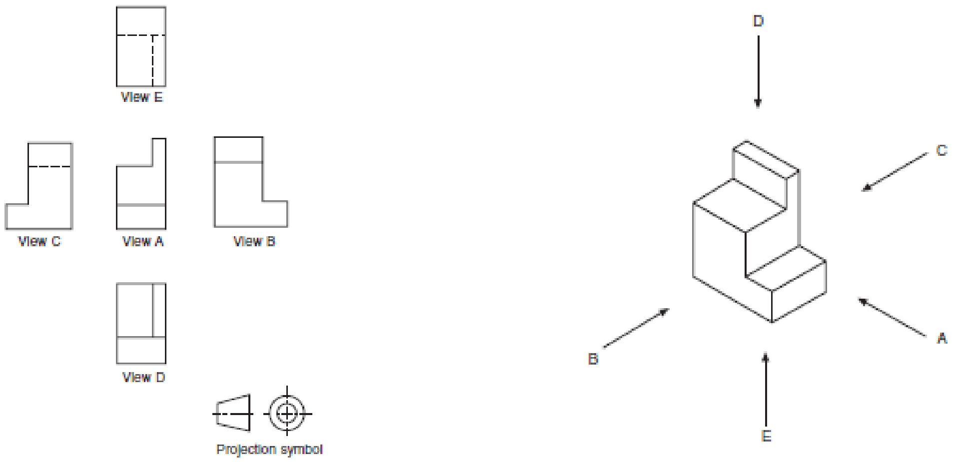


Fig. 4.3 First angle projection arrangement.

For the isometric in figure 4.1, the orthographic projection according to first angle shows the following views to descript the geometry.

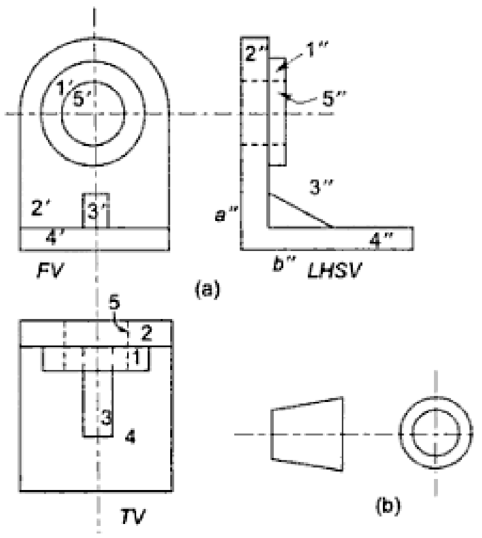


Fig. 4.4 Views arrangement for first angle projection.

4.2 Third angle projection

The difference between first and third angle projection is in the arrangement of views and, with reference to the illustration in Fig. 4.5, views are now positioned as follows:

1- View B from the left is placed on the left.
2- View C from the right is placed on the right.
3- View D from above is placed above
4- View E from below is placed underneath.

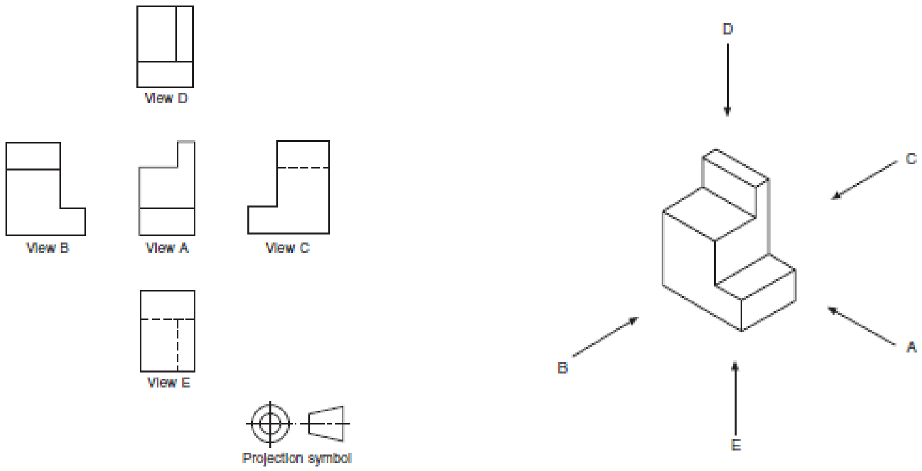


Fig. 4.5 Third angle projection arrangement

For the isometric in figure 4.1, the orthographic projection according to third angle shows the following views to descript the geometry, Fig. 4.6.

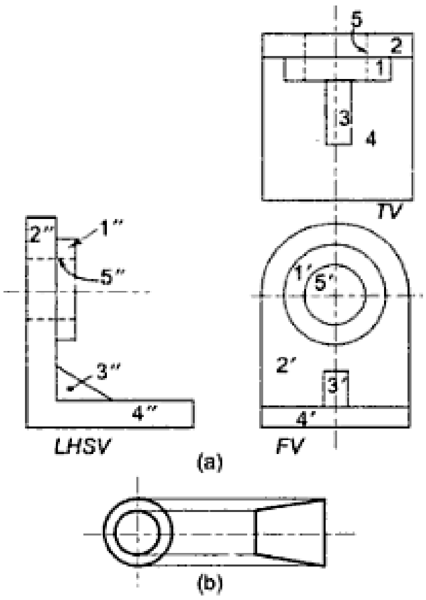


Fig. 4.6 Views arrangement for third angle projection.

From the previous figures we can conclude that, for the first angle projection:

1- The plan is drawn below the front view.

2- The right hand side view is drawn on the left of the front view.

3- The left hand side view is drawn on the right of the front view.

But for the third angle projection

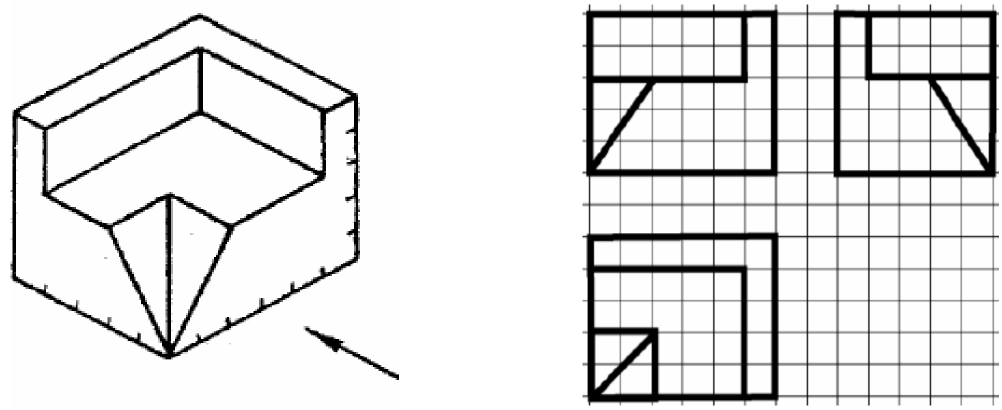
l - The plan is drawn above the front view.

2- The right hand side view is drawn on the right of the front view.

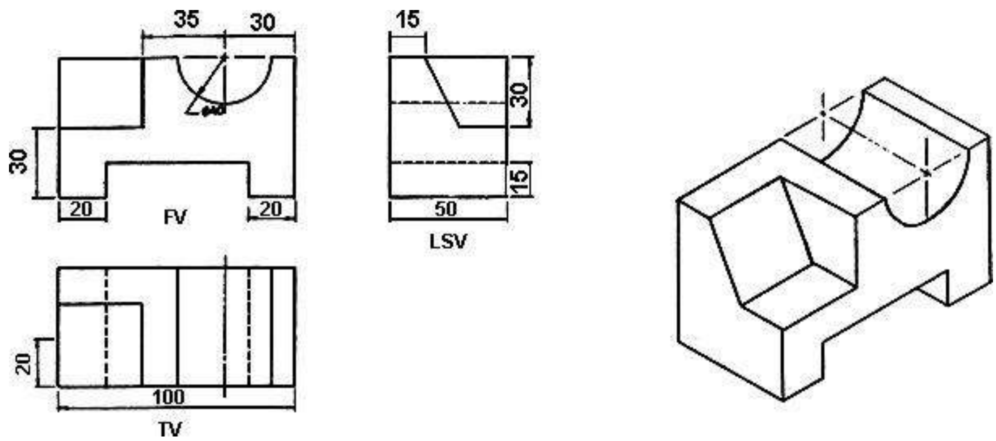
3- The left hand side view is drawn on the left of the front view.

4.3 Projections Examples

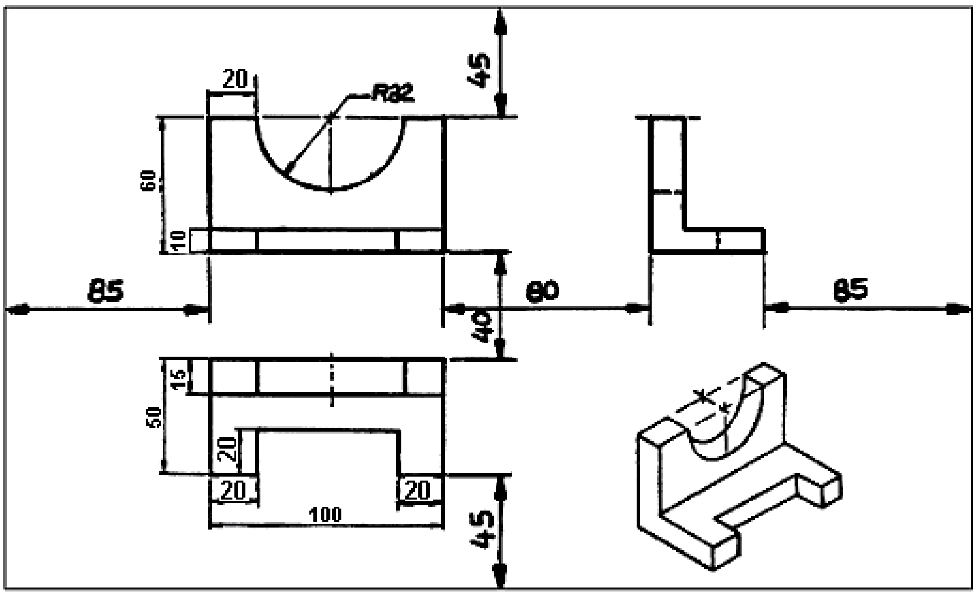
1.



2.

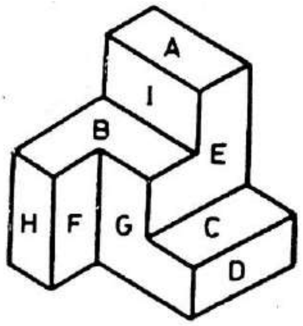


3.

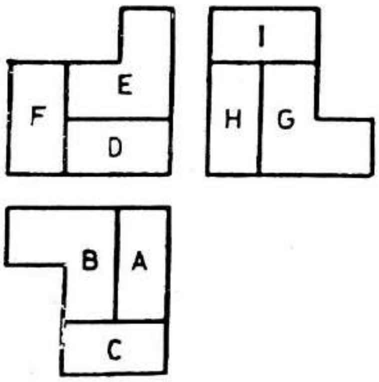


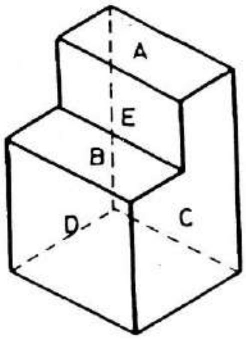
Note: Dotted lines indicate hidden edges and corners

4.

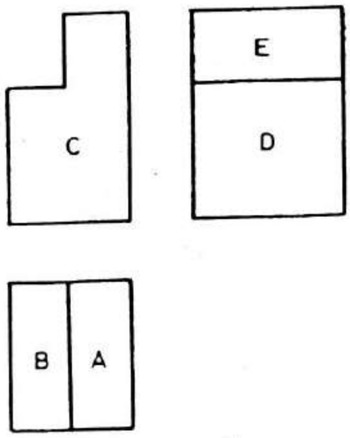

A
I
B
E
H
F
G
C
D

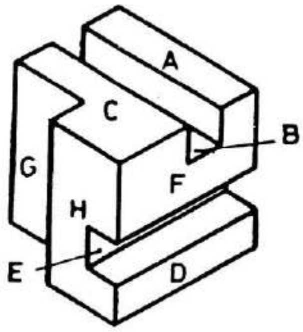
(1)


F
E
D
I
H
G
B
A
C

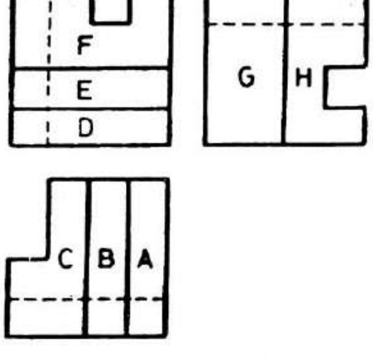

A
E
B
D
C

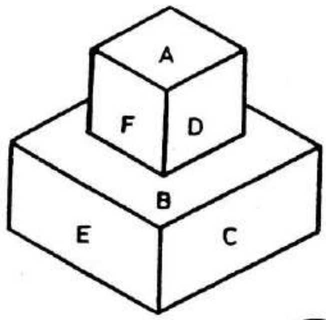
(2)


C
E
D
B
A

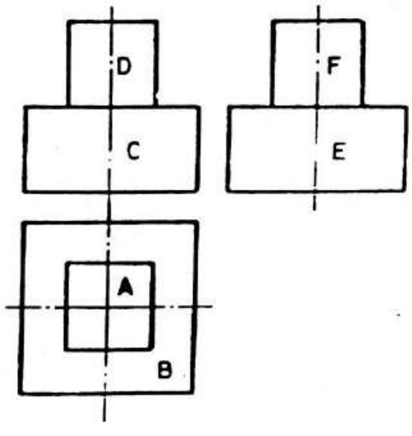

A
C
B
G
F
H
E
D

(2)

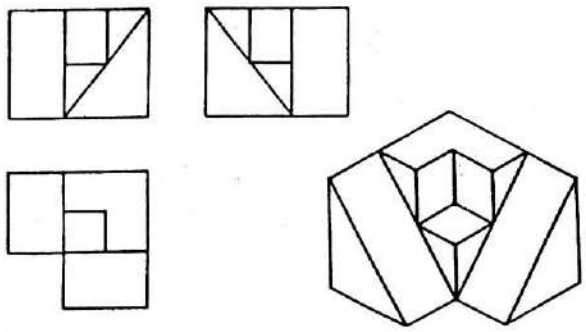

F
E
D
G
H
C
B
A

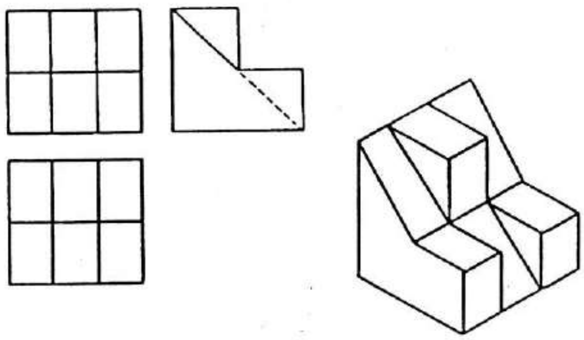

A
F
D
B
E
C

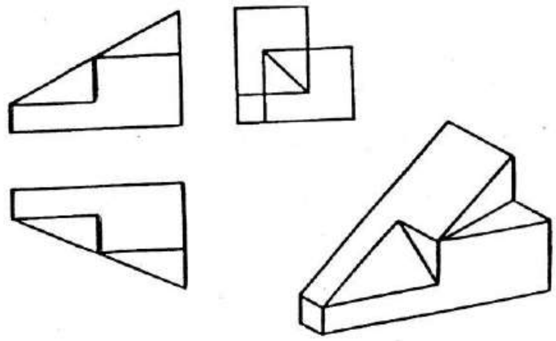
(3)

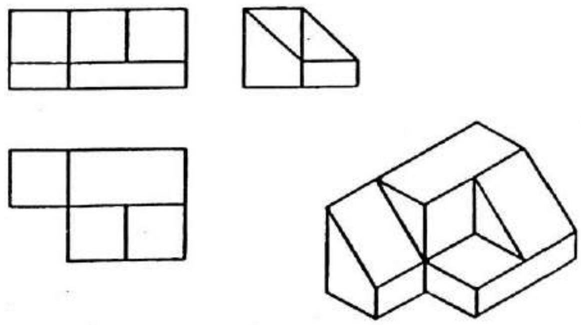

D
C
F
E
A
B

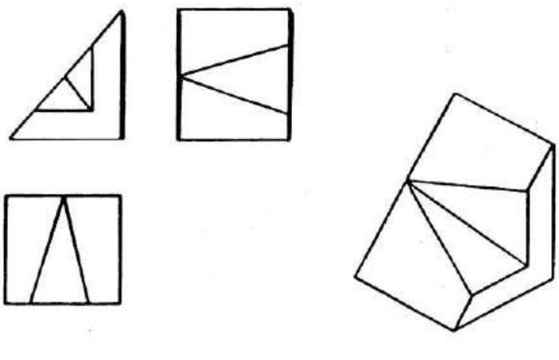
5.

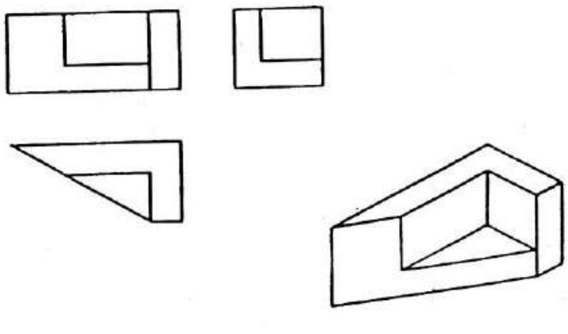


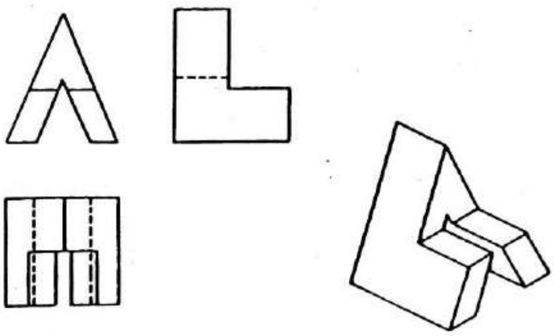


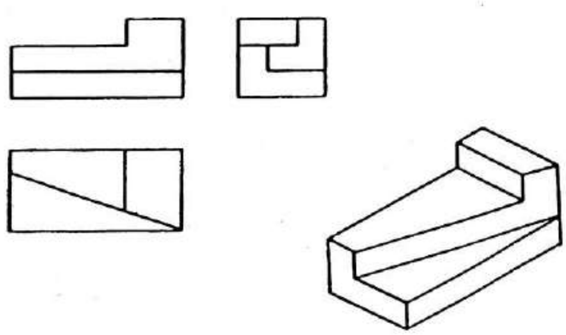






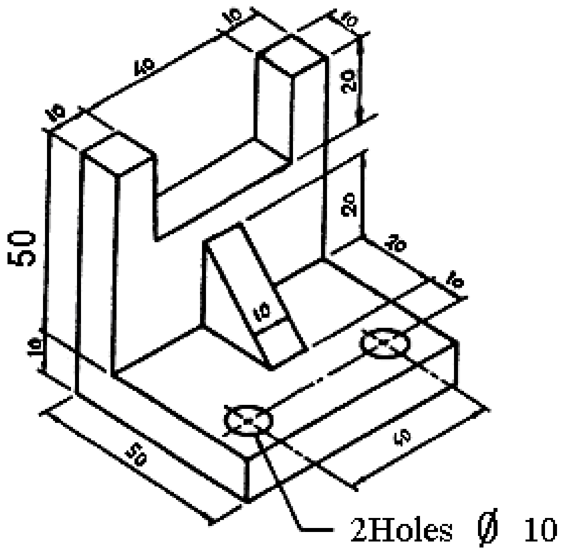


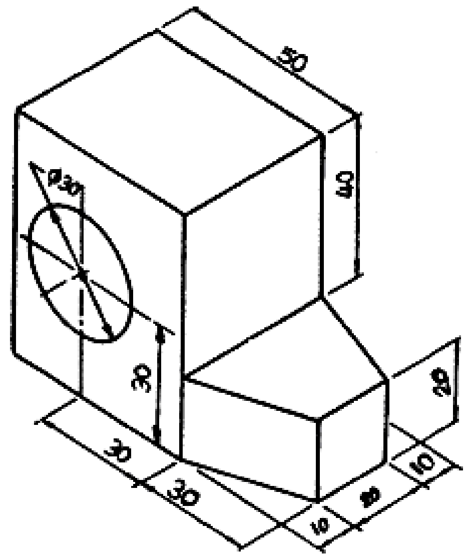


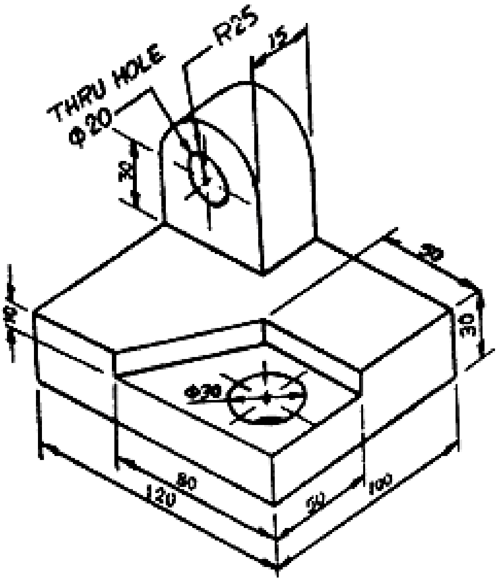


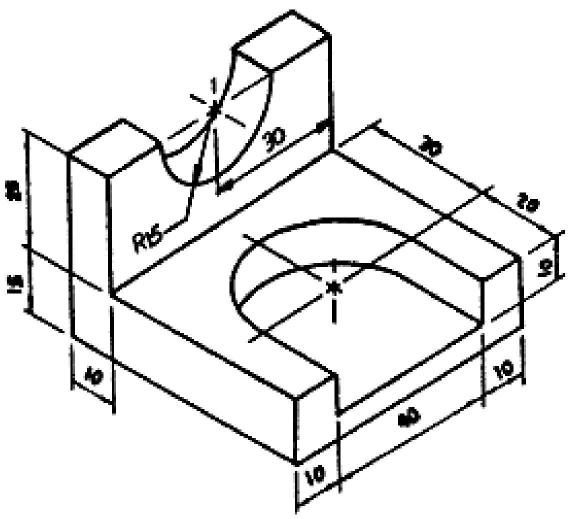
Exercise

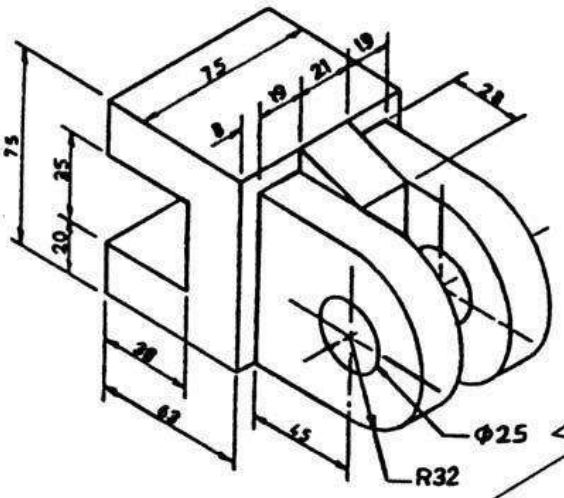
Draw the three views for the following isometrics:

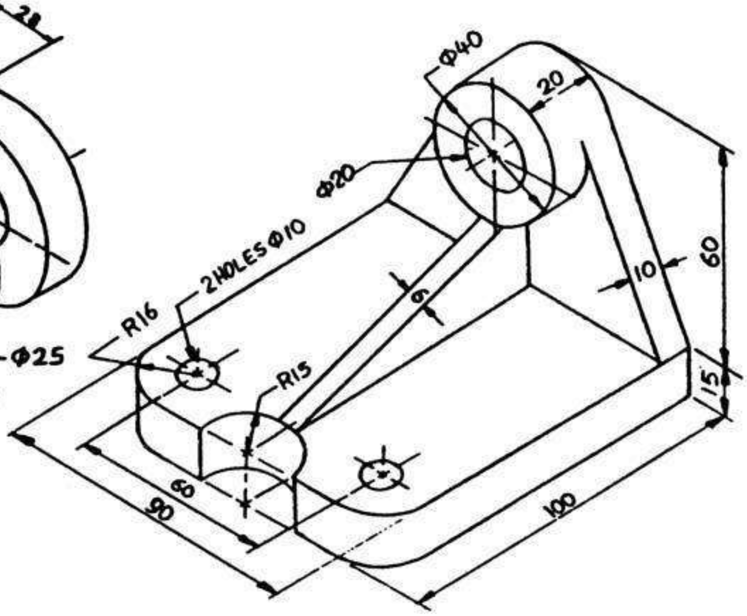


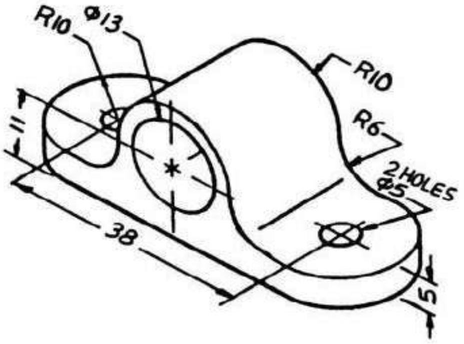


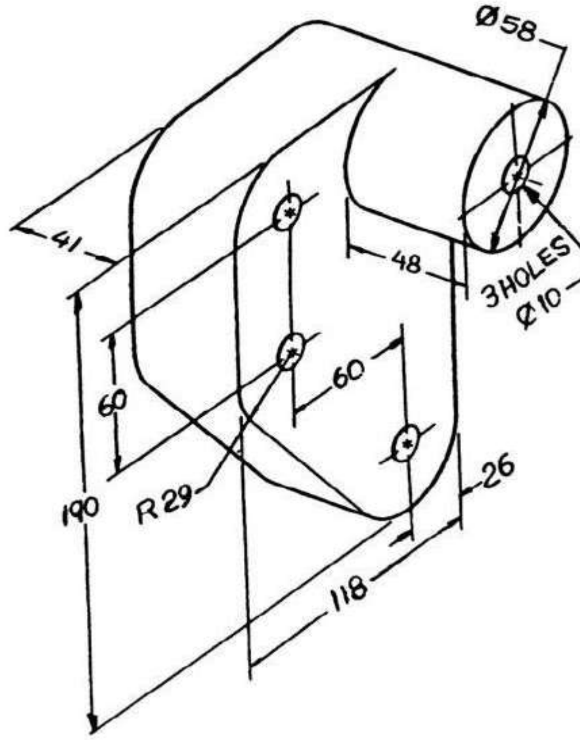


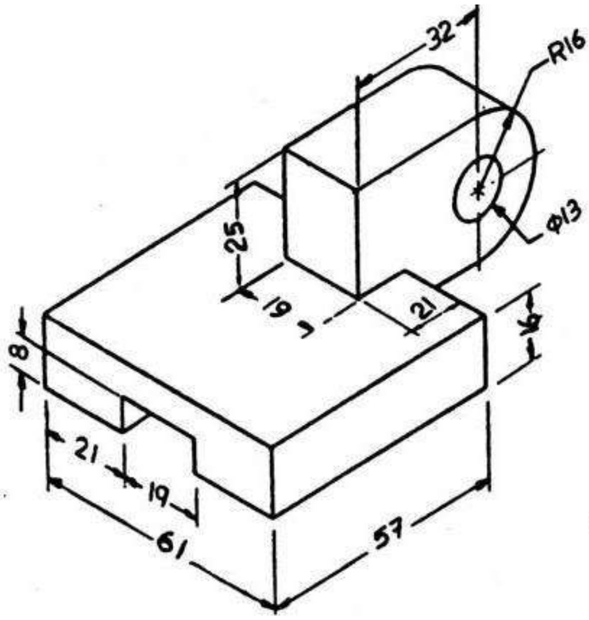


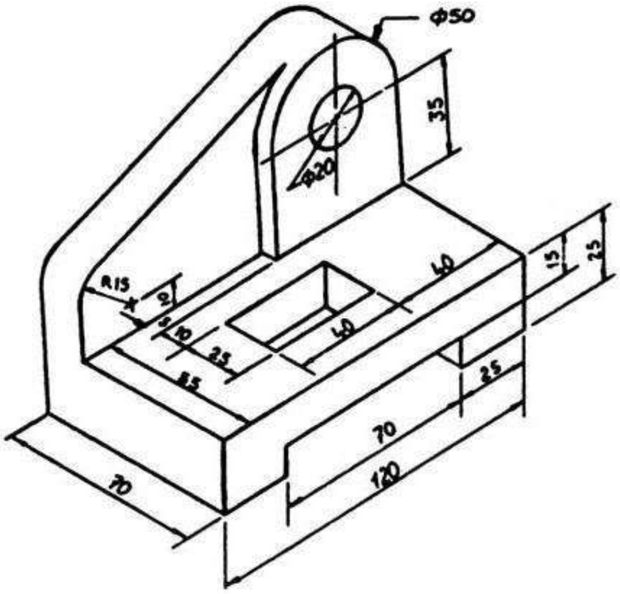

75
8
19
21
19
28
73
25
20
38
62
45
Φ25
R32

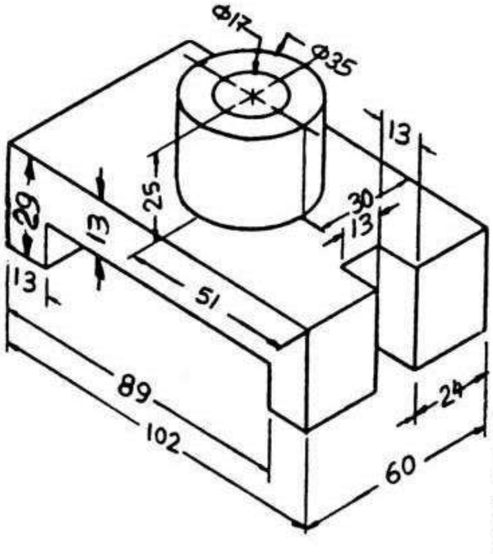

Φ40
20
Φ20
2 HOLES Φ10
R16
R15
6
10
60
15
60
90
100


R10
Φ13
R10
R6
11
2 HOLES Φ5
38
5


Ø58
41
48
3 HOLES Ø10
60
60
190
R29
26
118


32
R16
Φ13
25
19
21
16
8
21
19
61
57


Φ50
35
Φ20
R15
15
25
40
40
25
85
25
70
120
70


Φ17
Φ35
13
25
30
13
29
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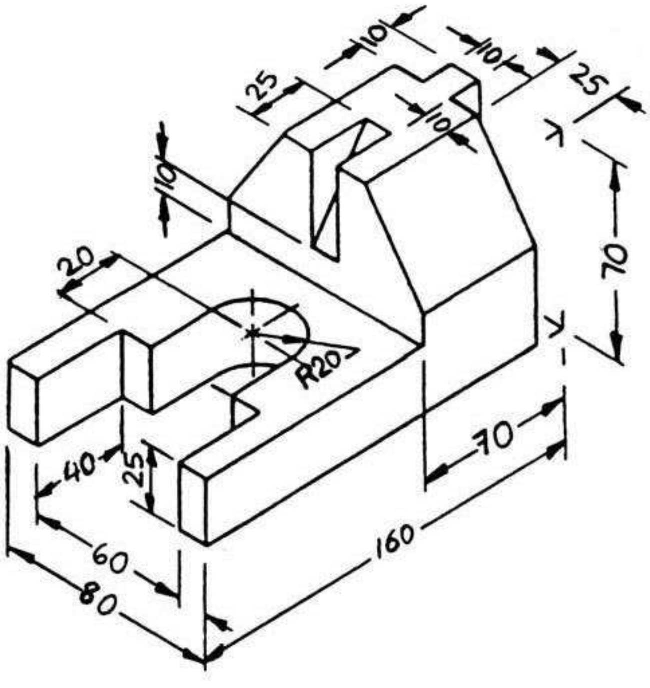
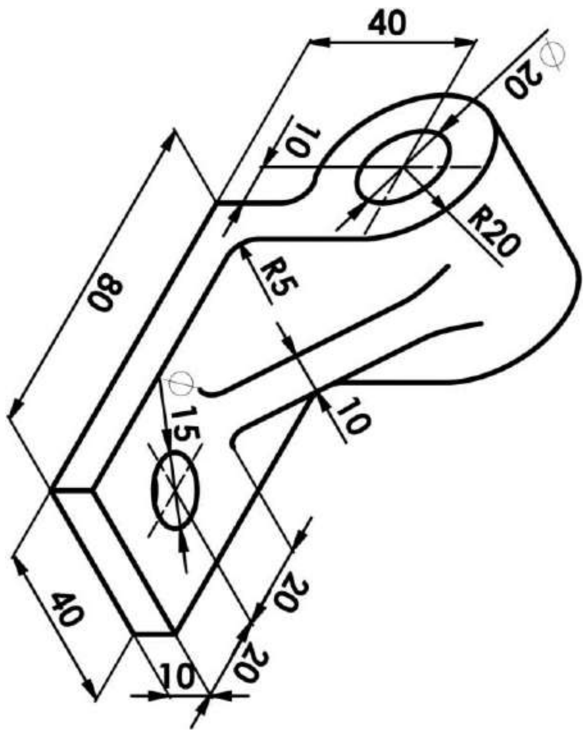
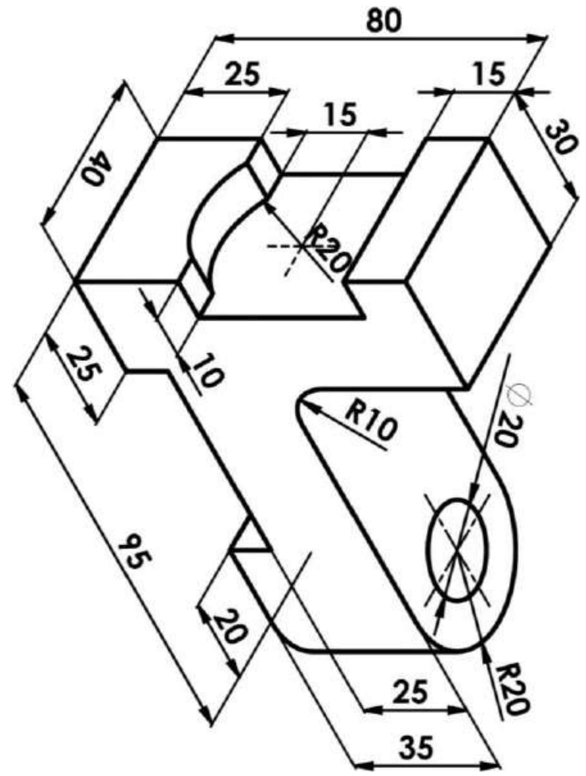
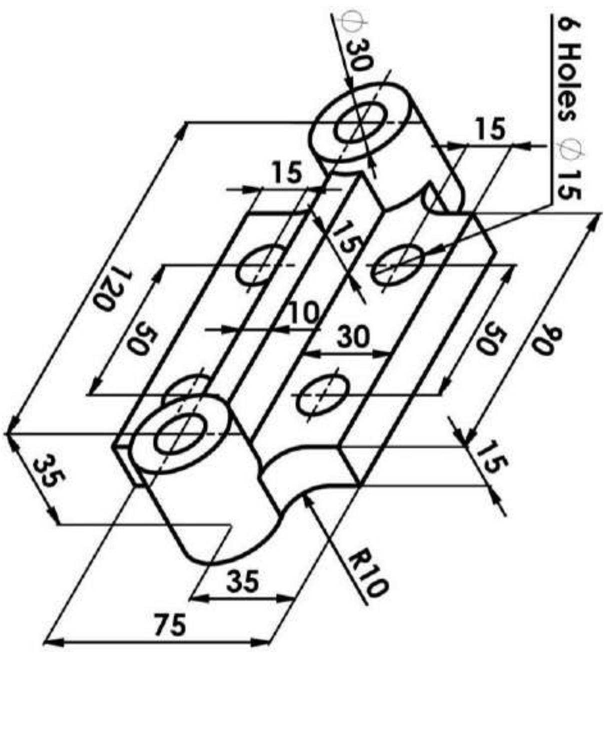
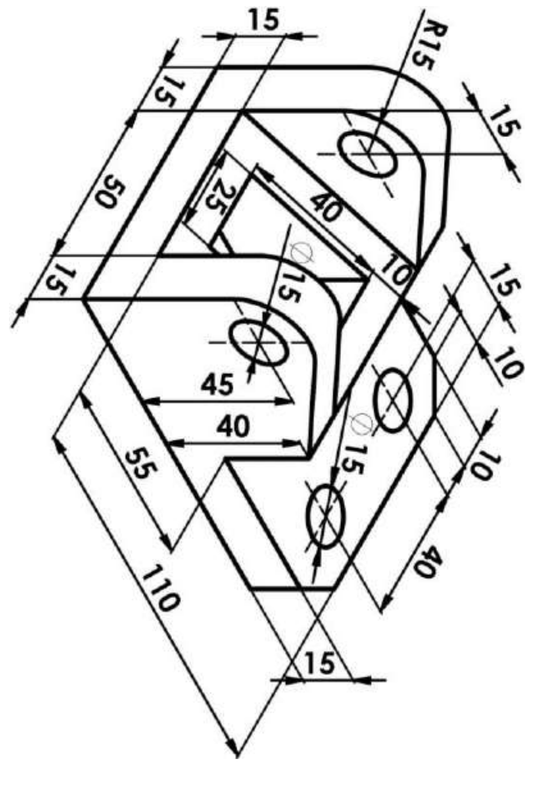
13
51
89
102
24
60

