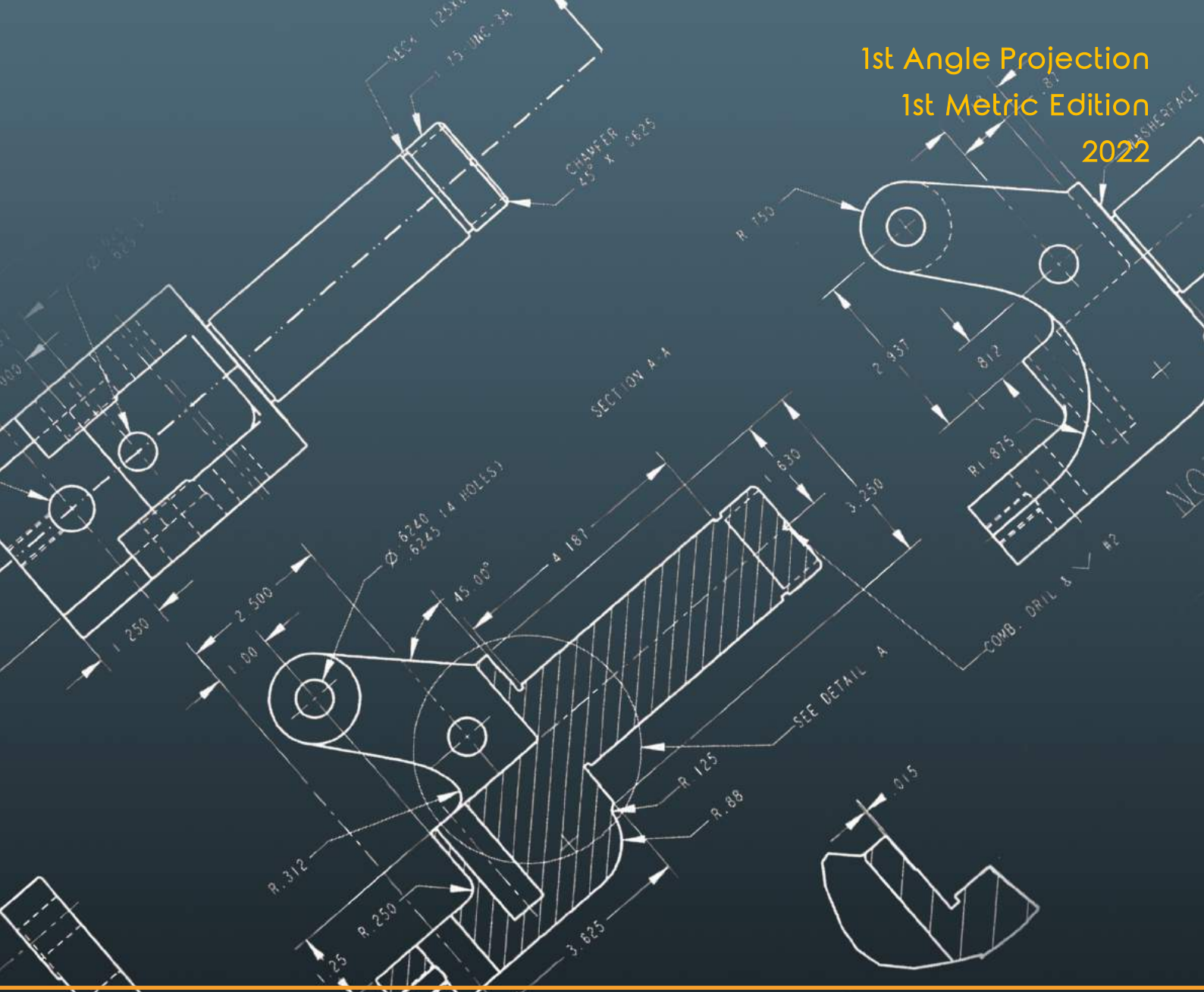
1st Angle Projection
1st Metric Edition
2022



TECHNICAL DRAWING

Lecture Notes

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Drawings are essential communication tools for engineers. They are used for conveying ideas, designs and details of work to others. Engineering students must be able to visualize three dimensional objects and present them on paper in order to be able to communicate with others during their study period and throughout their careers. This lecture note emphasizes visualization and presentation of three dimensional objects as well as the use of drawing instruments and the method of freehand sketching. An introduction to AutoCAD release 2020 will also be given.

The goal and objectives of studying "Technical Drawing" are as follows:

1. Analyze and visualize three dimensional objects.
2. Make good quality sketches to convey ideas to others.
3. High quality precise drawings using instruments and AutoCAD.

The first step in this course is to know the tools required for drawing in the lab, the different sizes of drawing papers, writing of letters in a drawing paper, the different types of lines used in engineering drawing, and how to add dimensions on the drawing.

Three different techniques are used is engineering drawing as follows:

1. free-hand sketching,
2. Using instruments,
3. Using computer graphics techniques.

The course is divided into four main sections as follows:

1. Orthographic Projection (Drawing of the 3 Views),
2. Pictorial Drawing,
3. Missing View,
4. Sectioning.

Students will find lots of solved examples, as well as, many problems for practicing the different topics of the course.

Dr. Hesham A. Hegazi

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This chapter explains and demonstrates the type of equipment and supplies used for manual drafting.

Manual drafting, also known as **hand drafting**, describes traditional drafting practice using pencil or ink on a medium such as paper or polyester film, with the support of drafting instruments and equipment.

DRAFTING PENCILS AND LEADS

Automatic pencils are common for manual drafting, sketching, and other office uses. The term **automatic pencil** refers to a pencil with a lead chamber that advances the lead from the chamber to the writing tip by the push of a button or tab when a new piece of lead is needed (see Figure 1). Automatic pencils hold leads of one width so you do not need to sharpen the lead. The pencils are available in several different lead sizes. Drafters typically have several automatic pencils. Each pencil has a different grade of lead hardness and is appropriate for a specific technique. This reduces the need to change leads constantly.

Figure 1 An automatic pencil. Lead widths are 0.3, 0.5, 0.7, and

LEAD GRADES

Lead grades of 2H and H are good in your automatic pencil for typical daily office use. The leads you select for line work and lettering depend on the amount of pressure you apply and other technique factors. Experiment until you identify the leads that give the best line quality. Leads commonly used for thick lines range from 2H to F, whereas leads for thin lines range from 4H to H, depending on individual preference.
Construction lines for layout and guidelines are very lightly drawn with a 6H or 4H lead. Figure 2 shows the different lead grades.

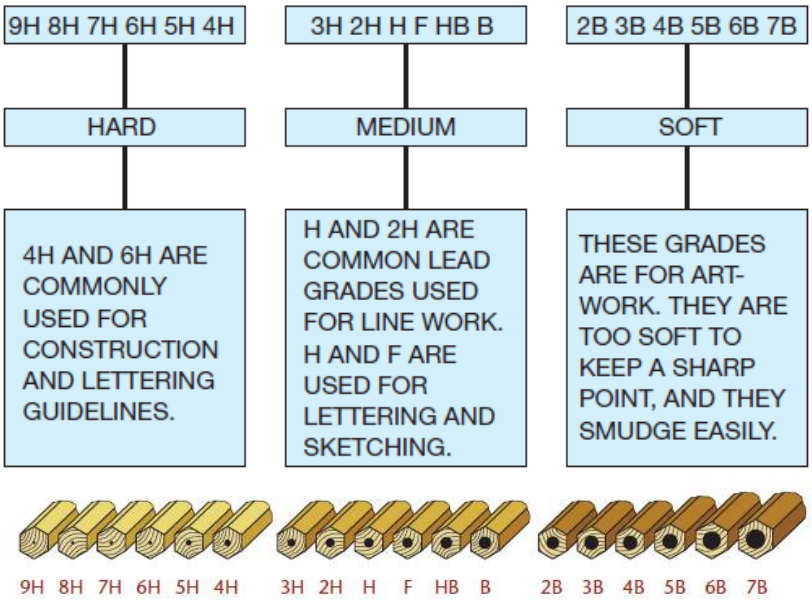


Figure 2 The range of lead grades

COMPASSES

A **compass** is an instrument used to draw circles and arcs. A compass is especially useful for large circles, but using one can be time consuming. Use a template, whenever possible, to make circles or arcs more quickly.
There are several basic types of compasses. A bow compass, shown in Figure 3, is used for most drawing applications.

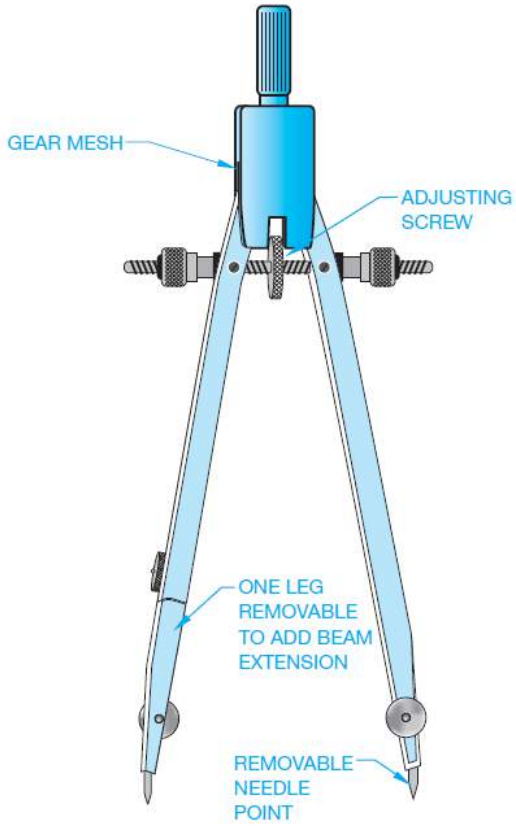


Figure 3 Bow Compass

TRIANGLES

There are two standard triangles. The 30°–60° triangle has angles of 30°–60°–90°. The 45° triangle has angles of 45°–45°–90° (see Figure 4). Some drafters prefer to use triangles in place of a vertical drafting machine scale.

Triangles can also be used as straightedges to connect points for drawing lines without the aid of a parallel bar or machine scale. Use triangles individually or in combination to draw angled lines in 15° increments (see Figure 5). Also available are adjustable triangles with built-in protractors that are used to make angles of any degree up to a 45° angle.

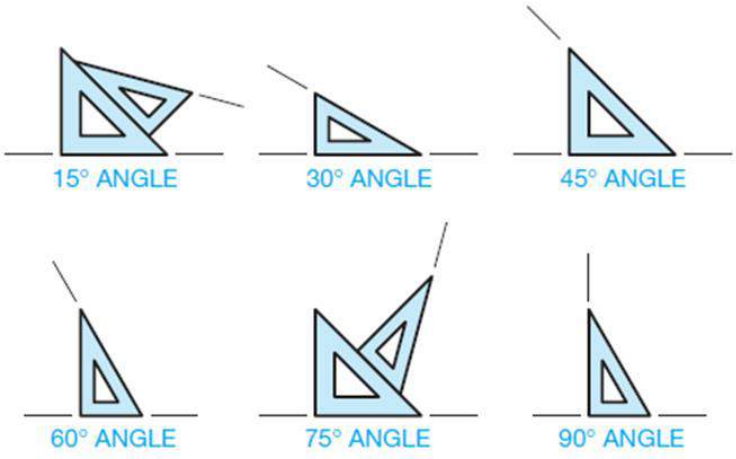


Figure 5 Angles that may be made with the 30°–60° and 45° triangles

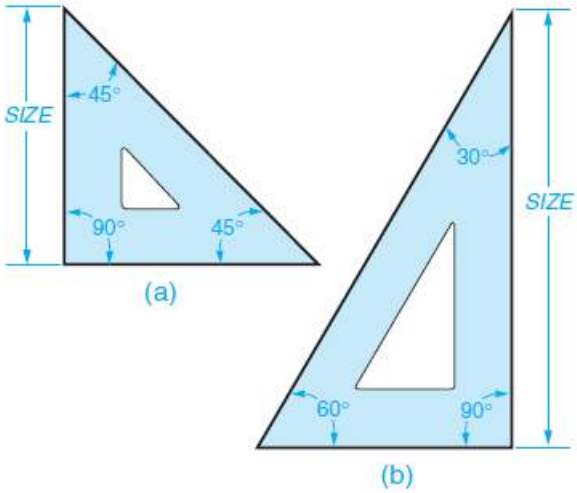


Figure 4 (a) 45° triangle. (b) 30°–60° triangle.

TEMPLATES

Manual drafting **templates** are plastic sheets with accurate shapes cut out for use as stencils to draw specific shapes. The most common manual drafting templates are circle templates for drawing circles and arcs. Templates for drawing other shapes, such as ellipses, and for letters are also common. Templates are also available for specific requirements and drafting disciplines. For example, use architectural templates to draw floor plan and other symbols to scale. Electronic drafting templates have schematic symbols for electronic schematic drawings.

CIRCLE TEMPLATES

Circle templates are available with circles in a range of sizes beginning with 1/16 in. (1.5 mm). The circles on the template are marked with their diameters and are available in fractions, decimals, or millimeters. Figure 2.10 shows the parts of a circle. Figure 2.11 shows examples of circle templates. A popular template is one that has circles, hexagons, squares, and triangles.

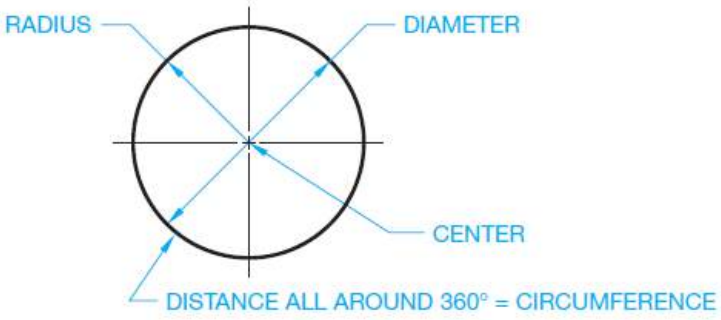


Figure 6 Parts of a Circle

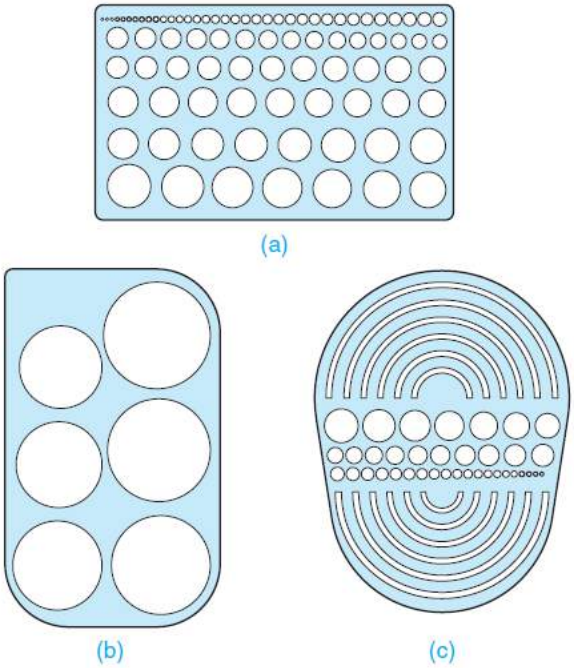


Figure 7 Circle templates. (a) Small circles. (b) Large, full circles. (c) Large half circles.

DRAWING BOARD

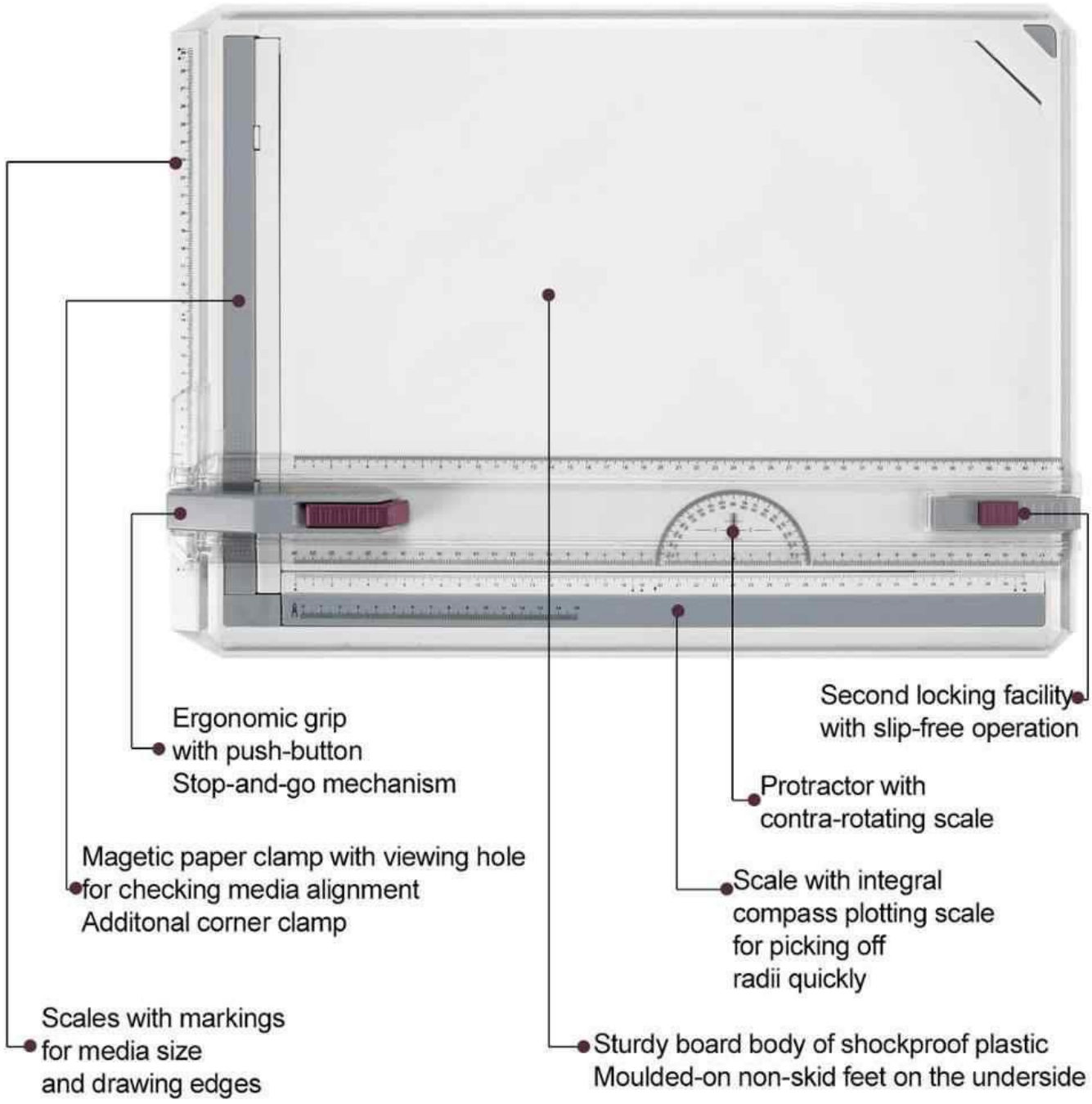


Figure 8 Drawing Board Components

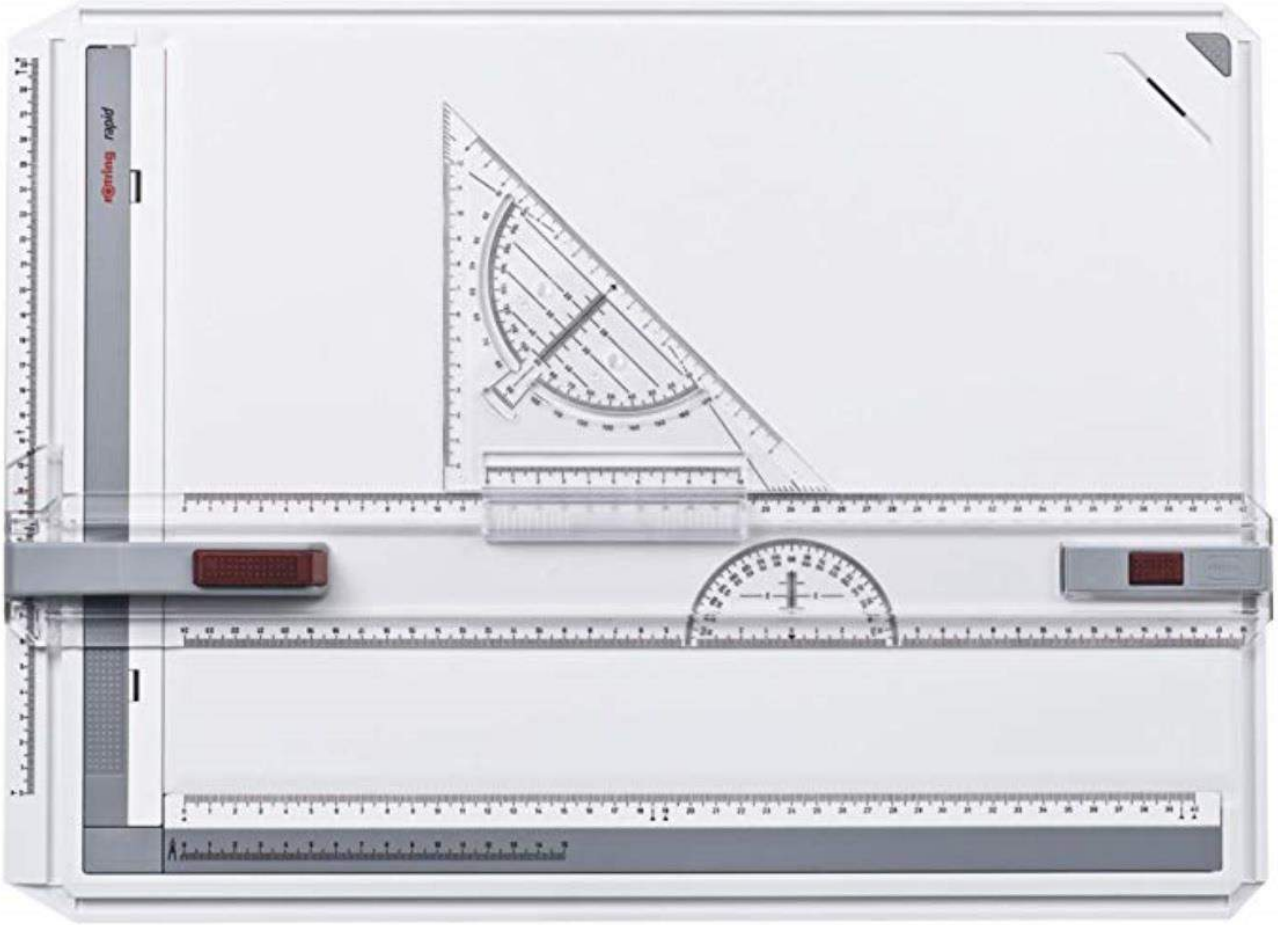


Figure 9 Usage of a triangle to draw vertical lines

Most professional drawings follow specific standards for sheet size and format. The ASME Y14.1 and ASME Y14.1M standards specify the exact sheet size and format for engineering drawings created for the manufacturing industry. Following sheet size and format standards to improve readability, handling, filling, and reproduction; this will also help ensure that all necessary information appears on the sheet.

When selecting a sheet size, consider the size of objects drawn; the drawing scale; the amount of additional content on the sheet, such as a border, title block, and notes; and drafting standards. In general, choose a sheet size that is large enough to show all elements of the drawing using an appropriate scale and without crowding. For example, the dimensioned views of a machine part that occupies a total area of 15 in. × 6 in. (381 mm × 153 mm), can typically fit on a 17 in. × 11 in. (B size) or 420 mm × 297 mm (A3 size) sheet. A larger sheet will likely display too much blank area and is an unnecessary use of material. A smaller sheet will not support the dimensioned views and the remaining elements of the drawing, such as the title block.

The following is a list of standard and optional items found on common sheet size according to ASME standards:

- Border.
- Zoning.
- Title block.
- Angle of projection block.
- Dimensioning and tolerancing block.
- Revision history block.
- Revision status of sheets block.
- Revision status notation.
- Margin drawing number block.
- Application block.
- Microfilm alignment arrows.

ASME Y14.1M, Metric Drawing Sheet Size and Format, specifies the following common metric drawing sheet sizes. The M in the title of the document Y14.1M means all specifications are given in metric.

Figure 1 shows standard metric sheet sizes. The additional sheet sizes A1.0, A2.1, A2.0, A3.2, A3.1, and A3.0 apply to specific elongated sizes. Elongated sizes are horizontally longer than standard sheets.

Size Designation	Size in Millimeters	
	Vertical	Horizontal
A0	841	1189
A1	594	841
A2	420	594
A3	297	420
A4	210	297

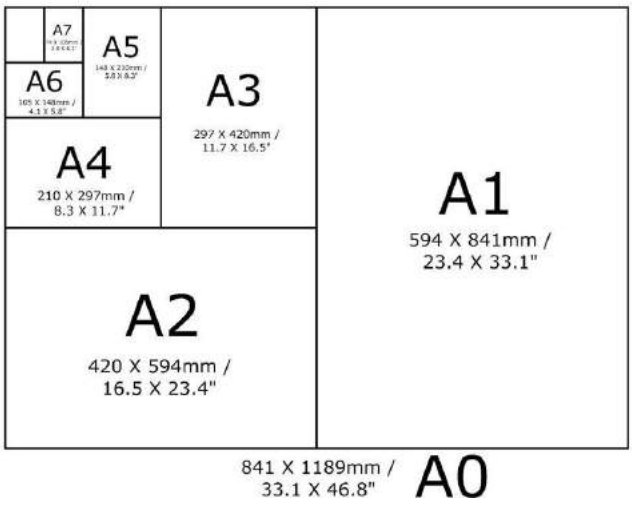


Figure 1 Metric Sheet Sizes

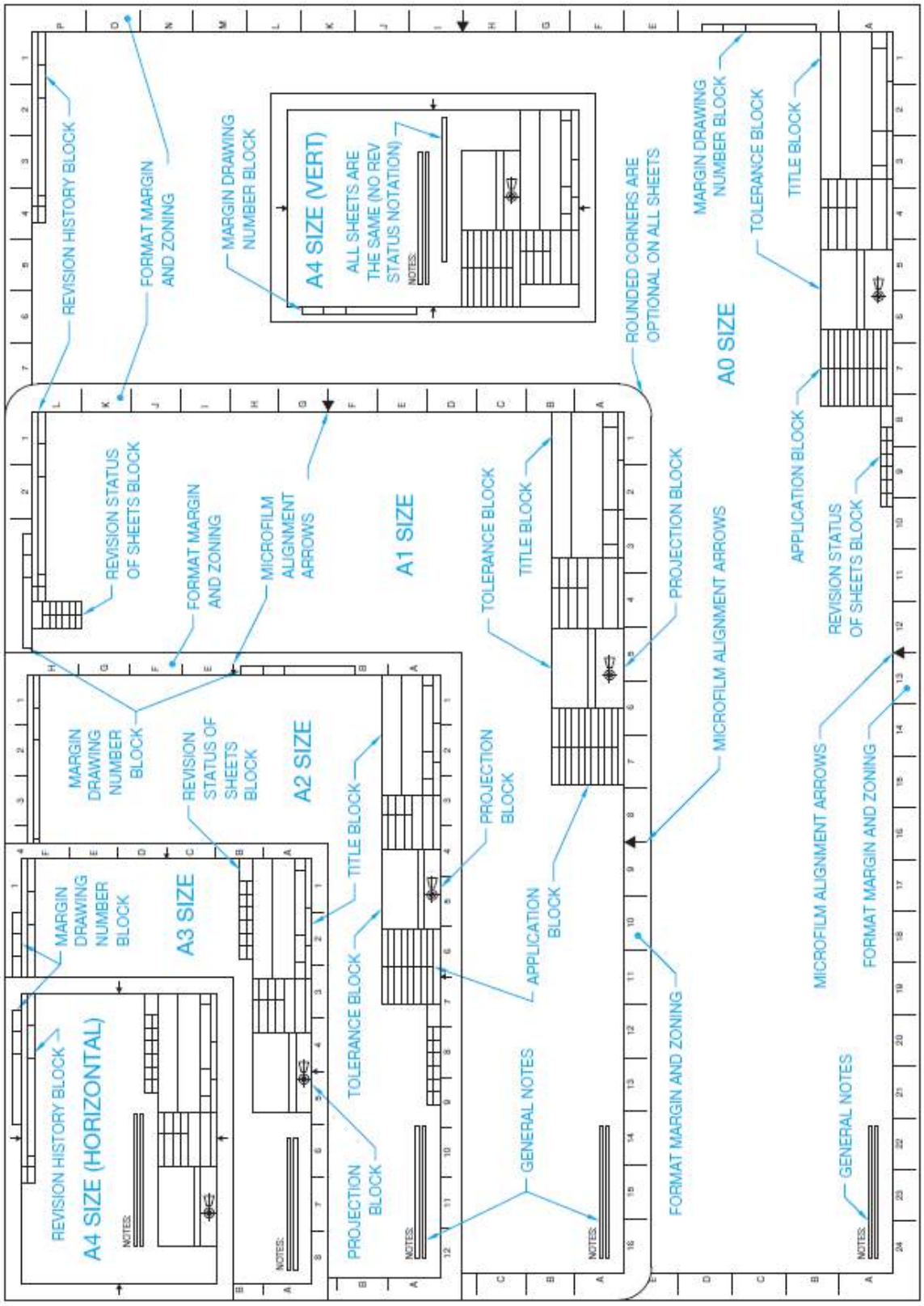


Figure 2 Detailed Sheet Sizes

Title Block

The title block is located in the lower right corner of the format. Refer to Figure 2 for dimensions for a typical title block for A-, B-, and C-size sheets (standard one).

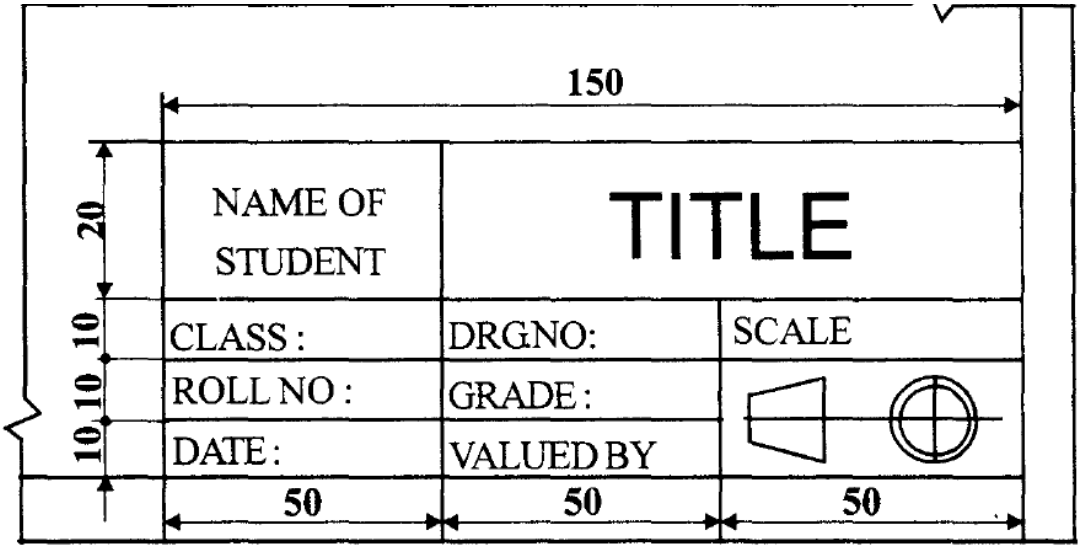


Figure 3 Title Block Dimensions

- **Name** Show the name of the originating company or business (and address if desired). Refer to Figure 3.
- **Drawing Title** Briefly describe the item using a singular noun or noun phrase and modifiers if necessary to distinguish it from similar items. Do not use the terms "for" or "or" in the title. For example, "Dust Cap" would be preferred over "Cap or Cover for Dust Protection," which is too wordy.
- **Drawing Number** Give each drawing a unique number, using the company's numbering system.
- **Sheet Revision Block** Track the drawing version using the number of the revision. The original release of the drawing typically shows revision 0.
- **Approval Block** List the name(s) of the person(s) approving the drawing and the date it was approved. Additional areas of this block can be used for various design activities, if separate approval is required. For example, a company may use separate areas for structural design or manufacturing engineering approvals (Figure 4).
- **Scale** List the predominant scale for the drawing. Drawings may include details at other scales, which should be noted below the detail. If the drawing is not made to a particular scale, note NONE in the scale area.

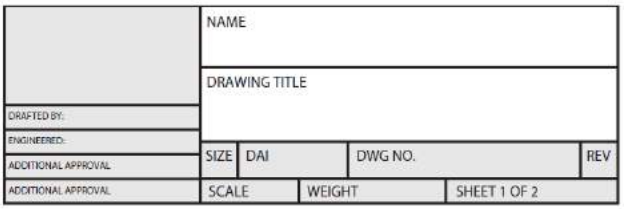


Figure 4 Company Name and Drawing Title

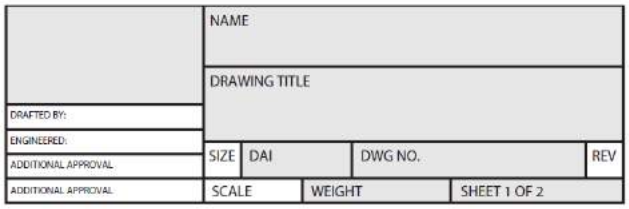


Figure 5 Approval Block, Scale, Revision

As mentioned previously, every company can make its own unique *Title Block and we are using a custom one in our course*

TITLE: **ASSIGNMENT # 2 CW/HW**		University Logo
Name: ID:	Grade:	A3
SCALE:	SHEET 1 OF 2	Course Name

And you also may use a simpler one in your drawings

NAME		HW	8
ID		SEC.	
MISSING VIEW			
University Name		Course Name	

Lettered text is often necessary to completely describe an object or to provide detailed specifications. Lettering should be legible, be easy to create, and use styles acceptable for traditional drawing and CAD drawing.

Engineering drawings use single-stroke sans serif letters because they are highly legible and quick to draw. (Sans serif means without serifs, or spurs.) The sans serif letters used for drawings also referred to as Gothic. (Serif letters are sometimes called Roman, but today that term is commonly used for the upright form of the letters.) A font is the name for a set of letters with the same style. Figure 1 shows the distinctions among ***Roman, italic, serif,*** and ***sans serif*** fonts.

Lettering is a standard feature available in computer graphics programs. With CAD software, you can add titles, notes, and dimensioning information to a drawing. Several fonts and a variety of sizes may be selected. When modifications are required, it is easy to make lettering changes on the drawing by editing existing text.

Freehand lettering ability has little relationship to writing ability. You can learn to letter neatly even if you have terrible handwriting. There are three necessary aspects of learning to letter:

1. Knowing the proportions and forms of the letters (to make good letters, you need to have a clear mental image of their correct shape)
2. Spacing of letters and words for legibility
3. Practice

Figure 1 Letters Styles

STANDARD

The standard for lettering was established in 1935 by the American National Standards Institute (ANSI). This standard is now conveyed by the American Society of Mechanical Engineers document ASME Y14.2, Line Conventions and Lettering. Letters and numbers should be opaque and clearly spaced. Lettering can be vertical or inclined, but only one style should be used throughout the drawing. Uppercase letters are used on drawings unless lowercase letters are required for a specific application. The lettering style used when revising a drawing should match the original drawing lettering style.

Lettering Guidelines

Most hand-drawn notes use lettering about 3 mm high. Light horizontal guidelines are useful for producing consistent letter heights. CAD notes are set using the keyboard and sized to be in the range of 3 mm tall according to the plotted size of the drawing. Lettering heights vary with the size of the sheet and the intended use of the drawing.

CAD drawings typically use a Gothic (sans serif) lettering style but often use a Roman (serif) style for titles. When adding lettering to a CAD drawing, a good rule of thumb is not to use more than two fonts within the same drawing. See Figure 2 for a sample of the fonts available using CAD. You may want to use one font for the titles and a different font for notes and other text.

Figure 2

Keep in mind that if you open a drawing created with software such as AutoCAD, you must have the fonts available that were used in the drawing, otherwise the software will have to substitute different fonts. This can be a problem because the horizontal spacing may be different and the text will no longer fit correctly. It may be tempting to use many different fonts in a drawing because of the wide variety available, but this tends to look distracting on the drawing. Drawings that use too many lettering styles and sizes have been jokingly referred to as having a "ransom note" lettering style.

Guidelines for Hand Lettering

Use extremely light horizontal ***guidelines*** to keep letter height uniform, as shown in Figure 3. Capital letters are commonly made 3 mm high, with the space between rows of lettering being from three-fifths to full height of the letters. Do not use vertical guidelines to space the distance from one letter to the next within a word or sentence. This should be done by eye while lettering. If necessary, use a vertical guideline at the beginning of a column of hand-lettered text to help you line up the left edges of the following rows. Beginners can also use randomly spaced vertical guidelines to practice maintaining the correct slant.

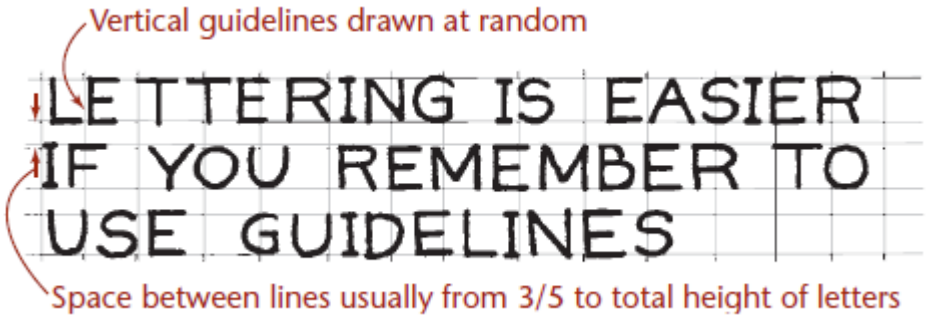


Figure 3 Using Guidelines

Vertical and Inclined Letters and Numbers

The proportions of vertical capital letters and numbers are shown in Figure 4. The letter shapes are probably a little wider than your usual writing. Hand lettering and text added to engineering drawings is typically upper case. Lowercase letters are rarely used except for large volumes of notes or when there is some other particular need for it. Lowercase letters are shown in Figure 5. The lower part of the letter (or descender) is usually two thirds the height of the capital letter.

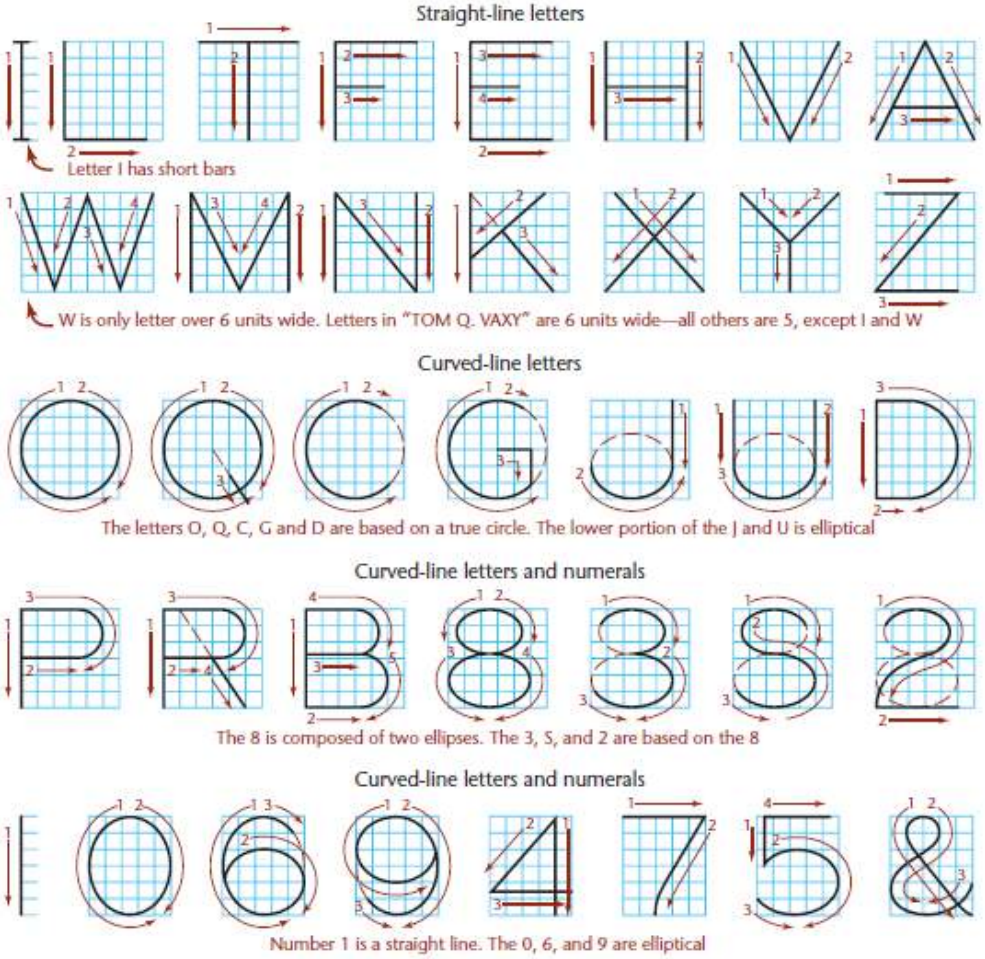


Figure 4

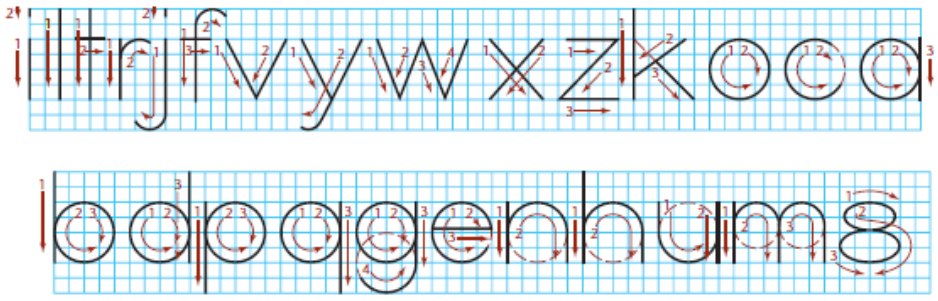


Figure 5

Inclined (italic) capital letters and numerals are shown in Figure 6. They are similar to vertical lettering, except the slope is about 68° from the horizontal. Although you may practice hand lettering slanted at approximately this angle, it is important in CAD drawings to always set the amount of incline for the letters at the same value within a drawing so that the lettering is consistent. Inclined lowercase letters, shown in Figure 7, are rarely used.

Keep in mind that only one style of lettering or font, either vertical or inclined, should be used throughout a drawing.

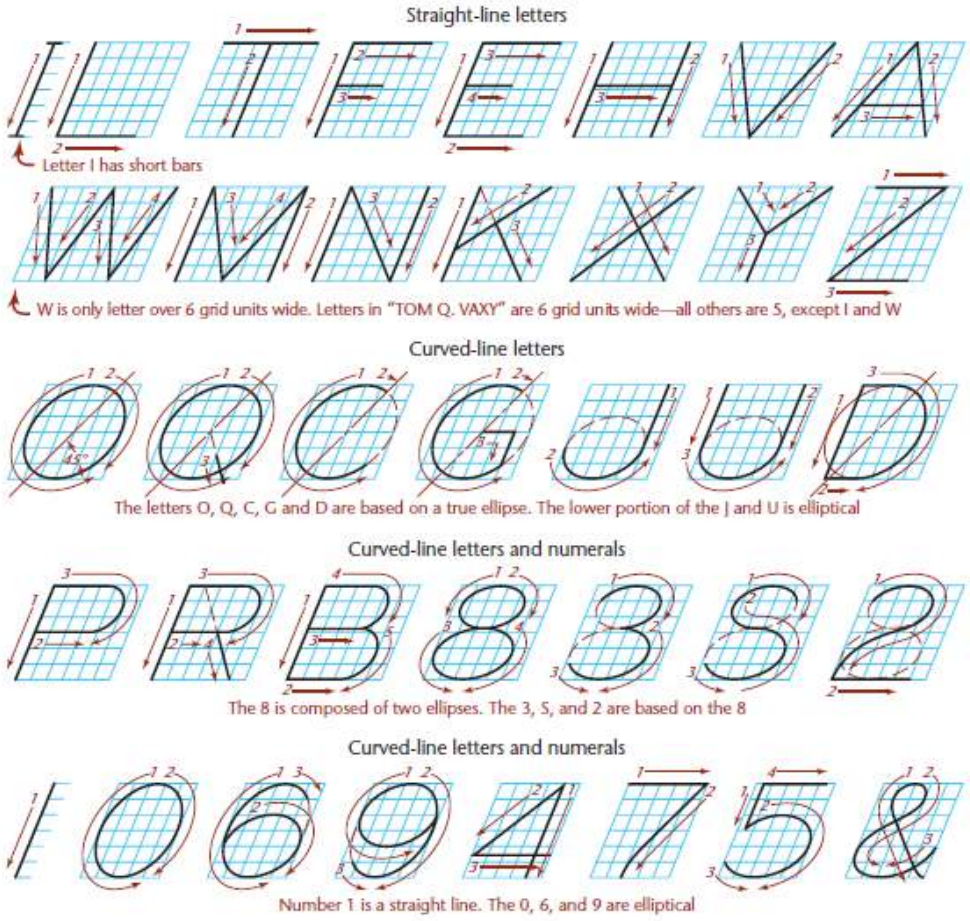


Figure 6

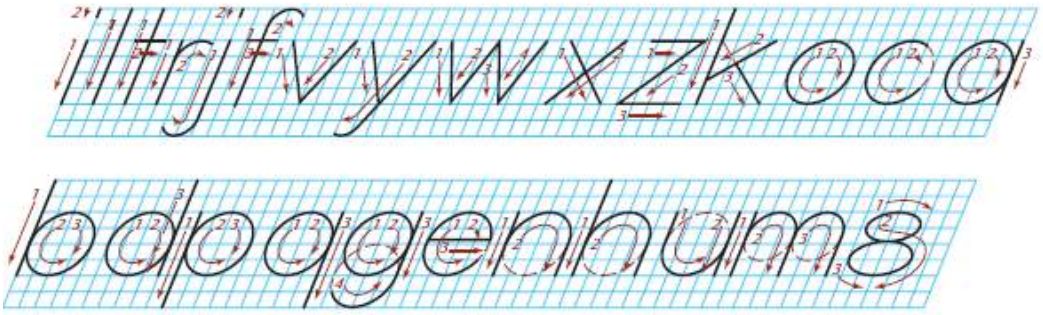


Figure 7

The meaning of each line on a technical drawing is indicated by its width (thick or thin) and its particular line style. The person who reads the drawing will depend on these line styles to know if a line is visible or hidden, if it represents a center axis, or if it conveys dimension information.

To make your drawings easy to read, make the contrast between thick and thin lines distinct. ***Thick lines*** (0.6 mm) should be twice the width of ***thin lines*** (0.3 mm), as shown in Figure 1. The line gage in Figure 2 shows various widths.

Figure 1 Thick and Thin Drawing Lines

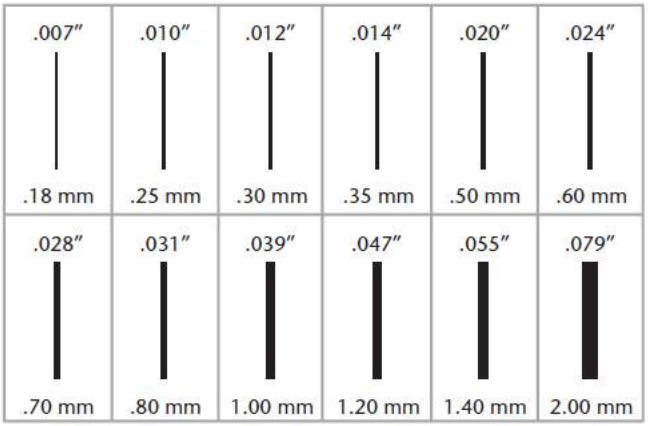


Figure 2 Line gauge

Figure 3 shows freehand line technique. You may find it helpful to use 1/8″ graph paper at first to get a feel for the length of dashes used in hidden lines and centerlines. Soon you will be able to estimate the lengths by eye.

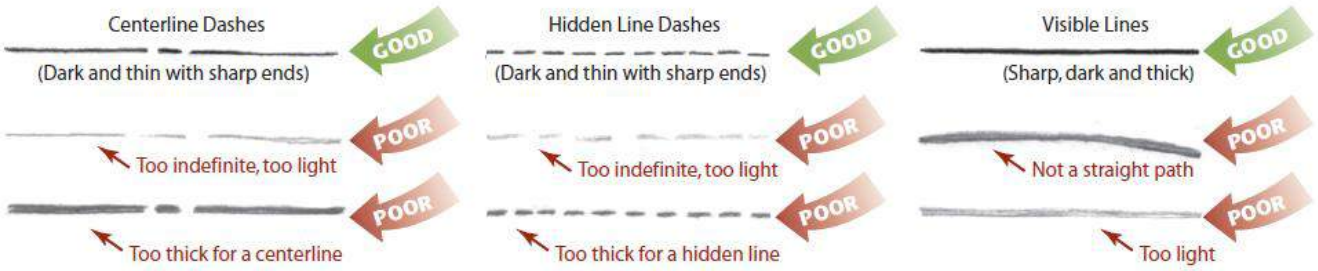


Figure 3 Good and Poor Freehand Line Technique

In the following the figure you can find the most commonly used types of lines

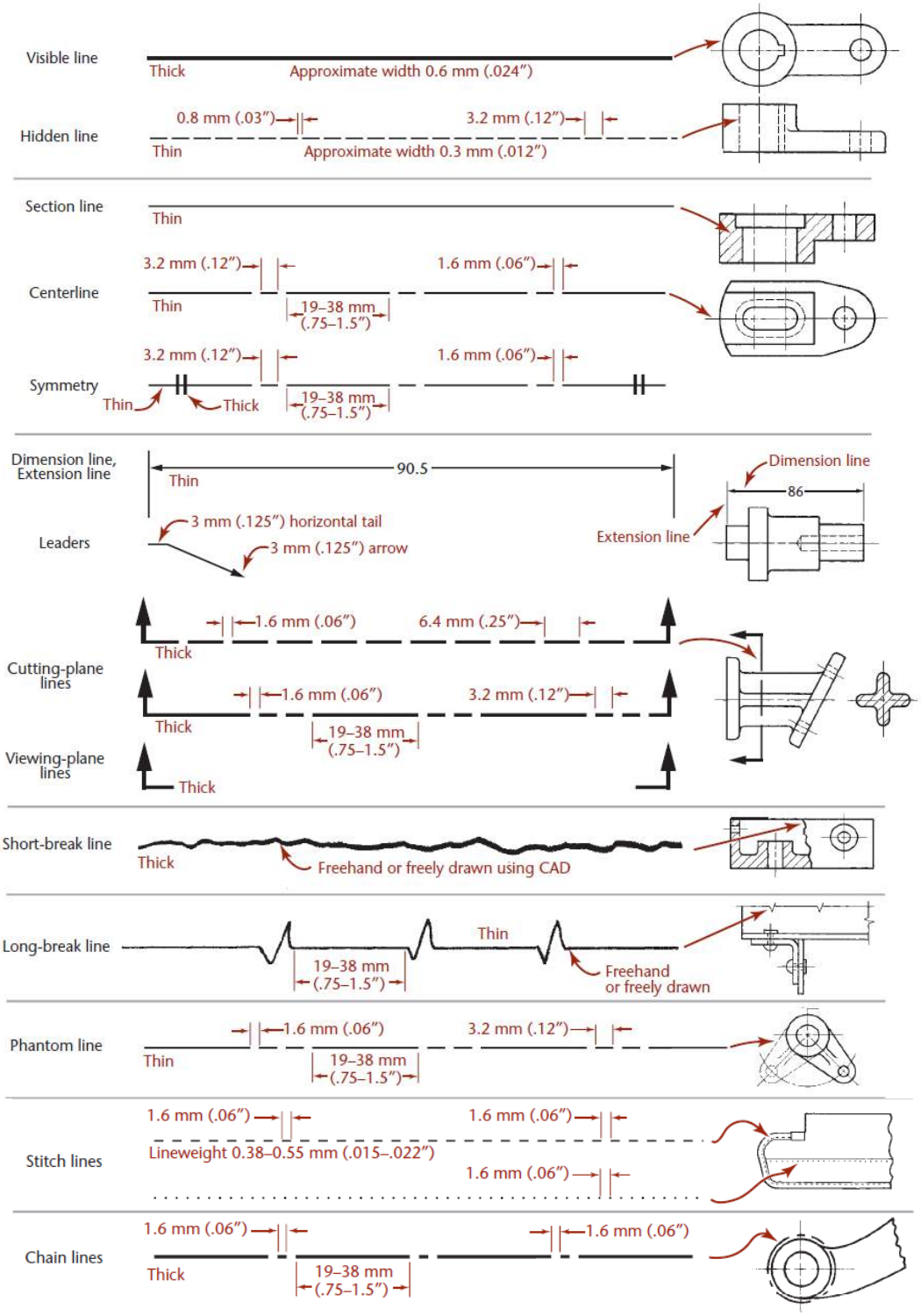
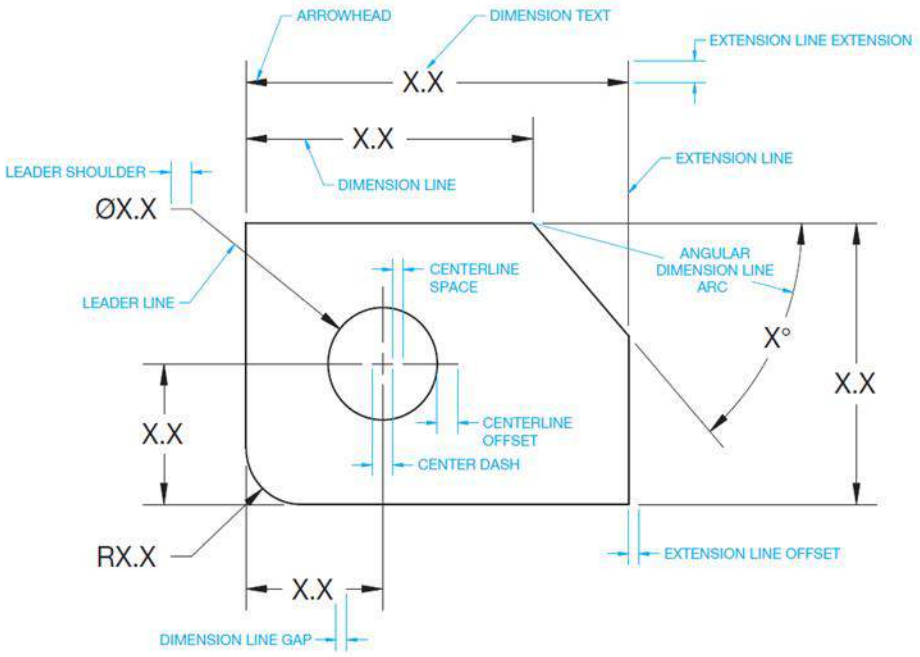


Figure 4 Line Types

It is essential to describe not only the shape of the features you design but also their sizes and locations. Dimensions and notes define the size, finish, and other requirements to fully define what you want manufactured. Standards organizations prescribe how dimensions should appear and the general rules for their selection and placement in the drawing and in digital models, but it takes skill and practice to dimension drawings so that their interpretation is clear and unambiguous. Whether you are creating 2D drawings or 3D models, CAD systems are great for producing dimensions that follow standards for the appearance of the dimensions themselves. However, the job of selecting which dimension to show or where to place it in a drawing takes a level of intelligence that is not part of most CAD systems. Those important decisions are still up to the CAD user—or in other words, *you*.

- DIMENSIONING COMPONENTS

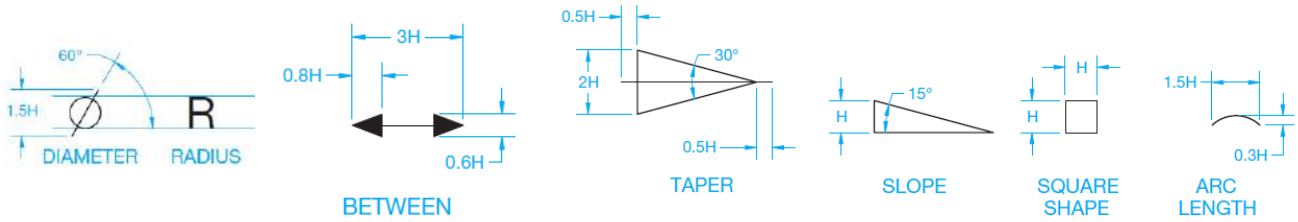
- **Dimension lines** indicate the length of the dimension. They are thin lines capped on the ends with arrowheads and broken along the length, providing a space for the dimension numeral. The **gap** between the dimension line and the dimension numeral varies but is commonly 1.5 mm.
- An **angular dimension line** is an arc with the center of the arc from the vertex of the angle.
- **Dimension text** is normally 3 mm high, centered in the space provided in the dimension line.
- A **leader line** is a thin line used to connect a specific note to a feature on the drawing. The leader line can be at any angle between 15 – 75, with 45 preferred. There is a horizontal shoulder between 3–6 mm, centered where it meets the text.



- **Arrowheads** are used to cap the dimension line and leader line ends. These are discussed in detail later.
- **Extension lines** are thin lines used to establish the extent of a dimension. They start with a small offset of 1.5 mm from the object and extend 3 mm past the last dimension line. Extension lines do not extend between views.
- The **centerline space** is commonly 1.5 mm. This is the space between the short and long dashes of the centerline. The **centerline offset** is the part of the centerline extending beyond the circle or other related feature. The centerline extension is generally 3–6 mm.

- MAIN DIMENSIONING SYMBOLS

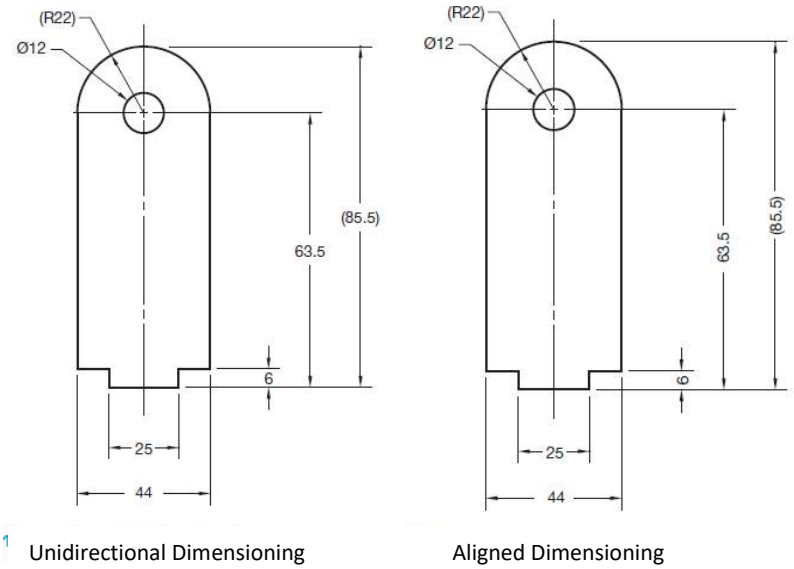
Symbols are commonly used in drafting to replace words, to simplify the drawing, to aid in clarity, and to ease drawing presentation.



Dimensioning systems refers to the manner in which dimensions are applied to drawings for different applications.

- **Aligned dimensioning** requires that all numerals, figures, and notes be aligned with the dimension lines so they can be read from the bottom for horizontal dimensions and from the right side for vertical dimensions see the figure Aligned dimensioning is commonly used in architectural and structural drafting.

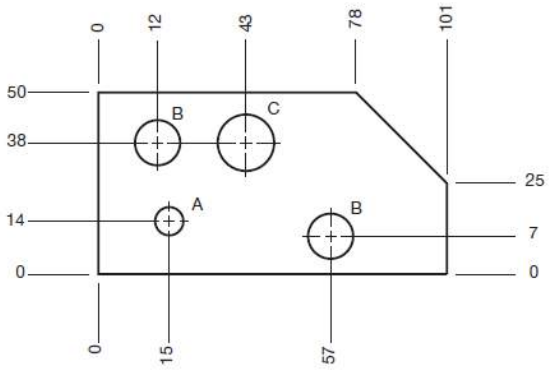
- **Unidirectional dimensioning** is commonly used in mechanical drafting for manufacturing. Unidirectional dimensioning requires that all numerals, figures, and notes be lettered horizontally and be read from the bottom of the drawing sheet.



Unidirectional Dimensioning

Aligned Dimensioning

- Omitting dimension lines is common for drawings in industries that use computer-controlled machining processes and when unconventional dimensioning practices are required because of product features. **Rectangular coordinate dimensioning without dimension lines** is a type of dimensioning that includes only extension lines and text aligned with the extension lines. Each dimension represents a measurement originating from datums or coordinates. Often identification letters label holes or similar features. A table, keyed to the identification letters, indicates feature size or specifications. Alternately, features sizes, such as holes, can be dimensioned using traditional leaders and notes rather than using a table. Use of the table is common, so you should confirm the desired practice with your employer or instructor. Rectangular coordinate dimensioning without dimension lines is popular for specific applications, such as precision sheet metal part drawings and electronics drafting, especially for chassis layout. Rectangular coordinate dimensioning without dimension lines is also called ordinate dimensioning.



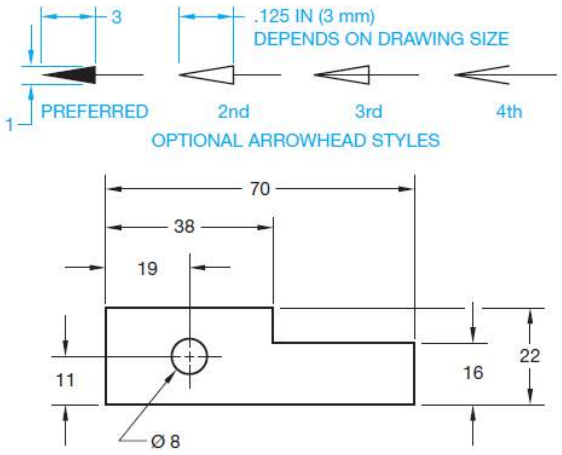
HOLE SYMBOL	HOLE DIA	QTY
A	6	1
B	9	2
C	12	1

Rectangular coordinate dimensioning without dimension lines.

Arrowheads

Arrowheads are used to terminate dimension lines and leaders. Properly drawn arrowheads should be three times as long as they are wide. All arrowheads on a drawing should be the same size. Do not use small arrowheads in small spaces.

Individual company preference indicates if arrowheads are filled in solid or left open as shown in the figure. Most companies prefer the appearance of the filled-in arrowhead. The filled-in arrowheads look better and make the dimension easier to read.

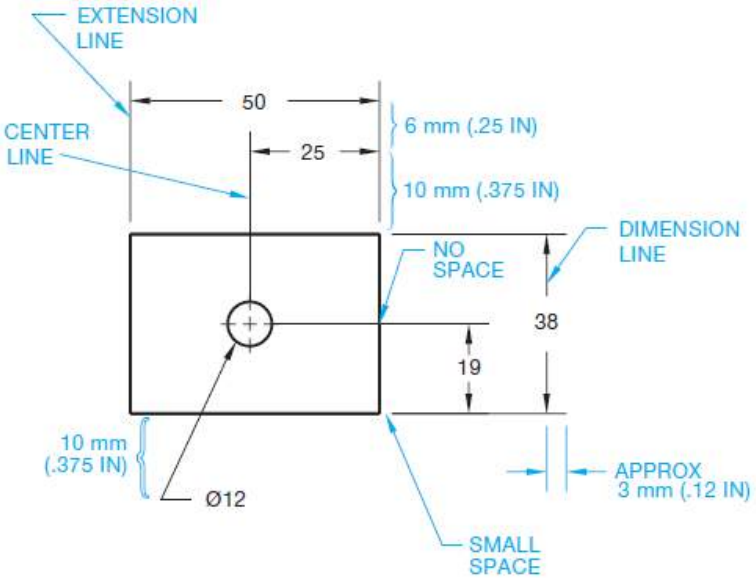


Dimension Line Spacing

Dimension lines are drawn parallel to the feature or object being dimensioned. Dimension lines should be placed at a uniform distance from the object, and all succeeding dimension lines should be equally spaced. The first dimension line should be a minimum of 10 mm from the object, and the second dimension line should be a minimum distance of 6 mm from the first dimension line. All additional dimension lines should be spaced equally, with the same space as the distance from the first to second dimension line. Figure 10.21 shows the minimum acceptable distances for spacing dimension lines.

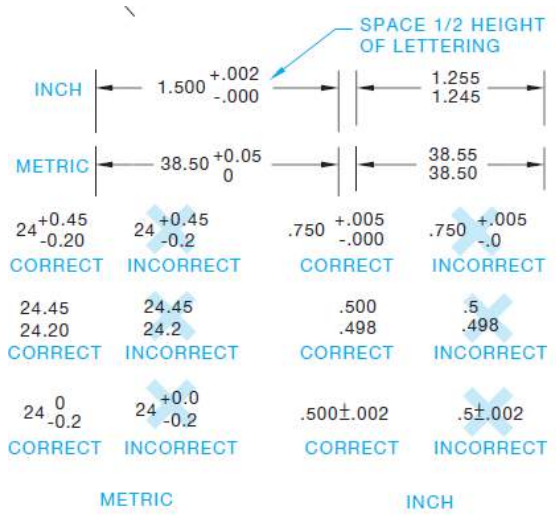
In actual practice, the minimum distance is normally too crowded. Judgment should be used based on space available and information presented. Never crowd dimensions, if possible.

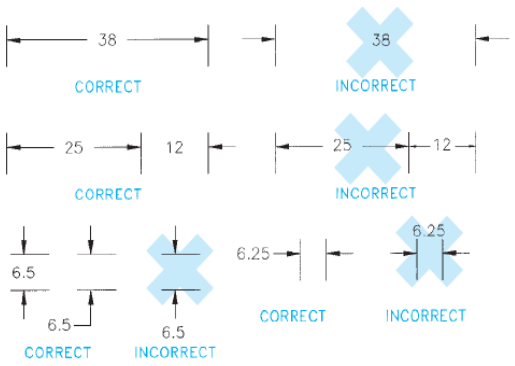
Drafters typically prefer a distance of 12–24 mm and a spacing of 12–20 mm for the following dimension lines. The first dimension line is normally spaced farther from the object than the spacing for additional dimension lines. Always place the smallest dimensions closest to the object and progressively larger dimensions outward from the object. Group dimensions and place dimensions between views when possible.



Minimum dimension line spacing

The following figures show several dimensioning options. Evaluate each example carefully as you dimension your own drawing assignments. Also there are some correct and incorrect dimensioning practices. Keep in mind that some computer aided drafting programs do not necessarily acknowledge all of the rules or accepted examples. Some flexibility on your part is needed to become accustomed to the potential differences that may exist between the recommended applications and the CAD software format.



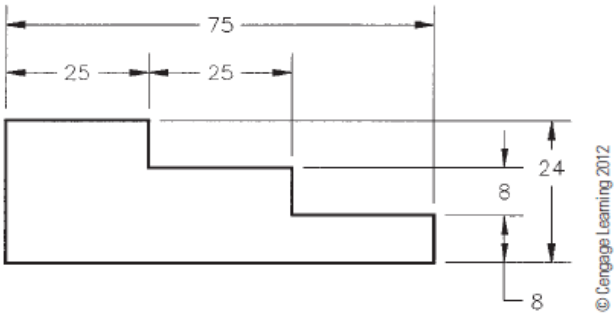


Correct and incorrect dimensioning

Dimensioning Methods

- **Chain Dimensioning**

Chain dimensioning also known as **point-to-point dimensioning**, is a method of dimensioning from one feature to the next. Each dimension is dependent on the previous dimension or dimensions.
The next figure shows the common mechanical drafting practice of providing an overall dimension while leaving one of the intermediate dimensions blank.
Chain dimensioning is commonly used in architectural drafting and related construction industries.

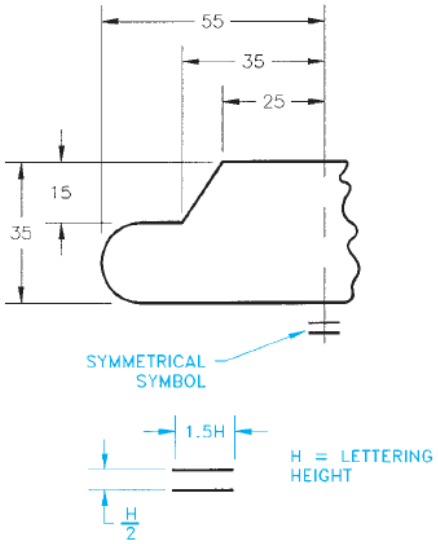


Chain Dimensioning

- **Baseline Dimensioning**

Baseline dimensioning is a common method of dimensioning machine parts whereby each feature dimension originates from a common surface, axis, or center plane. Each
dimension in baseline dimensioning is independent.

The following figure shows how baseline dimensions can be placed symmetrical about a center plane. It also shows the use of the symmetrical symbol. In this application, the baseline dimensions originate from the center plane of the part. The symmetrical symbol is used shows that both sides of the object is symmetrical when the object is too large to fit on the sheet and a portion is broken away. A short break line is used to represent the break. While this practice is an option for very large parts, it should be avoided when possible because the application can cause confusion. Drawing the entire view on a larger sheet is normally preferred.