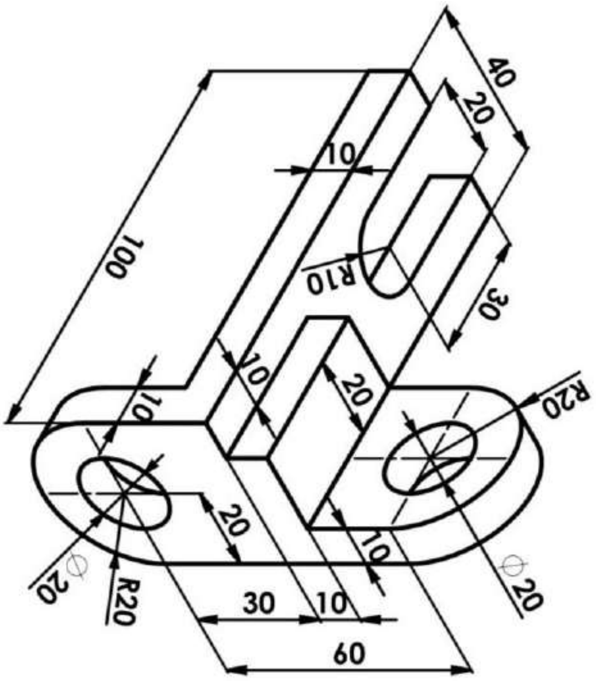

10
10
25
25
10
10
70
20
R20
40
25
70
60
80
160

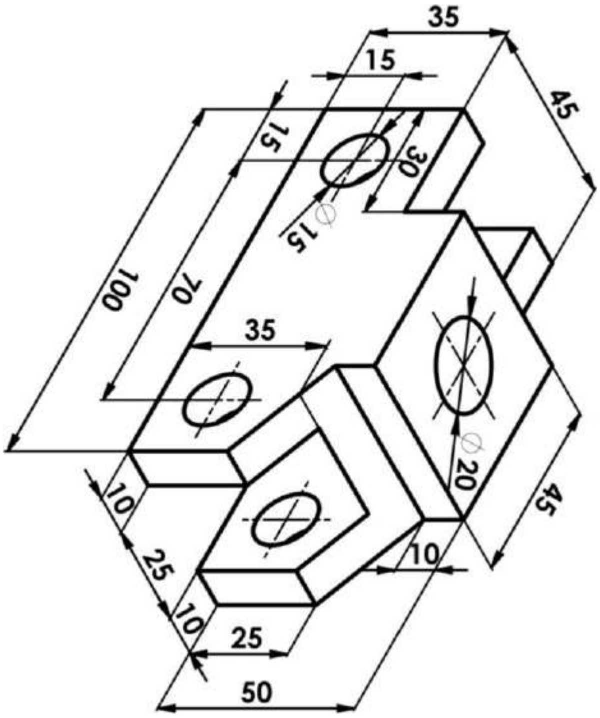

40
⌀20
10
R20
R5
80
⌀15
10
20
20
40
10

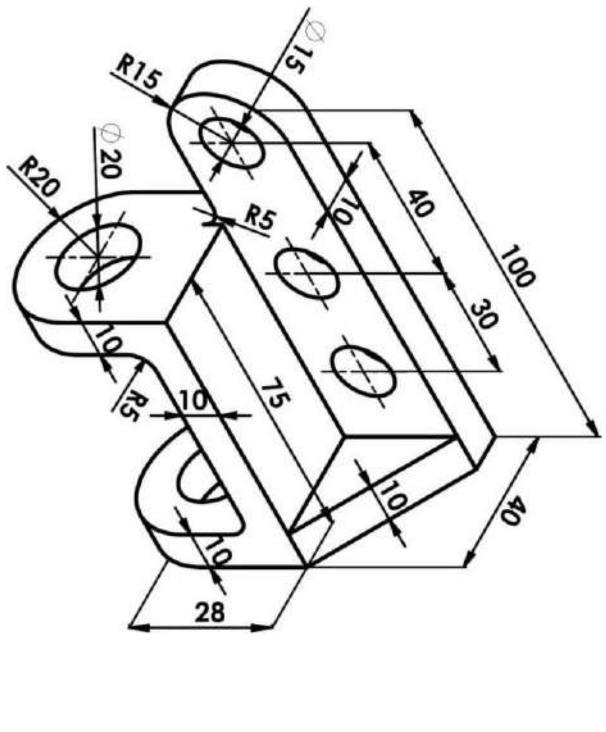

80
25
15
15
30
40
R20
25
10
R10
⌀20
95
20
25
35
R20

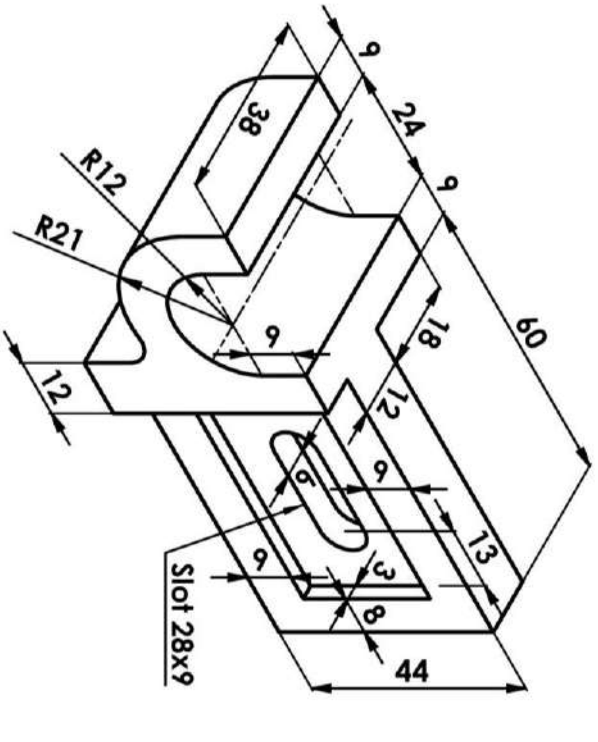

⌀30
6 Holes ⌀15
15
15
15
120
50
10
30
50
90
15
35
R10
35
75

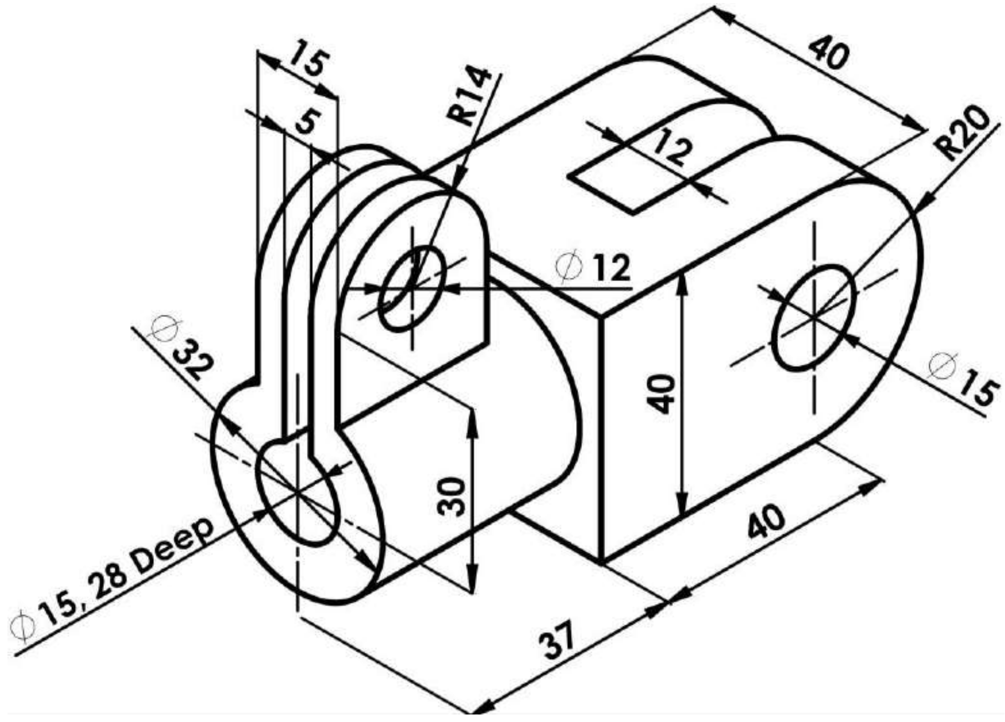

15
R15
15
15
50
25
40
⌀15
10
15
15
10
45
40
⌀15
10
55
40
110
15


100
40
20
10
R10
30
10
20
R20
10
20
⌀20
R20
30
10
10
⌀20
60

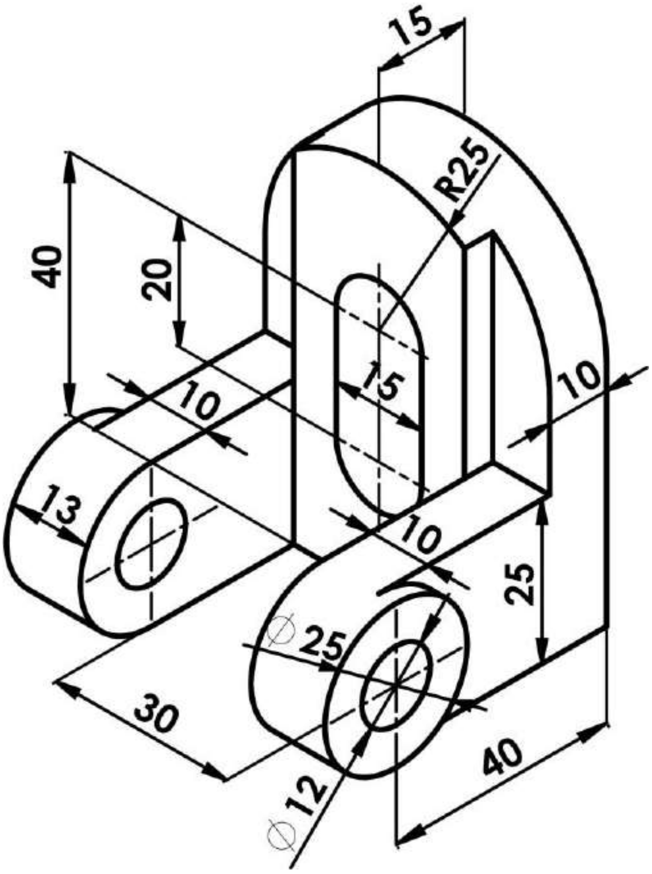

35
15
45
15
30
⌀15
100
70
35
⌀20
45
10
25
10
10
25
50


⌀15
R15
⌀20
R20
R5
40
100
10
30
75
10
R5
10
10
40
10
28

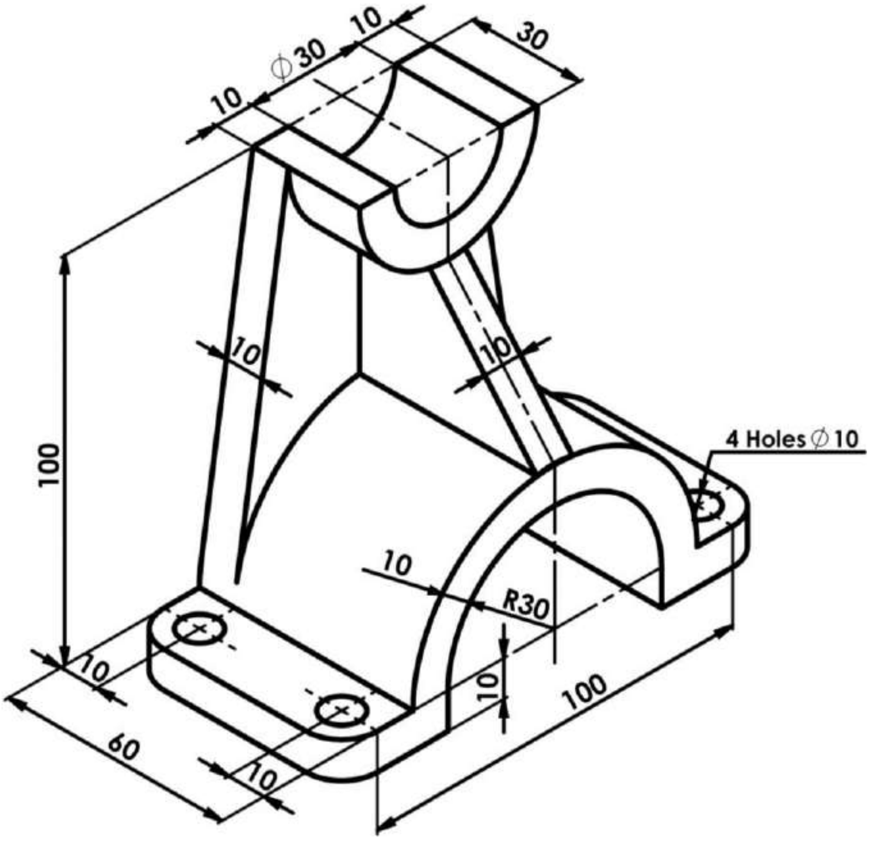

9
24
38
R12
9
R21
9
18
60
12
12
9
9
13
9
3
8
44
Slot 28x9



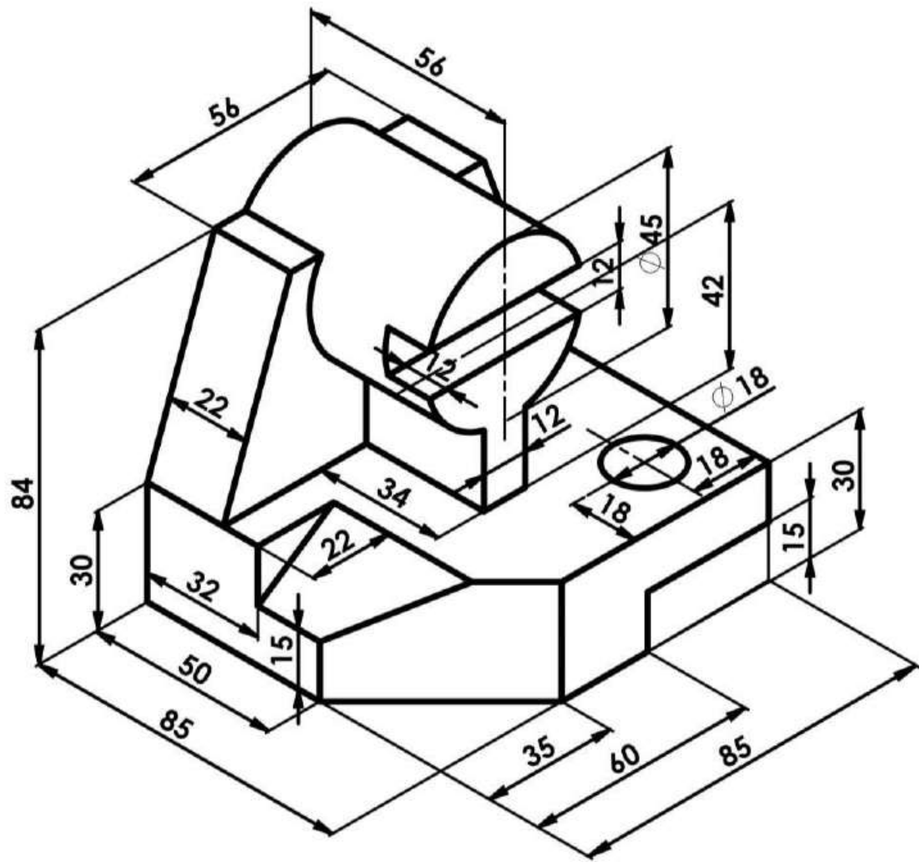
15
5
R14
40
12
R20
Ø12
Ø32
40
Ø15
30
Ø15, 28 Deep
40
37



15
R25
40
20
10
15
10
13
10
25
Ø25
30
40
Ø12



10
⌀30
10
30
100
10
10
4 Holes ⌀10
10
R30
10
10
100
60
10



56
56
⌀45
12
42
12
22
⌀18
12
18
84
34
18
30
22
15
30
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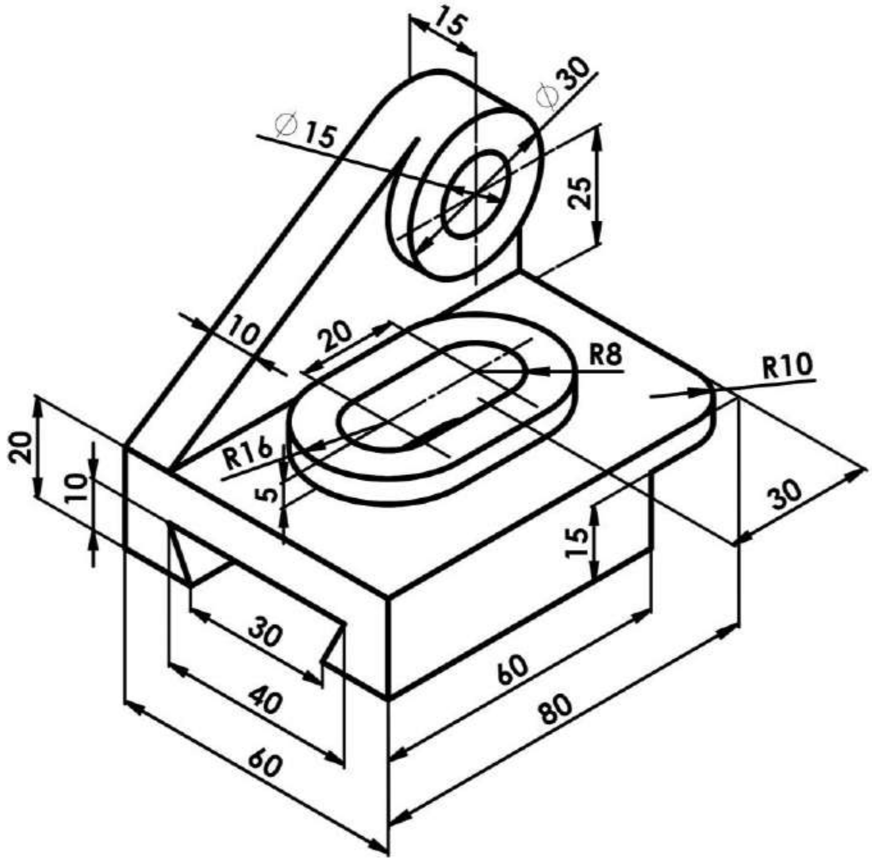
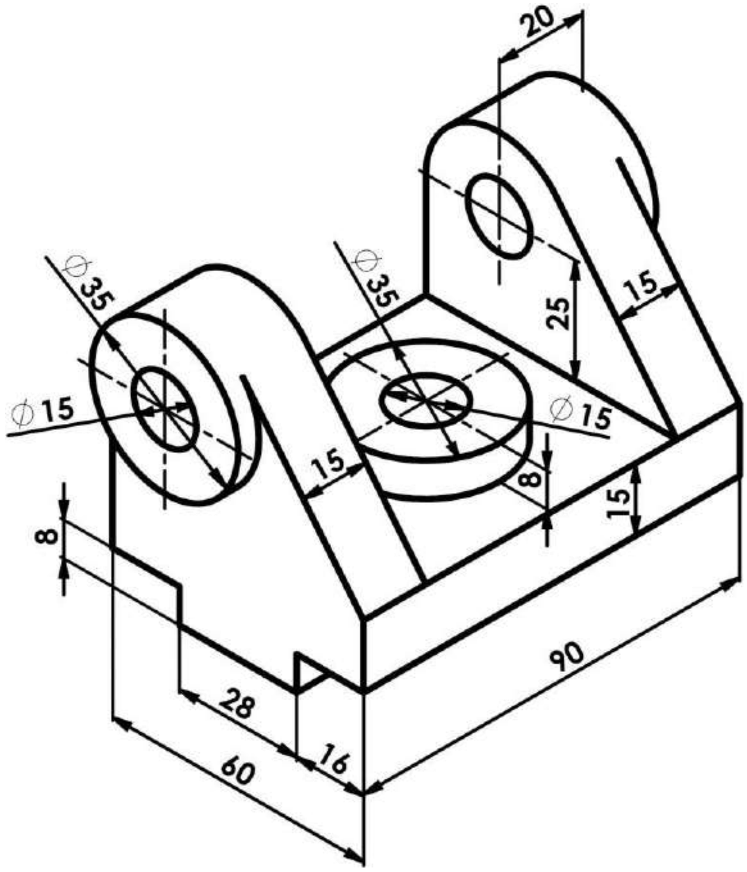
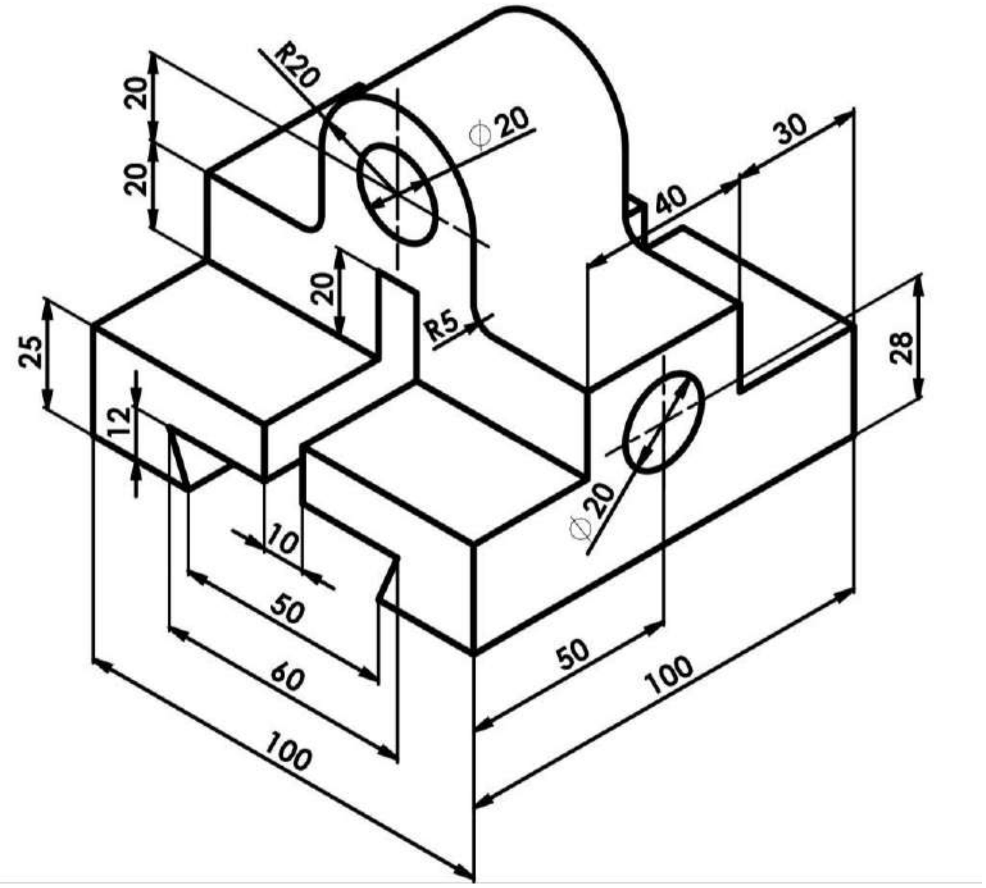
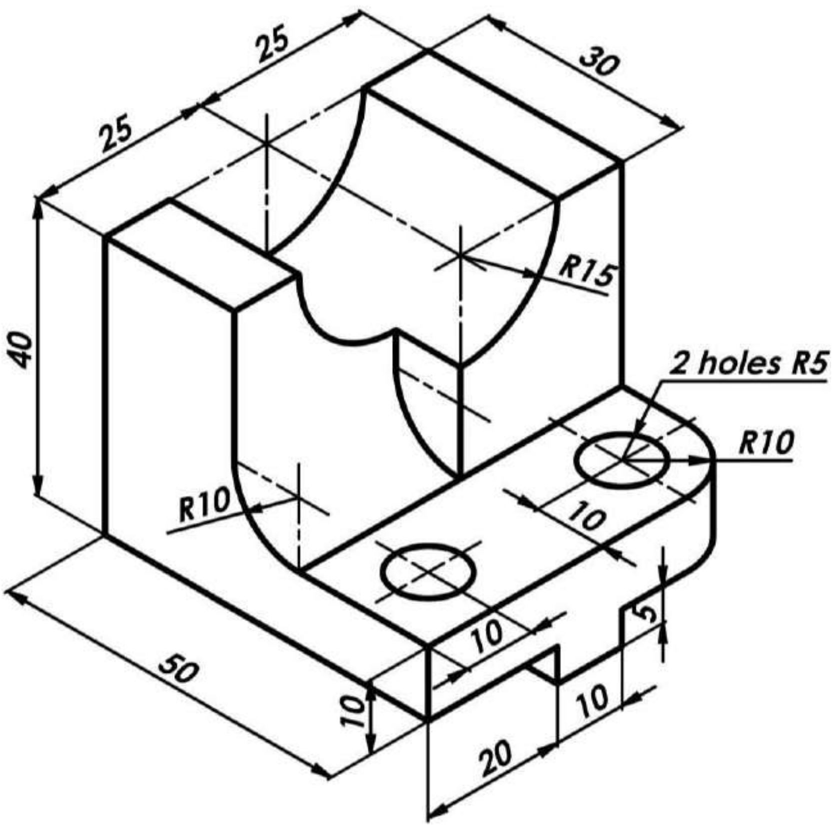
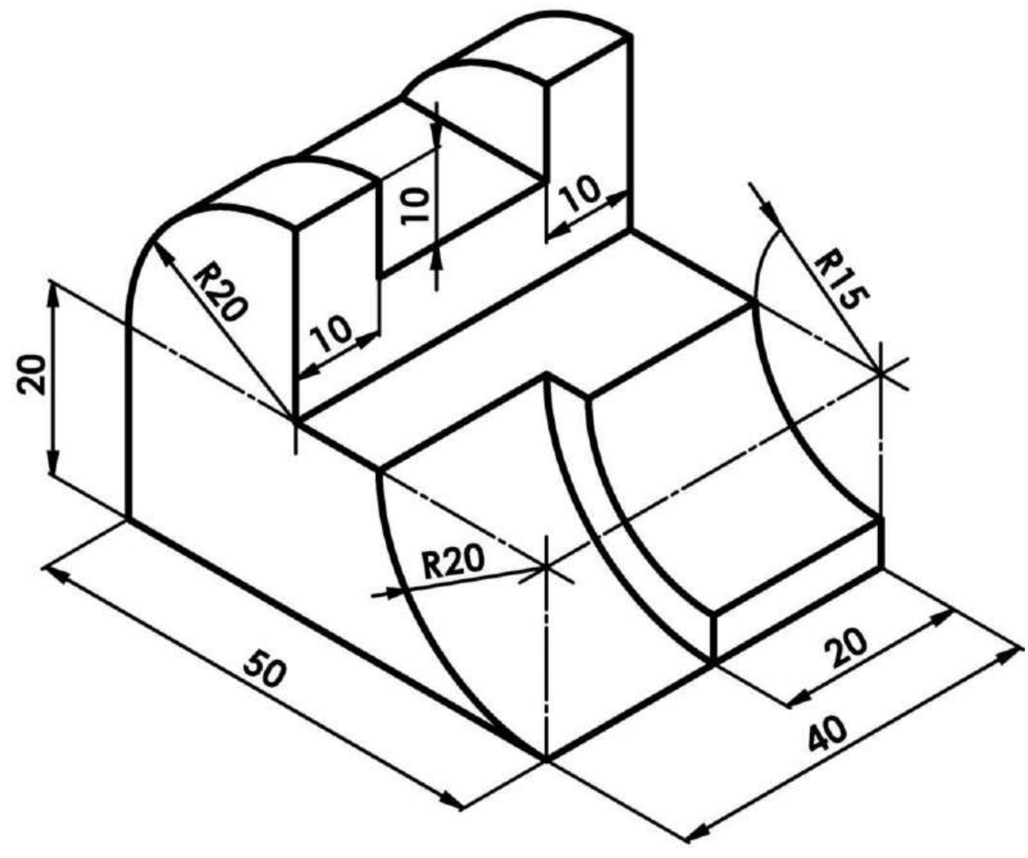
50
85
35
60
85