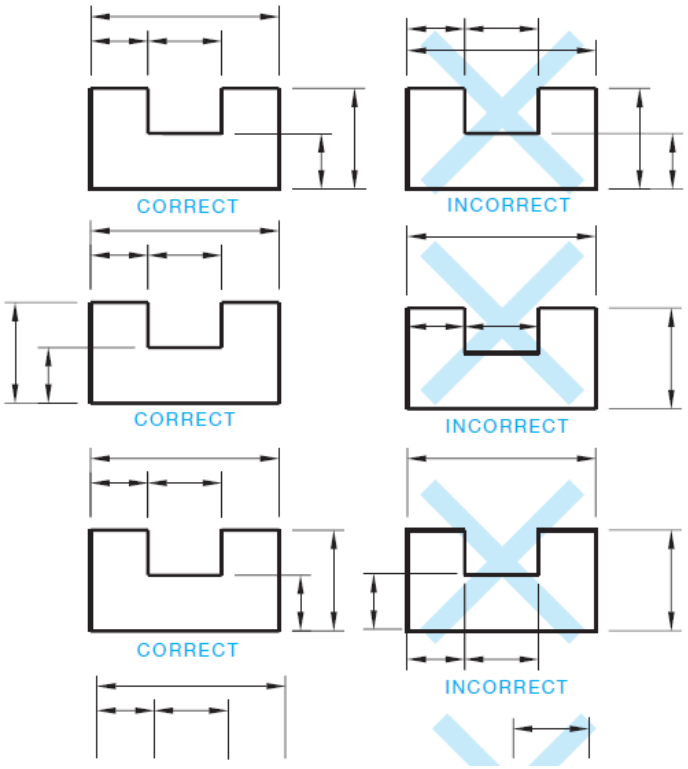


PREFERRED DIMENSIONING PRACTICES

The drawings in the following figure show correct and incorrect dimensioning practices. Good judgment should be used when placing dimensions. Basic rules apply to the placement of dimensions, but you will find situations in the real world when it is impossible to follow the rules. When it is necessary to break a dimensioning rule, do it

The following provides basic guidelines to follow when placing dimensions:

- Avoid crossing extension lines, but *do not* break extension lines when they do cross.
- *Never* cross extension lines over dimension lines. When there is no other solution, break the extension line where it crosses over the dimension line. *Never* break a dimension line.
- Break extension lines when they cross *over or near* an arrowhead, such as when an extension line crosses a leader line near the arrowhead. It is not necessary to break the extension line in other cases. Your CADD drafting application may not allow for breaks in extension lines. Confirm this with your CADD instructions.
- *Avoid* dimensioning over or through the object.
- *Avoid* dimensioning to hidden features.
- *Avoid* unnecessary long extension lines.
- *Avoid* using any line of the object as an extension line.
- Dimension *between views* when possible.
- *Group* adjacent dimensions.
- Dimension to views that provide the *best shape description.*

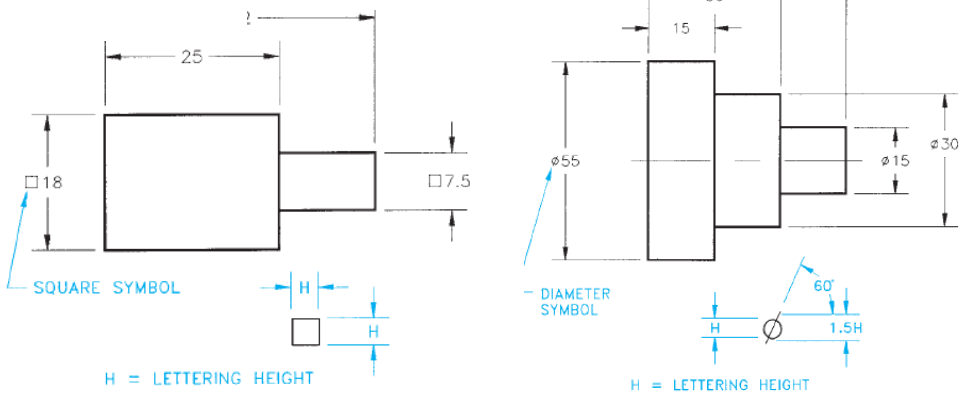


Correct and incorrect dimensioning examples.

Dimensioning Symmetrical Objects, Cylinders, and Square Features

Dimension cylindrical shapes in the view where the cylinders appear rectangular. The diameters are identified by the diameter symbol, and the circular view can be omitted.

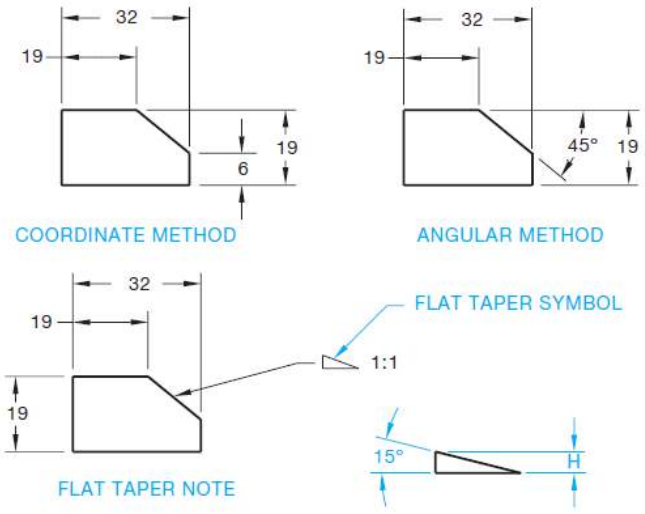
Square features are dimensioned in a similar manner using the square symbol shown in the figure



Dimensioning Angles

Angular surfaces can be dimensioned as coordinates, as angles in degrees, or as a flat taper (see Figure). Angles are measured in degrees using the degree symbol (°). There are 360° in a circle. Each degree contains 60 minutes. Minutes are identified with the minute symbol ('). Each minute has 60 seconds, identified with the seconds symbol (").

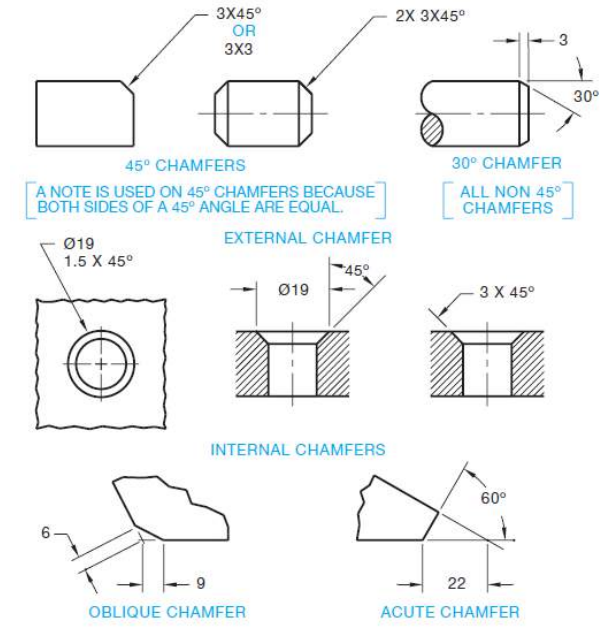
Notice in the figure that the dimension line for the 45° angle is drawn as an arc. The radius of this arc is centered at the vertex of the angle.



Dimensioning angular surfaces and flat taper symbol.

Dimensioning Chamfers

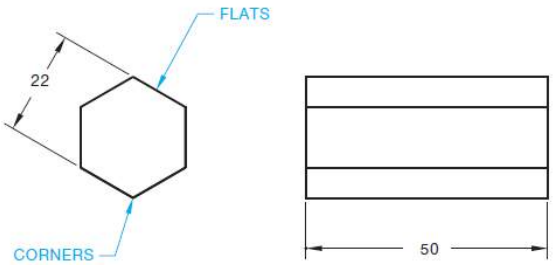
A **chamfer** is a slight surface angle used to relieve a sharp corner.
Chamfers of 45° are dimensioned with a note, while other chamfers require an angle and size dimension, or two size dimensions as shown in Figure. A note is used on 45° chamfers because both sides of a 45° angle are equal. When placing the 45° chamfer note, the size is followed by the (X) symbol and then the 45° angle without spaces. For example, 3X45°. Both sides of a 45° angle are equal. For this reason, another option is to place the value for the dimension of the sides in the note and leave out the 45° angle, such as 3X3.



Dimensioning chamfers

Dimensioning Hexagons and Other Polygons

Dimension **Hexagons** and other polygons across the flats in the views where the true shape is shown. Provide a length dimension in the adjacent view.

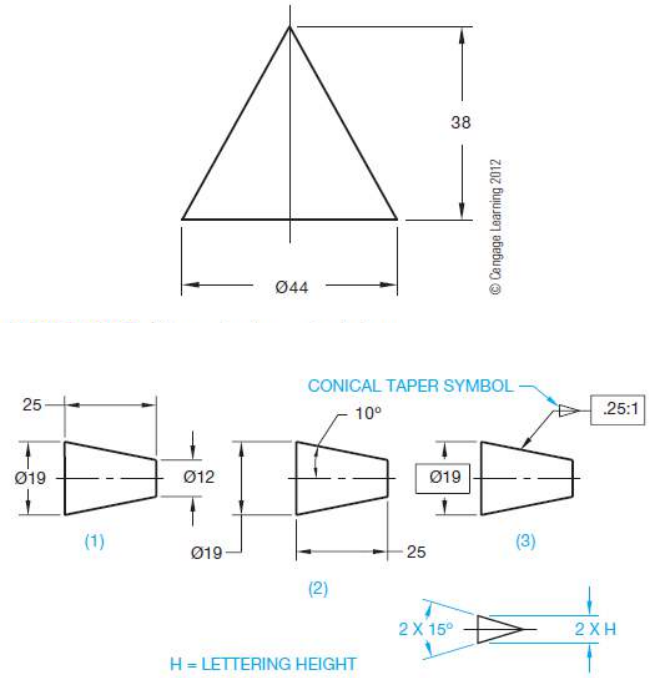


Dimensioning Hexagons

Dimensioning Conical Shapes

Conical shapes should be dimensioned when possible in the view where the cone appears as a triangle as shown in the figure

The circular view can be omitted because the base is dimensioned with a diameter. A conical taper can be treated in one of three possible ways, as shown in the figure. The circular views are omitted in these examples.



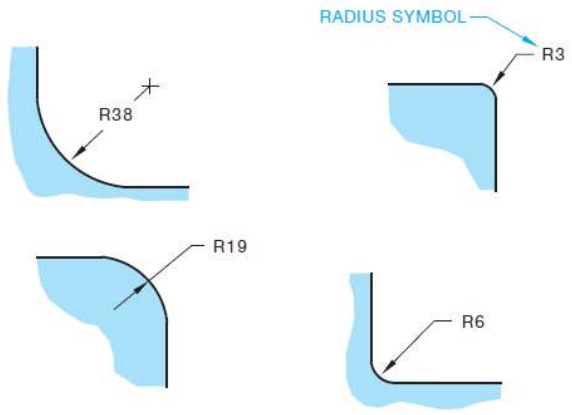
3 Ways of Dimensioning Conical Shapes

Dimensioning Arcs

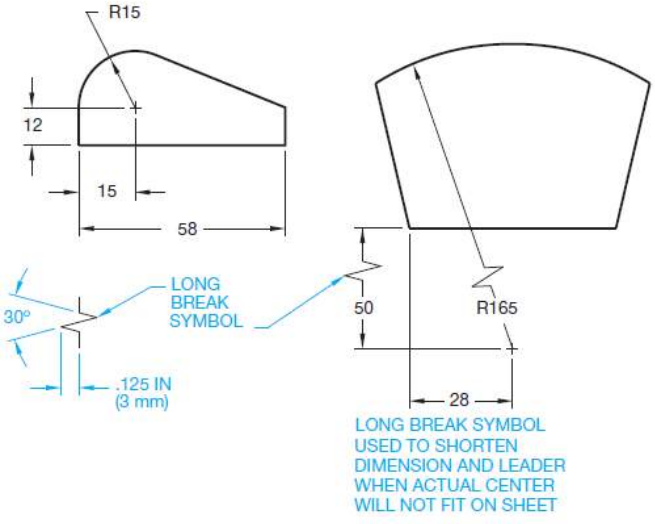
Arcs are dimensioned with leaders and radius dimensions in the views where they are shown as arcs. The leader can extend from the center to the arc or point to the arc as shown in the figures. The letter *R* precedes all radius dimension values. Depending on the situation, arcs can be dimensioned with or without their centers located. It is common to leave the center marks off small arcs and the arc center location is not dimensioned. This depends on the specific application and the company or school preference.

Figure 2 shows a very large arc with the center moved closer to the object. To save space, a break line is used in the leader and in the shortened locating dimension to indicate that the dimension is not in true length. The given dimension value is the accurate location dimension.

A design can be created that has a series of tangent arcs. When this is done, the curved outline made up of two or more arcs is dimensioned by providing the radii of all arcs and locating the arc centers with coordinate dimensions.



Dimensioning arcs with no centers located and using the radius symbol.



Dimensioning arcs with centers located and using the long break symbol for a very long radius dimension.

Freehand sketching is an effective way to get an idea across when words fail. In this way, graphic language becomes an important aid to verbal language. It provides a quick and easy way to convey new ideas between engineers. Most original engineering ideas or inventions are recorded for the first time in the form of a sketch. Usually, engineers start new designs using freehand sketches. These sketches are then conveyed to drafters for more detailed drawings.

Sketches are often used instead of complete drawings where changes of design must be made quickly. The greatest use of sketches, however, is in formulating, expressing, and recording new ideas.

Freehand sketching required a pencil, a soft eraser, and some papers. Grid papers are very useful in order to align the drawings of multiple views, as shown in Figure (1).

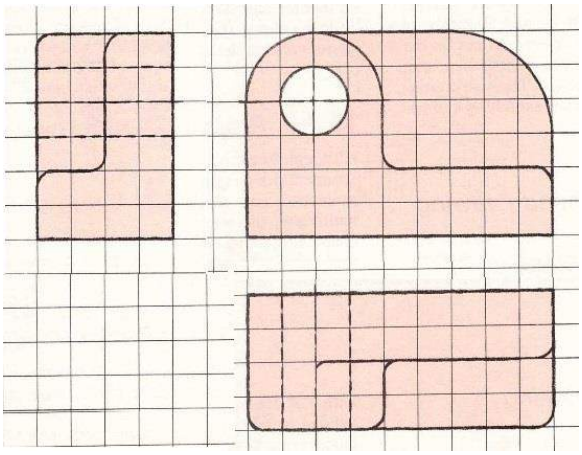


Figure 1 Sketch of Multiple Views on Grid Paper

1- Types of Sketch Lines

Freehand lines differ in their appearance from mechanical lines. Mechanical lines are done using tools. An effort should be made in order to draw freehand lines straight and uniform in weight. Figure (2) shows the different types of lines used in freehand sketching from the thickness and lightness required.

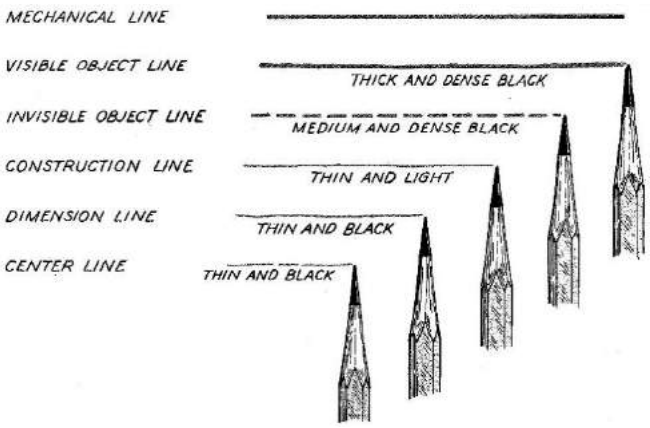


Figure 2 Types of Sketch Lines

2- Straight Lines

When sketching a straight line, it is advisable to first mark the end points with very light dots or small crosses, while horizontal lines are sketched from left to right as shown in Figure (3). Keep your eye on the point toward which you are drawing, not on the pencil point. Steps for sketching a straight line are as follows:

1. Mark the end points.
2. Make few trial motions between the end points.
3. Sketch a very light line between the two end points.
4. Darken the finished line in one stroke. It should be distinct, black, and uniform.

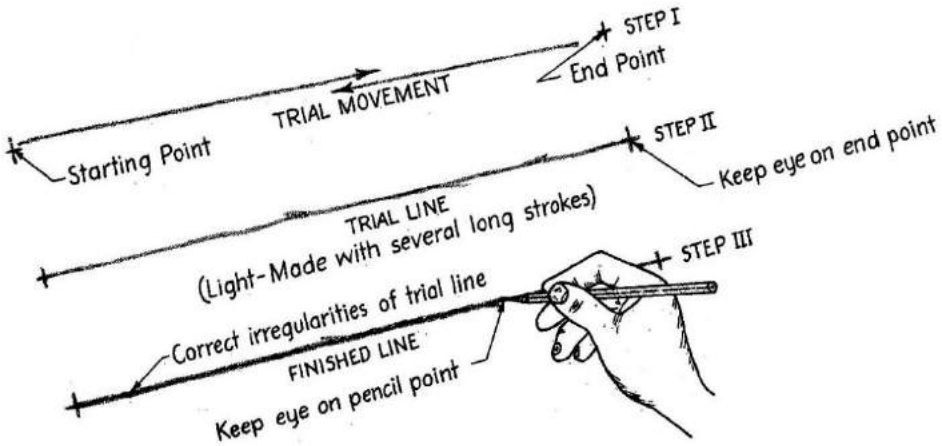


Figure 3 Steps for Sketching a Straight Line

Sketching straight lines with different angles are shown in Figure (4).

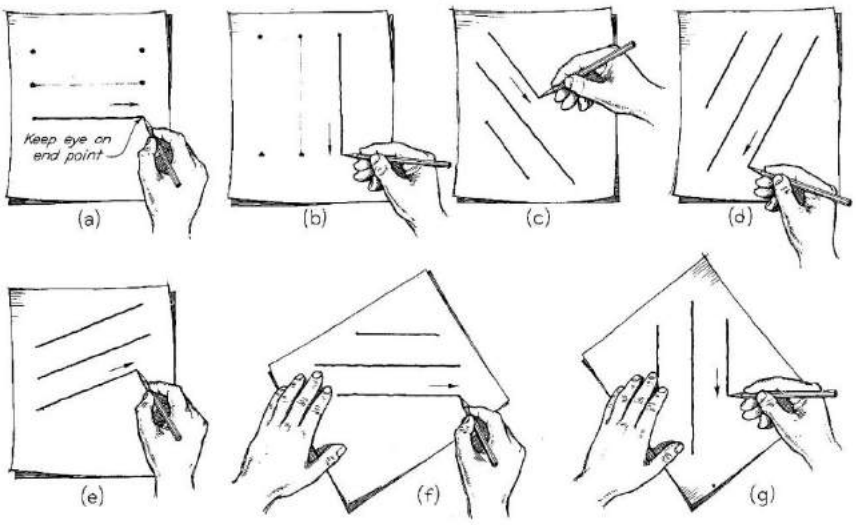


Figure 4 Sketching Straight Lines with Different Angle

3- Sketching Circles

Circles can be divided into small and large circles. Small circles can be sketched in one motion by first making radial distances on perpendicular center lines. While large circles additional radial distances on 450 are required. Figure (5) shows the steps required to draw a circle using eight sketch rays at 450. The steps required for larger circles are shown in Figure (6).

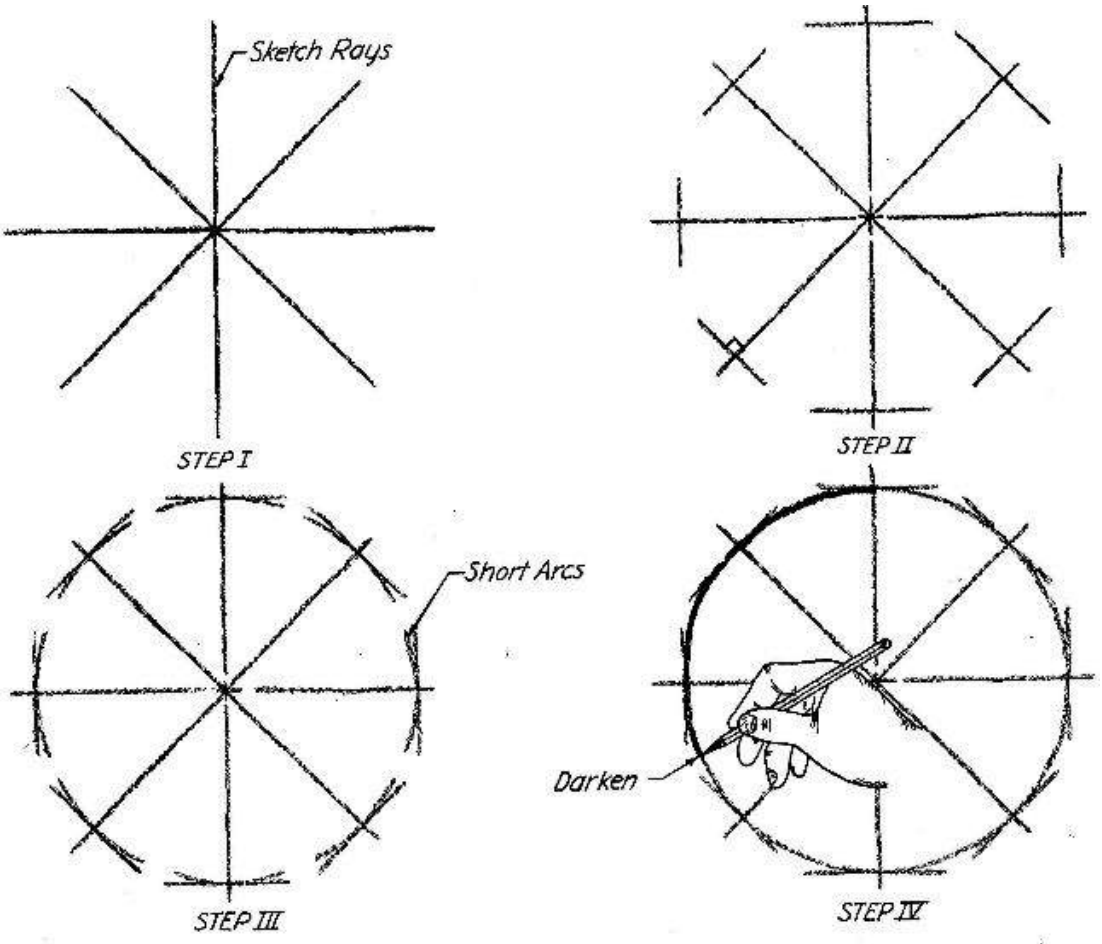


Figure 5 Steps for Sketching a Circle

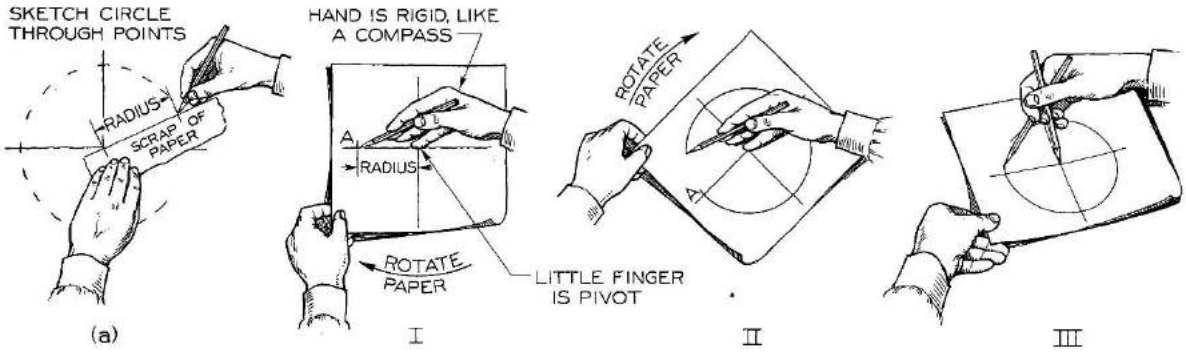


Figure 6 Steps for Sketching Larger Circle

4- Sketching Ellipses

Ellipses have two different axes. The long axis of an ellipse is called major axis while the short one is called the minor axis. To sketch an ellipse, Figure (7), (a) lightly sketch the enclosing rectangle and mark the approximate mid-points of the sides. Then (b), sketch light tangent arcs at the mid-points, and complete the ellipse, (c). A second method is to start with the major and minor axes and sketch the ellipse as shown in (d) and (f).

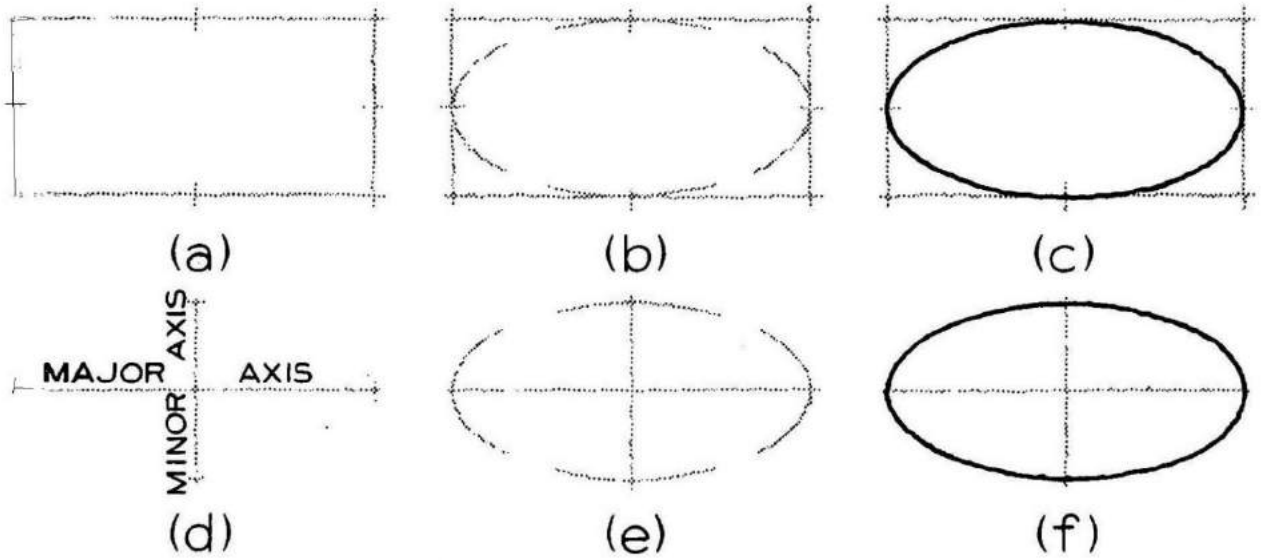


Figure 7 Steps for Sketching Ellipses

5- Sketching a complete View

In Figure (8), it is required to look at the "Lock Plate" from the direction of the arrow indicated in order to draw a view. Three steps are proposed. I) Sketch main areas lightly to keep the proportions of the entire view. II) Lightly construct the arcs and circles. III) Then heavy-in all final lines, making them clean-cut and dark.

Figure (9) shows another example of sketching a different model using grid paper.

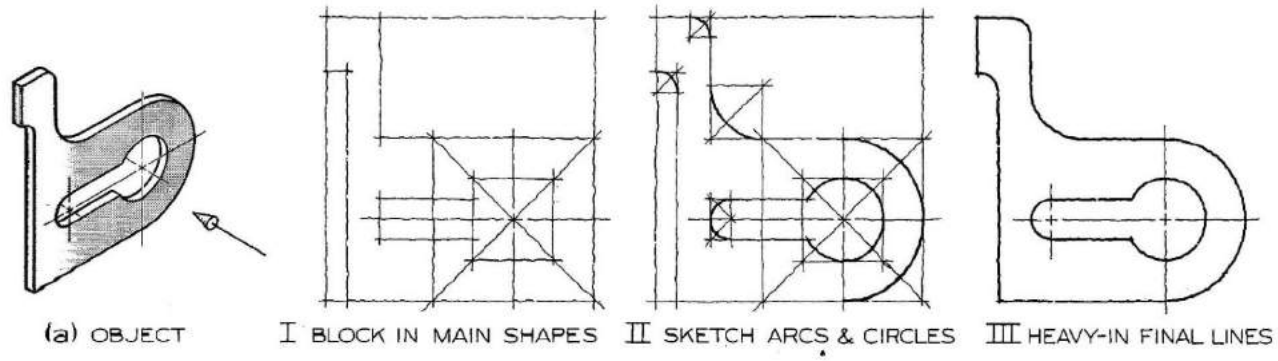


Figure 8 Steps for Sketching a Complete View

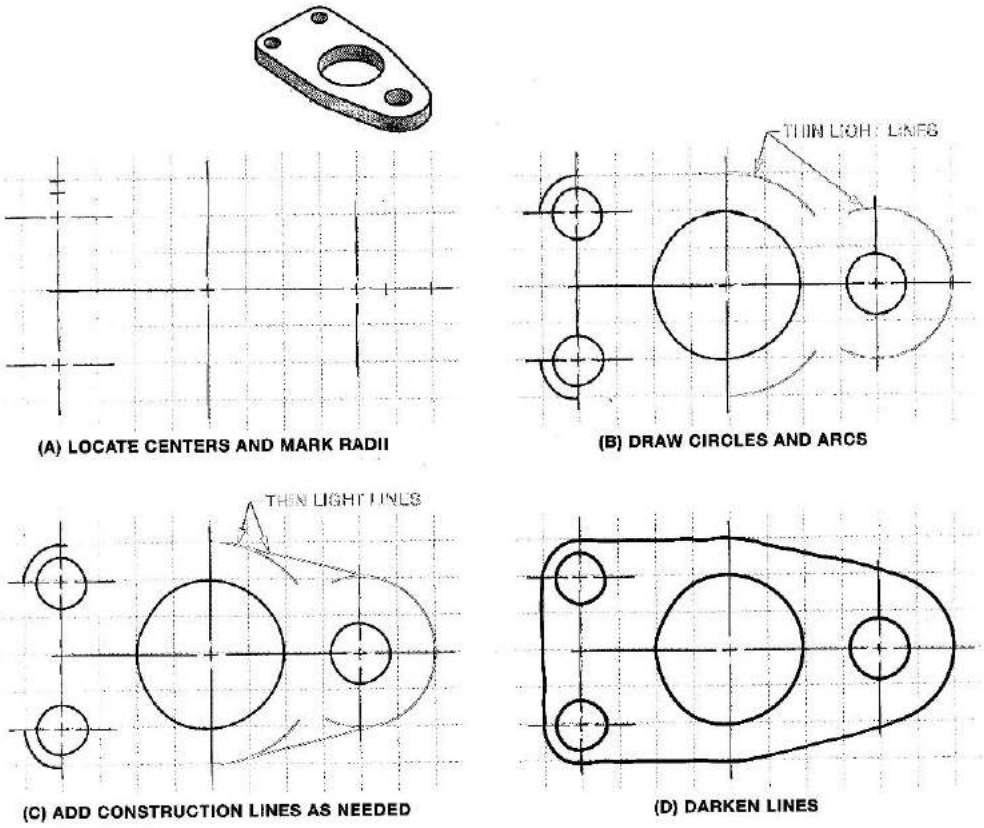
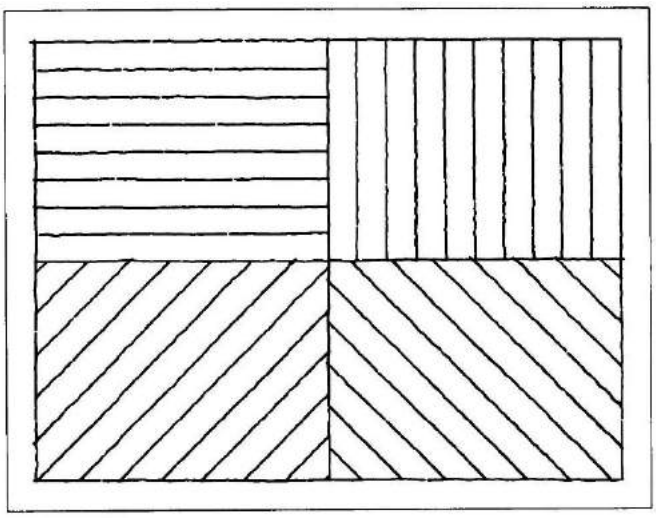
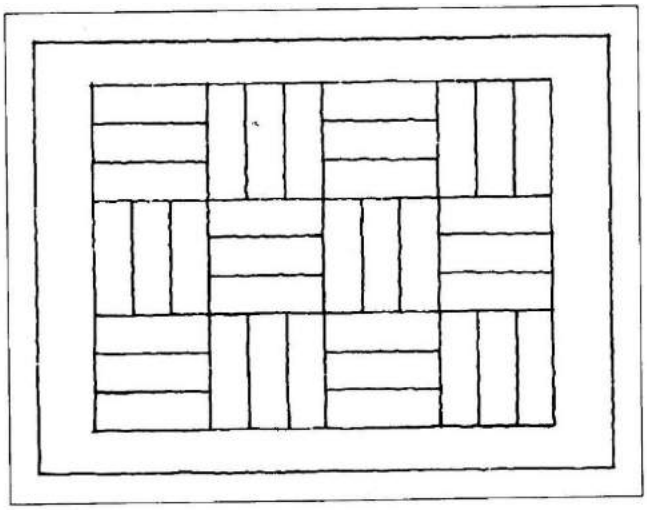
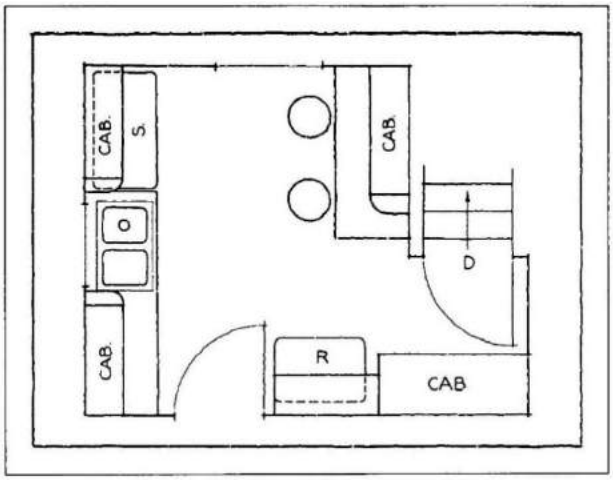
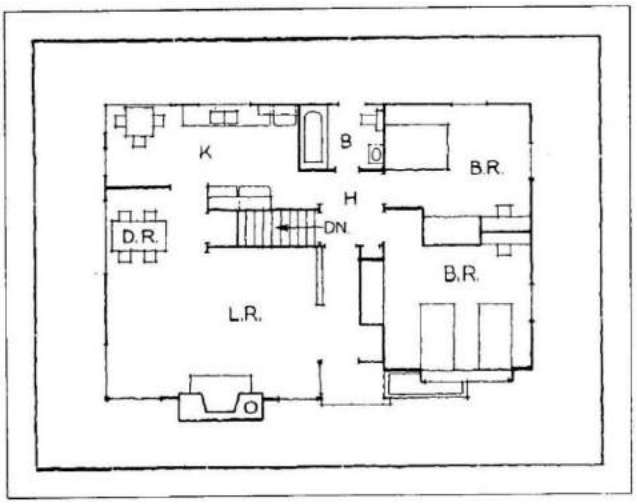


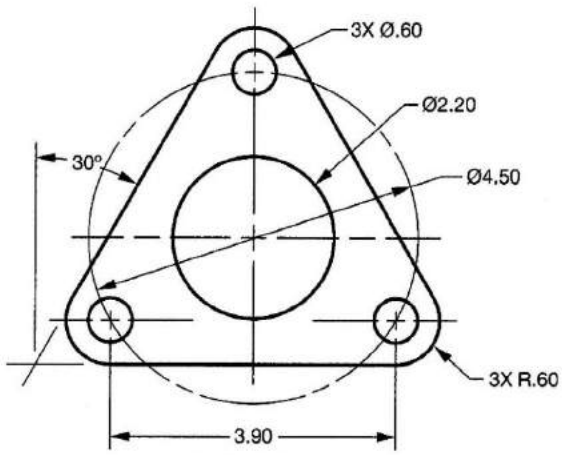
Figure 9 Steps for Sketching a Complete View Using Grid Paper



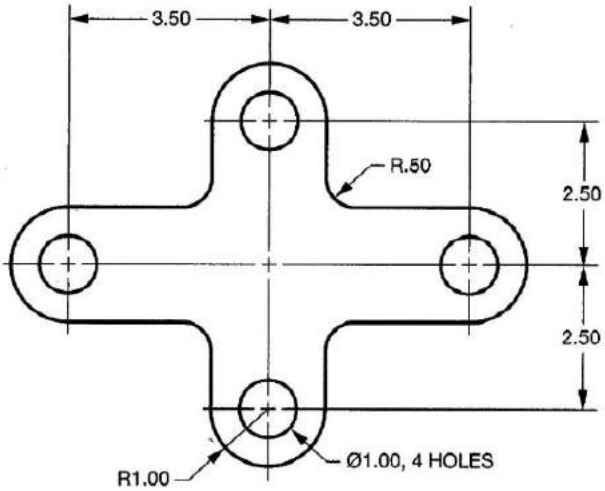




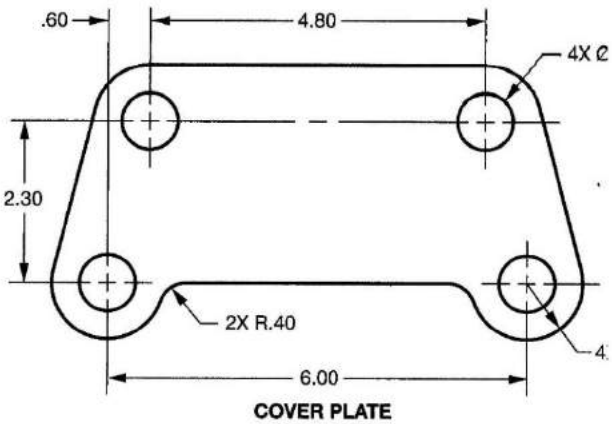




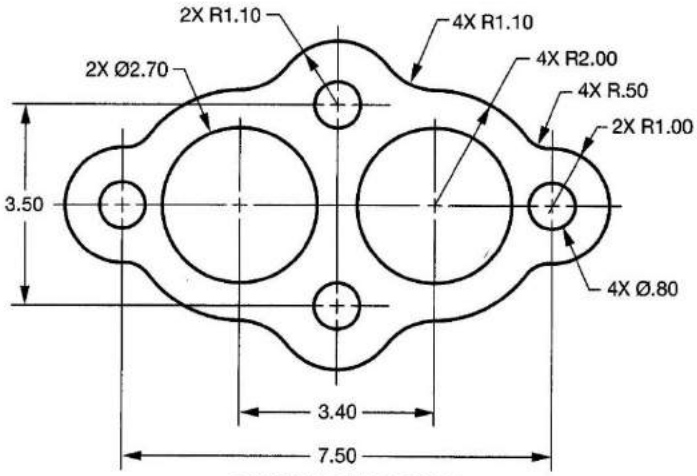
BASE PLATE



OFFSET LINK



COVER PLATE



CARBURETOR GASKET

This chapter deals with problems on geometrical construction which are mostly based on plane geometry and which are very essential in the preparation of engineering drawings. The solution of these geometric problems is pure graphical using the drawing tools. Geometry is the basis of all technical drawings. The knowledge of the principles of geometric construction and its applications are essential to engineers. An engineer, must know how to draw various types of lines which can be a straight line, a circle, an arc of a circle, a circular curve etc.

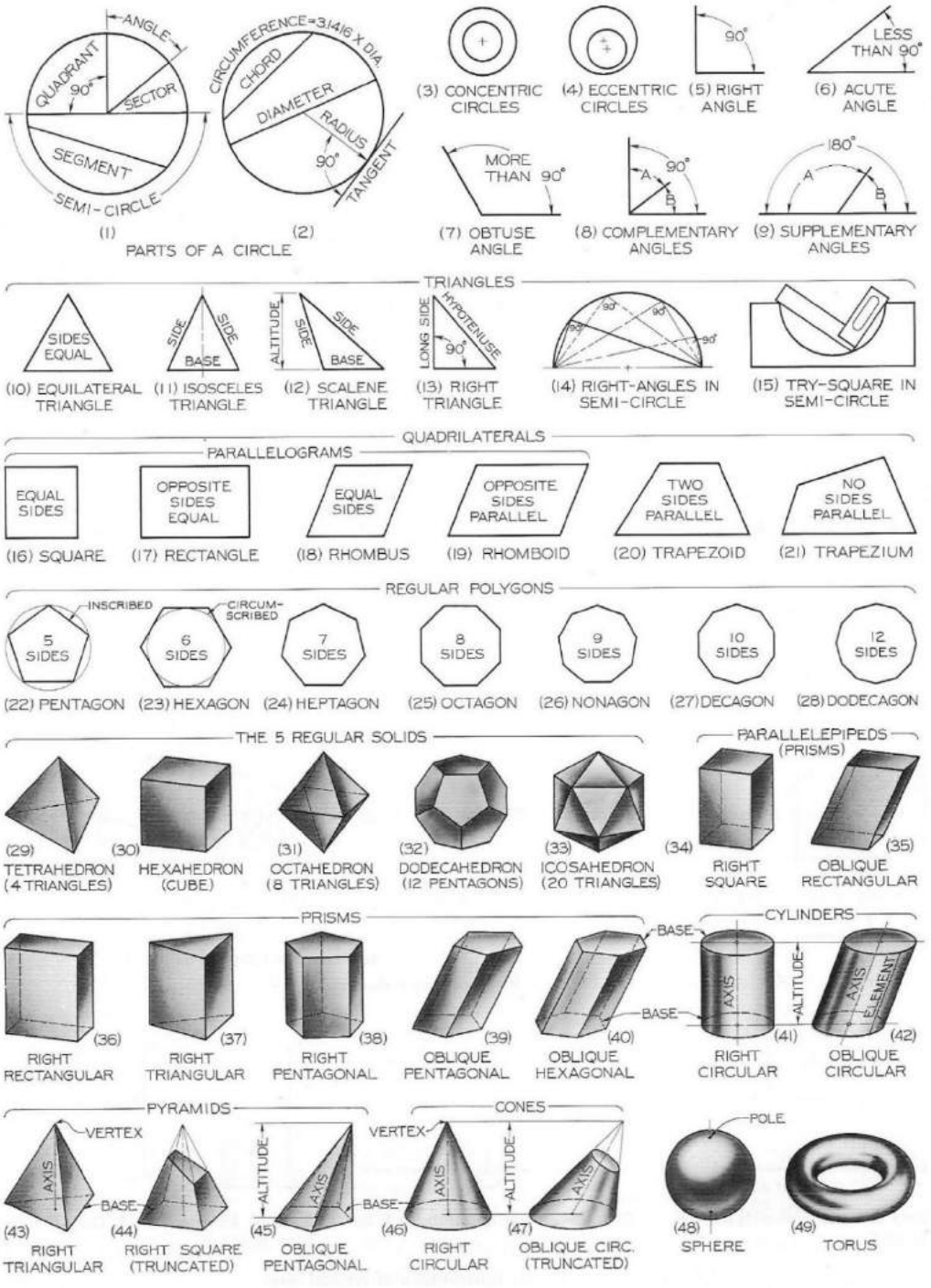


Figure 1 Geometric Shapes

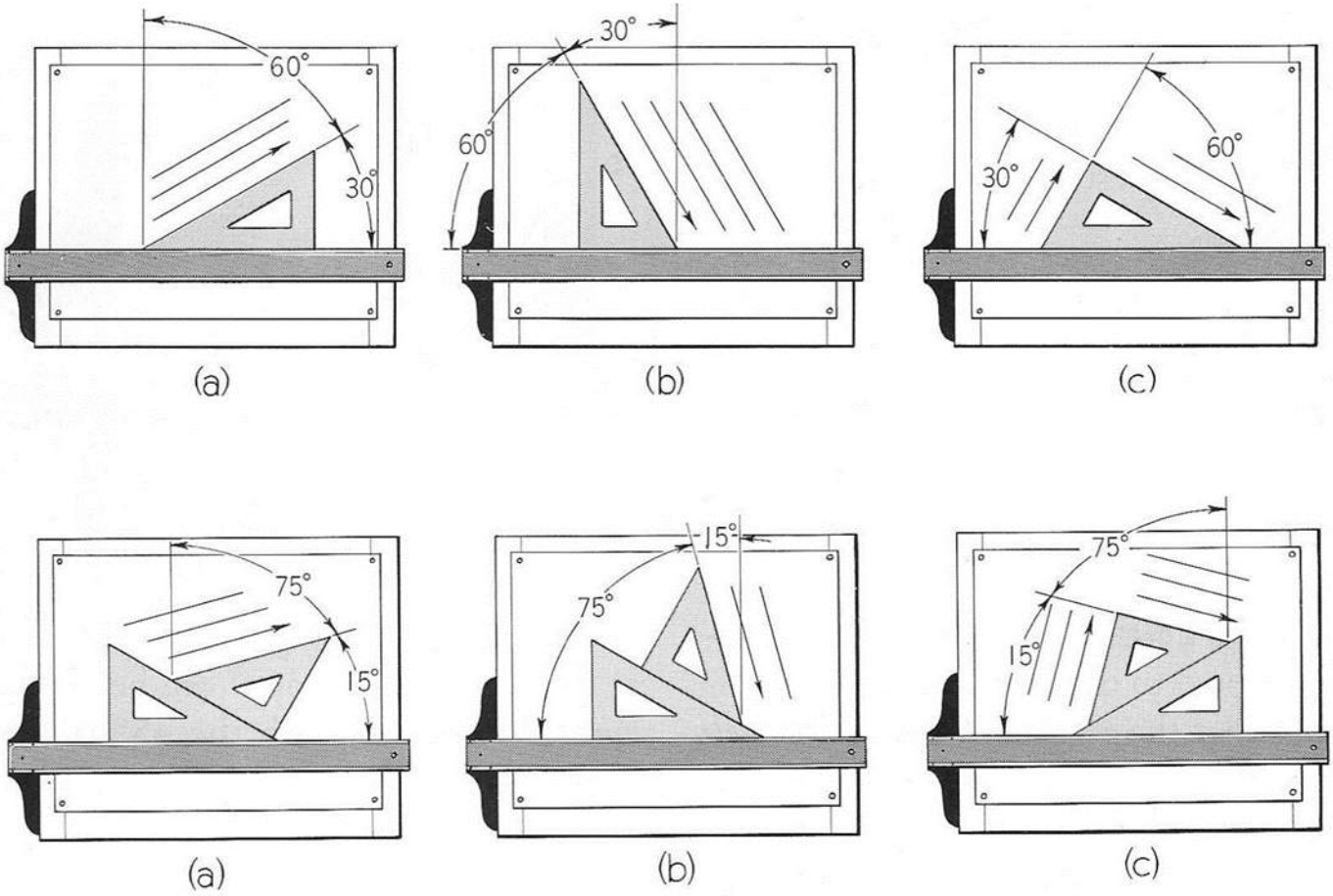


Figure 2 Different Angles Done by Triangles

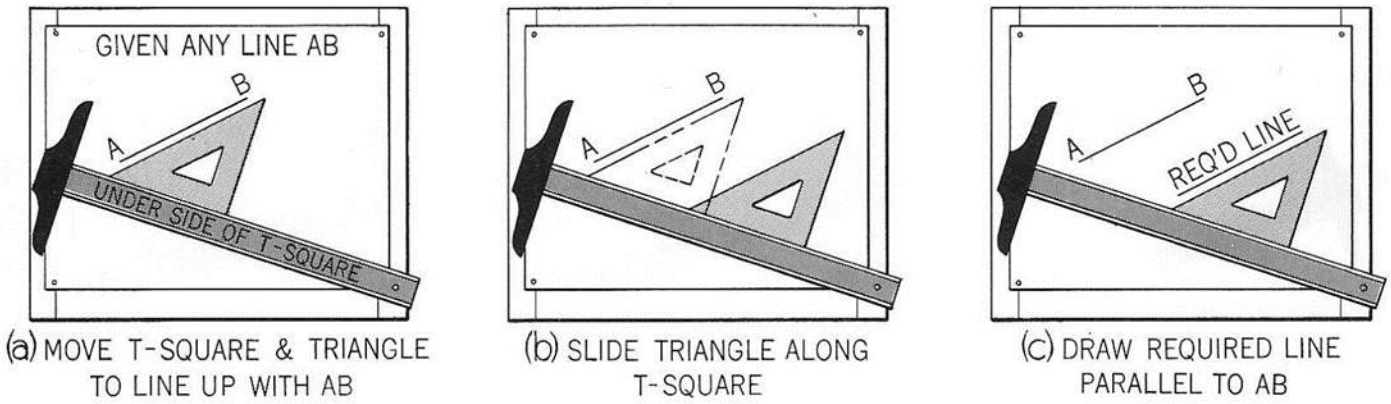


Figure 3 Drawing Parallel Lines Using Triangles

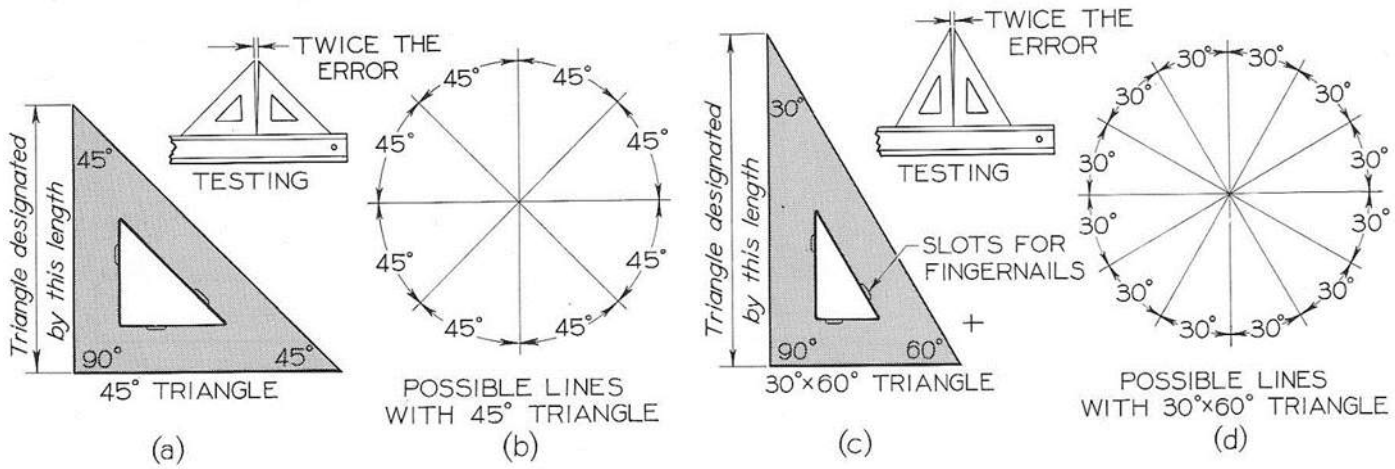
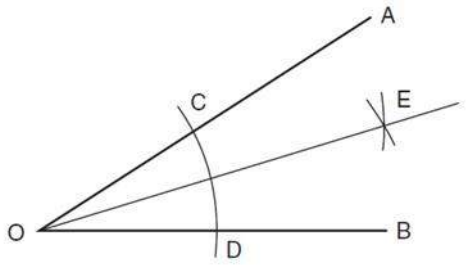
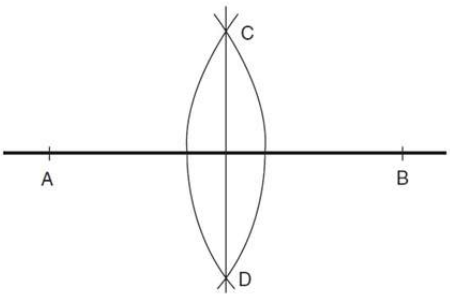
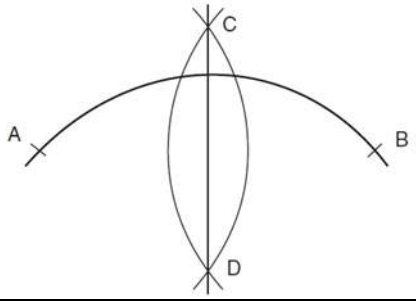


Figure 4 Possible Line Angles Using Triangles

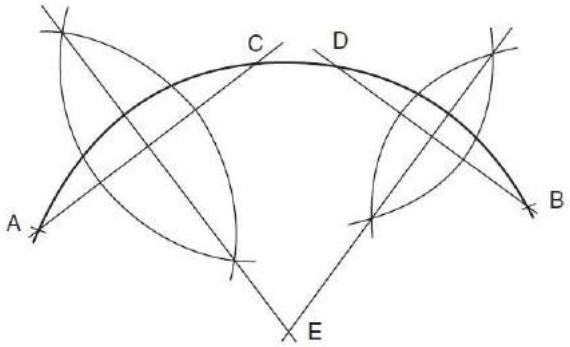
Different Geometric Construction Operations

To bisect a given angle AOB 1. With centre O, draw an arc to cut OA at C and OB at D. 2. With centres C and D, draw equal radii to intersect at E. Line OE bisects angle AOB.	
To bisect a given straight line AB 1. With centre A and radius greater than half AB, describe an arc. 2. Repeat with the same radius from B, the arcs intersecting at C and D. Join C to D and this line will be perpendicular to and bisect AB.	
To bisect a given arc AB 1. From centre A and with a radius greater than half AB, describe an arc. 2. Repeat with the same radius from B, the arcs intersecting at C and D. 3. Join C to D to bisect the arc AB.	

To find the centre of a given arc AB

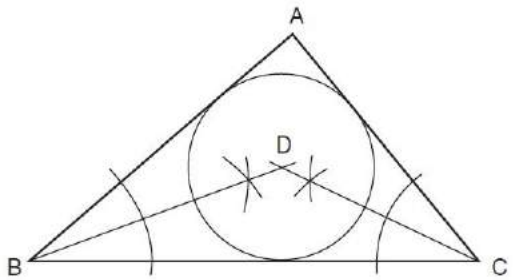
1. Draw two chords, AC and BD.
2. Bisect AC and BD as shown; the bisectors will intersect at E.

The centre of the arc is point E.



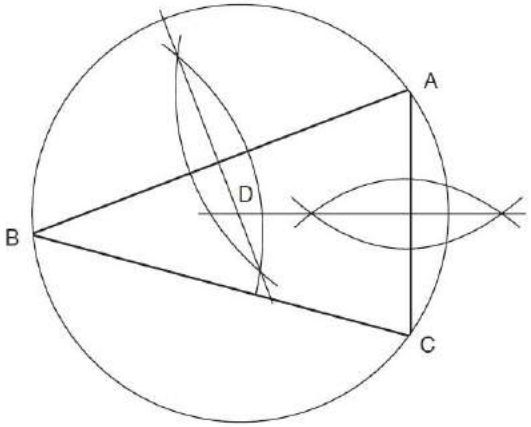
To inscribe a circle in a given triangle ABC

1. Bisect any two of the angles as shown so that the bisectors intersect at D.
2. The centre of the inscribed circle is point D.



To circumscribe a circle around triangle ABC

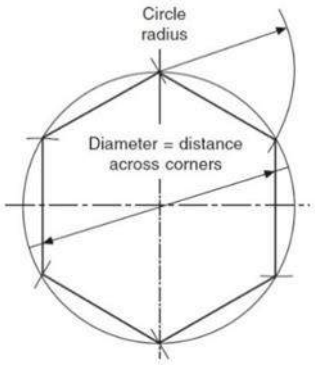
1. Bisect any two of the sides of the triangle as shown, so that the bisectors intersect at point D.
2. The centre of the circumscribing circle is point D.



To draw a hexagon, given the distance across the corners (the corners mean there are 2 or more other corners in between so that the connecting line passes through the centre point)

Method A

1. Draw vertical and horizontal centre lines and a circle with a diameter equal to the given distance.
2. Step off the radius around the circle to give six equally spaced points, and join the points to give the required hexagon.

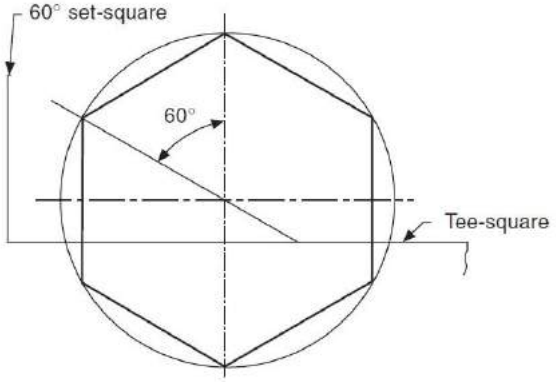


To draw a hexagon, given the distance across the corners

Method B

1. Draw vertical and horizontal centre lines and a circle with a diameter equal to the given distance.
2. With a 60° set-square, draw points on the circumference 60° apart.

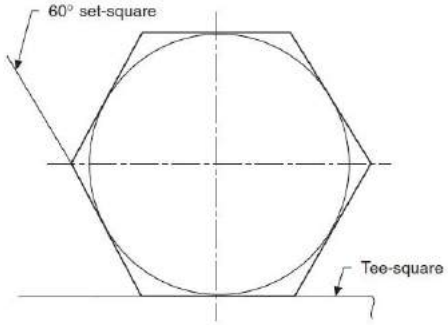
Connect these six points by straight lines to give the required hexagon.



To draw a hexagon, given the distance across the flats

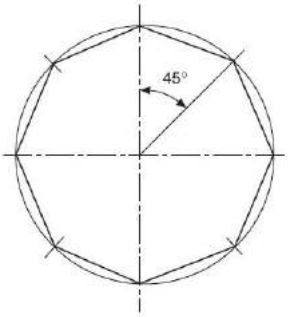
1. Draw vertical and horizontal centre lines and a circle with a diameter equal to the given distance.

Use a 60° set-square and tee-square as shown, to give the six sides.



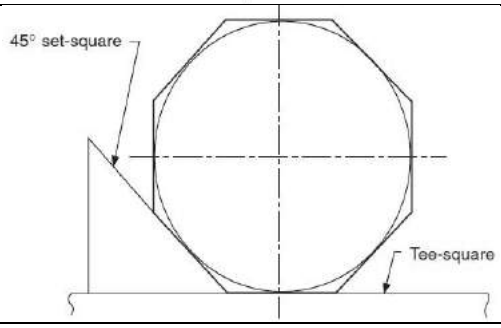
To draw a regular octagon, given the distance across corners

Repeat the instructions but use a 45° setsquare, then connect the eight points to give the required octagon.



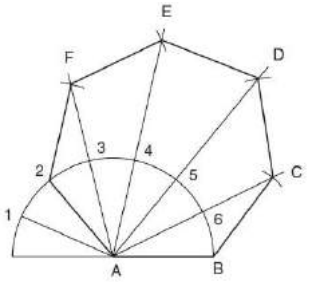
To draw a regular octagon, given the distance across the flats

Repeat the instructions but use a 45° setsquare to give the required octagon.



To draw a regular polygon, given the length of the sides

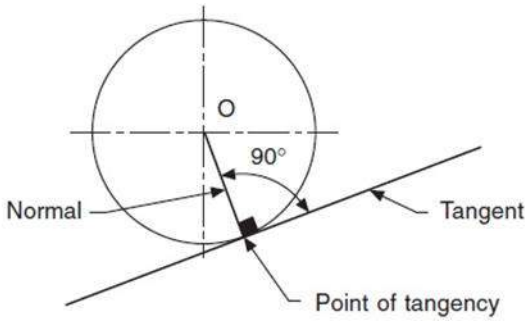
Note that a regular polygon is defined as a plane figure which is bounded by straight lines of equal length and which contains angles of equal size. Assume the number of sides is seven in this example.



1. Draw the given length of one side AB, and with radius AB describe a semi-circle.
2. Divide the semi-circle into seven equal angles, using a protractor, and through the second division from the left join line A2.
3. Draw radial lines from A through points 3, 4, 5, and 6.
4. With radius AB and centre on point 2, describe an arc to meet the extension of line A3, shown here as point F.
5. Repeat with radius AB and centre F to meet the extension of line A4 at E.
6. Connect the points as shown, to complete the required polygon.

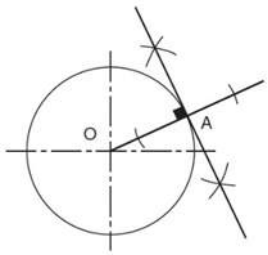
To draw a tangent to a point A on the circumference of a circle, centre O

If a disc stands on its edge on a flat surface it will touch the surface at one point. This point is known as the point of tangency, as shown in the figure and the straight line which represents the flat plane is known as a tangent. A line drawn from the point of tangency to the centre of the disc is called a normal, and the tangent makes an angle of 90° with the normal.
The following constructions show the methods of drawing tangents in various circumstances.



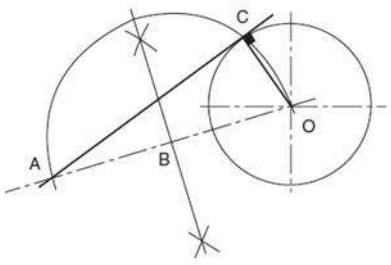
To draw a tangent to a point A on the circumference of a circle, centre O

Join OA and extend the line for a short distance. Erect a perpendicular at point A by the method shown.



To draw a tangent to a circle from any given point A outside the circle

Join A to the centre of the circle O. Bisect line AO so that point B is the mid-point of AO. With centre B, draw a semi-circle to intersect the given circle at point C. Line AC is the required tangent.

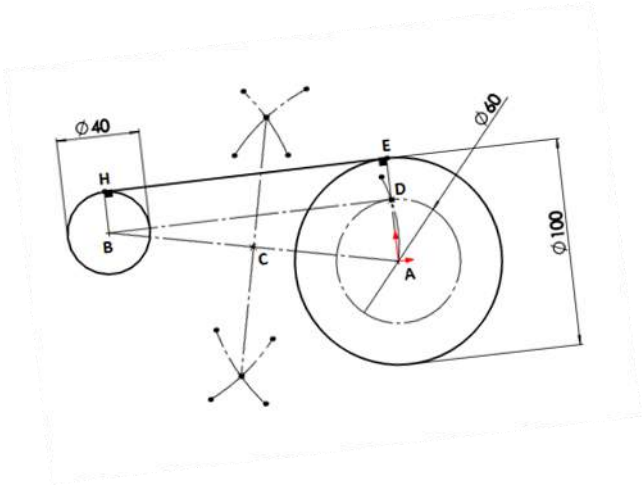


To draw an external tangent to two Circles

1. Join the centres of the circles by line AB, bisect AB
2. Draw a circle from point A with Radius equal to the difference between the 2 radii.
3. From point C, draw an arc with radius equal to CA and intersects with the new circle at point D

Note that BD is tangent to the new circle at point D

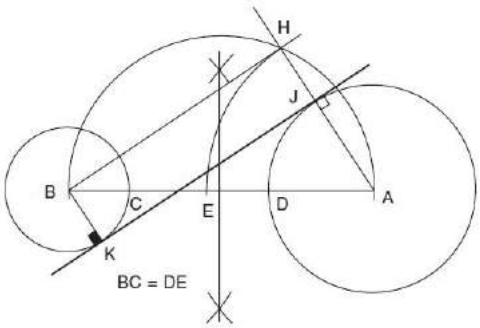
4. Draw a line from A that goes through D and intersects with the large circle at point E
5. From point B draw a line normal to BD and intersects with the small circle at H
6. Line EH is the required tangent line



To draw an internal tangent to two circles

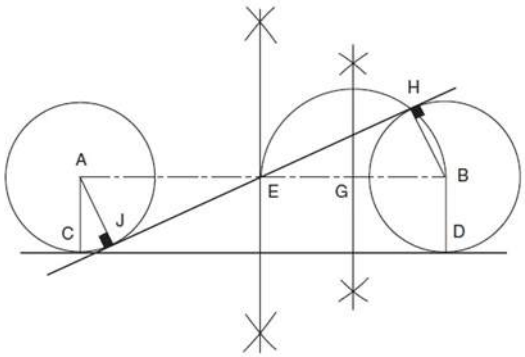
1. Join the centres of the circles by line AB, bisect AB and draw a semi-circle from the midpoint of AB with R equal to half AB.
2. From point A, with R equal to the sum of the 2 radii (the small and large ones), draw an arc to cut the semi-circle in H.
3. Join AH; this line crosses the larger circle circumference at J.
4. Draw line BH and from J draw a line parallel to BH to touch the smaller circle at K. Line JK is the required tangent.

Note that angle AHB lies in a semi-circle and will therefore, be 90°. AJ and BK are normals.



To draw internal and external tangents to two circles of equal diameter

Join the centres of both circles by line AB. Erect perpendiculars at points A and B to touch the circumferences of the circles at points C and D. Line CD will be the external tangent.
Bisect line AB to give point E, then bisect BE to give point G. With radius BG (and from point G), describe a semi-circle to cut the circumference of one of the given circles at H. Join HE and extend it to touch the circumference of the other circle at J. Line HEJ is the required tangent. Note that again the angle in the semi-circle, BHE, will be 90°, and hence BH and AJ are normals.

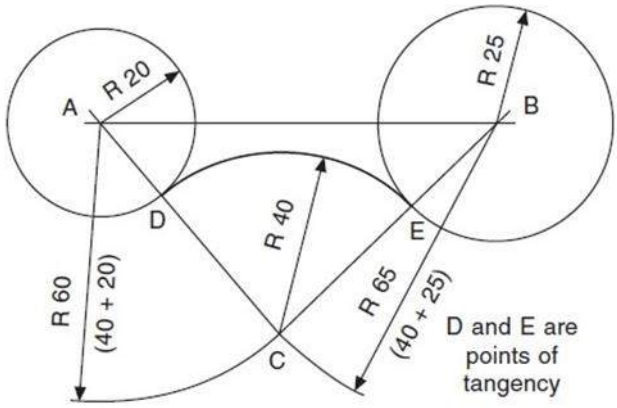


To draw a curve of given radius to touch two circles when the circles are outside the radius

Assume that the radii of the given circles are 20 and 25 mm, spaced 85 mm apart, and that the radius to touch them is 40 mm.

With centre A. describe an arc equal to 20 + 40 = 60 mm.
With centre B, describe an arc equal to 25 + 40 = 65 mm.
The above arcs intersect at point C. With a radius of 40 mm, describe an arc from point C as shown,

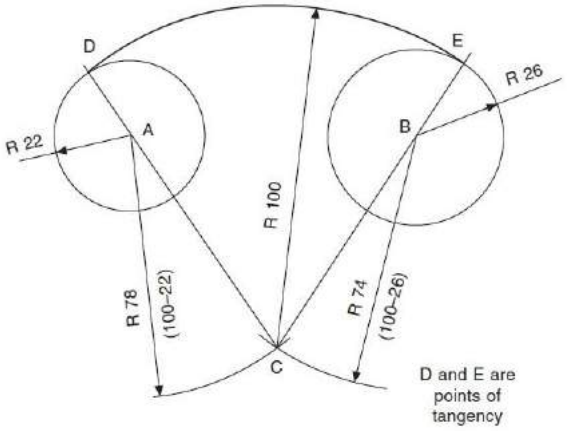
Note that the points of tangency between the arcs lie along the lines joining the centres AC and BC. It is particularly important to note the position of the points of tangency before lining in engineering drawings, so that the exact length of an arc can be established.



To draw a curve of given radius to touch two circles when the circles are inside the radius

Assume that the radii of the given circles are 22 and 26 mm, spaced 86 mm apart, and that the radius to touch them is 100 mm.

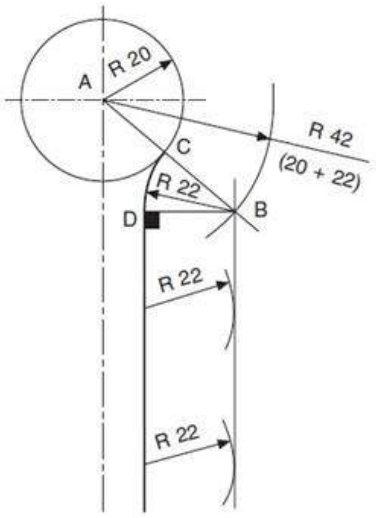
With centre A, describe an arc equal to 100 – 22 = 78 mm.
With centre B, describe an arc equal to 100 – 26 = 74 mm.
The above arcs intersect at point C. With a radius of 100 mm, describe an arc from point C, and note that in this case the points of tangency lie along line CA extended to D and along line CB extended to E.



To draw a radius to join a straight line and a given circle

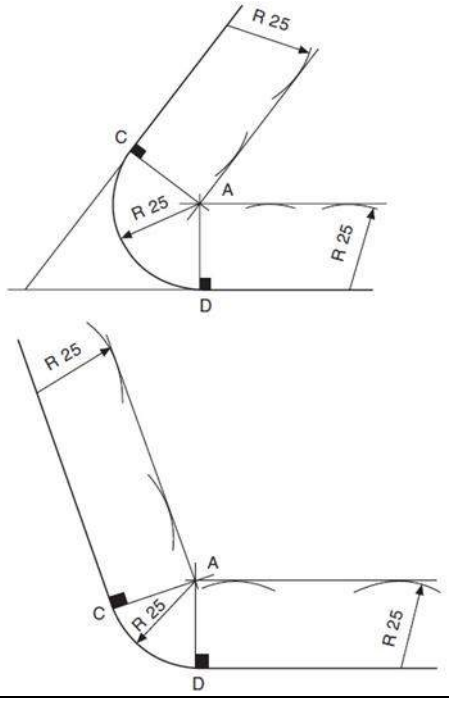
Assume that the radius of the given circle is 20 mm and that the joining radius is 22 mm.
With centre A, describe an arc equal to 20 + 22 = 42 mm.
Draw a line parallel to the given straight line and at a perpendicular distance of 22 mm from it, to intersect the arc at point B.

With centre B, describe the required radius of 22 mm, and note that one point of tangency lies on the line AB at C; the other lies at point D such that BD is at 90° to the straight line.



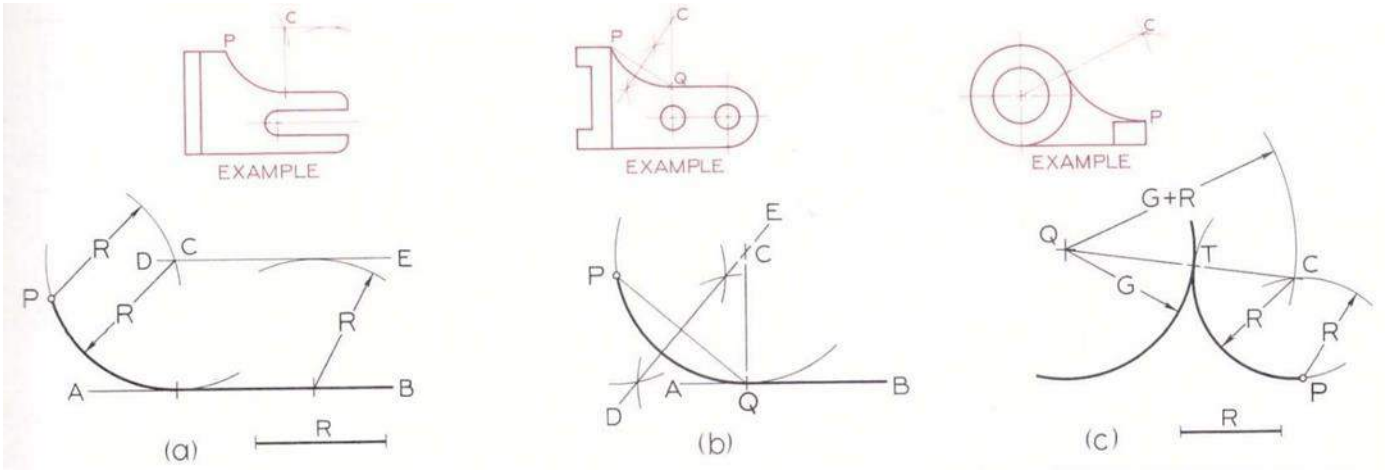
To draw a radius which is tangential to given straight lines

Assume that a radius of 25 mm is required to touch the lines shown in the figures. Draw lines parallel to the given straight lines and at a perpendicular distance of 25 mm from them to intersect at points A. As above, note that the points of tangency are obtained by drawing perpendiculars through the point A to the straight lines in each case.

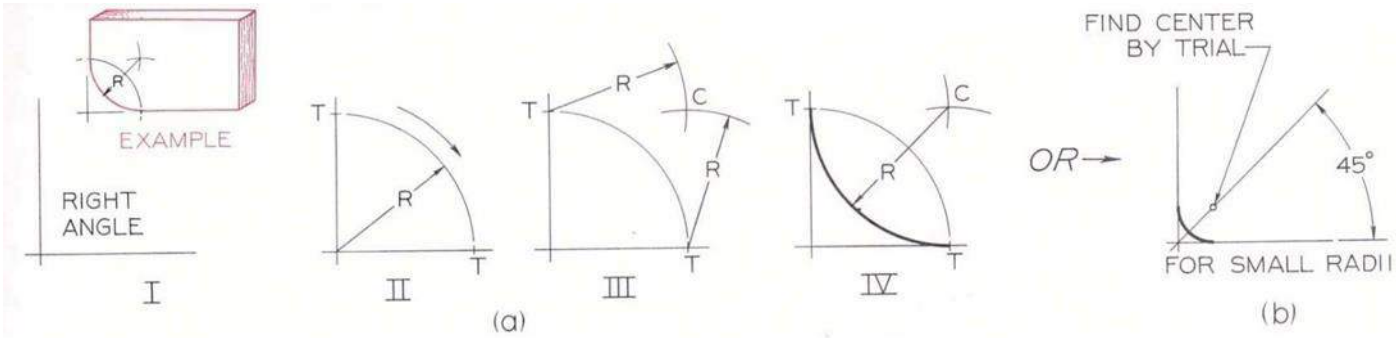


Examples

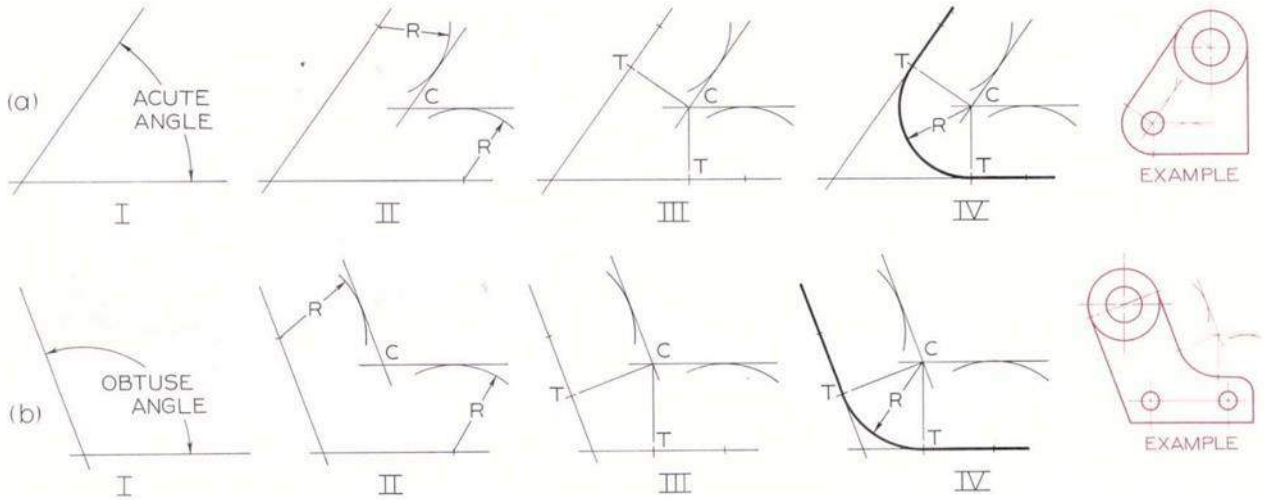
Example (1):



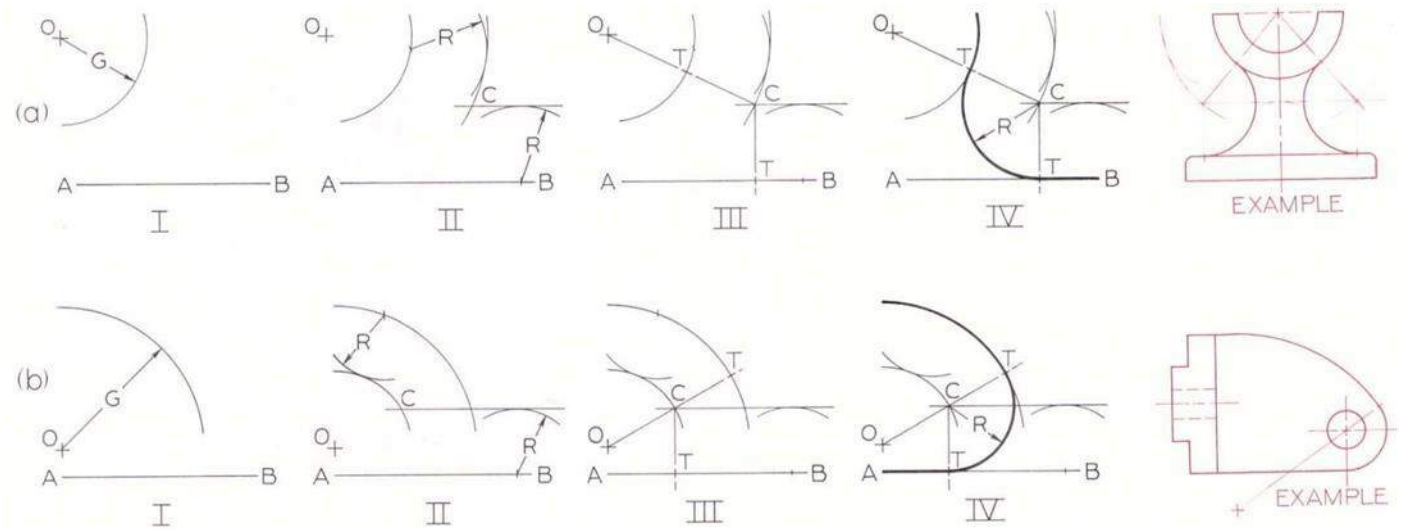
Example (2):



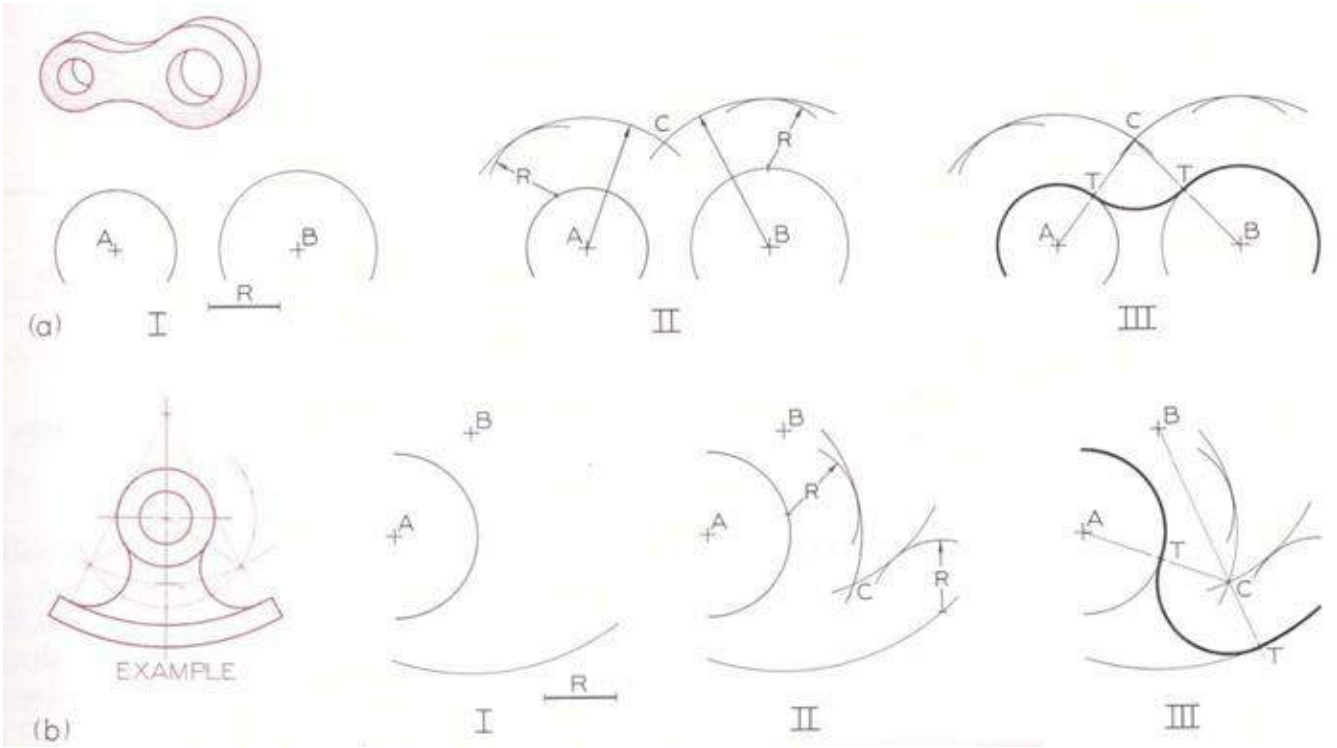
Example (3):



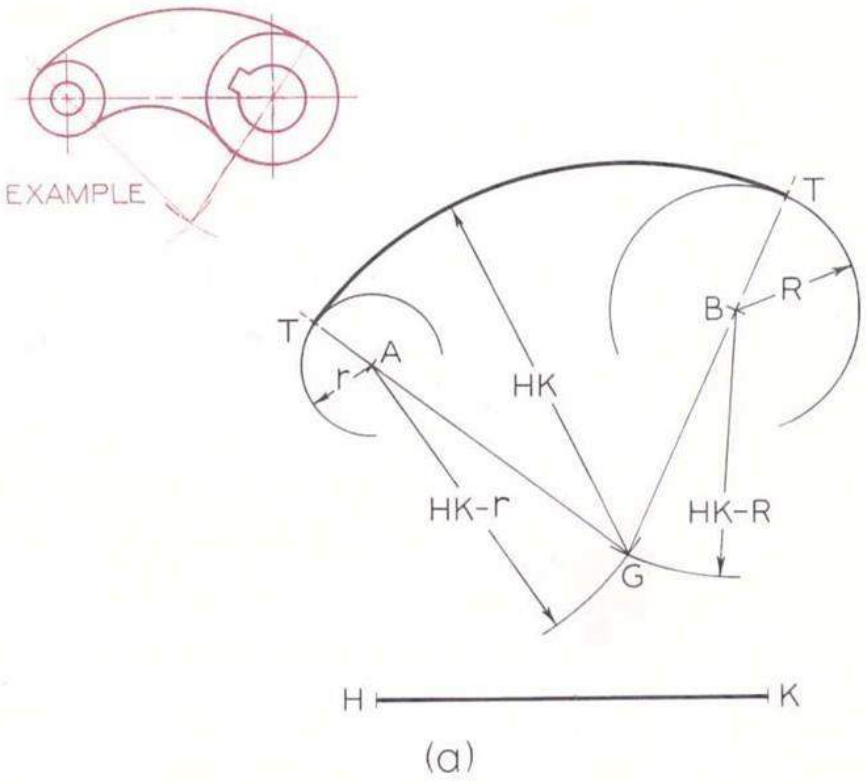
Example (4):



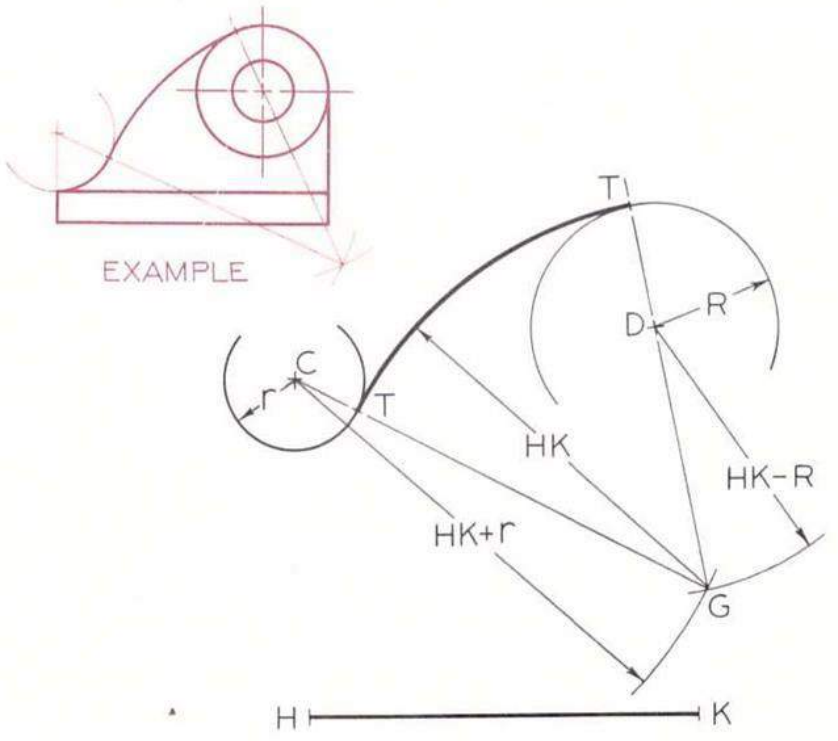
Example (5):



Example (6):



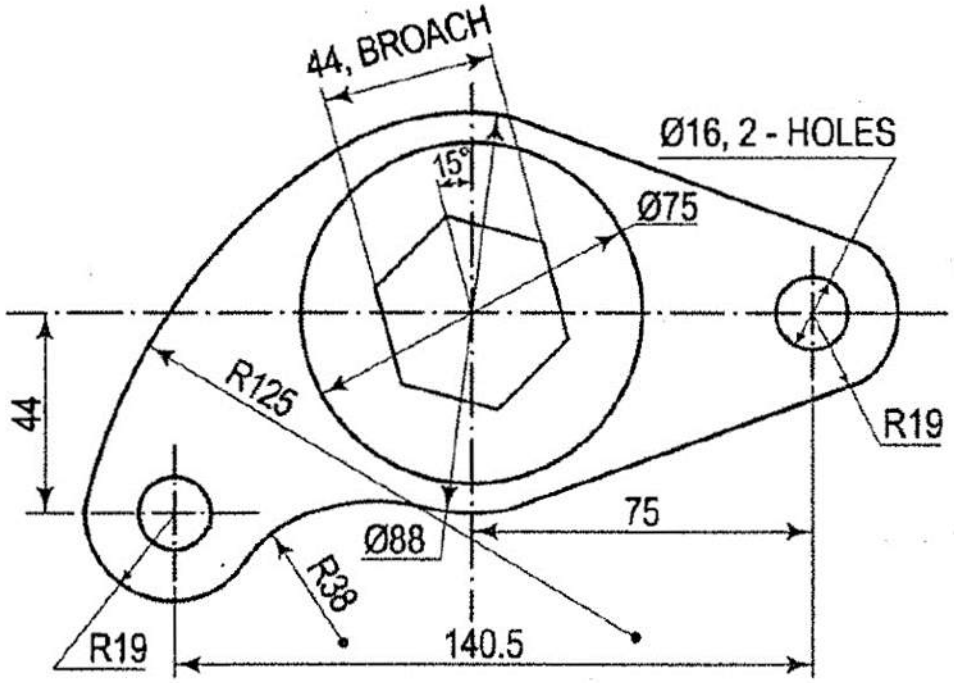
(a)



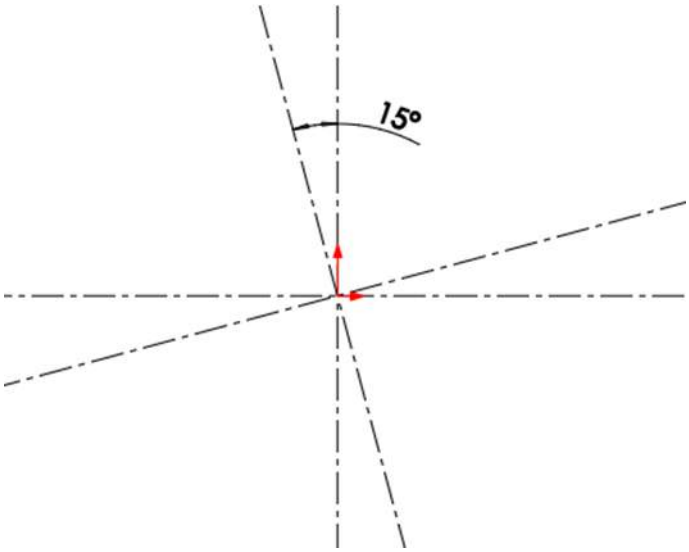
(b)

Solved Example

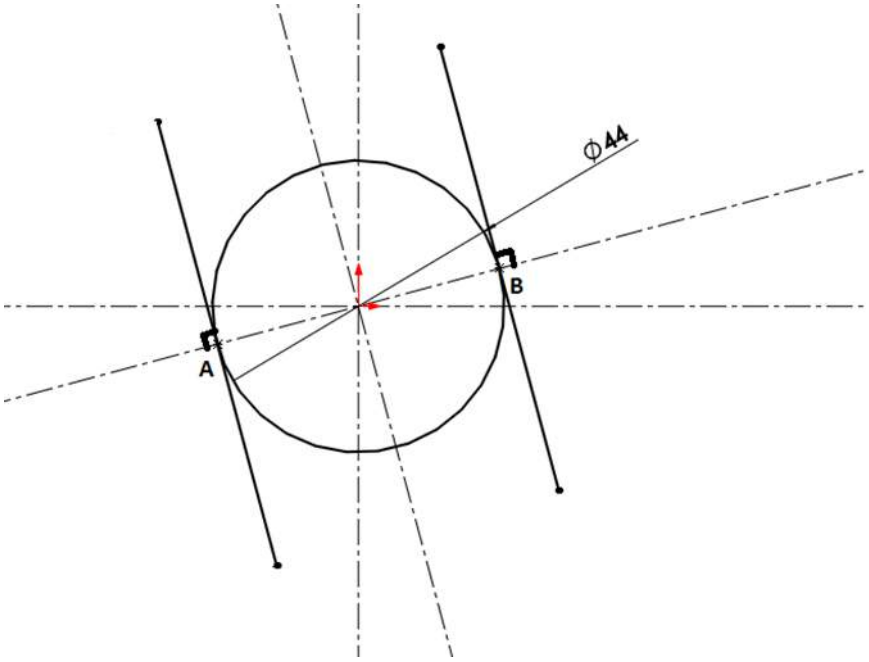
Let us take an example that contains as much as possible of the given geometric relations



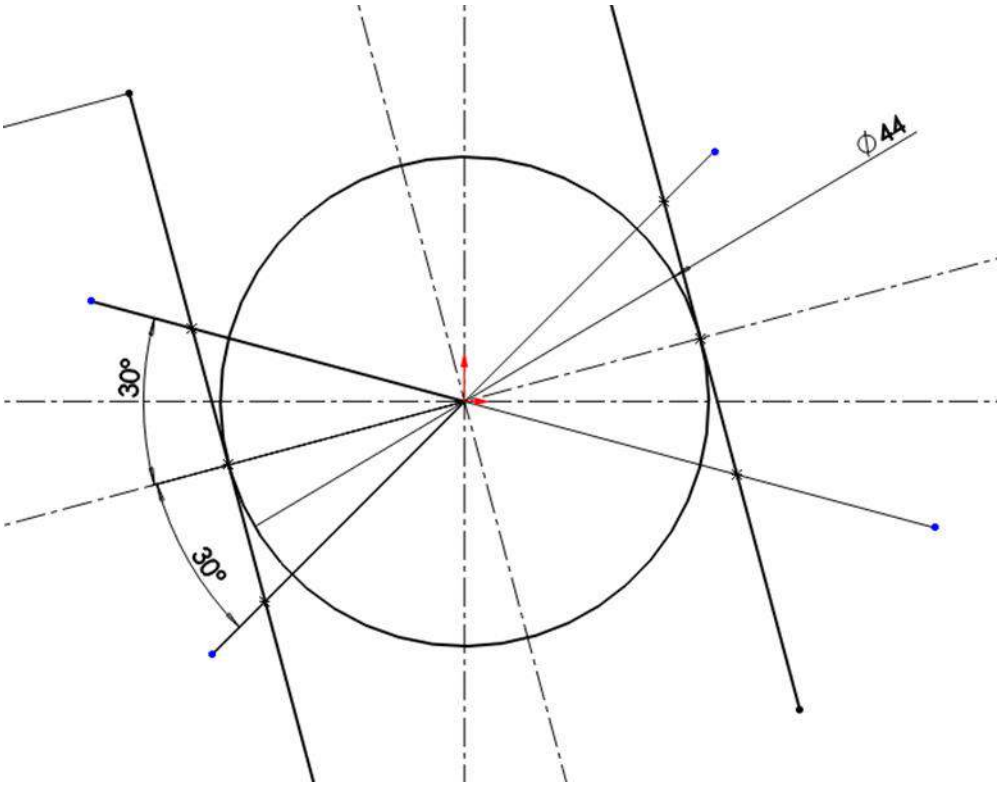
1. The best way to start any drawing is to start with the main centre lines, here we have horizontal and vertical centre lines and also inclines ones for the hexagon.



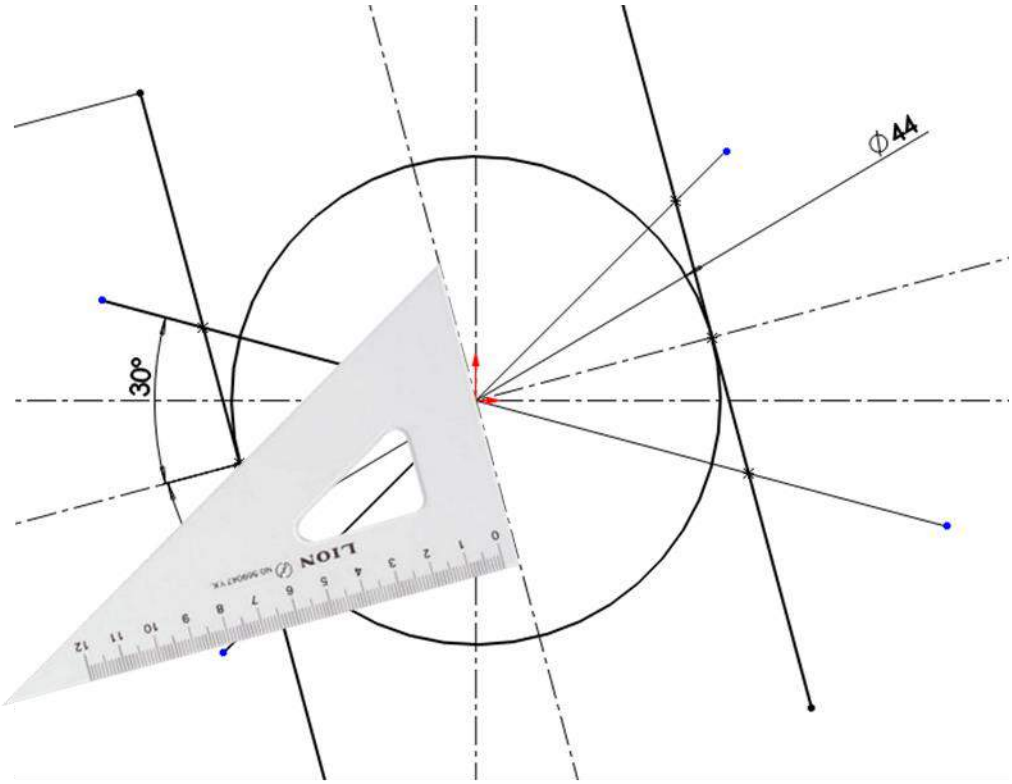
2. To draw the hexagon, he we have a distance between 2 sides (not between corners). Here we are dealing with the inclined centre lines, draw a circle with diameter equal to 44 mm then from the quadrant points (A and B), draw 2 lines normal to the inclined centre line as shown



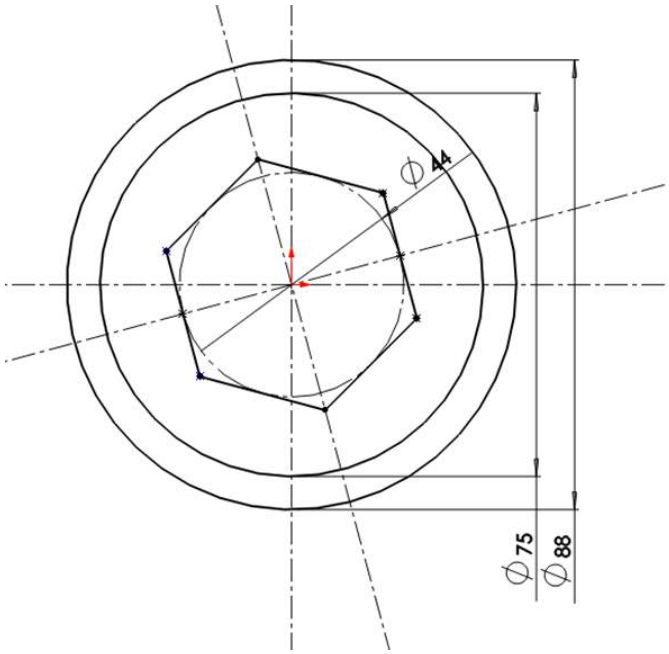
3. Draw 2 lines crossing through the centre point so that the angle between each line and the centre line (the inclined one with 15 degrees from the horizontal) is 60 degrees, these lines will intersect with the existing lines at 4 points specifying 4 corners



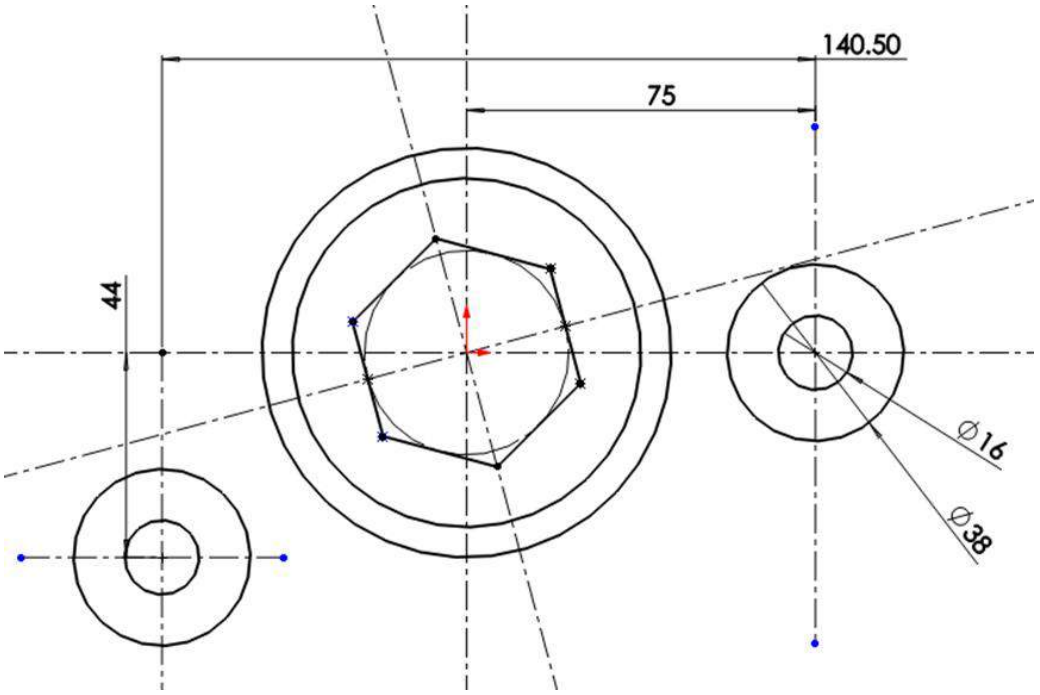
4. Put the 60 triangle at 2 points (the half left or the half right) so that the 60 degrees is between the inclined centre #2 and the 60-degree edge



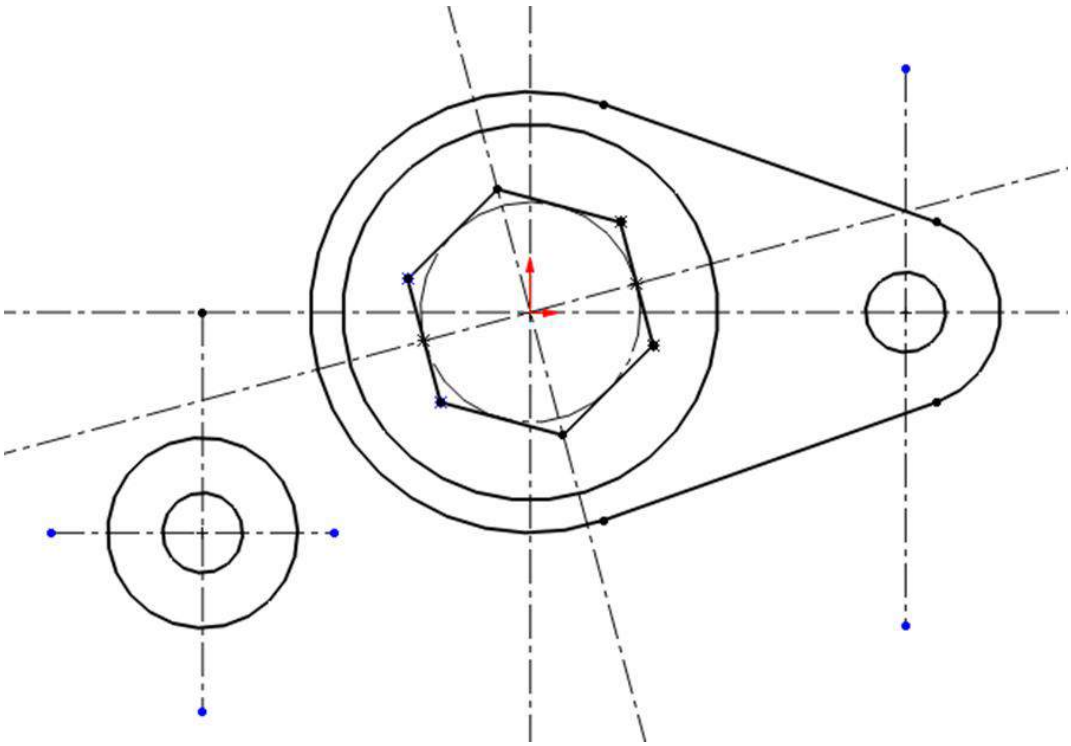
5. Draw the 2 concentric circles of diameters 75 and 88 mm



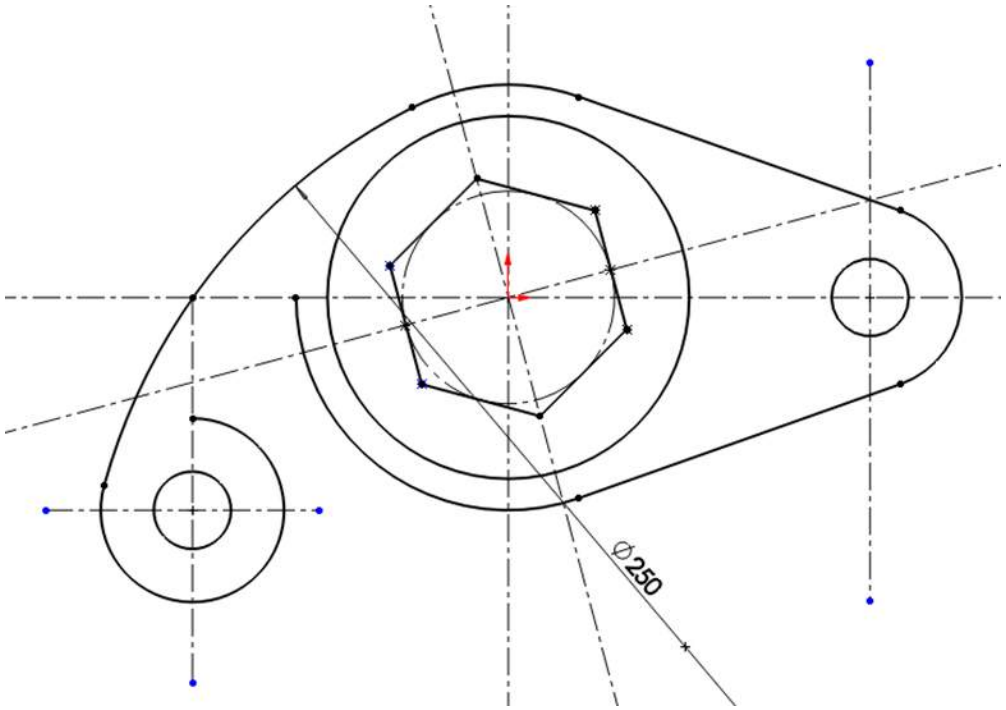
6. Assign the other 2 centres with the specified dimensions and draw the 4 circles, 2 of them with D = 16 mm And the other 2 with R = 19 mm



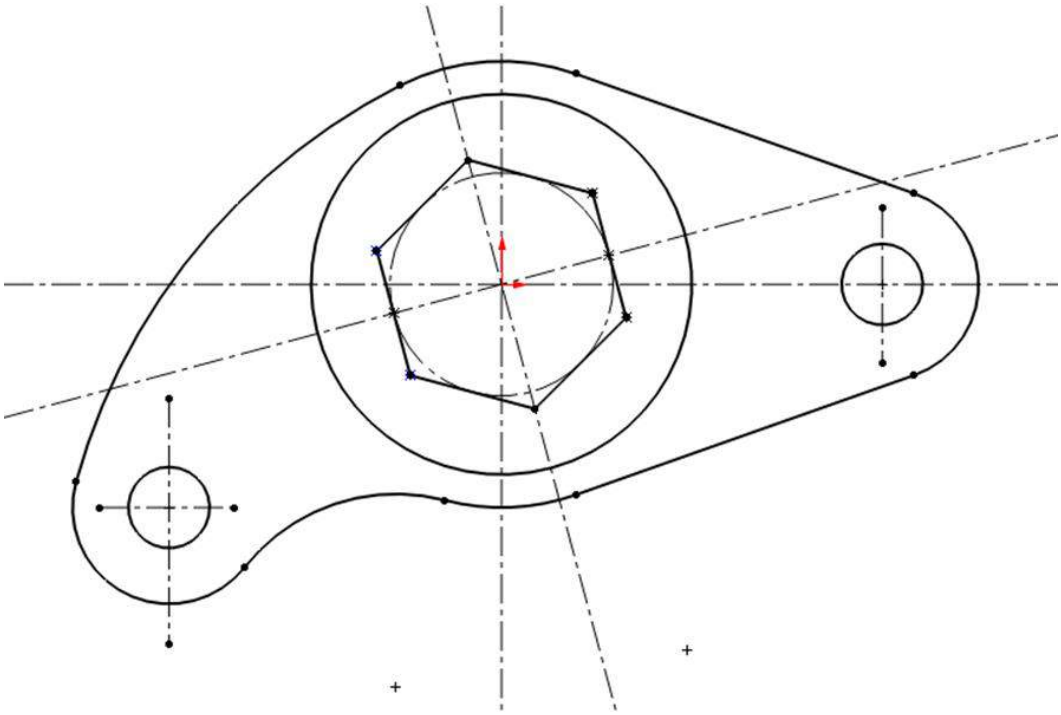
7. To draw a tangent line between the 88 mm diameter circle and the 38 mm diameter one, follow the instructions given in page G-07



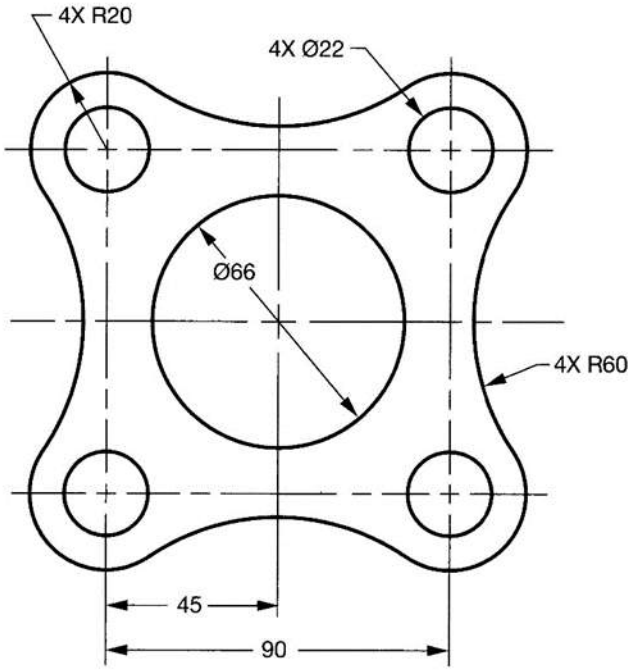
8. The next step is to draw the R-125 mm arc that is tangent to the D-88 mm circle and to the D-38 mm one Follow the steps given in the page G-08 to draw this arc



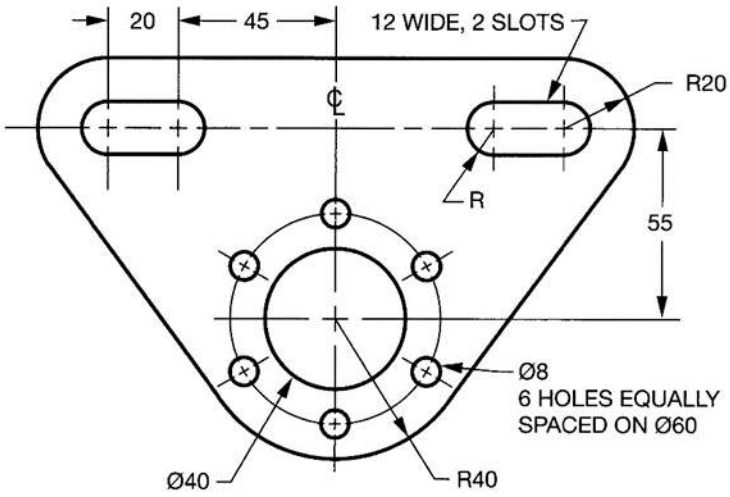
9. The last step is to draw the R-38 mm arc that is tangent to the D-88 mm and the D-38 mm circles. Again follow the steps given in page G-08 to draw this arc



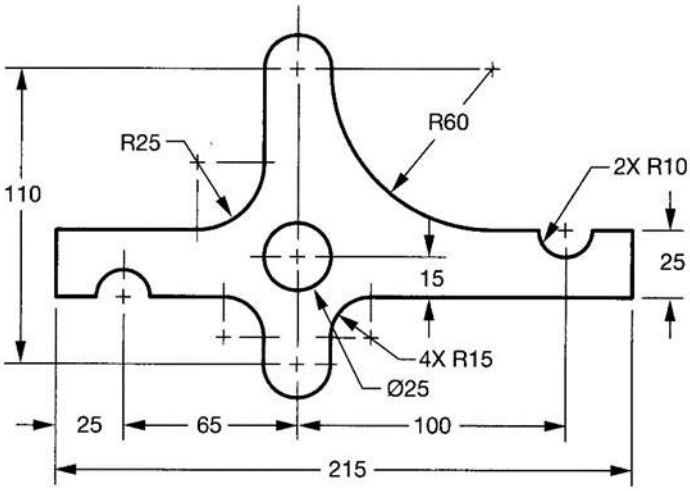
Problems



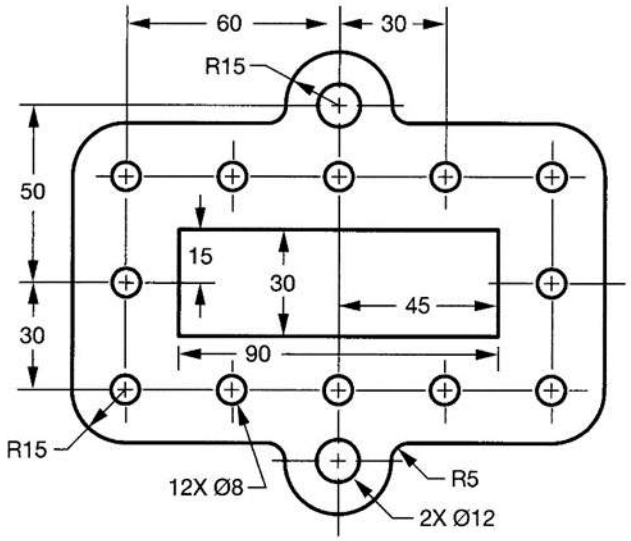
ANCHOR PLATE



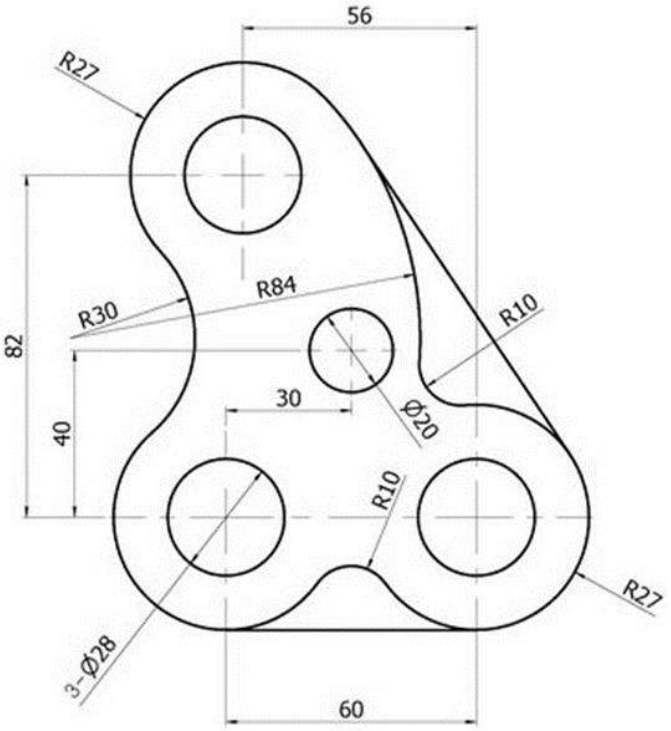
SHAFT SUPPORT



PAWL



GASKET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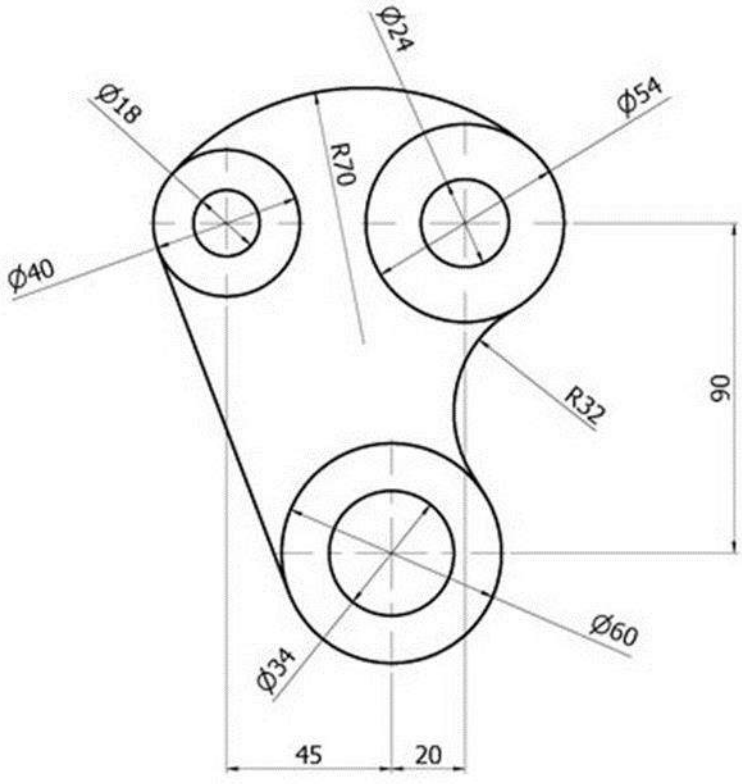
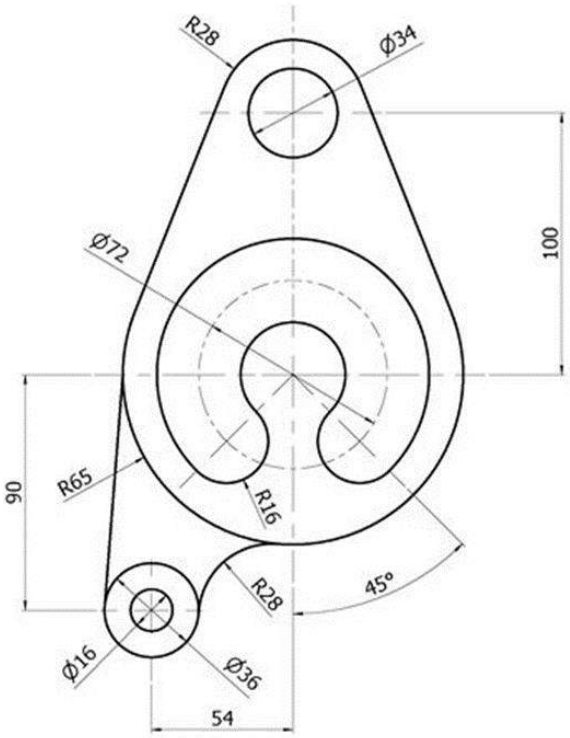
R27
R84
R30
R10
82
30
Ø20
40
R10
R27
3-Ø28
60


Ø24
Ø18
Ø54
R70
Ø40
R32
90
Ø60
Ø34
45
20


R28
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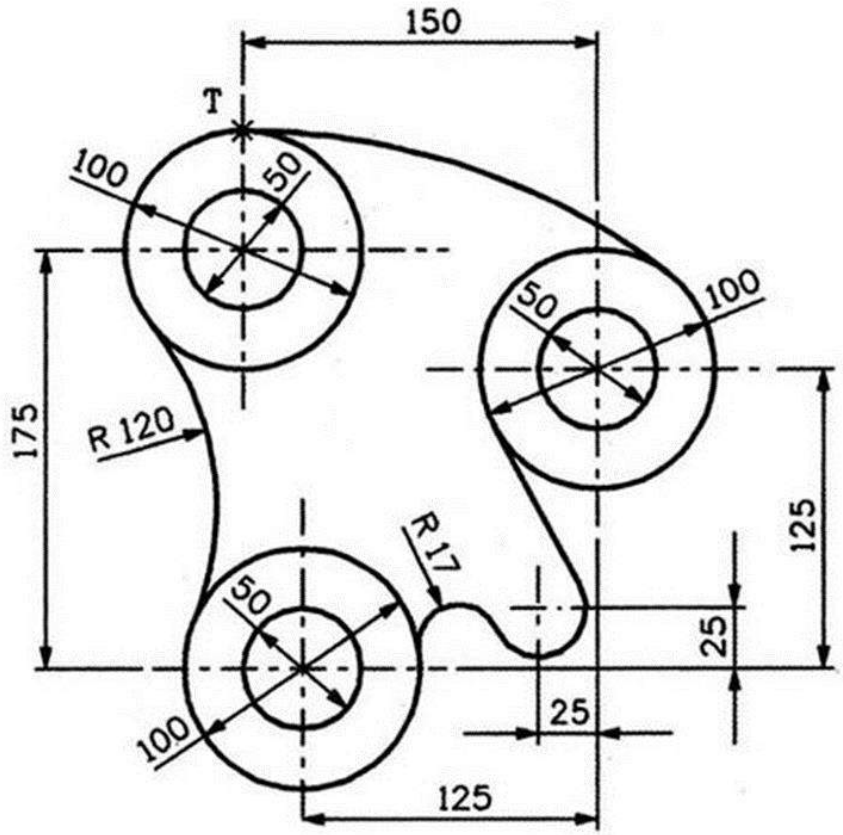
Ø72
100
90
R65
R16
R28
45°
Ø16
Ø36
54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T
100
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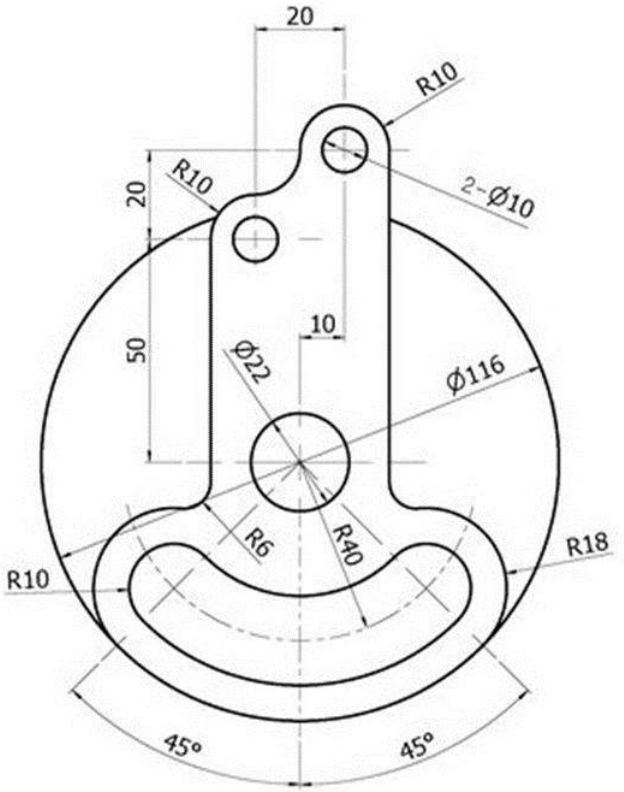
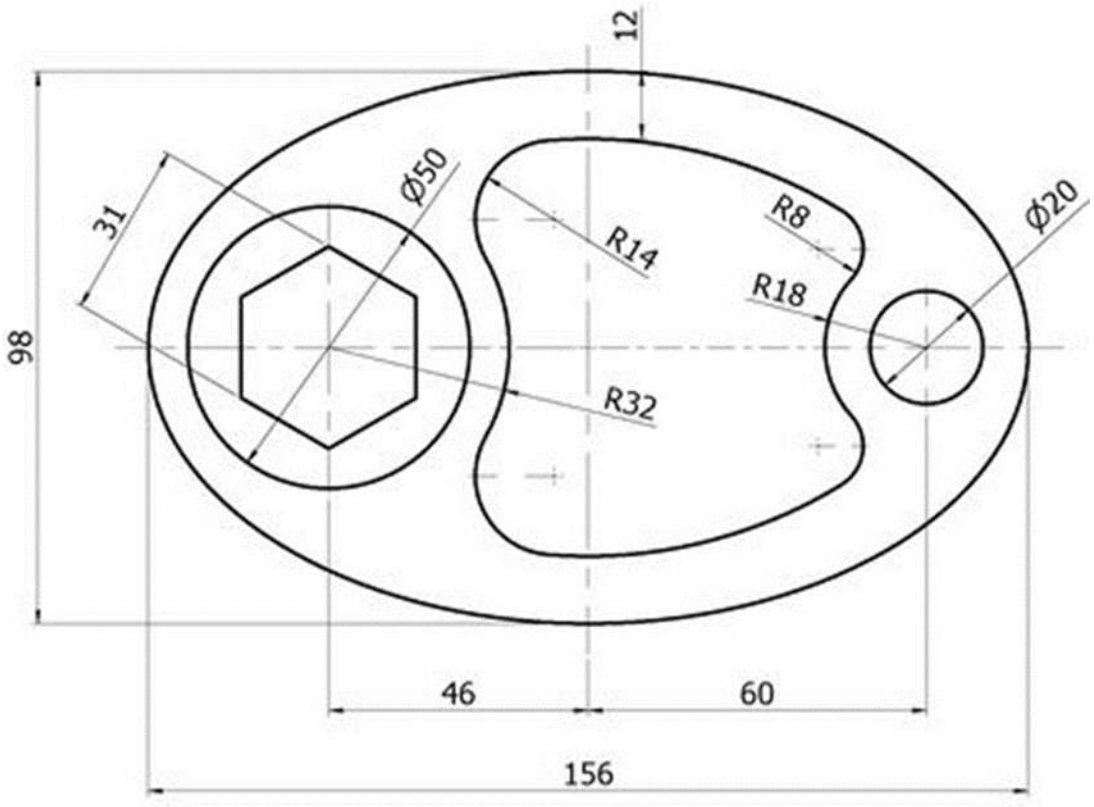
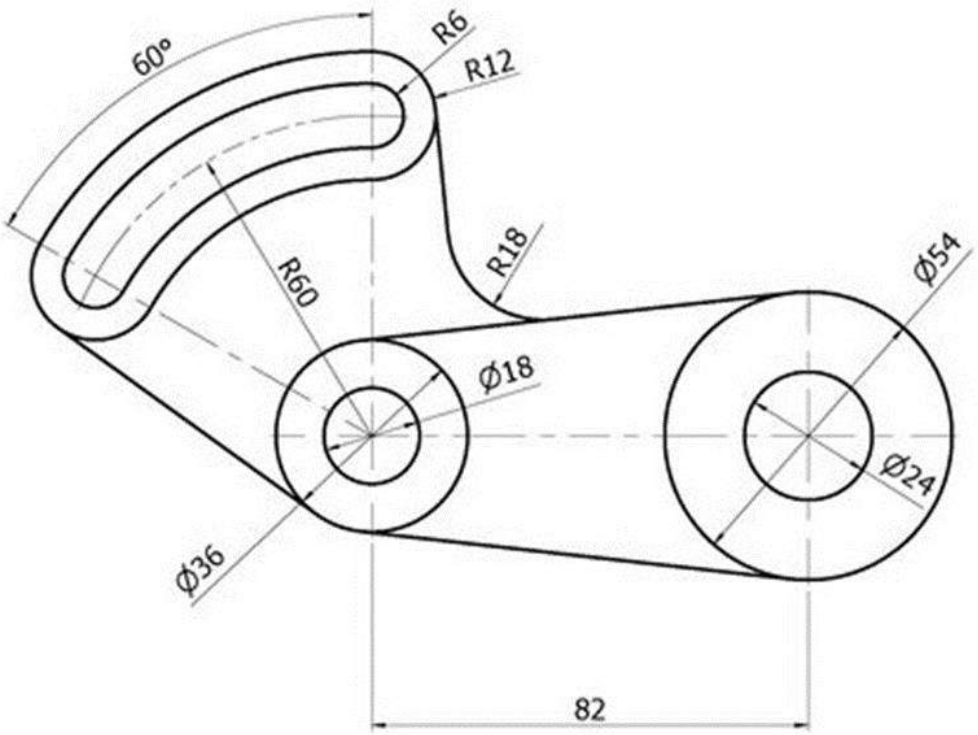
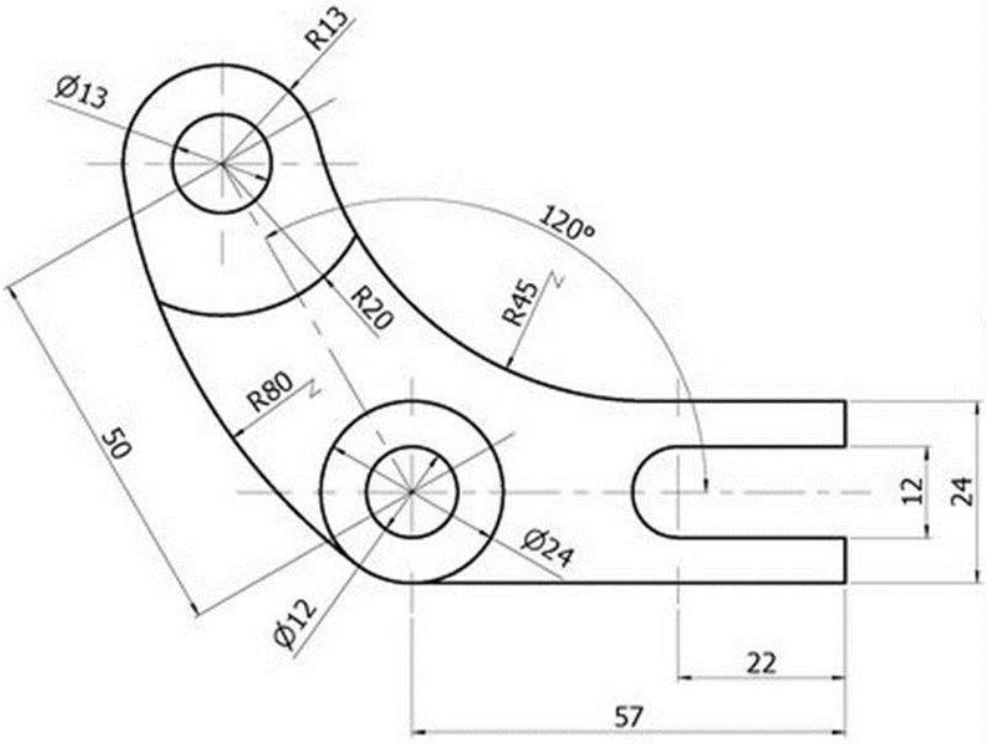
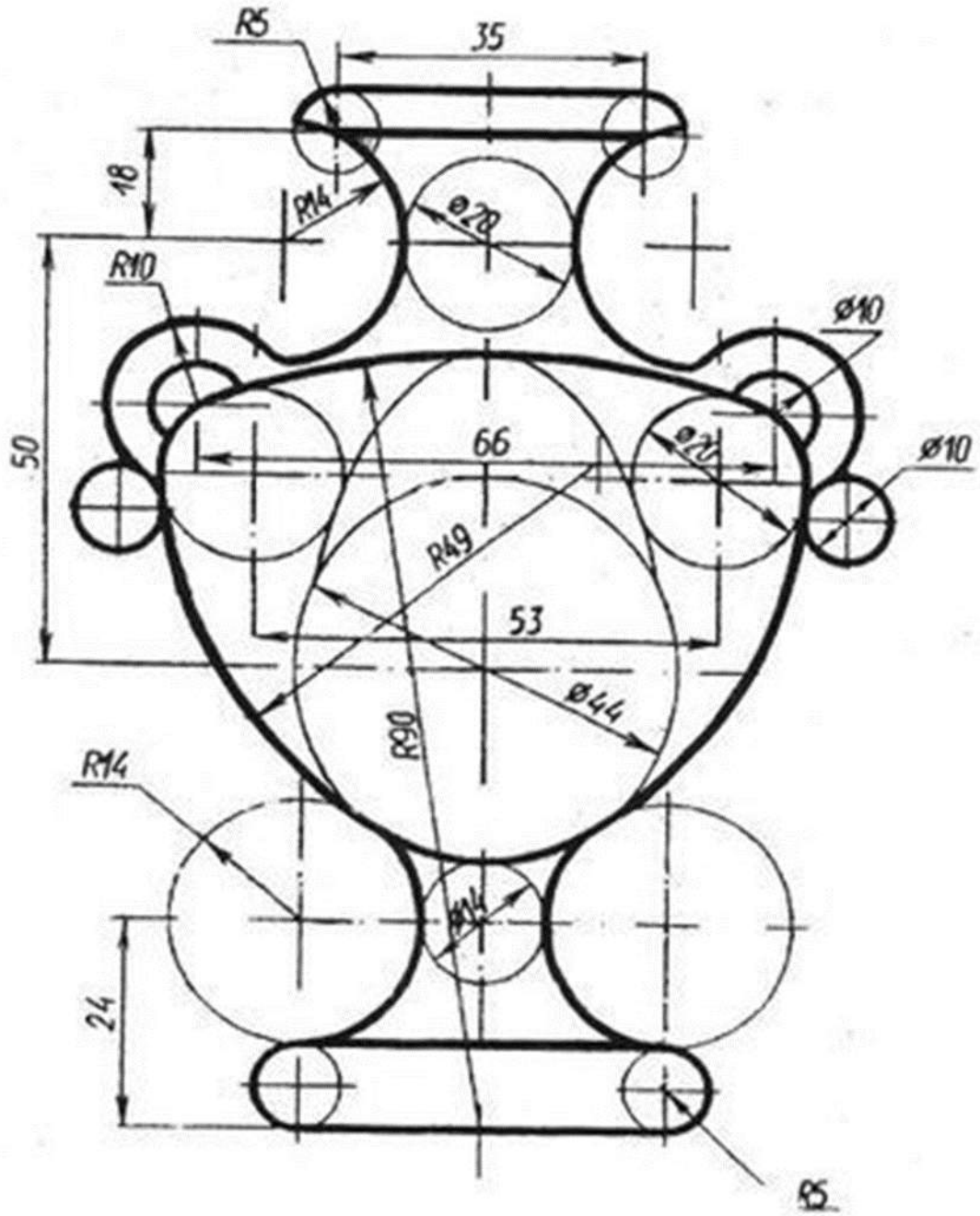
50
100
175
R 120
R17
50
25
125
25
100
125