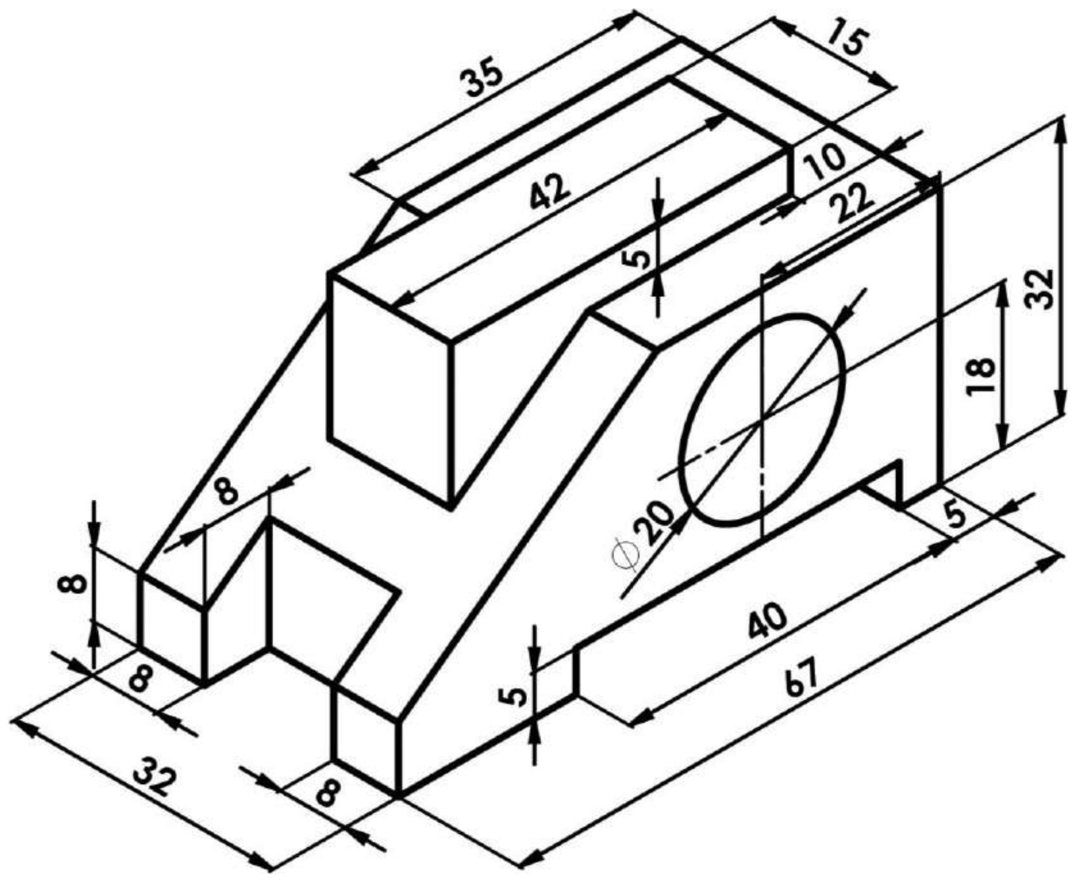

15
Ø30
Ø15
25
10
20
R8
R10
R16
20
10
5
15
30
30
40
60
60
80

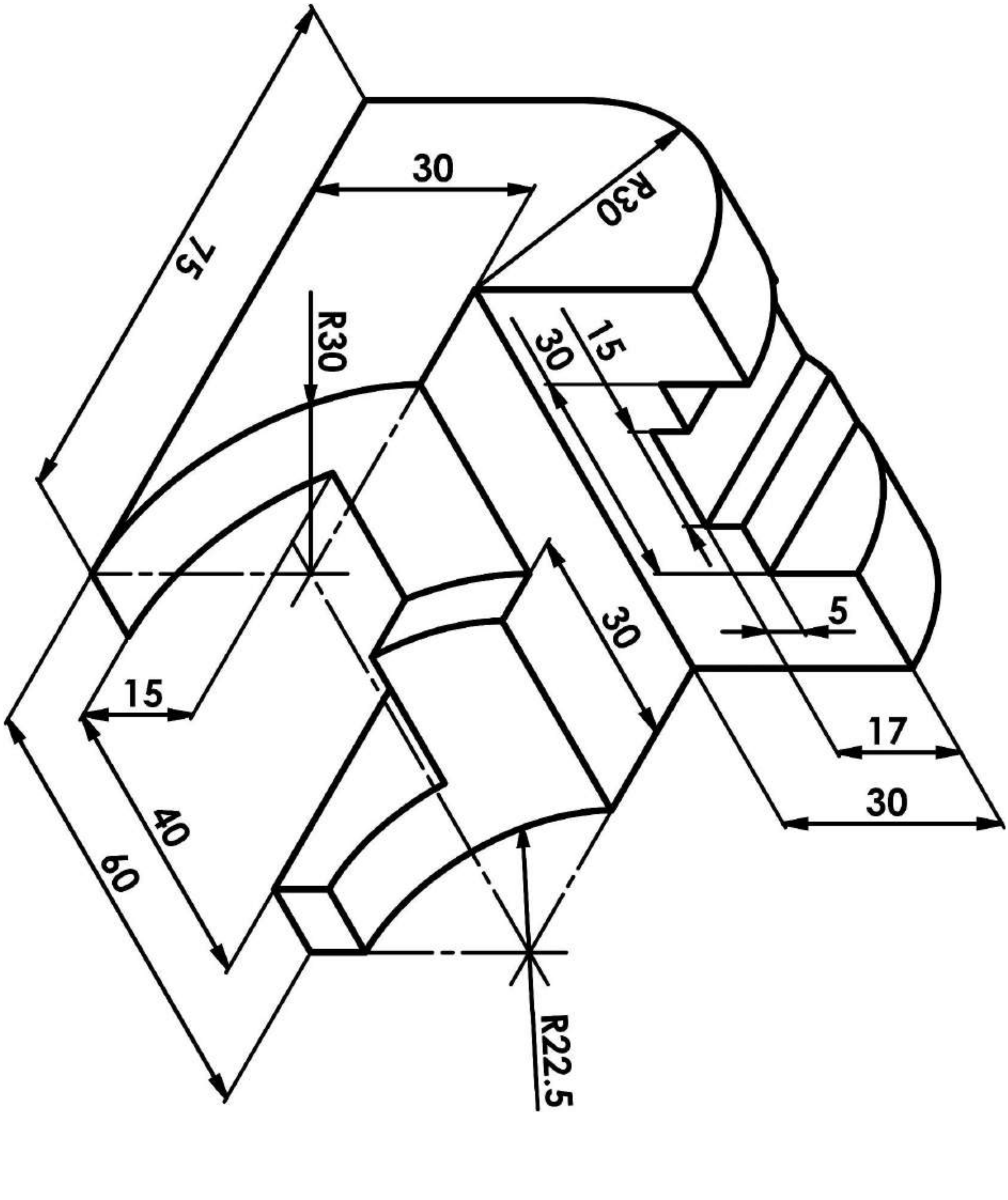

20
Ø35
Ø35
15
25
Ø15
Ø15
15
8
15
8
28
90
60
16

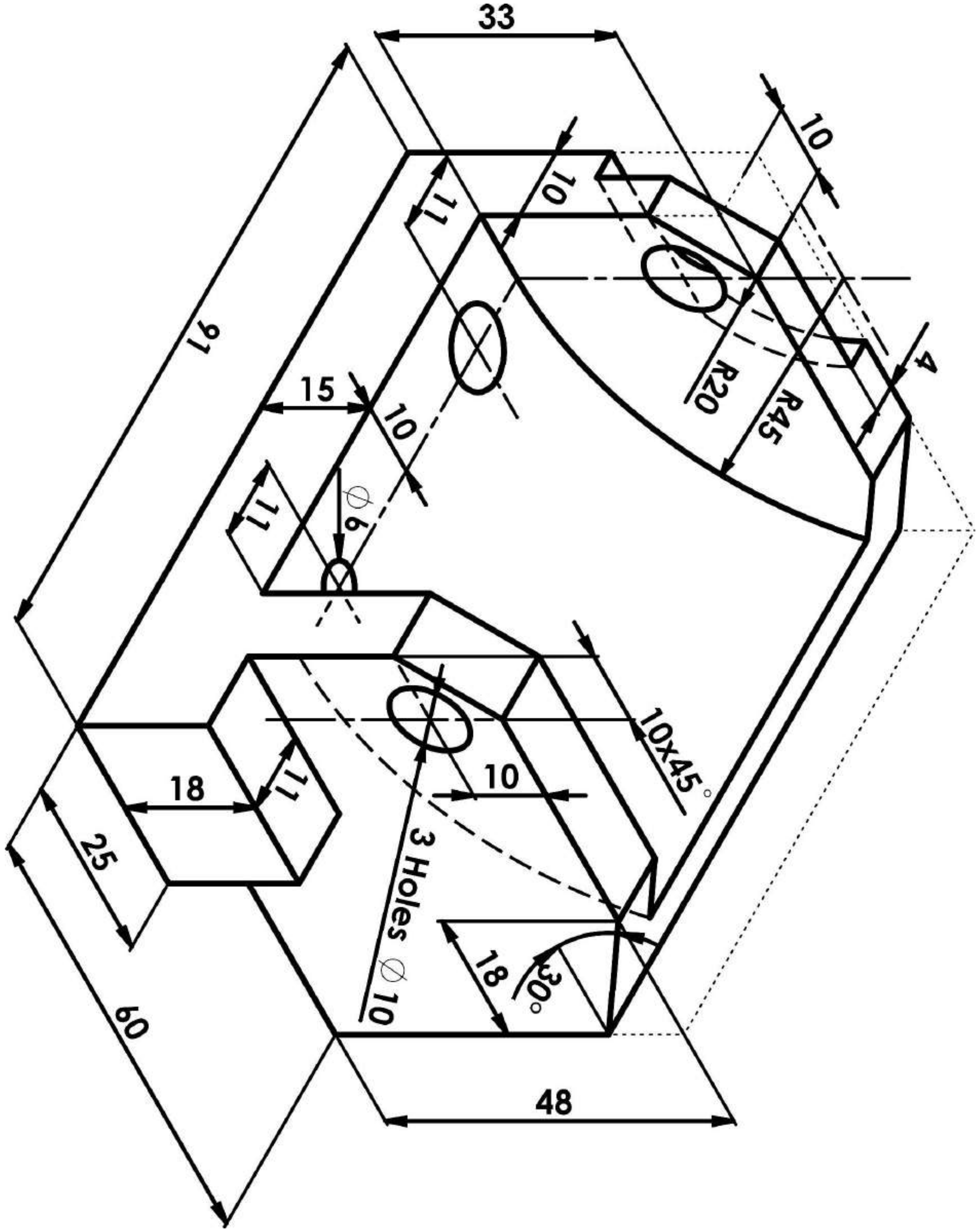

R20
20
20
ϕ20
30
40
20
R5
25
28
12
ϕ20
10
50
50
60
100
100

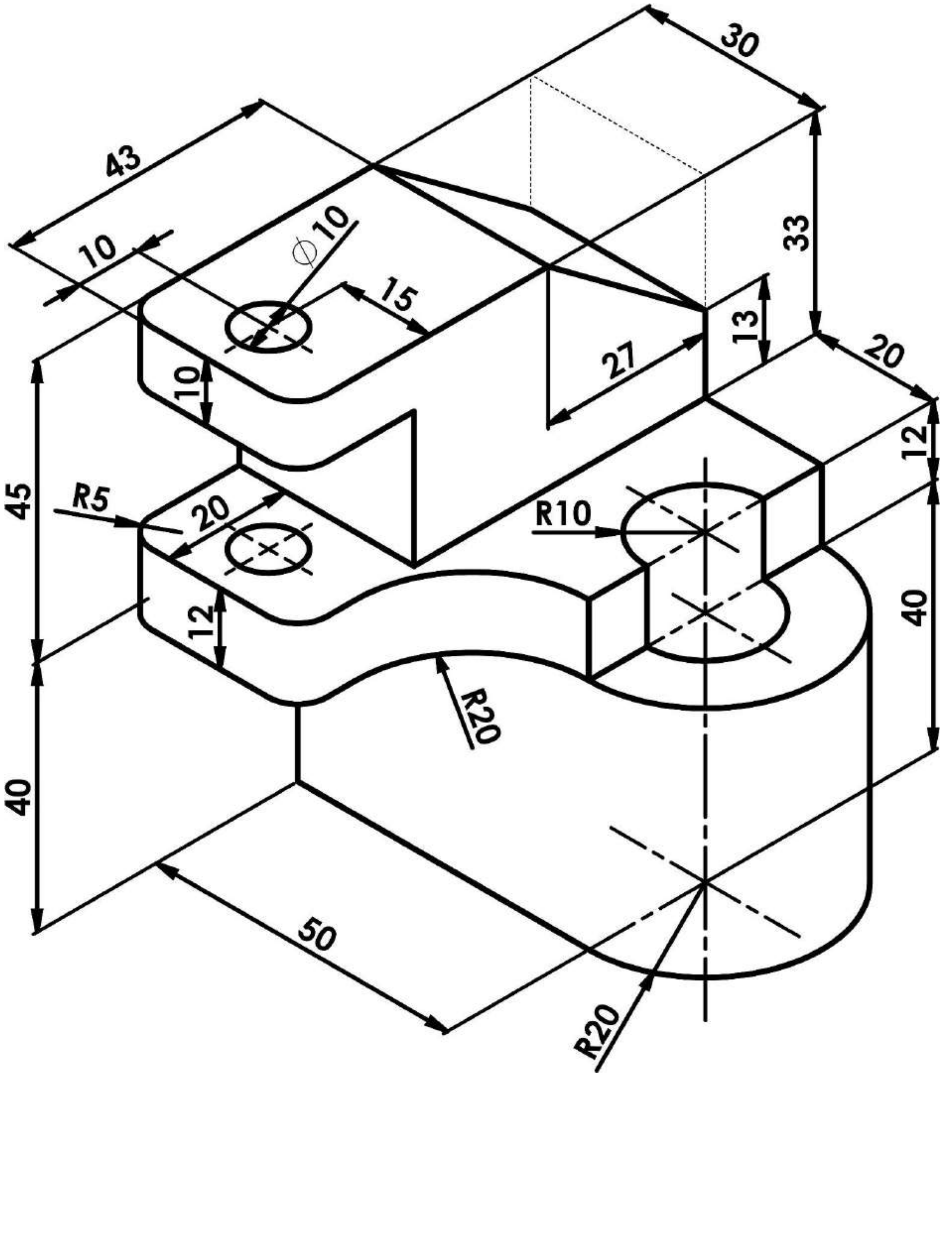

25
25
30
40
R15
2 holes R5
R10
R10
10
50
10
5
10
10
20

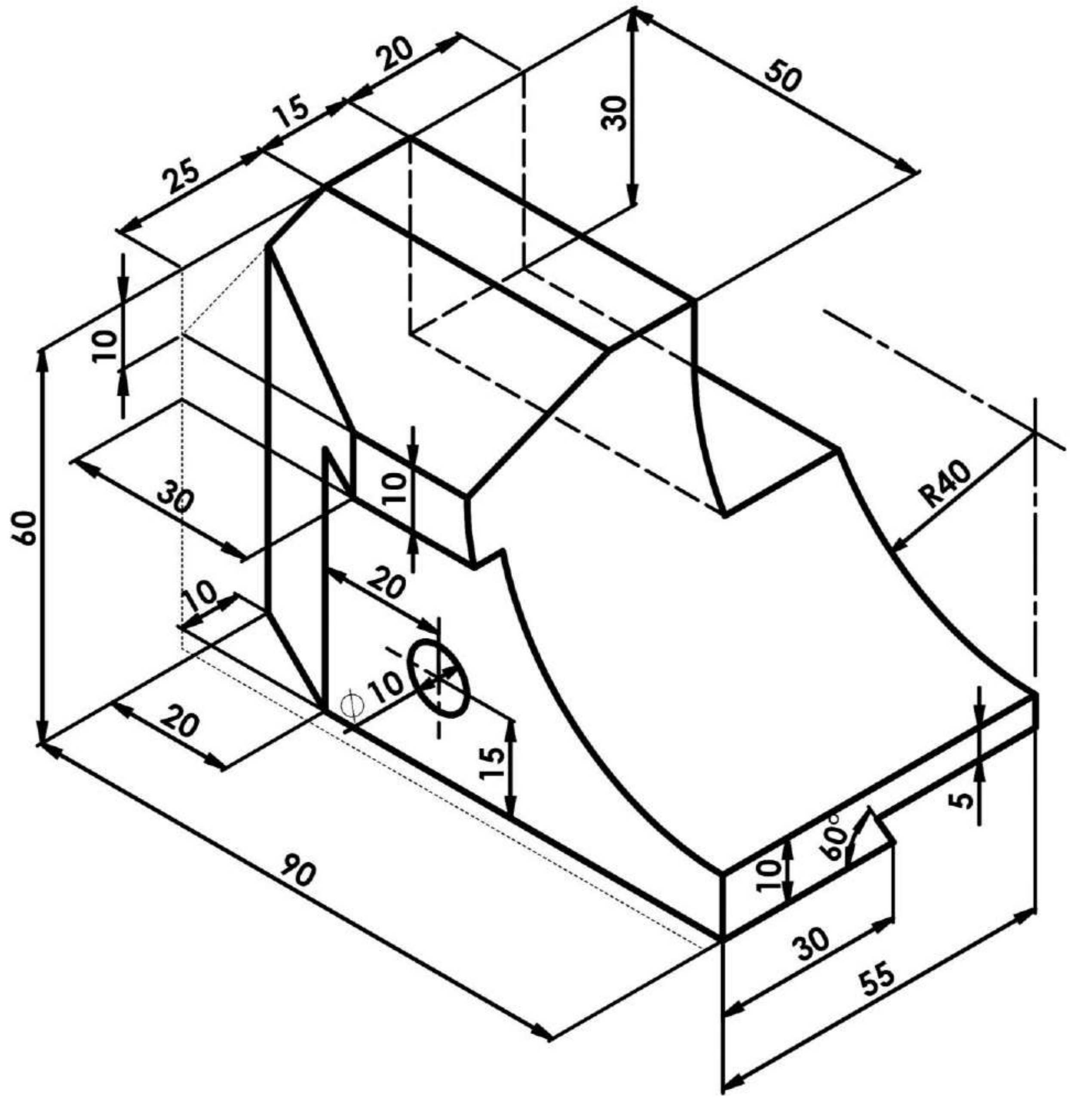

10
10
10
R20
R15
20
R20
50
20
40


35
15
42
10
22
5
32
18
8
8
8
ϕ20
5
40
67
5
32
8


75
30
R30
R30
15
30
30
5
15
17
30
40
60
R22.5


33
10
10
11
91
15
10
Ø6
11
R20
R45
4
10x45°
10
18
11
25
3 Holes Ø10
18
30°
60
48


30
43
33
10
ϕ10
15
13
27
20
10
12
45
R5
20
R10
40
12
R20
40
50
R20



Note: use previous chapter isometric exercise to draw the three views

Chapter 5

Auxiliary Projections

So far we have been able to draw four different views of the same block. In most engineering drawings these are sufficient but there are occasions when other views are necessary, perhaps to clarify a particular point. Figure 5.1 shows examples where a view other than front elevation, side view, or plan is needed to show very important features of geometry.

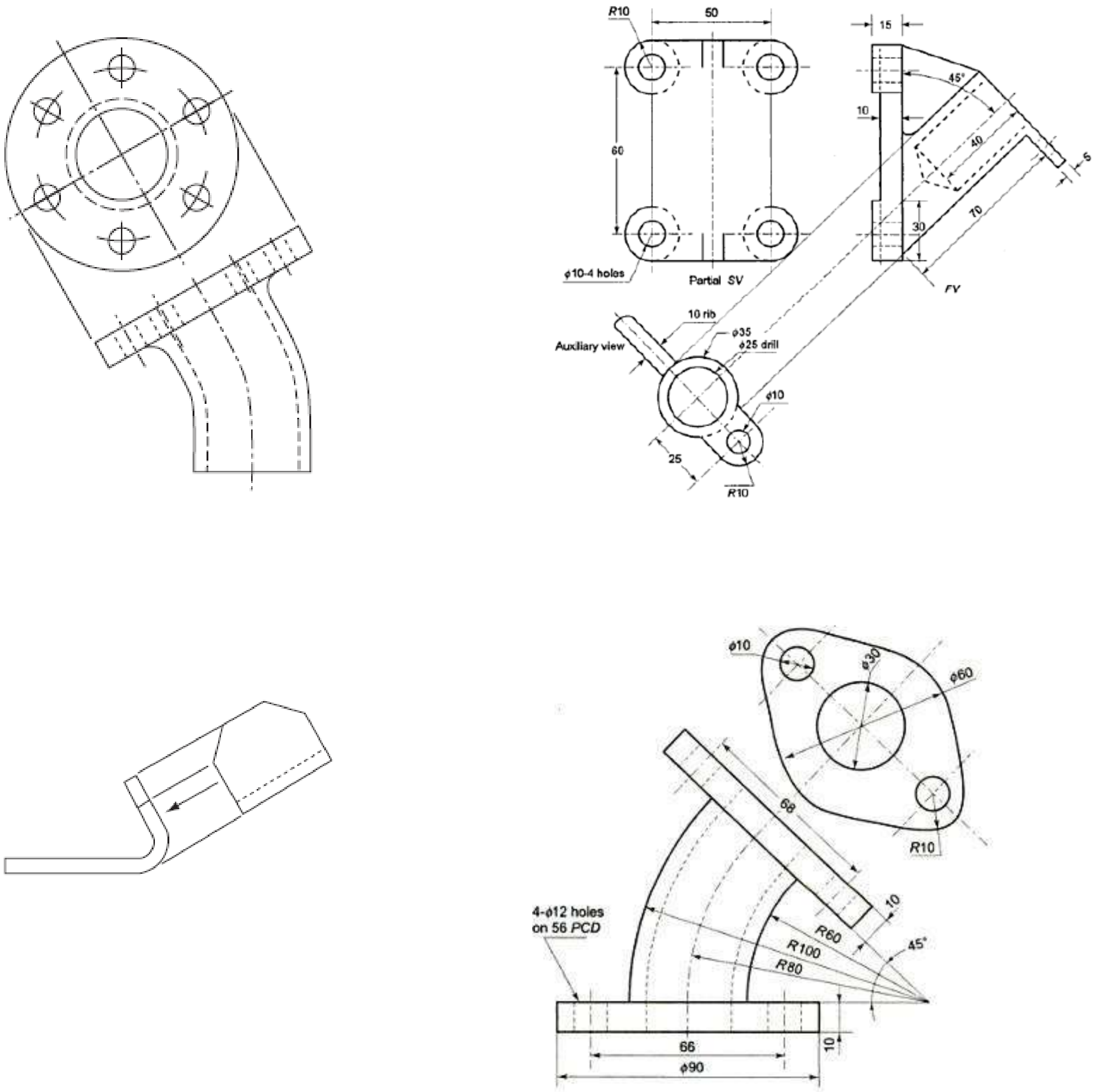


Fig. 5.1 Auxiliary plans for a number of mechanical parts

5.1 Auxiliary views for first angle projection

Extra views are called auxiliary elevations (AE) or auxiliary plans (AP). Figure 5.2 shows an (AE) and an (AP) of the shaped block at first angle projection. One is projected from the plan at 30 ° and the other from the (FE) at 45 °. Projection lines are drawn at those angles and the heights, H and h, are marked off on one AE and the width W on the other.

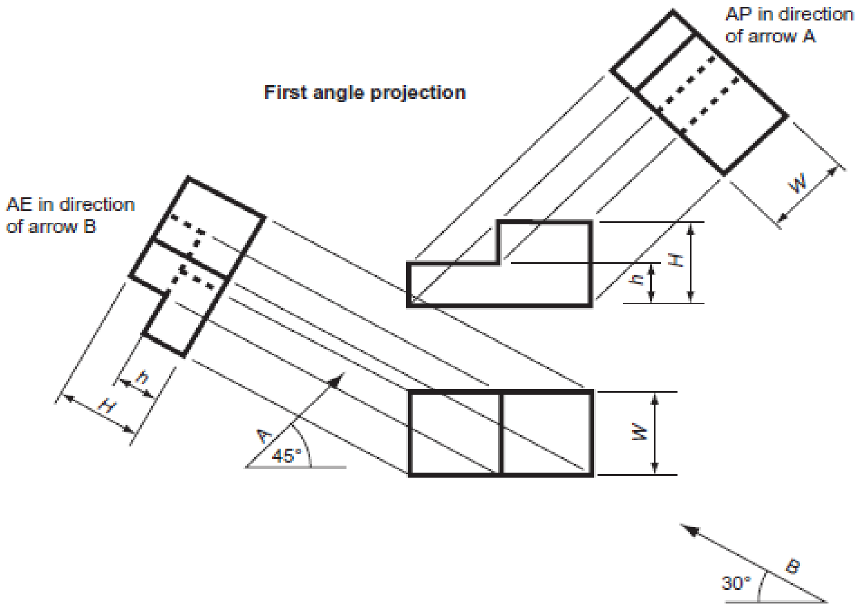


Figure 5.2 Auxiliary elevation & auxiliary plan for first angle projection.

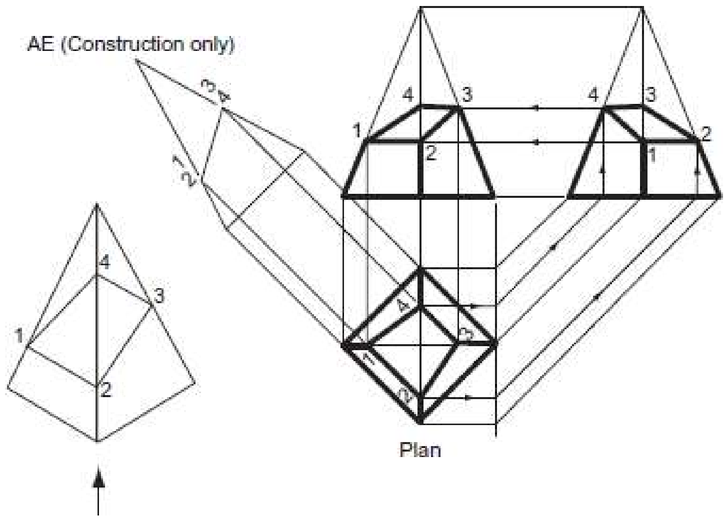


Fig. 5.3 views of a square pyramid.

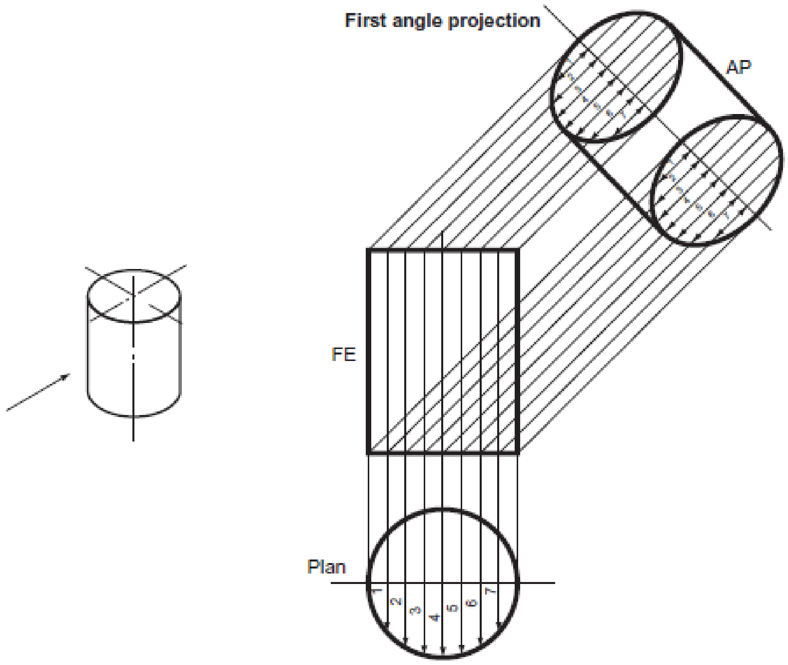


Fig. 5.4 Views of a vertical cylinder.

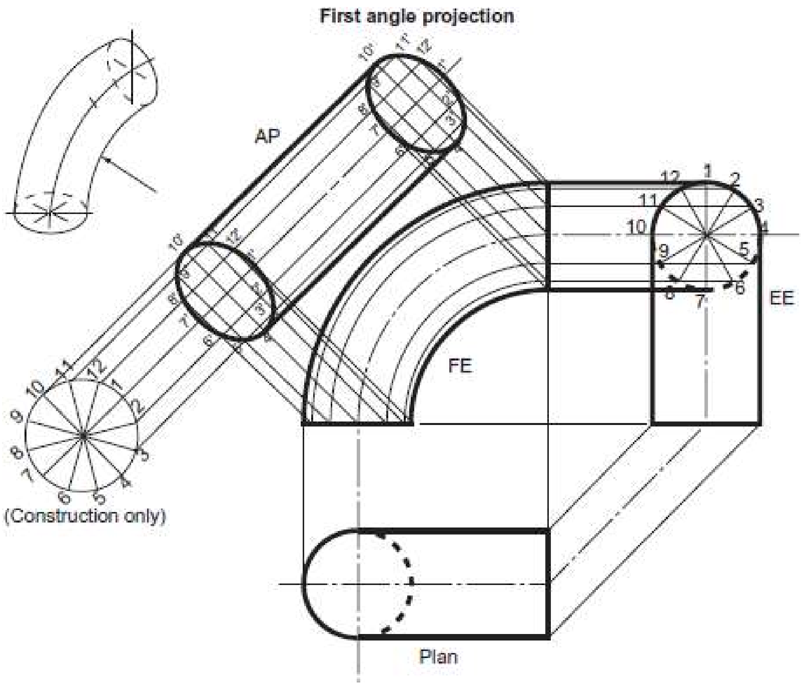


Fig. 5.5 Views of a curved cylinder

5.2 Auxiliary Views for third angle projection

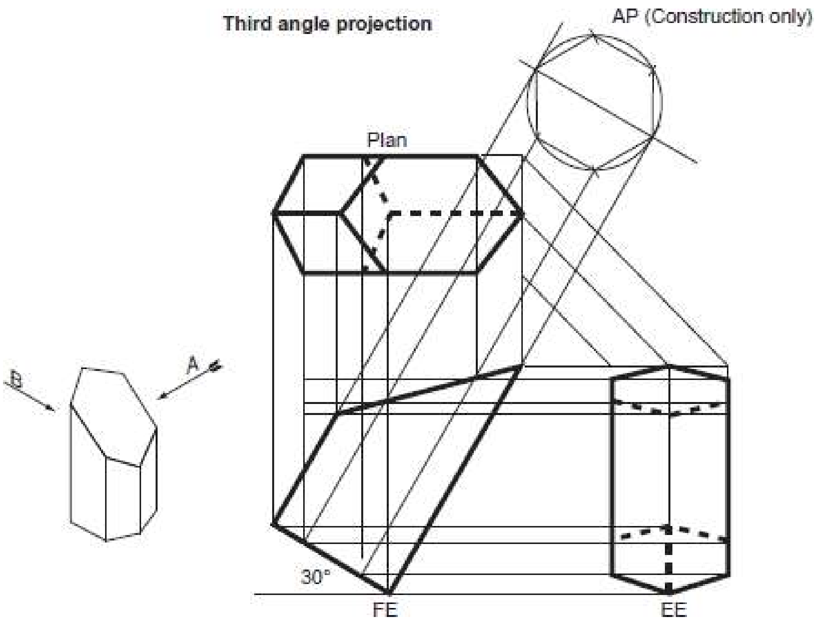


Fig. 5.6 views of a hexagonal prism.

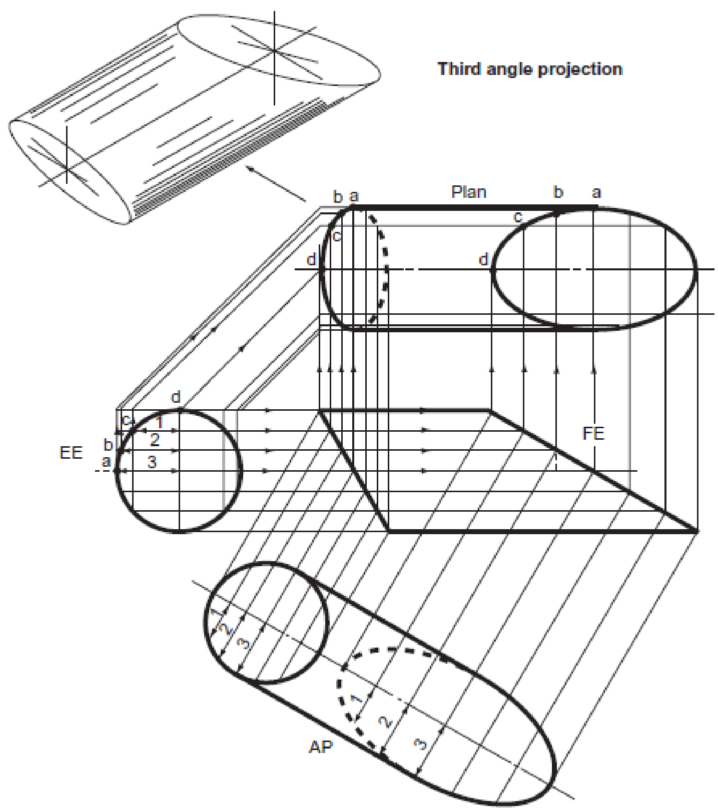


Fig. 5.7 Views of semi cylinder.

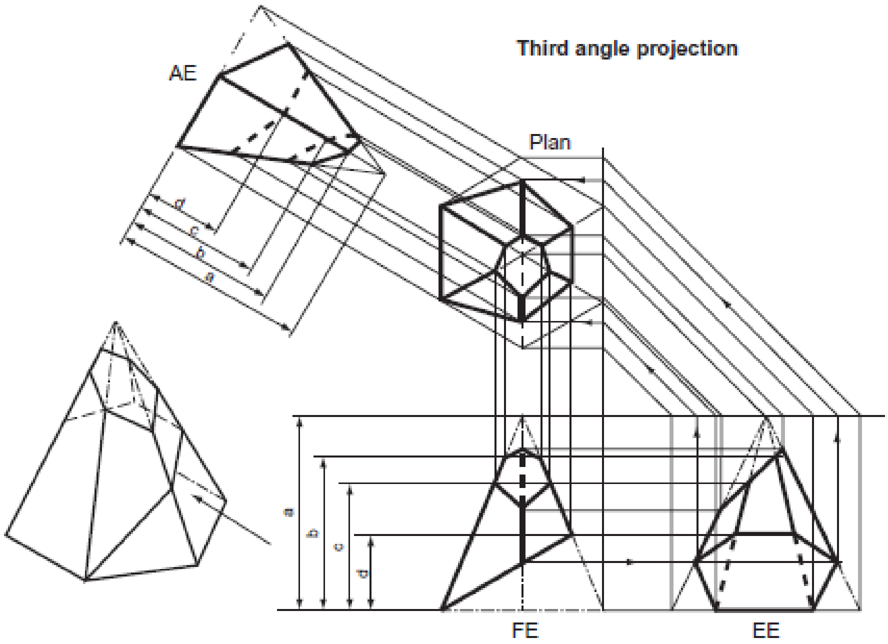


Fig. 5.8 Views of a hexagonal pyramid

5.3 Partial auxiliary view

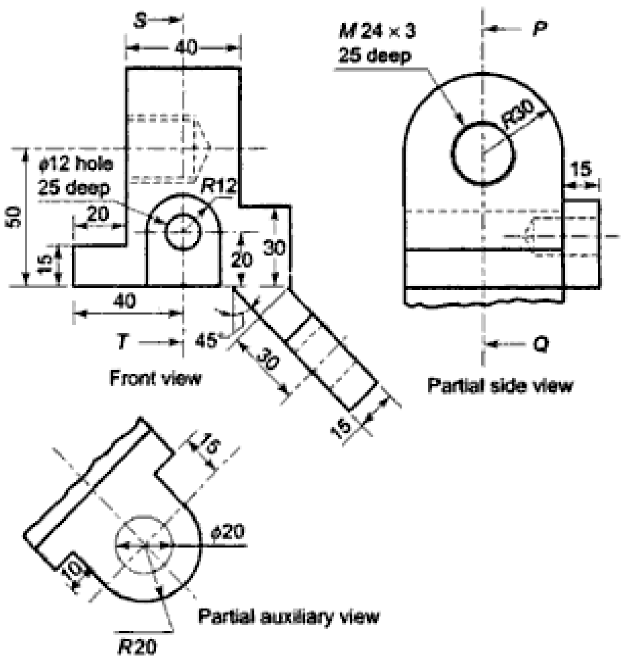


Fig. 5.9 partial auxiliary view

Chapter 6

Intersection of solids

Many objects are formed by a collection of geometrical shapes such as cubes, cones, spheres, cylinders, prisms, pyramids, etc., and where any two of these shapes meet, some sort of curve of intersection results. It is necessary to be able to draw these curves to complete drawings in orthographic projection or to draw patterns and developments.

6.1 Intersection of cylinders

The following drawings show some of the most commonly intersections. If a cylinder intersects another one, number of points should be followed to produce the intersection curve between them as shown in figure 6.1. The arrows indicate the direction in which the projections have been taken.

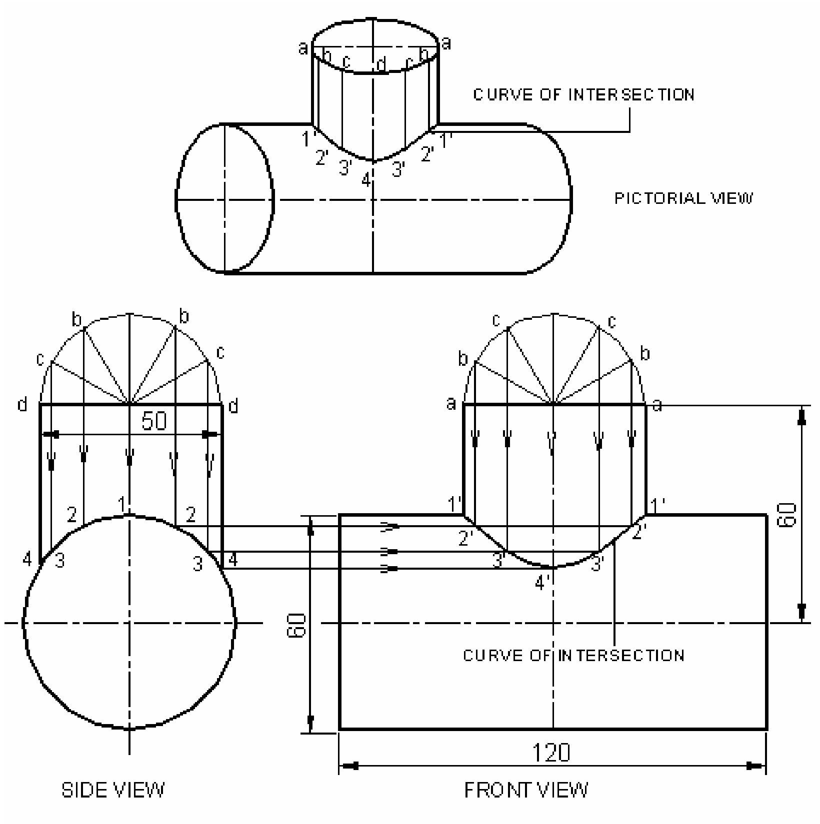


Figure 6.1 The projection views for two perpendicular intersecting cylinders.

A. Intersection between cylinders with different diameters

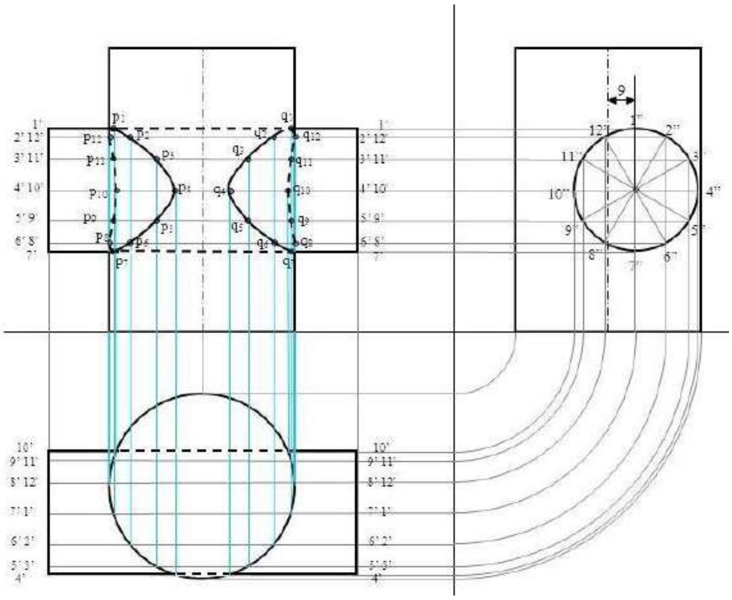


Fig. 6.2 Intersection between perpendicular cylinders with different diameters

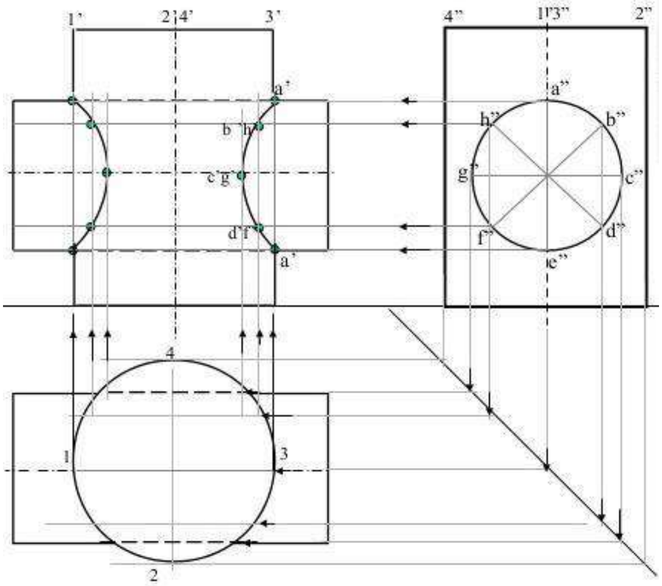


Fig. 6.3 Intersection between perpendicular cylinders with different diameters

B. Intersection between cylinders with equal diameters

When two cylinders of equal diameters intersect as shown in Fig. 6.4, the line at the intersection is straight and at 45^0.

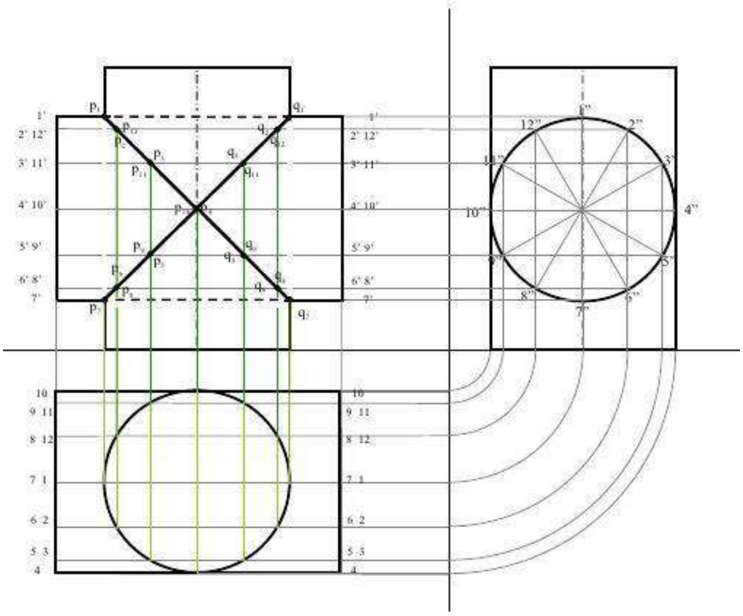


Fig: 6.4 Intersection between perpendicular cylinders with equal diameters

C. Intersection between inclined cylinders

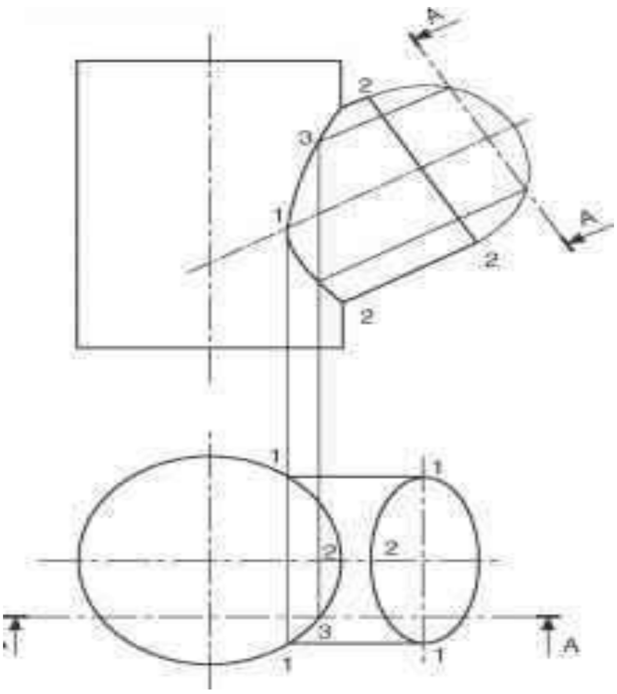


Fig. 6.5 Intersection between inclined cylinders with different diameters

6.2 Intersection between cylinder and cone

A cylinder passing through a cone, will give a circle intersecting the side view, fig. 6.6. The plotting of more points from more sections will give the intersection curves shown in the front elevation and the plan.

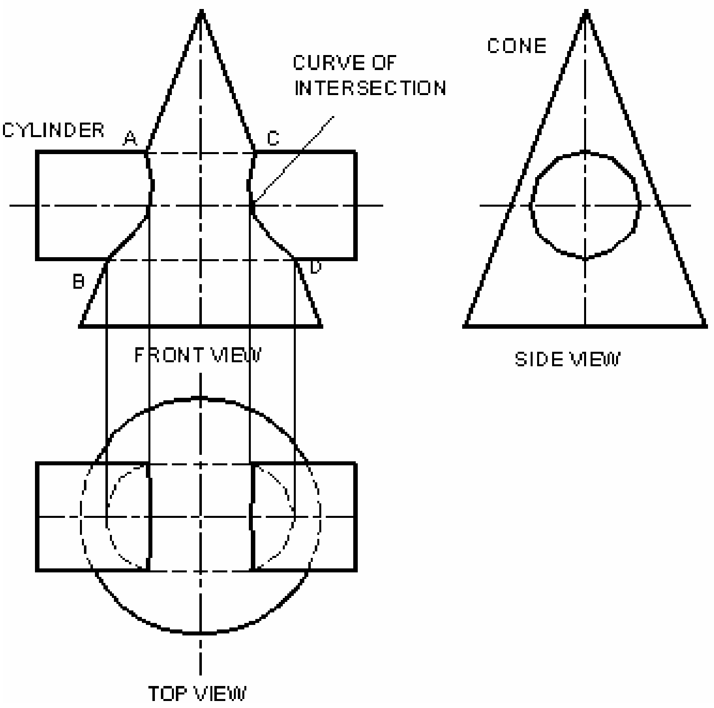
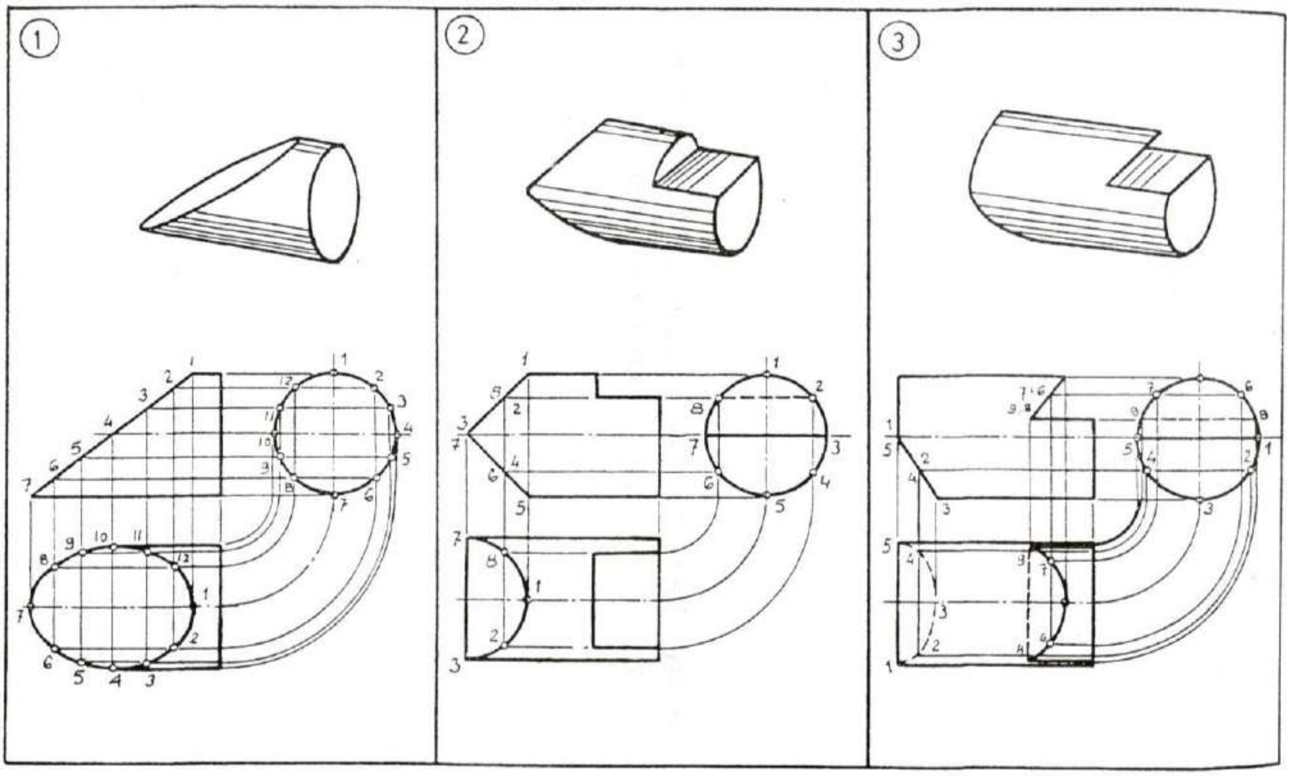
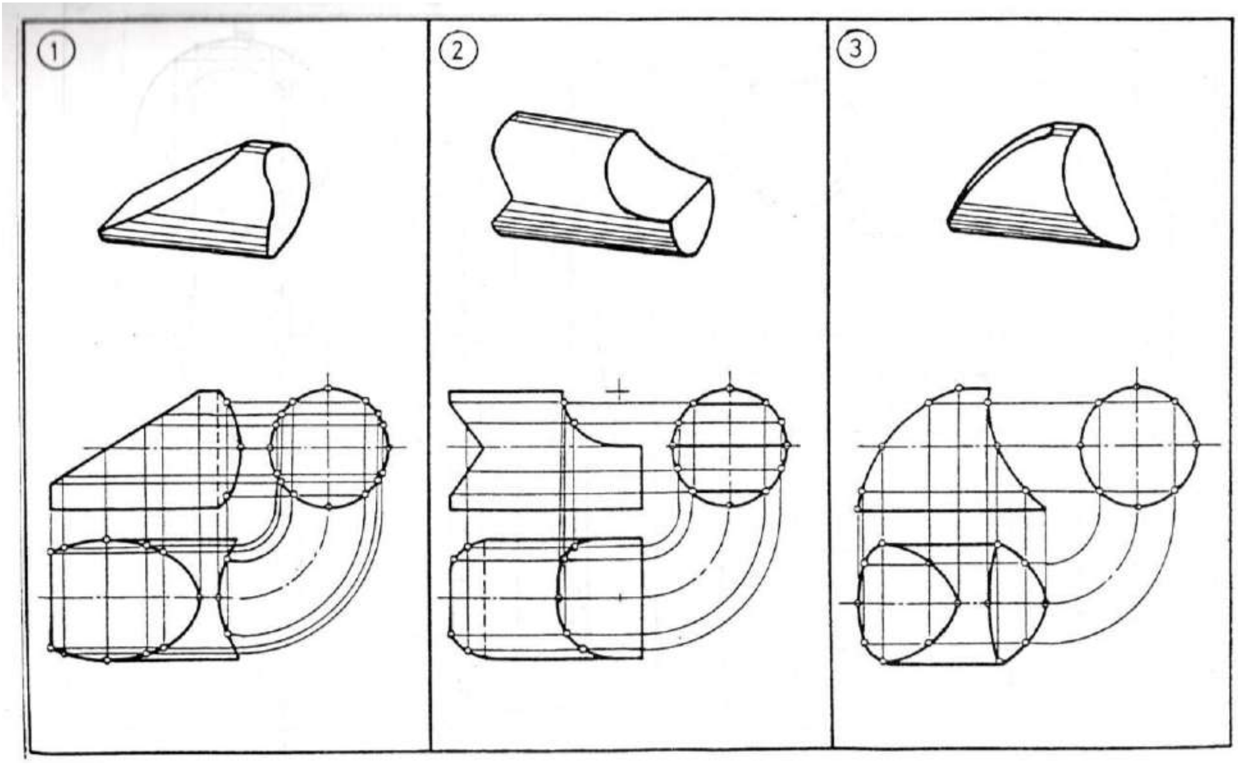
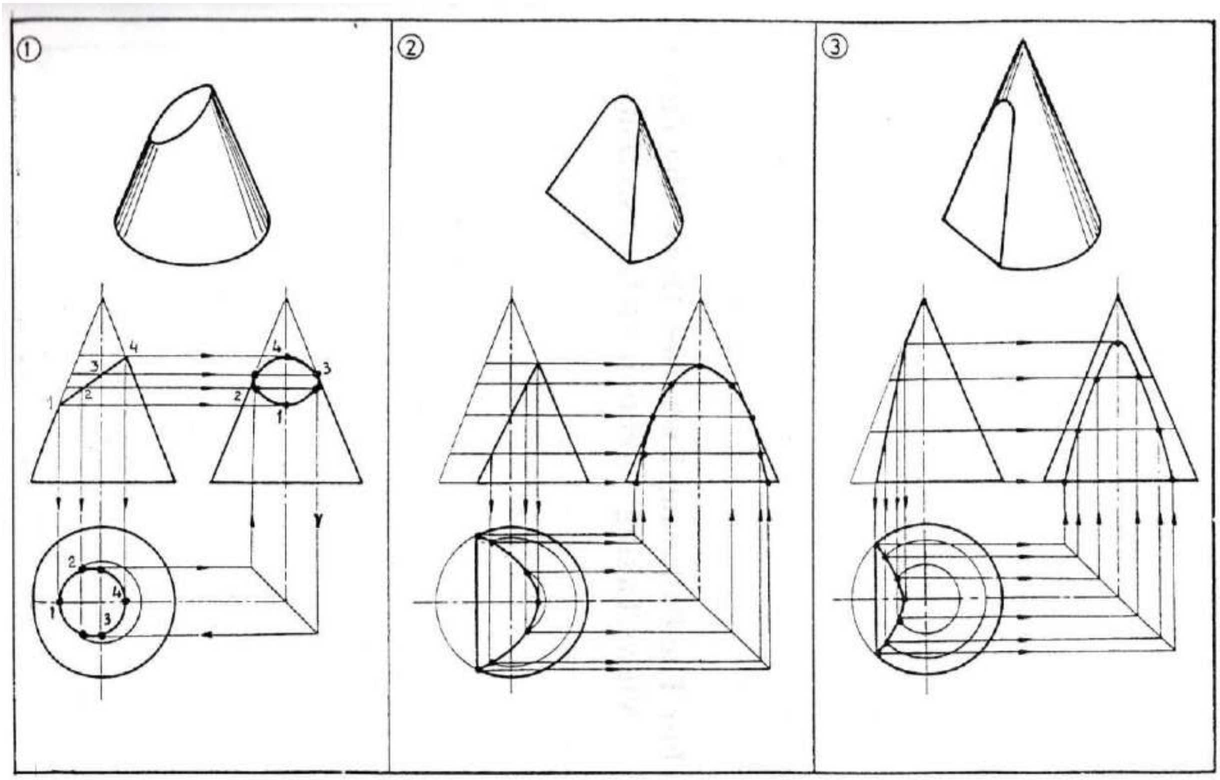