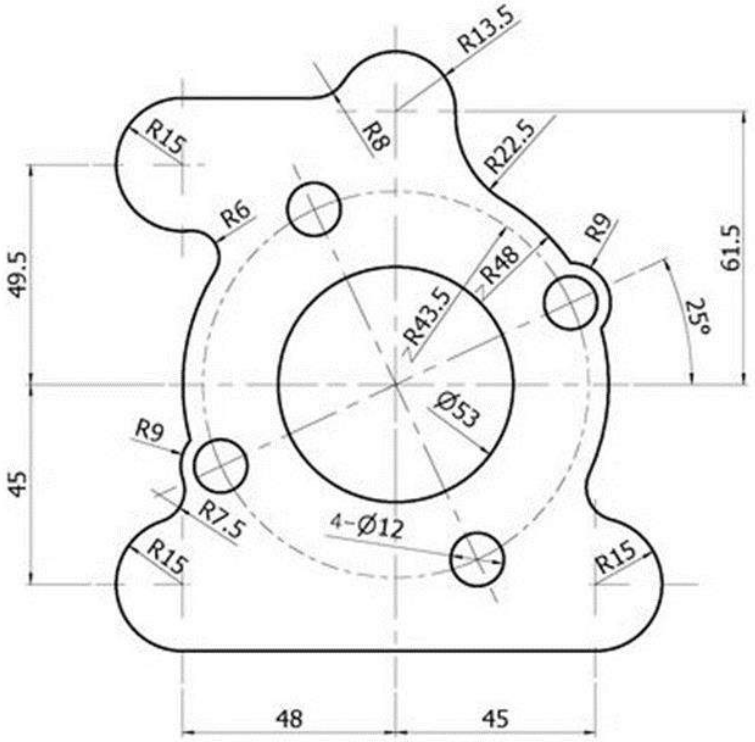

20
R10
R10
20
2-Ø10
10
50
Ø22
Ø116
R6
R40
R18
R10
45°
45°

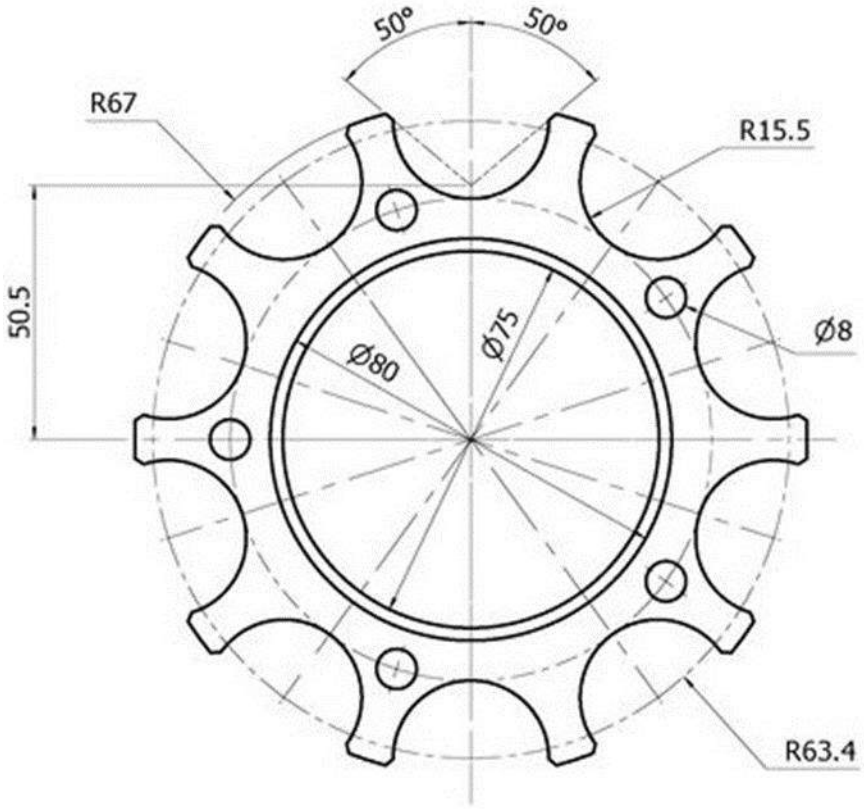

12
31
Ø50
R14
R8
Ø20
R18
98
R32
46
60
156

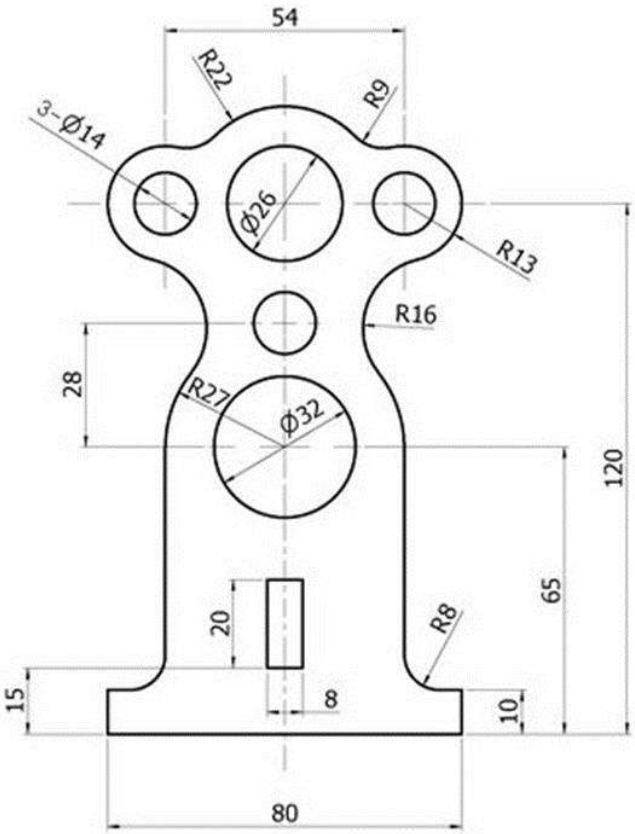

60°
R6
R12
R18
R60
Ø54
Ø18
Ø24
Ø36
82

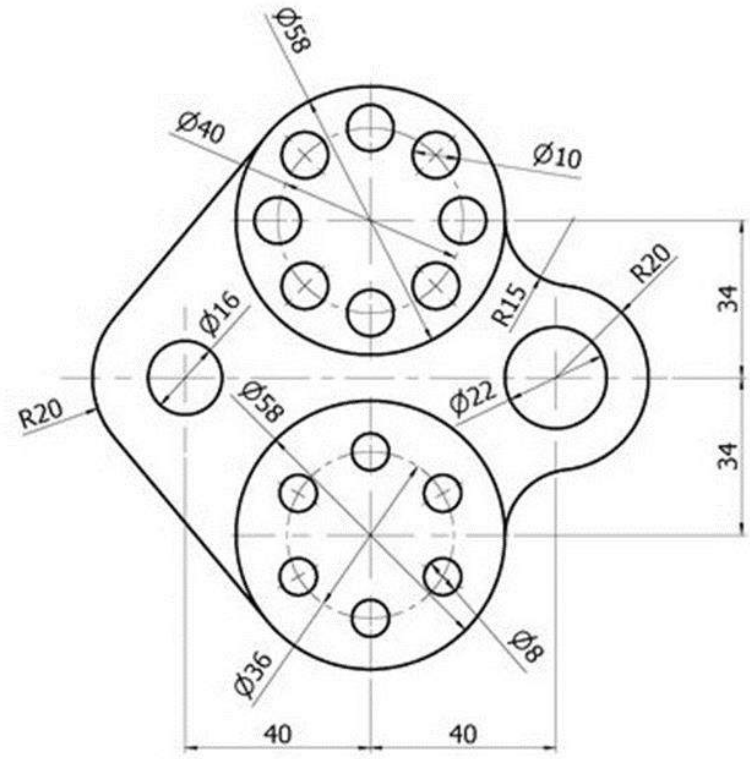

R13
Ø13
120°
R20
R45
R80
50
12
24
Ø24
Ø12
22
57

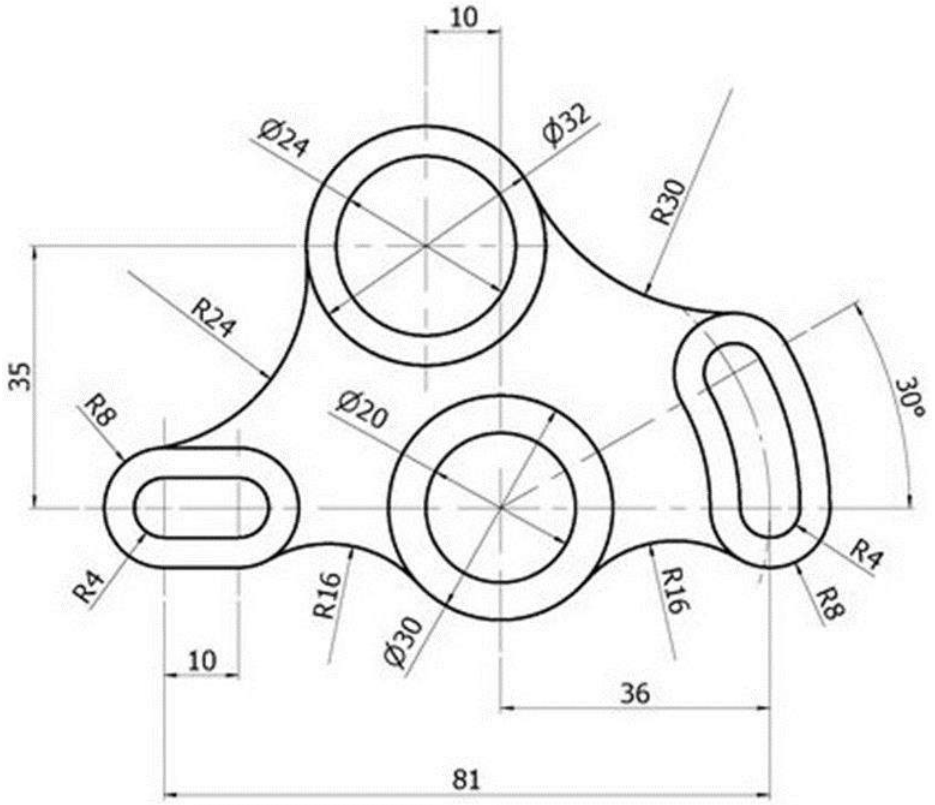

R5
35
18
R14
ø20
R10
ø10
50
66
ø20
ø10
R49
53
ø44
R90
R14
ø14
24
R5

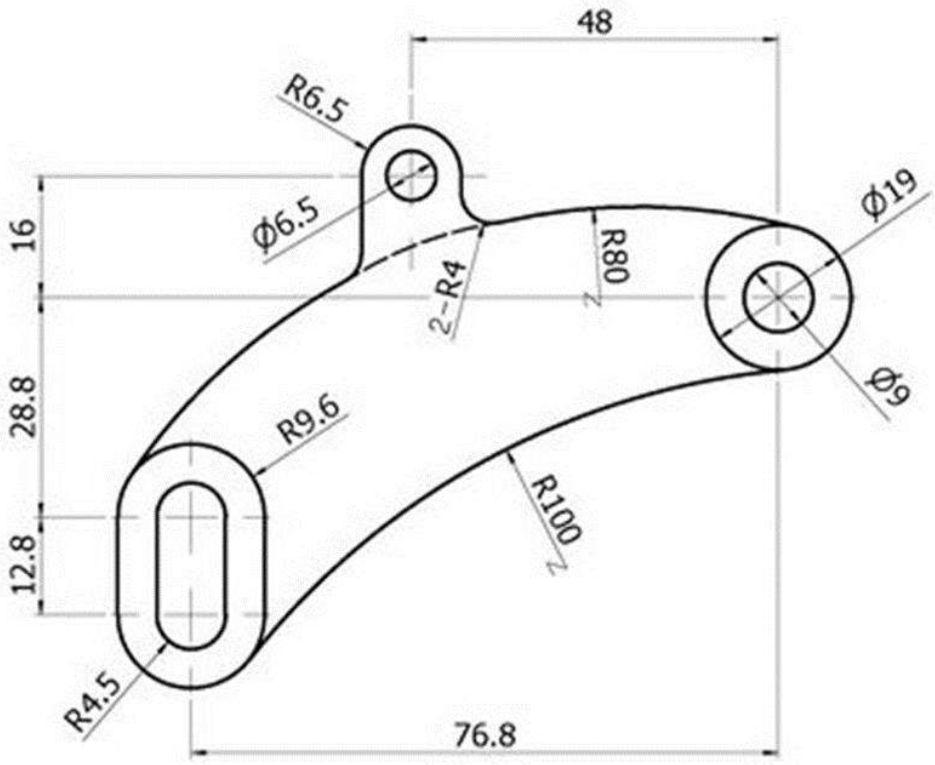

R13.5
R15
R8
R22.5
R6
R9
R48
R43.5
49.5
61.5
25°
Ø53
R9
45
R7.5
4-Ø12
R15
R15
48
45

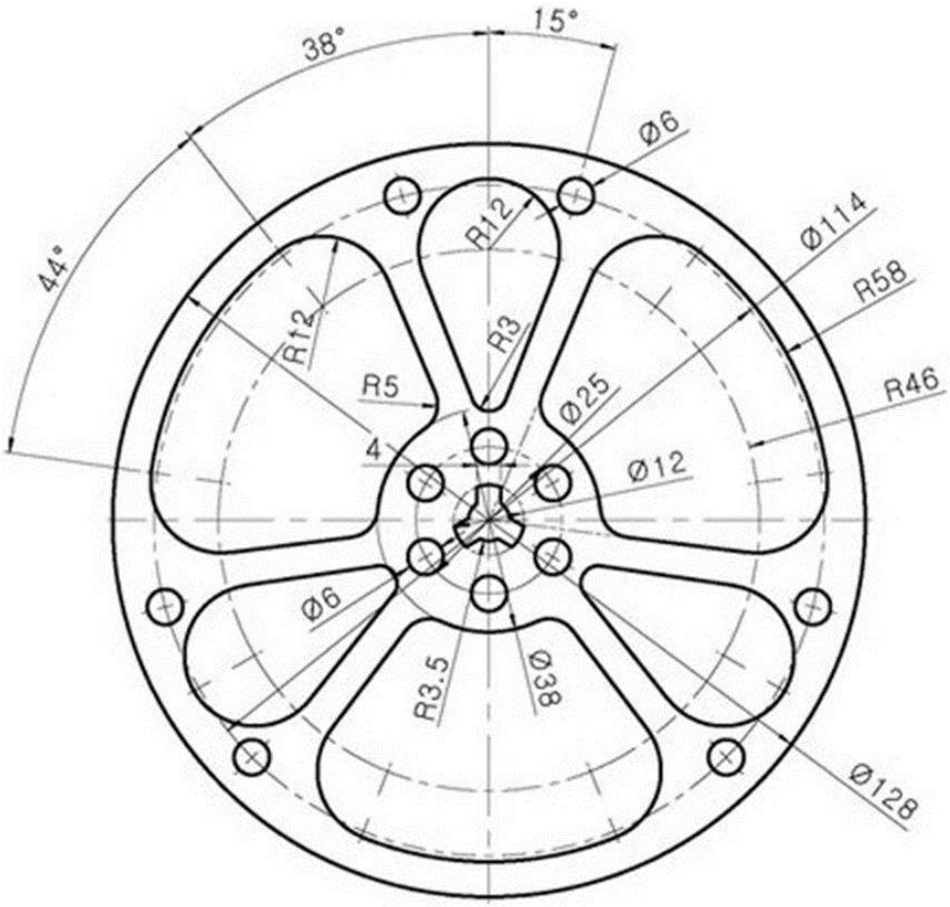

50°
50°
R67
R15.5
50.5
Ø80
Ø75
Ø8
R63.4

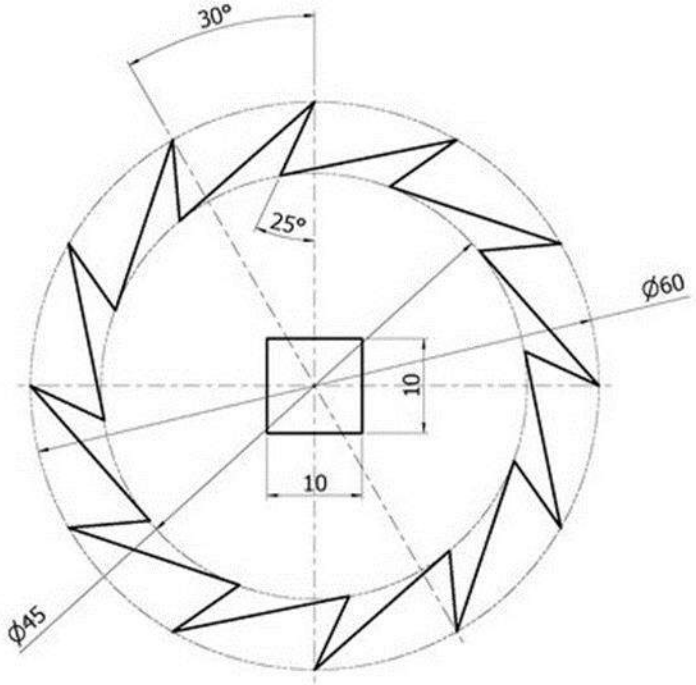

54
R22
R9
3-Ø14
Ø26
R13
R16
28
R27
Ø32
120
65
20
R8
15
8
10
80

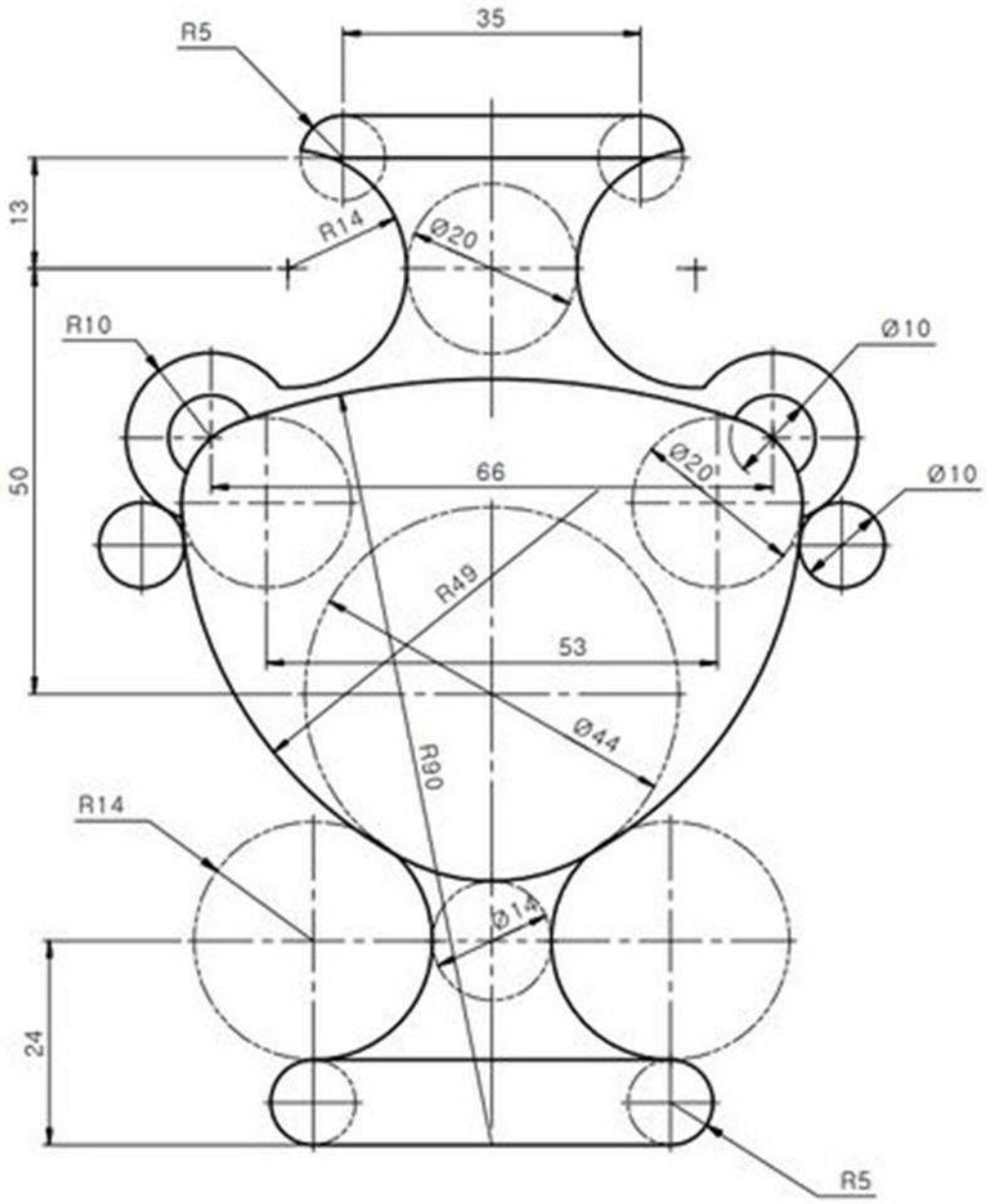

Ø58
Ø40
Ø10
R20
R15
Ø16
34
R20
Ø58
Ø22
34
Ø36
Ø8
40
40

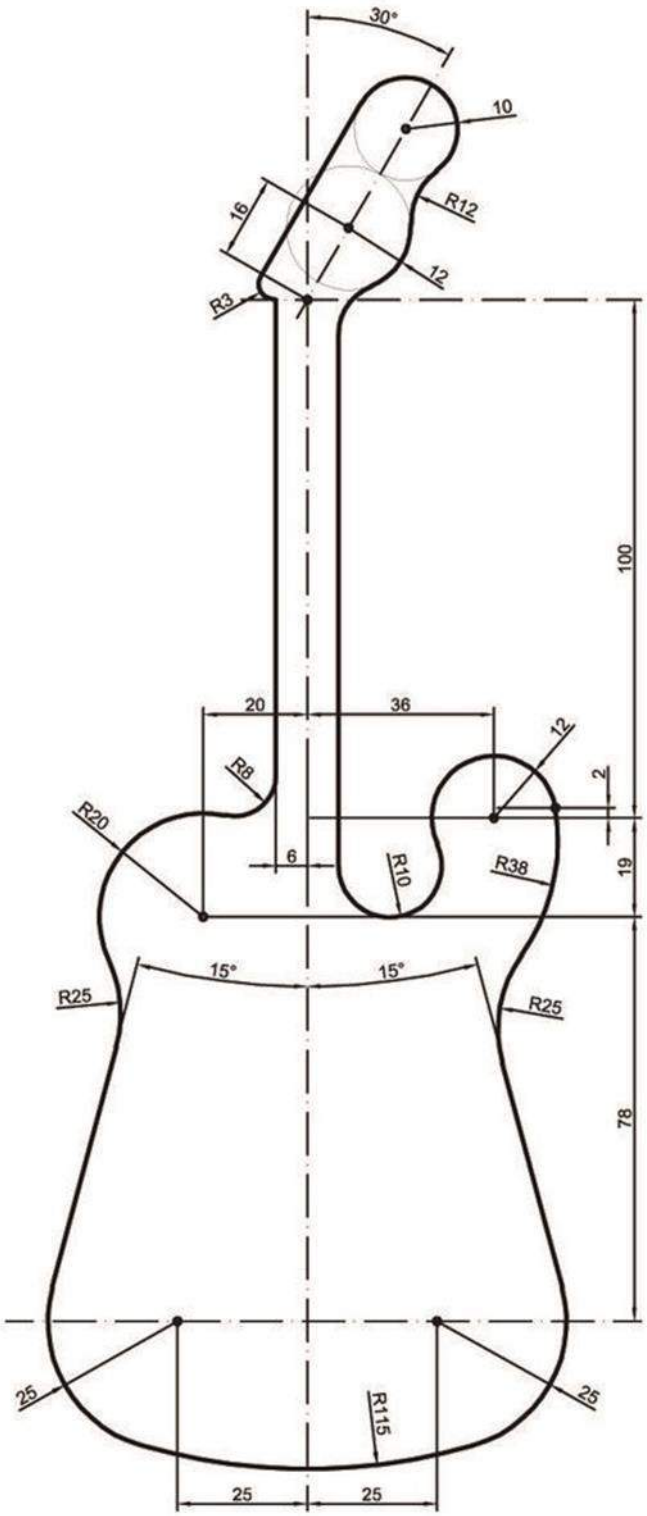

10
Ø24
Ø32
R30
R24
35
R8
Ø20
30°
R4
R16
R16
R4
R8
Ø30
10
36
81

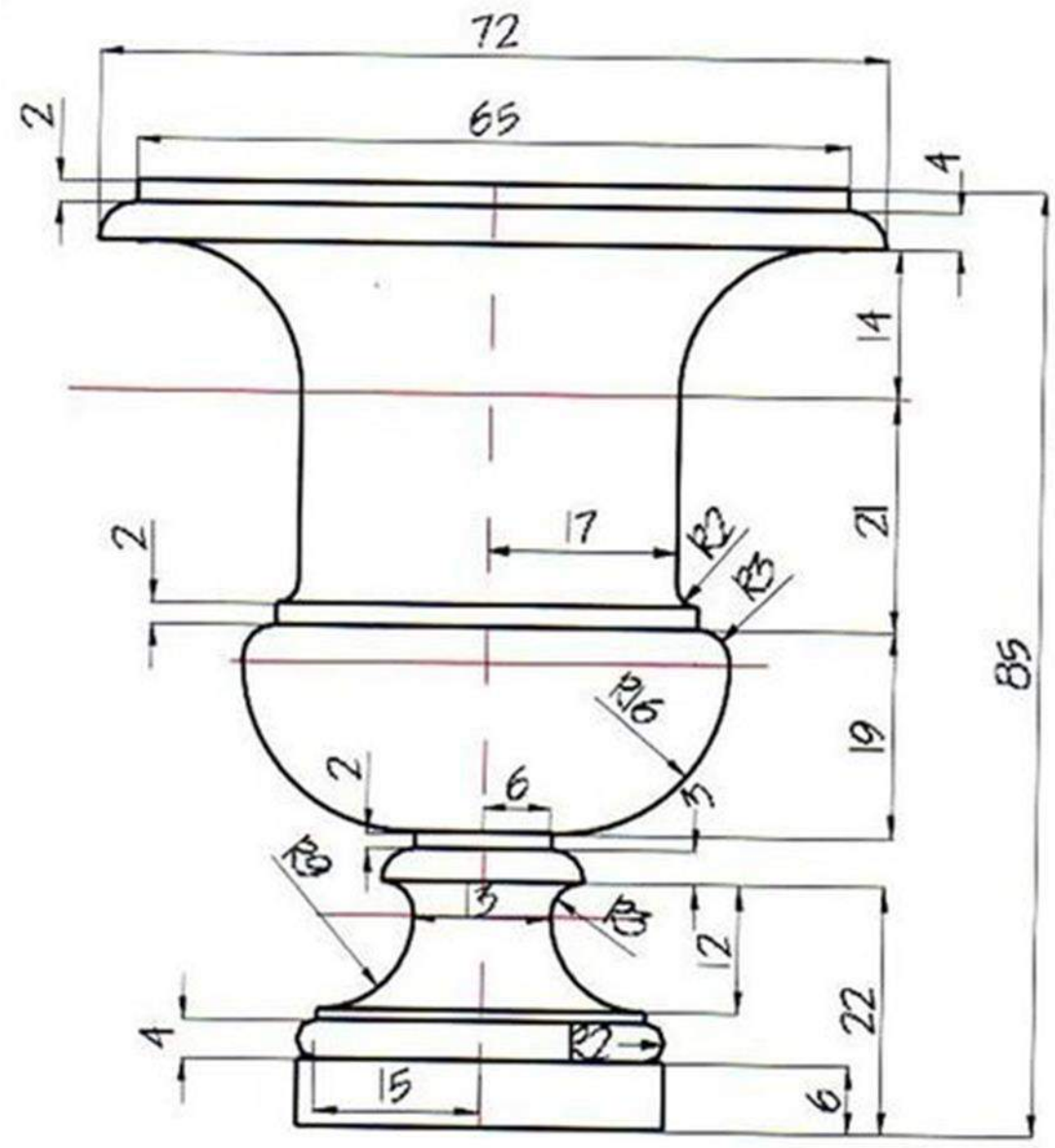

48
R6.5
16
Ø6.5
2-R4
R80
Ø19
28.8
R9.6
R100
Ø9
12.8
R4.5
76.8


38°
15°
44°
Ø6
R12
Ø114
R58
R12
R3
R5
Ø25
R46
4
Ø12
Ø6
R3.5
Ø38
Ø128


30°
25°
ϕ60
10
10
ϕ45


R5
35
13
R14
Ø20
R10
Ø10
50
66
Ø20
Ø10
R49
53
Ø44
R90
R14
Ø14
24
R5


30°
10
R12
16
12
R3
100
20
36
12
2
R8
R20
6
R10
R38
19
15°
15°
R25
R25
78
25
R115
25
25
25


72
65
2
4
14
21
85
19
17
R2
R3
R16
2
6
3
R8
3
R3
12
22
4
R2
15
6

Orthographic Projection is such a type of drawing in which Parallel Projection is used for the preparation of the drawing of an object. These lines are perpendicular to the plane. In this drawing, it is assumed that the object is at infinity. The shape of an object is seen in actual size in such drawing. A plane is an imaginary surface on which pictures are prepared imaginably. Then it is transferred to the paper. This plane is faced toward the object whose view is to be prepared as shown in Figure (1).

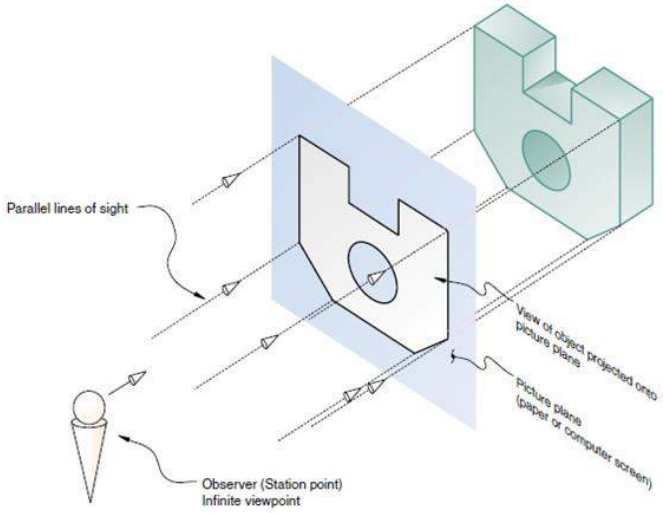


Figure 1 Orthographic Projection Concept

The total number of views to observe an object are shown in Figure (2). These views are Front, rear, Right Side, Left Side, Top, and Bottom views. Due to the use of dashed lines to represent hidden features, the number of views can be reduced to three. These are Front View, Side View, and Top View.

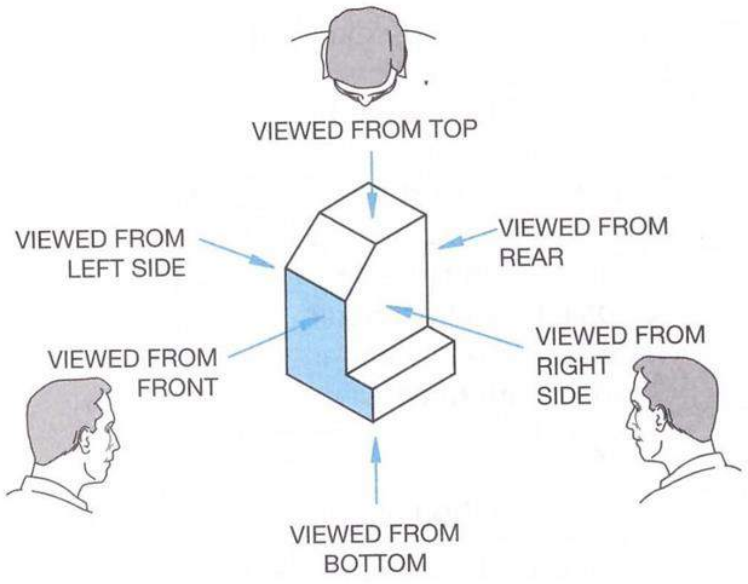


Figure 2 The Six Principal Views

UNDERSTANDING PROJECTIONS

To make and interpret drawings, you need to understand projections and the standard arrangement of views. You also need to be familiar with the geometry of solid objects and be able to visualize a 3D object that is represented in a 2D sketch or drawing. The ability to identify whether surfaces are normal, inclined, or oblique in orientation can help you visualize.
Common features such as vertices, edges, contours, fillets, holes, and rounds are shown in a standard way, which makes drawings simpler to create and help prevent them from being misinterpreted.

Views of Objects

A photograph shows an object as it appears to the observer but not necessarily as it is. It cannot describe the object accurately, no matter what distance or which direction it is taken from, because it does not show the exact shapes and sizes of the parts. It would be impossible to create an accurate 3D model of an object using only a photograph for reference because it shows only one view. It is a 2D representation of a 3D object.

Drawings are 2D representations as well, but unlike photos, they allow you to record sizes and shapes precisely. In engineering and other fields, a complete and clear description of the shape and size of an object is necessary to be sure that it is manufactured exactly as the designer intended. To provide this information about a 3D object, typically a number of systematically arranged views are used.

The system of views is called multiview projection. Each view provides certain definite information. For example, a front view shows the true shape and size of surfaces that are parallel to the front of the object. An example of a 3D object and its front view projection is shown in Figure 3. Figure 4 shows the six views of a house.

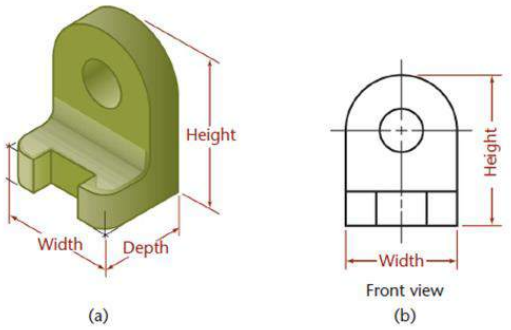


Figure 3 Front View of an object

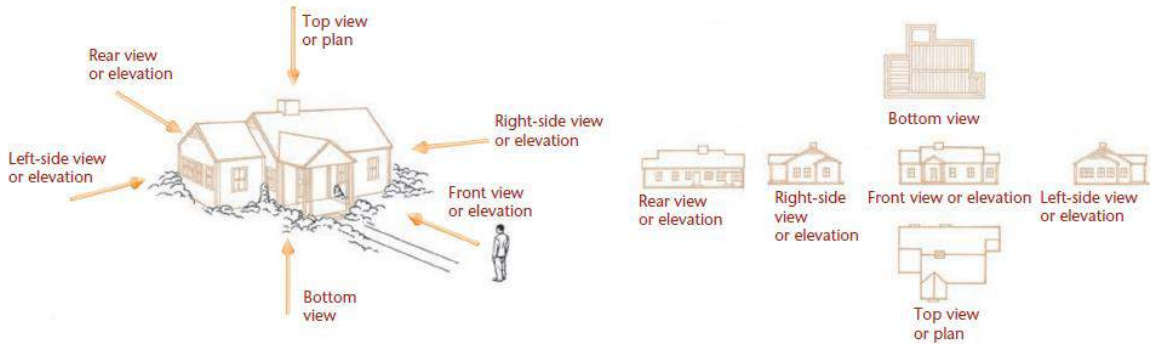


Figure 4 Six Views of a House

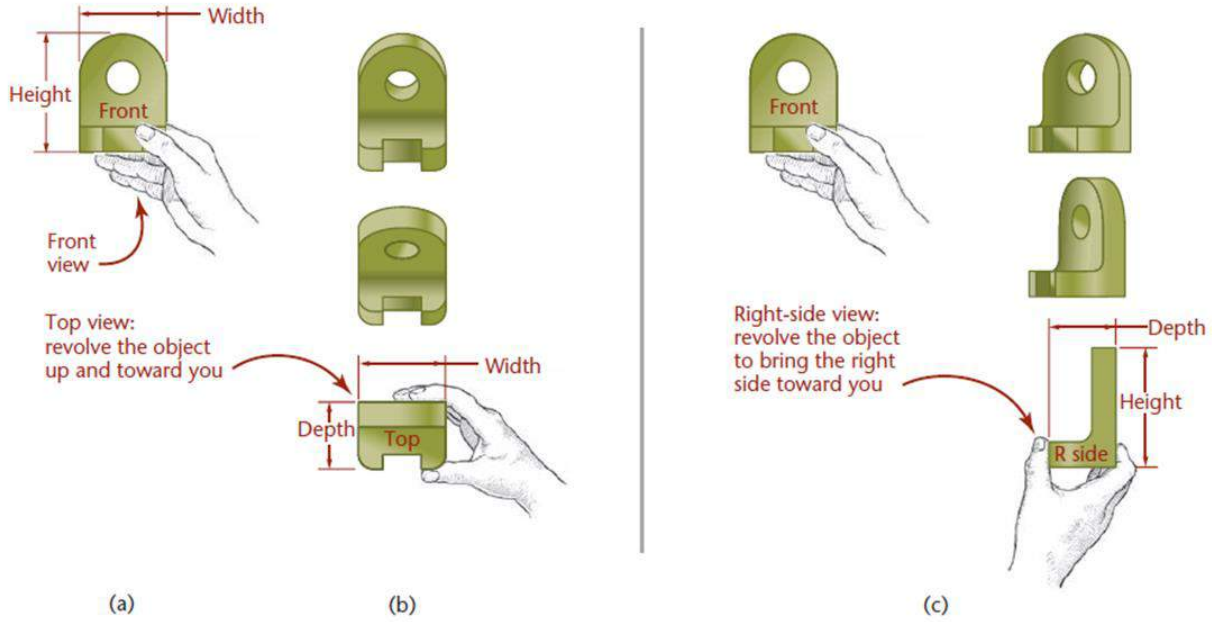


Figure 5 Revolving the Object to Produce Views. You can experience different views by revolving an object, as shown. (a) First, hold the object in the front view position. (b) To get the top view, tilt the object toward you to bring the top of the object into your view. (c) To get the right-side view, begin with the object's front view facing you and revolve it to bring the right side toward you. To see views of the rear, bottom, or left side, you would simply turn the object to bring those sides toward you.

Projection Method

Figure 6 illustrates the front view of an object drawn using an orthographic projection. Imagine a sheet of glass parallel to the front surfaces of the object. This represents the plane of projection. The outline on the plane of projection shows how the object appears to the observer.

In orthographic projection, rays (or projectors) from all points on the edges or contours of the object extend parallel to each other and perpendicular to the plane of projection. The word *orthographic* means "at right angles."

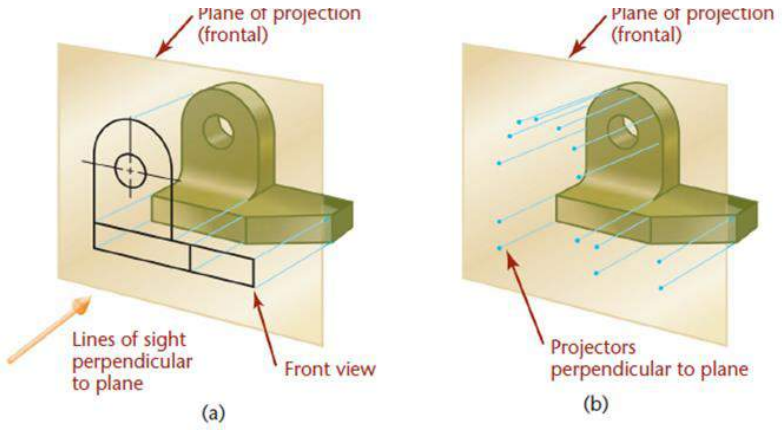


Figure 6 Projection of an Object

The Glass Box

One way to understand the standard arrangement of views on the sheet of paper is to envision a ***glass box***. If planes of projection were placed parallel to each principal face of the object, they would form a box, as shown in Figure 7. The outside observer would see six standard views (front, rear, top, bottom, right side, left side) of the object through the sides of this imaginary glass box.

To organize the views of a 3D object on a flat sheet of paper, imagine the six planes of the glass box being unfolded to lie flat, as shown in Figure 8. Think of all planes except the rear plane as hinged to the frontal plane. The rear plane is usually hinged to the left-side plane. Each plane folds out away from the frontal plane. The representation of the hinge lines of the glass box in a drawing are known as folding lines. The positions of these six planes after they have been unfolded are shown in Figure 9.

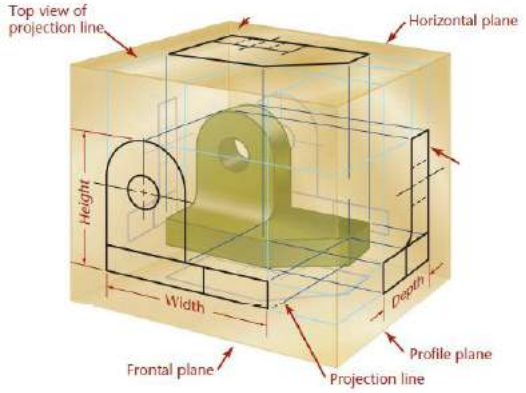


Figure 7 The Glass Box

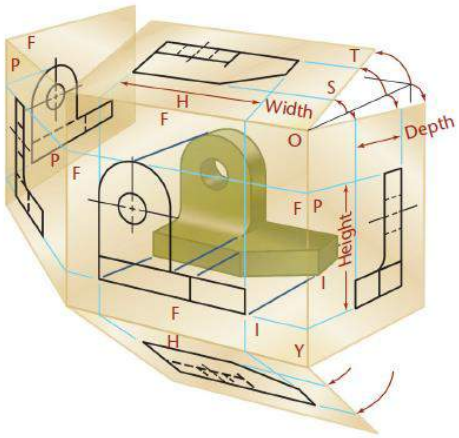


Figure 8 Unfolding the Box

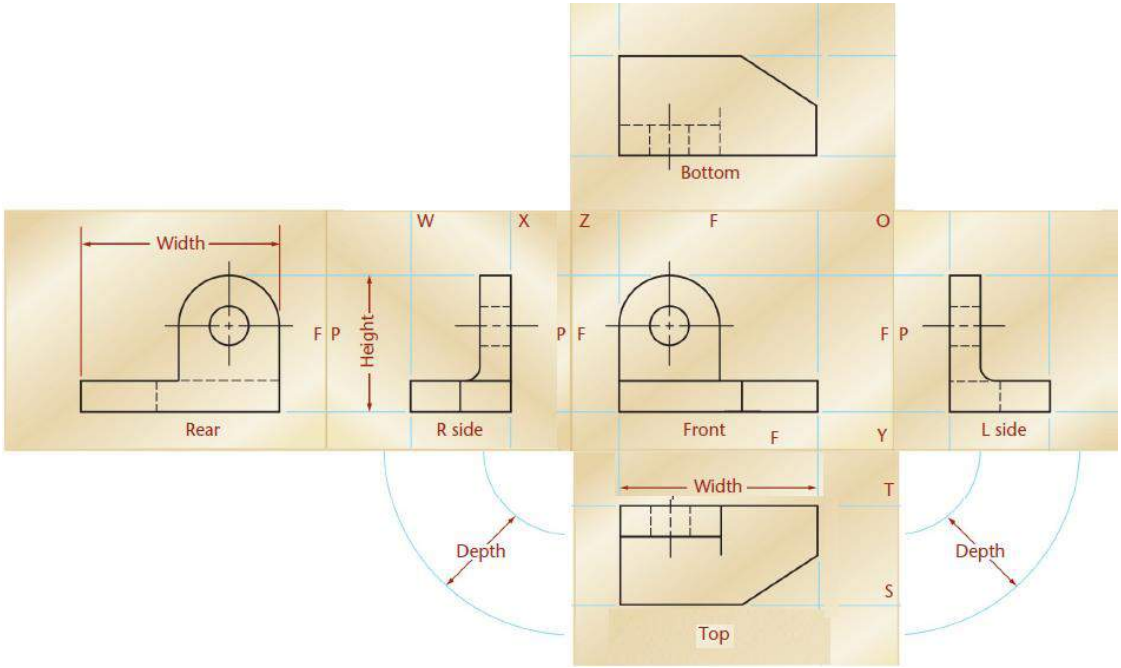


Figure 9 The Glass Box Unfolded

Necessary Views

A sketch or drawing should contain only the views needed to clearly and completely describe the object. These minimally required views are referred to as the ***necessary views***. Choose the views that have the fewest hidden lines and show essential contours or shapes most clearly. Complicated objects may require more than three views or special views such as partial views.

Many objects need only two views to clearly describe their shape. If an object requires only two views, and the left-side and right-side views show the object equally well, use the right-side view. If an object requires only two views, and the top and bottom views show the object equally well, choose the top view. If only two views are necessary and the top view and right-side view show the object equally well, choose the combination that fits best on your paper.

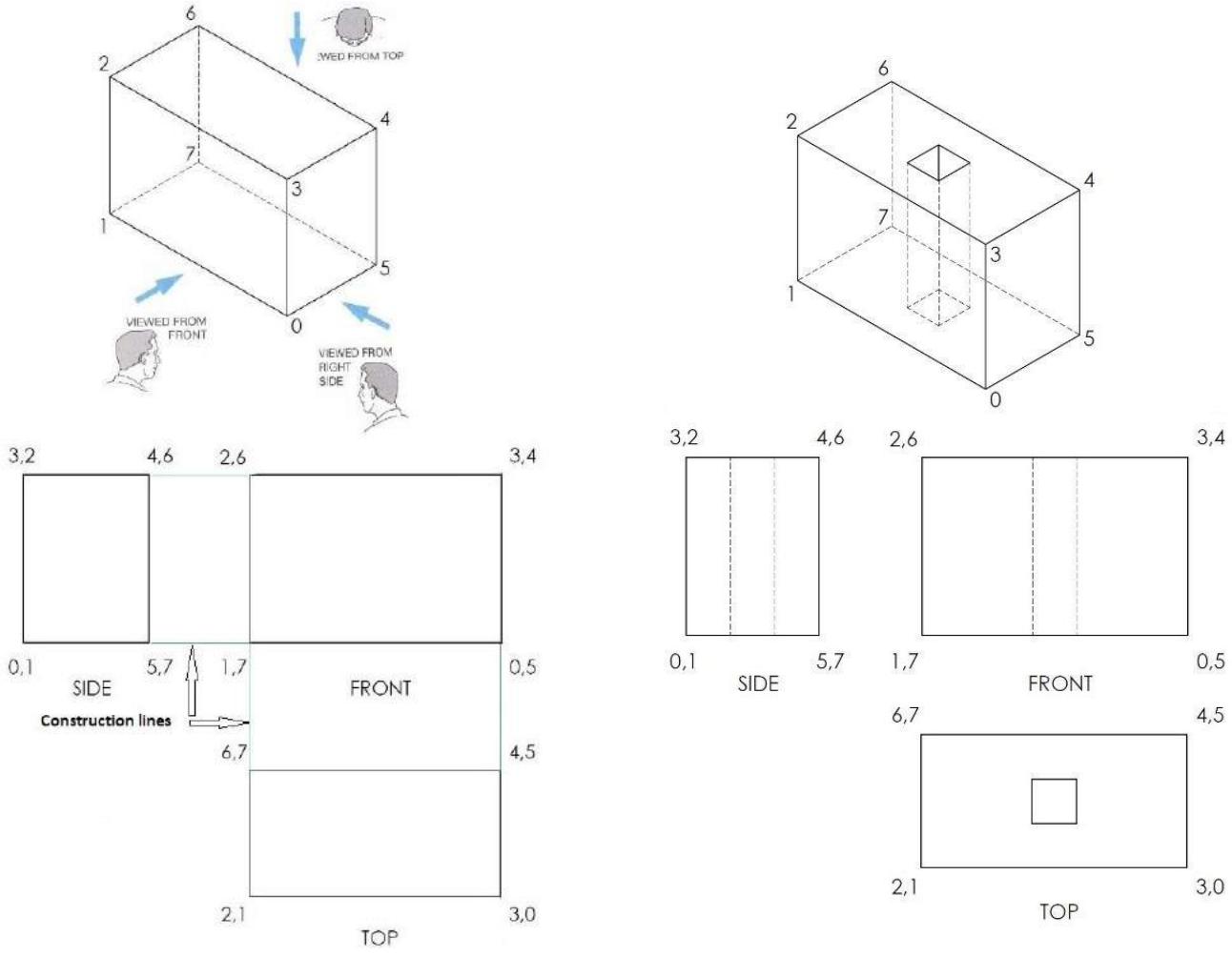
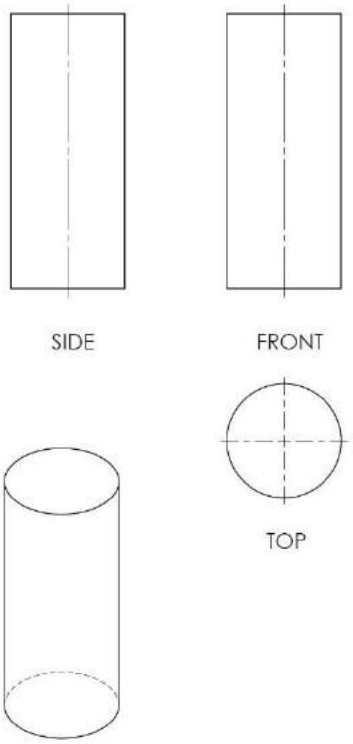
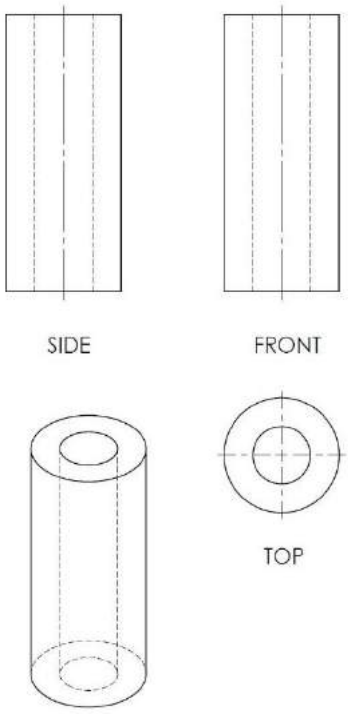
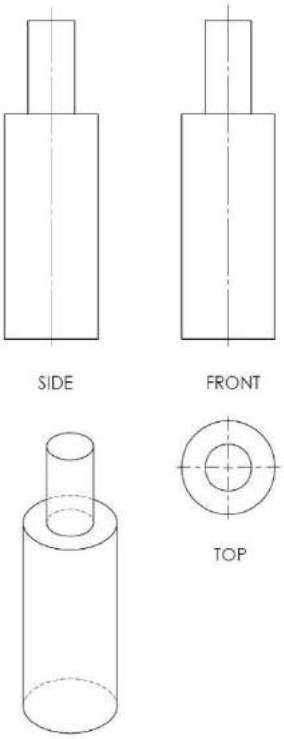


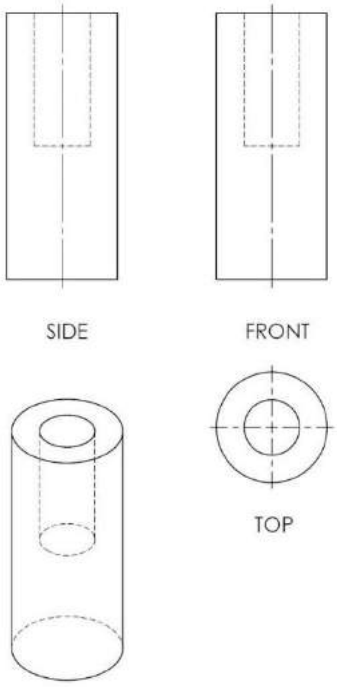
Figure 10 Projection of Flat Surfaces

Projection of Cylinders

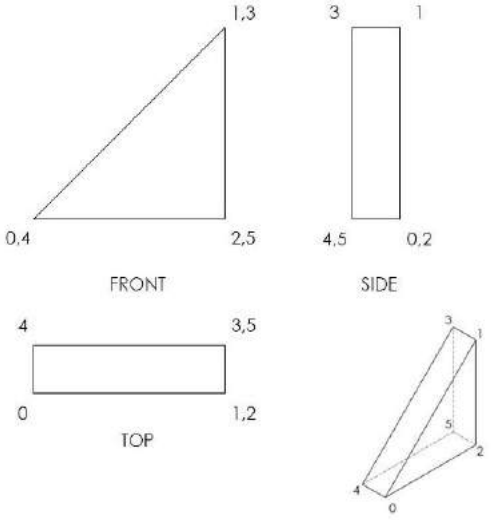


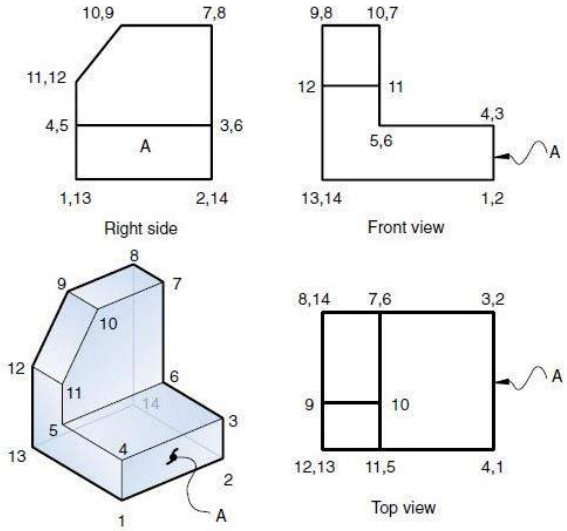


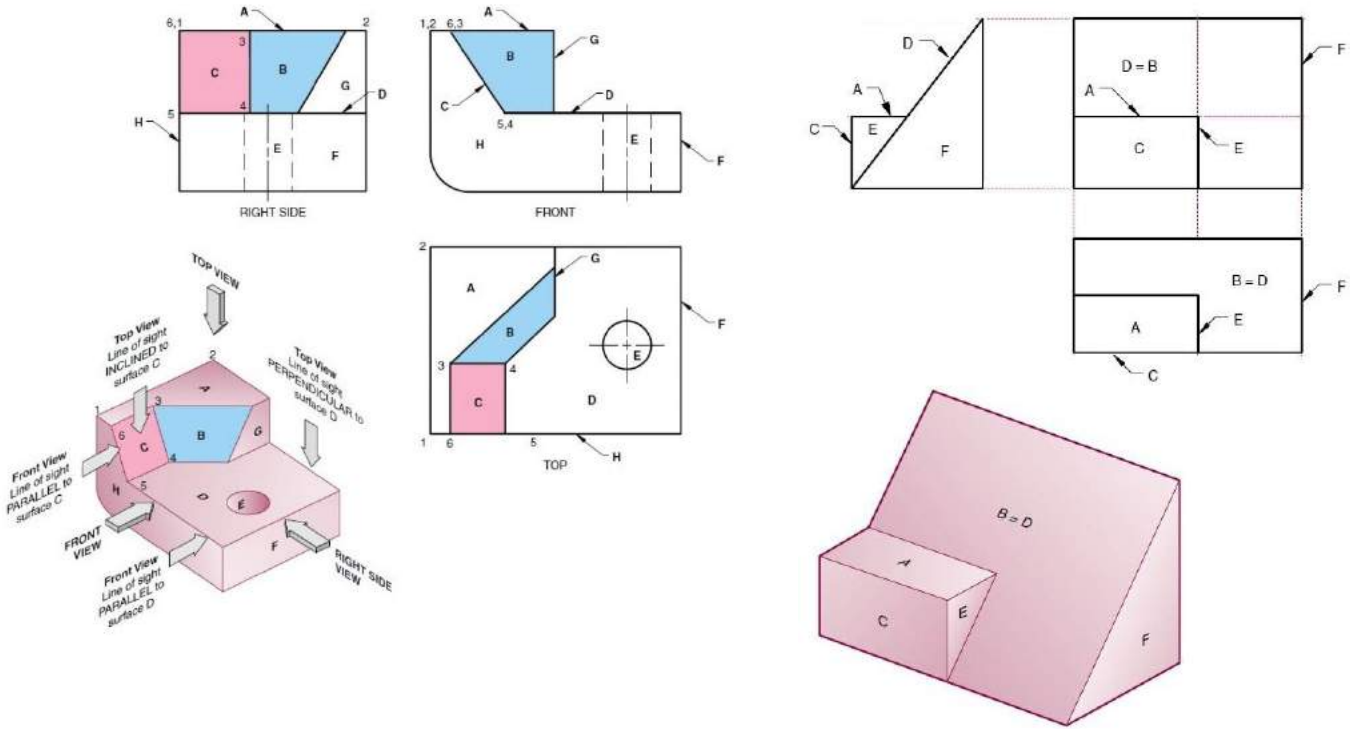




Projection of Inclined Surfaces

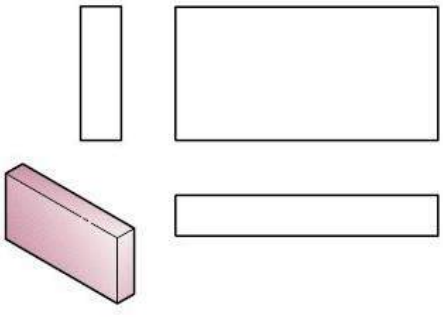




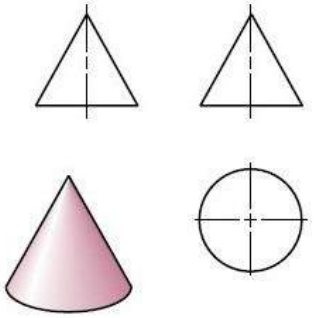


The following models show you some basic shapes, how it is seen in the main 3 views and the intersection between more than one of them

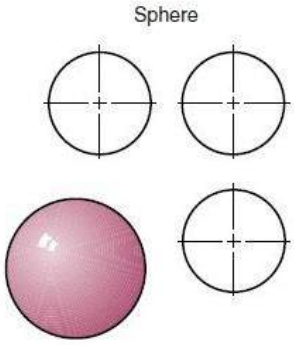
Rectangular prism



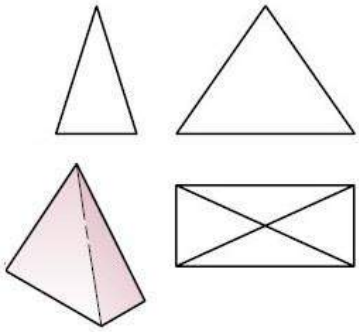
Cone



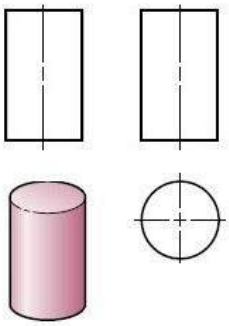
Sphere



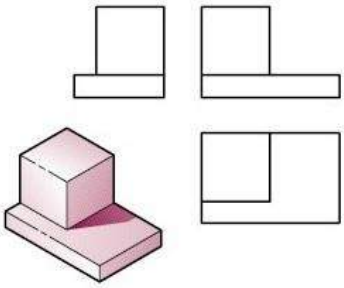
Pyramid



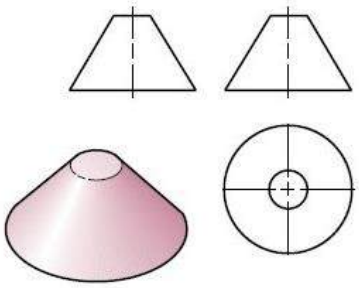
Cylinder



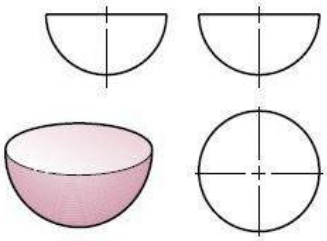
Prism and cube



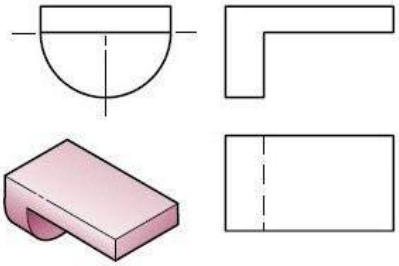
Truncated cone



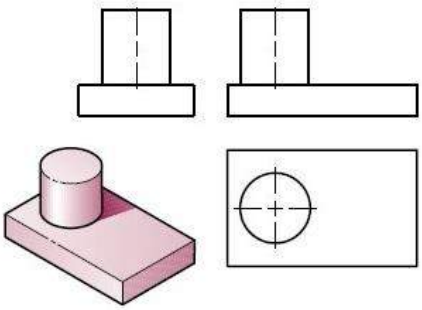
Partial sphere



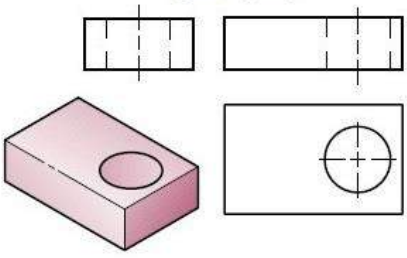
Prism and partial cylinder

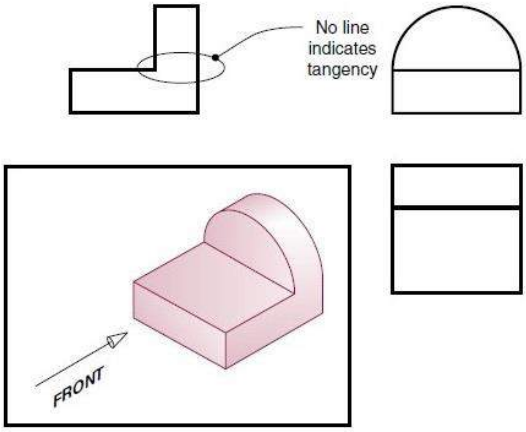


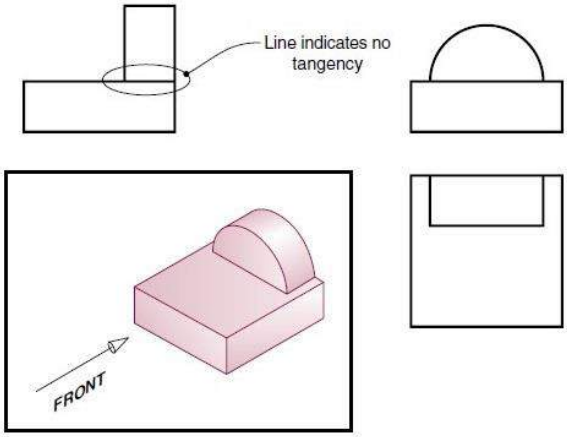
Prism and cylinder

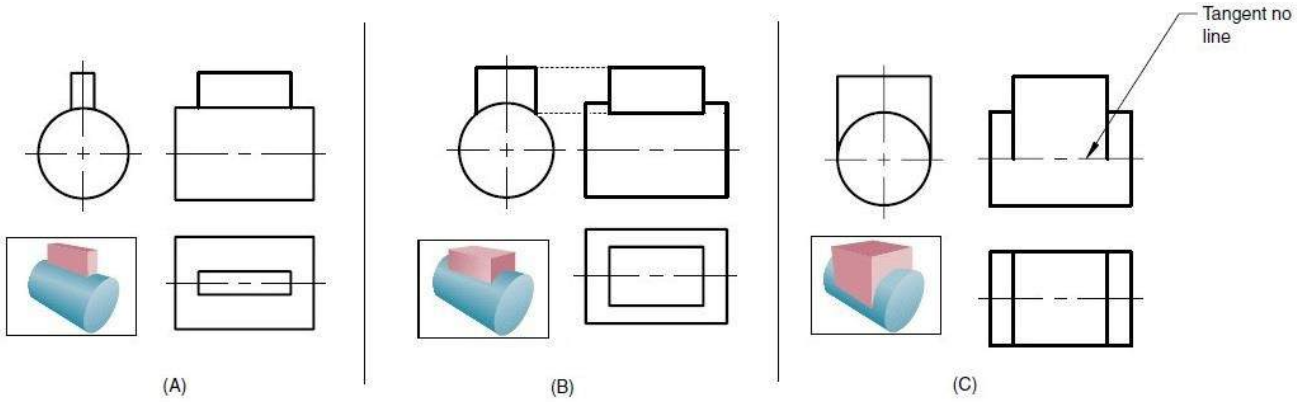


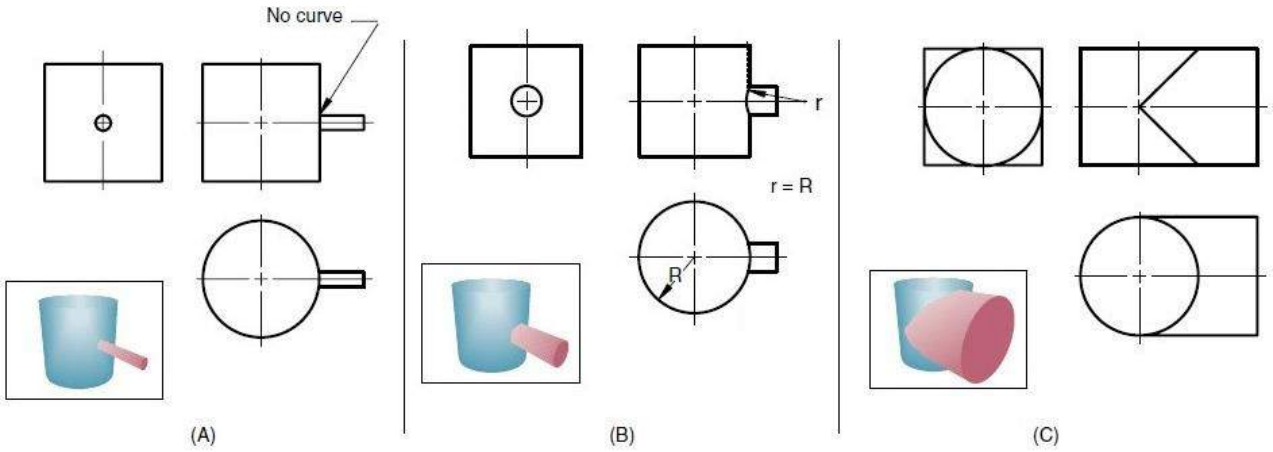
Prism and negative cylinder (hole)

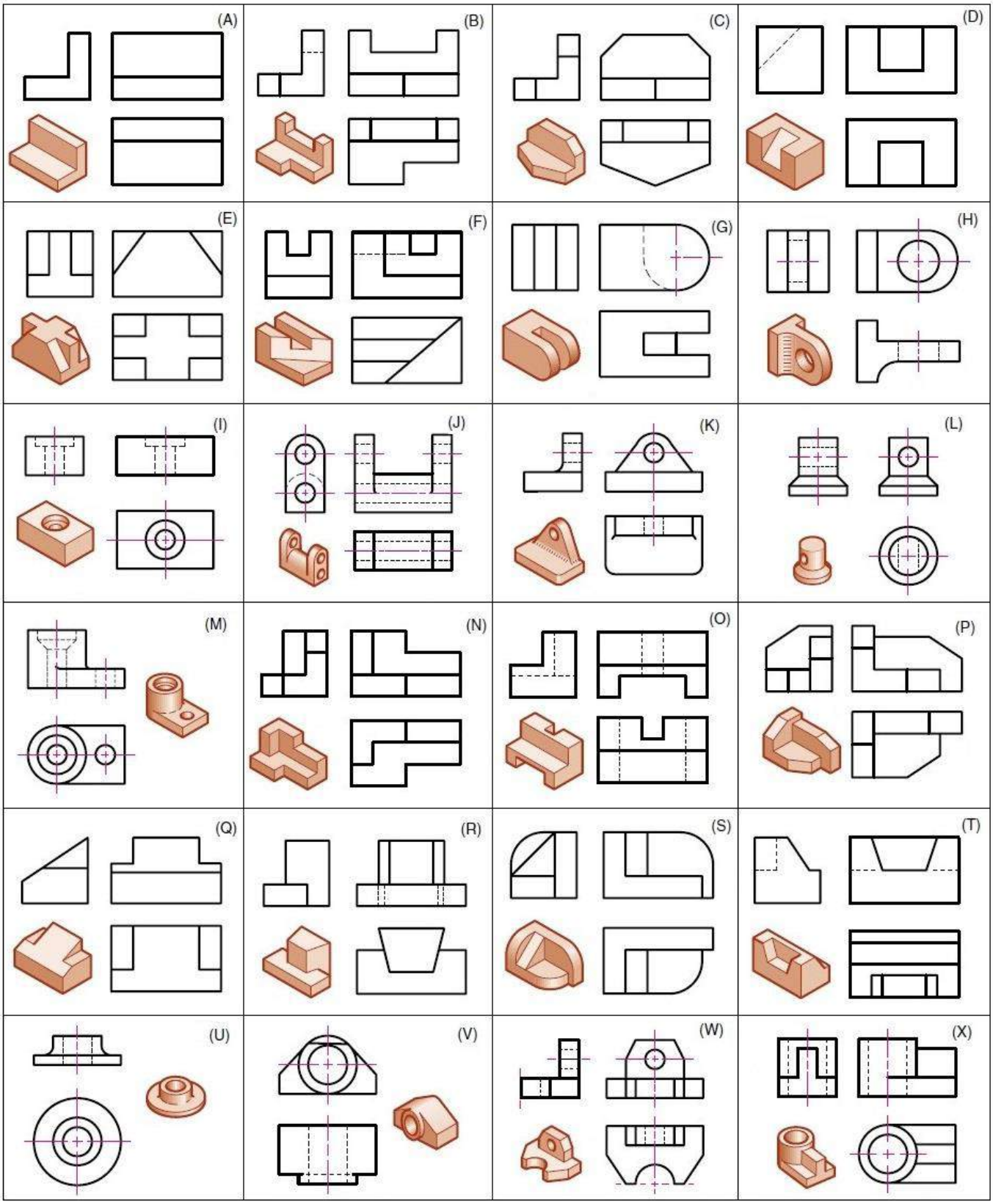


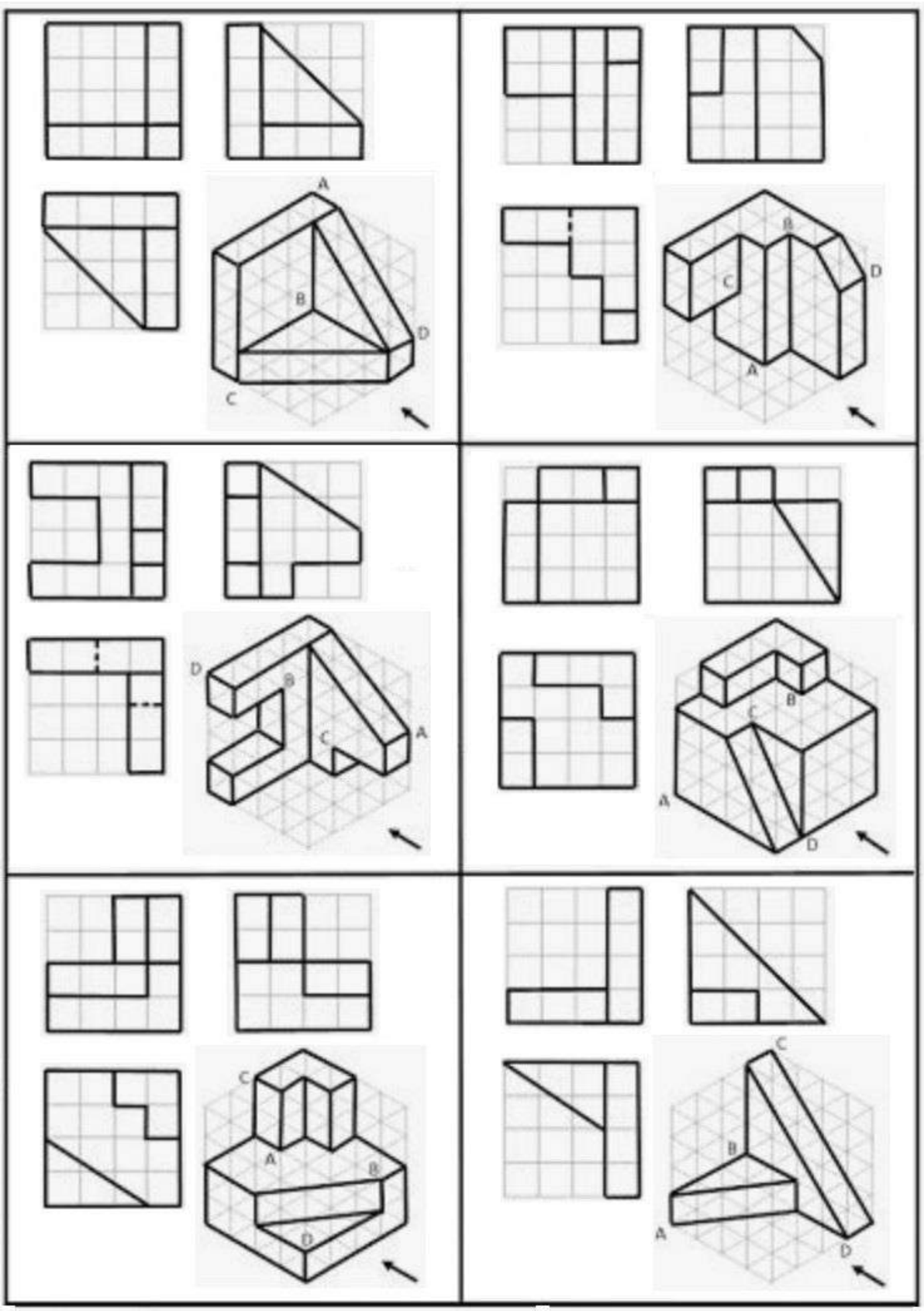

No line indicates tangency
FRONT


Line indicates no tangency
FRONT


Tangent no line
(A)
(B)
(C)


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


(A)
(B)
(C)
(D)
(E)
(F)
(G)
(H)
(I)
(J)
(K)
(L)
(M)
(N)
(O)
(P)
(Q)
(R)
(S)
(T)
(U)
(V)
(W)
(X)


A
B
C
D
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D
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C
A
C
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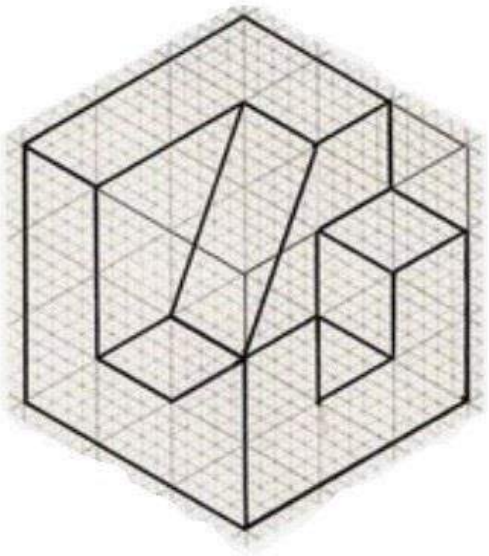
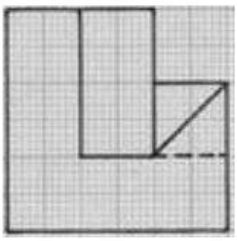
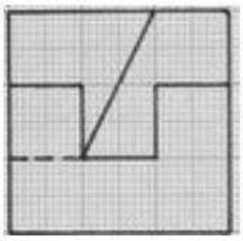
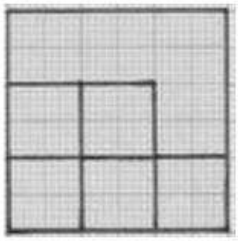
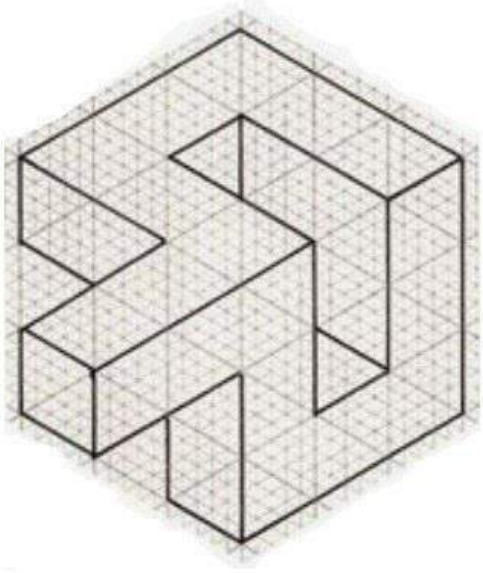
C
A
B
D
C
B
A
D

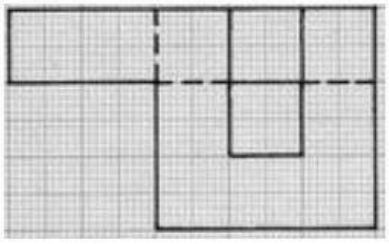


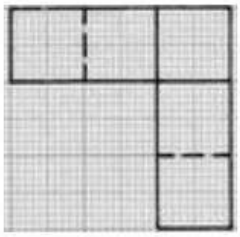


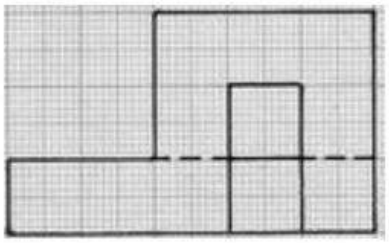




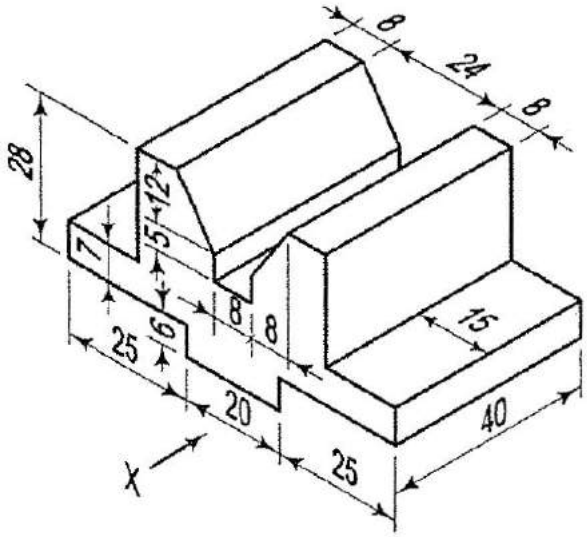


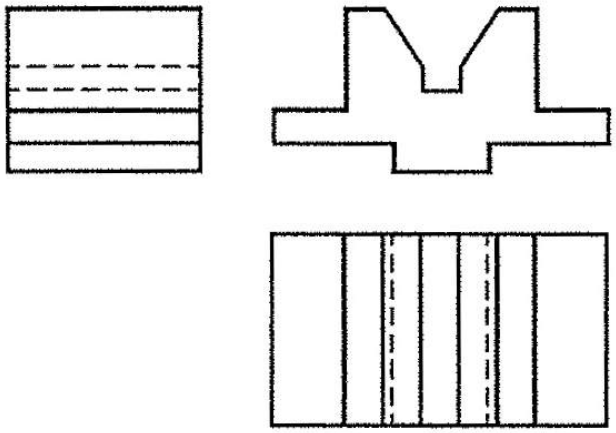


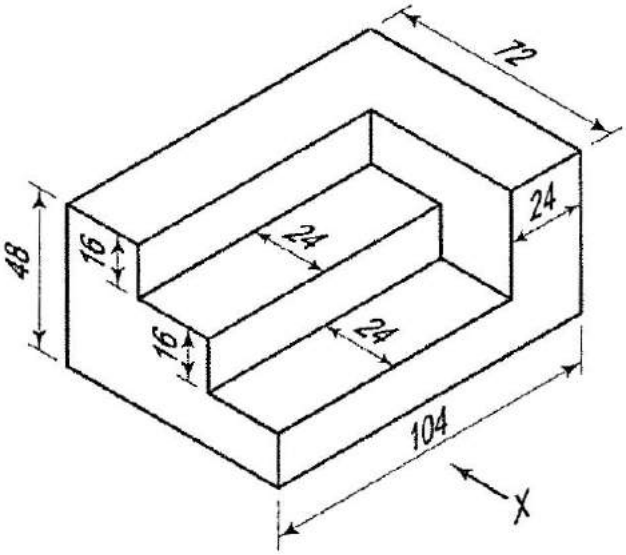


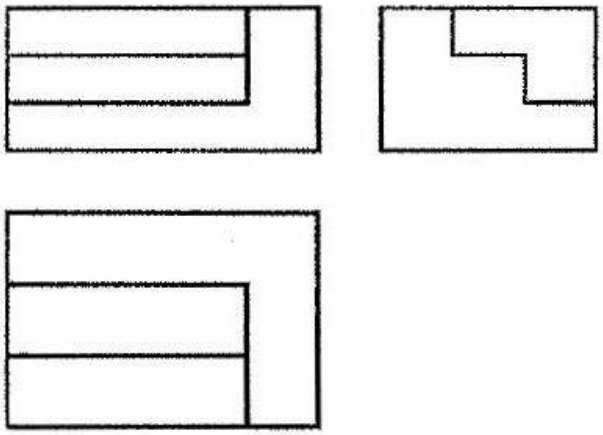


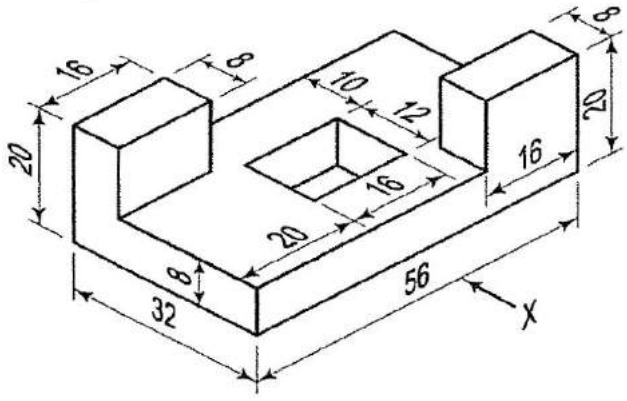
Solved Models

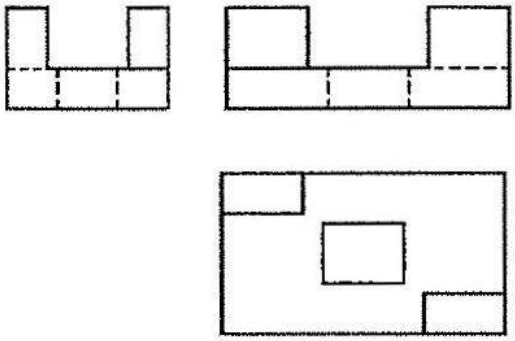


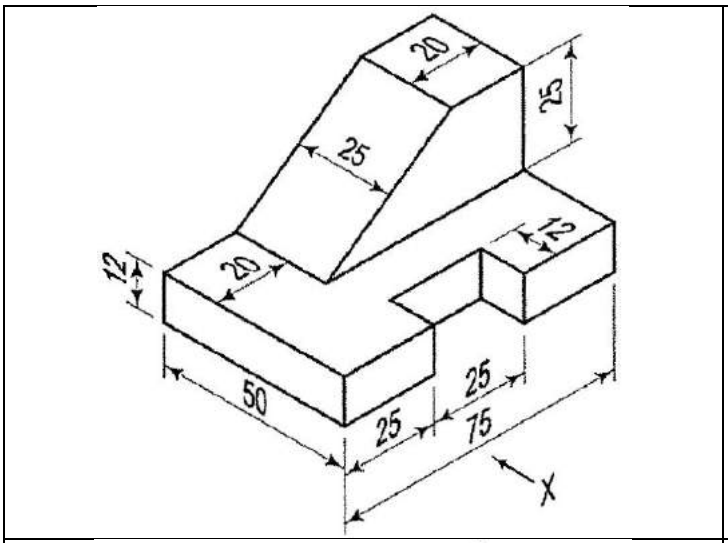








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25
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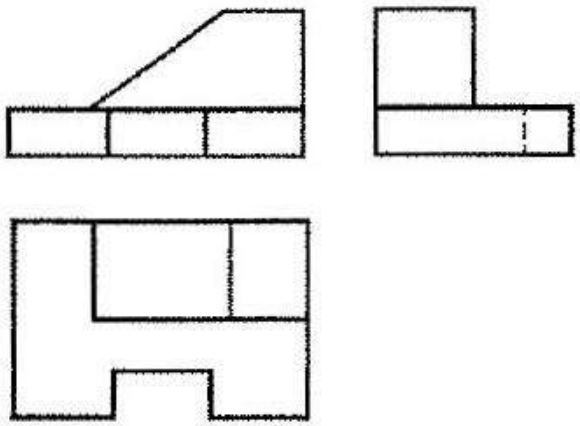
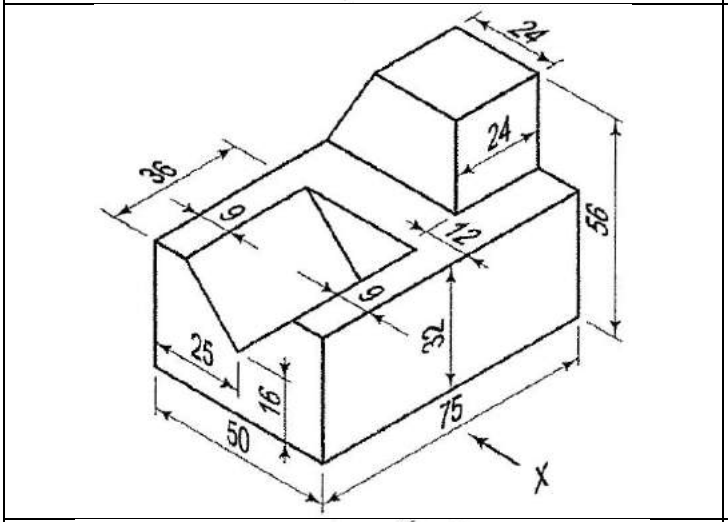
12
20
12
50
25
25
75
X

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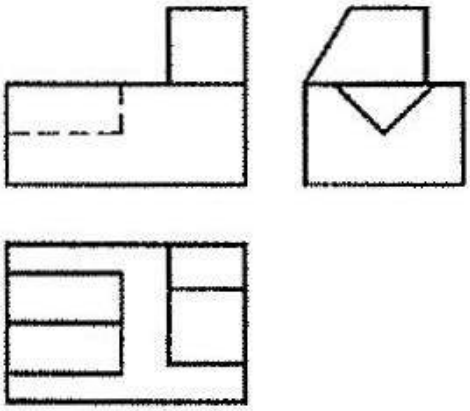

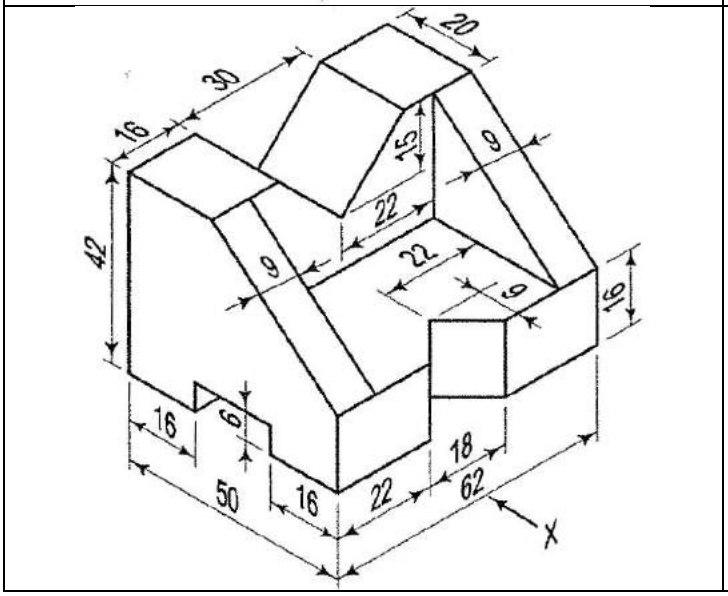
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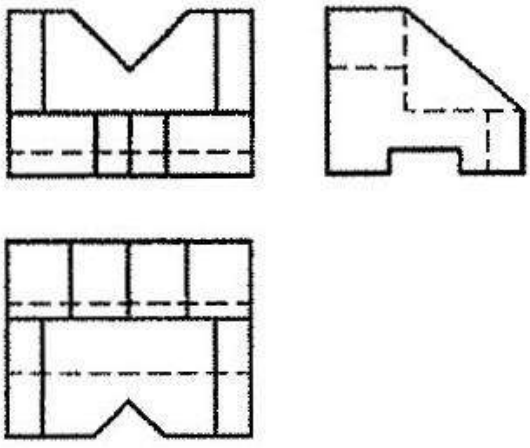

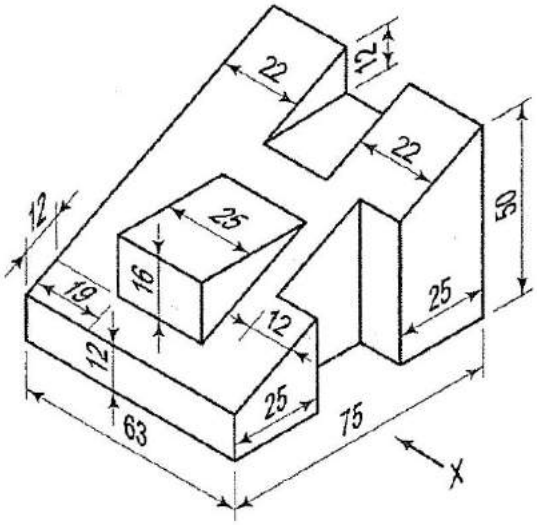
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75
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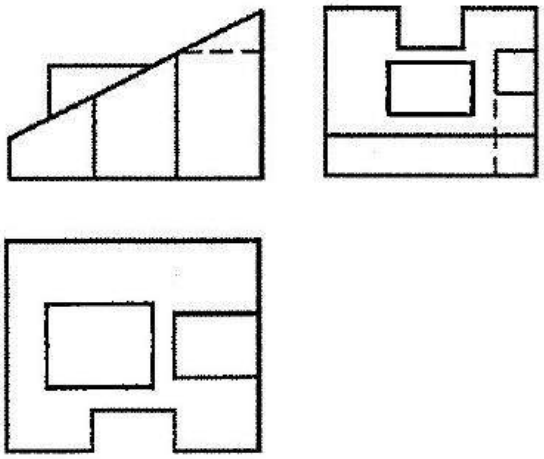

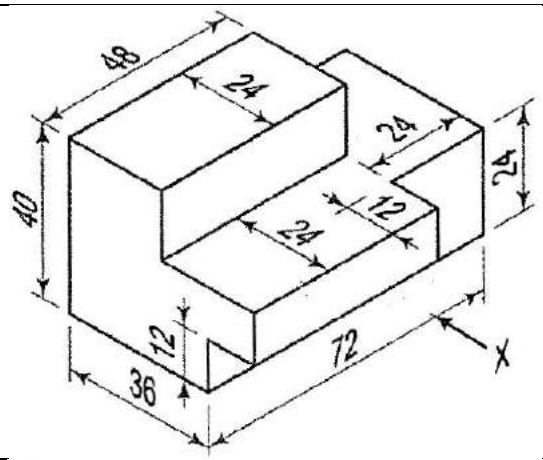
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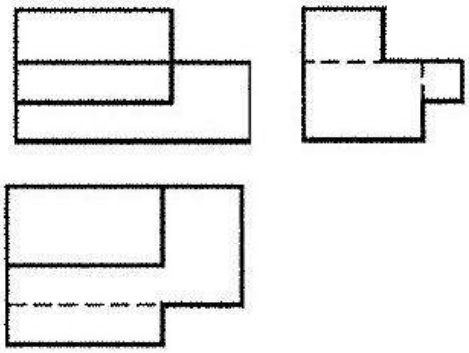

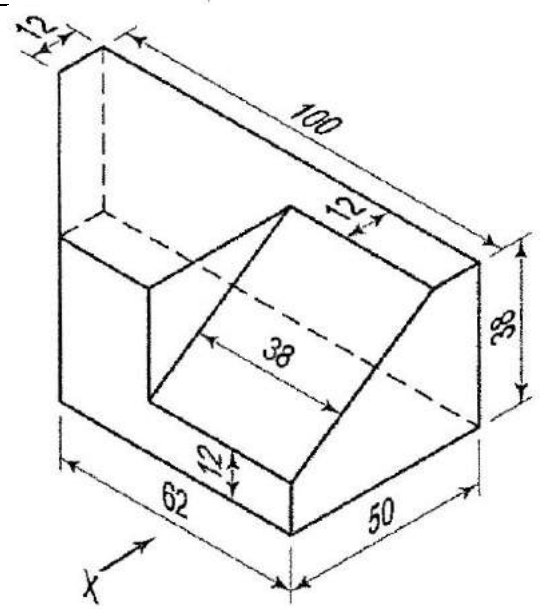
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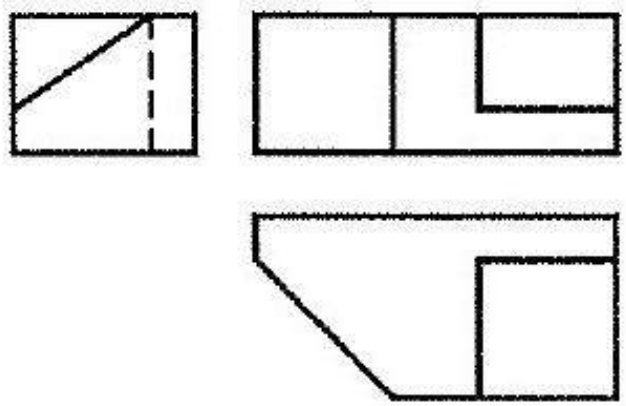

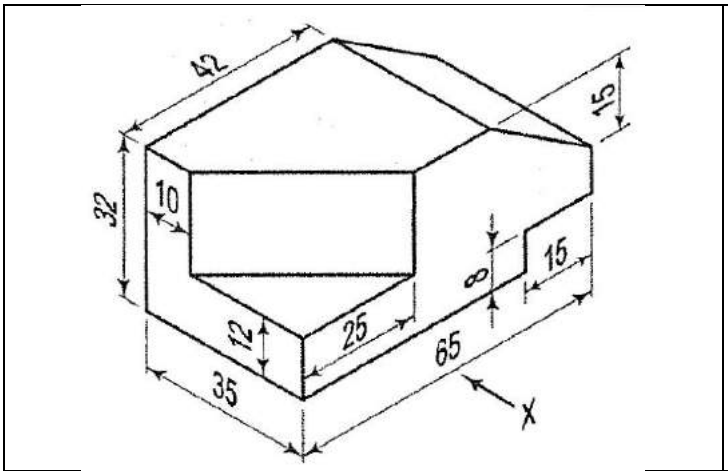
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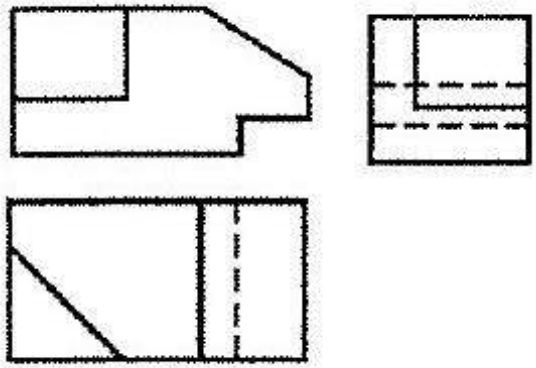

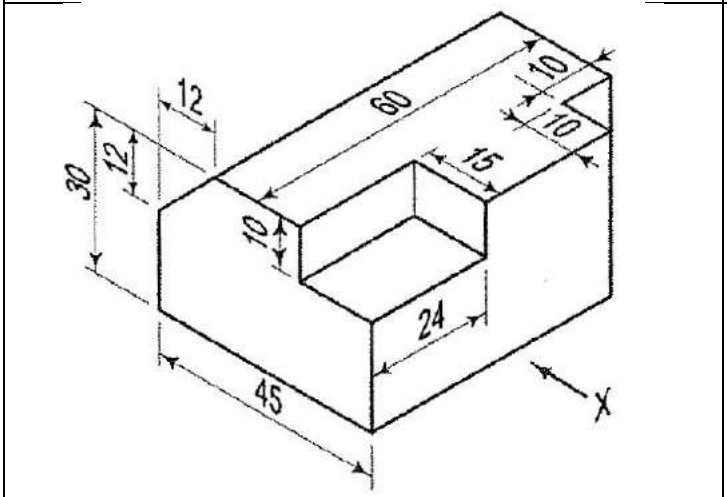
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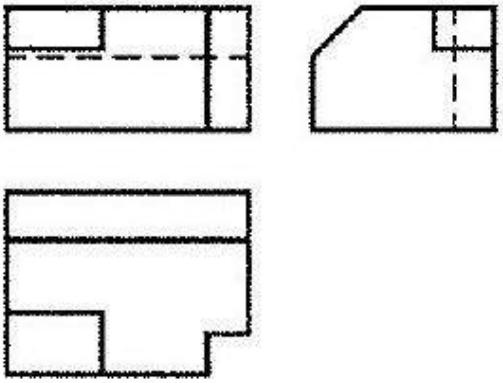

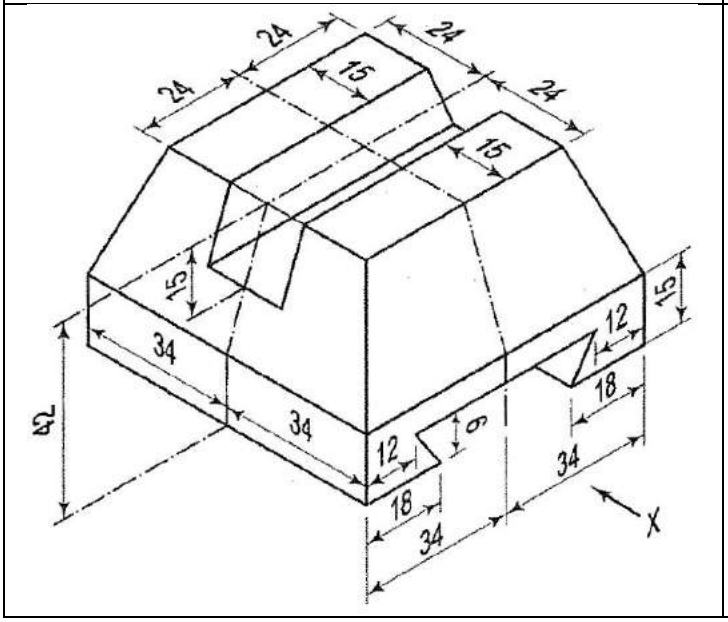
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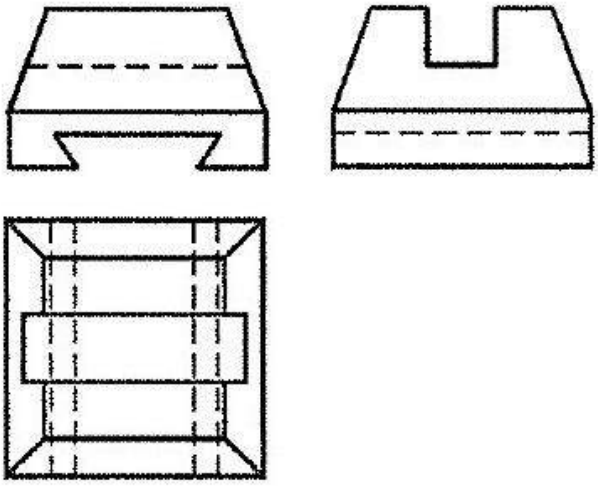

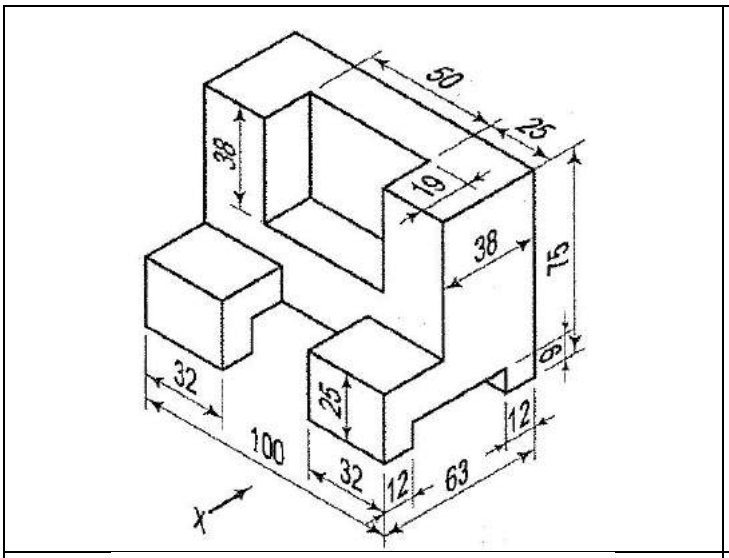
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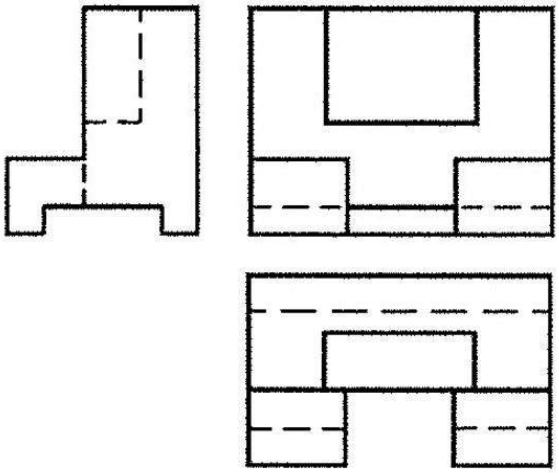

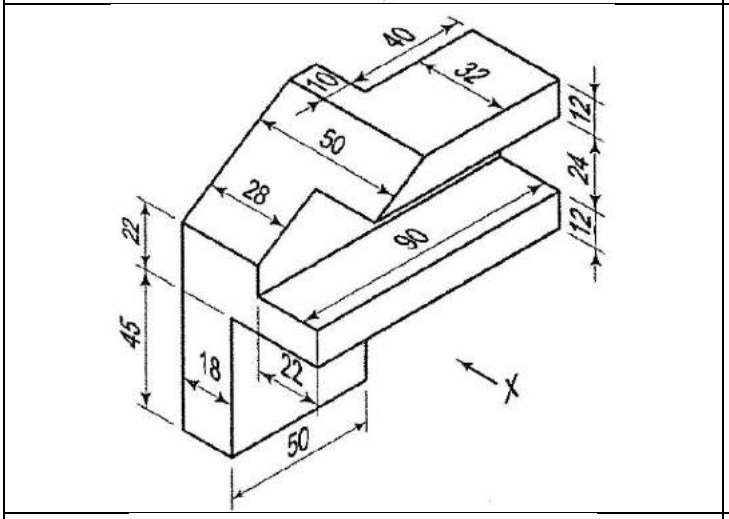
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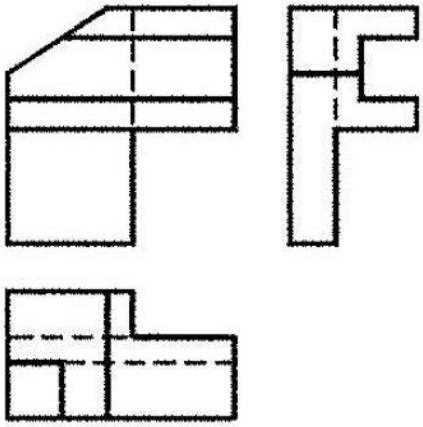

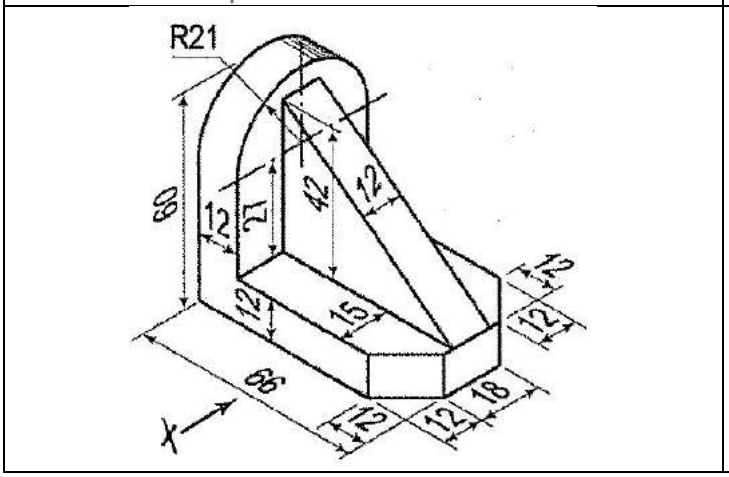
R21
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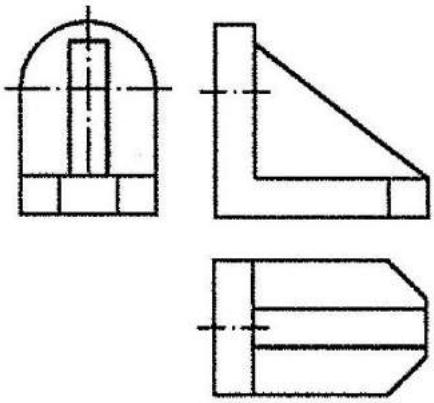
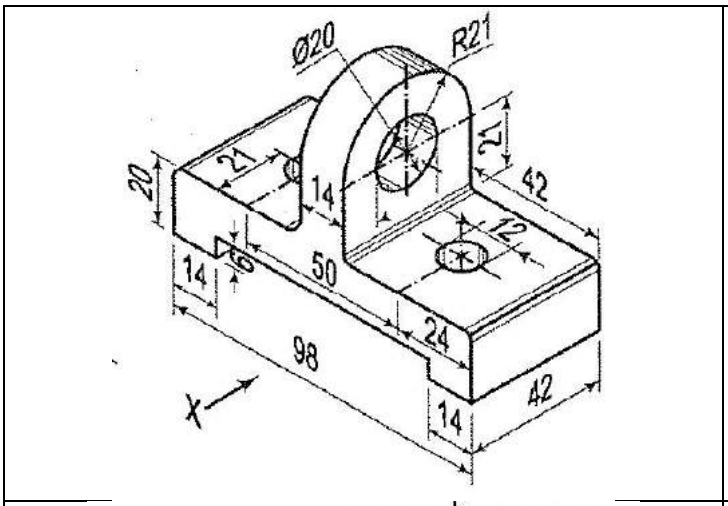
66
12
12
18
X

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98
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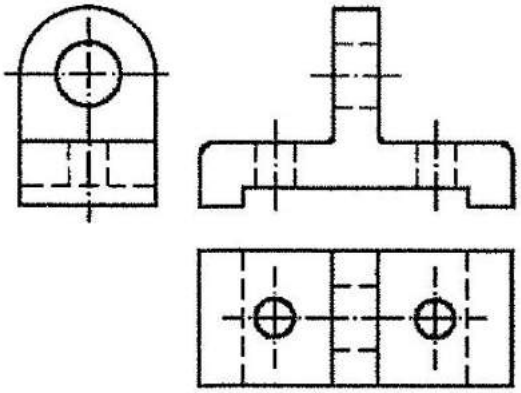
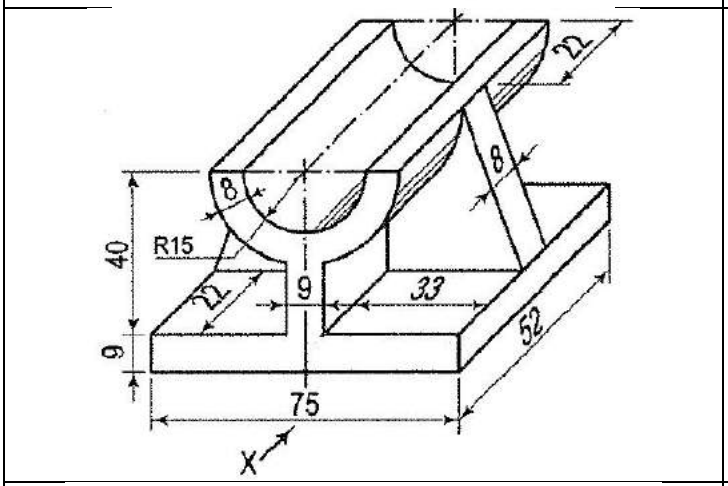
14
X

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52
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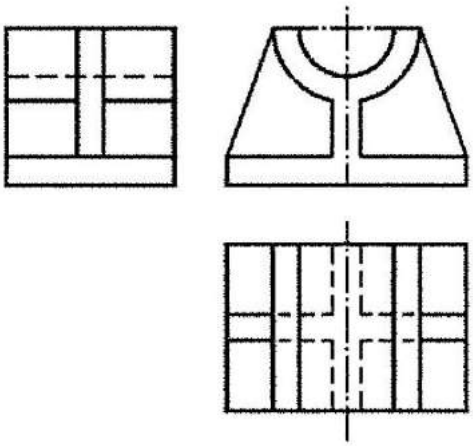
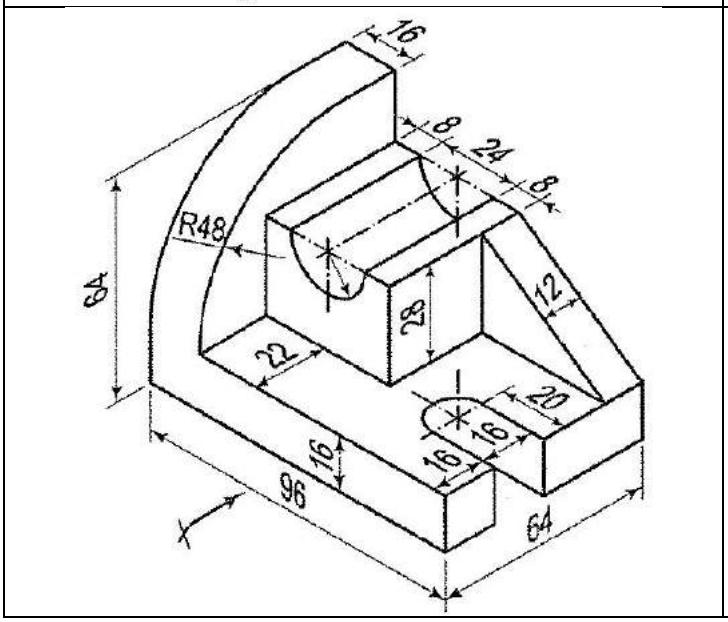
75
X

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16
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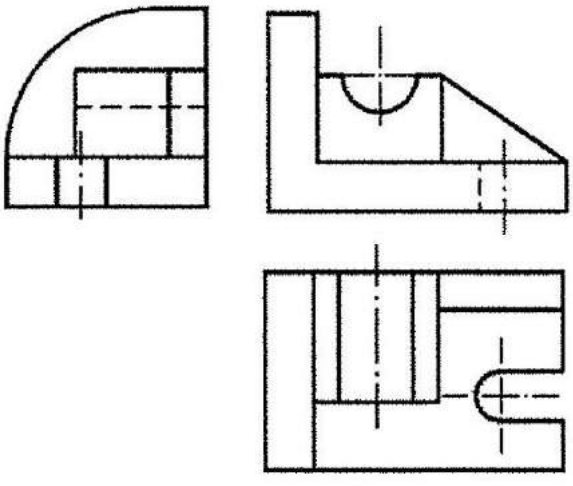

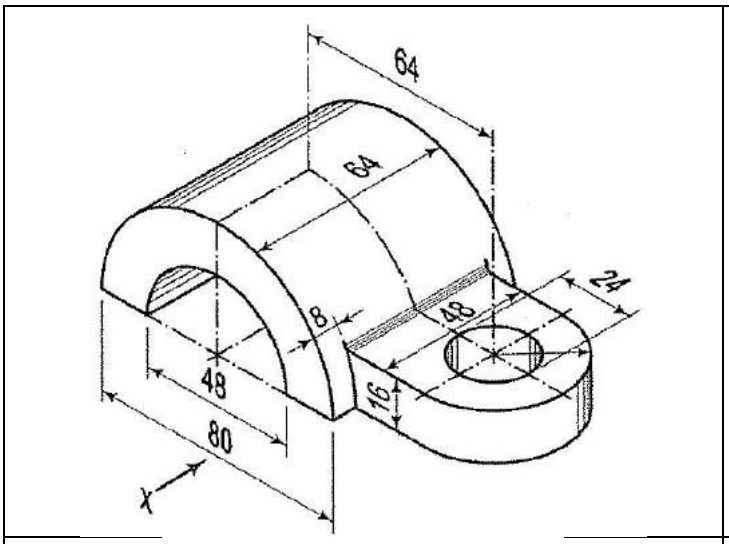
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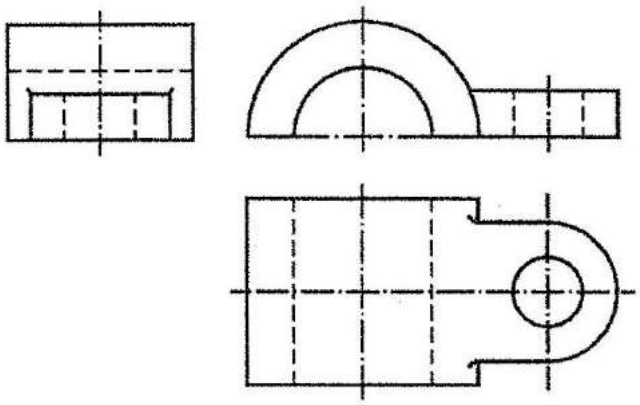
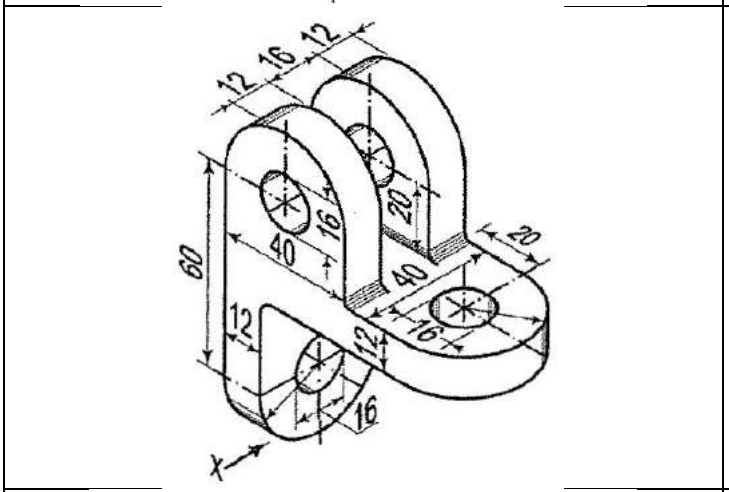
48
16
80
X

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16
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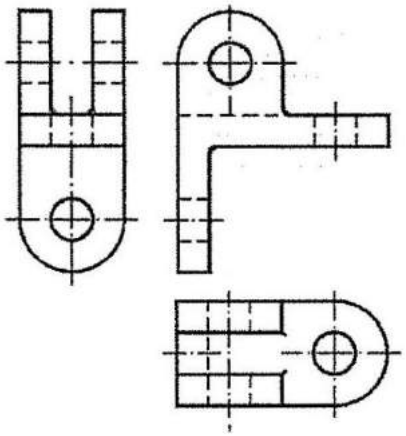

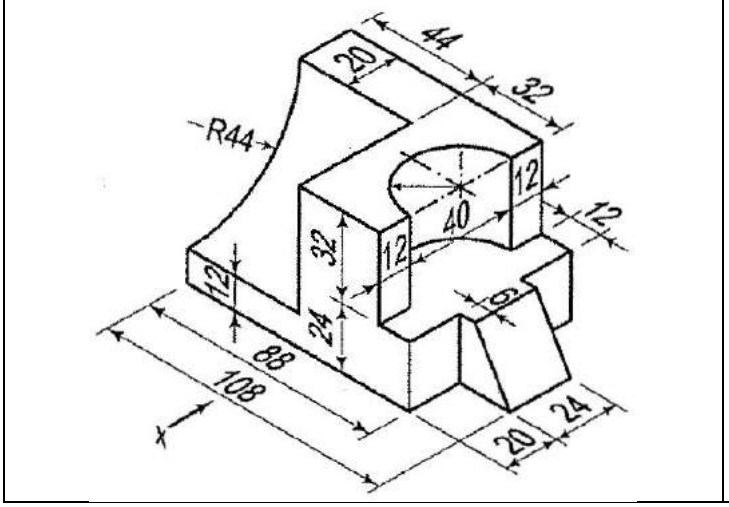
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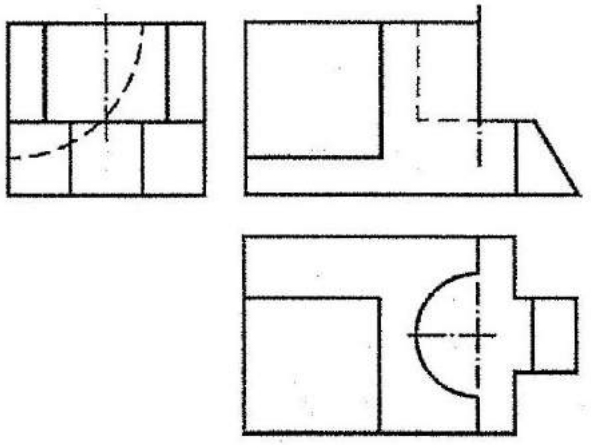

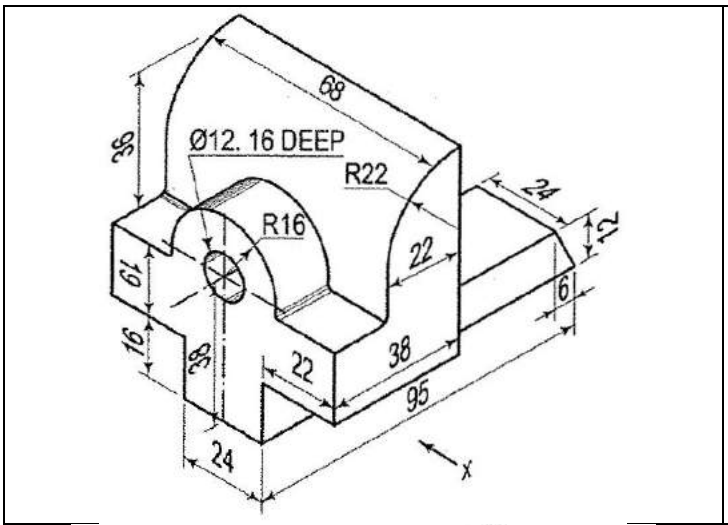
68
Ø12. 16 DEEP
R22
R16
24
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22
6
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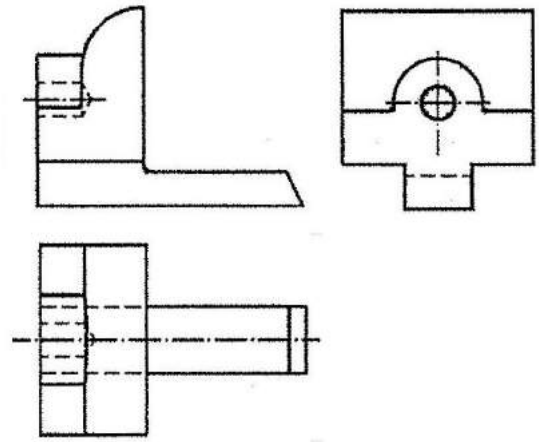
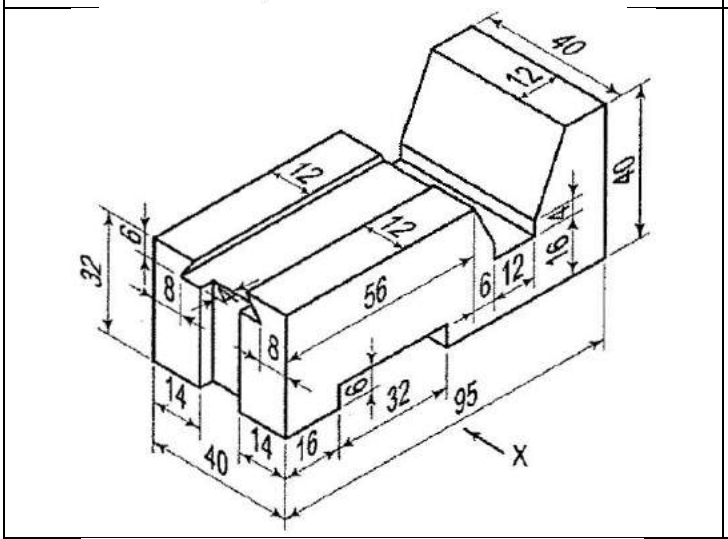
95
22
24
X

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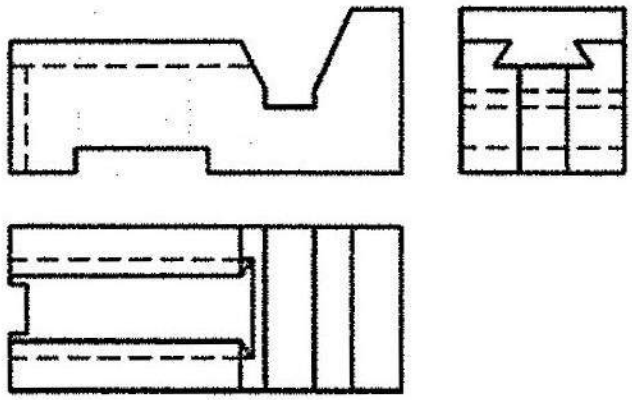
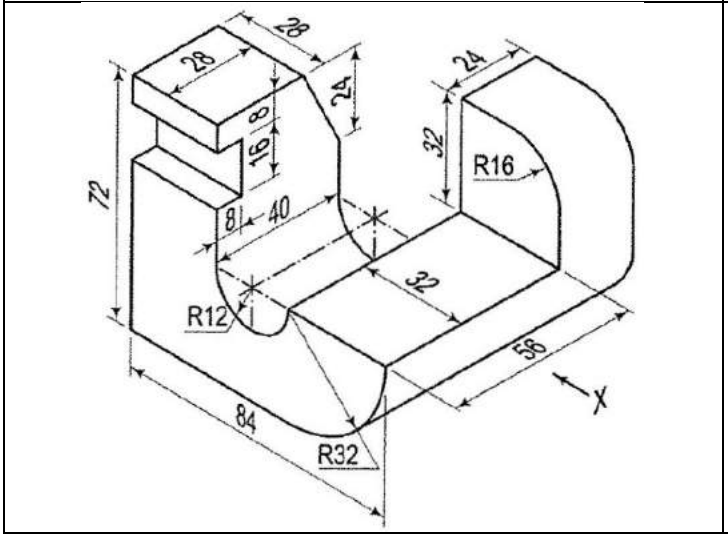
40
X

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24
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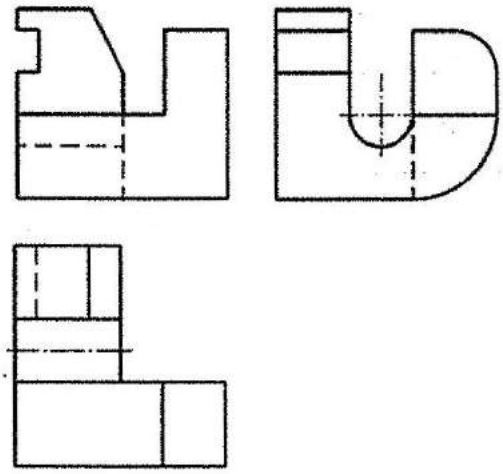
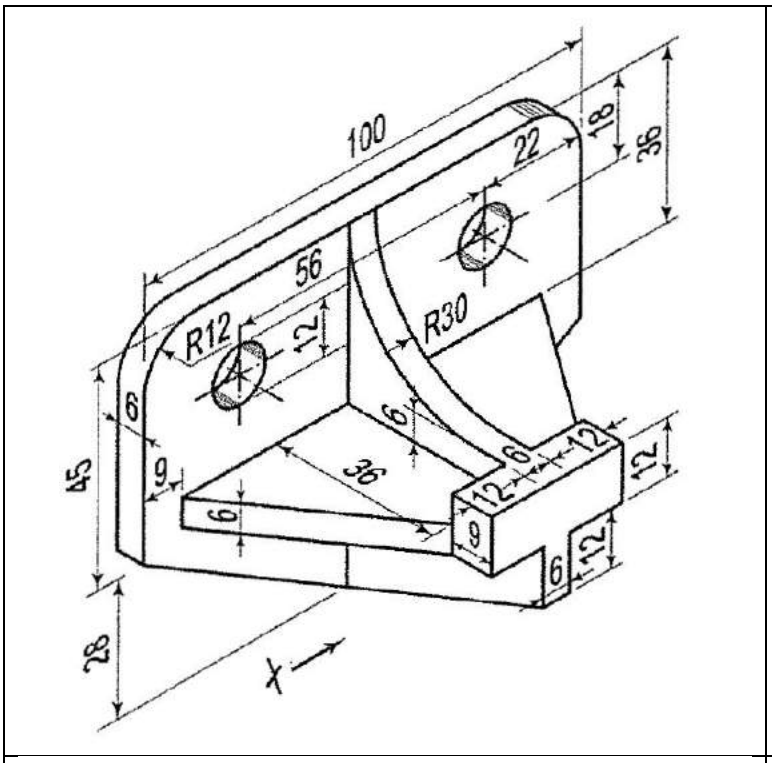
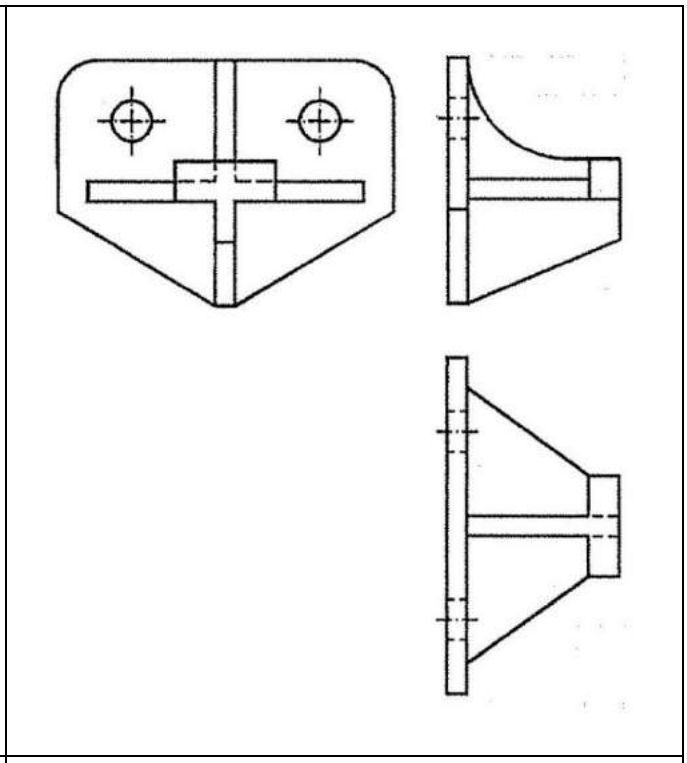
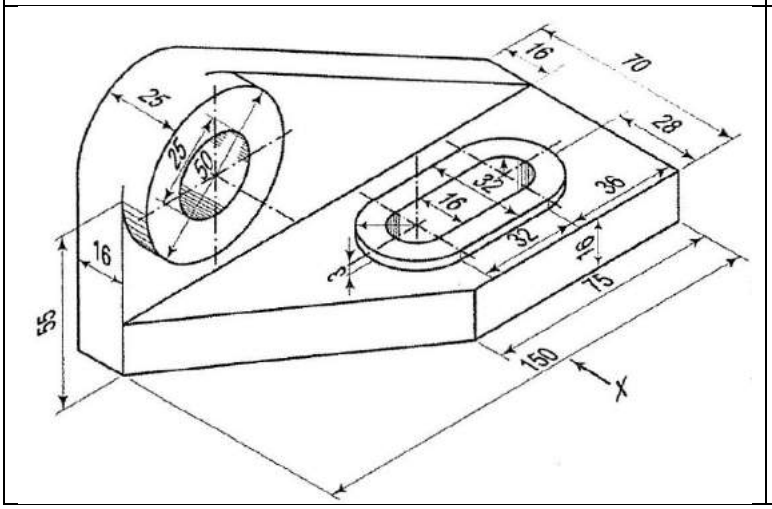
16
72
32
R16
8
40
32
R12
58
84
R32
X




100
22
18
36
56
R12
12
R30
6
45
9
36
6
12
12
9
6
12
28
X




16
70
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28
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36
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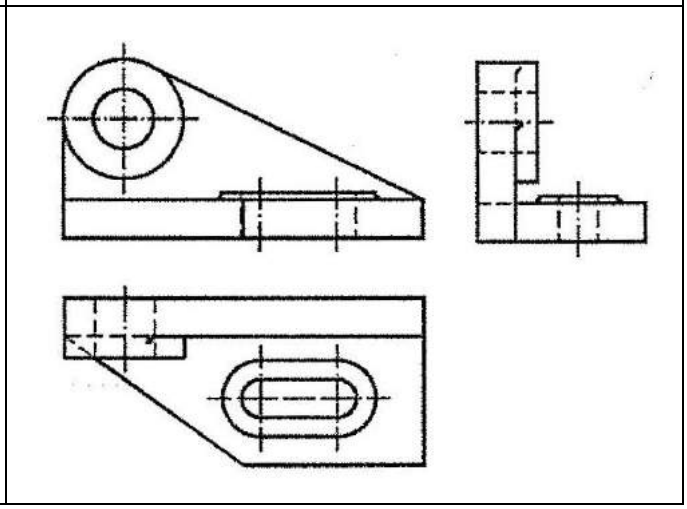
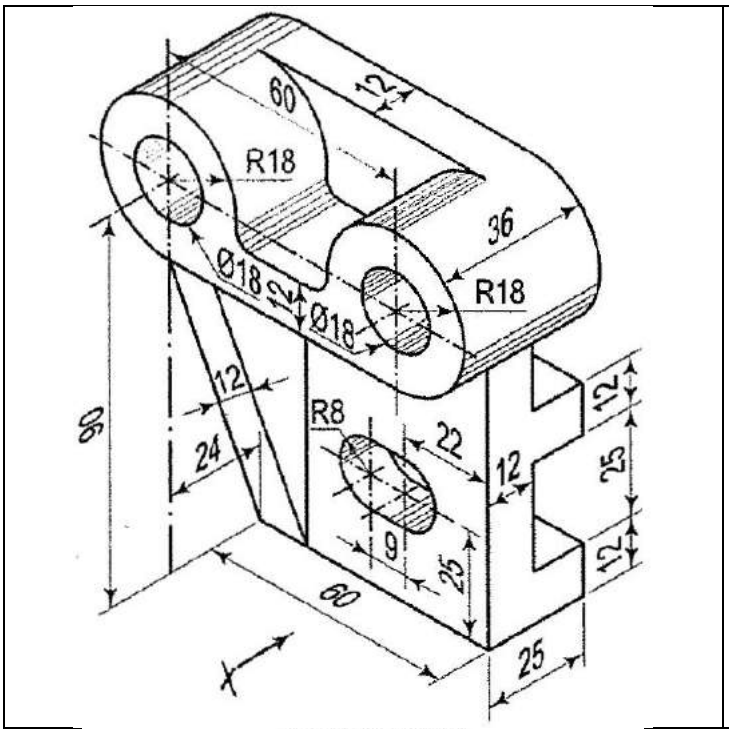
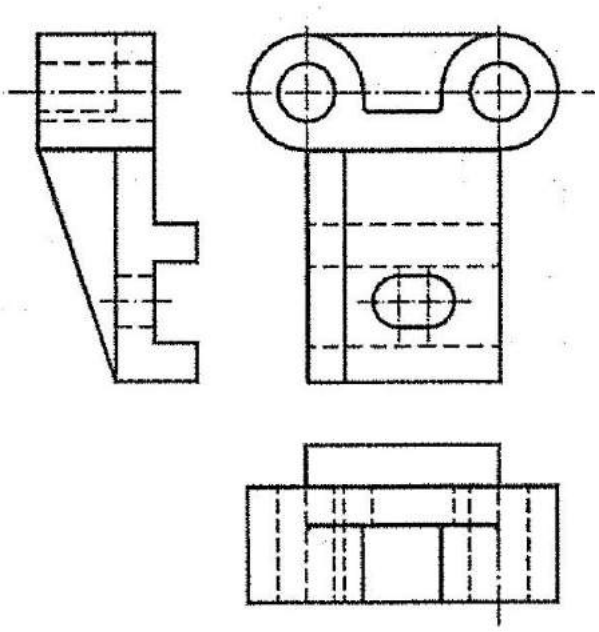
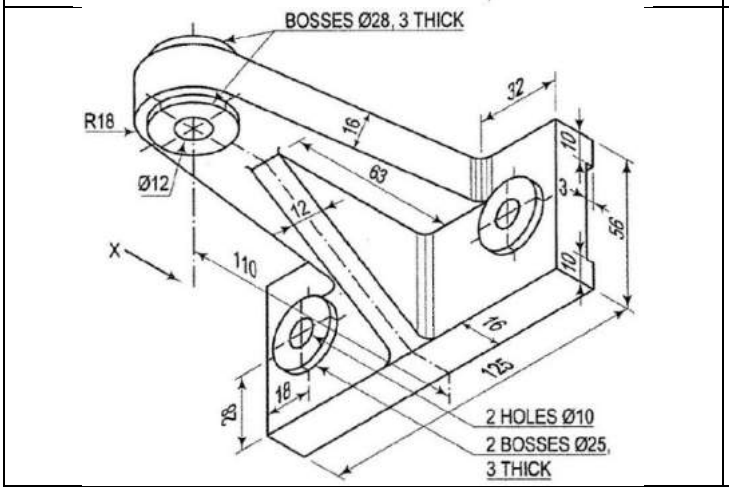
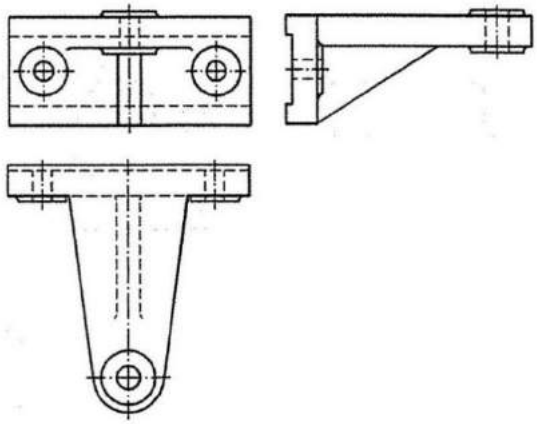
3
40
75
55
150
X

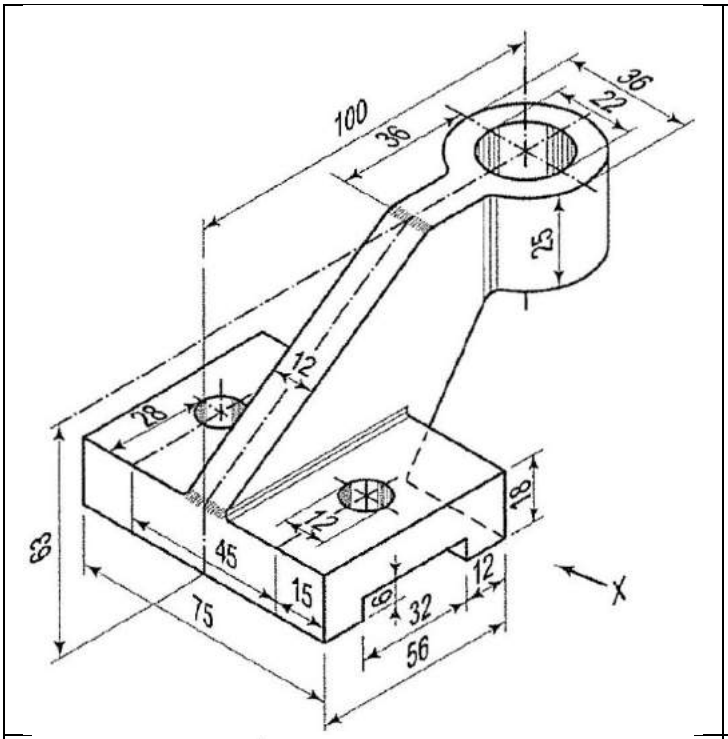



60
12
R18
36
Ø18
Ø18
R18
12
R8
22
12
24
80
12
25
9
25
12
60
25
X




BOSSES Ø28, 3 THICK
R18
Ø12
32
16
63
12
110
X
10
3
10
55
16
125
18
28
2 HOLES Ø10
2 BOSSES Ø25,
3 THICK




100
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35
22
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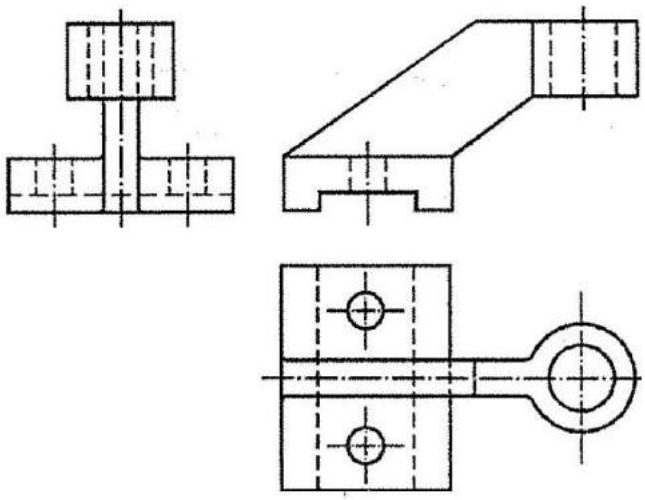
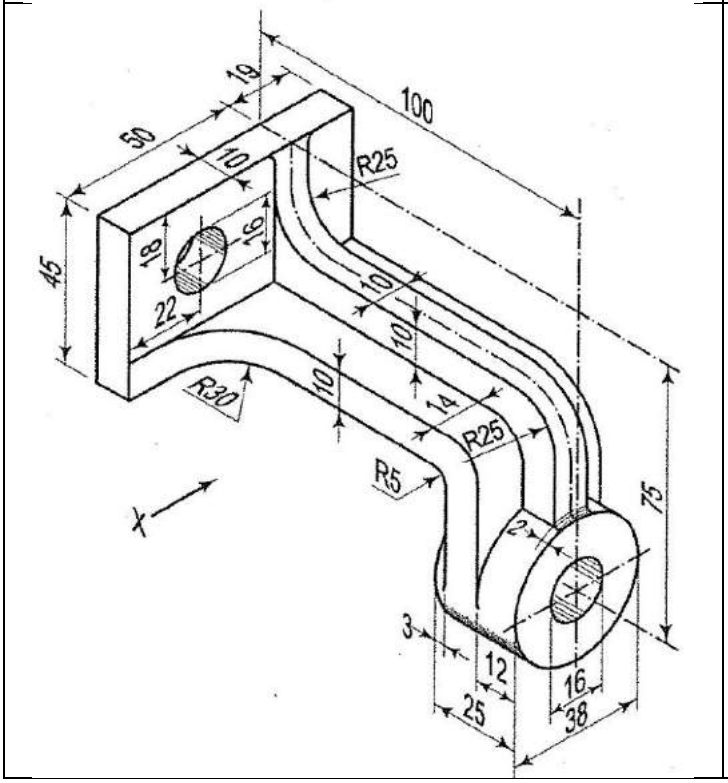
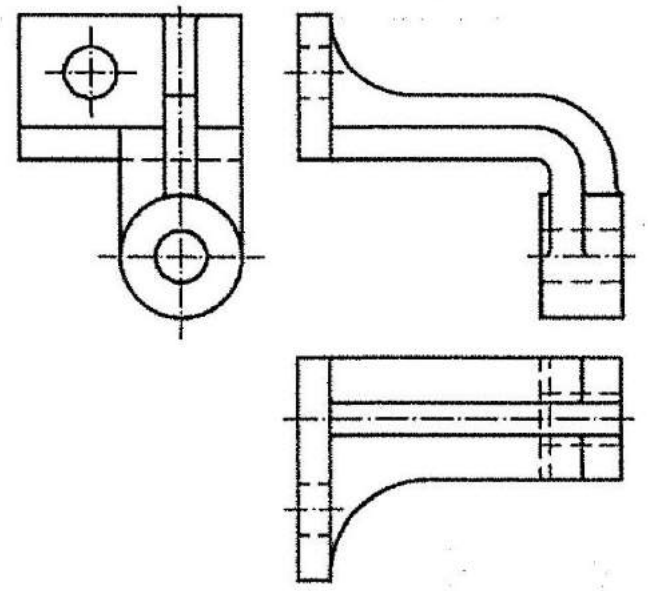
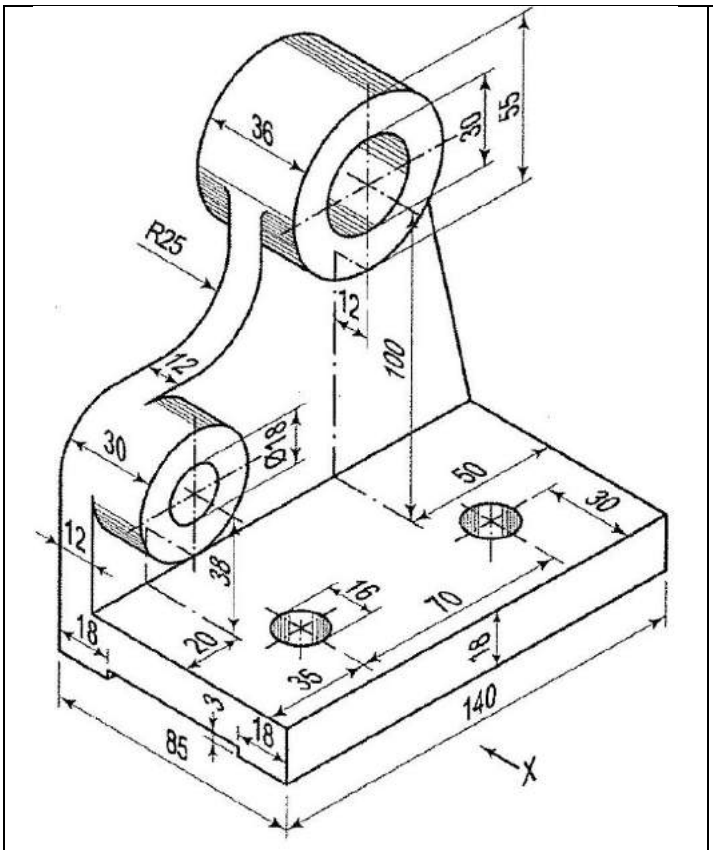
12
28
63
45
15
75
12
18
12
8
32
56
X




100
19
50
10
R25
45
19
19
22
10
10
R30
10
14
R25
R5
75
X
2
3
12
25
16
38




36
30
55
R25
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100
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Ø18
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16
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18
20
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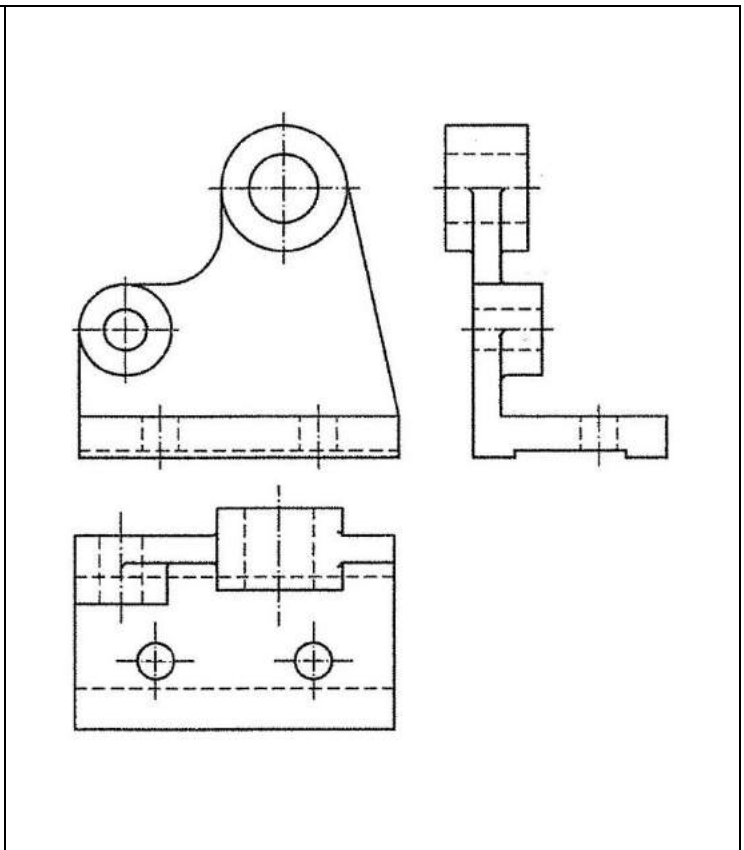
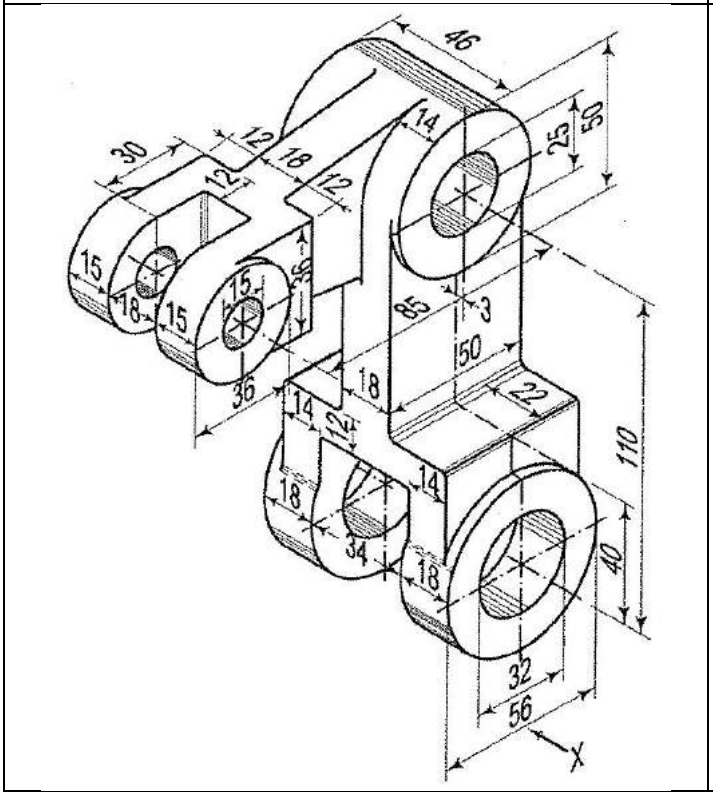
35
3
18
140
85
X