Fig. 6.6 Intersection curve between cylinder and cone

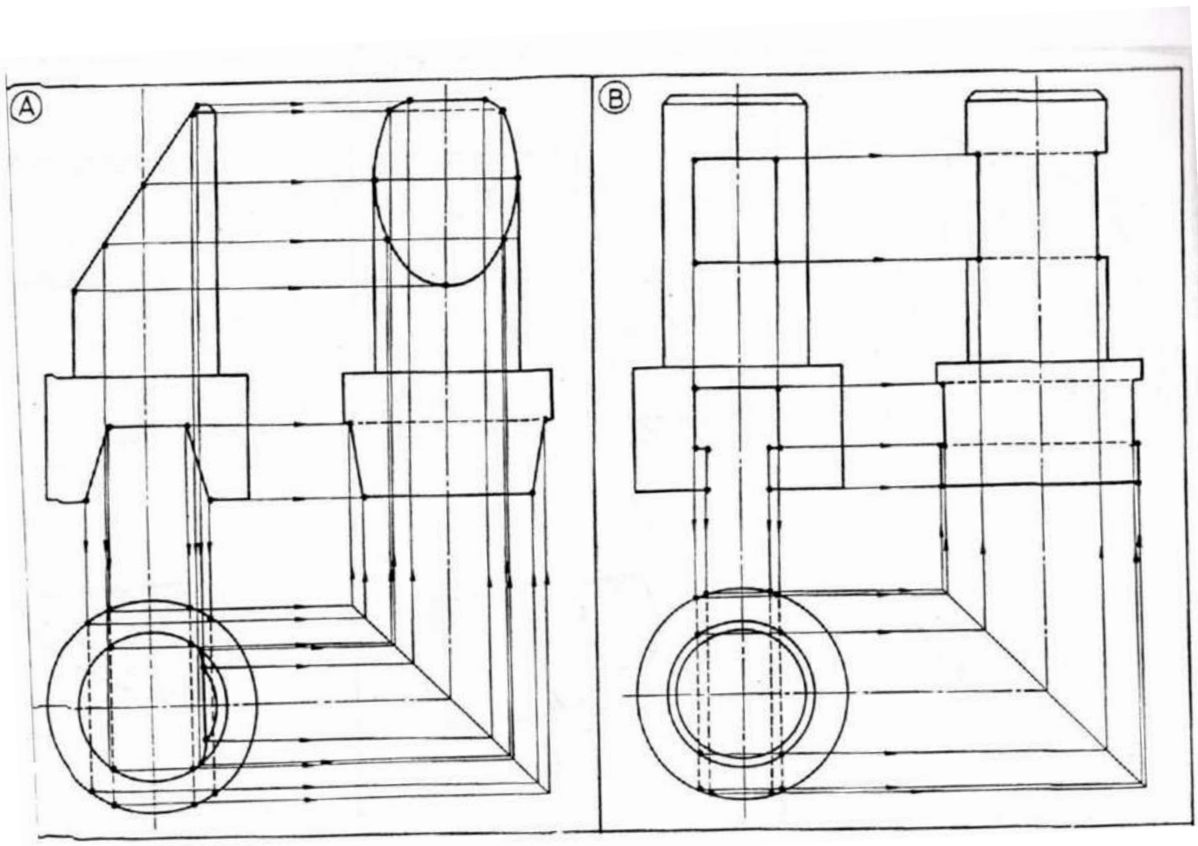
6.3 Examples



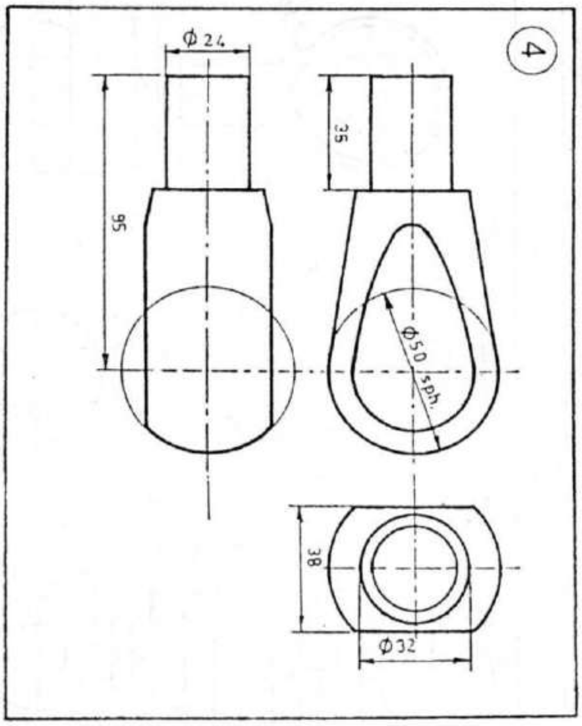


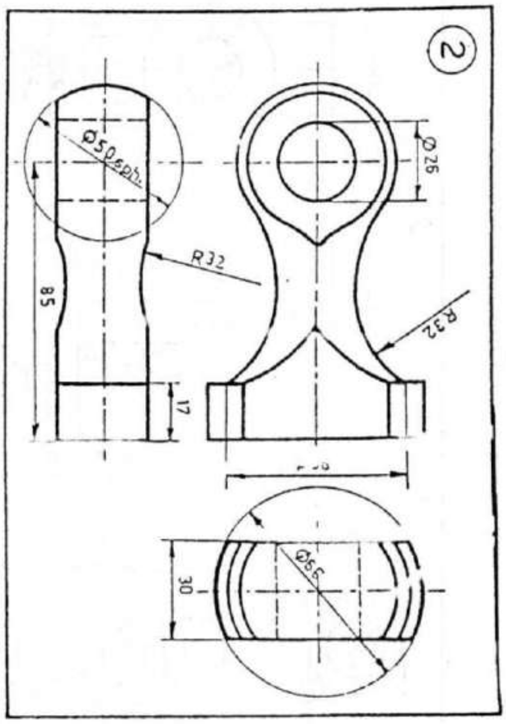


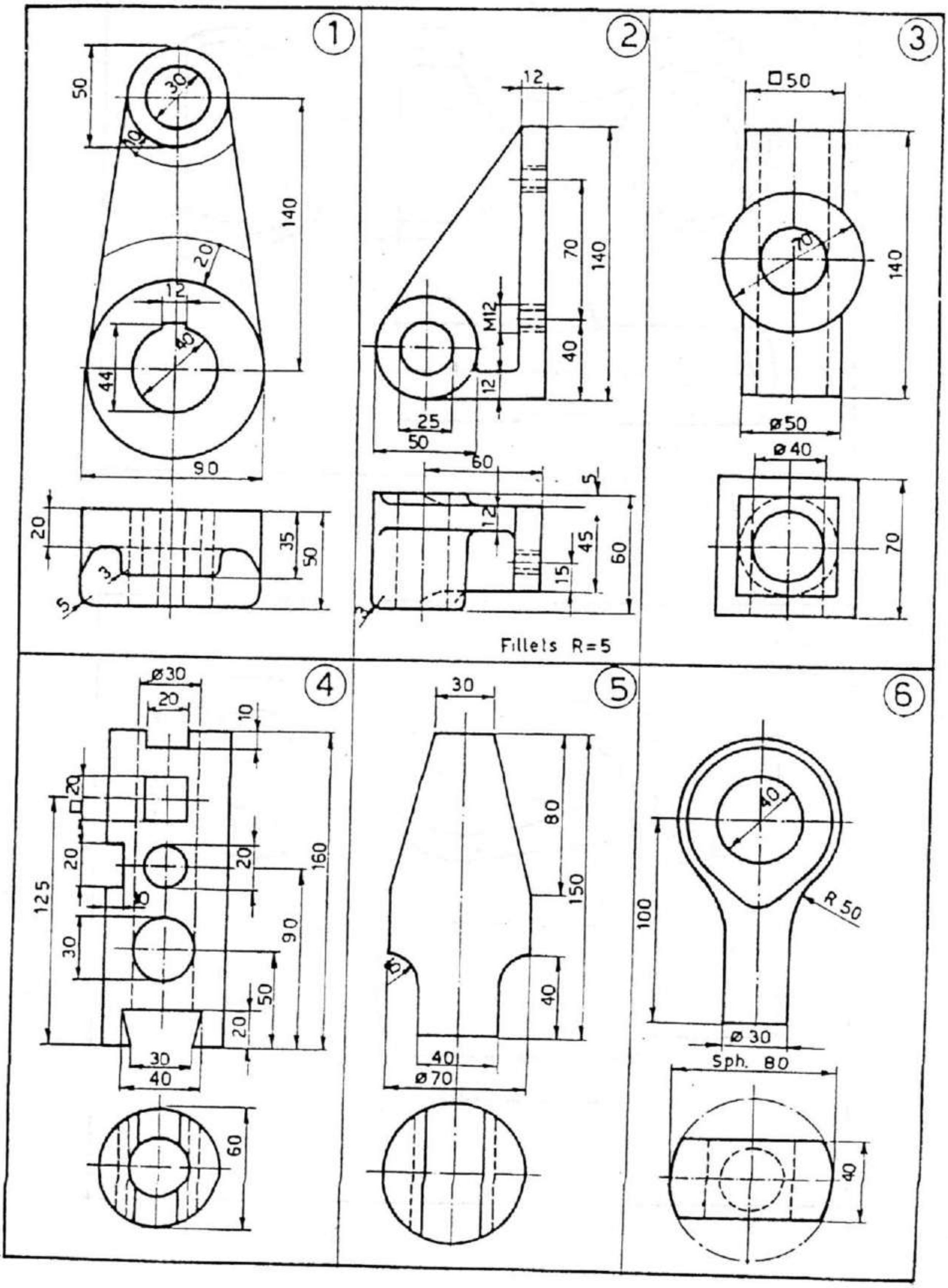
1
2
3
4



A
B


4
Φ24
95
35
Φ50 sph.
38
Φ32


2
Φ50 sph.
Φ26
R32
85
R32
17
Φ66
30


1
2
3
4
5
6
Fillets R=5
Sph. 80
R 50
Ø 30
Ø 70
Ø 50
Ø 40
Ø30
□50
M12

Chapter 7
Dimensions

7.1 Dimension Lines

Dimensions are represented by thin, light, continuous lines drawn outside the outline wherever possible. They are necessary to indicate the extent of a measurement. Fig. 7.1 shows the use of projection and dimension lines with appropriate measurements. Figure 7.2 illustrates the correct and incorrect methods of employing center lines and projection lines for dimensioning purposes.

7.2 Linear Dimensions

In the SI system of units, dimensions of engineering drawing are usually expressed in millimeters. It is not necessary to write the symbol "mm" after every number. A general note such as “all dimensions are in millimeters” in the title block is sufficient.

7.3 Angular Dimensions

Angular dimensions should be stated in degrees, degrees, and minutes. or in degrees, minutes, and seconds, for example. 36.5° or. 36° 30'.

7.4. Methods of Dimensioning

Commonly, there are two methods of indicating dimensions, namely:

A- Unidirectional: where the dimensions are drawn parallel to the bottom of the drawing, i.e., horizontal.

B- Aligned: where the dimensions are drawn parallel to the related dimension line and are readable from the bottom or right-hand side of the drawing.

The two methods of dimensioning are illustrated in Fig.7.3

7.5. Staggered Dimensions

When a number of parallel dimensions are close together, they should be staggered to ensure clear reading, as show in Fig. 7.4.

7.6. Overall Dimensions

When a length consists of a number of dimensions, an overall dimension may be shown outside the dimensions concerned (see Fig. 7.5); the end projection lines are extended to allow this. When an overall dimension is shown, however, one or more of the dimensions which make up the overall length is omitted. This is done to allow for variations in sizes which may occur during production. The omitted dimension is always a non-functional dimension that is one which does not affect the function of the product. A functional dimension is that one which is necessary for the product operation; this type of dimensions is essential.

7.7. Auxiliary Dimensions

When all the dimensions which make up the overall length are given, the overall dimension may be added as an auxiliary dimension. This is indicated by enclosing the dimension in brackets. Figure 7.6 illustrates the use of an auxiliary dimension, namely (100). If the overall length dimension is important, then one of the intermediate dimensions is redundant, for example, the width of the narrow groove in the center. May be inserted as an auxiliary.

7.8. Dimensions not to Scale

When it is desirable to indicate that a dimension is not drawn to scale, the dimension is underlined with a full, heavy line.

7.9. Incomplete Dimensions

Where a dimension defines a large distance or diameter, that cannot be completely inserted on a drawing, the free end is terminated in a double arrowhead pointing in the direction the dimension would take if it could be competed.

7.10. Diameters

a. End View

The symbol o may be used to precede the figure indicating a hole or cylinder. Figure 7.7 illustrates some methods used in dimensioning circles, ranging from small to large diameters.

b. Side View

This may be indicated, as shown in Fig. 7.8(a). Using the symbol Φ preceding the dimension, or using leaders which, are at right angles to the outline in conjunction with the symbol ϕ as shown in Fig.7.8(b).

7.11. Dimensioning Radii and Small Spaces

Dimensions of radii are preceded by the letter R. Figure 7.9 illustrates the methods of dimensioning these features. Leaders should pass through the centers of arcs to which they refer.

7.12. Dimensioning Spherical Surfaces

Spherical surfaces are dimensioned as shown in Fig. 7.10. Note the distinction made between spherical diameters and spherical radii.

7.13 Dimensioning Squares

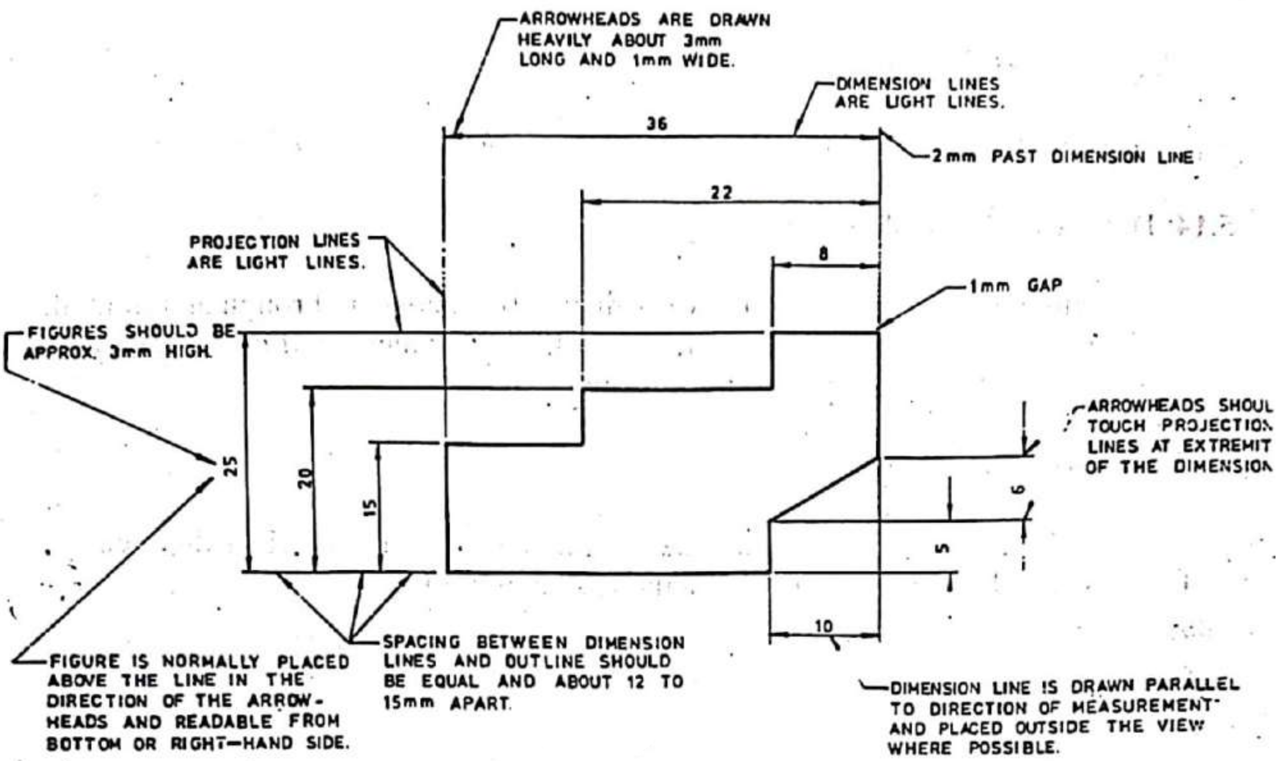
The symbol is used to indicate a square section as shown in Fig 7.11

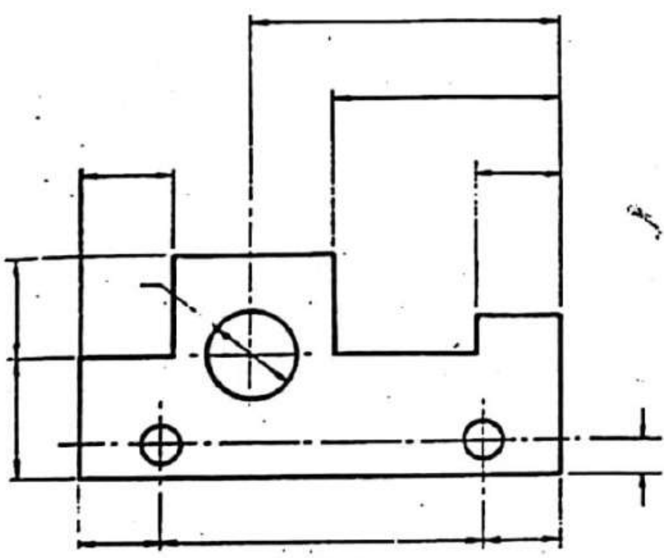
7.14 Dimensioning Holes

In dimensioning holes, you have to distinguish between through and limited depth holes; this must be specified as well as the diameter. If no indication is given, the hole is taken as a through one as shown in Fig 7.12.

7.15 Dimensioning Flanges

Bolt holes on flanges may be positioned around the CD. (Pitch circle diameter) by either of the two methods shown in Fig. 7.13.





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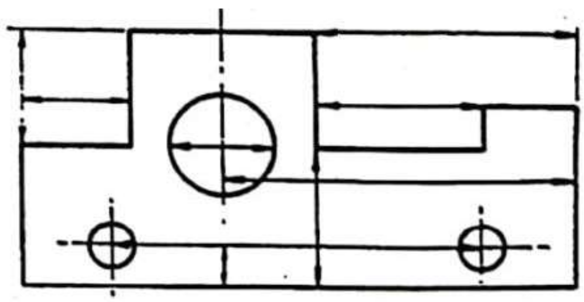

INCORRECT

Fig. 7.1

Fig. 7.2

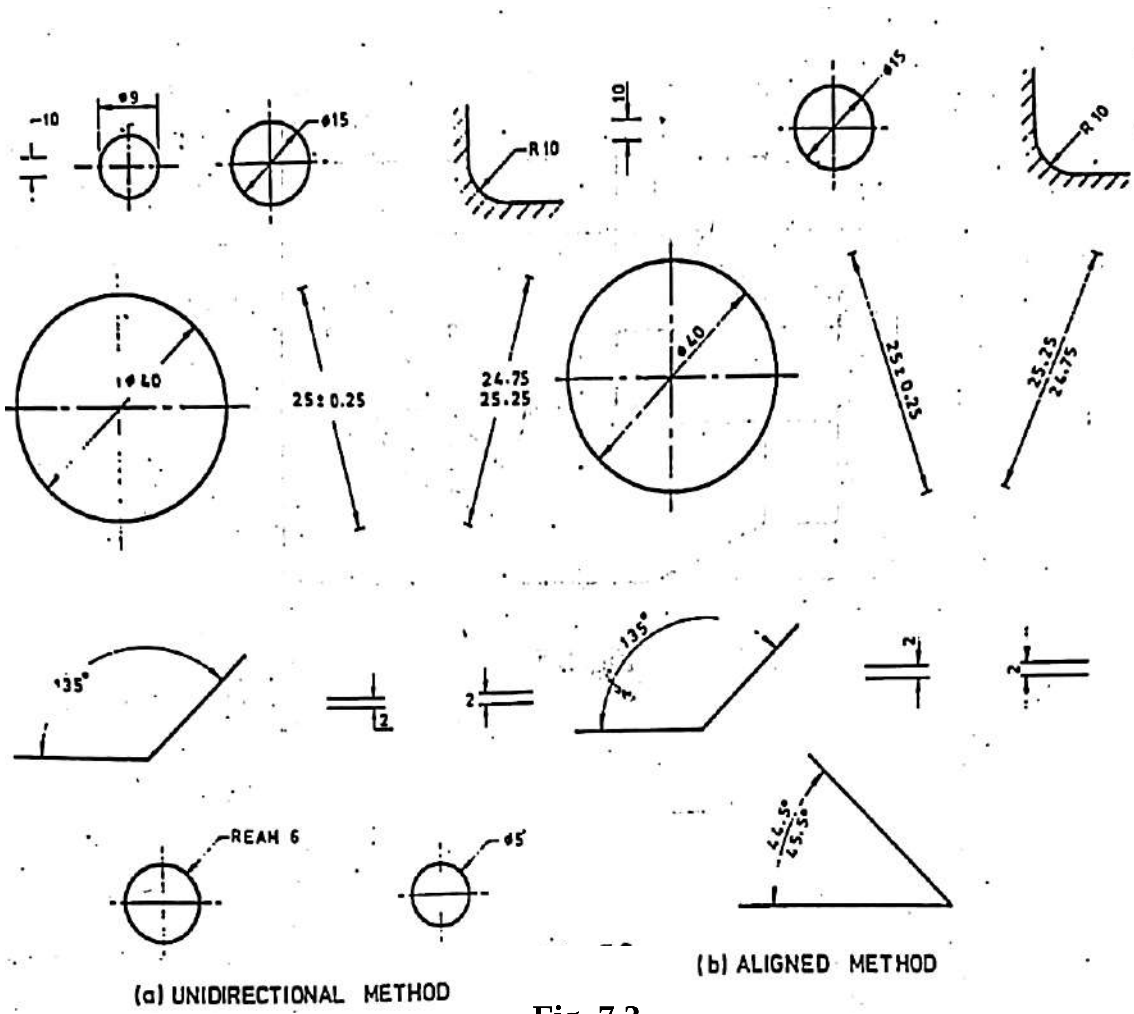

Fig. 7.3

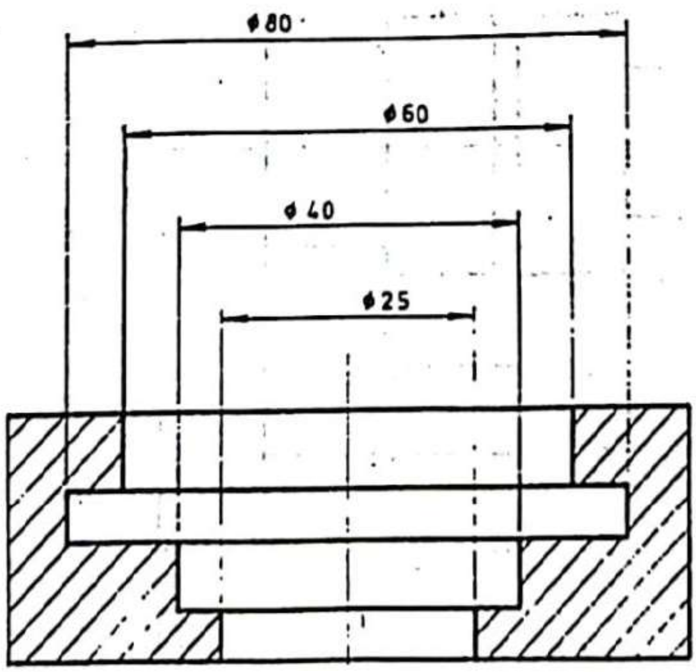

Fig. 7.4

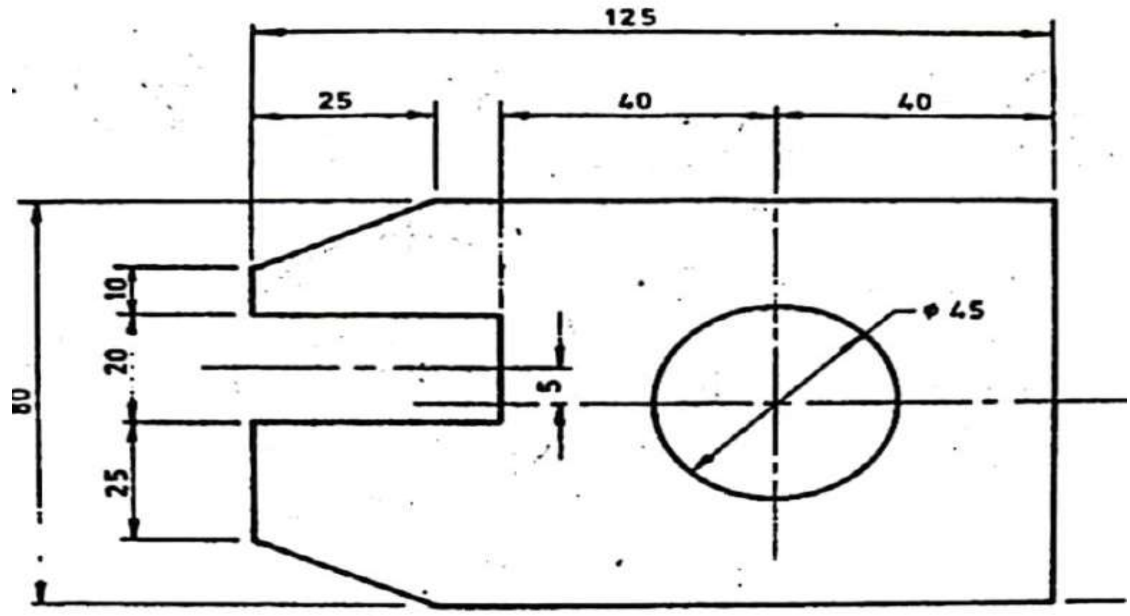

Fig. 7.5

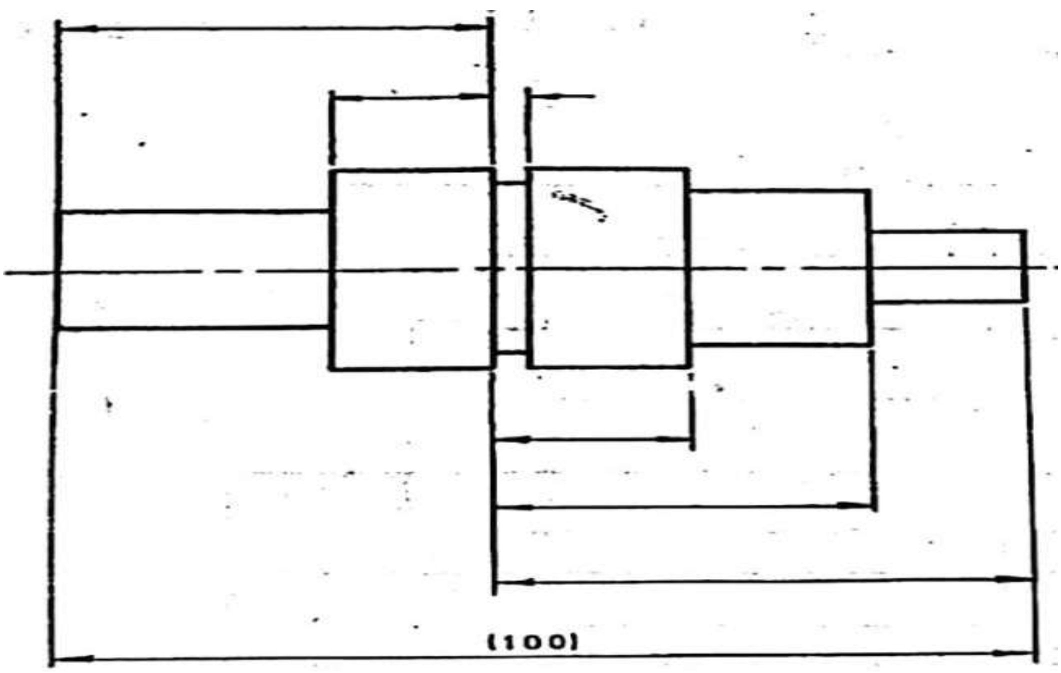

Fig. 7.6

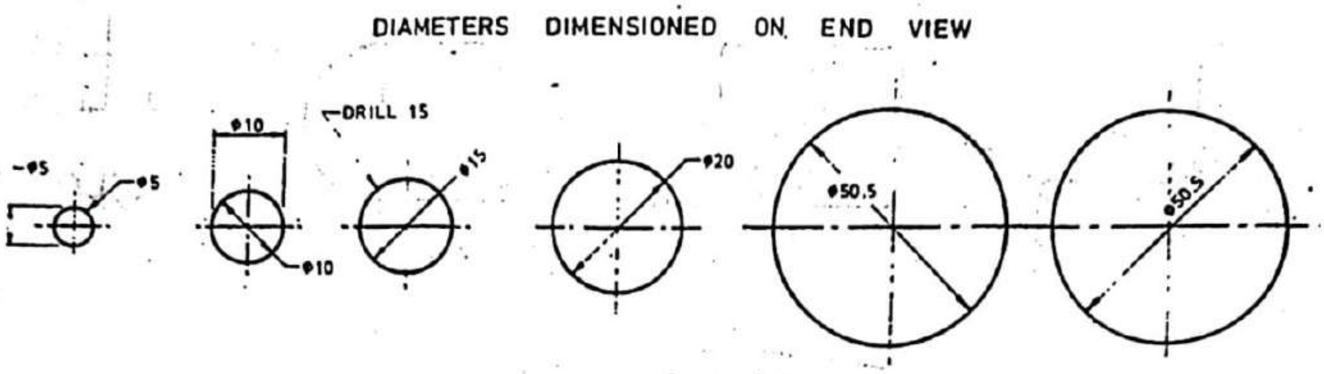

Fig. 7.7

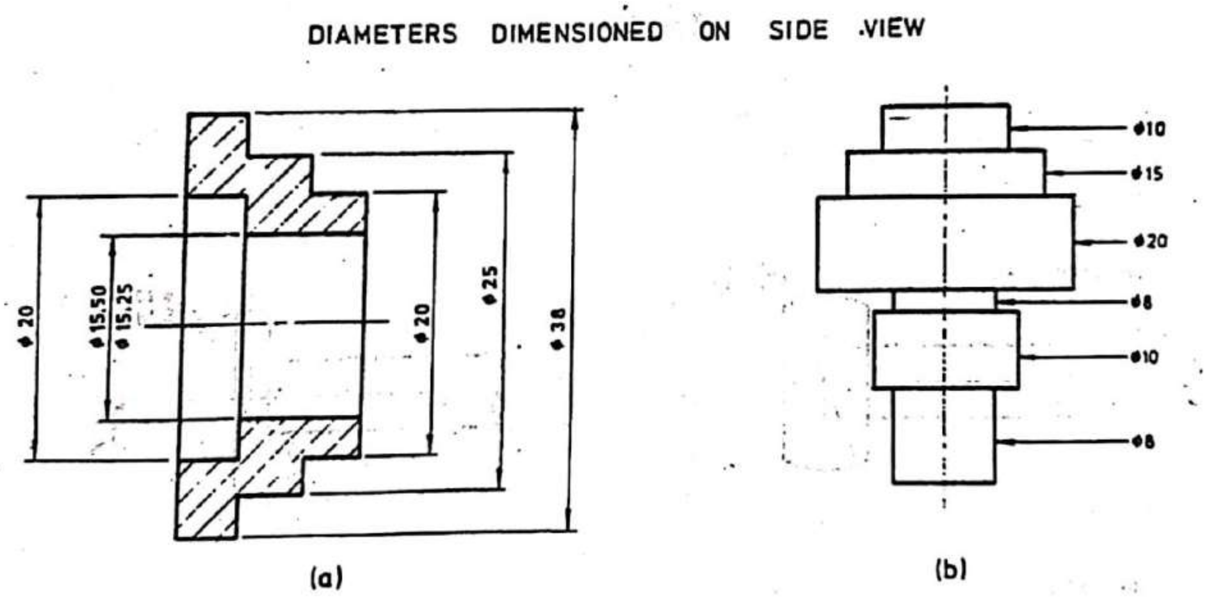

Fig.7.8

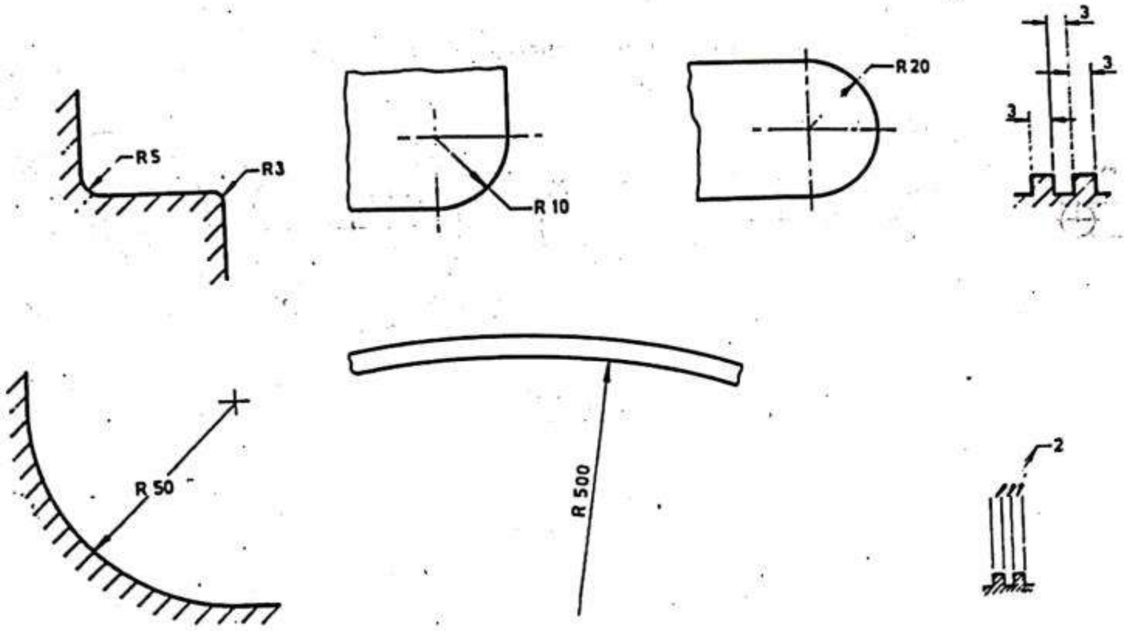

Fig.7.9

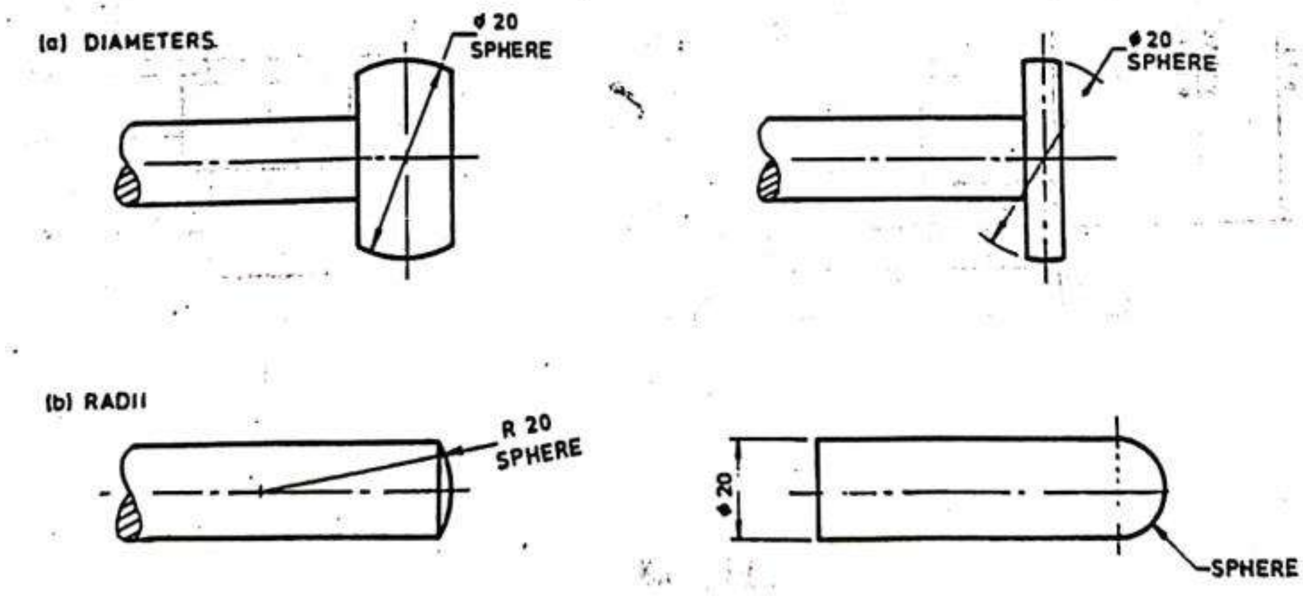

Fig. 7.10

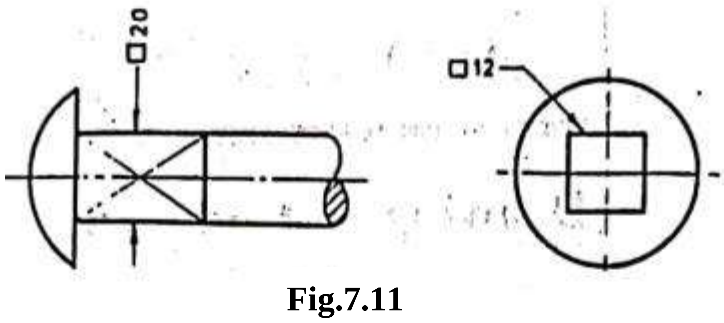

Fig.7.11

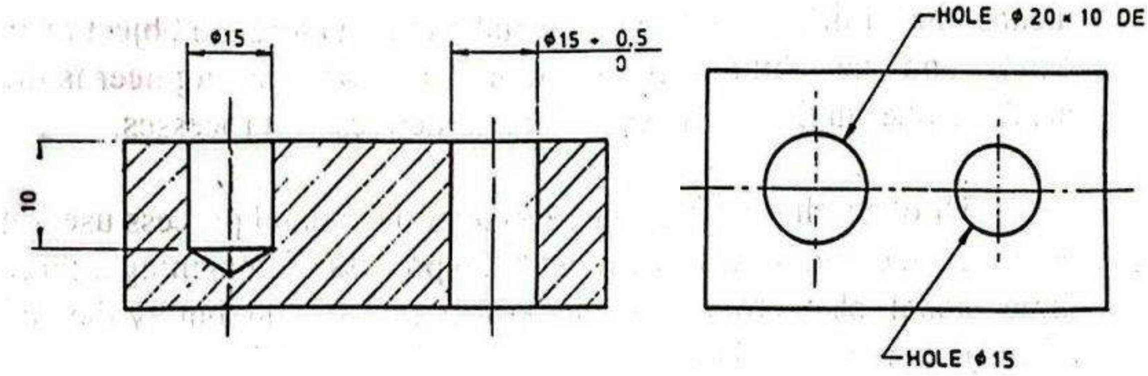

Fig. 7.12 illustrates methods of dimensioning holes using both end and side views

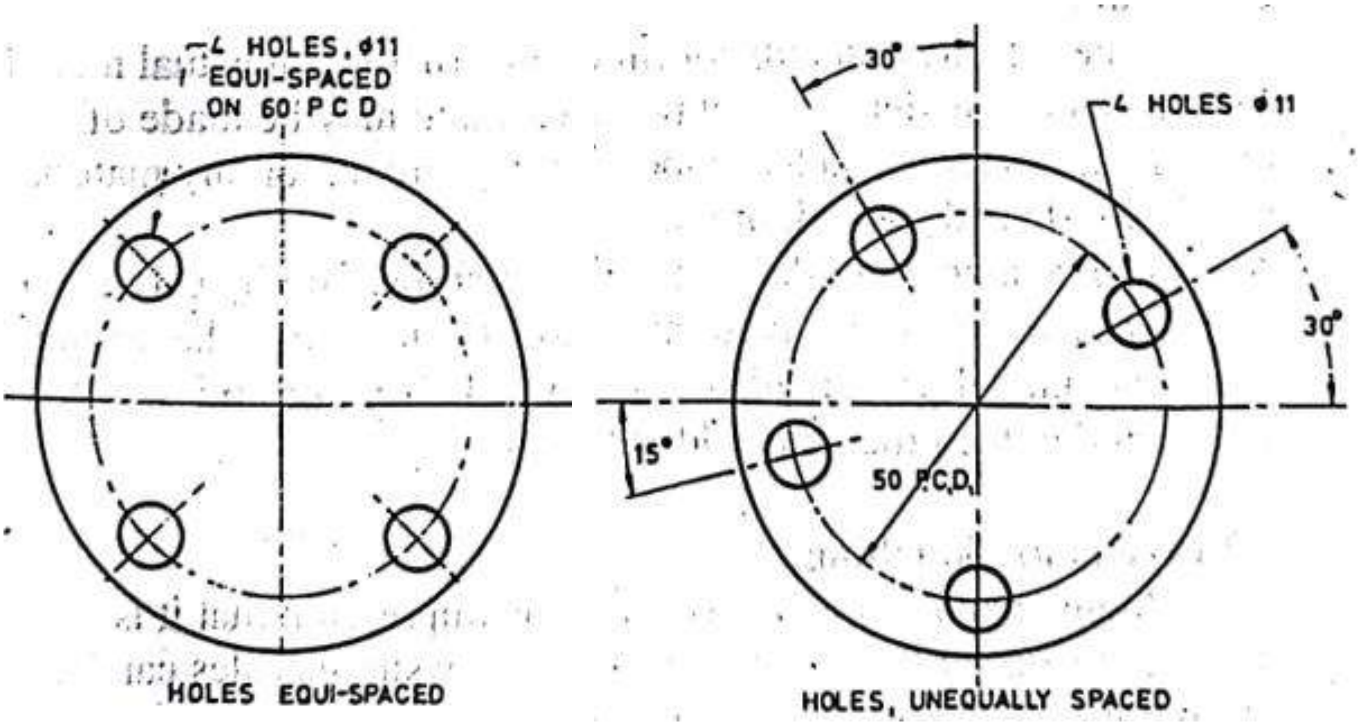

Fig. 7.13

7.16 The Missing View

Constructing a third projection from two given projections is a basic step in making and reading drawings. To successfully solve the required exercise, you have to read the given two views and look for their dimensions to draw an area for the third one as a first step to find the missing view as plan or side view.

Follow the following method to deduce the missing view without the need to imagine the isometric of the object.

7.16.1 Deducing the plan

To find the plan form given elevation and side view, follow the following steps

1- Draw the maximum three views area as in fig. 7.14

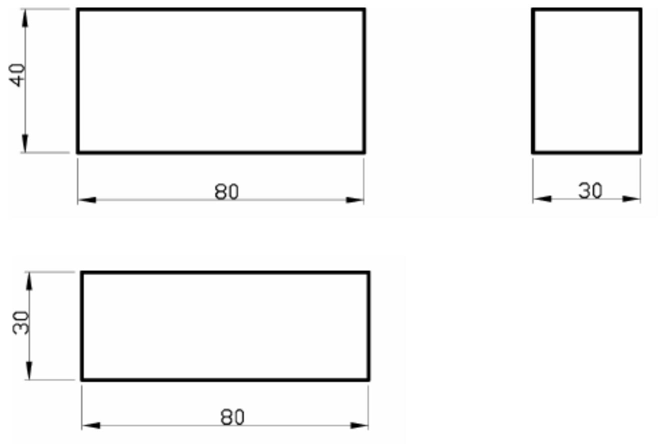

Fig.7. 14 The maximum area for the three views

This step provides you with the **area** of the missed view and its **position** according to the other two given views, as shown in fig.7. 15

2- Draw the given two views to start deducing the third one, fig. 7.16
3- Find the first horizontal plane dimensions from the elevation and side view. This plane is represented as a horizontal line in the elevation with length (L1) and width (W1), it will

be positioned as the arrows direction shown in fig.7. 17

4- Repeat the processes again for the next horizontal line to draw the second plane in the missing view, fig.7. 18

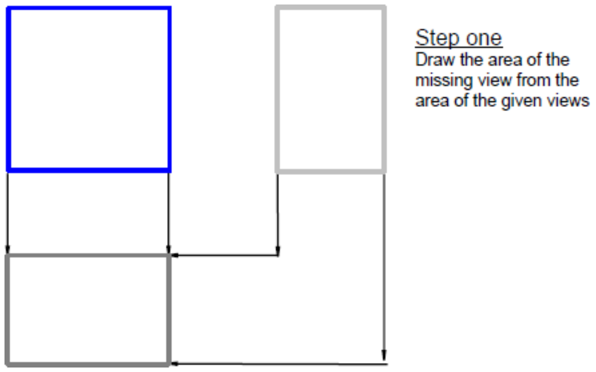

Fig.7.15 Area and position for missing plan

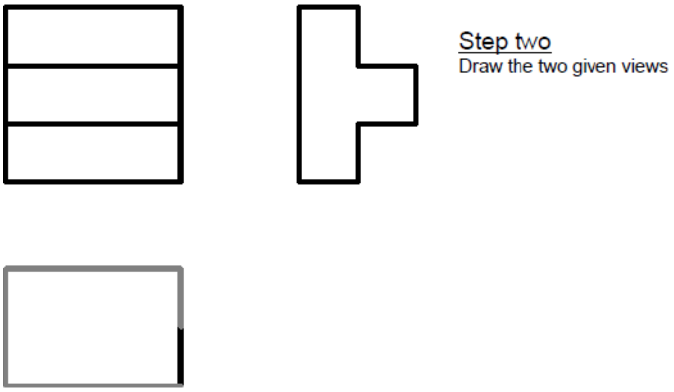

Fig.7.16 Elevation and side view drawing step

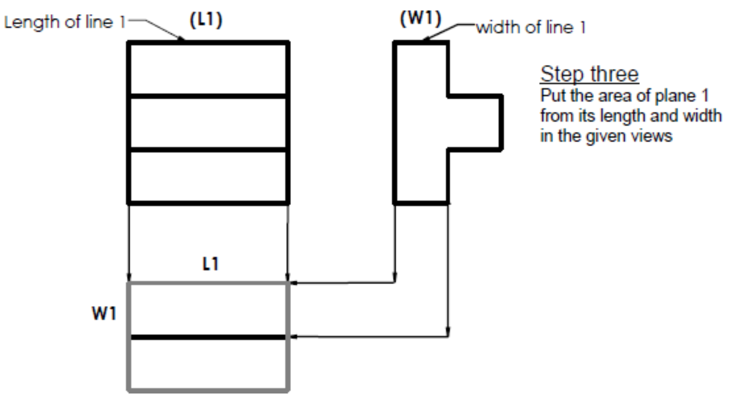

Fig.7. 17 Draw the first horizontal plan

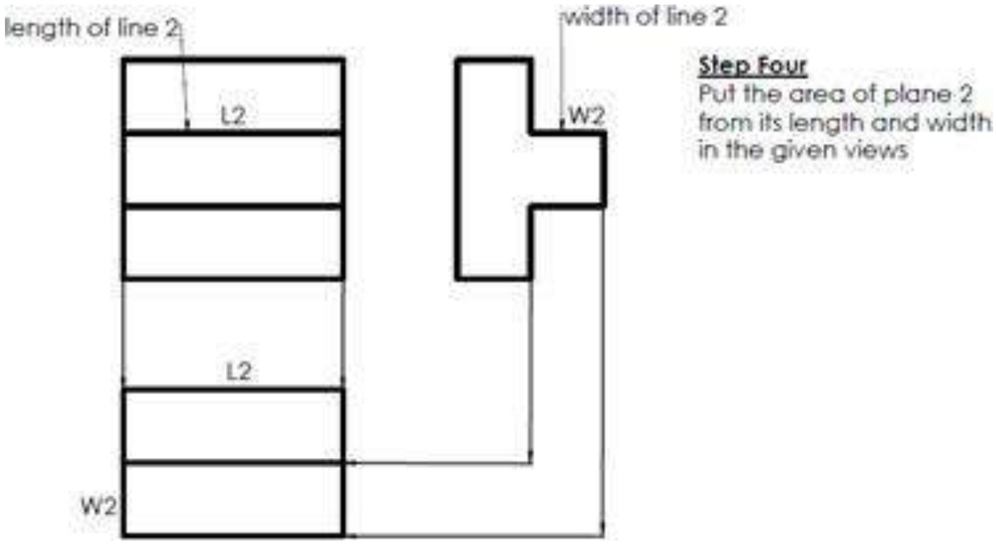

Fig.7. 18 Draw the second horizontal plan

Step three can be repeated several times for all horizontal lines in elevation and side view.

7.16.2 Deducing the side view

To find side view from given elevation and plan, follow the following steps

1- Draw the maximum three views area as in figure7. 19
2- Draw the given two views to start deducing the third one, fig.7.20
3- Find the first vertical plane dimensions from the elevation and plan.
   This plane is represented as a vertical line in the elevation with high (H1) and plan with width (W1), it will be positioned as the arrows direction shown in fig.7. 21
4- Repeat the processes again for the next vertical line to draw the second vertical plane in the missing view, fig.7. 22

Step three can be repeated a number of times for all vertical lines in elevation and plan, fig.7. 23

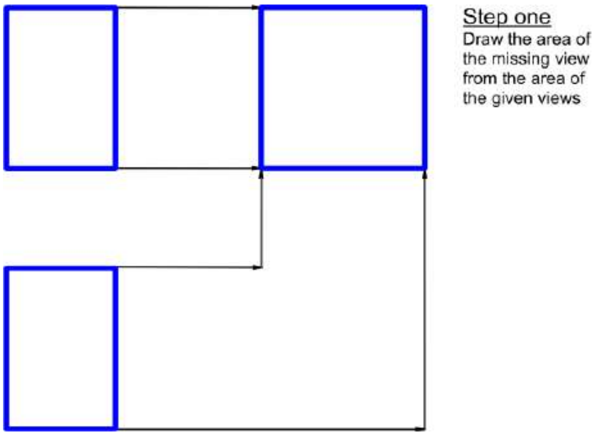

Fig.7. 19 Area and position for missing plan

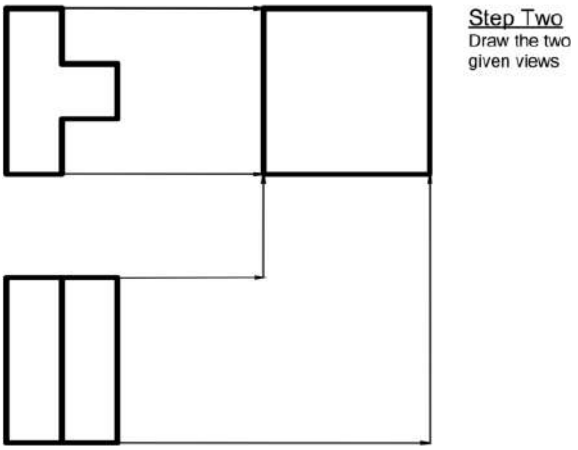

Fig.7. 20 Elevation and plan views drawing step.

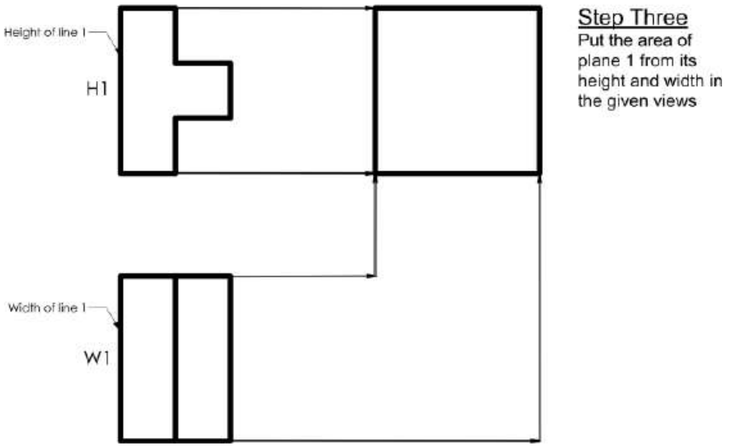

Fig.7. 21 Draw the first vertical plan

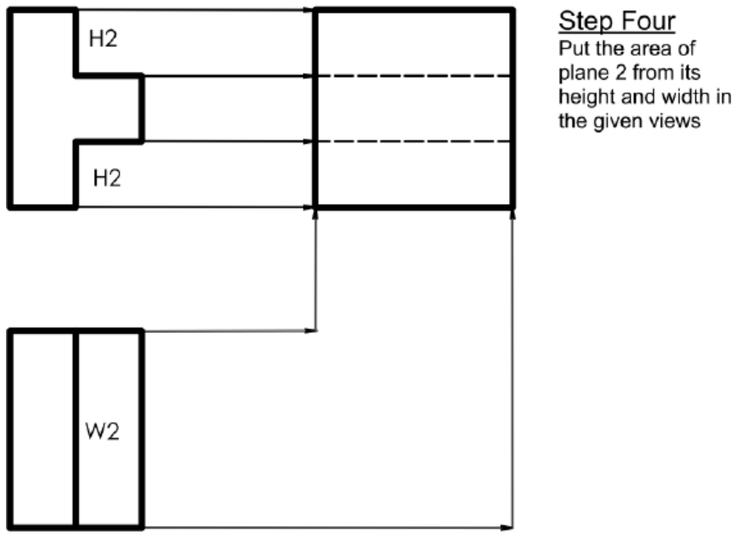

Fig.7.22 Draw the second horizontal plan

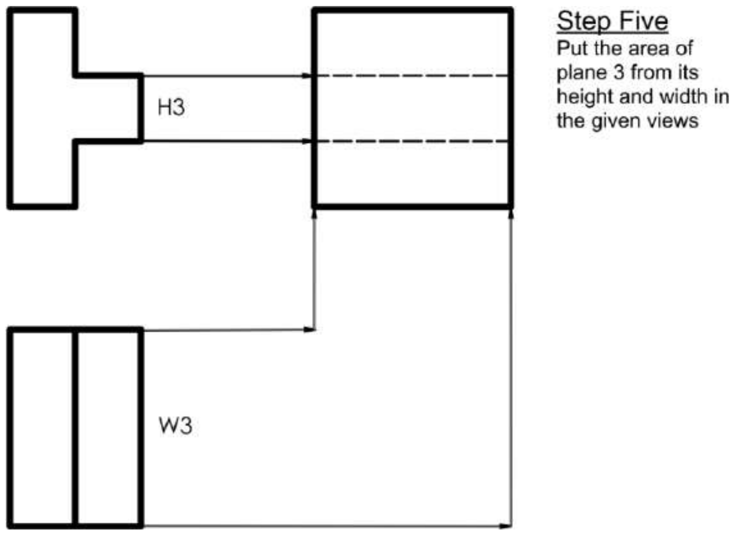

Fig.7. 23 Draw the third horizontal plan

7.16.3 Solved Examples

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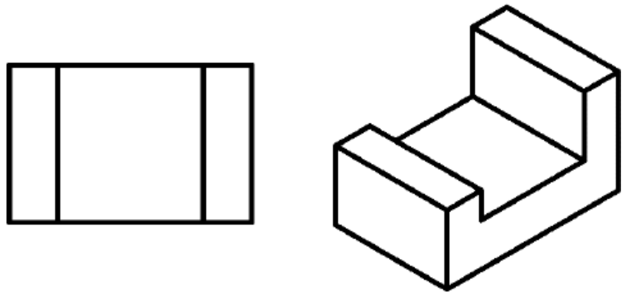

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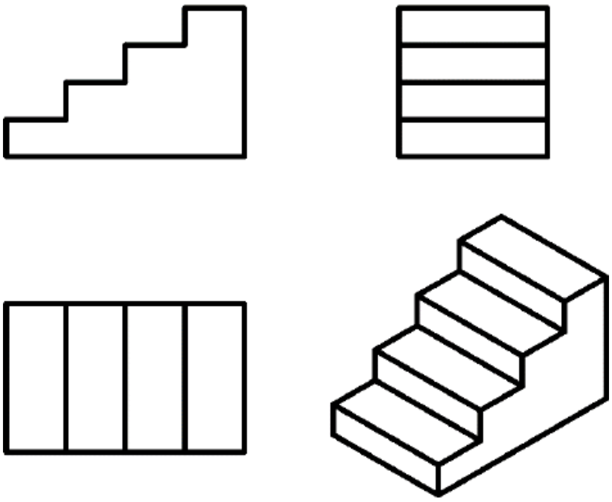

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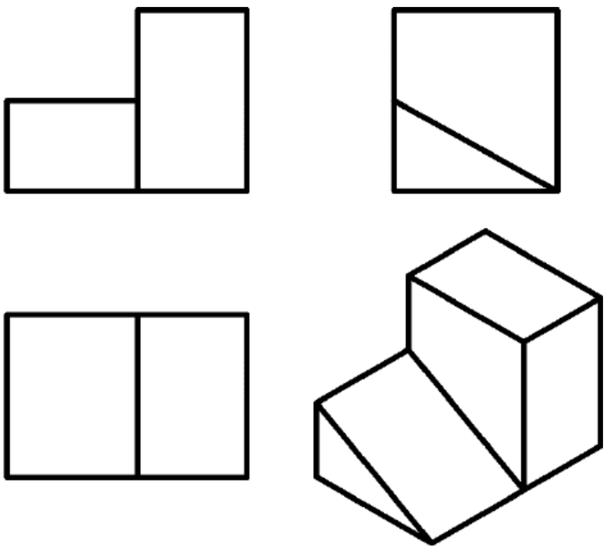

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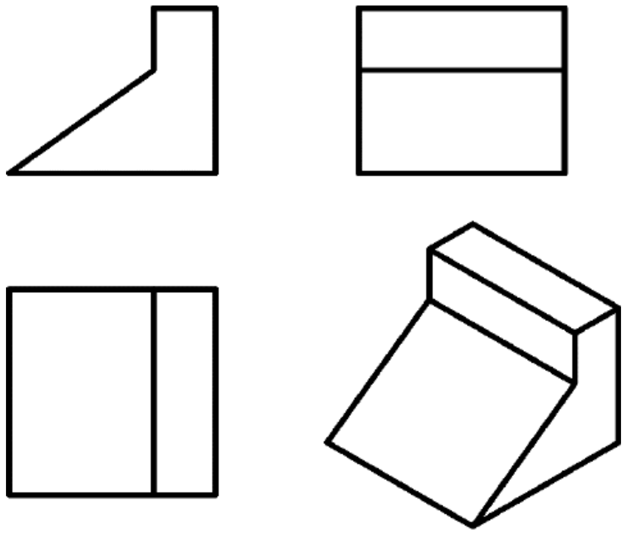

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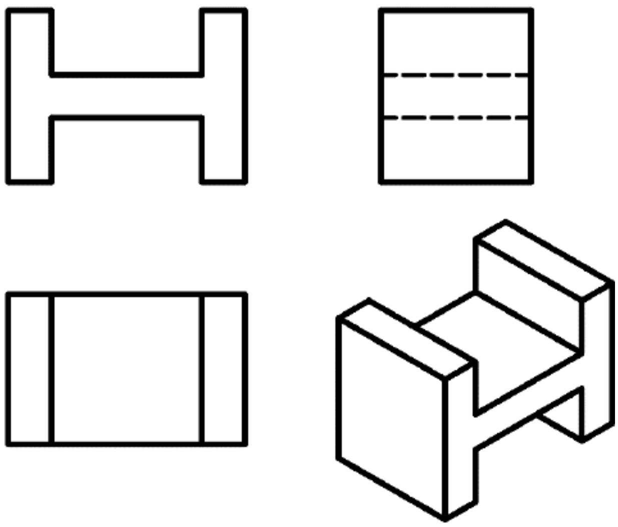

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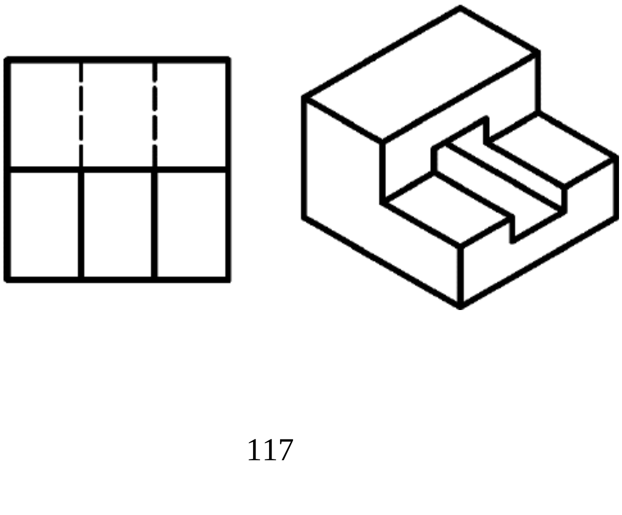
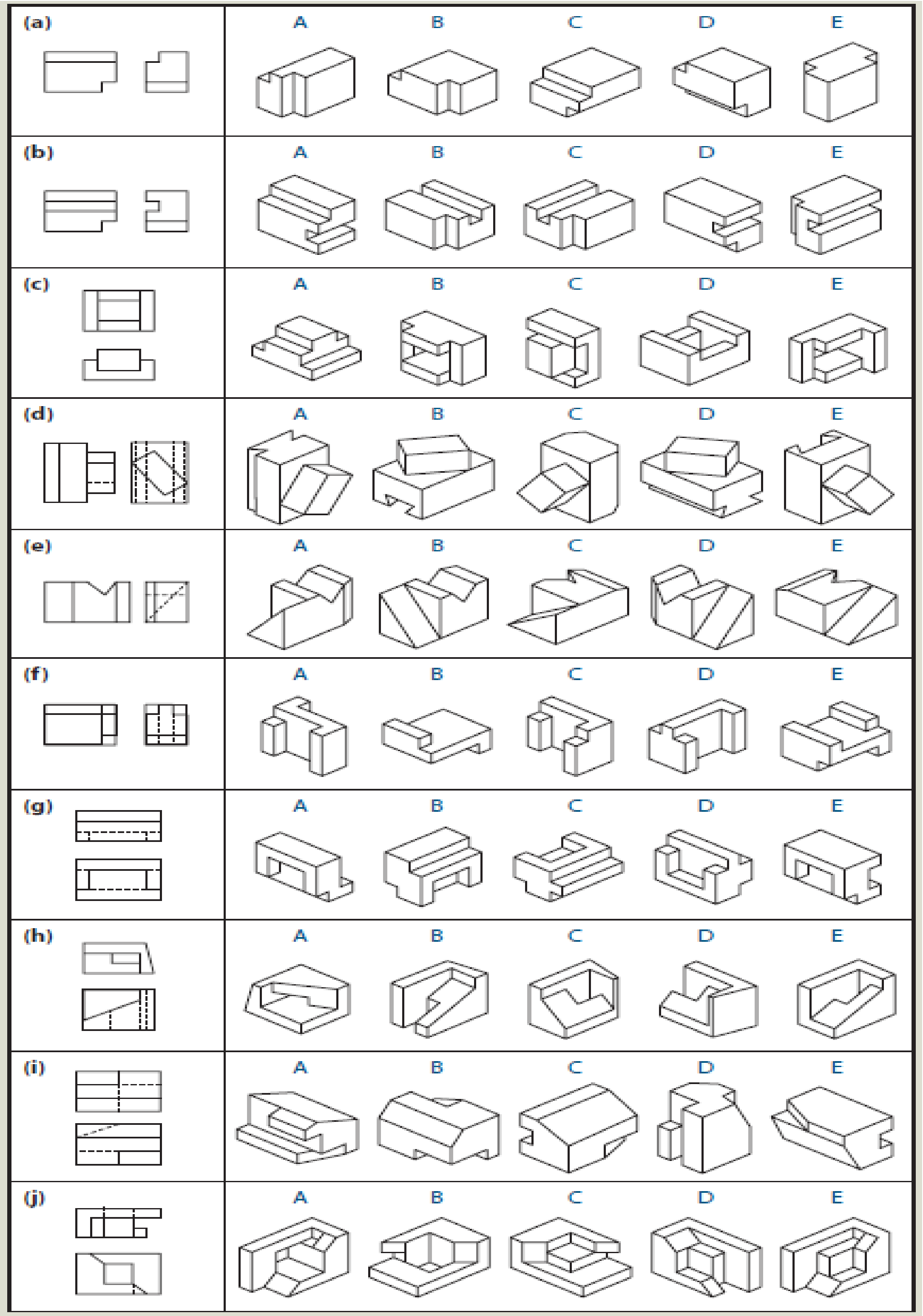
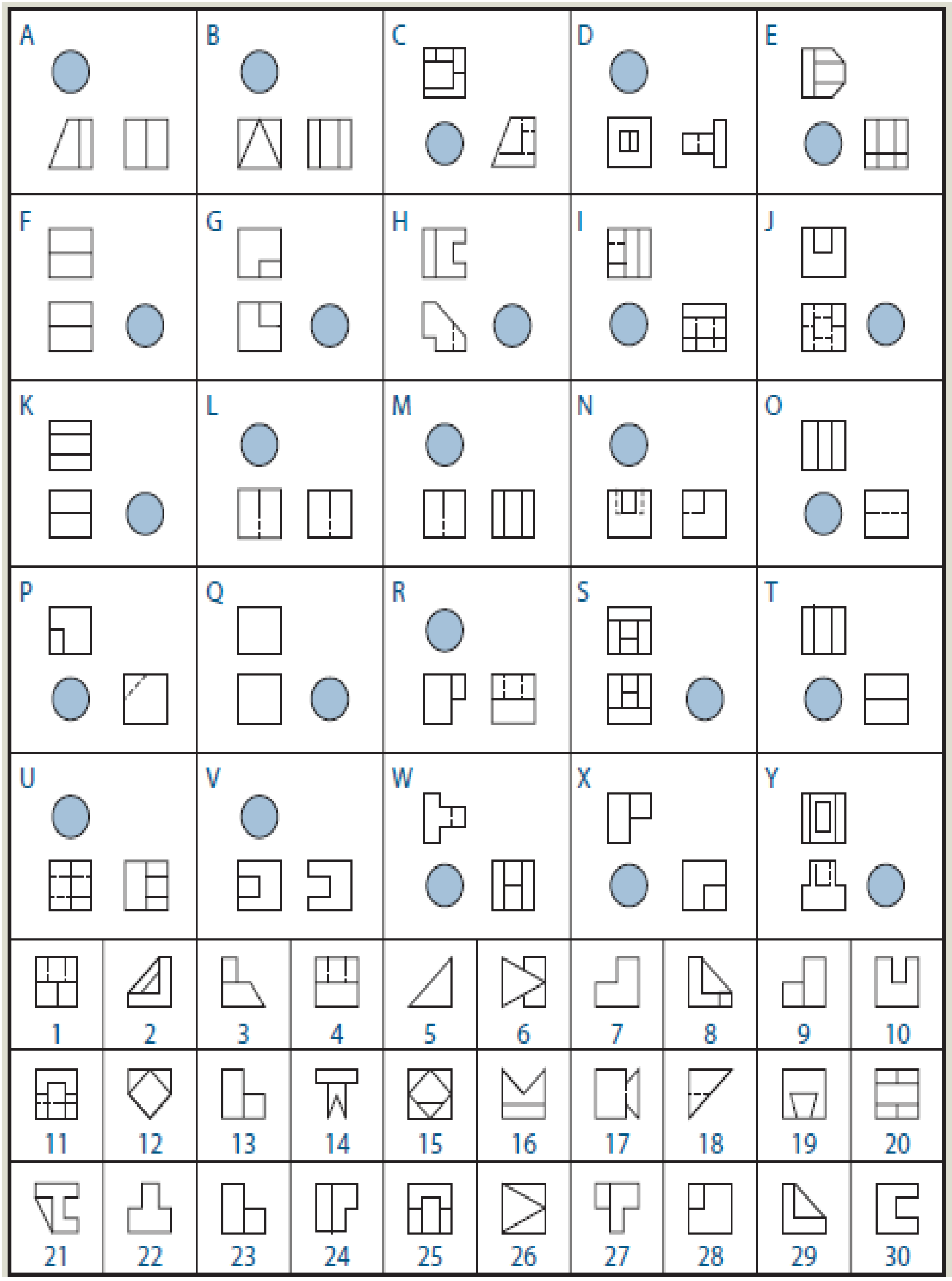


Exercise

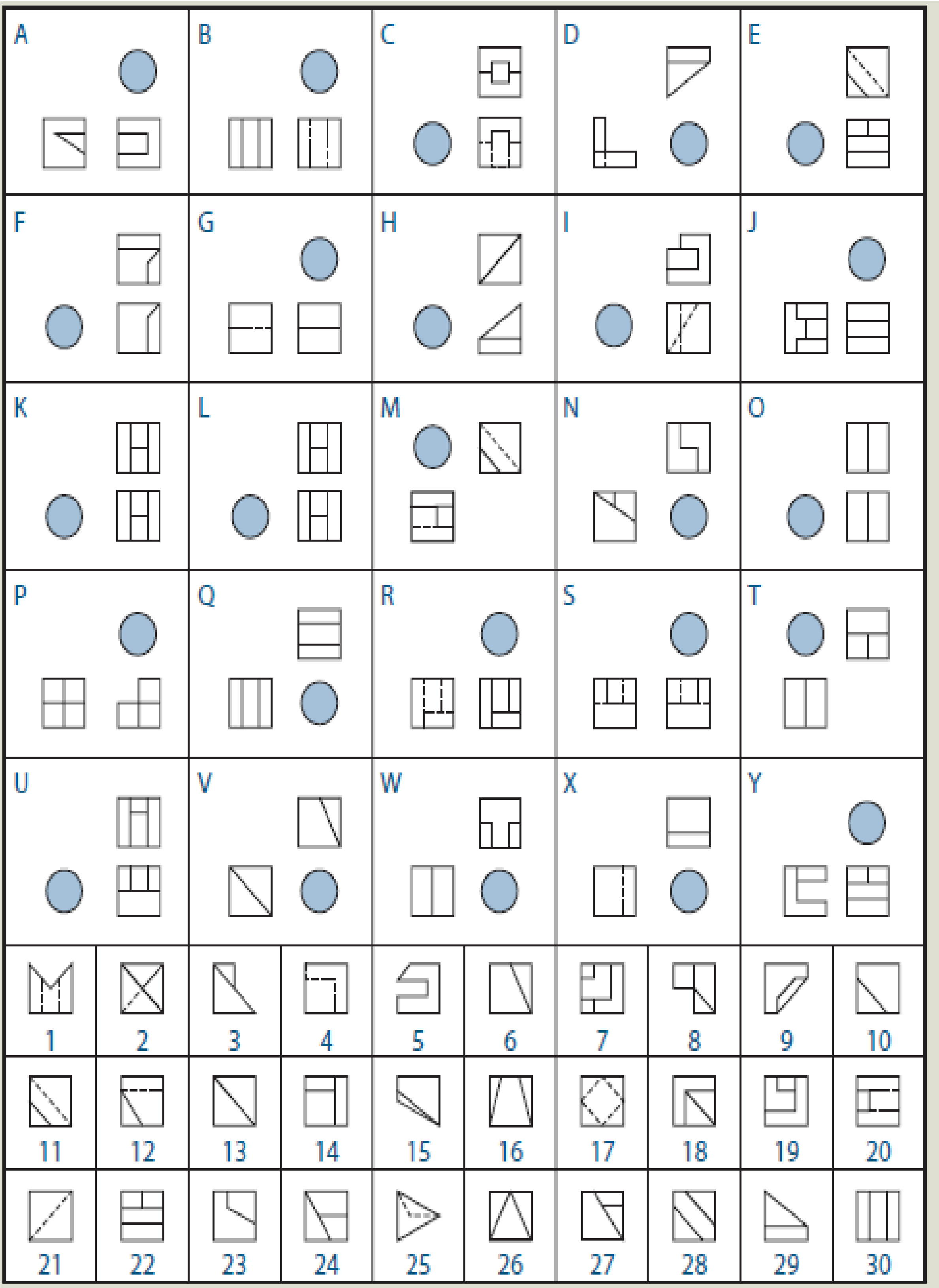
1- For each row shown in the following figure, select the suitable view of the object that will produce the orthographic views that are given.



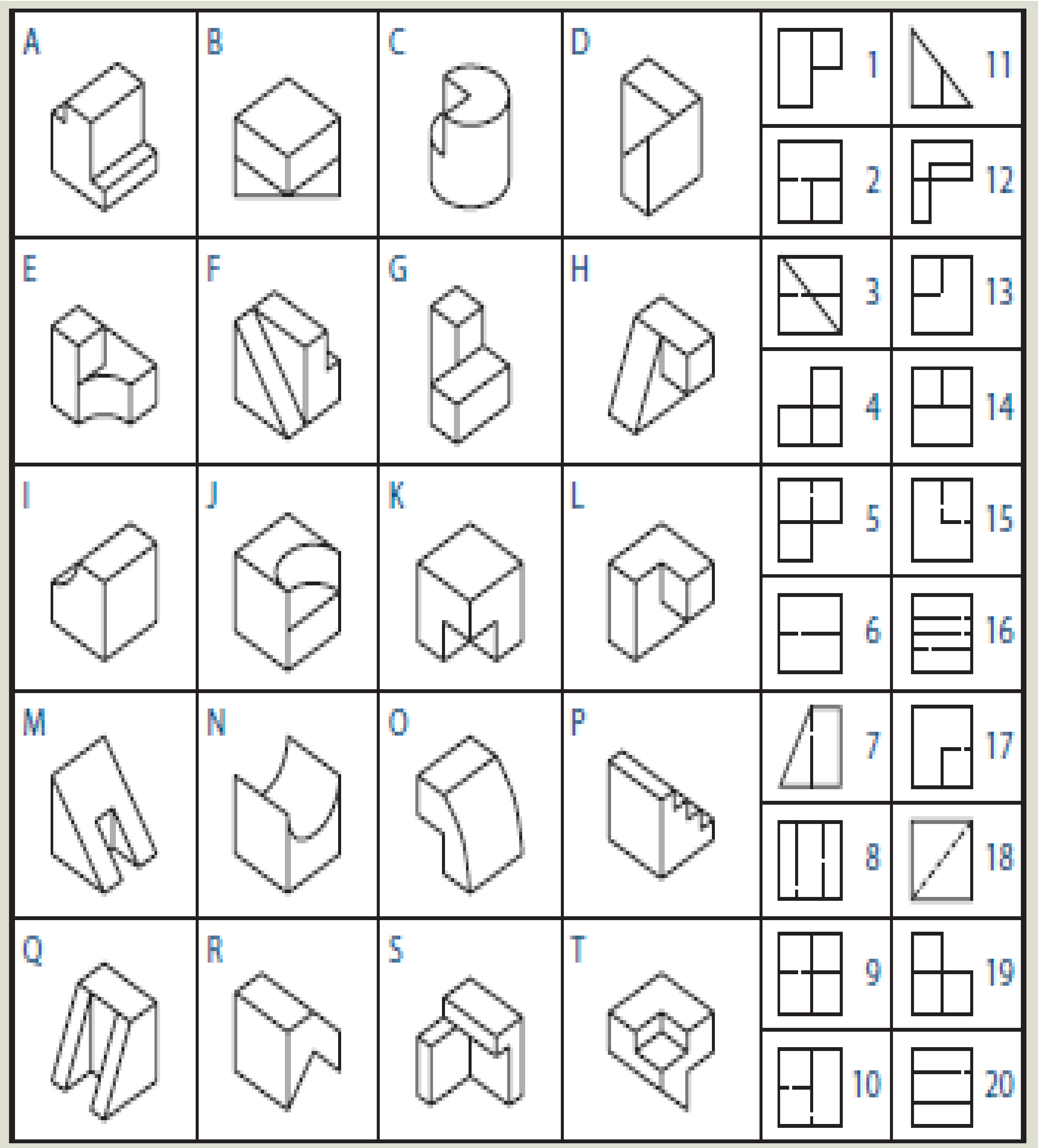
2- In each lettered cell shown in the following figure, the circle represents the location of a missing view. Select the correct view from the thirty views proposed. A view may be used more than once.