46
30
12
18
12
14
25
50
12
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15
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150
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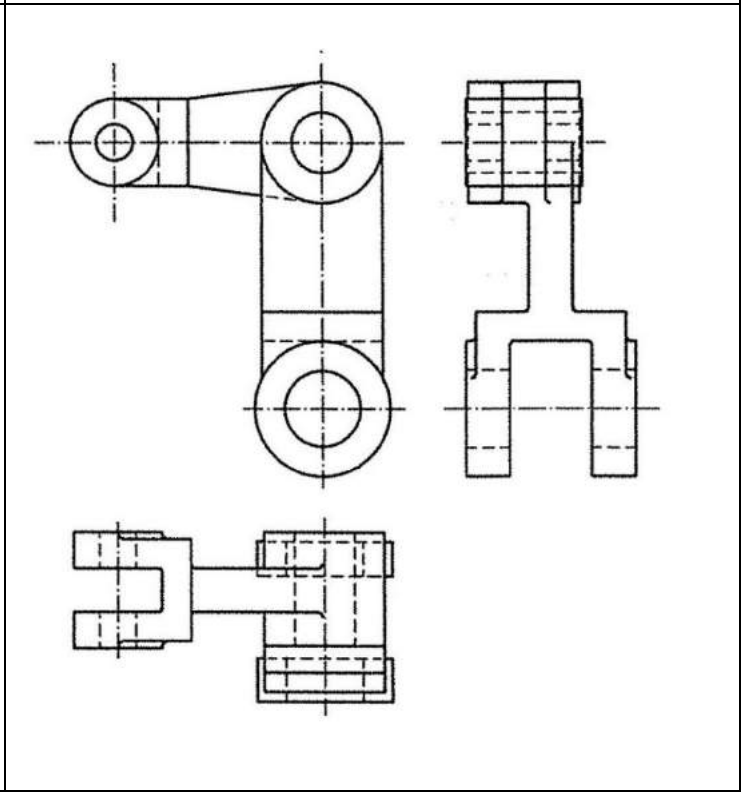
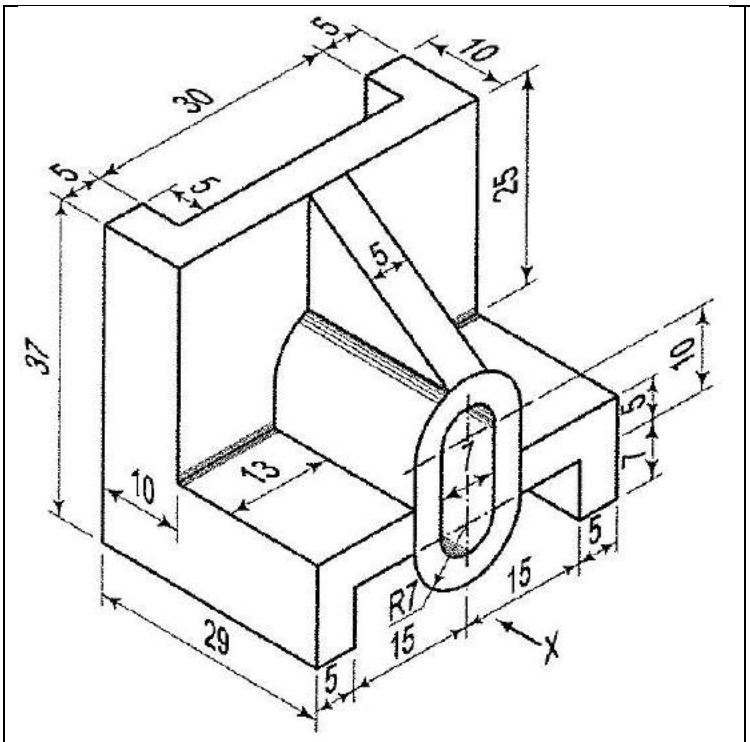
32
56
X




5
10
30
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5
25
5
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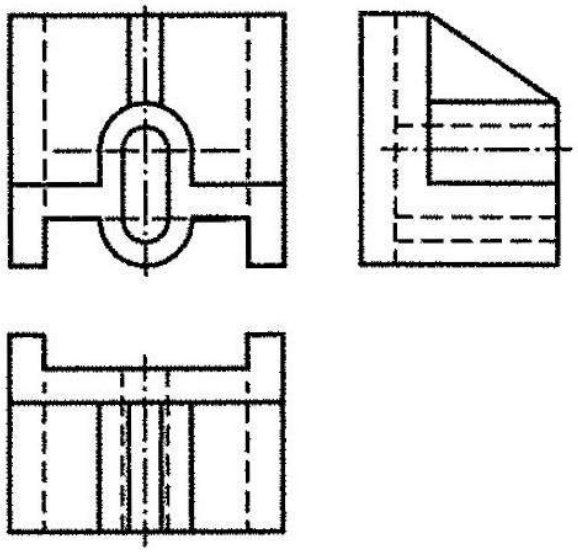
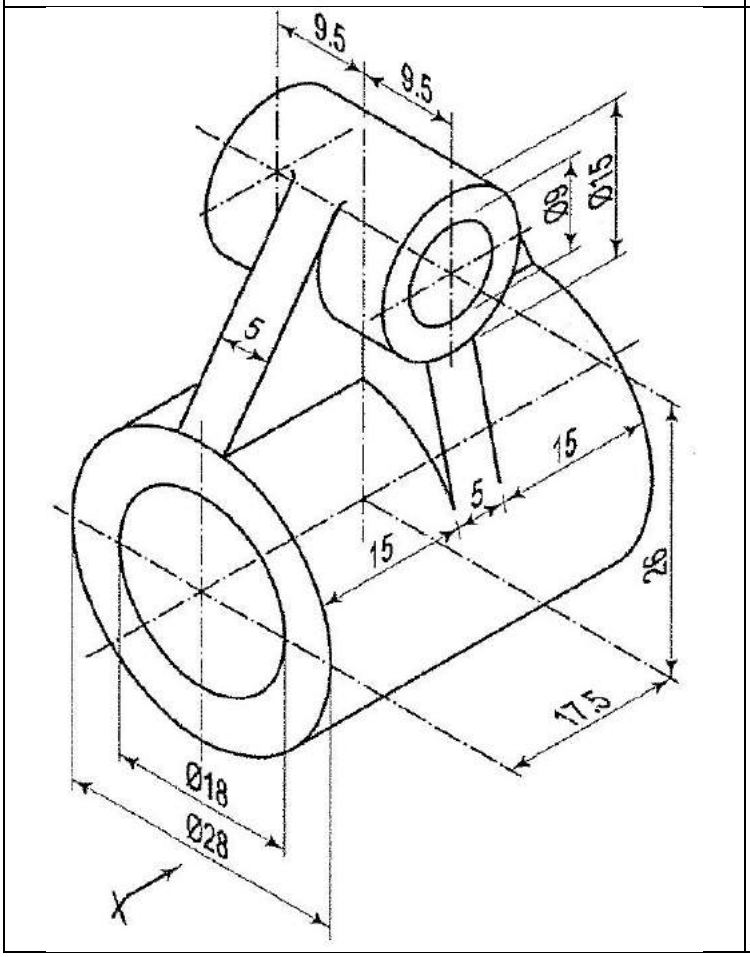
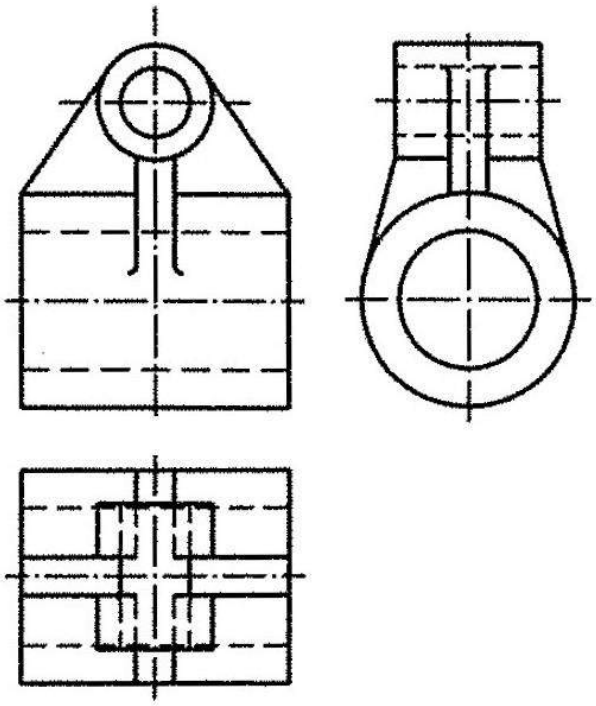
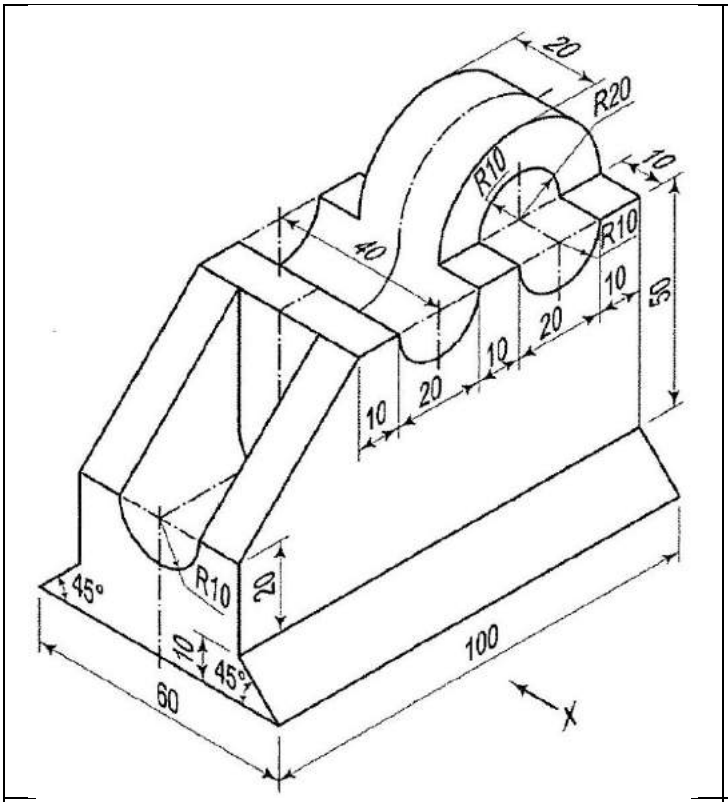
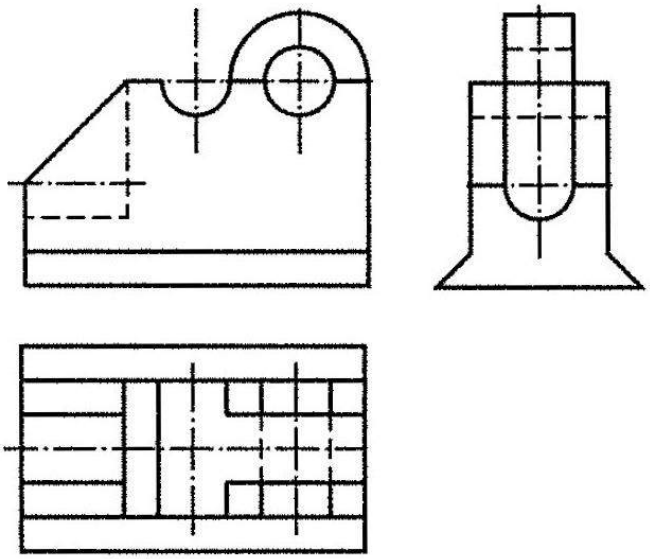
7
10
13
5
R7
15
29
5
15
X

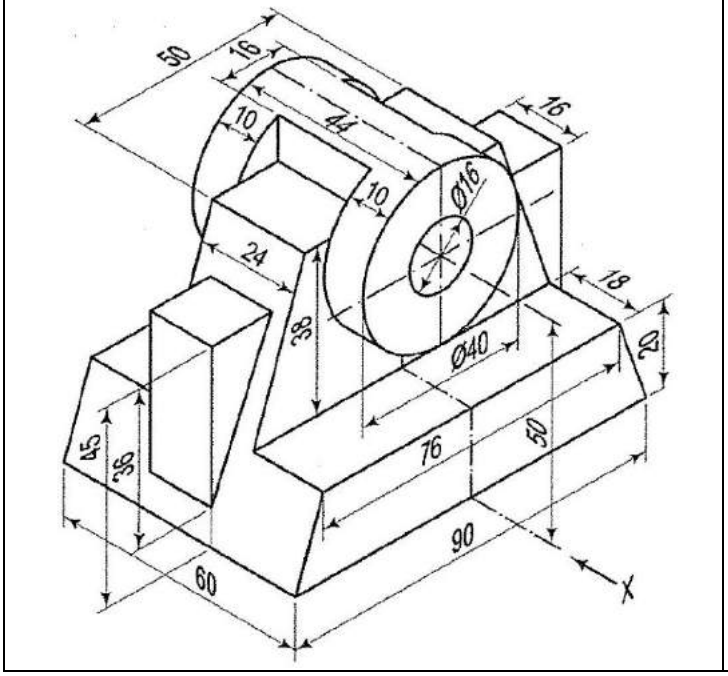


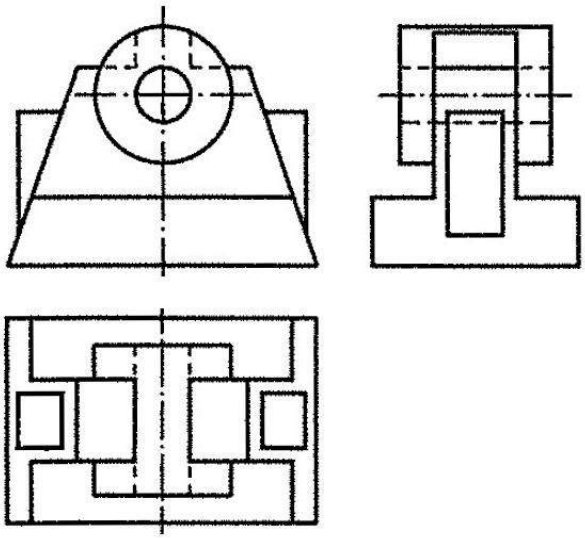

9.5
9.5
Ø8
Ø15
5
15
5
15
26
17.5
Ø18
Ø28
X

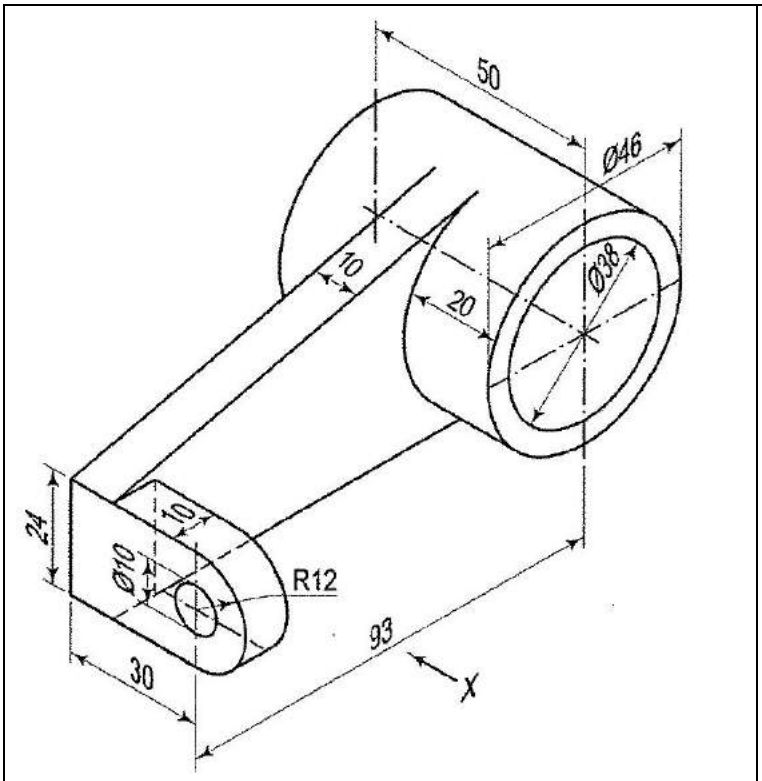


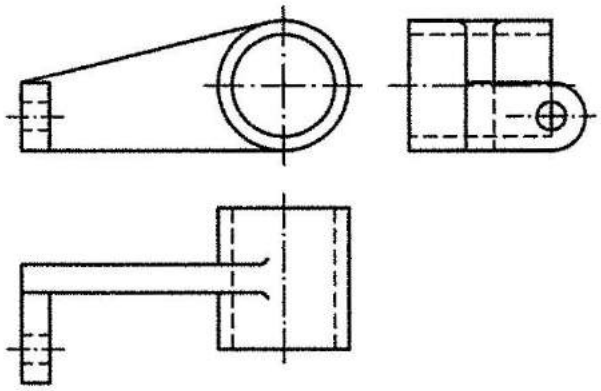

20
R20
R10
10
40
R10
10
50
10
20
10
20
45°
R10
20
10
45°
60
100
X

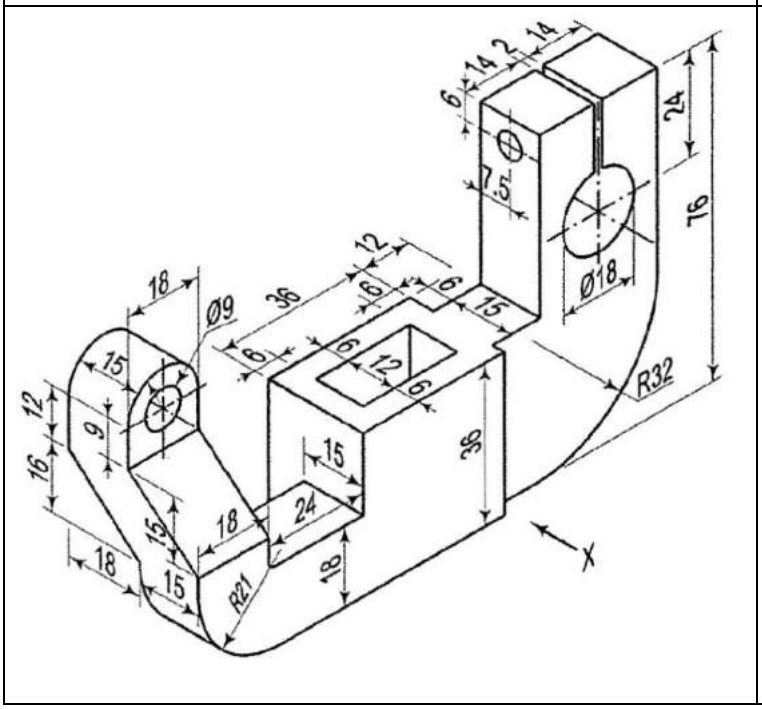



50
16
10
44
16
10
Ø16
24
18
36
Ø40
20
45
36
76
50
60
90
X




50
Ø46
10
20
Ø38
24
10
Ø10
R12
30
93
X




14
2
14
6
24
7.5
79
12
18
Ø9
36
6
6
6
15
Ø18
6
6
12
6
R32
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15
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16
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18
24
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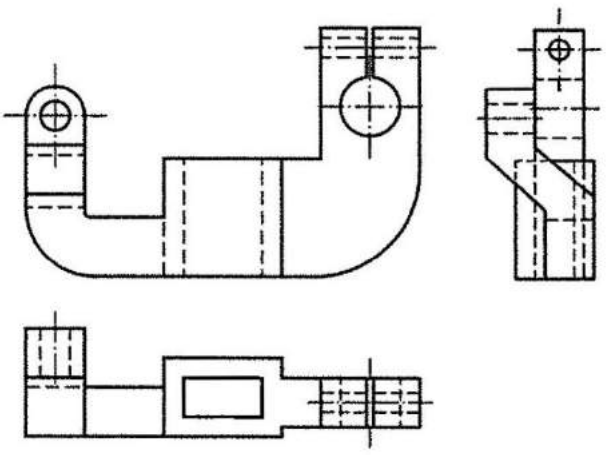
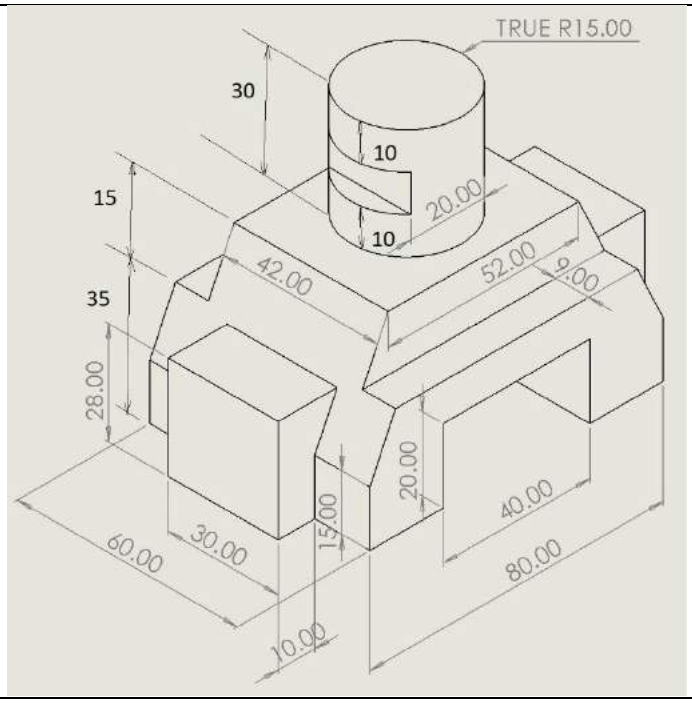
18
15
R21
18
X




TRUE R15.00
30
10
15
20.00
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42.00
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9.00
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40.00
60.00
30.00
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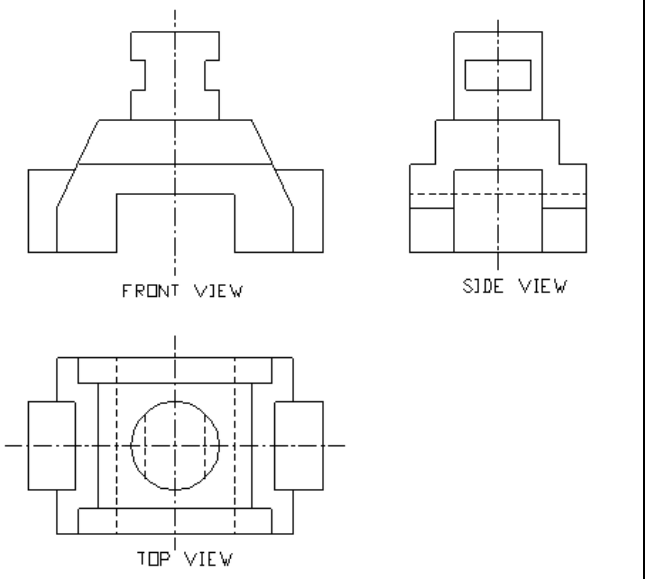
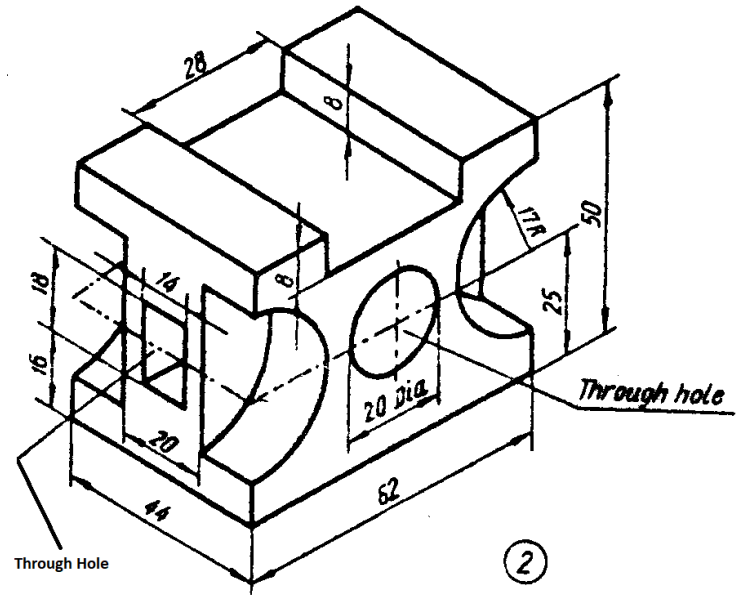
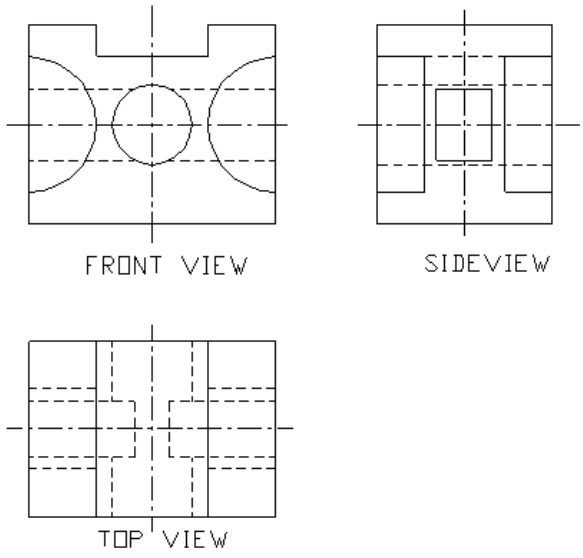
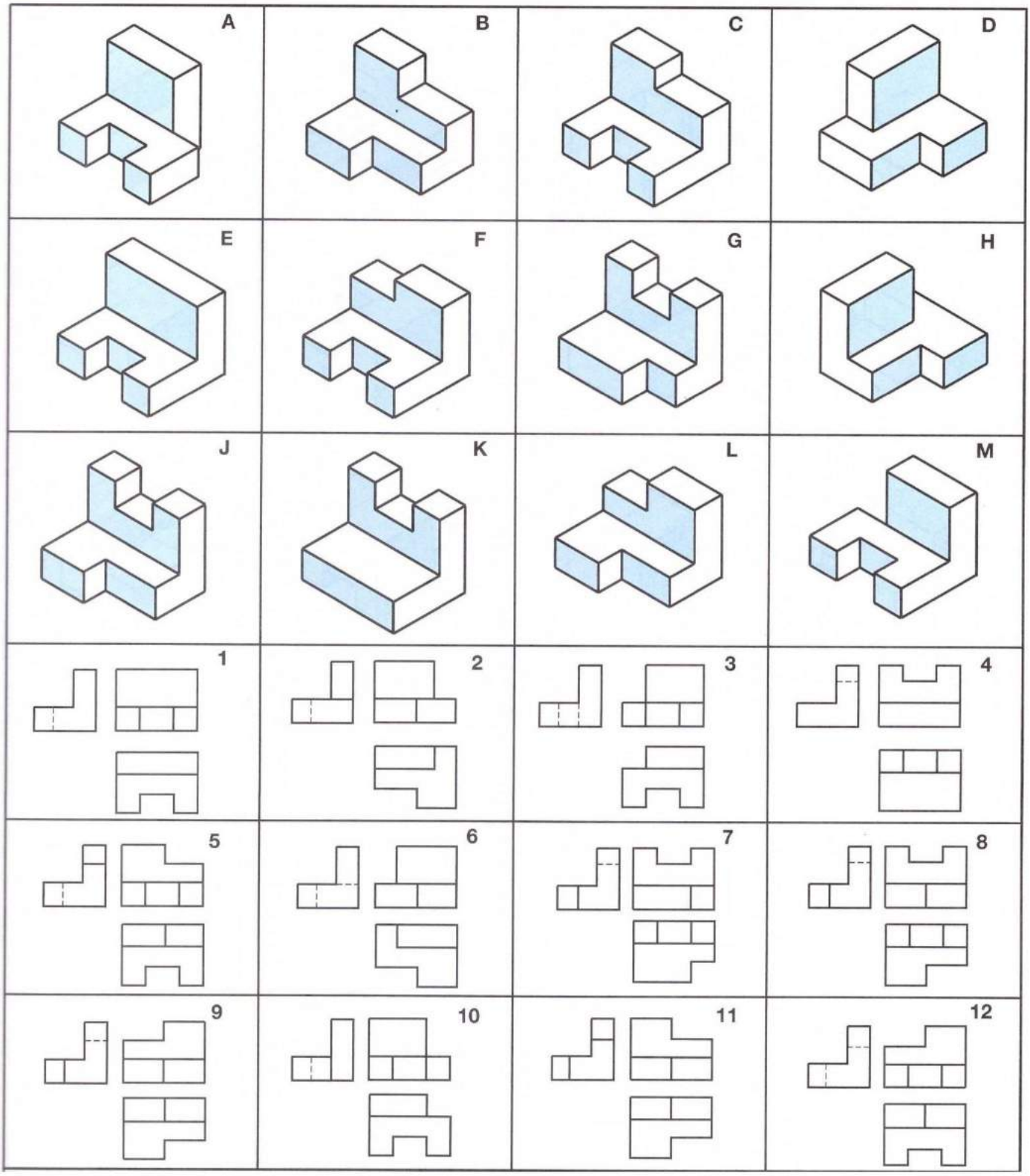
80.00
10.00

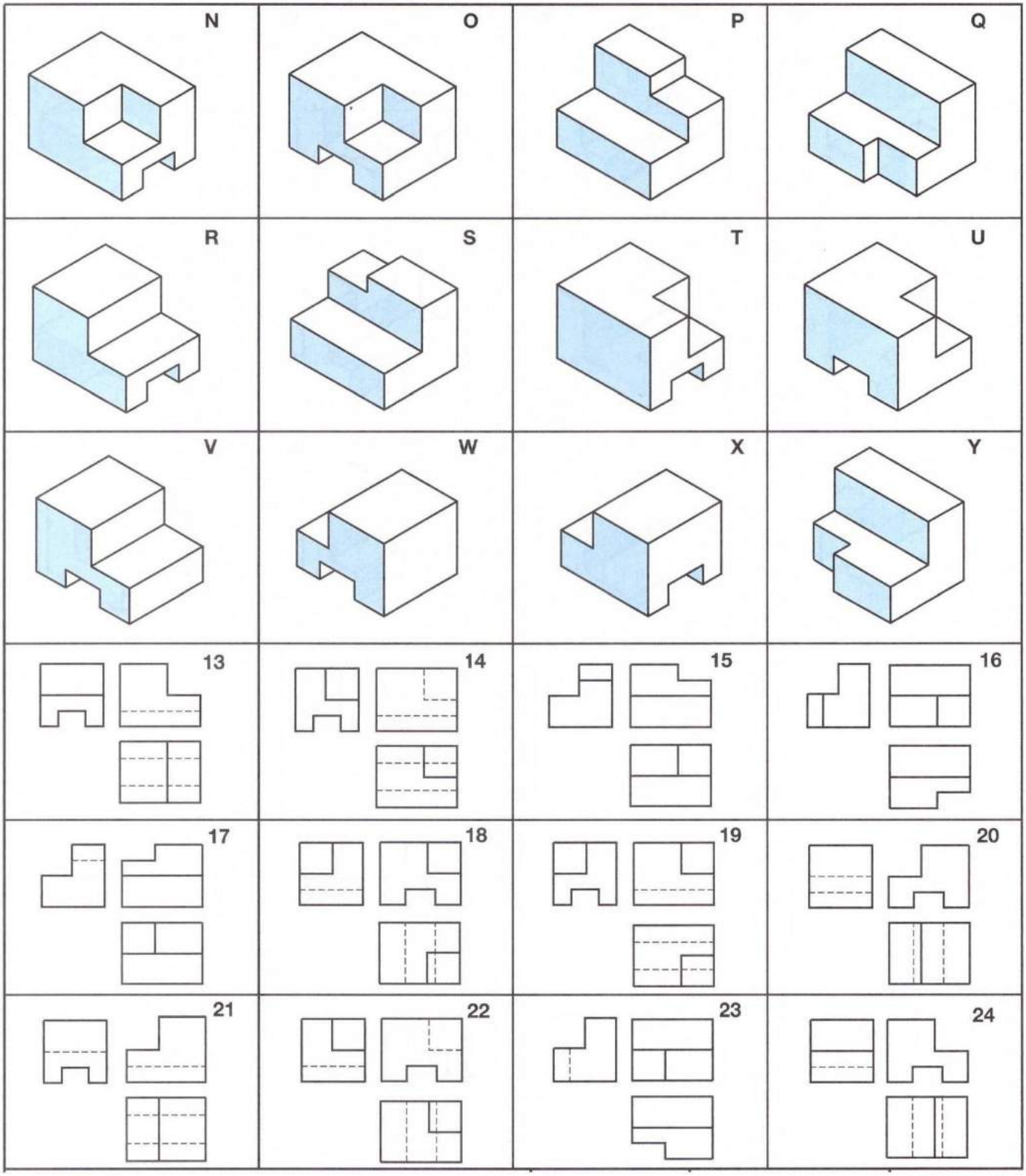

FRONT VIEW
SIDE VIEW
TOP VIEW


28
8
17R
50
18
14
8
25
16
20 Dia
Through hole
20
44
62
Through Hole
2

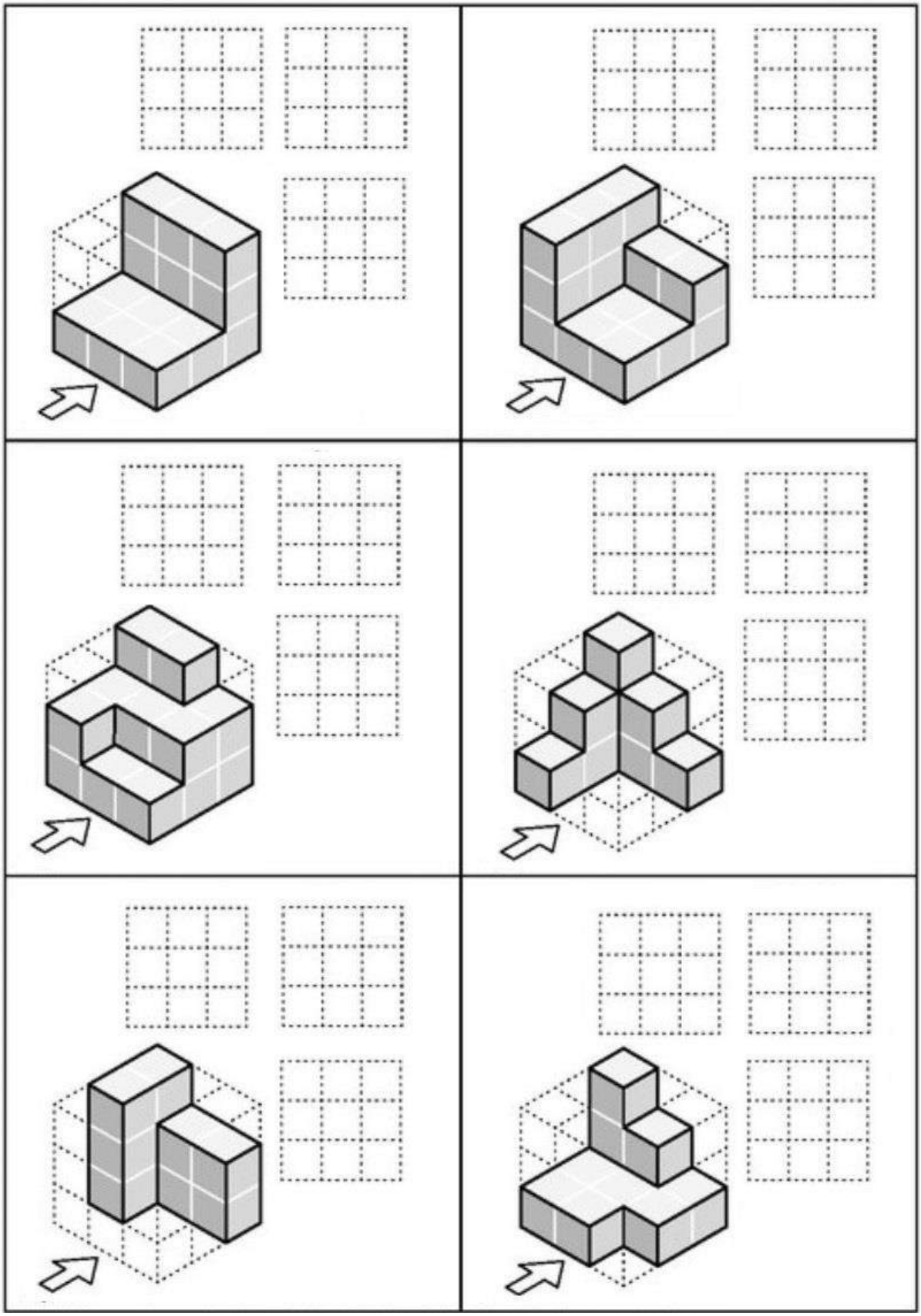

FRONT VIEW
SIDEVIEW
TOP VIEW

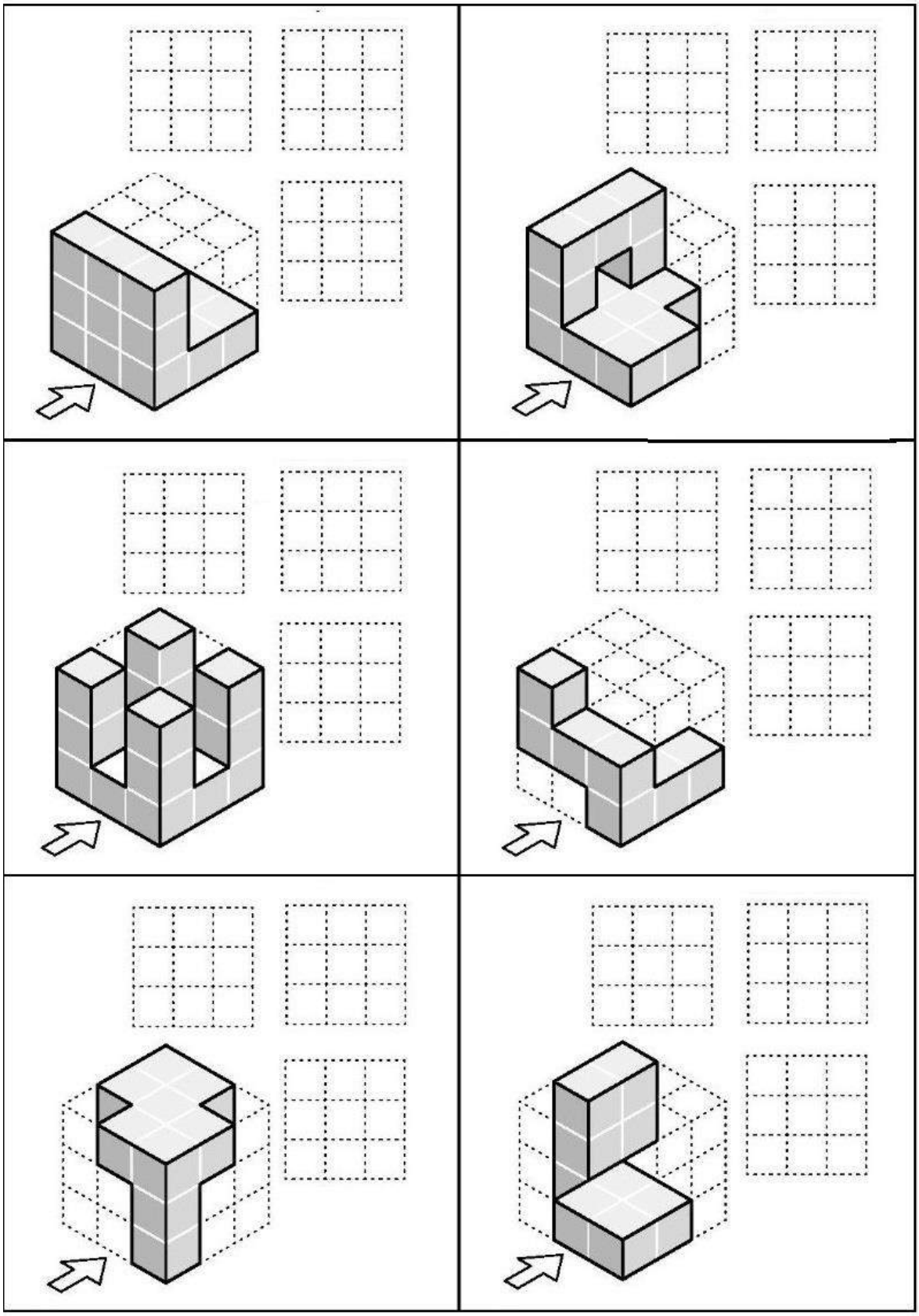


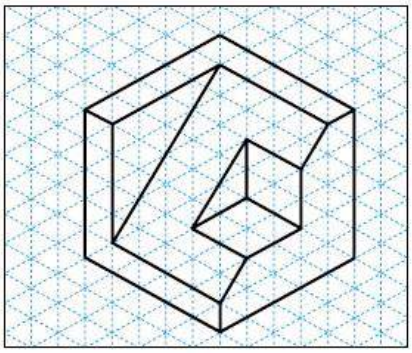
Find the correct match between the letters and numbers



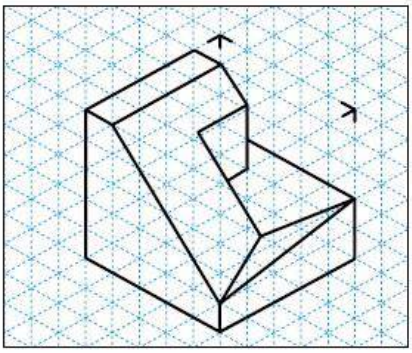
Find the correct match between the letters and numbers



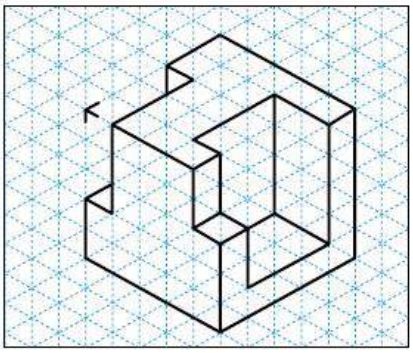




(1)



(2)



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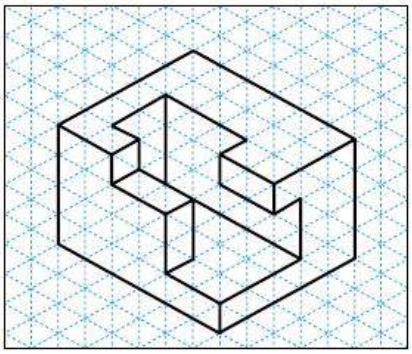

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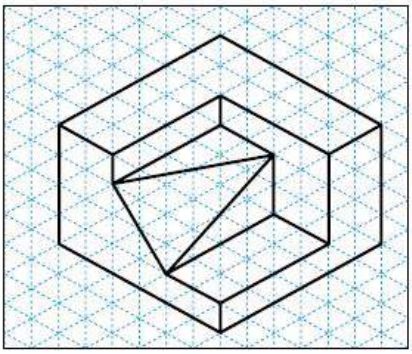

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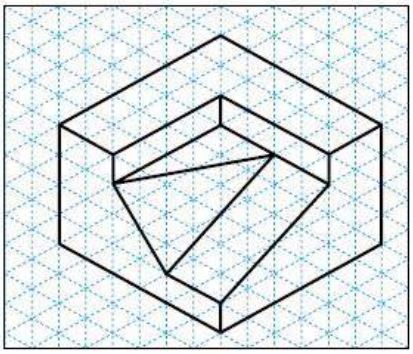

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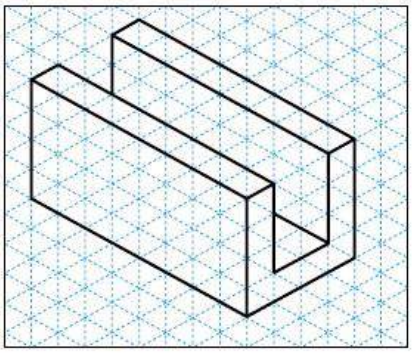

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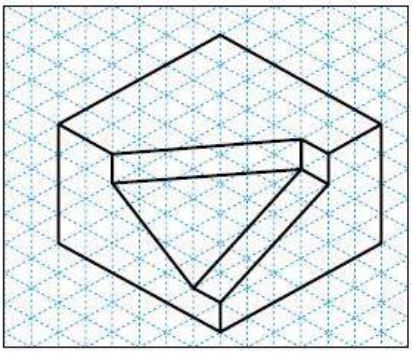

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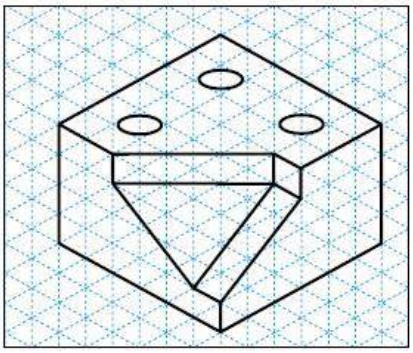

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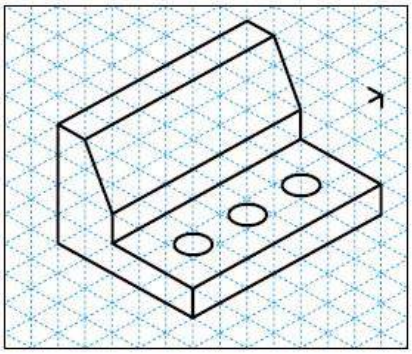

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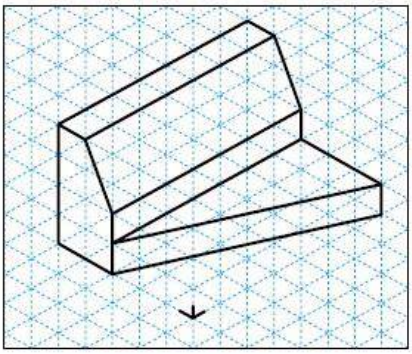

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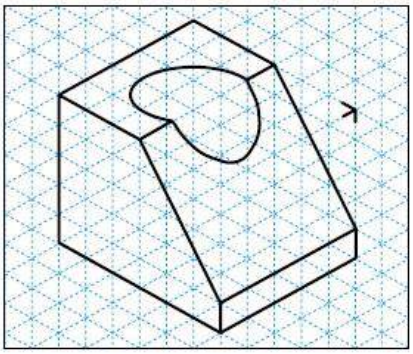

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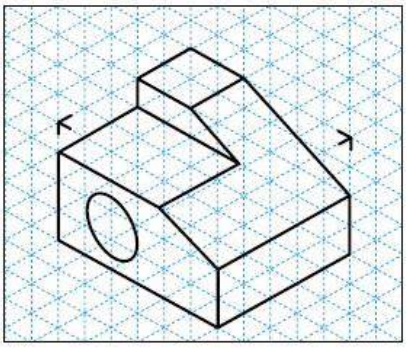

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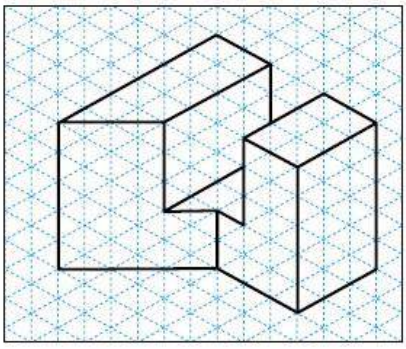

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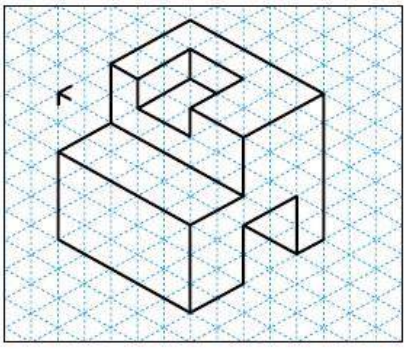

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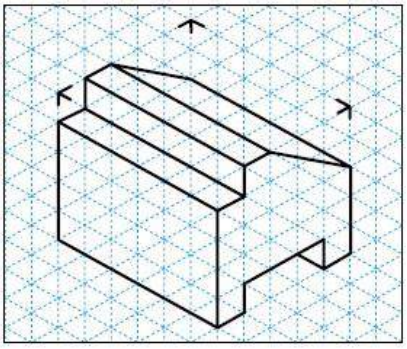

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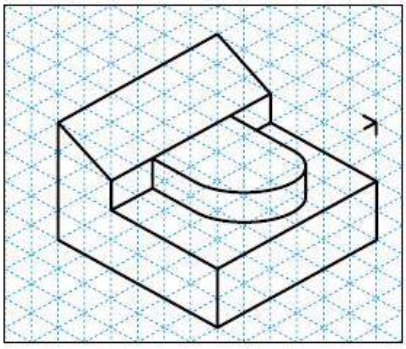

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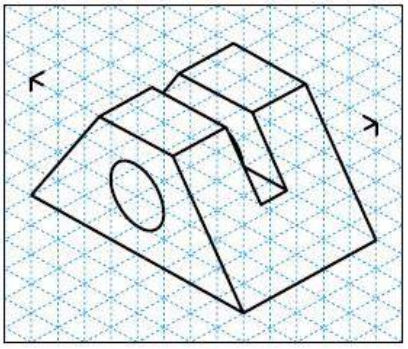

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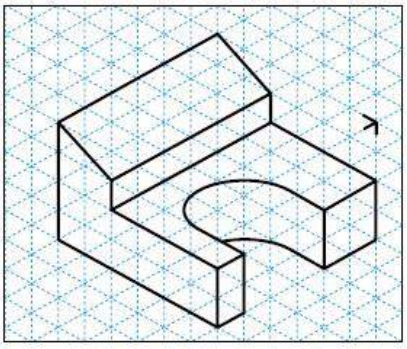

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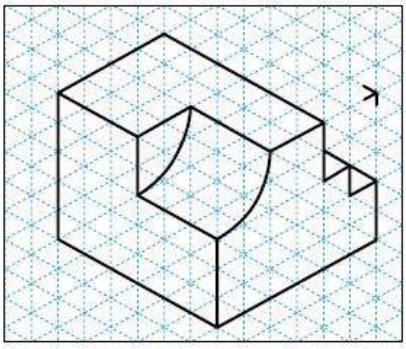

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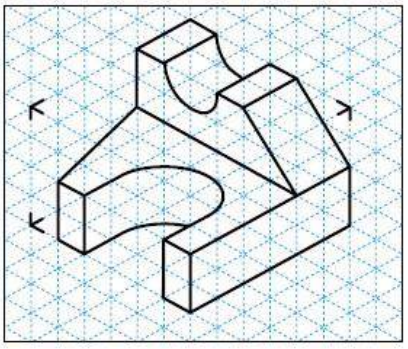

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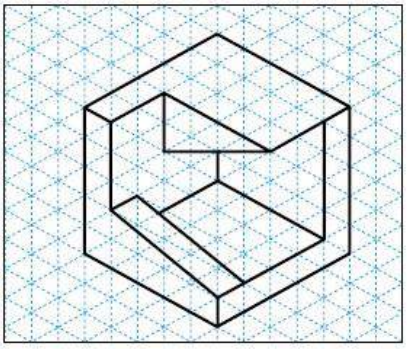

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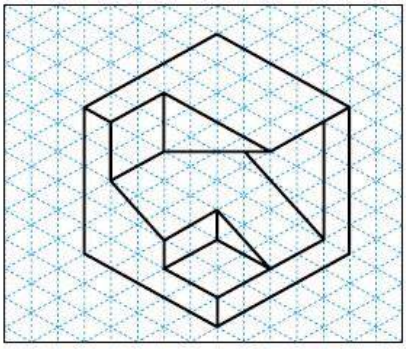

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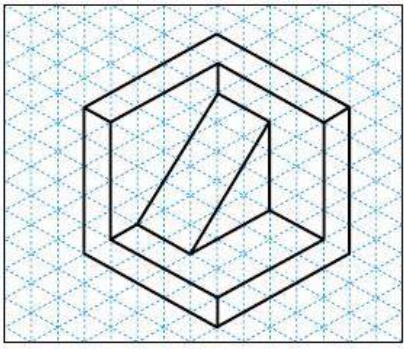

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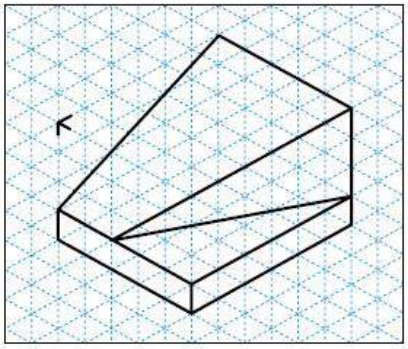

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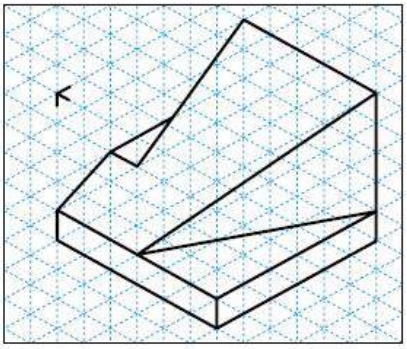

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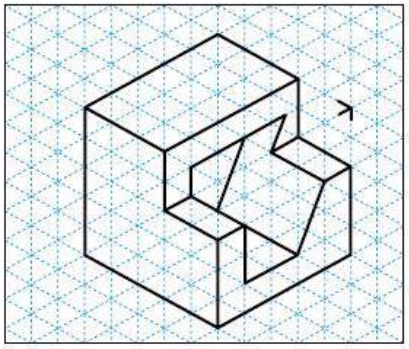

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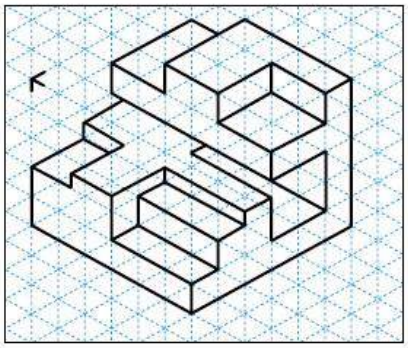

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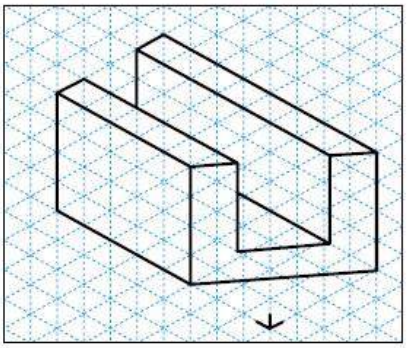

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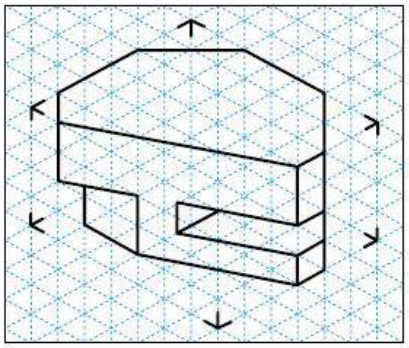

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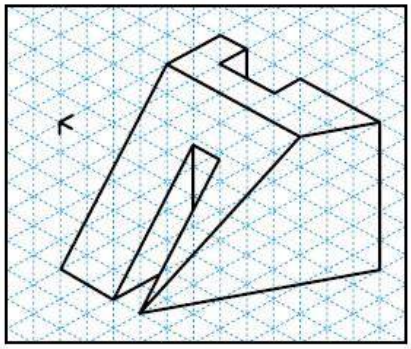

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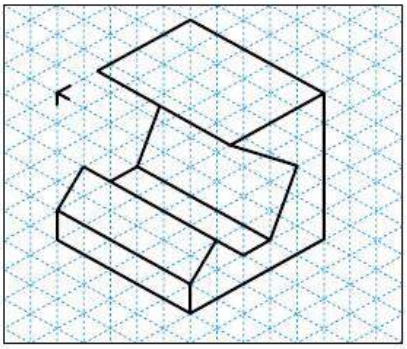

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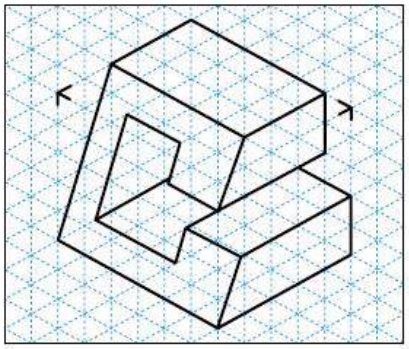

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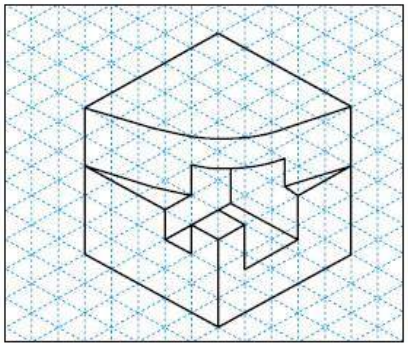

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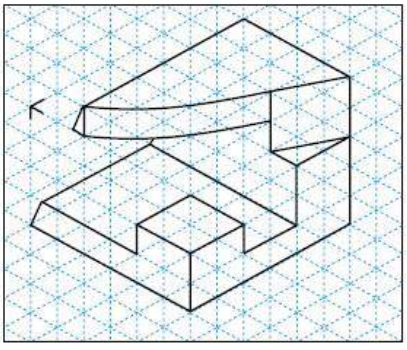

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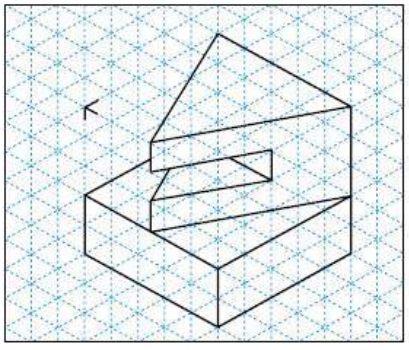

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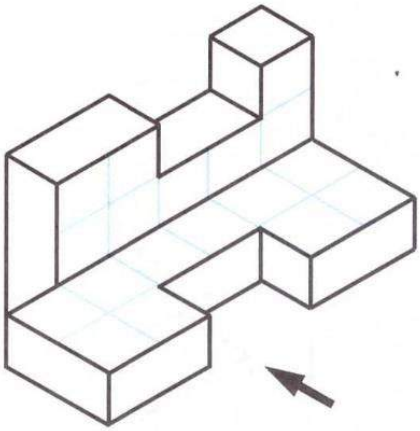

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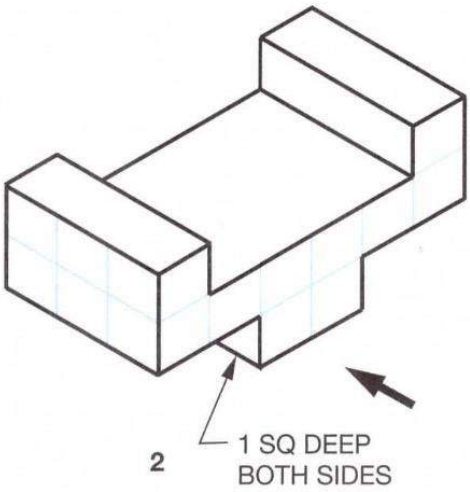

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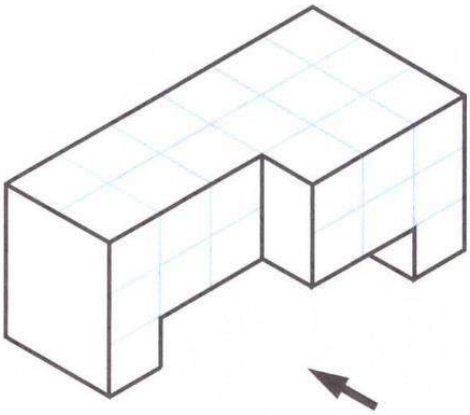

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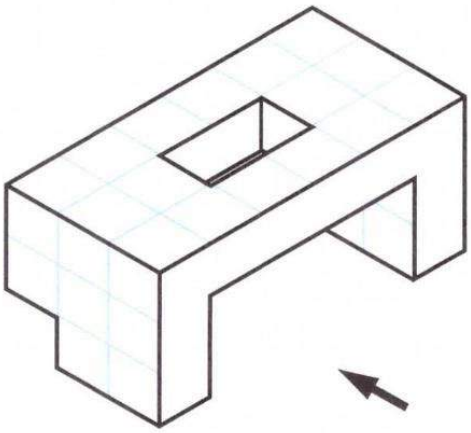

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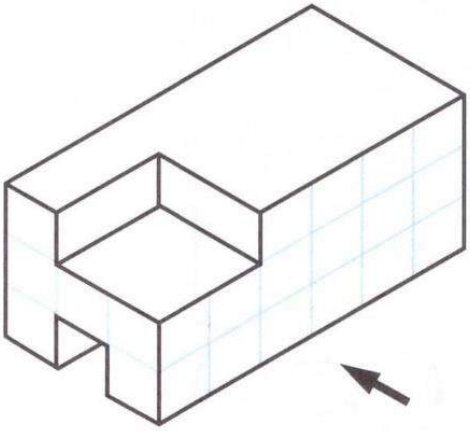

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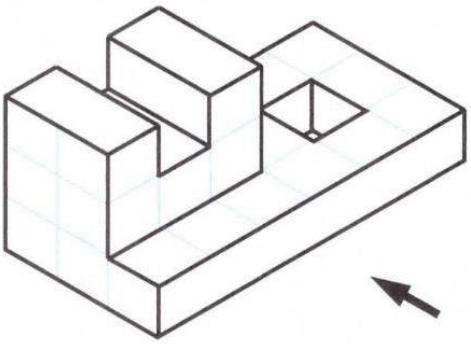
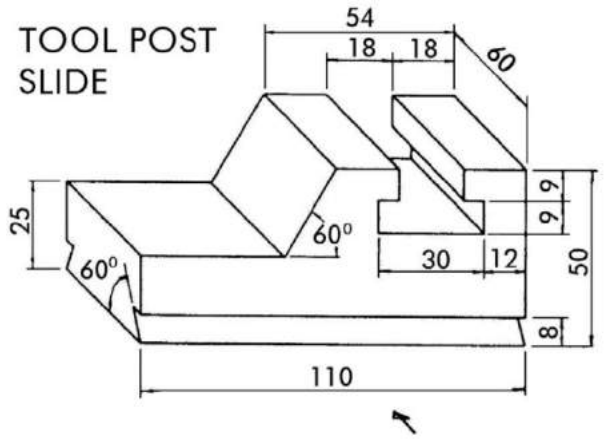
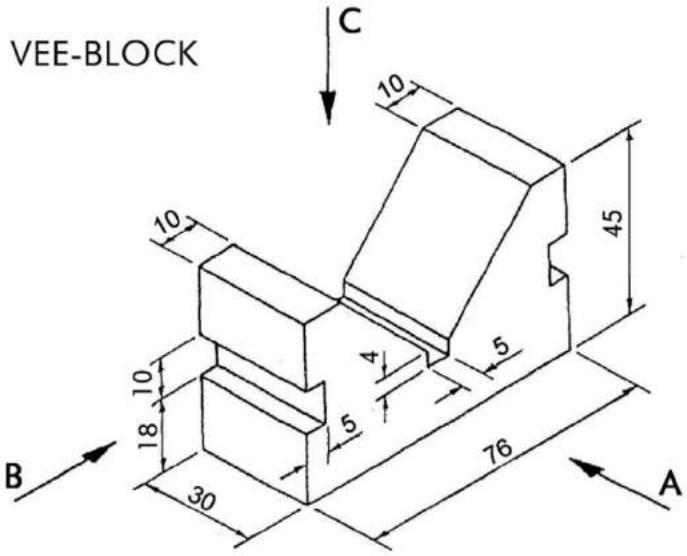


6

TOOL POST SLIDE



VEE-BLOCK



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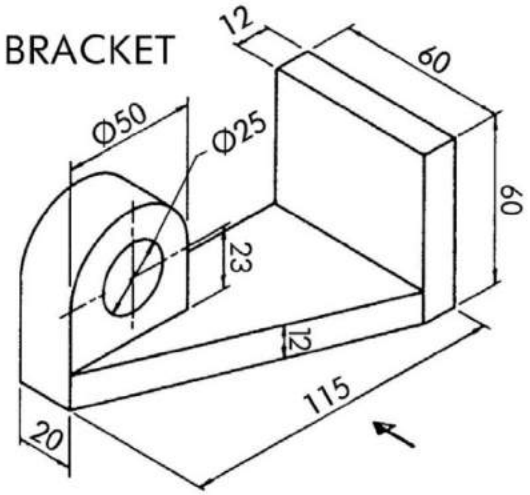
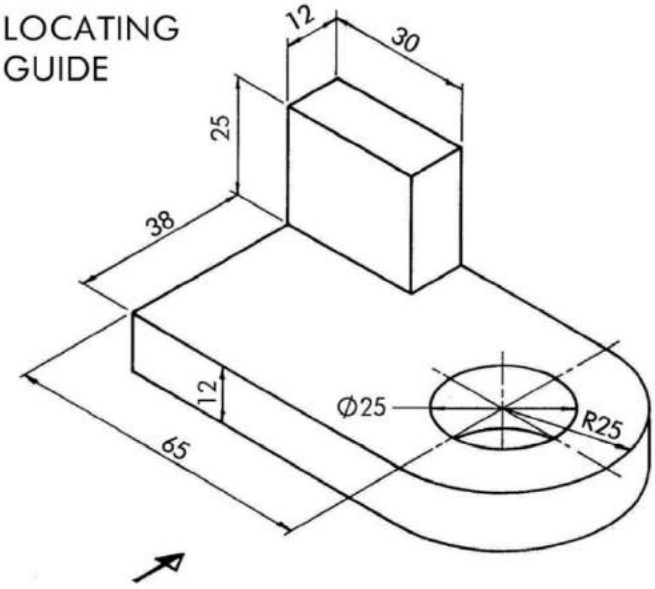
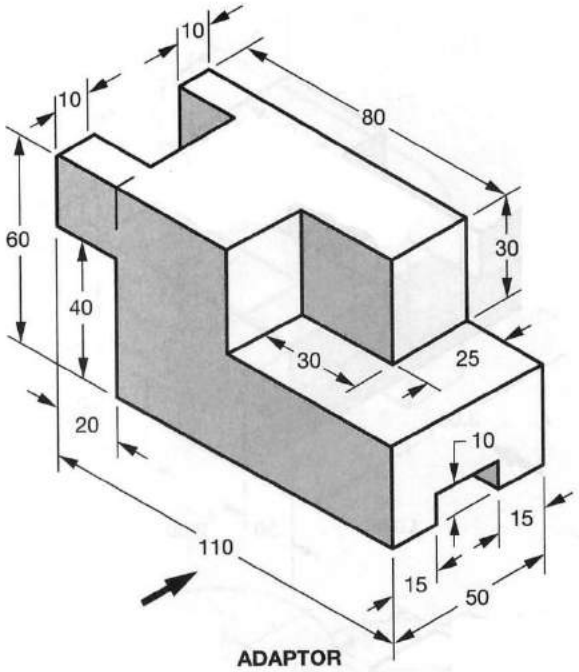
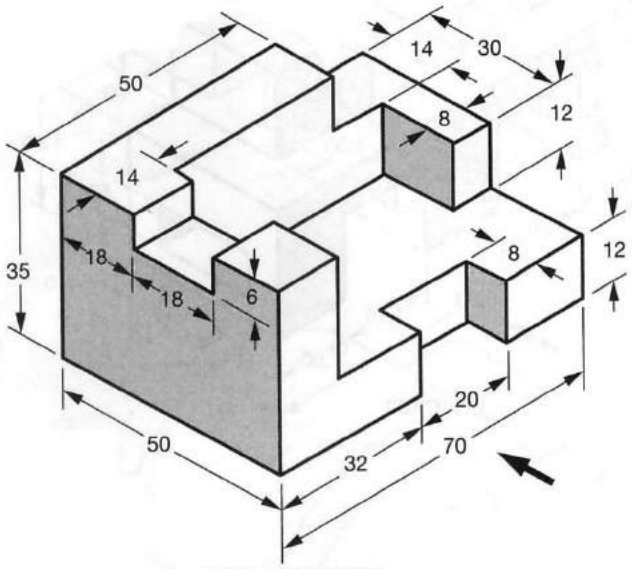