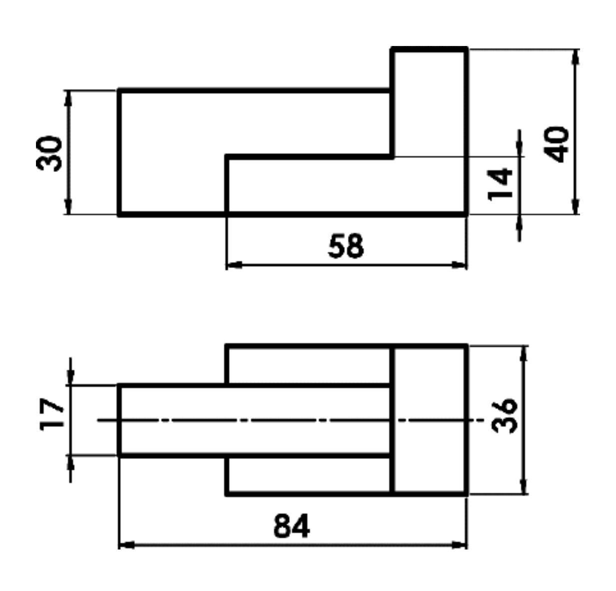
3- In each lettered cell shown in Figure P11.4, the circle represents the location of a missing view. Select the correct view from the thirty views proposed. A view may be used more than once.

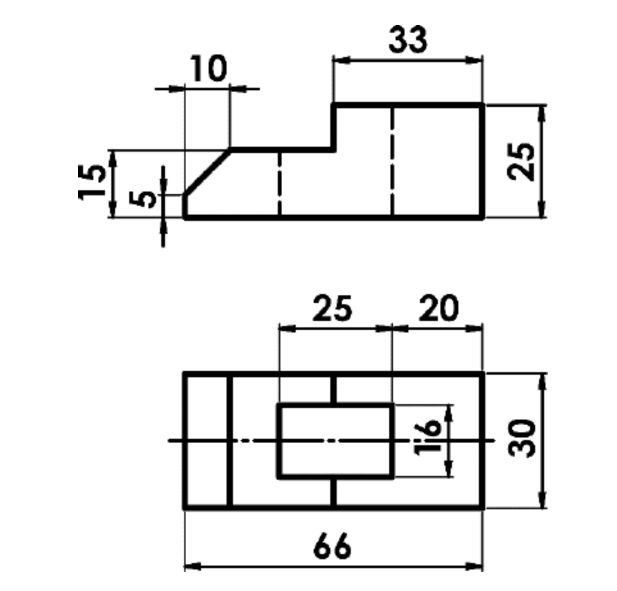


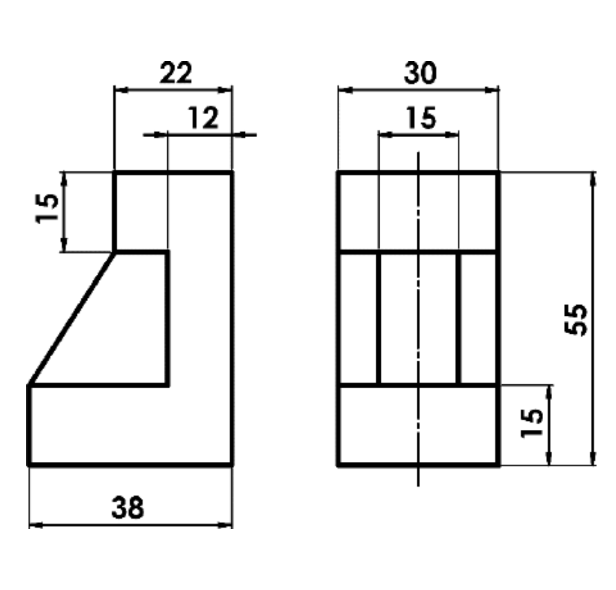
4- For the object shown in each lettered cell shown in the following figure, select one of the twenty views that correctly shows a top, front, or right-side view of the object having one or more hidden lines.

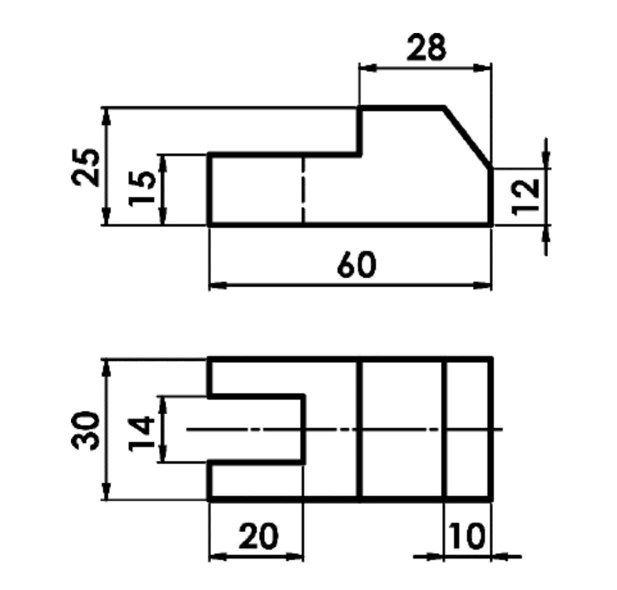


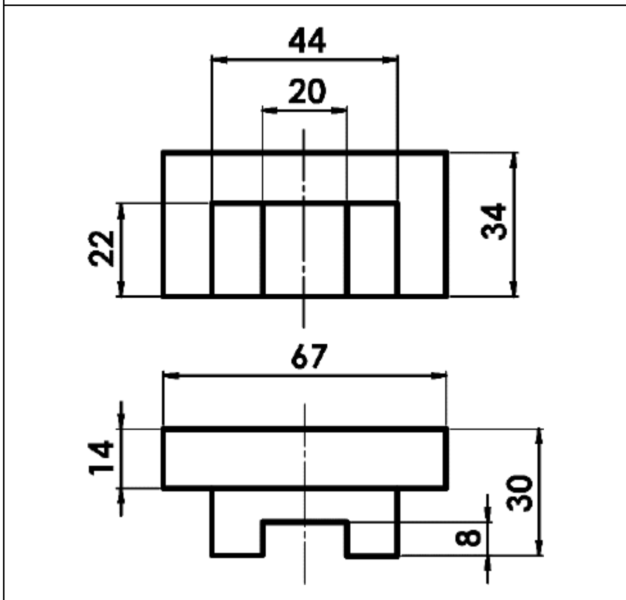
5- For the given views find the third view

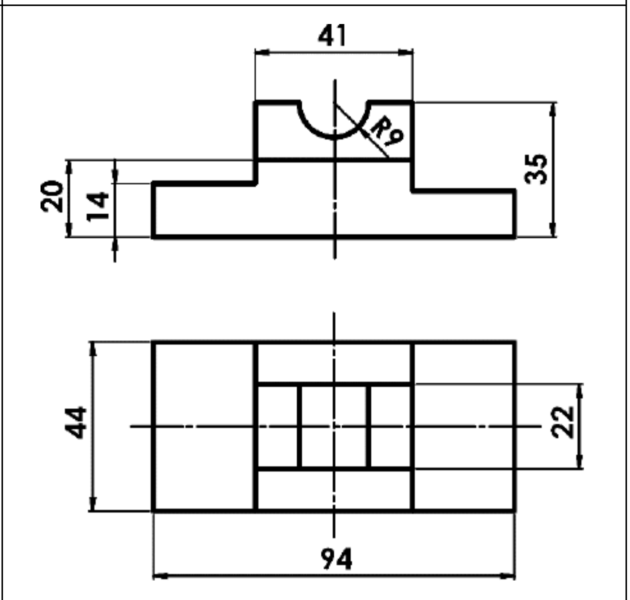


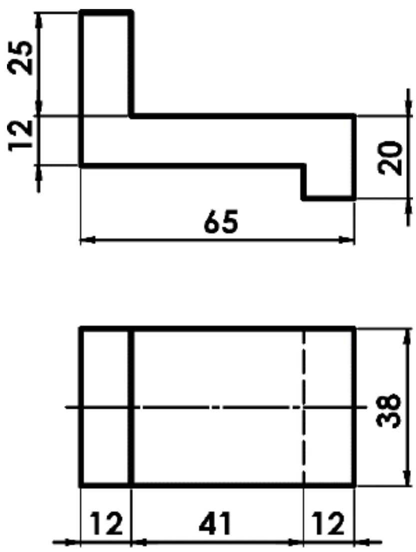








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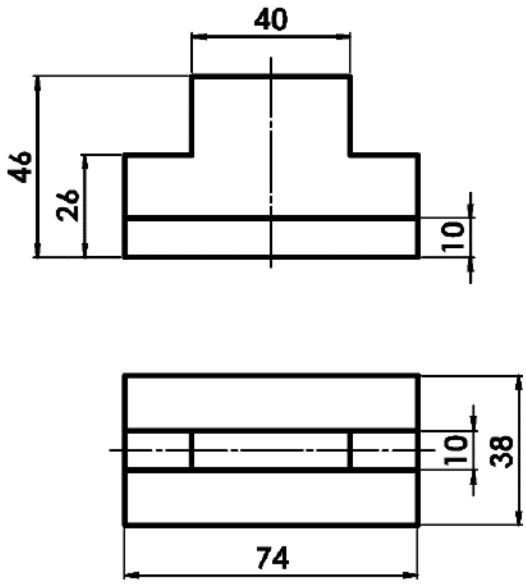
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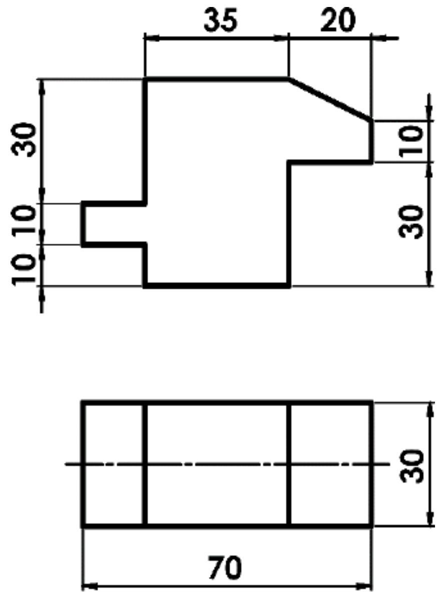
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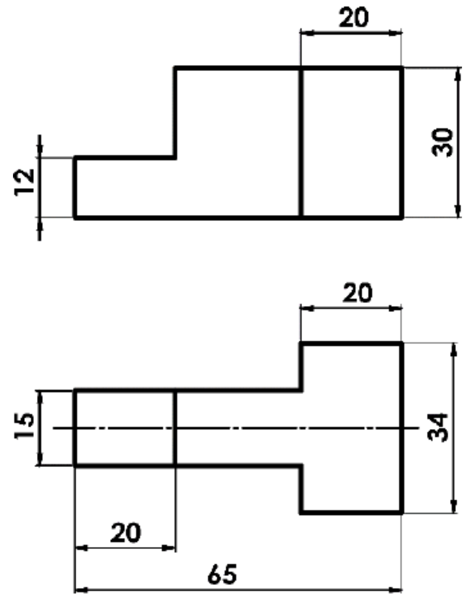
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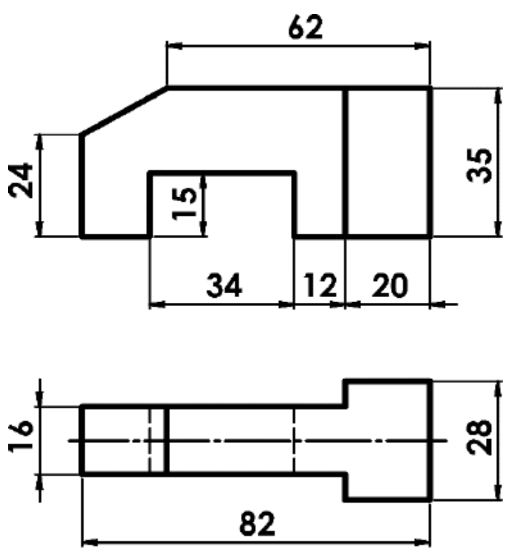
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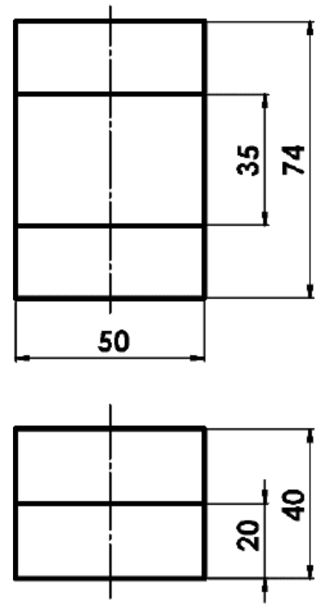
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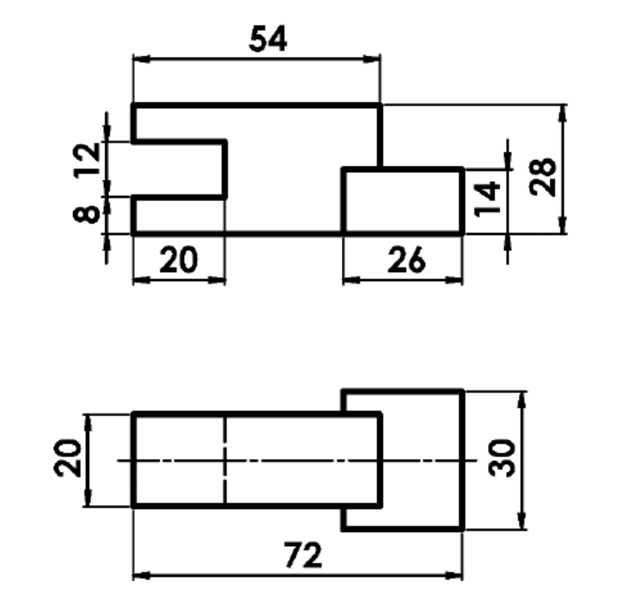
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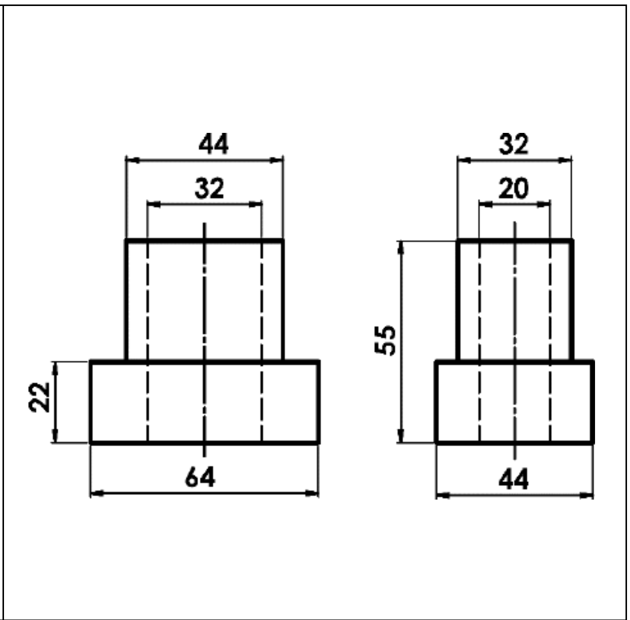
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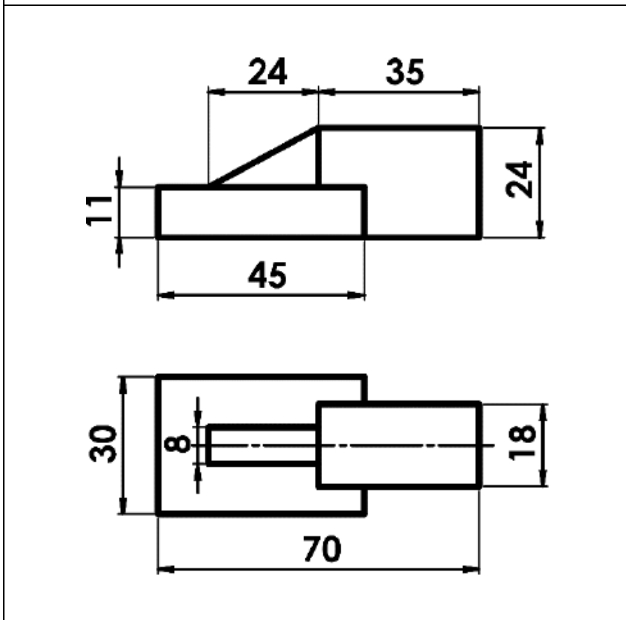
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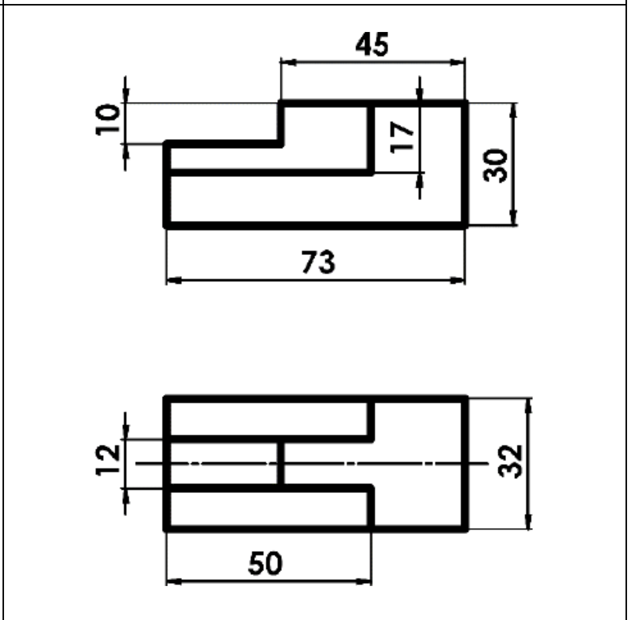
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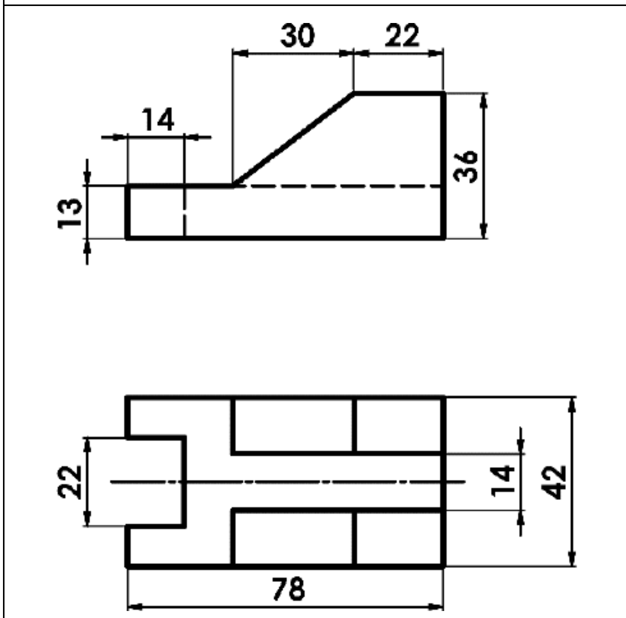
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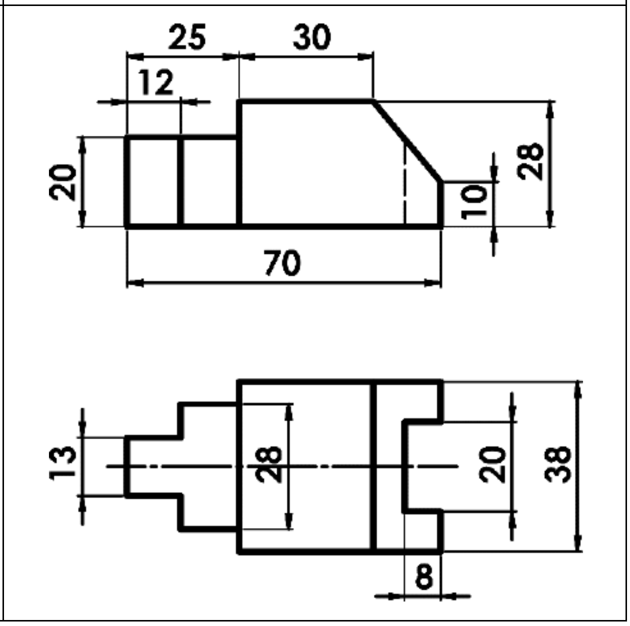
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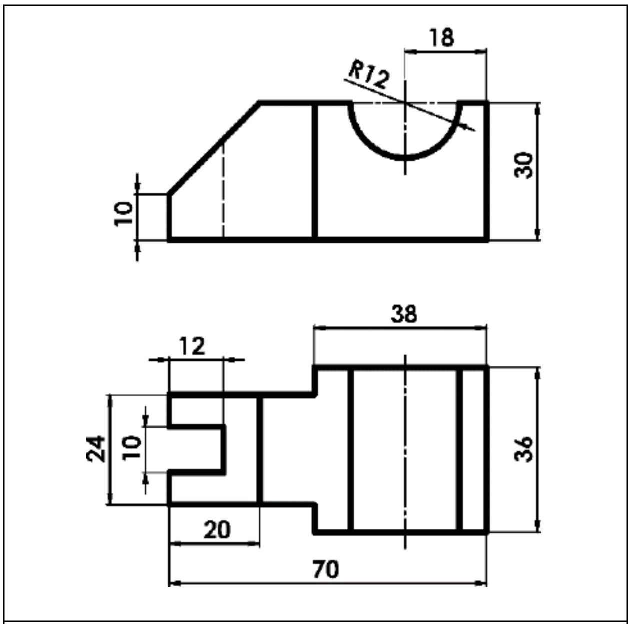
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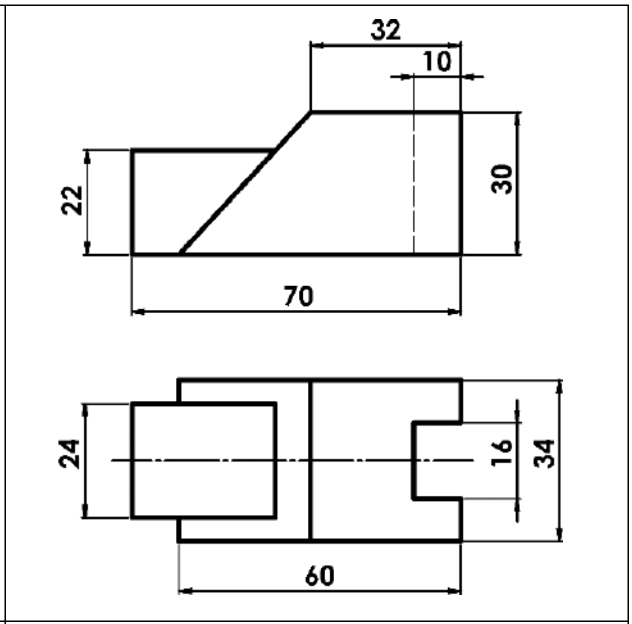
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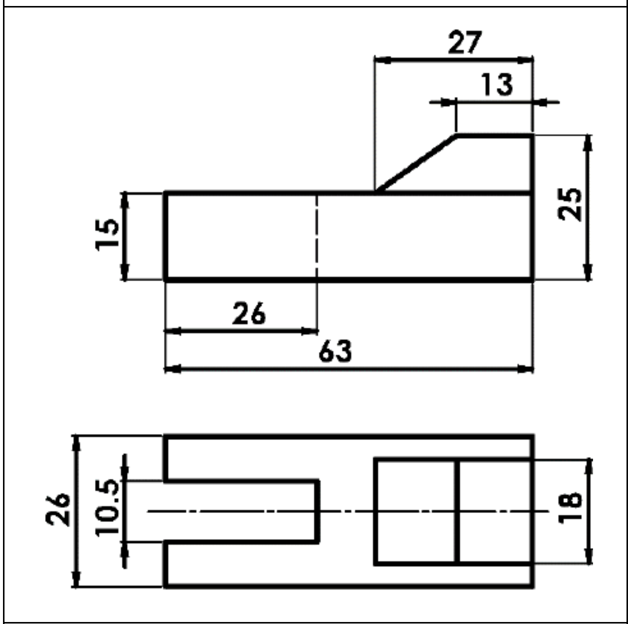
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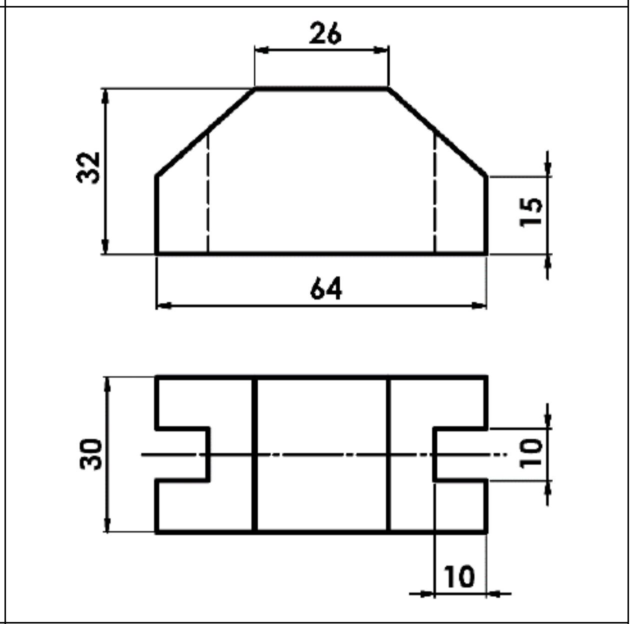
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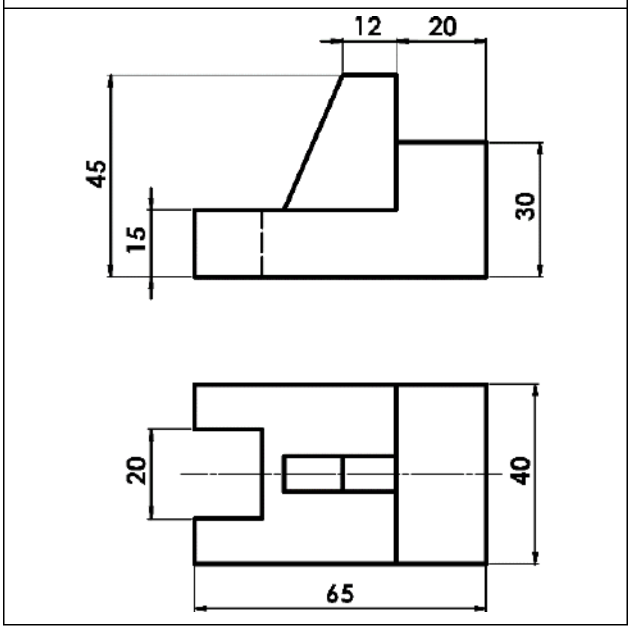
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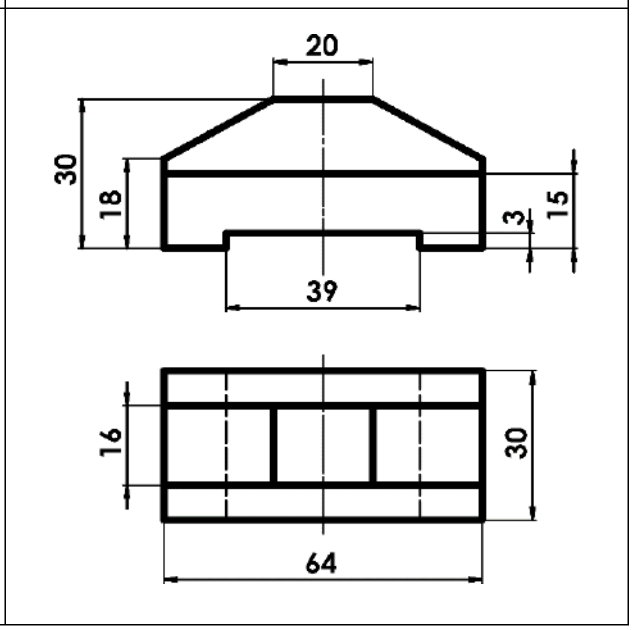
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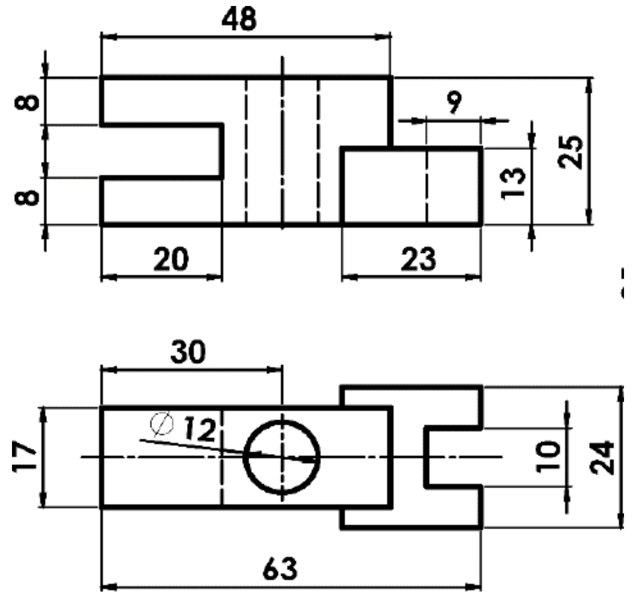
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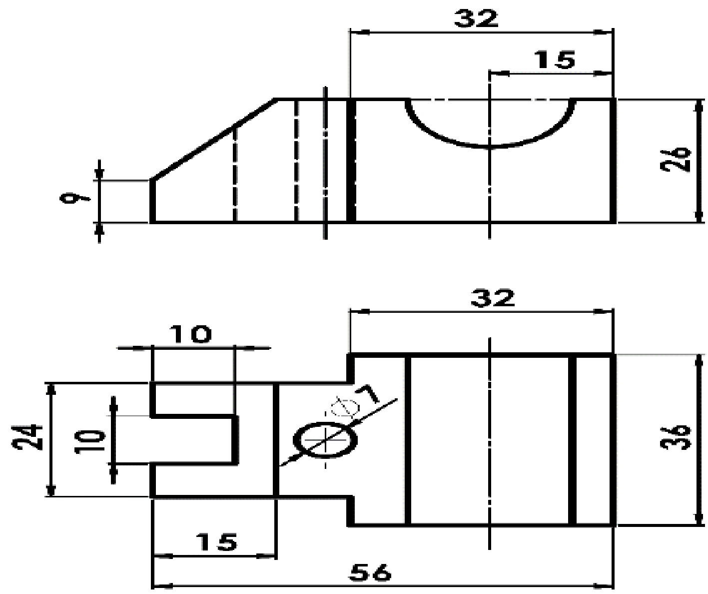
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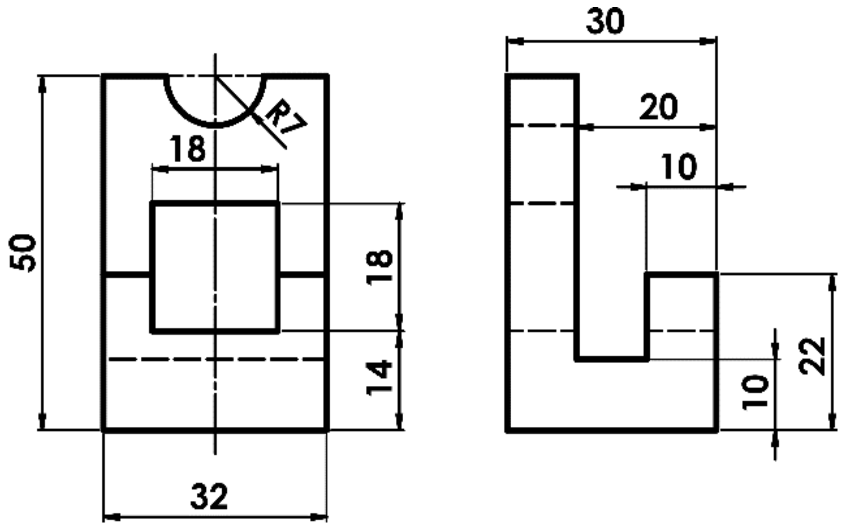
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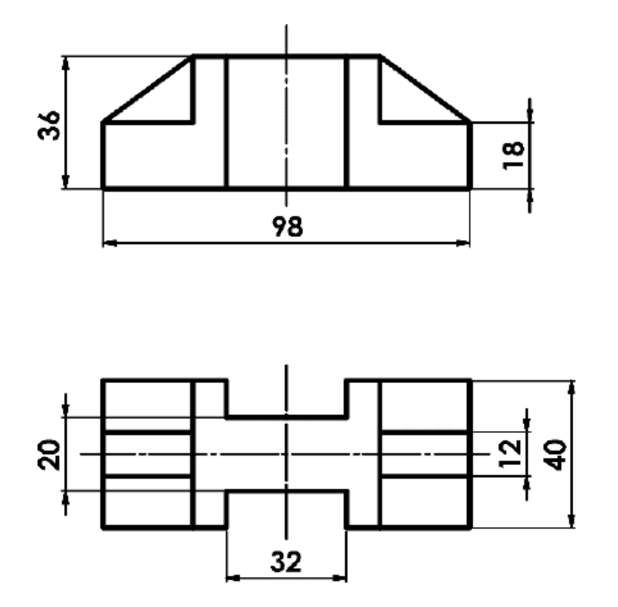
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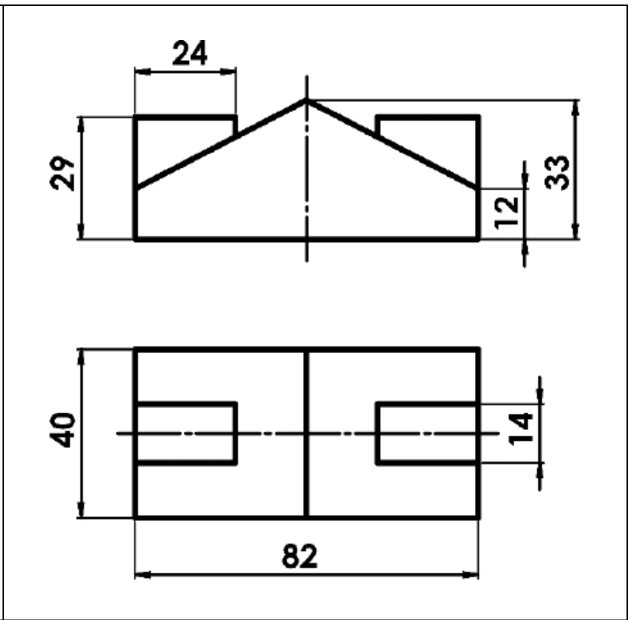
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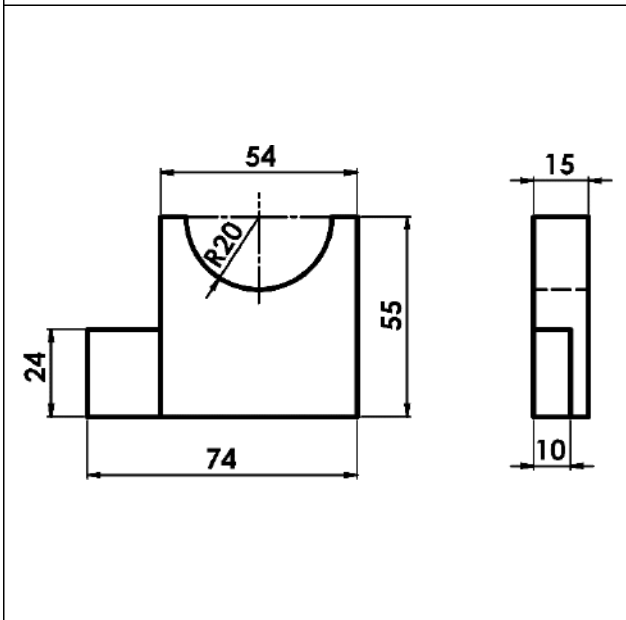
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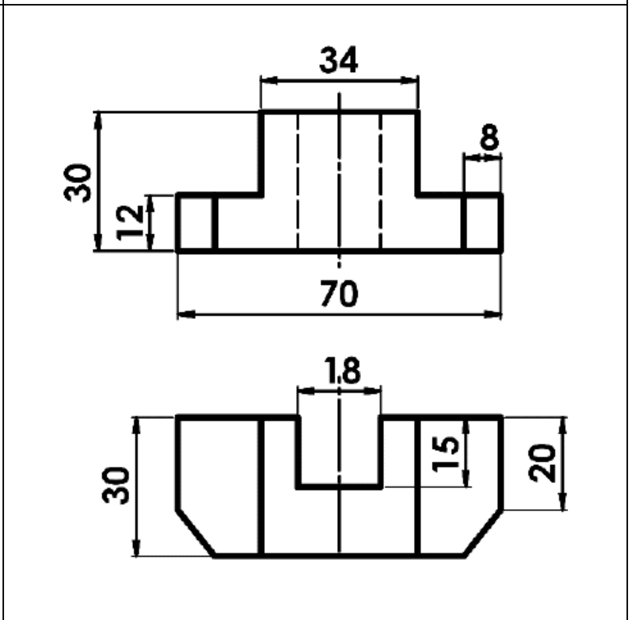
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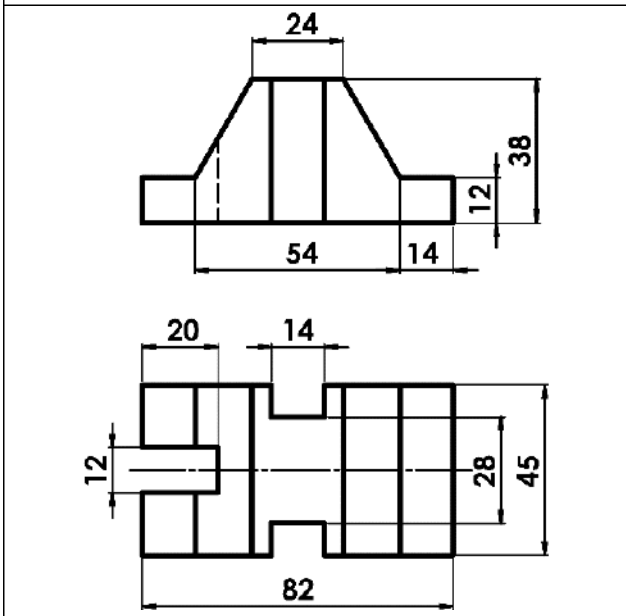
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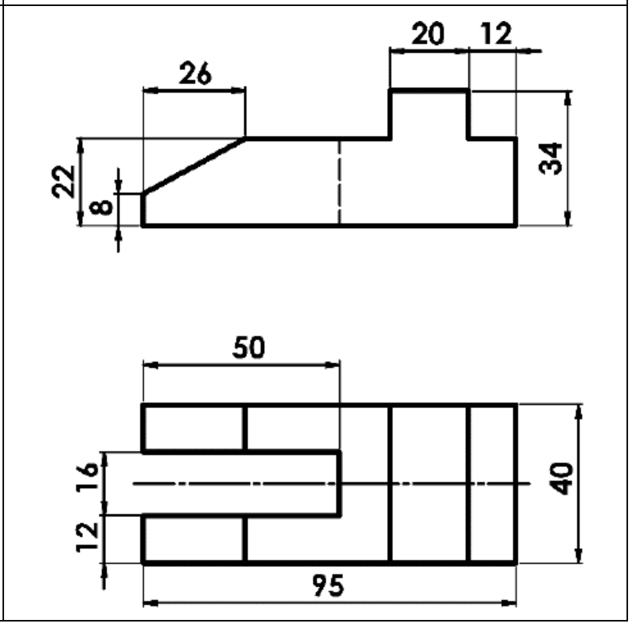
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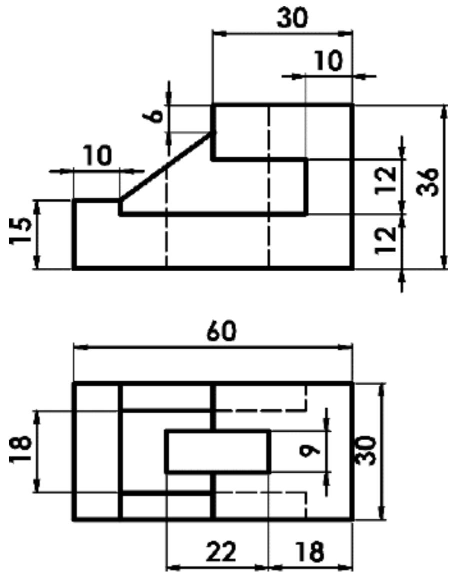
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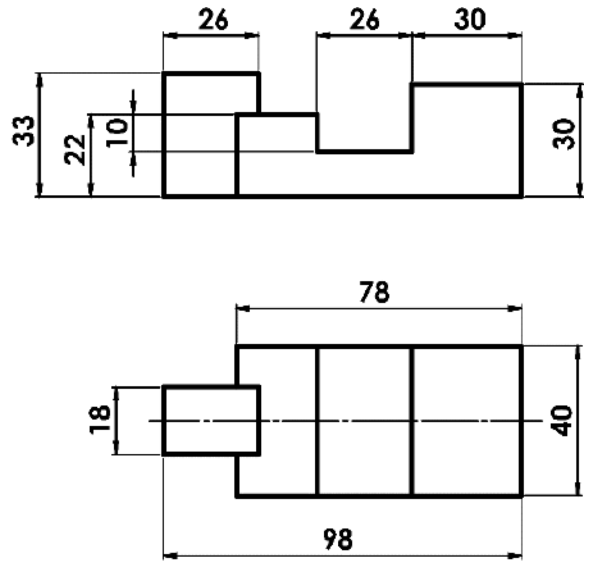
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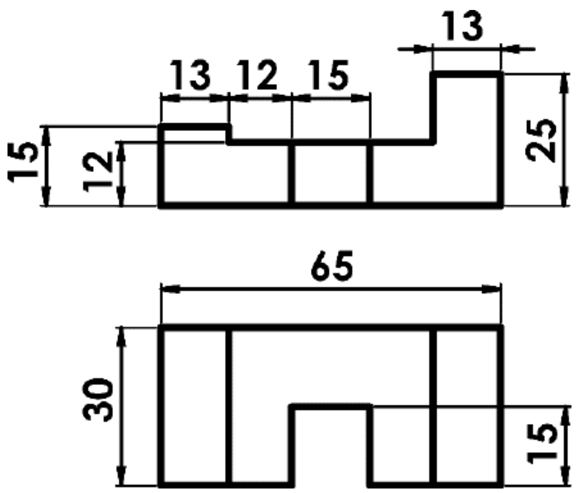
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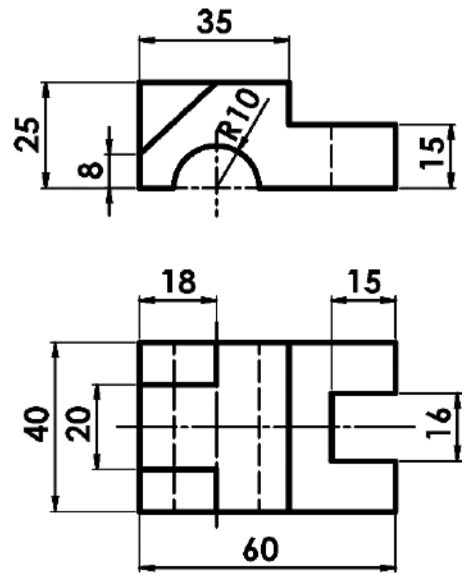
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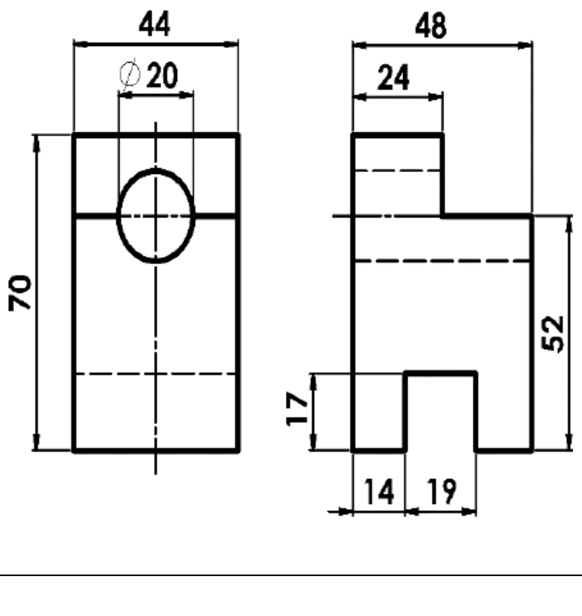
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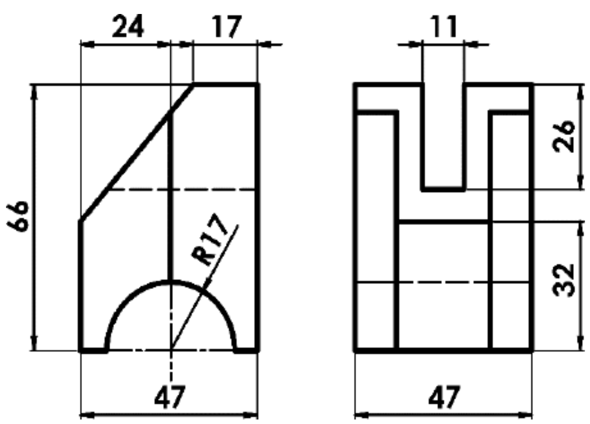
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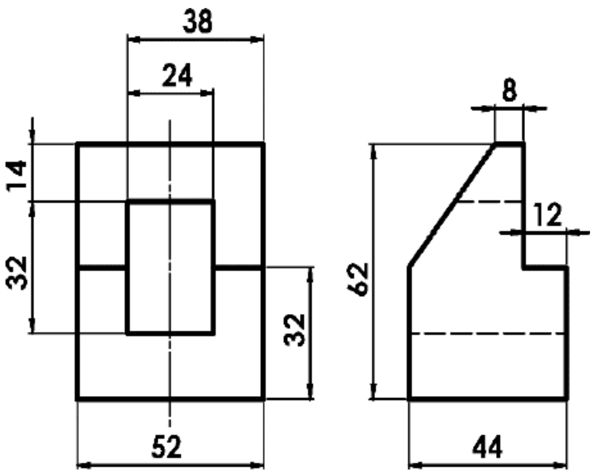
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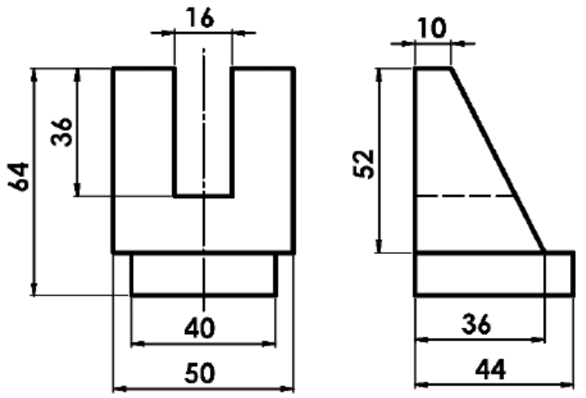
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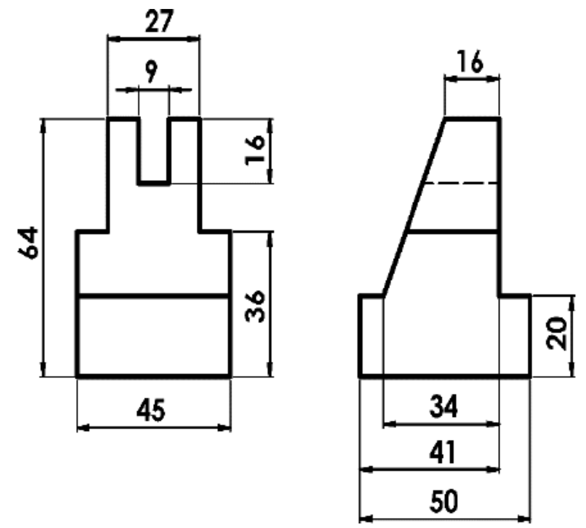
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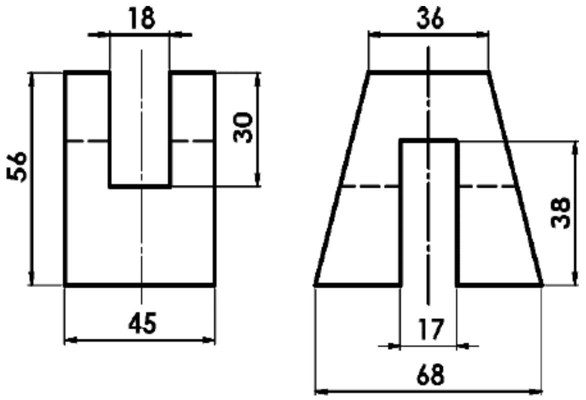
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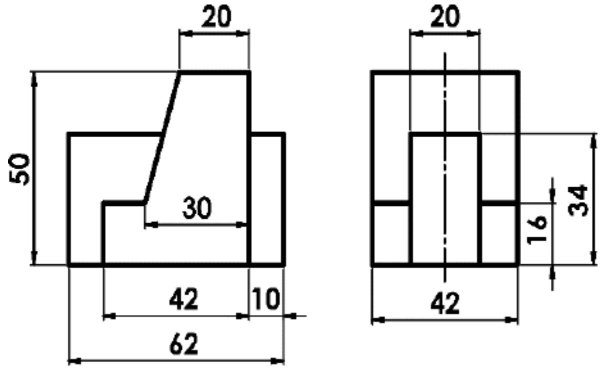
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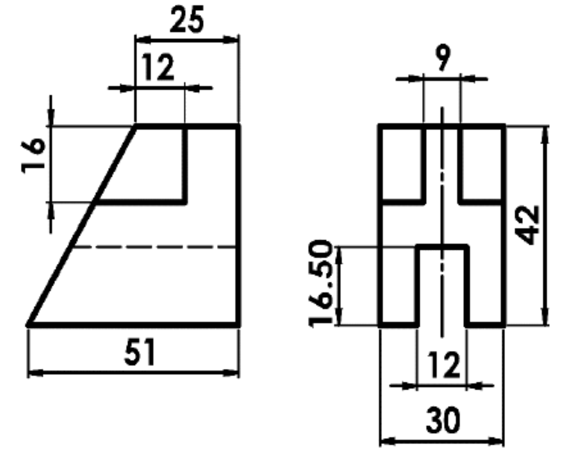
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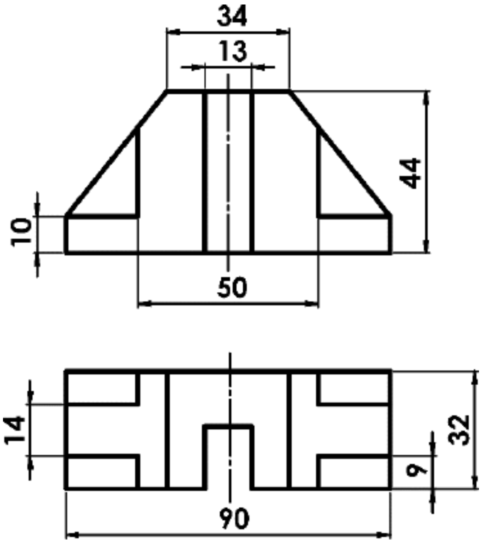
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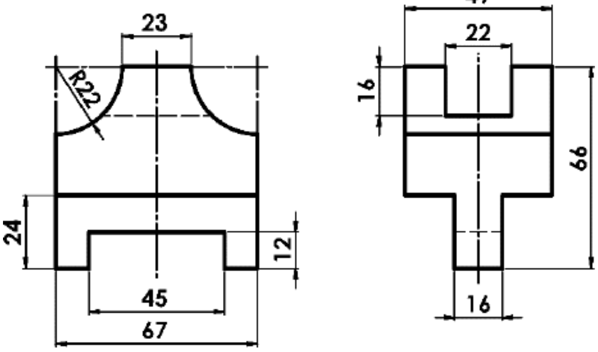
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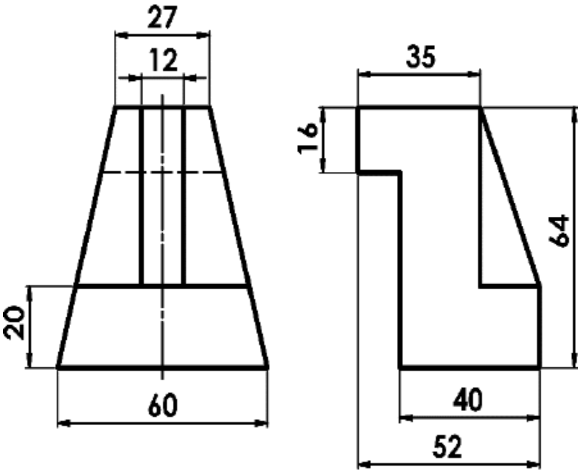
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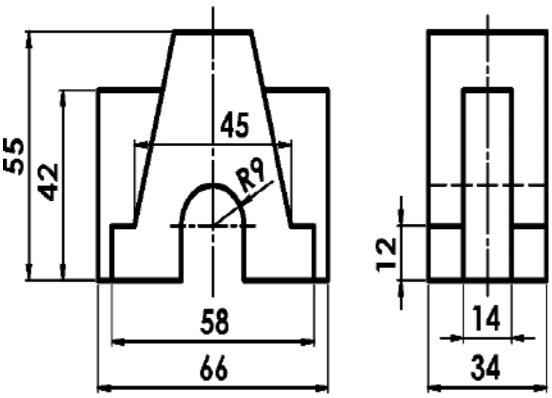
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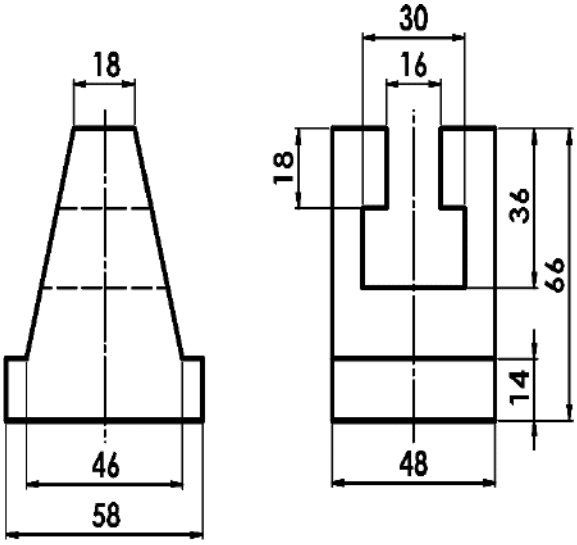
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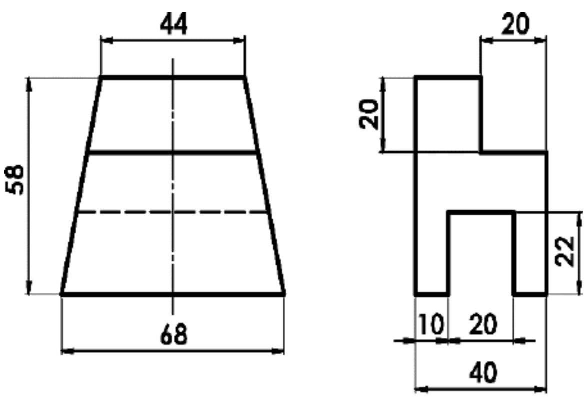
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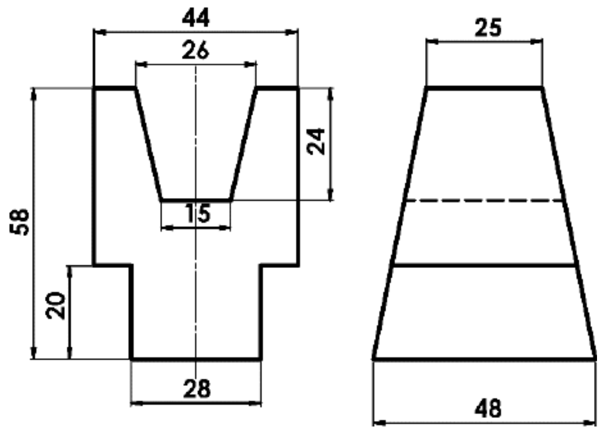
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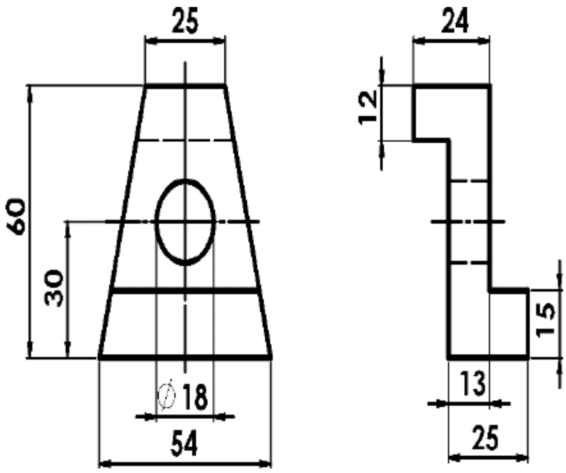
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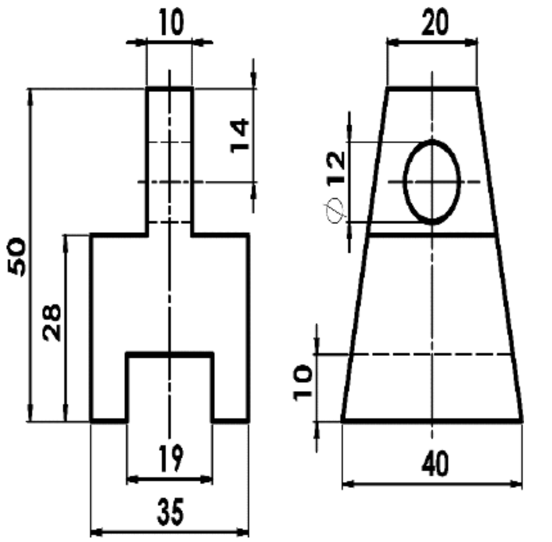
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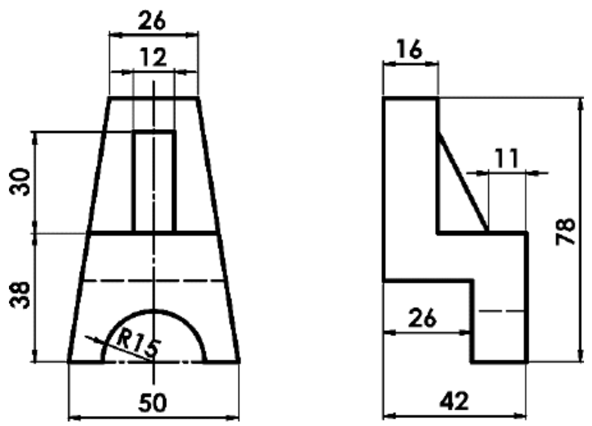
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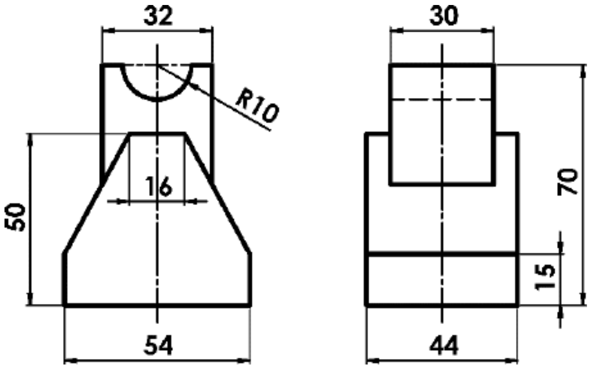
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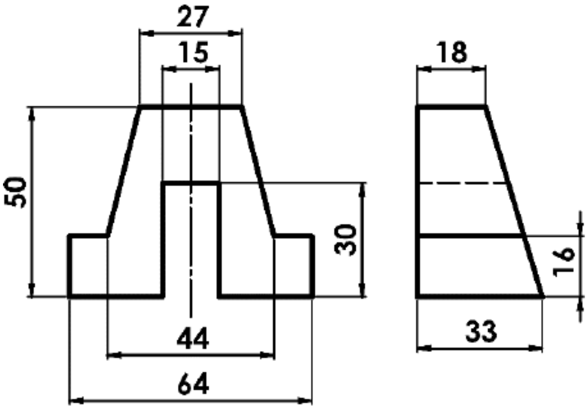
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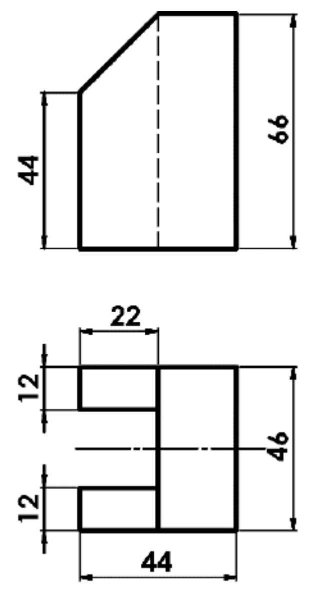
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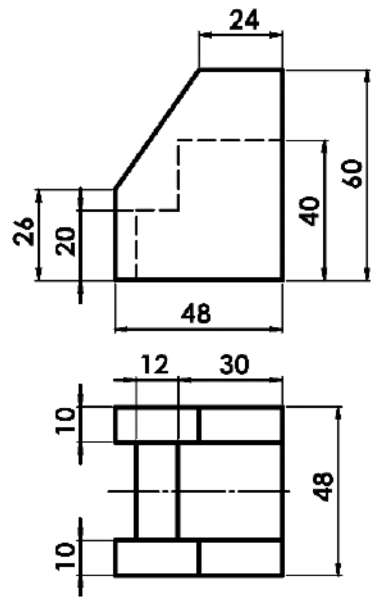
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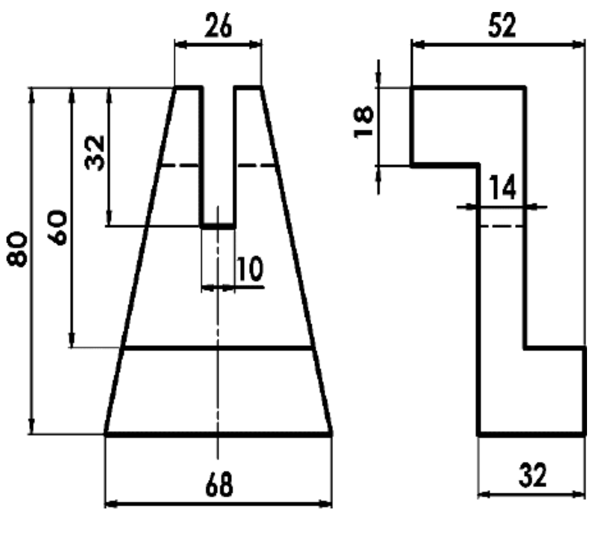
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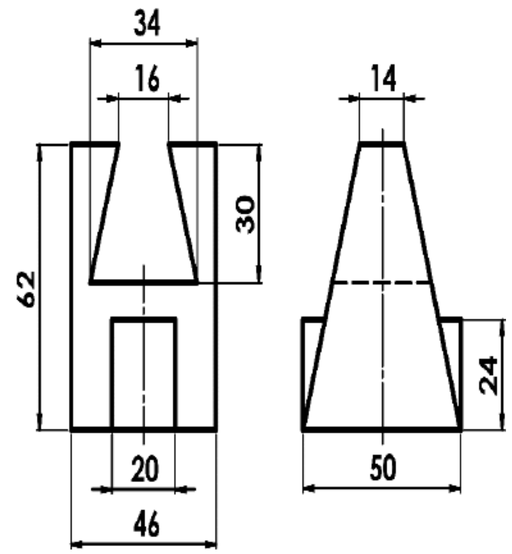
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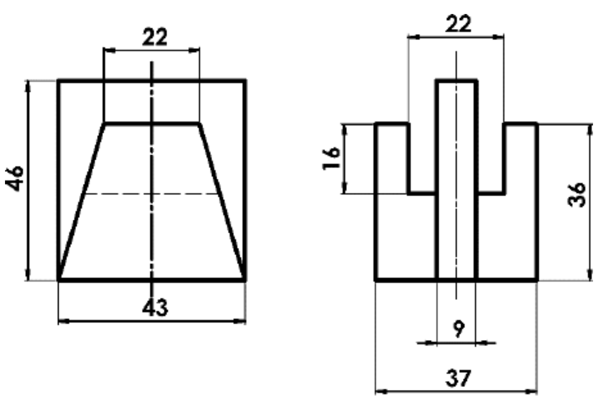
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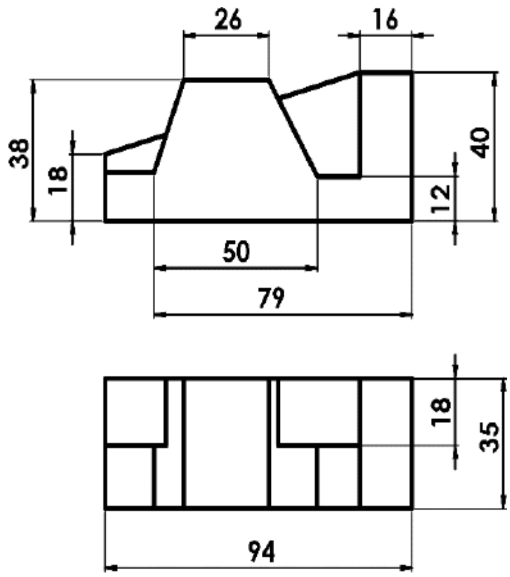
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Chapter 8