LOCATING GUIDE

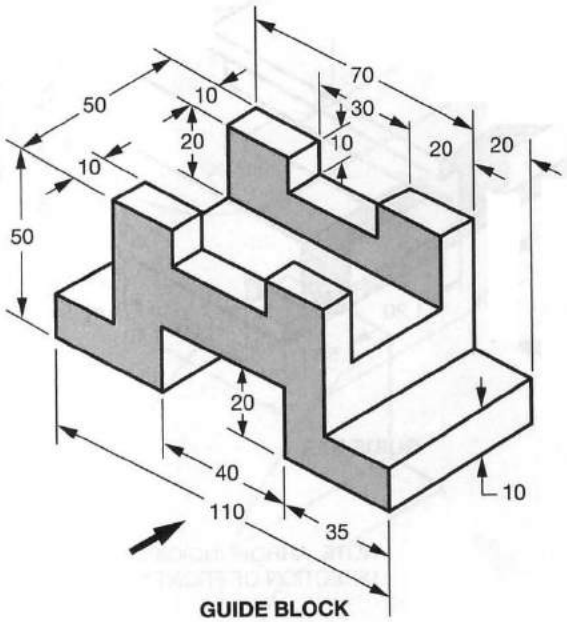




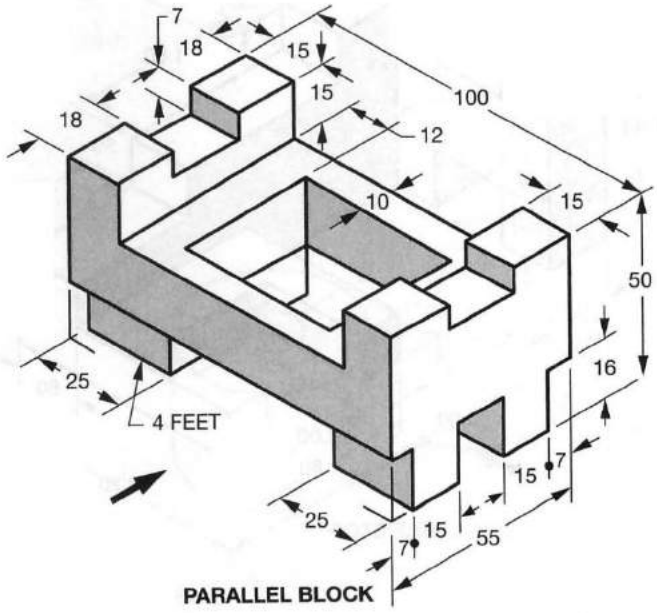
ADAPTOR



ADJUSTING GUIDE

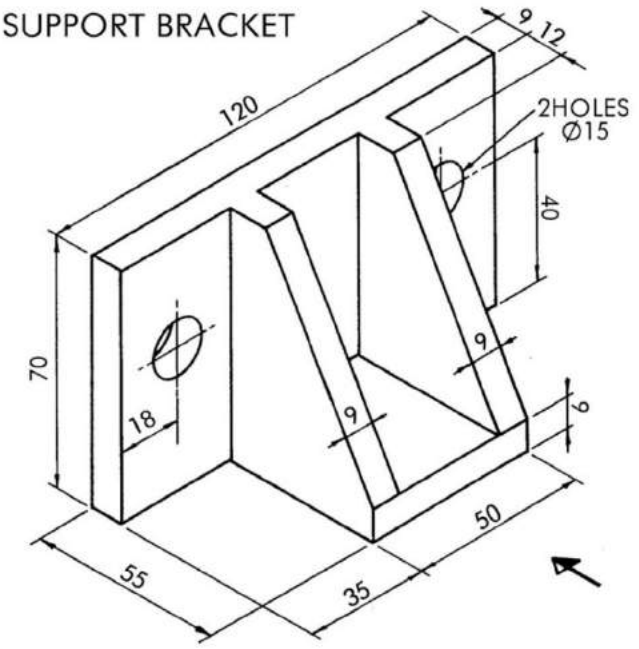


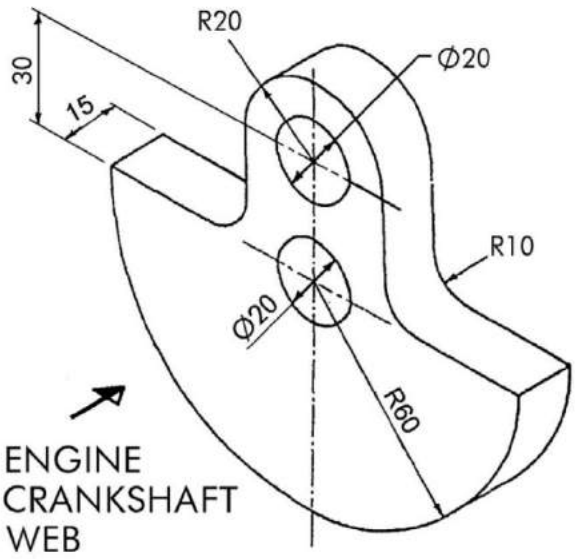
GUIDE BLOCK



PARALLEL BLOCK

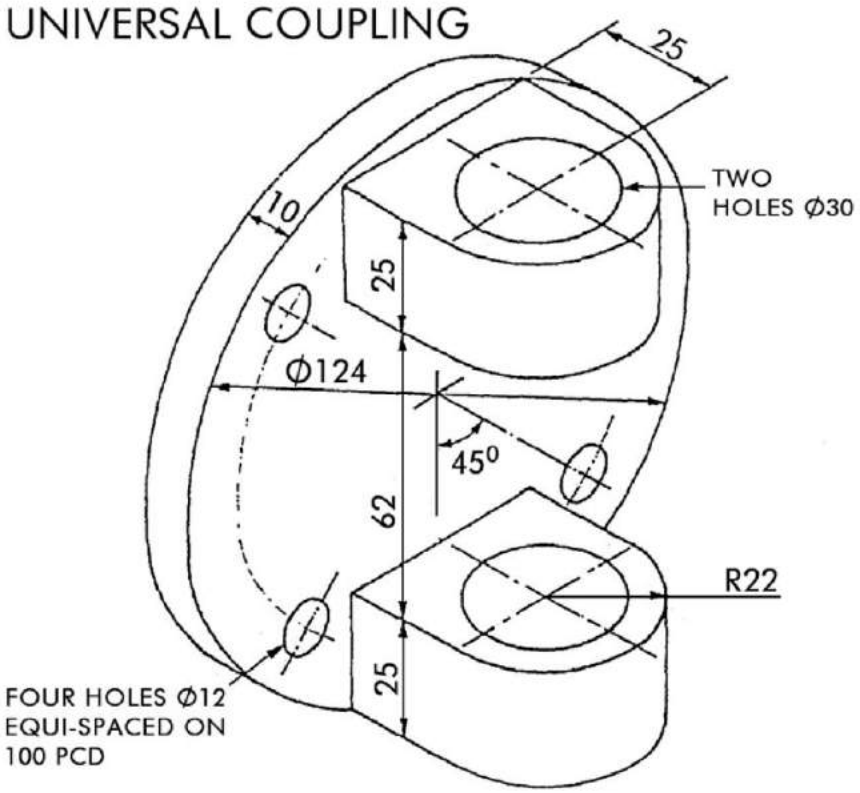
SUPPORT BRACKET

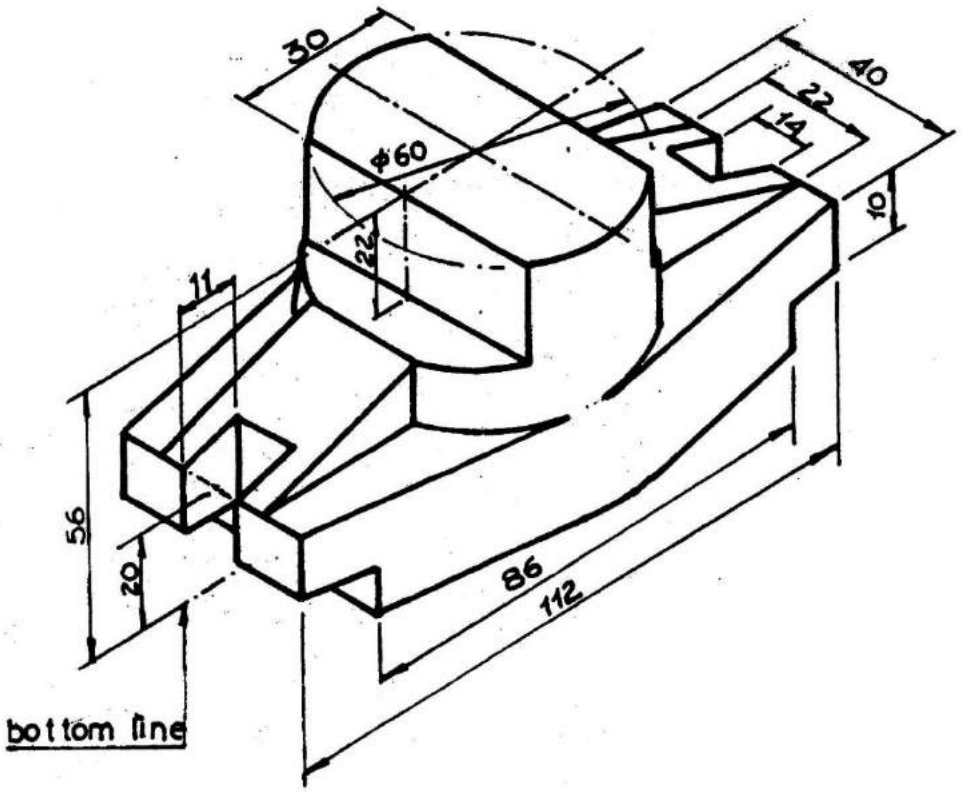


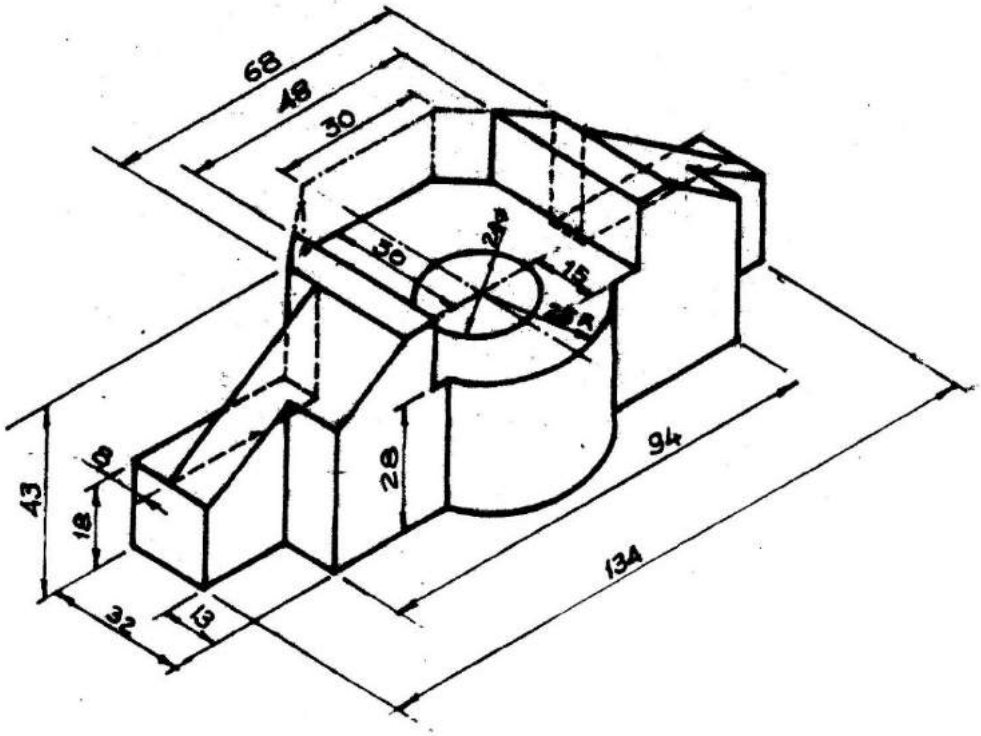


ENGINE
CRANKSHAFT
WEB

UNIVERSAL COUPLING




30
40
22
14
ϕ60
10
22
11
56
20
86
112
bottom line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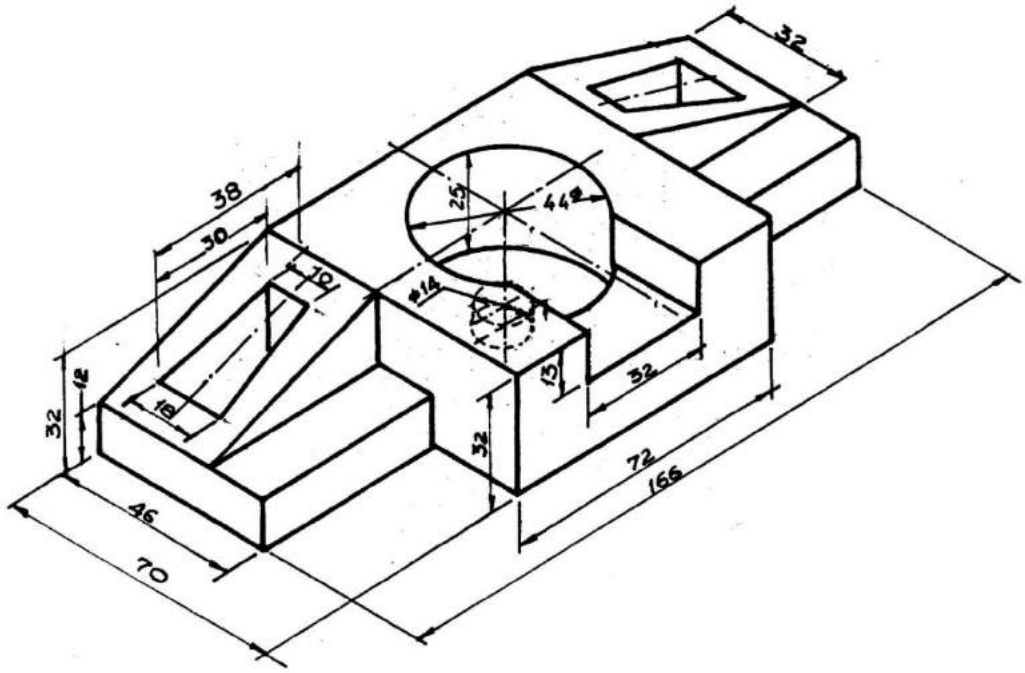
13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16
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72
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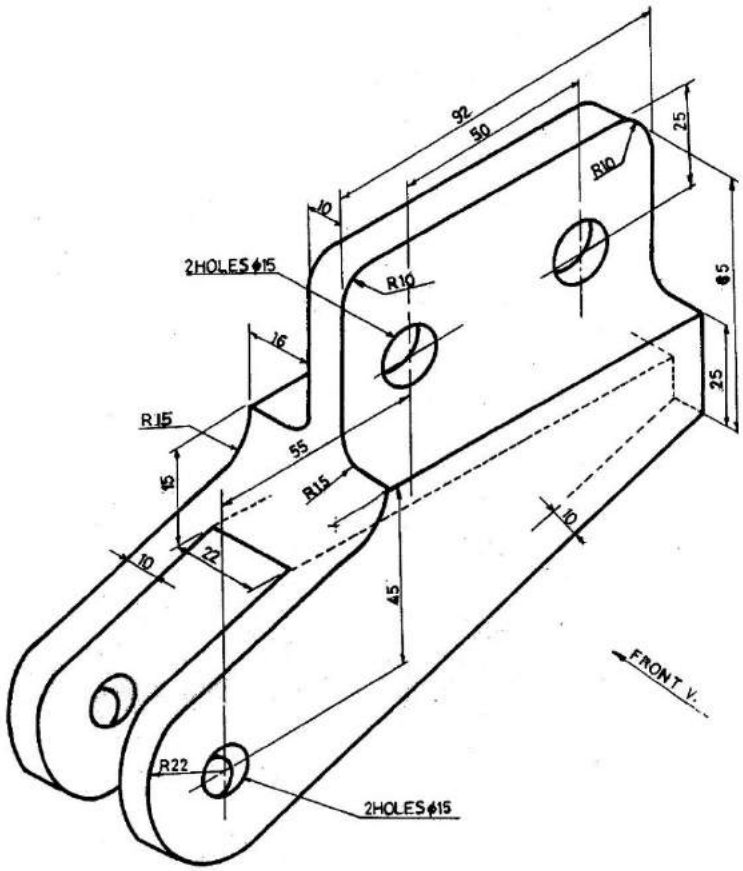
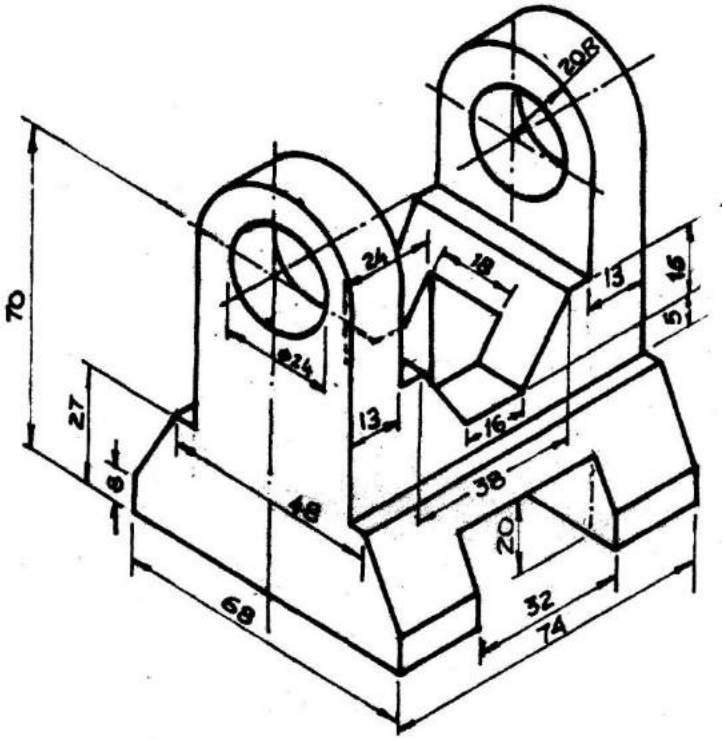
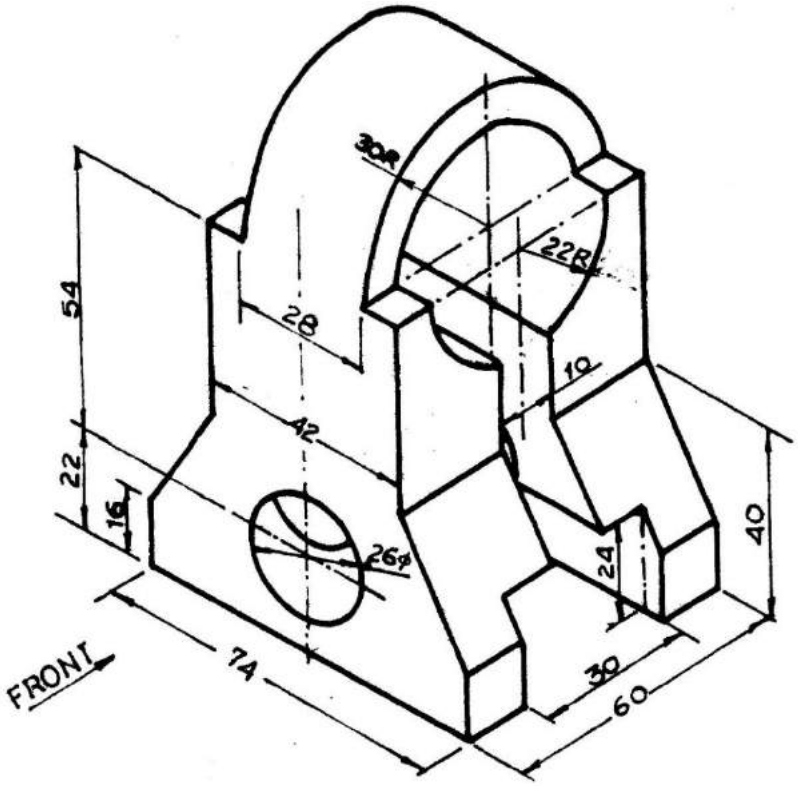
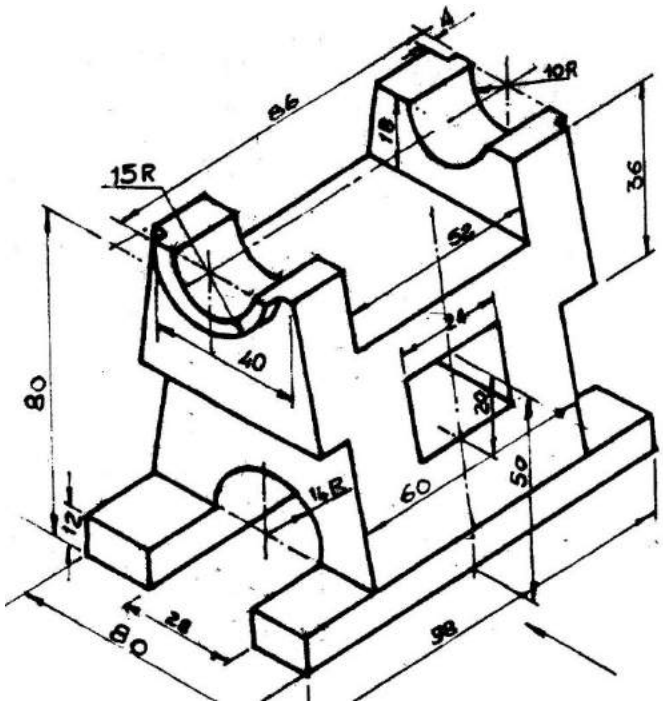
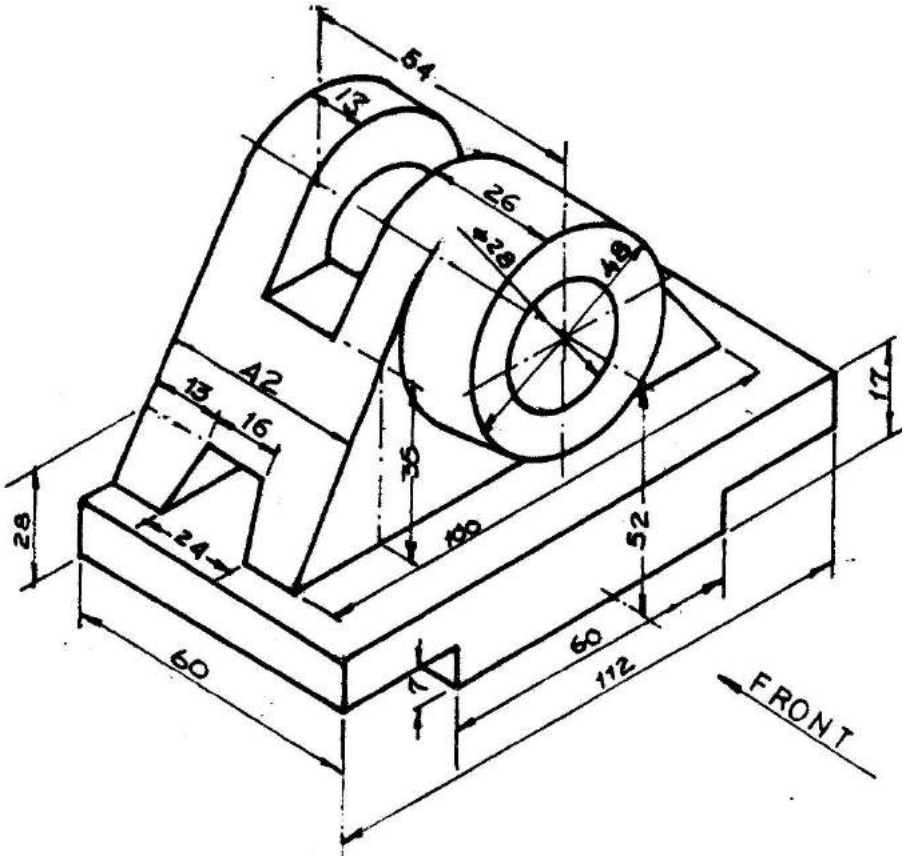
46
70

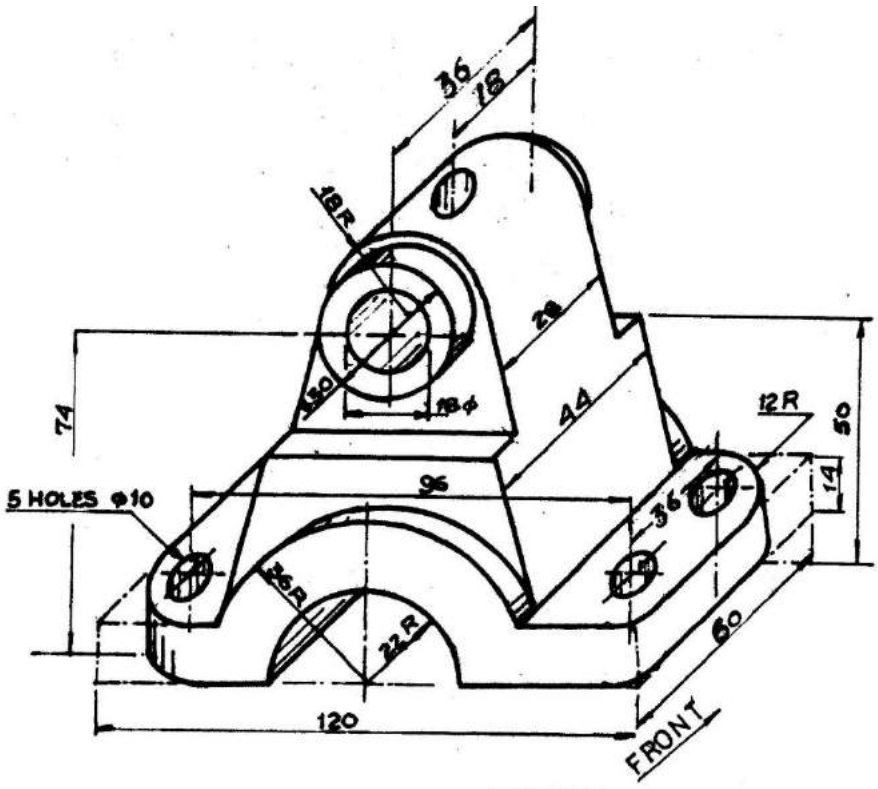

92
50
10
25
R10
2HOLES ϕ15
R10
ϕ5
16
25
R15
55
15
R15
10
22
10
45
FRONT V.
R22
2HOLES ϕ15

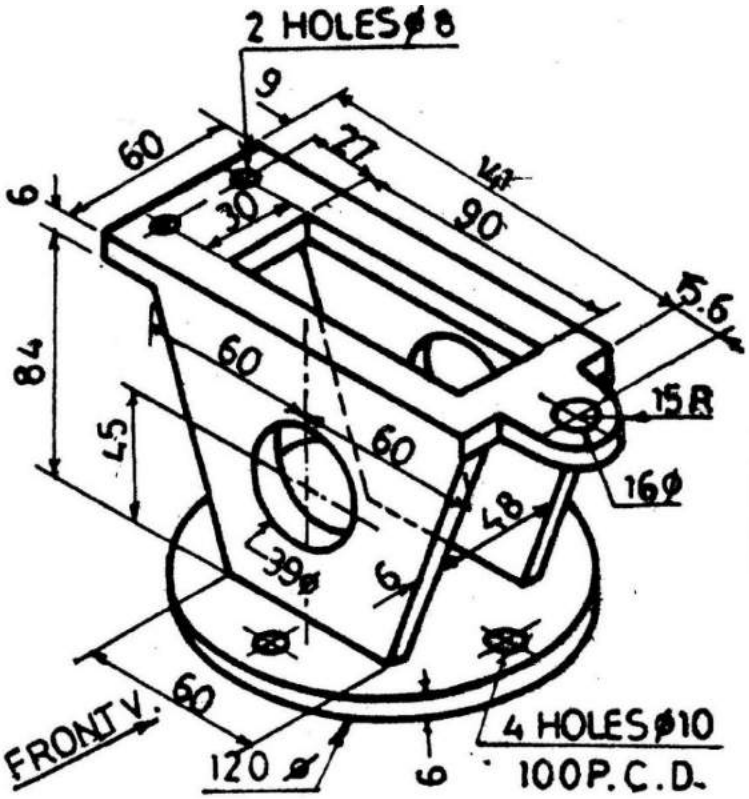

20R
70
24
18
13
15
5
Ø14
27
13
16
8
48
38
20
68
32
74

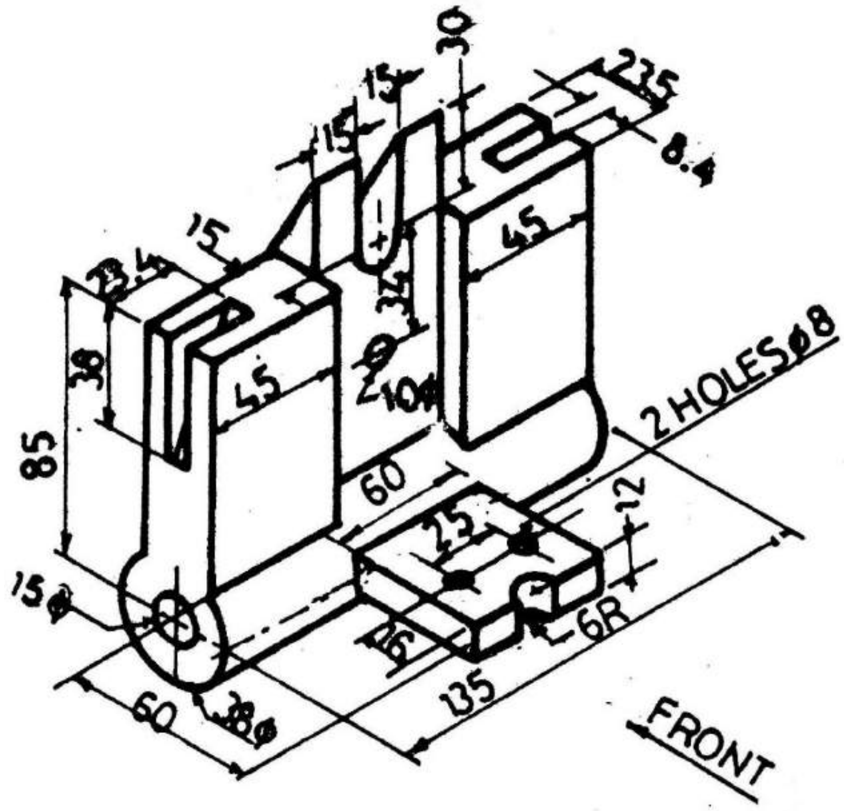

30R
22R
54
28
10
42
22
16
Ø9
24
40
FRONT
74
30
60

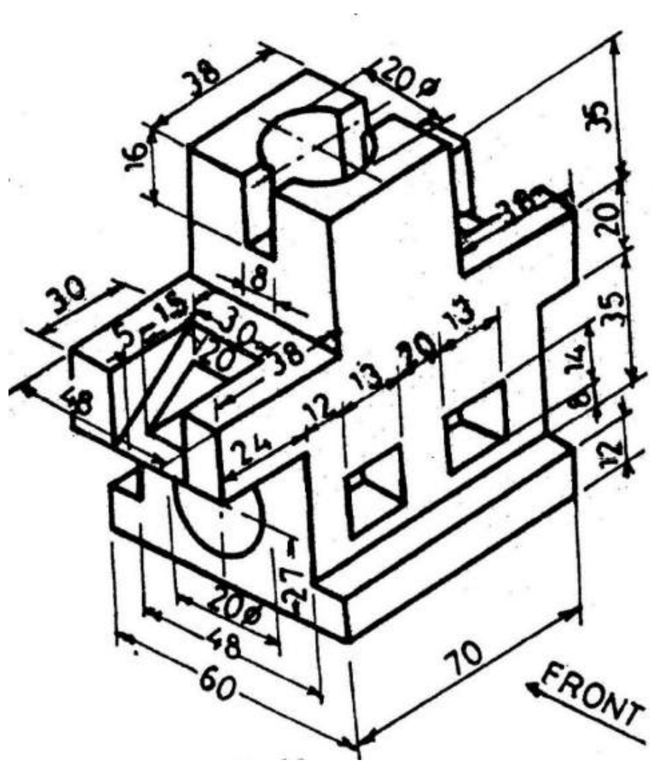

4
10R
ø6
15R
18
36
52
24
40
80
28
12
14R
60
50
80
28
98

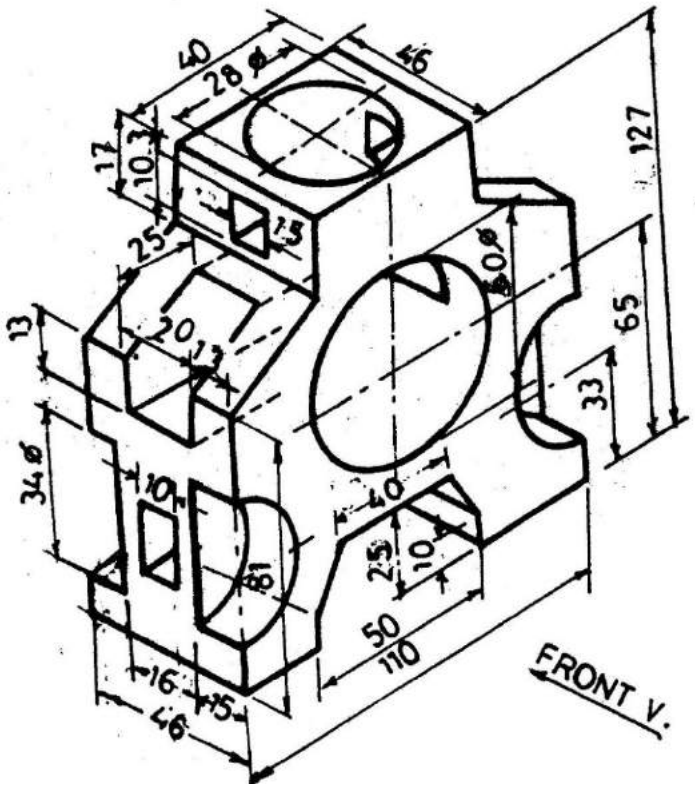

54
13
26
28
48
A2
13
16
17
35
28
24
100
52
60
60
112
FRONT

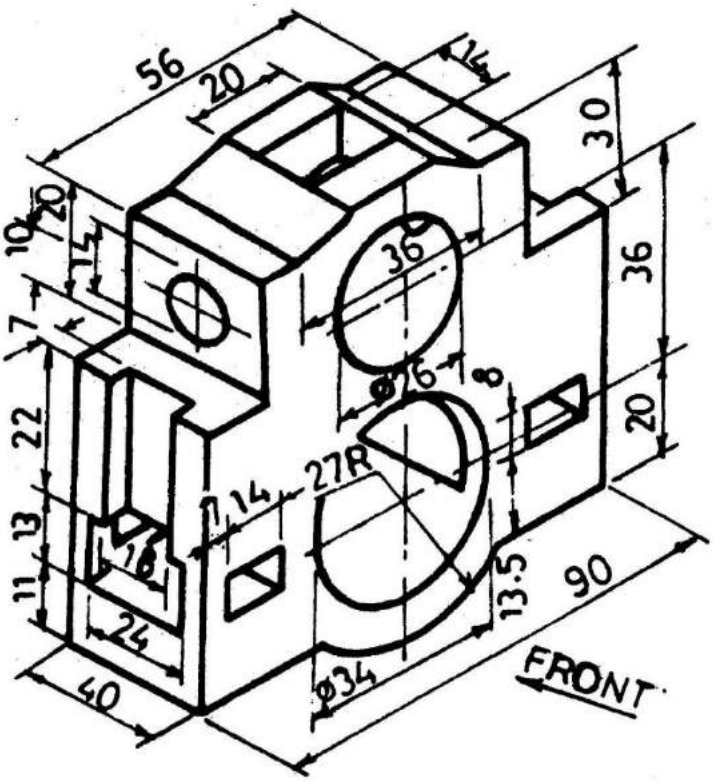

36
18
18R
28
44
74
30
18φ
12R
50
14
5 HOLES φ10
96
36
36R
22R
60
120
FRONT


2 HOLES φ8
9
60
6
27
143
30
90
5.6
84
60
15R
45
60
16φ
48
39φ
6
FRONT V.
60
120
6
4 HOLES φ10
100 P.C.D.


30
23.5
8.4
15
15
15
23.4
38
45
45
35
10ø
85
60
25
12
2 HOLES ø8
15ø
6R
16
135
60
38ø
FRONT


38
20ø
35
16
38ø
20
35
8
30
15
5
30
20
38
13
20
13
14
8
12
48
24
12
20ø
27
48
60
70
FRONT


40
28 ϕ
46
17
10
3
25
13
20
13
13
34 ϕ
10
81
40
25
10
50
110
16
15
46
127
ϕ10
65
33
FRONT V.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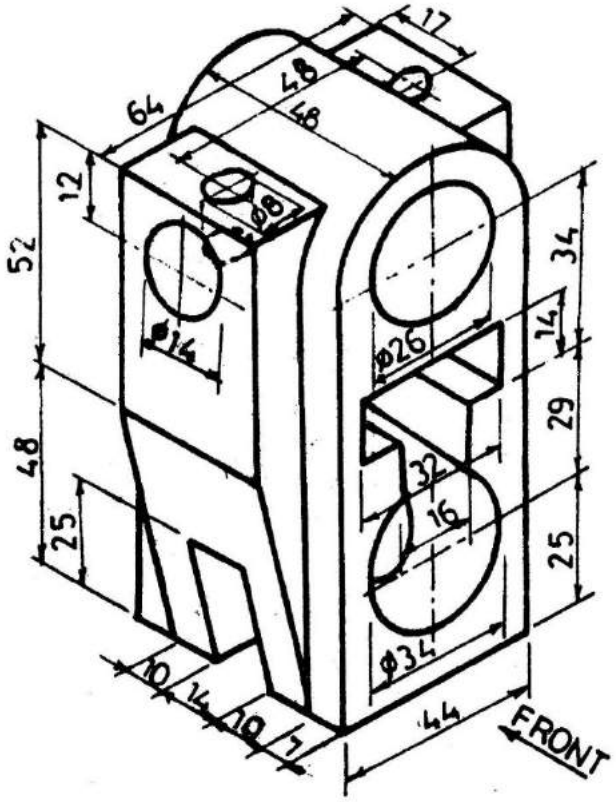
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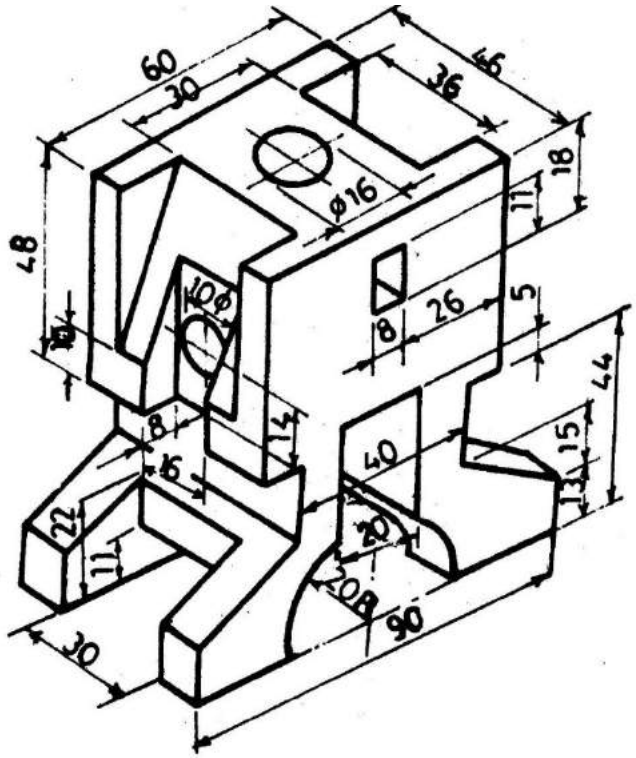
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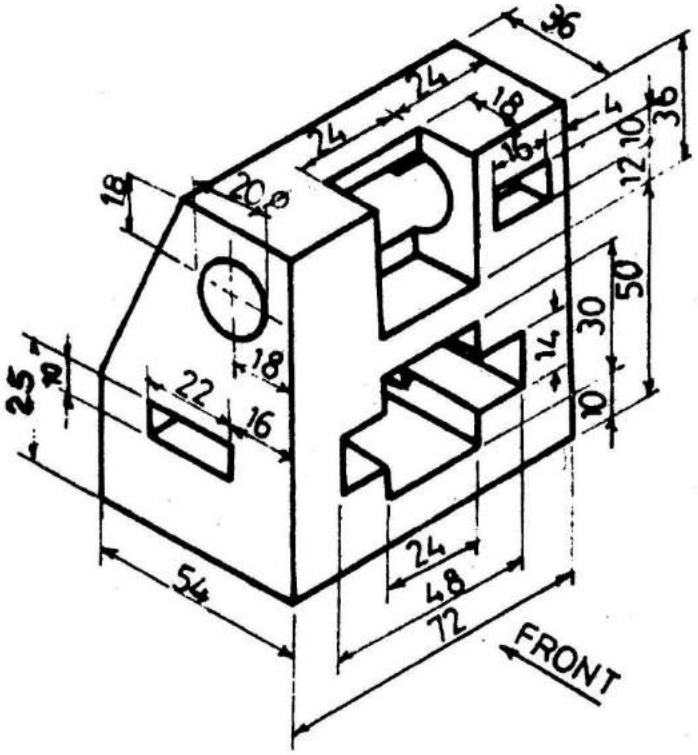
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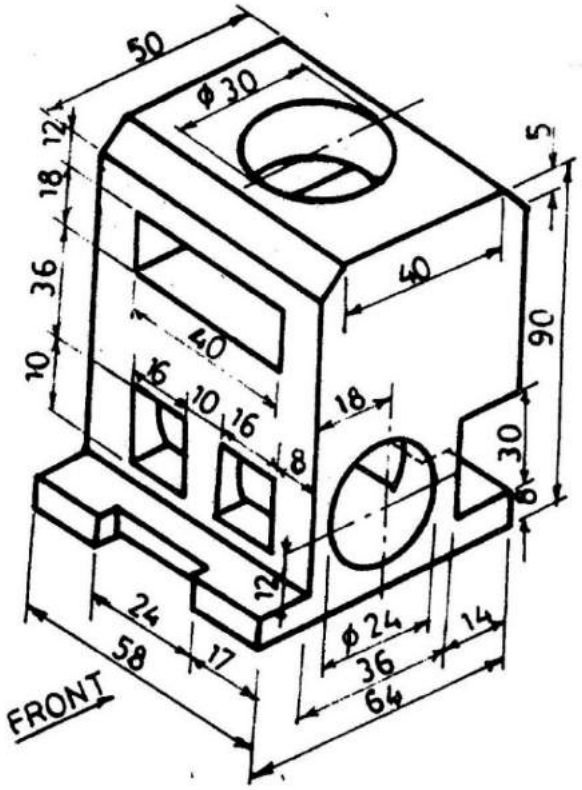
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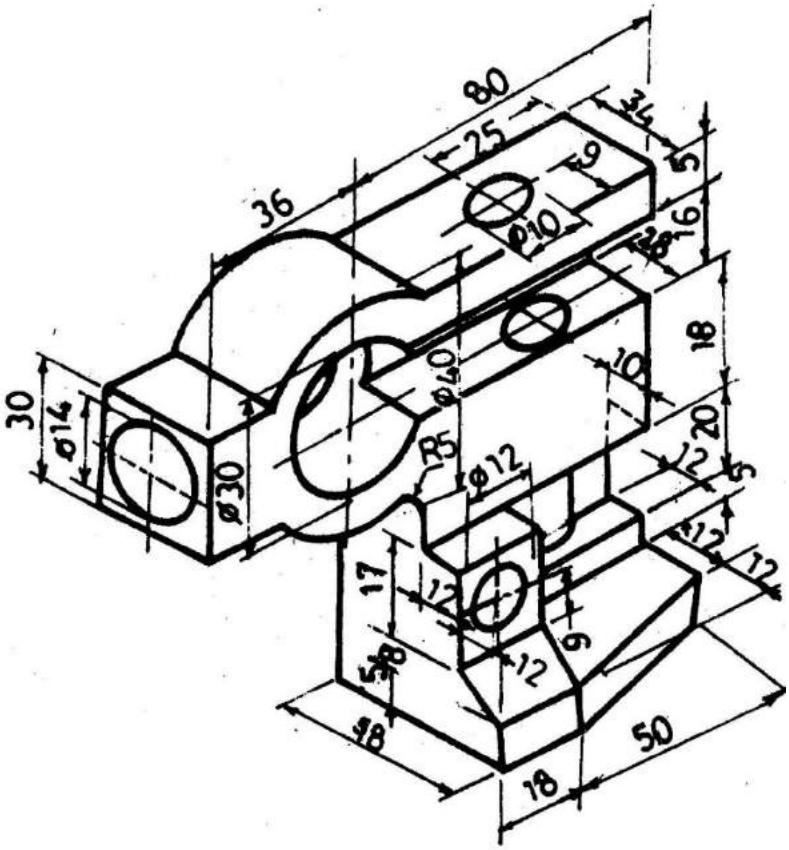
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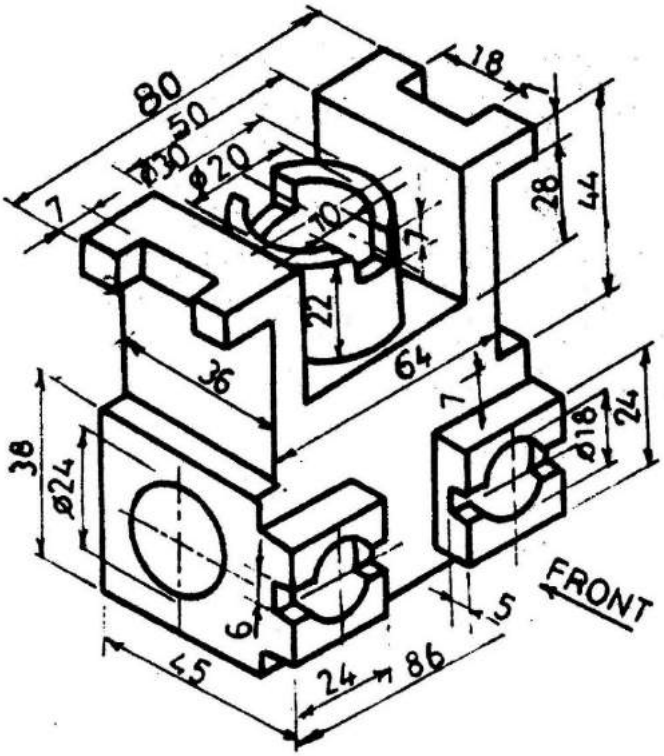
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86

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

1. Isometric projection
2. Oblique projection
3. Prespective projection.

In this chapter you will learn about the pictorial drawings, and a focus on Isometric and Oblique drawings will be illustrated.

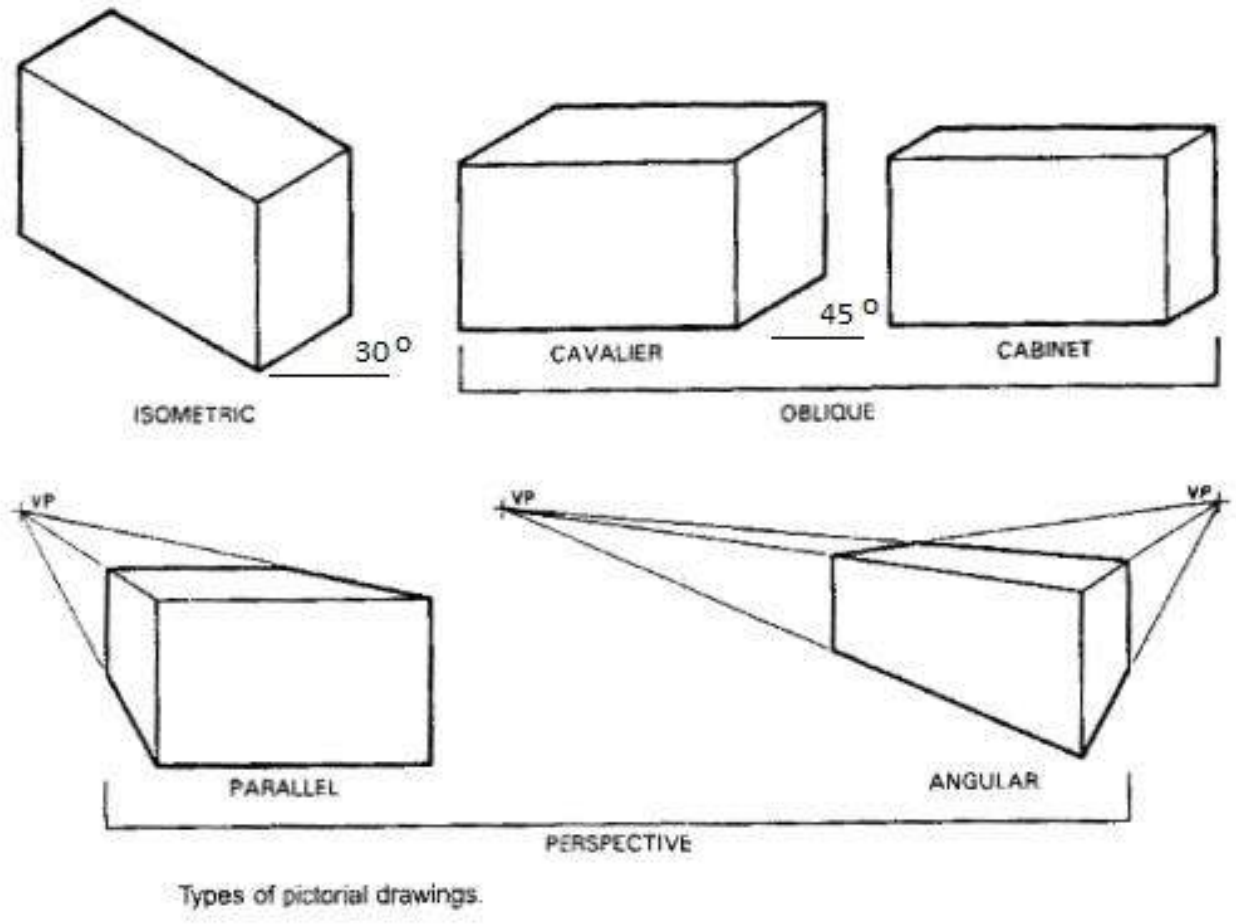


Figure 1 Classifications of Pictorial Drawings

1- ISOMETRIC DRAWING

Isometric projection is a type of pictorial projection in which the three dimensions of a solid are not only shown in one view, but their actual sizes can be measured directly from it. In an isometric view, lines for the hidden edges are generally not shown. A special grid is required for Isometric drawing. In this grid, both ground lines in the Front and Side will be tilted 30^0 about the horizontal axis. Horizontal lines in the Front, Side or Top will be tilted by 30^0 in Isometric, depending on the direction. Vertical lines will stay vertical, it is important to say that true vertical lines, not the projection of inclined lines. The principle of Isometric is illustrated in Figure 2.

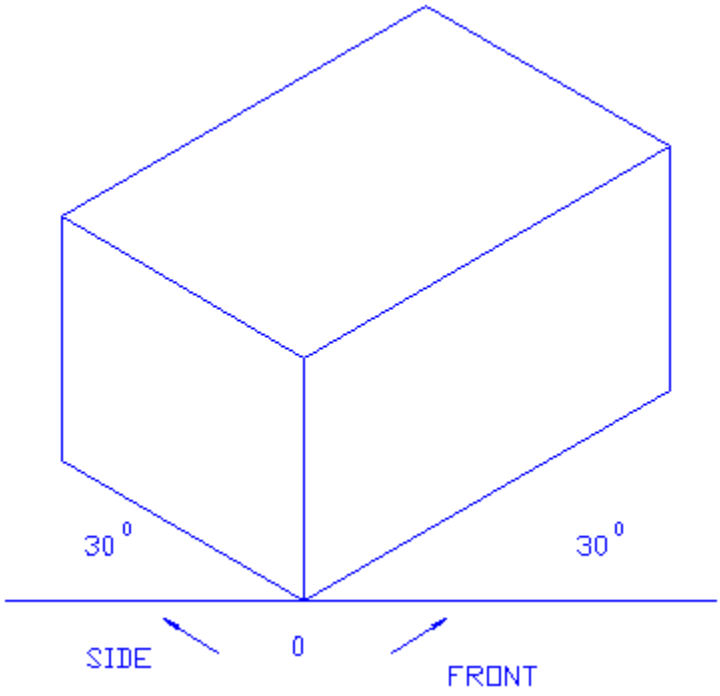


Figure 2 Principles of Isometric

Figure 3, shows the three views of a box, each corner has a number starting from (0) to (7). The isometric view of this model is shown, while the location of each corner is shown in the 3D model. Corner (7) is hidden inside the model, this is why it is not shown.

In order to start the Isometric, we will start first with the Front view. The Front view (0, 1, 2, 3, 0) now will be transformed to the Isometric view. Since the Front view is a right, then in Isometric will be at right. Line (0, 1) is horizontal in the Front view, then in Isometric will be inclined by 30^0 in the direction of the Front. Line (1, 2) is vertical, then in Isometric stays vertical, and so on.

After finishing the Front in the Isometric, we will move to the Side view with the same concept, but, in the direction of the Side. In order to find the Top, only point (6) is missing. Draw a parallel line from 4 to line (3, 2), and a line from 2 parallel to (3, 4). At the intersection we will find point 6.

Point 7, the hidden point is just below point 6.

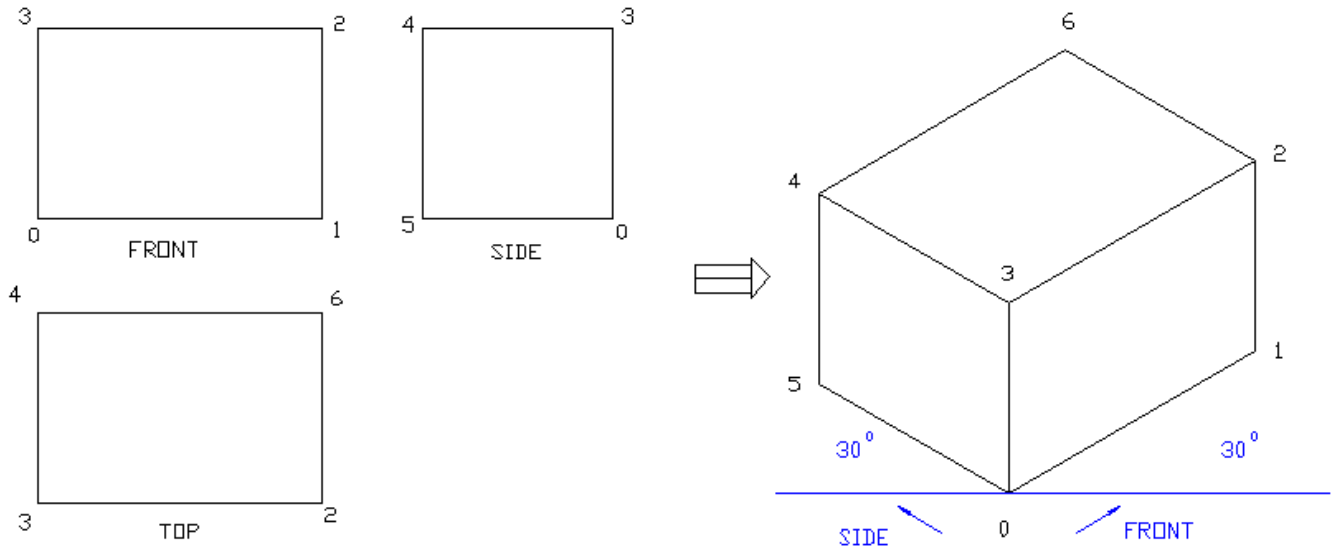


Figure 3 From Views to Isometric

Figure 4, shows the Isometric view of a model having inclined lines. In order to draw inclined lines in Isometric, the simple way to do it, is to find the two ends and then connect.

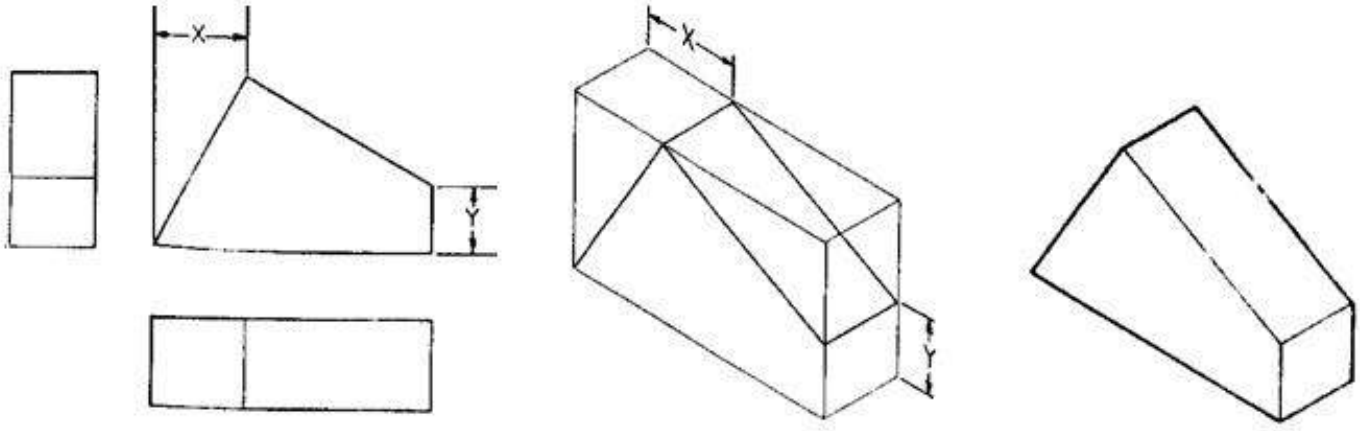
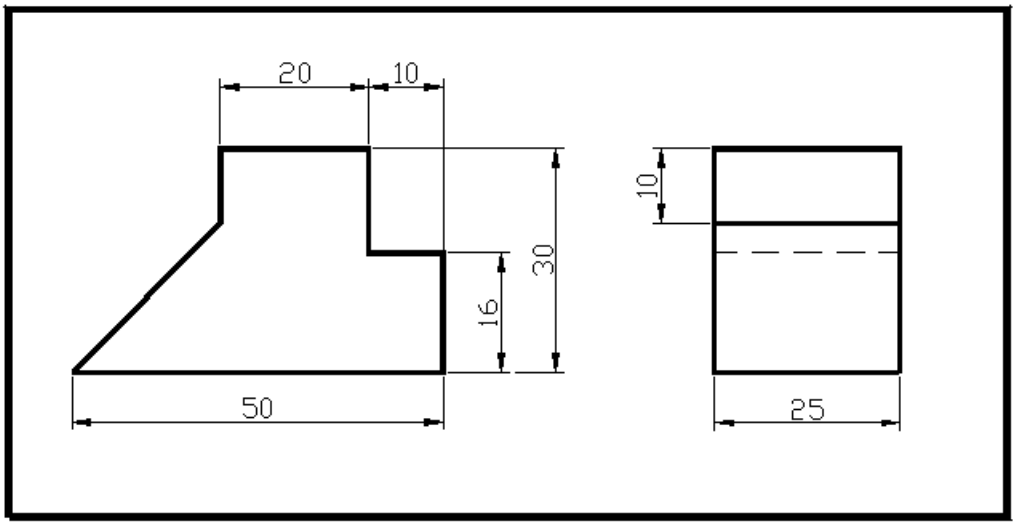
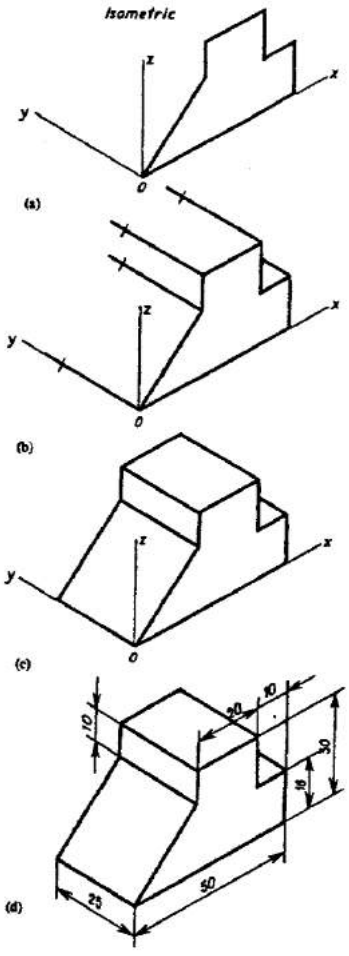
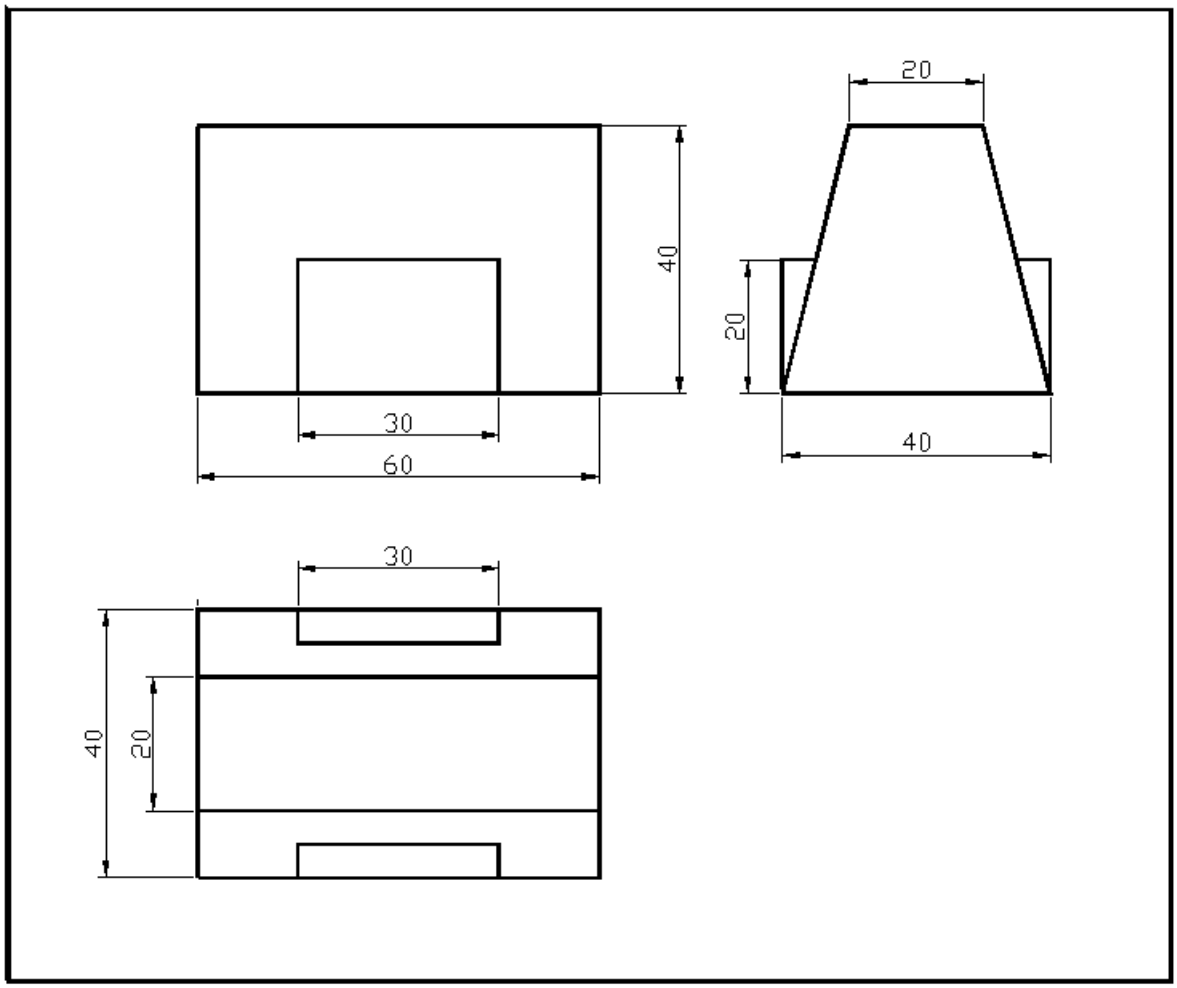


Figure 4 Isometric Applied to Inclined Lines

Solved Models



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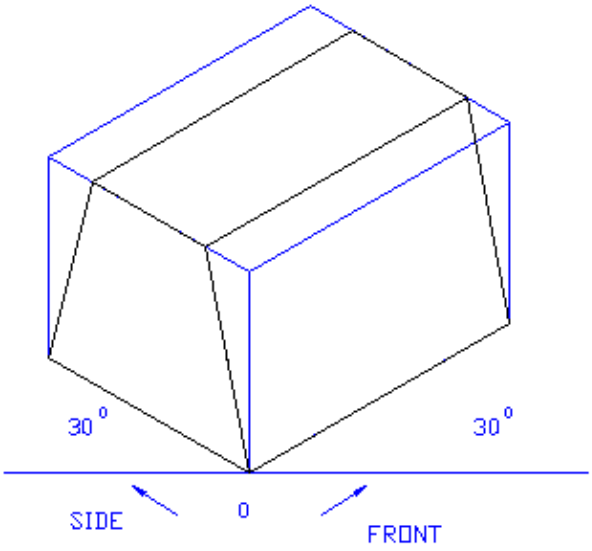
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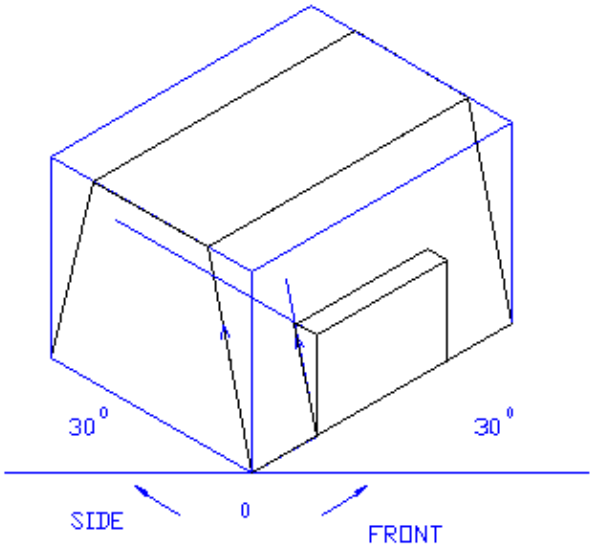
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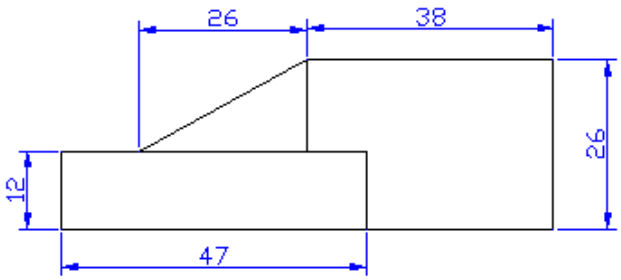
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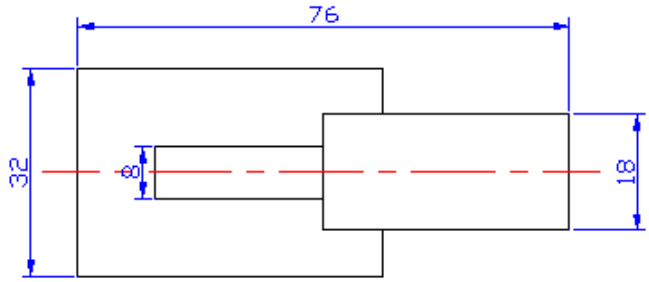
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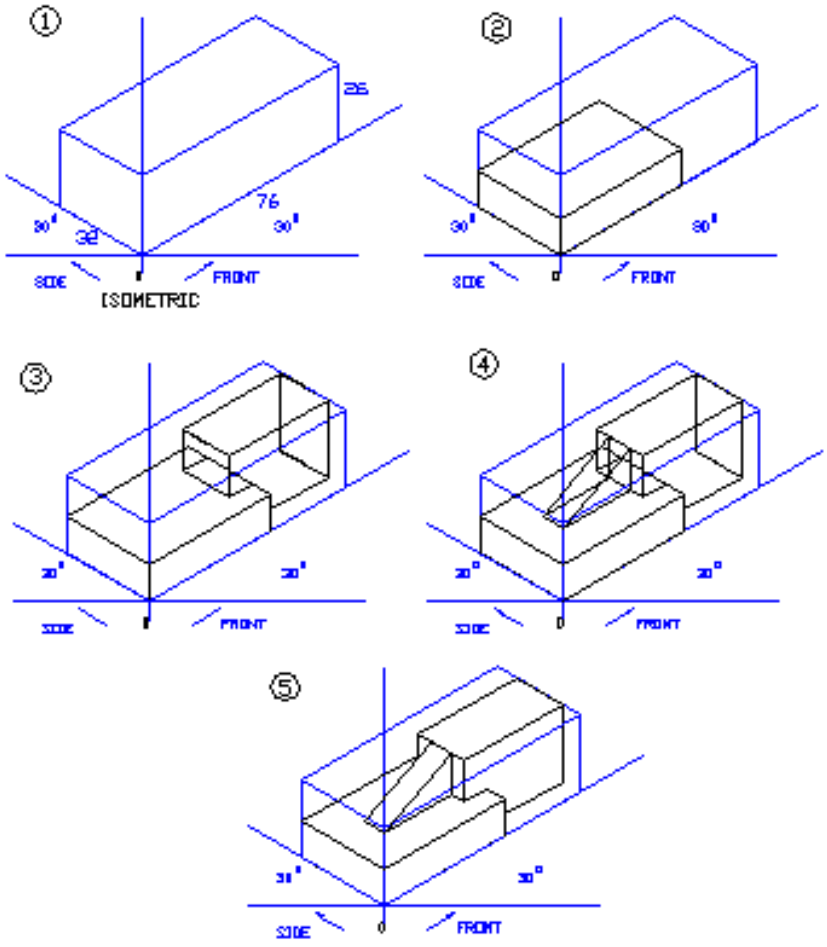
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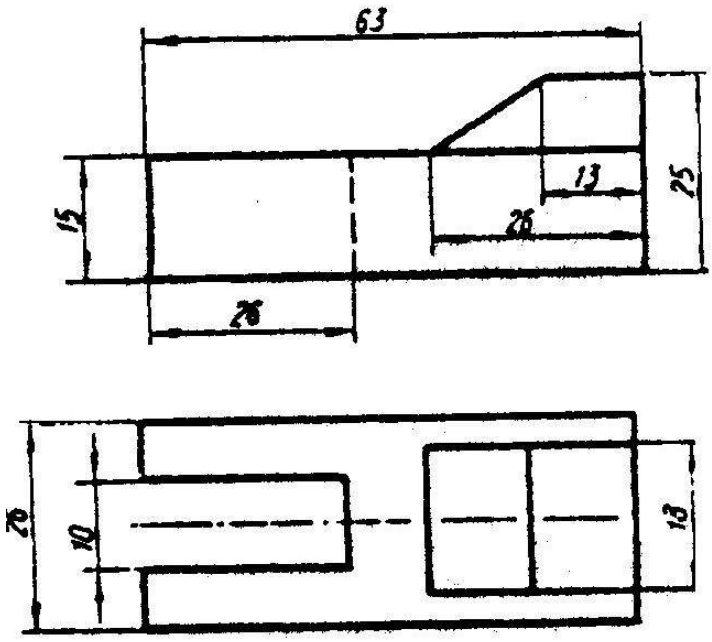
ISOMETRIC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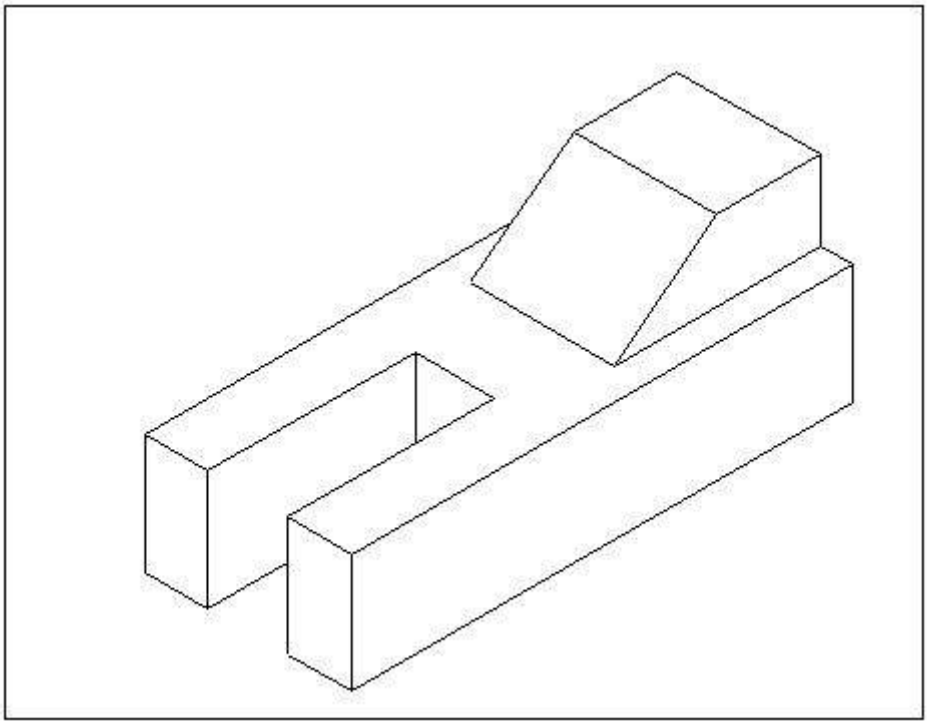
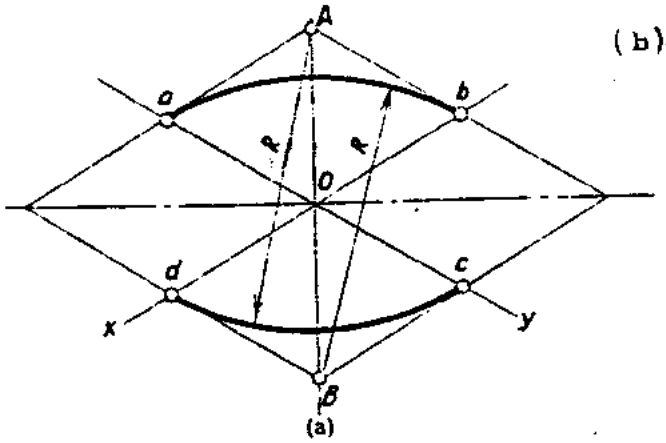
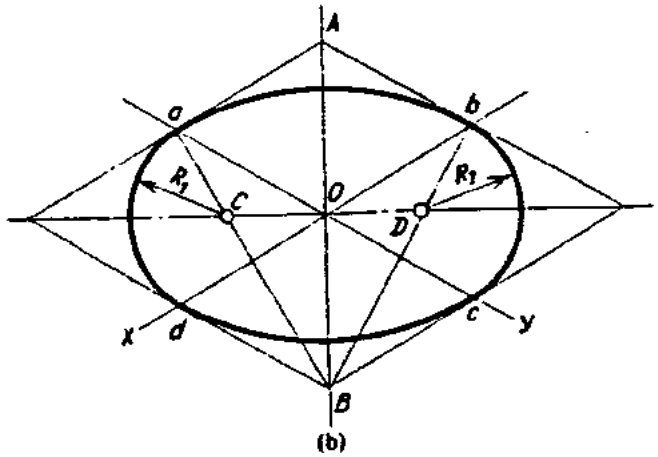