Computer Aided Drafting

The term Computer Aided Drafting (CAD) applies to a wide range of programs that allow the user to create drawings, plans electronically. AutoCAD® is one such program that main claim to fame. It is relatively easy to use and very comprehensive in its ability to create 2D and some 3D drawings. It is very popular, around seventy percent of the CAD users in the world use AutoCAD.

In previous chapters, traditional drawings were applied using essential drawing tools such as paper, pencils, drafter, compasses, eraser, scale etc., which in turn consumes more time and power especially in complex drawings. One of the most drawbacks with traditional drawing is information sharing. For example, if an engineer is drawing a design for a machine component and suddenly the manufacturer decided to modify some dimensions of innermost part of the component; in such situations one cannot modify the drawing already drawn, the component should be redrawn.

CAD is an electronic tool that enables to make quick and accurate drawings with the use of a computer. Drawings created with CAD have several advantages over drawings created on a drawing board. CAD drawings are neat, clean and highly presentable. Electronic drawings can be modified quite easily and can be presented in a variety of formats. There are hundreds of CAD programs available in the CAD industry today. Some are intended for general drawing work while others are focused on specific engineering applications. There are many programs used to do 2D drawings, 3D drawings, renderings, shadings, engineering calculations, space planning, structural design, piping layouts, plant design, project management, etc.

8.1 History of CAD

In 1883 Charles Barbage developed idea for computer. First CAD demonstration is given by Ivan Sutherland (1963). A year later, Rum produced the first commercial CAD system. Many changes have taken place since then, with the advancement of powerful computers, it is now possible to do all the designs using CAD including two-dimensional drawings, solid modeling, complex engineering analysis, production and manufacturing. New technologies are constantly invented which make this process quicker, more versatile and more powerful.

8.2 Advantages of CAD

(i) Detail drawings may be created more quickly and making changes is more efficient than correcting drawings drawn manually.
(ii) It allows different views of the same object and 3D pictorial view, which gives better visualization of drawings
(iii) Designs and symbols can be stored for easy recall and reuse.
(iv) By using the computer, the drawing can be produced with more accuracy.
(v) Drawings can be more conveniently filed, retrieved and transmitted on disks and tape.
(vi) Quick Design Analysis, also Simulation and Testing Possible.

In this chapter you will learn how to navigate through and create drawings within AutoCAD's 2-D drawing workspace. Entities within the AutoCAD® environment are created within a coordinate framework. Objects such as Lines, Circles, Polygons, etc.… are created using a set of defining coordinates points. Although this may sound complex, AutoCAD® makes creating professional looking engineering drawings very simple. By the end of this chapter you will be able to create and edit complex geometries.

8.3 Introduction to AutoCAD®

AutoCAD® allows you to visualize, document and share a design idea in either a 2-D or 3-D environment. This chapter will focus on AutoCAD's 2-D drawing capabilities. AutoCAD® gives you the tools to create accurate and professional looking detailed and assembly drawings. Figure 8.1 shows a detailed drawing created in AutoCAD's 2-D drawing workspace.

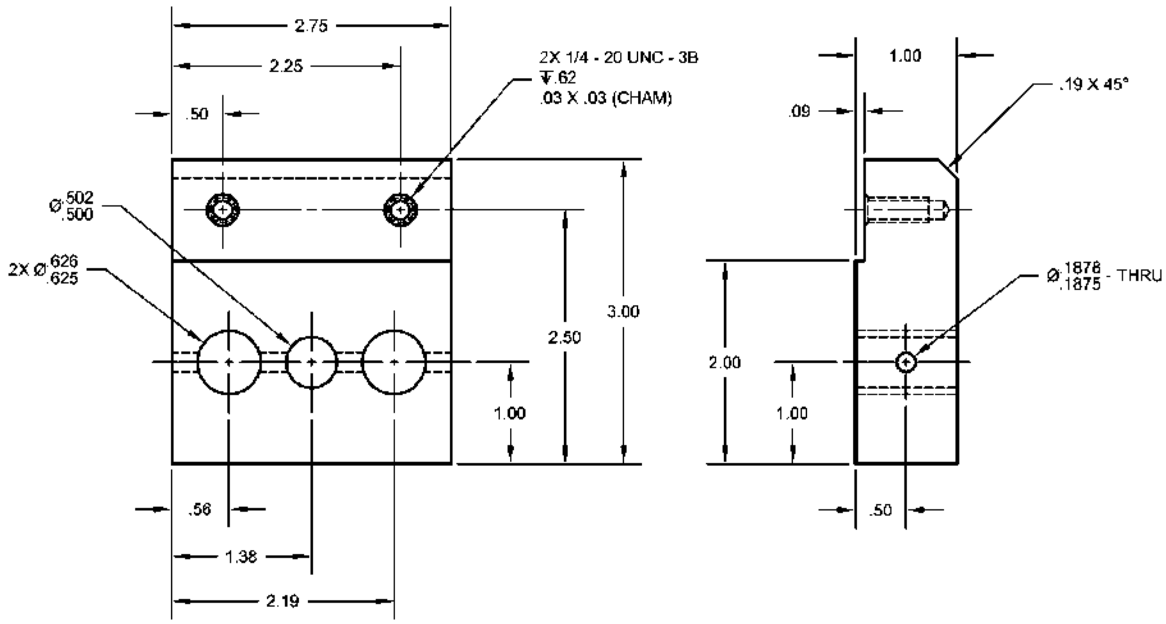


Fig. 8.1 AutoCAD® drawing example

8.3.1 AutoCAD's Workspaces and User Interface

AutoCAD® has three predefined workspaces: Drafting & Annotation, 3D Basics and 3D Modeling. Each workspace allows for easy access to operations that are relevant to the current workspace. AutoCAD's user interface is workspace dependent.

We will focus on the Drafting & Annotation workspace in this book. The Drafting & Annotation workspace user interfaces are shown in Figure 8.2. The important areas of the interfaces are identified and will be discussed in the sections indicated.

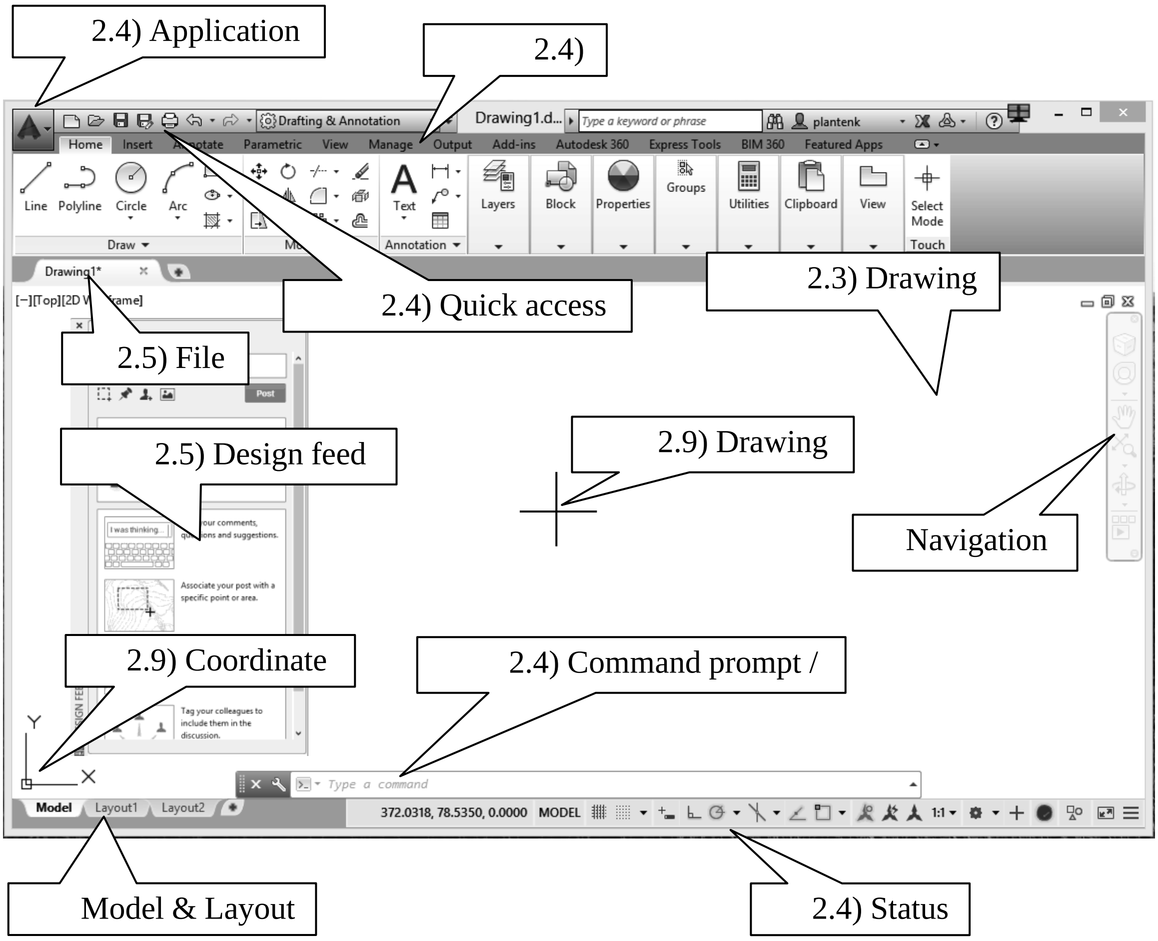


Fig. 8.2 Drafting & Annotation workspace user interface

8.3.2 The Drawing Area

The drawing area/window is the place where you create and view your drawing. The background color of the drawing window may be changed to suit the user's preference as shown in Figure 8.3.

The drawing area is as large as you need it to be. The usable drawing area does not just consist of the area that you can see. You can pan around the drawing area using the Pan command to reveal areas of your drawing that are out of view. You can also zoom in and out to reveal of the drawing area. Both the Pan and Zoom commands are in the Utilities ribbon panel in the Hometab or the Standard toolbar. You can also pan and zoom by manipulating the mouse wheel.

- **Pan**: Hold down the mouse wheel and move the mouse.
- **Zoom**: Rotate the mouse wheel.
- **Zoom All**: Double click the mouse wheel.

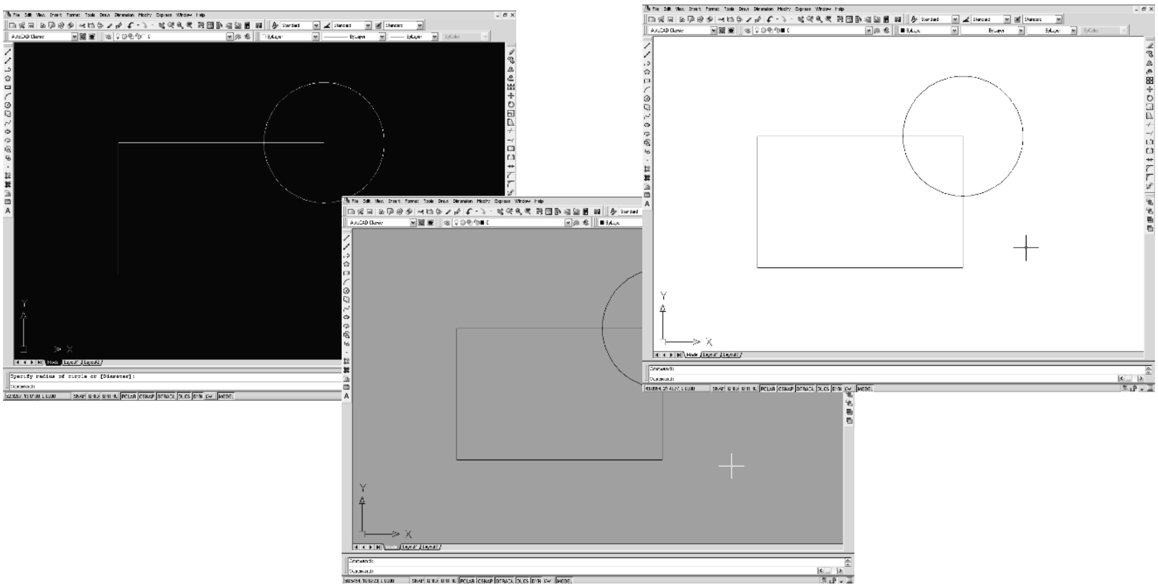


Fig. 8.3 Drawing area background color

<u>Changing the drawing area's background color</u>

1) <u>Application button:</u>  Select the ***Options***  icon at the bottom of the menu.
2) <u>*Options* window – *Display* tab:</u> Select the ***Colors…*** button.
3) <u>*Drawing Window Colors* window:</u>
   a) <u>*Context* field</u>: Select ***2D model space***
   b) <u>*Interface element* field:</u> Select ***Uniform background***
   c) <u>*Color* field:</u> Select your color preference.
   d) Select the ***Apply & Close*** button.

Because the drawing area is so large, it is a good idea to indicate the region that you wish to use. This is your drawing size or limits. This is usually the area that will be printed. You can change your drawing size using the **LIMITS** command.

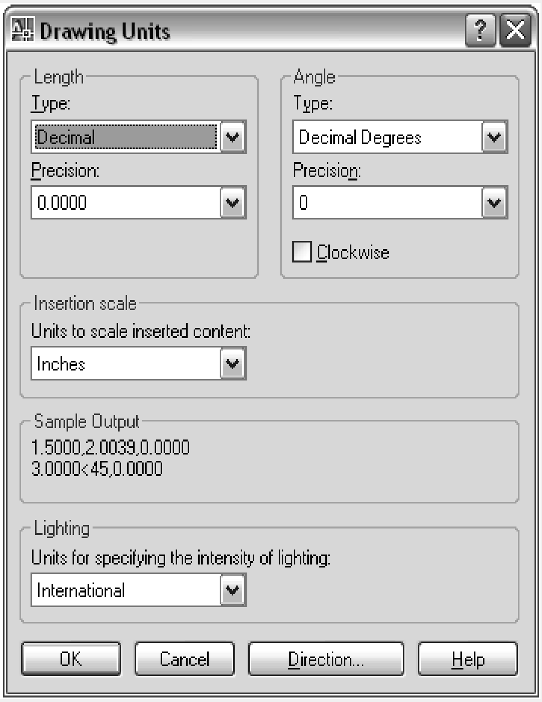
Setting your drawing size

1) Command: **limits**
2) `Specify lower left corner or [ON/OFF] <0.0000,0.0000>:` **`Enter`** (The lower left corner of your limits should always remain 0,0.)
3) `Specify upper right corner <420.0000,297.0000>:` **280,216** (This changes a Metric drawing area to the equivalent of an 11 x 8.5 sheet of paper.)

The units (i.e., inches, millimeters, feet) used to draw objects in the drawing area can be selected using the **Units** command.

Setting your drawing units, precision and angle directions

1) Command: **Units**
2) *Drawing Units* window: Use this window to set your units and precision.



8.4 Starting, Saving, and Opening Drawings

8.4.1 Starting a new drawing

When starting a new drawing (QNEW), you have a choice of either starting from the Create New Drawing window or the Select Template window. The Create New Drawing windowallows you to set up a drawing to your preferences. You may set parameters such as the units (Imperial or Metric), the size of the drawing, and the degree of precision. The Select Template window allows you to choose from predefined templates. Figure 8.4 shows both startup windows. The **STARTUP** variable is used to choose what is displayed when the application is started, or which window will appear when you start a new drawing. It has 4 values that may be set (i.e., 0, 1, 2, and 3). However, for starting a new drawing, only 0 and 1 are of interest. If STARTUP = 0, then the Select Template window will appear. If STARTUP = 1, then the Create New Drawing window will appear.

Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views. Template drawings are distinguished from other drawing files by the .dwt file extension. Several template drawings are included in AutoCAD®. You can make additional template drawings by changing the extensions of drawing file names to dwt.

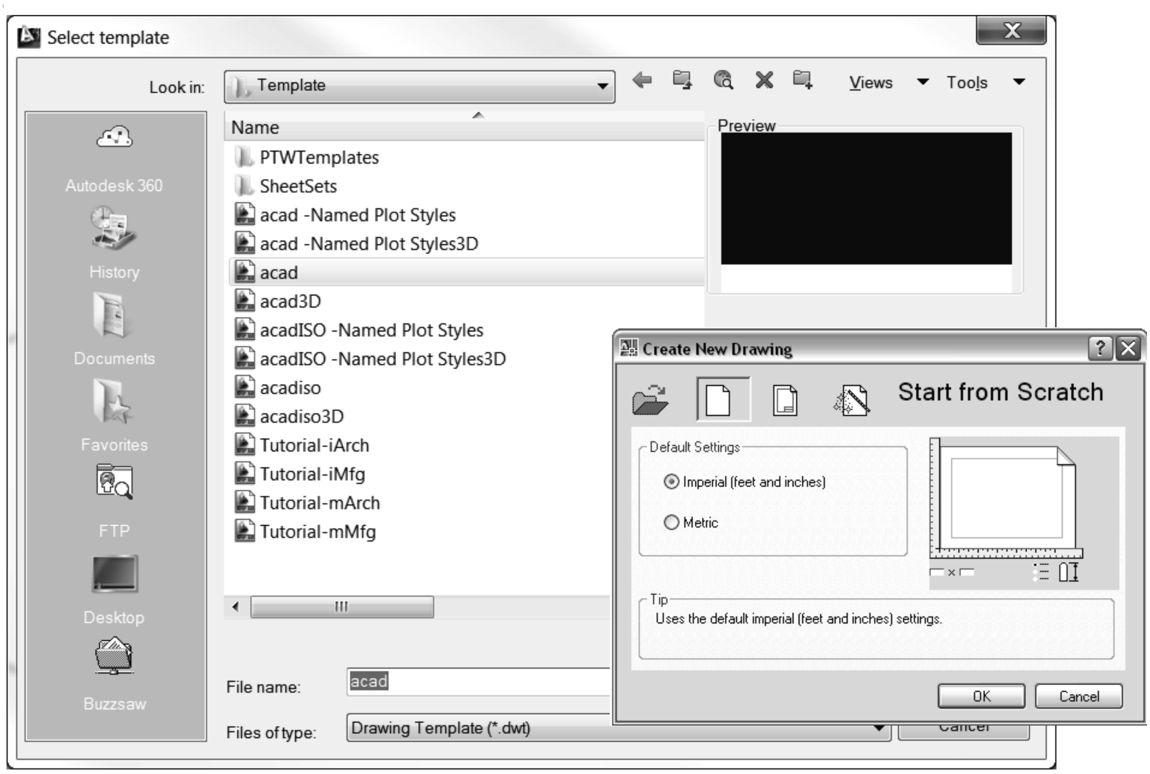
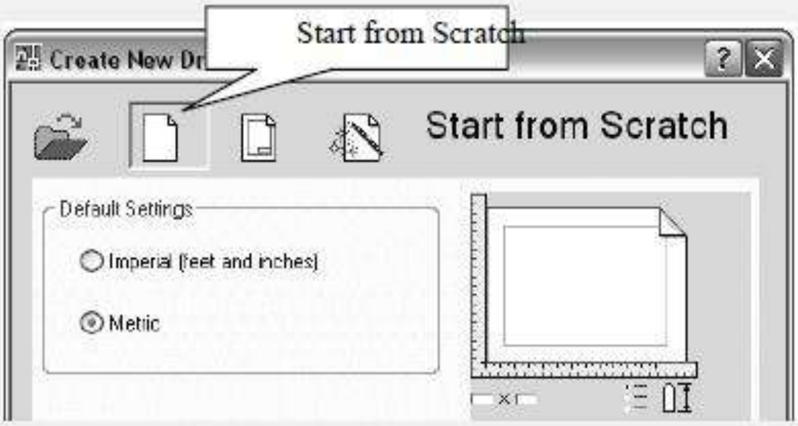


Fig. 8.4 Startup window

Starting a new drawing using the *Create New Drawing* window

1) Command: **startup**
2) `Enter new value for STARTUP <0>:` **1**

3) *Quick Access* toolbar:  or *Application* button: ***File – New…*** (**`Ctrl+N`**).
   The *Create New Drawing* window will appear.
4) *Create New Drawing* window: Activate the ***Start from Scratch*** button, activate either ***Imperial*** or ***Metric*** toggle, and then select ***OK***.



5) *Quick Access* toolbar: 
6) *Create New Drawing* window:
   a) Activate the ***Use a Wizard*** button.
   b) *Select a Wizard* field: Select ***Advanced Setup*** and then ***OK***.

7) The wizard will take you through a setup which will allow you to choose your drawings units, angle, angle measure, angle direction and drawing area.

8.4.2 Saving and opening a drawing

When saving (or open) a drawing (***Application*** button - ***Save*** or ***Save as*** or ***Open***), you have the option of saving (or opening) the following file types.

- **DWG** (DraWinG) is a binary file format used for storing two and three-dimensional design data and metadata. Most of what you draw will be saved in this format.
- **DWT** is a template file. These files are used as a starting point when starting a new drawing. They may contain drawing preferences, settings, and title blocks that you do not want to create repeatedly for every new drawing.
- **DXF** (Drawing Interchange Format or Drawing Exchange Format) is a CAD data file format developed by Autodesk® for enabling data interoperability between AutoCAD® and other programs.
- **DWS** is a standards file. To set standards, you create a file that defines properties for layers, dimension styles, line types, and text styles, and you save it as a standards file with the .dws file name extension.

8.4.3 File Tab

File tabs are displayed at the top of the drawing area as shown in Figure 8.5. They help provide an easy way for you to access all the open drawings in the application. The file tab usually displays the full name of the file. A preview of the file and all its spaces appears when you hover your mouse over the tab. The plus button on the right end of the file tabs displays the *Select Template* dialog box to create a new drawing. The following are the system variables that relate to the file tabs.

- **FILETAB** opens the file tabs.
- **FILETABCLOSE** closes the file tabs.
- **FILETABPREVIEW** controls the type of preview (1 = thumbnail, 0 = list).

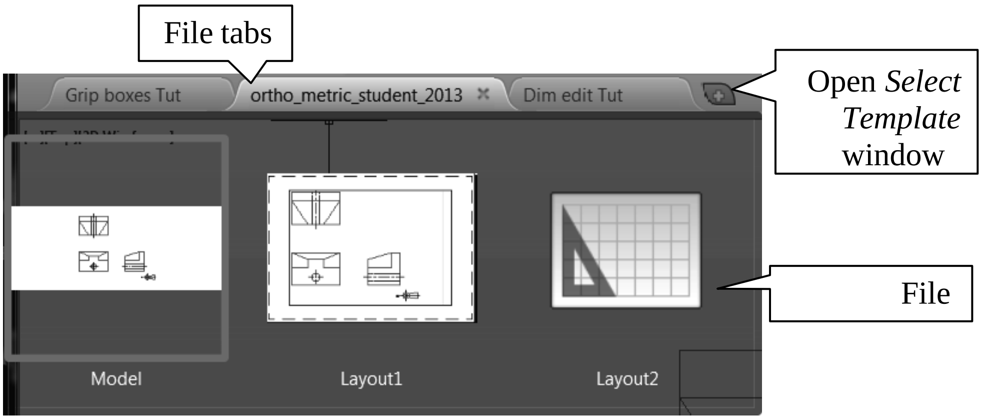


Fig. 8.5. File tab preview

8.5 Cartesian and polar coordinates

Objects created in AutoCAD such as lines and circles are defined by coordinate points. The user may specifically state the defining coordinate points using the Command window or use the geometry of other objects to define the coordinate points. The two main coordinate systems used by AutoCAD are the Cartesian coordinate system (x,y) and the polar coordinate system (r,θ).

8.5.1 Cartesian coordinate system

The Cartesian coordinate system consists of three mutually perpendicular axes. The axes are labeled the x-axis, the y-axis and the z-axis as shown in Figure 8.6. The point where all the axes meet is called the origin. The origin is defined to be the zero location (0,0,0). The location of any point in space can be identified by an x position, a y position and a z position relative to the origin. Since this chapter only deals with AutoCAD's 2-D capabilities, a point in space will be defined by an x andy value. The z value will always remain zero.

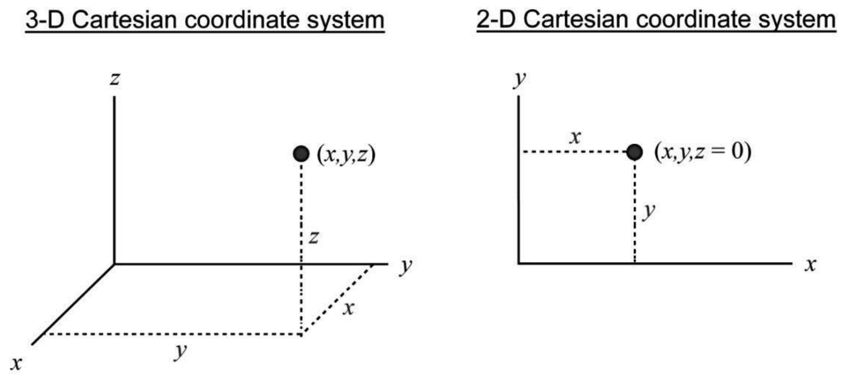


Fig. 8.6 The Cartesian coordinate system

To illustrate the use of Cartesian coordinates in AutoCAD, let's look at the sequence of commands used to draw the line shown in Figure 8.7. As shown, the line lies in the x-y plane. It starts at the point x = 2 and y = 1. It ends at the point x = 6 and y = 3. This is the command sequence used to create this line in AutoCAD:

- Command: **l** or **line**
- `LINE Specify first point`: **2,1**
- `Specify next point or [Undo]`: **6,3**
- `Specify next point or [Undo]`: Press **Enter**to end the LINE command.

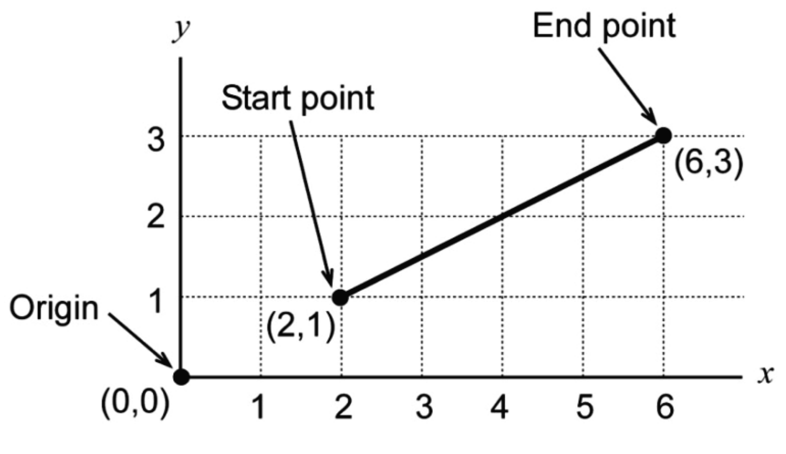


Fig. 8.7 A line defined by Cartesian coordinate points

8.5.2 Polar coordinate system

The 2-D polar coordinate system consists of two mutually perpendicular axes. The axes are labeled the x-axis and the y-axis as shown in Figure 8.8. The point where the axes meet is called the origin. The origin is defined to be the zero location (0,0). The location of any point in space can be identified by the radial coordinate r and the angular coordinate θ. The radial coordinate is the shortest measured distance between the origin and the point under consideration, and the angular coordinate θ is the angle between the radial coordinate line and the x-axis.

The angular coordinate θ is measured positive counterclockwise starting at the positive x-axis. Therefore, if a point lies on the positive x-axis, its angular coordinate is zero. If a point lies on the y-axis, its angular coordinate is 90 degrees. The angular coordinate directions are illustrated in Figure 8.8.

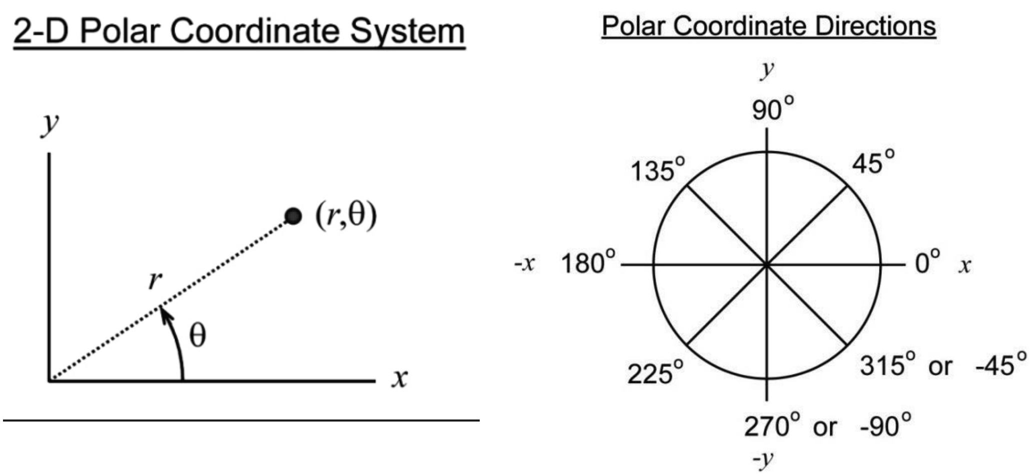


Fig. 8.8 The polar coordinate system

To illustrate the use of polar coordinates in AutoCAD®, let’s look at the sequence of commands used to draw the line shown in Figure 8.9. As shown, the line lies in the x-y plane. It starts at the point r = 2.5 and θ = 60 degrees. It ends at the point r = 7 and θ = 10 degrees. This is the command sequence used to create this line in AutoCAD®:

- Command: **l** or **line**
- `LINE Specify first point`: **2.5<60**
- `Specify next point or [Undo]`: **7<10**
- `Specify next point or [Undo]`: Press **Enter** to end the LINE command.

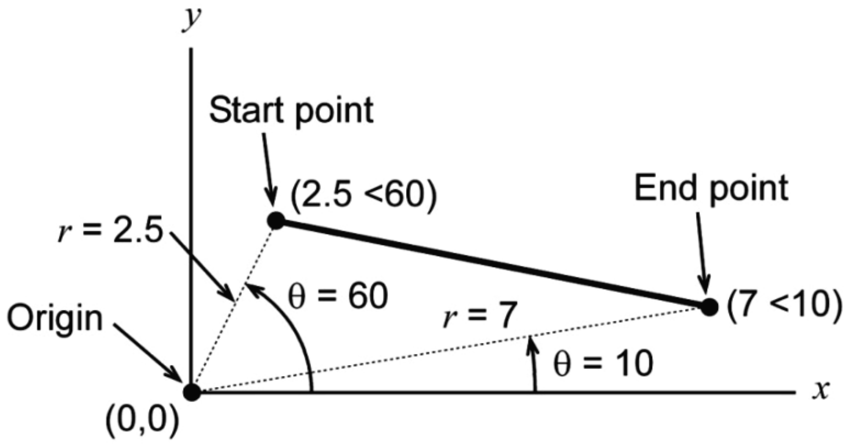


Fig. 8.9 A line defined by polar coordinate points

8.5.3 Relative coordinates

Many times, the start point of a line is unknown or the length and angle of the line is known but not the coordinate for the end point. Therefore, AutoCAD® allows you to enter coordinate points that are relative to the last point entered and not relative to the origin. It is like making the last point entered a temporary origin. The symbol **@** is placed before the coordinate point if it is to be relative to the last point entered.

To illustrate the use of relative coordinates, let's look at the sequence of commands used to draw the two lines shown in Figure 8.10. This is the command sequence used to create these lines in AutoCAD®:

Line 1

- Command: **l** or **line**
- `LINE Specify first point:` Using the mouse, select a point anywhere in the drawing area.
- `Specify next point or [Undo]:` **@4,2**
- `Specify next point or [Undo]:` Press **Enter**to end the LINE command.

Line 2

- Command: **l** or **line**
- `LINE Specify first point:` **2,1**
- `Specify next point or [Undo]:` **@5<25**
- `Specify next point or [Undo]:` Press **Enter**to end the LINE command.

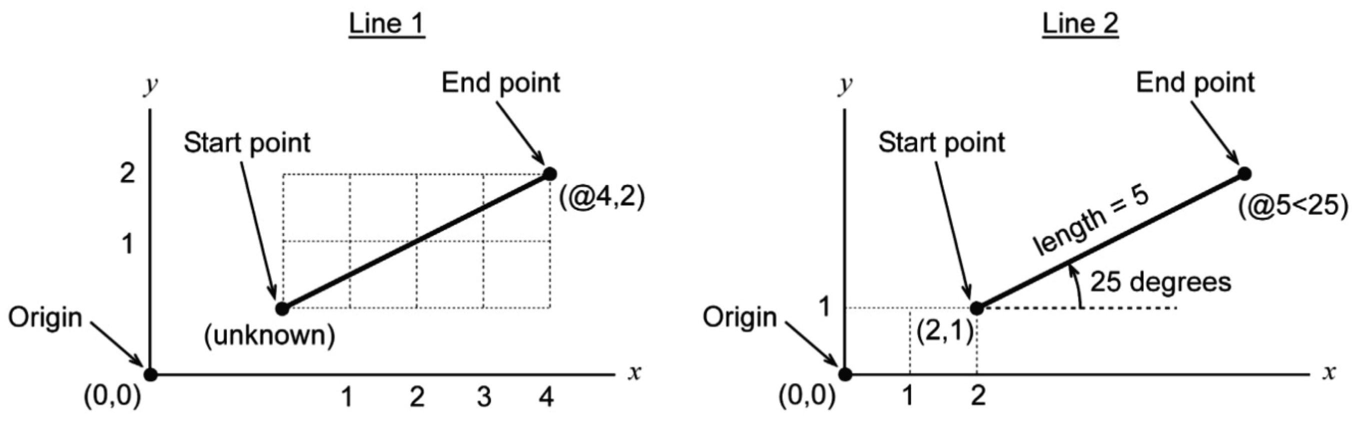


Fig. 8.10 Lines defined by relative coordinates

8.5.4 World coordinate system (WCS) and user coordinate system (UCS)

AutoCAD® has two different coordinate systems: the world coordinate system (WCS), which is fixed and cannot be moved, and the user coordinate system (UCS), which is movable. The WCS origin is always located in the same place. When opening a new drawing, it is in the bottom left corner of the drawing window. The UCS origin may be translated and its axes rotated using the commands found in the UCS panel. The UCS is very useful when drawing objects that are relative to a point on an existing object. Figure 8.11 shows the coordinate axes for the WCS and a translated and rotated UCS. Notice that the WCS coordinate axes have a box at the origin and the UCS does not. Visually this is how you can tell them apart. The properties of the coordinate axes may be changed to suit the user's preferences.

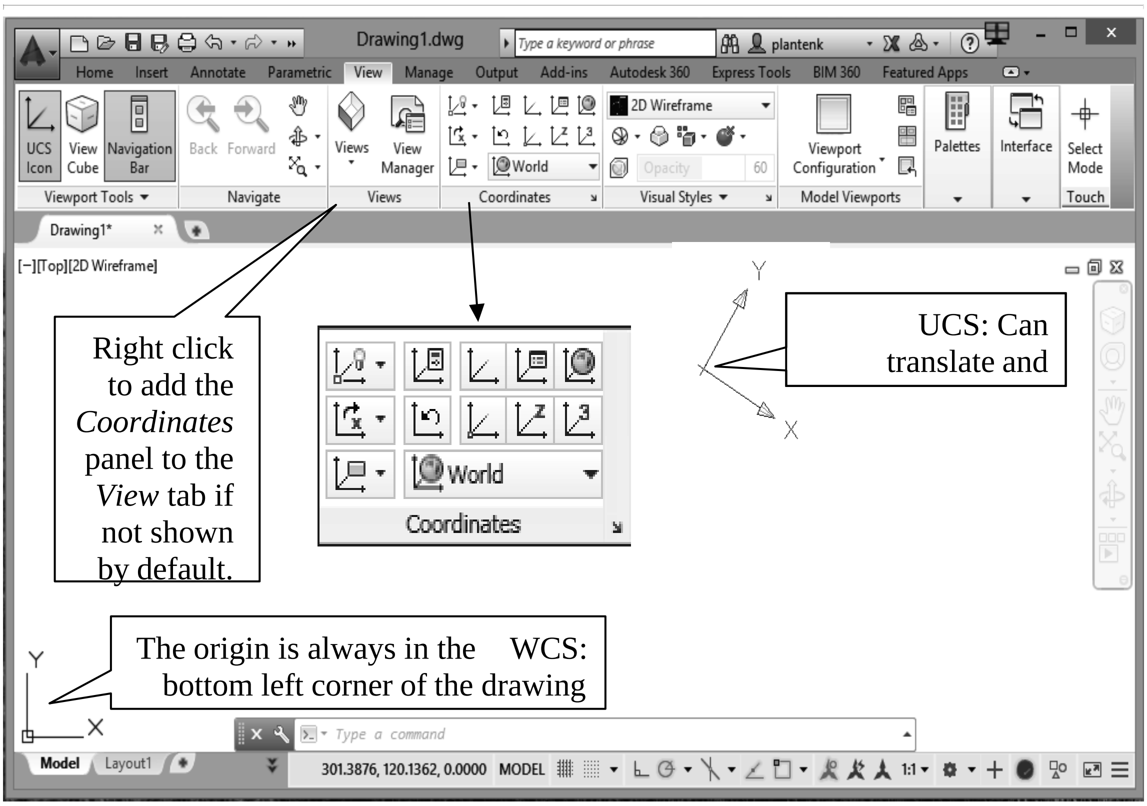


Fig. 8.11 AutoCAD's coordinate axes

8.6 Draw Commands

The Draw panel (Figure 8.12) contains commands that allow you to draw standard geometries. The Draw commands can be used to create new objects such as lines and circles. Most AutoCAD® drawings are composed purely and simply from these basic components. A good understanding of the Draw commands is fundamental to the efficient use of AutoCAD®. To determine the name of thecommand associated with each icon in the Draw panel, place the cursor over each icon in turn and the associated command name will pop up and then in a few secondsan extended tooltip will appear.

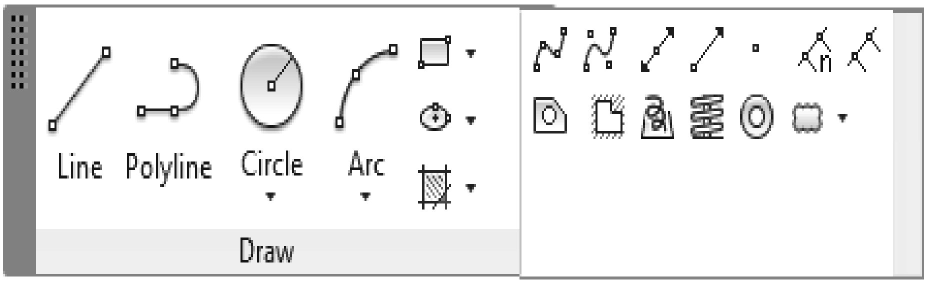


Fig. 8.12 Draw panel

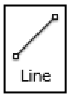
As is usual with AutoCAD®, the Draw commands can be accessed in one of several ways: typing the command in the Command window, from the Draw panel, and from the Draw tool palette.

8.6.1 Line

Lines are probably the simplest of AutoCAD's objects. Using the **LINE** command, a line can be drawn between any two points picked within the drawing area. A line drawn between two points is often called a vector. This terminology is used to describe the type of drawings that AutoCAD® creates. AutoCAD® drawingsare generically referred to as "vector drawings." Vector drawings are extremely useful where precision is the most important criterion because they retain their accuracy irrespective of scale.

With the LINE command you can draw a simple line from one point to another or you can continue picking points and AutoCAD® will draw a straight line between each picked point and the previous point. Each line segment drawn is a separate object and can be moved or erased as required. While the LINE command is active, you can un- enter the last point by using the **UNDO** option available in the Command window. You can also close a sequence of lines (connect the start and endpoint) using the **CLOSE** option.

The LINE command may be accessed in the following way.

- *Draw* panel: 

- *Command* window: **l** or **line**

8.6.2 Construction line

The construction line (**XLINE**) command creates a line of infinite length which passes through two picked points. Construction lines are very useful for creating construction frameworks or grids. Construction lines are not normally used as objects in finished drawings. Therefore, it is usual to draw all your construction lineson a separate layer which will be turned off or frozen prior to printing. Because of their nature, the ZOOM EXTENTS command ignores construction lines.

The construction line command may be accessed in the following way.

- *Draw* panel: 
- *Command* window: **xl** or **x line**

Construction line options (Specify a point or [Hor/Ver/Ang/Bisect/Offset]:).

- Hor: Creates a horizontal construction line.
- Ver: Creates a vertical construction line.
- Ang: Creates a construction line at a specified angle.
- Bisect: Create a construction line that bisects an angle defined by 3 points.
- Offset: Creates a construction line that is offset from an existing line by a specified distance.

8.6.3 Polyline

Polylines (PLINES) differ from lines in that they are more complex objects. A single polyline can be composed of several straight-line or arc segments. Polylines can also be assigned line widths to make them appear solid. Figure 2.13-2 shows a few polylines to give you an idea of the flexibility of this type of line. Because of their complexity, polylines use up more memory than the equivalent line. As it is desirable to keep file sizes as small as possible, it is a good idea to use LINEs rather than polylines unless you have a particular requirement.

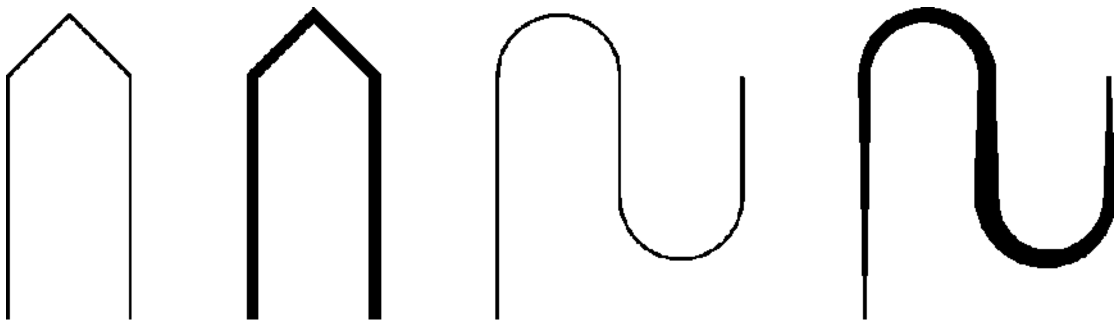
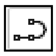


Fig. 8.13 Polylines

The PLINE (polyline) command may be accessed in the following way.

- *Draw* panel: 
- *Command* window: **pl** or **p line**

Polyline LINE options (Specify next point or [Arc/Half width/Length/ Undo/Width]).

- Half width: Enables you to set the half width of the start and end of the line.
- Length: Enables you to specify the length of the line.
- Undo: Removes the most recent segment added to the polyline.
- Width: Enables you to set the start and end widths of the line.

Polyline ARC options (Specify endpoint of arc or [Angle/Center/Close/ Direction/Half width/Line/Radius/Second pt./Undo/Width]).

- Angle: Enables you to specify the included angle of the arc segment.
- Center: Enables you to specify the center point of the arc.
- Direction: Enables you to specify the starting direction of the arc.
- Half width: Enables you to set the half width of the start and end of the arc.
- Radius: Enables you to specify the radius of the arc.
- Second pt.: Enables you to specify the second point of a three-point arc.
- Undo: Removes the most recent segment added to the polyline.
- Width: Enables you to set the start and end widths of the arc.

The **Undo** option is particularly useful. This allows you to unpick polyline vertices one at a time so that you can easily correct mistakes. Also, polylines may be edited after they are created using the command **PEDIT**.

8.6.4 Polygon

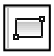
The **Polygon** command can be used to draw any regular polygon from 3 sides up to 1024 sides. This command requires four inputs from the user, the number of sides, a pick point for the center of the polygon, whether you want the polygon inscribed or circumscribed and then a pick point which determines both the radius of this imaginary circle and the orientation of the polygon. This command also allows you to define the polygon by entering the length of a side using the EDGE option.
The POLYGON command may be accessed in the following way.

- *Draw* panel: 
- *Command* window: **pol** or **polygon**

8.6.5 Rectangle

The **Rectangle** command is used to draw a rectangle whose sides are, by default, vertical and horizontal. However, you may draw a rectangle at a specified angle.

The RECTANGLE command may be accessed in the following way.

- Draw panel: 
- Command window: **rec** or **rectangle**

RECTANGLE options (Specify first corner point or [Chamfer/ Elevation/Fillet/Thickness/Width]:).

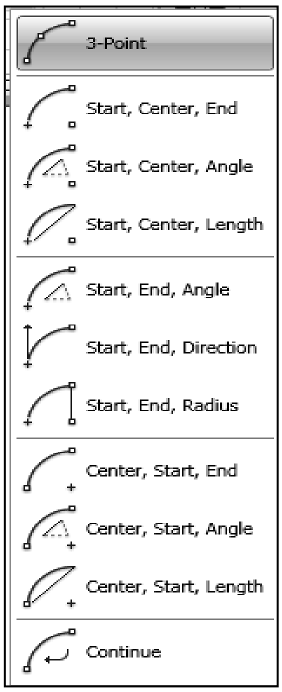
- Chamfer: Creates a rectangle with chamfered corners.
- Elevation: Enables you to specify the elevation of the rectangle.
- Fillet: Creates a rectangle with filleted corners.
- Thickness: Enables you to specify the thickness of the rectangle.
- Width: Enables you to specify the line width of the rectangle.

RECTANGLE options after picking the first corner point (Specify other corner point or [Area/Dimensions/Rotation]:).

- Area: Enables you to specify the area and length of the rectangle.
- Dimension: Enables you to specify the length and width of the rectangle.
- Rotation: Enables you to specify the angle of the line that connects the first and second corner of the rectangle.

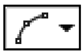
8.6.6 Arc

The Arc command allows you to draw an arc of a circle. There are numerous ways to define an arc; the default method uses three pick points, a start point, a second point and an end. Using this method, the drawn arc will start at the first pick point, pass through the second point and end at the third point. Other ways of defining an arc can be accessed through the fly-out menu under the ARC icon.

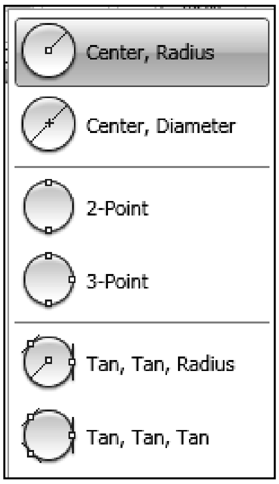


Arcs, by default, travel in the counter clockwise direction. This default direction may be changed by holding down the Ctrl key as you drag.

The ARC command may be accessed in the following way.

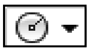
- *Draw* panel: 
- Command window: **a** or **arc**

8.6.7 Circle



The Circle command is used to draw circles. There are several ways that you can define a circle. The default method is to pick the center point and then to either pick a second point on the circumference of the circle or to enter the circle's radius in the Command window. Other ways of defining a circle can be accessed through the fly-out menu under the CIRCLE icon.

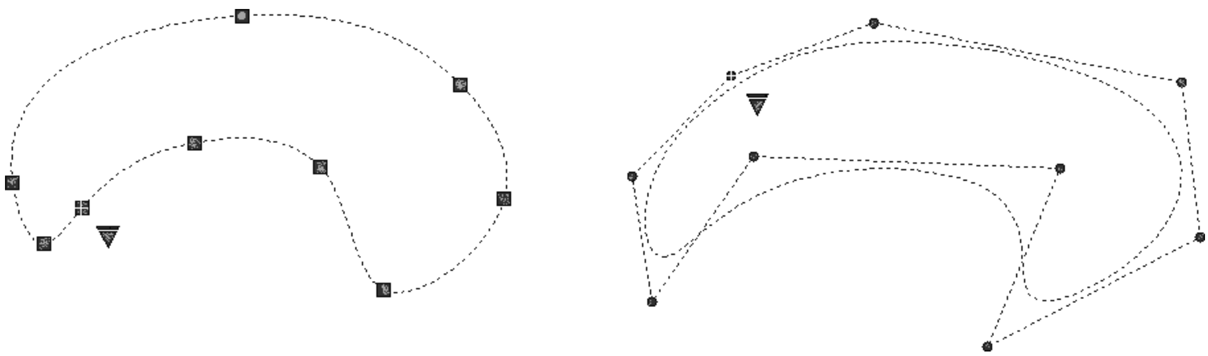
The CIRCLE command may be accessed in the following way.

- Draw panel*:* 
- Command window*:* c or circle

8.6.8 Spline

A **Spline** is a smooth curve that is fitted along a number of control points. Splines can be edited after they have been created using the **SPLINEDIT** command. Using this command, you can change the tolerance, add more control points, move control points and close a spline. However, if you just want to move spline control points, it is best to use the grips boxes.

You can create or edit splines using either control vertices or fit points (see Figure 8.14). The grip boxes are shown when you click on the spline. Creating a spline using control vertices has the advantage of fine control. The variables **CVSHOW** and **CVHIDE** control whether or not the control vertices are displayed. The options available within the spline command depend on the method used to create the spline.



Spline Fit　　　　　　　　　　　　Spline CV

Fig. 8.14. Spline methods

The SPLINE fit command may be accessed in the following way.

- Draw panel: (fit)  (CV) 
- Command window: **spl** or **spline**

Spline Fit options (Specify first point or [Method/Knots/Object]:)

- Method: Allows you to choose between the fit and control vertices methods.
- Knots: Allows you to choose the knot parameterization (chord, square root or uniform).

Spline Fit options (Enter next point or [start Tangency/tolerance])

- start Tangency: Allows you to specify the tangency of the first curved segment.
- tolerance: Allows you to control how closely the spline conforms to the control points. A low tolerance value causes the spline to form close to the control points. A tolerance of 0 (zero) forces the spline to pass through the control points.

Spline Fit options (Enter next point or [end Tangency/tolerance/Undo/Close])

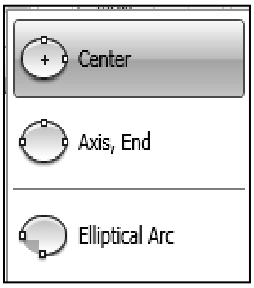
- <u>end Tangency:</u> Allows you to specify the tangency of the last curved segment.
- <u>Undo:</u> Removes the last point.
- <u>Close:</u> Closes the spline.

Spline CV options (Specify first point or [Method/Degree/Object]:)

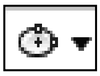
- <u>Degree:</u> Allows you to specify the degree of the spline.

8.6.9 Ellipse

The **Ellipse** command gives you a few different creation options. The default option is to pick the two end points of an axis and then a third point to define the eccentricity of the ellipse. Other ways of defining an ellipsecan be accessed through the fly-out menu under the ELLIPSE icon.



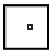
The ELLIPSE command may be accessed in the following way.

- Draw panel: 
- Command window: **el** or **ellipse**

8.6.10 Point

The **Point** command will insert a point marker in your drawing at a position which you pick or at any coordinate location which you enter in the Command window. Other ways of defining a point can be accessed through the fly-out menu. The default point style is a simple dot, which is often difficult to see but you can change the point style to something more easily visible or elaborate using the point style dialogue box.

The POINT command may be accessed in the following way.

- Draw panel: 
- Command window: **po** or **point**

You can access the Point Style window with the command **DDPTYPE** or you can access it from the pull-down menu at **Format - Point Style**… To change the point style (**PDMODE**), just pick the picture of the style you want and then click the OK button (see Figure 8.15). You may also change the point size (**PDSIZE**) in this window. You will need to use the **REGEN** (regenerate) command to update your existing points. Any new points created after the style has been set will automatically display in the new style.

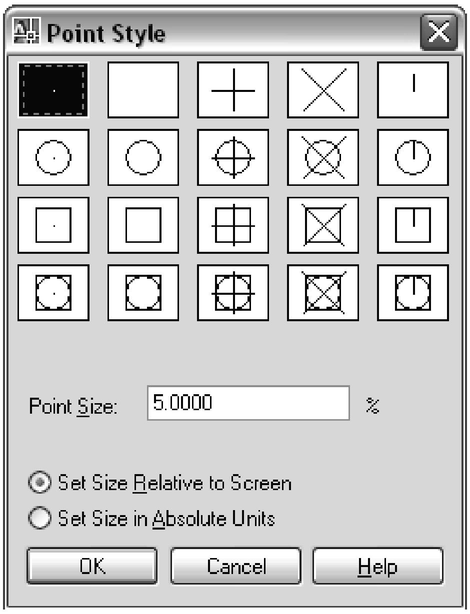


Fig. 8.15 Point Style window

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