Drawing of Circles



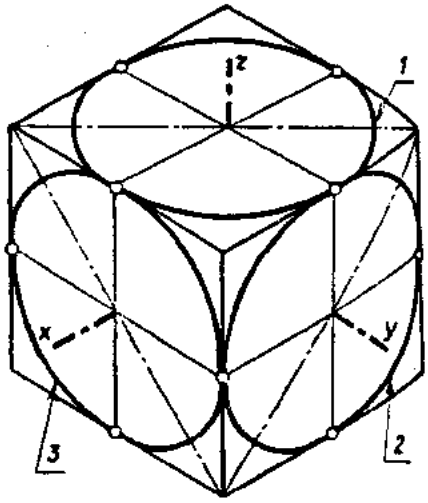
(a)

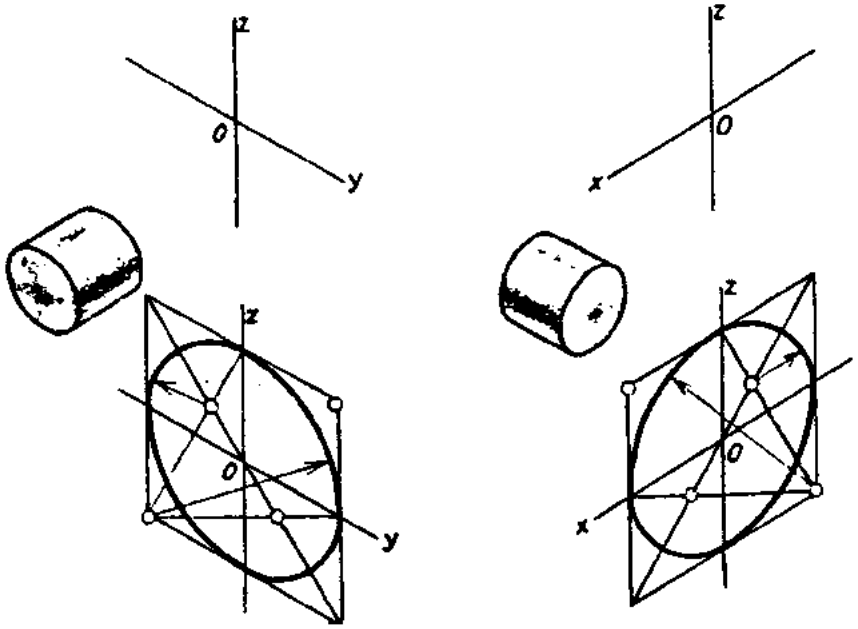
(b) Four-center method of constructing a circle in isometric

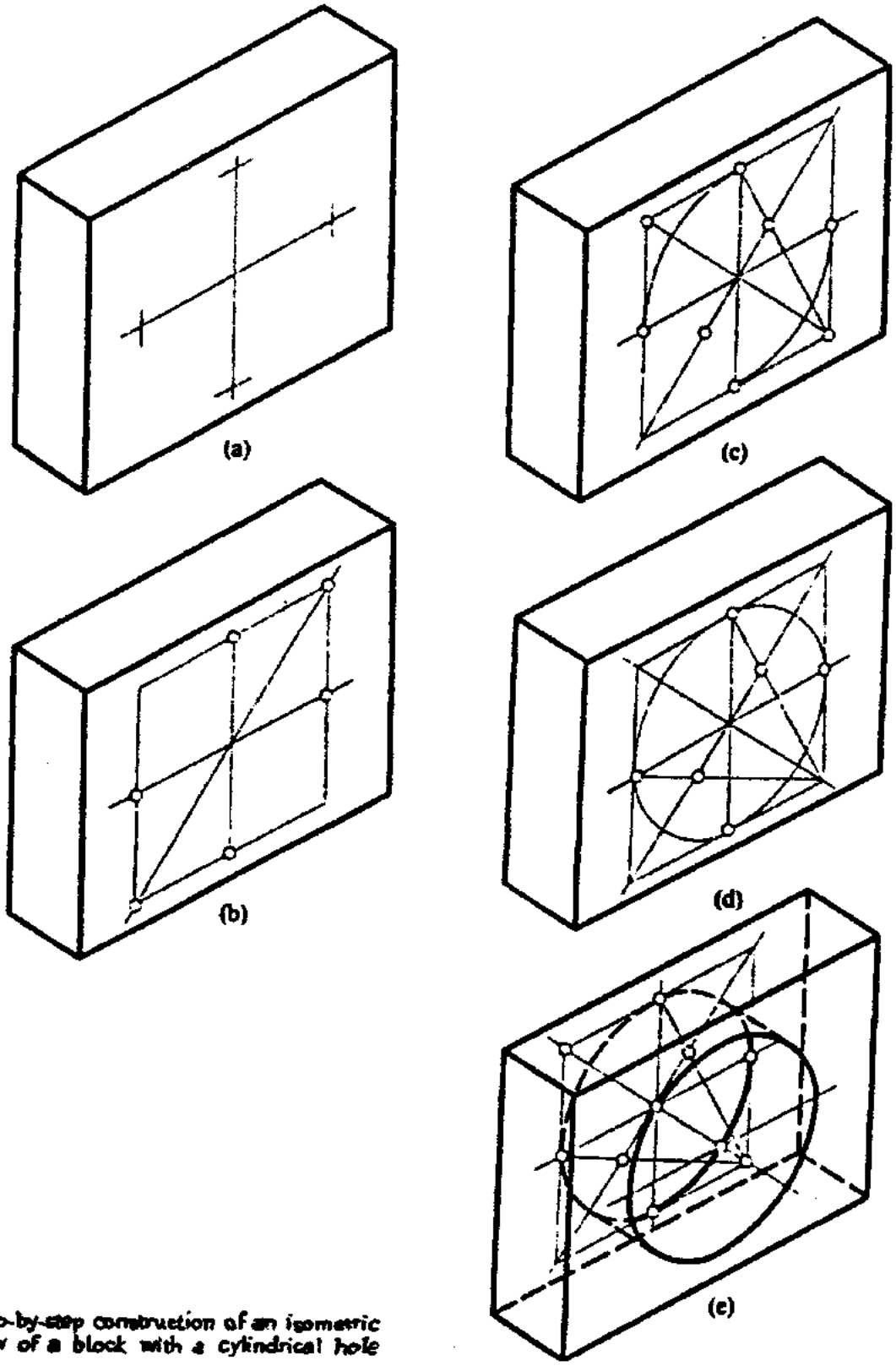


(b)

(a) Circles in isometric

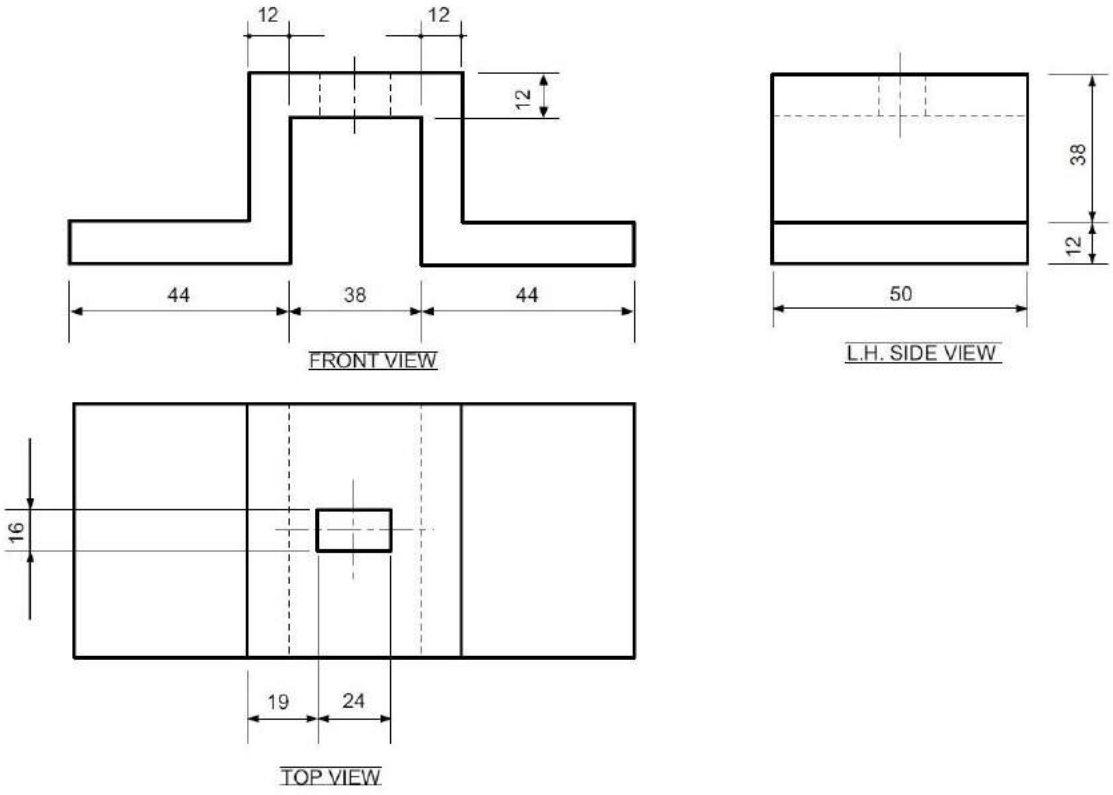


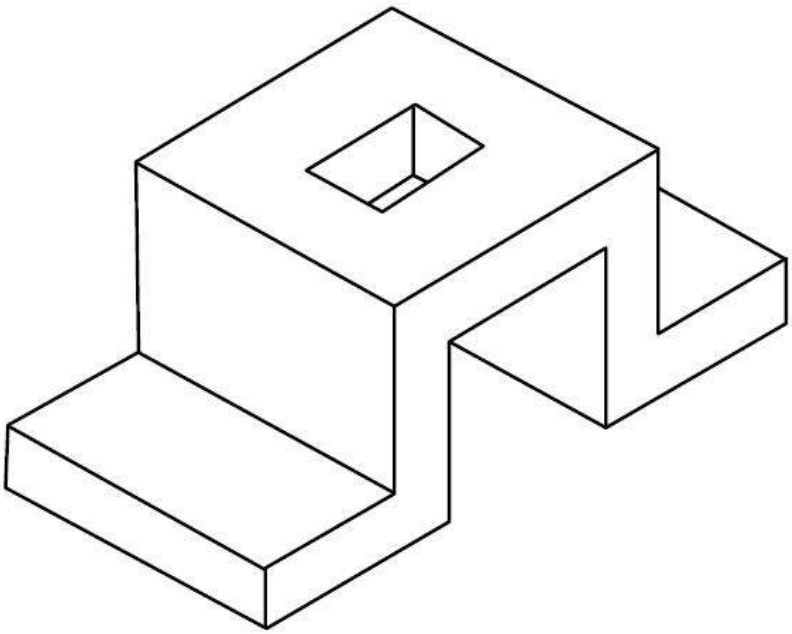


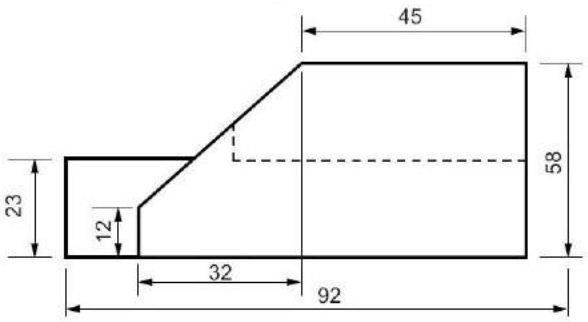


Step-by-step construction of an isometric view of a block with a cylindrical hole

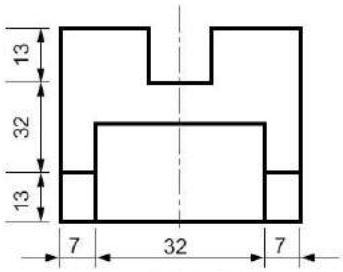
Solved Models (Isometric)



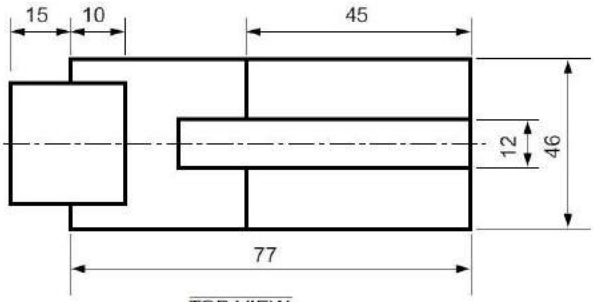




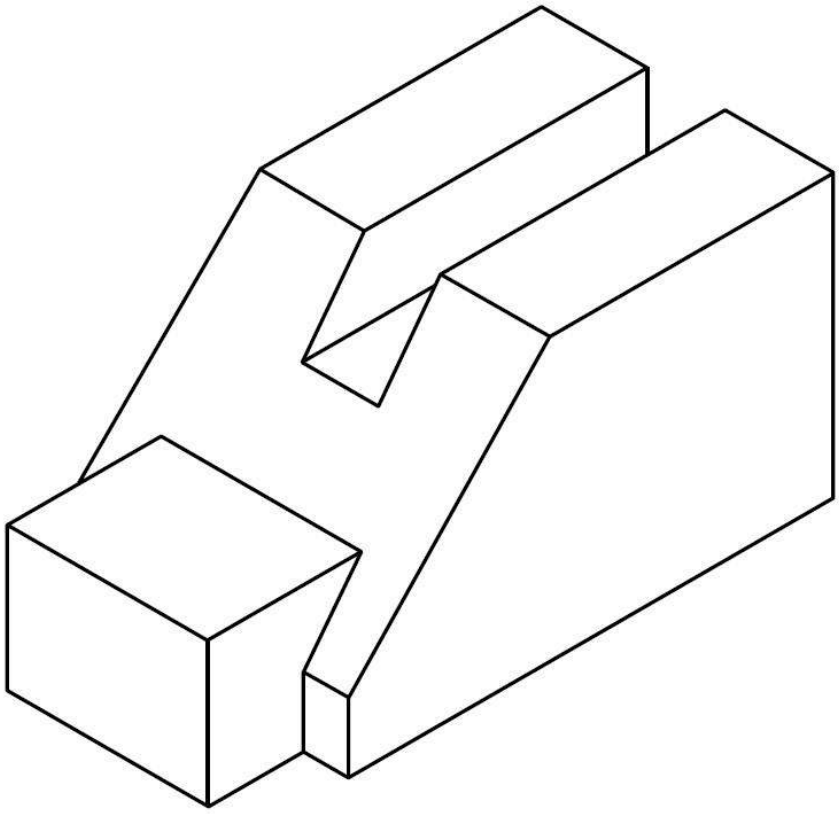
FRONT VIEW

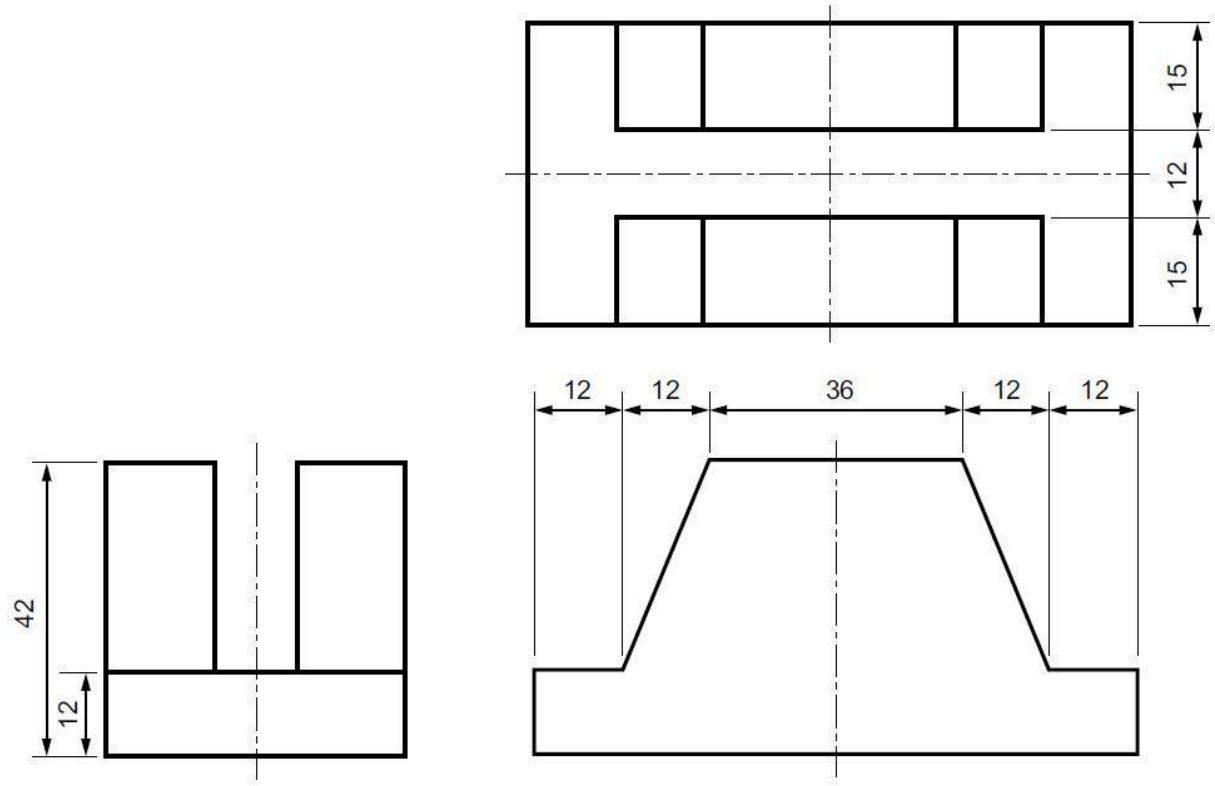


L.H. SIDE VIEW



TOP VIEW

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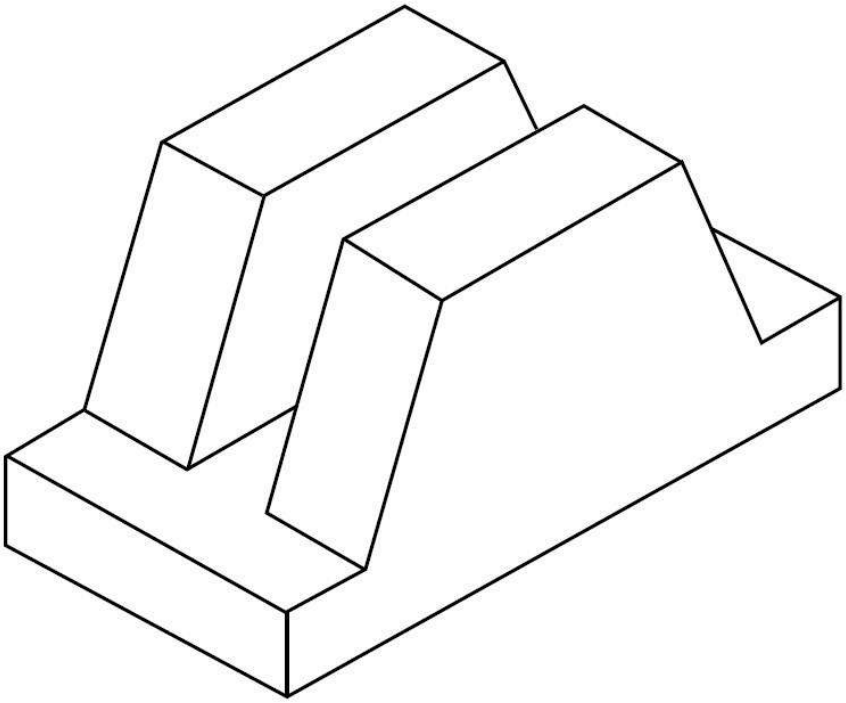

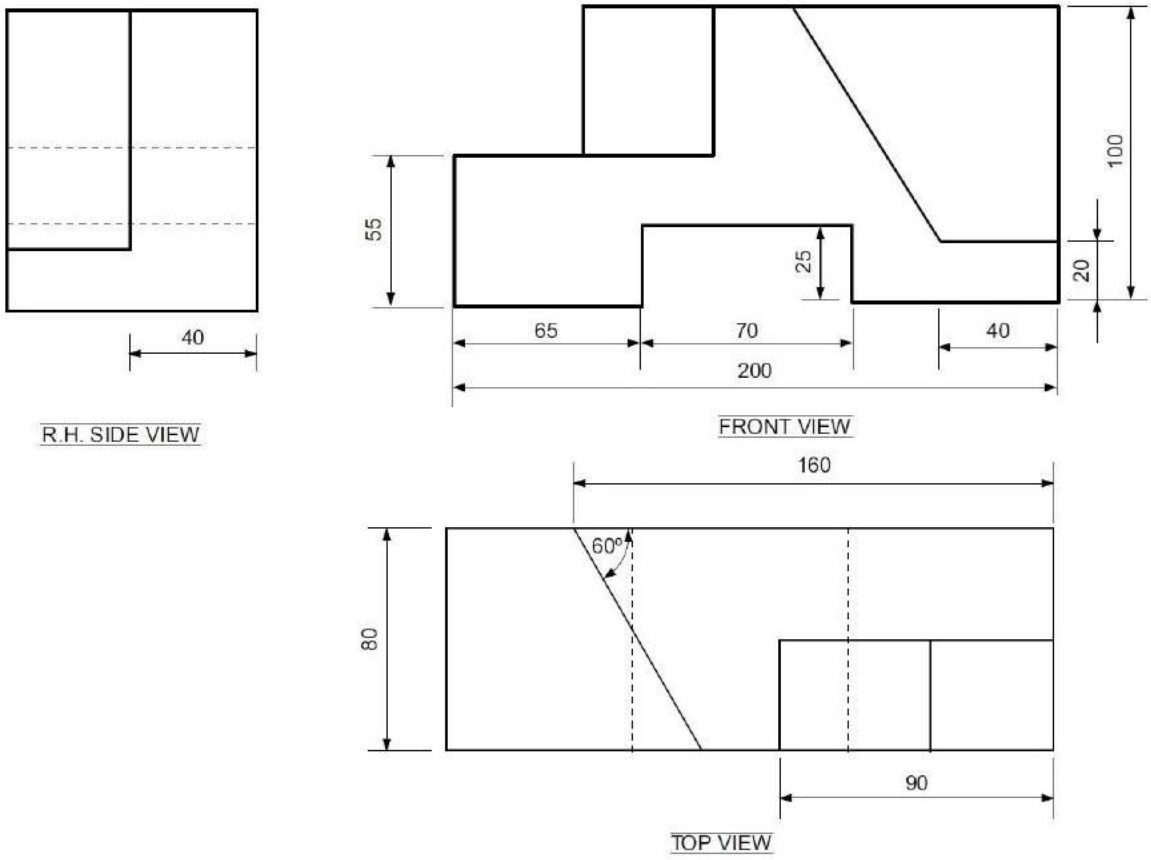
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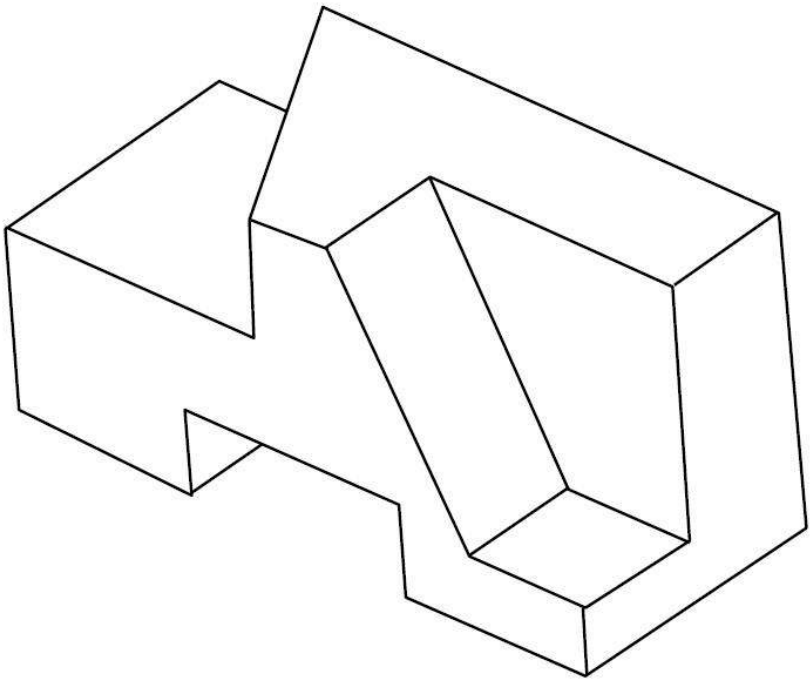

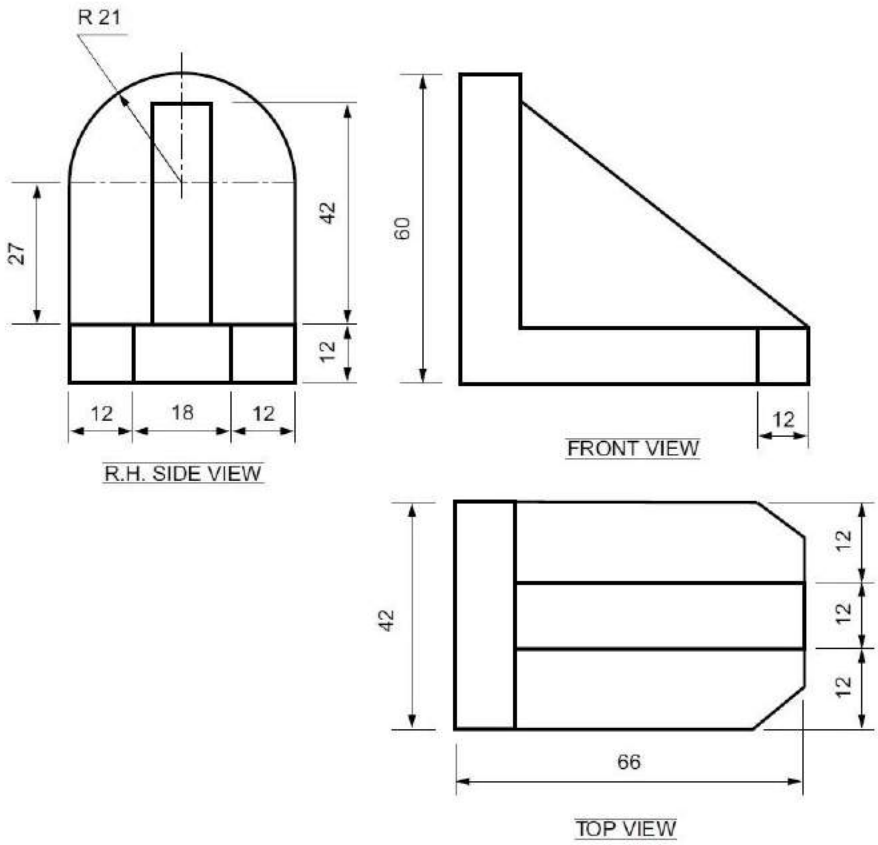
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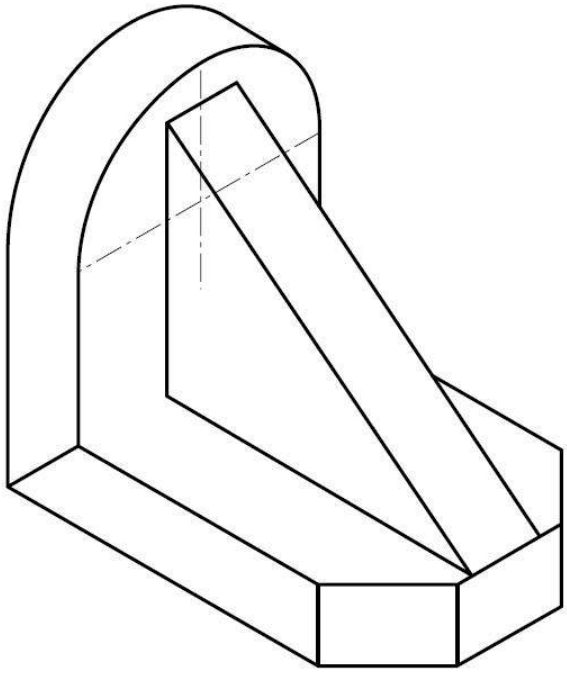

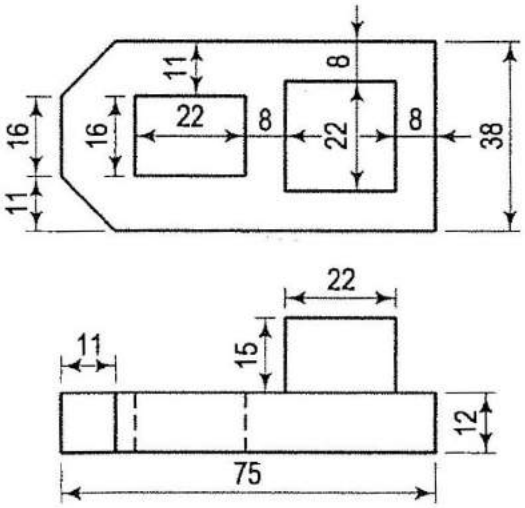
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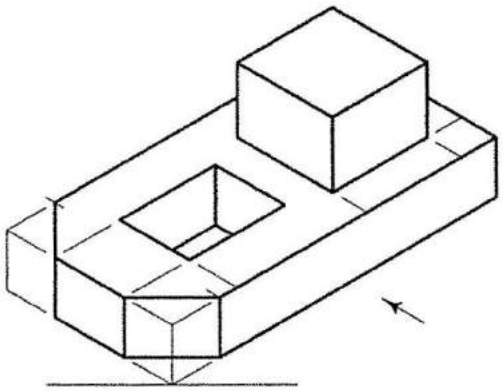

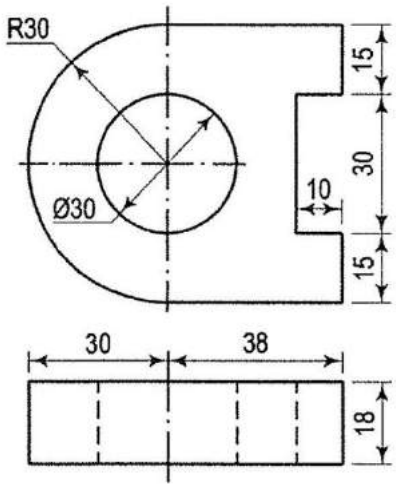
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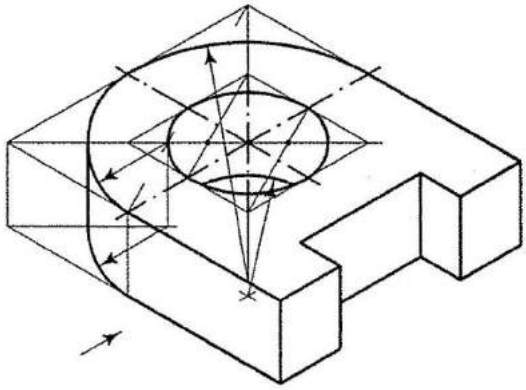

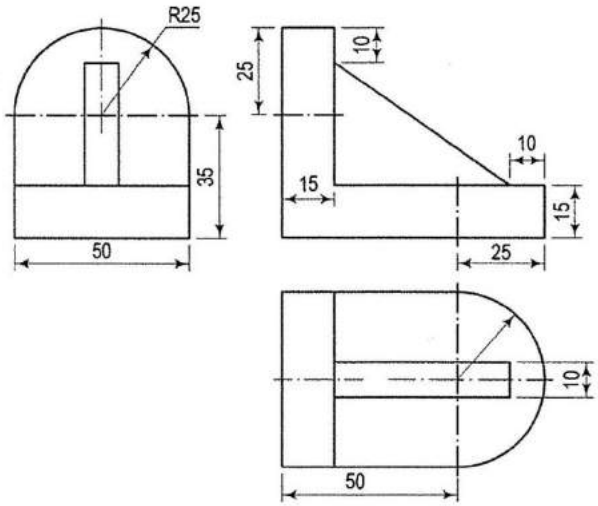
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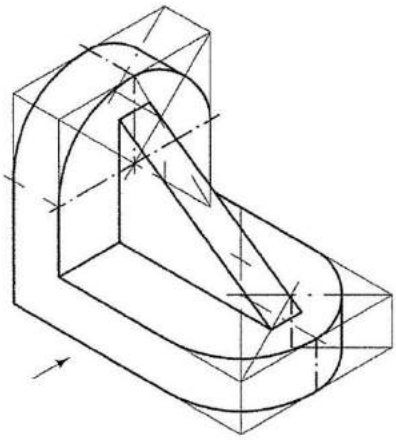

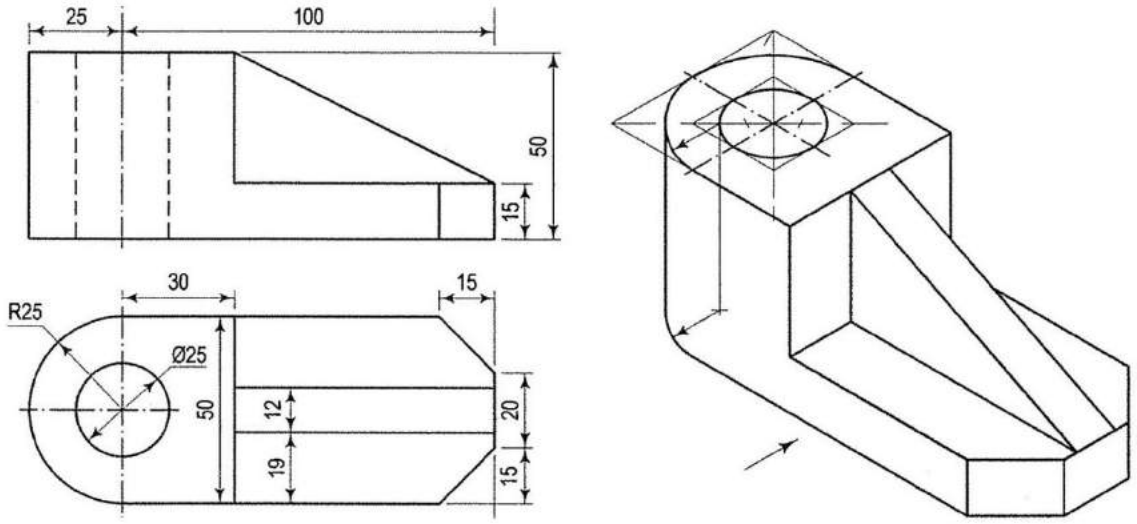
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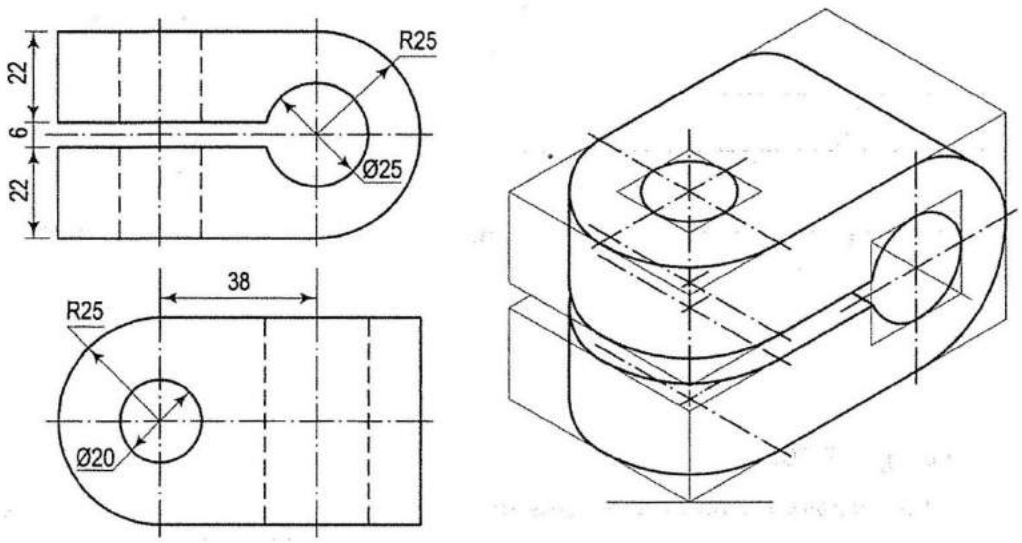
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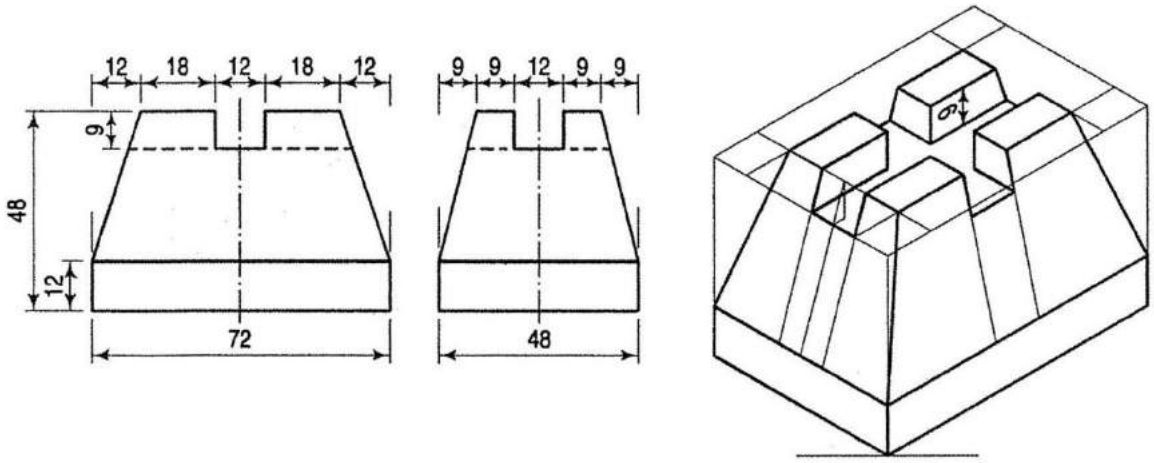
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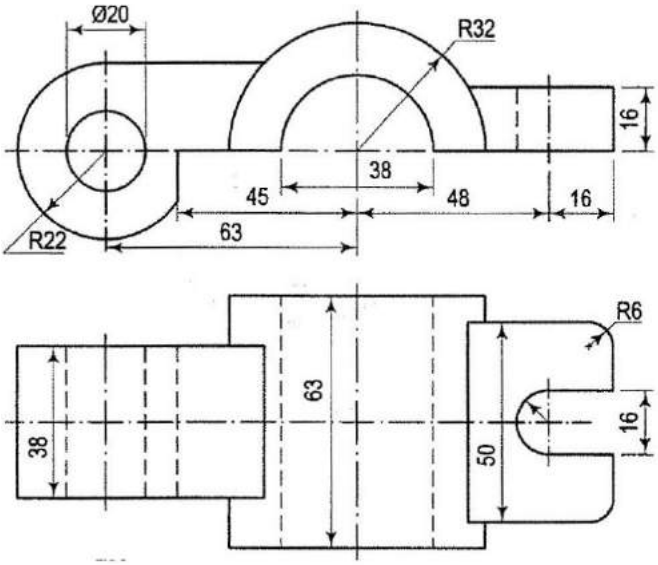
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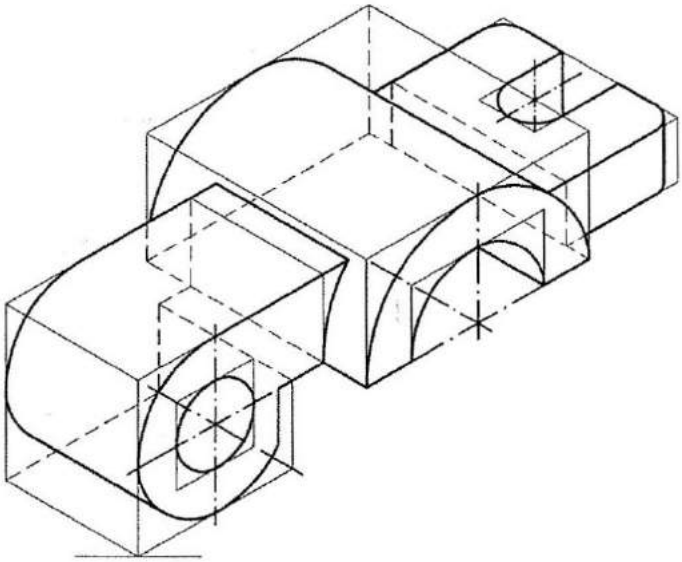
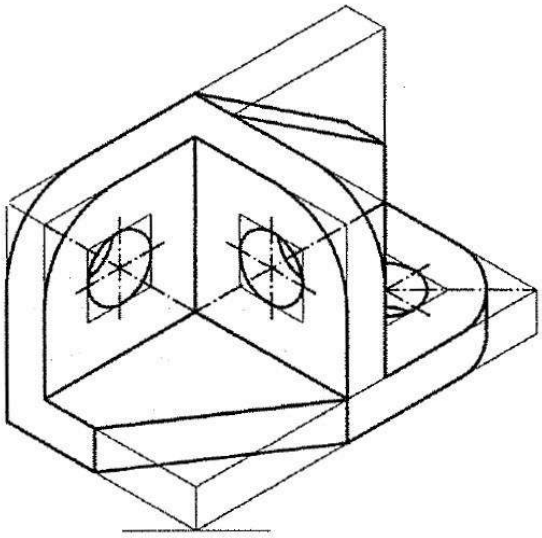
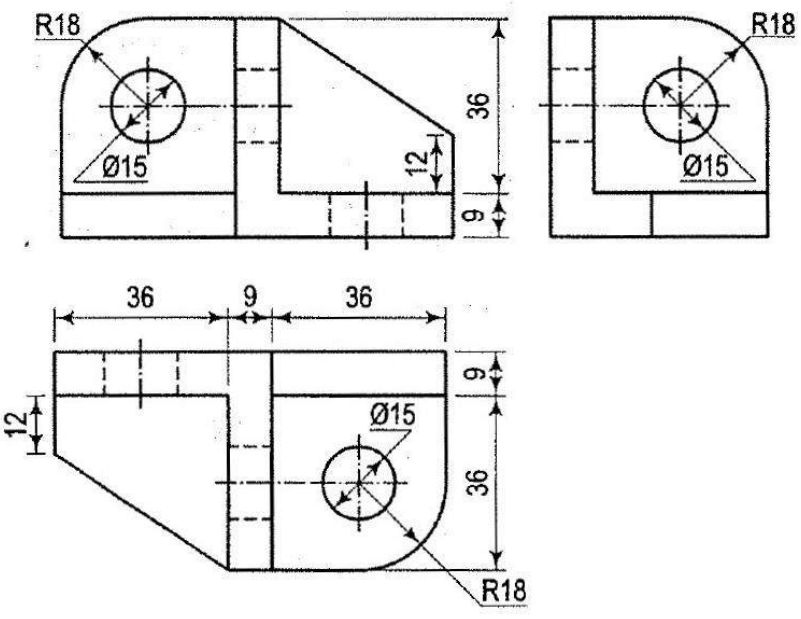
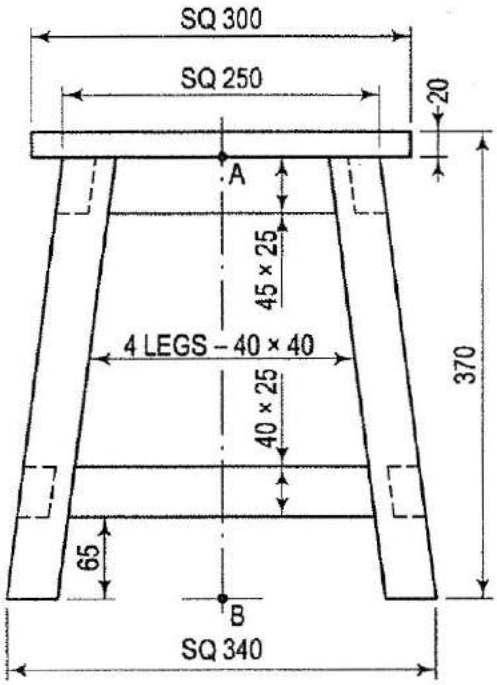
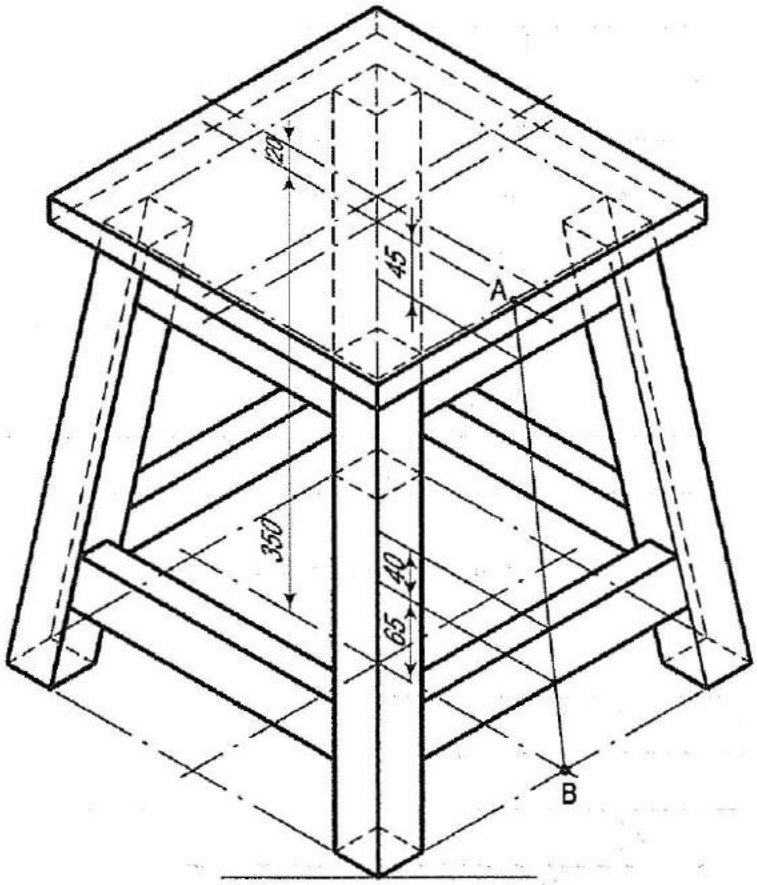
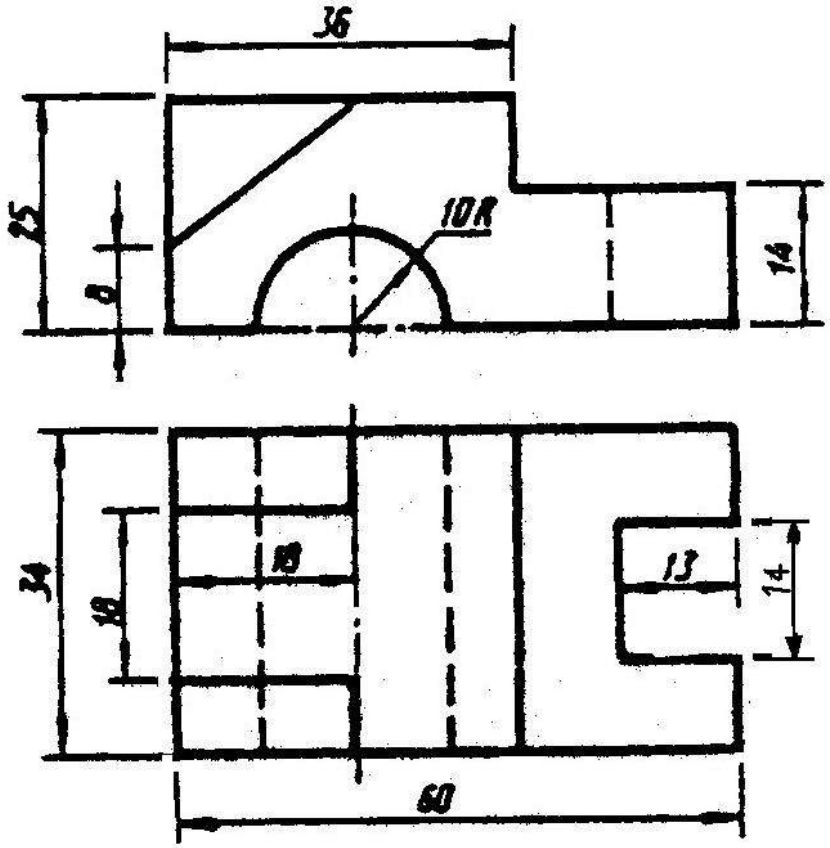





R18
Ø15
12
36
9
R18
Ø15
36
9
36
9
12
Ø15
36
R18


SQ 300
SQ 250
20
A
45 × 25
4 LEGS – 40 × 40
40 × 25
370
65
B
SQ 340


120
45
A
350
40
65
B


36
25
8
10R
14
34
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18
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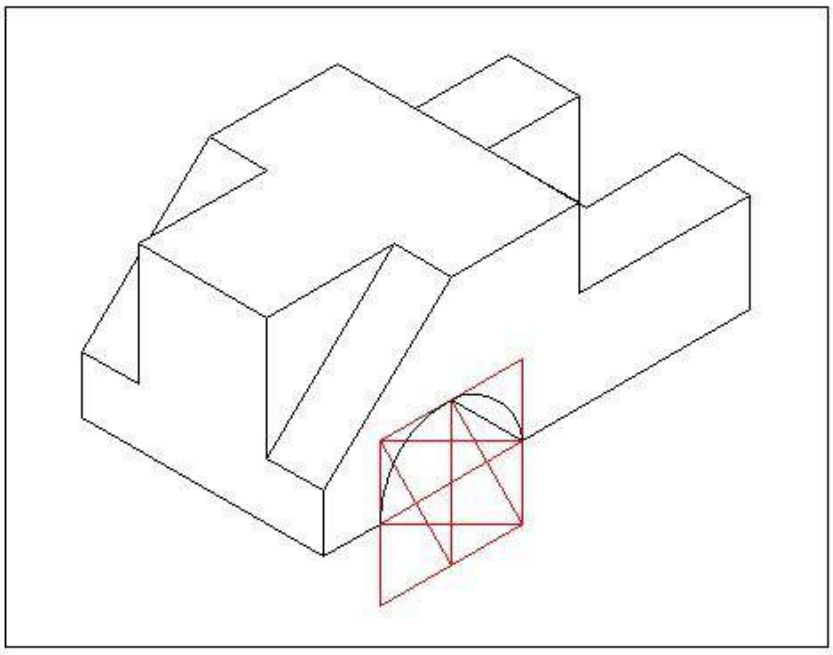
14
60



2- OBLIQUE DRAWING

Oblique drawing is another way to show 3D models on a 2D piece of paper. Oblique drawing is considered a quick and easy way to show an idea in 3D. We have two types of Oblique drawing. The cavalier type, with full depth at 45^0 and cabinet type with half scale along the depth at 45^0 also.

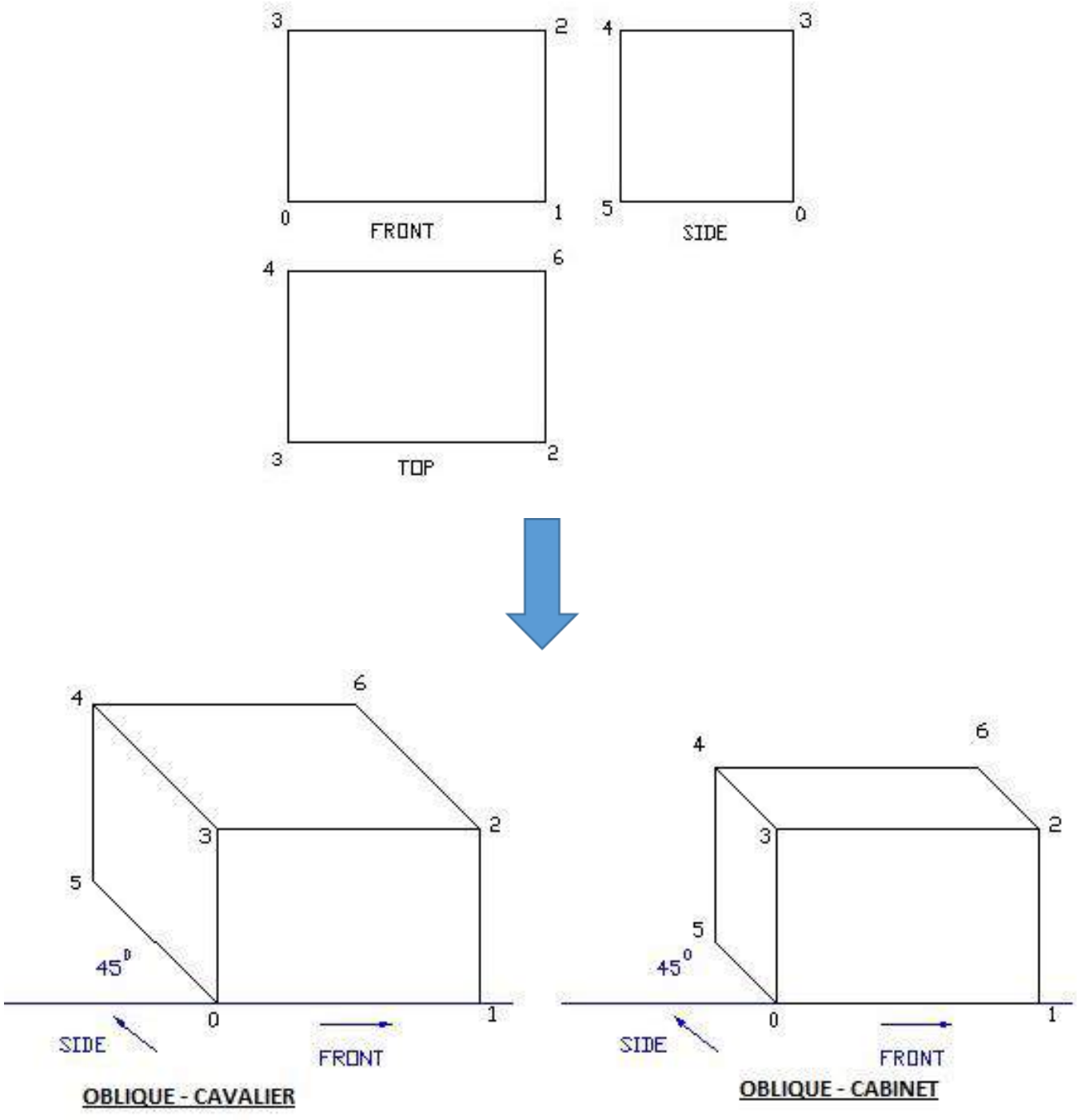
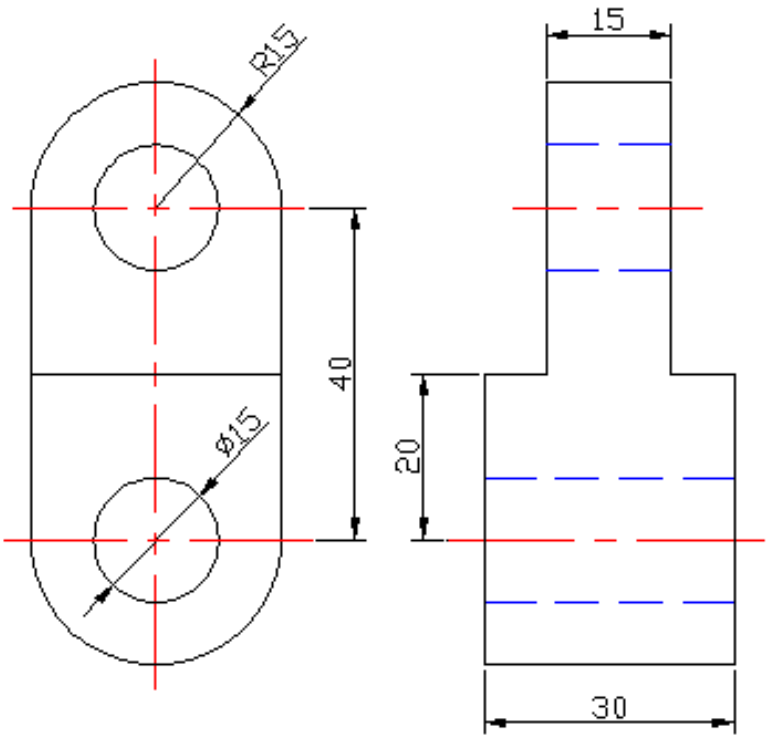
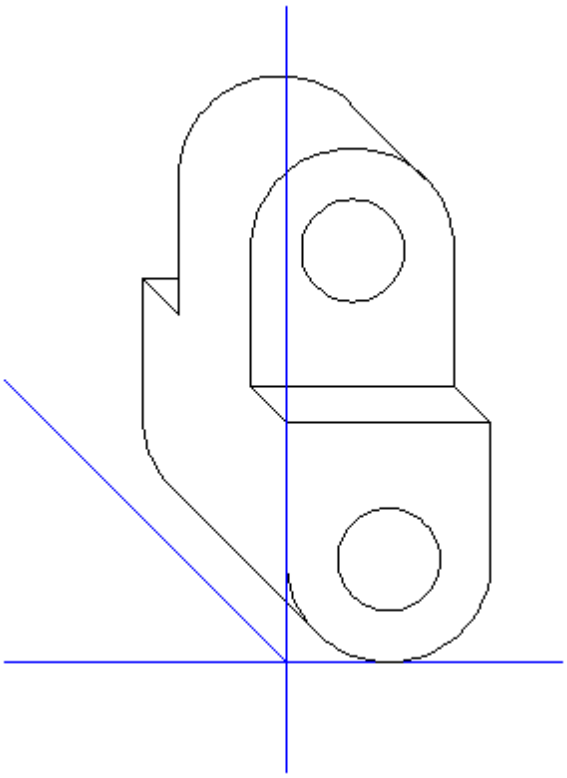
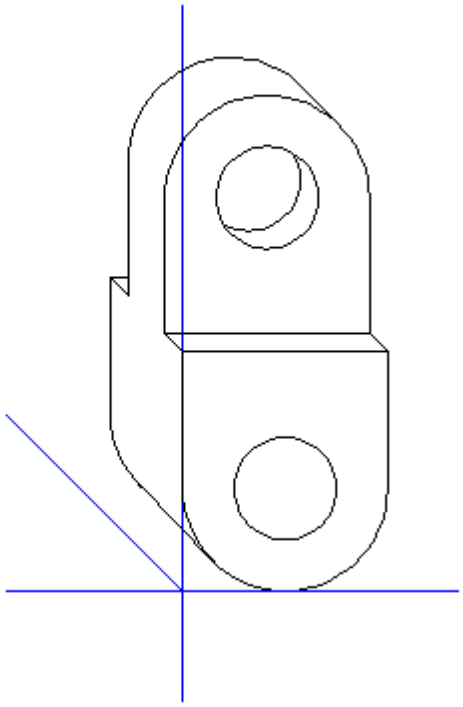


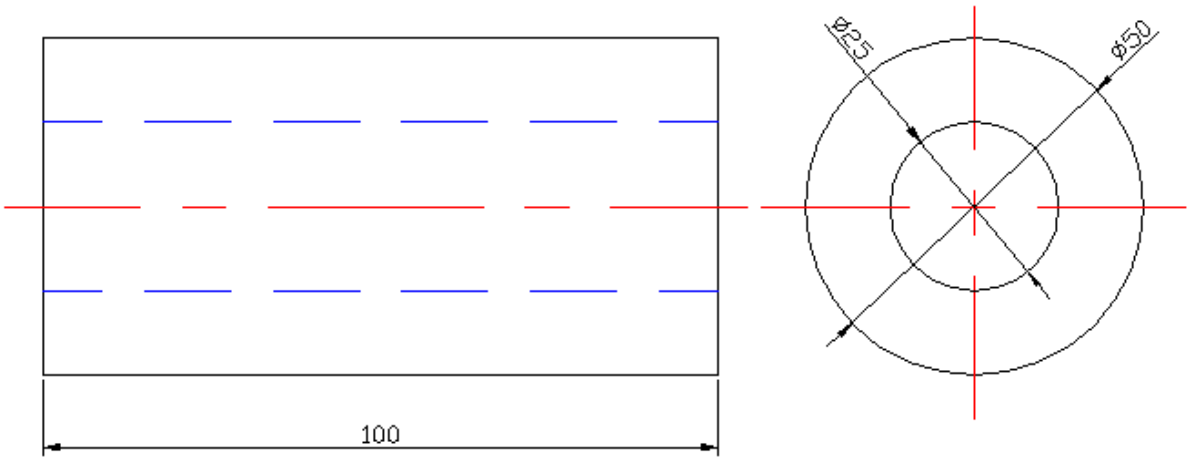
Figure 5 Oblique Drawing

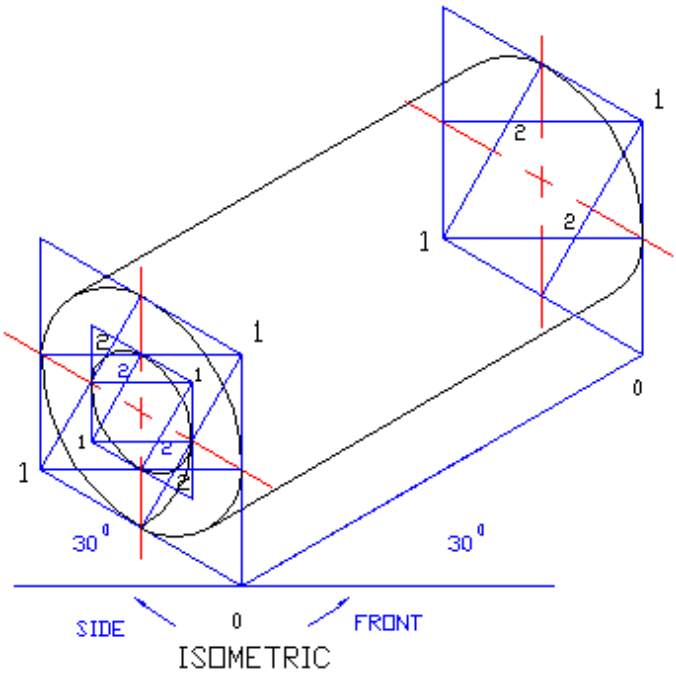
Solved Models



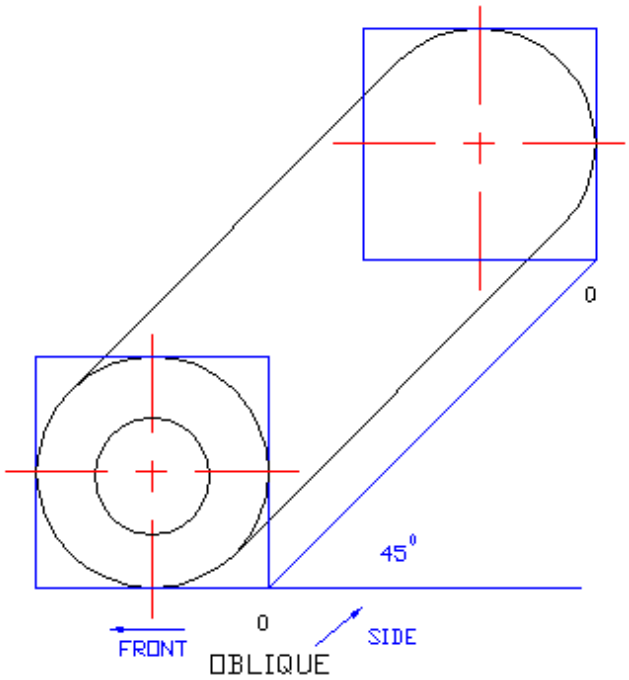






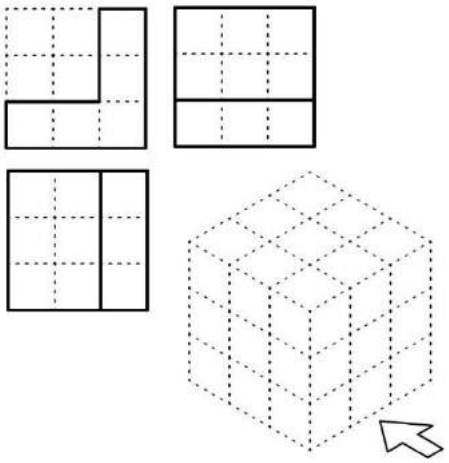


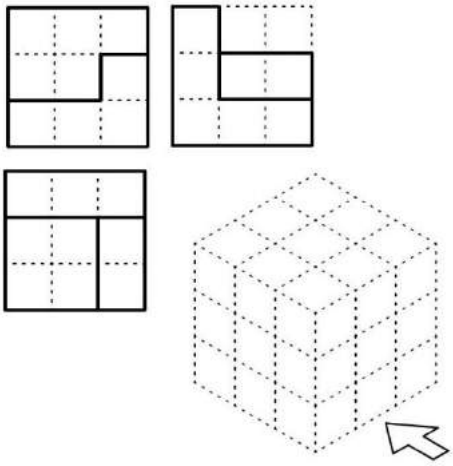
ISOMETRIC

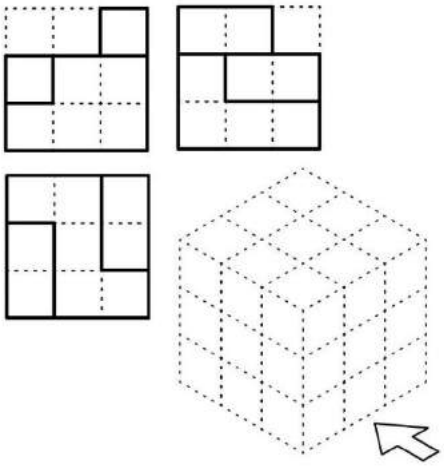


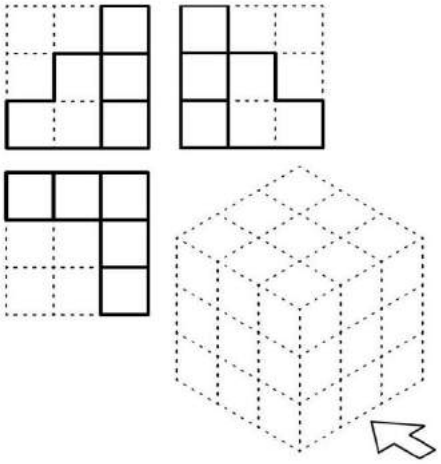
OBLIQUE

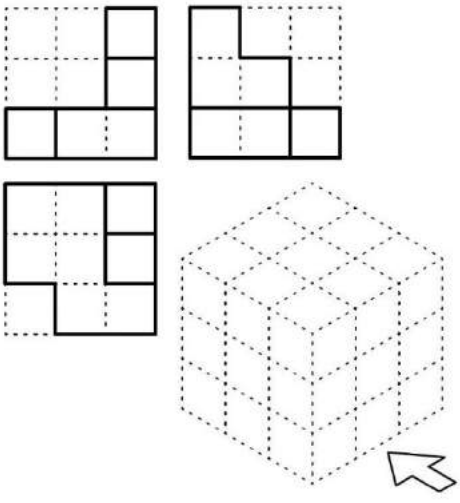
Problems

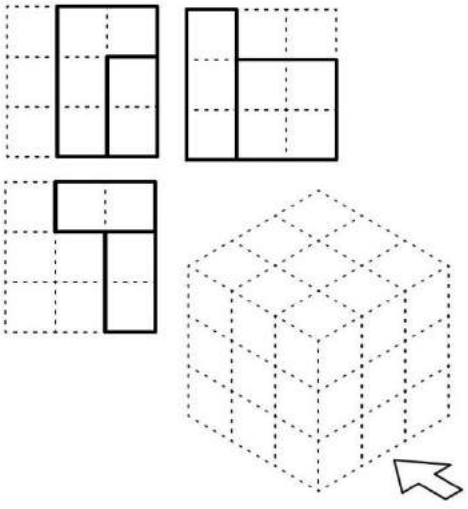


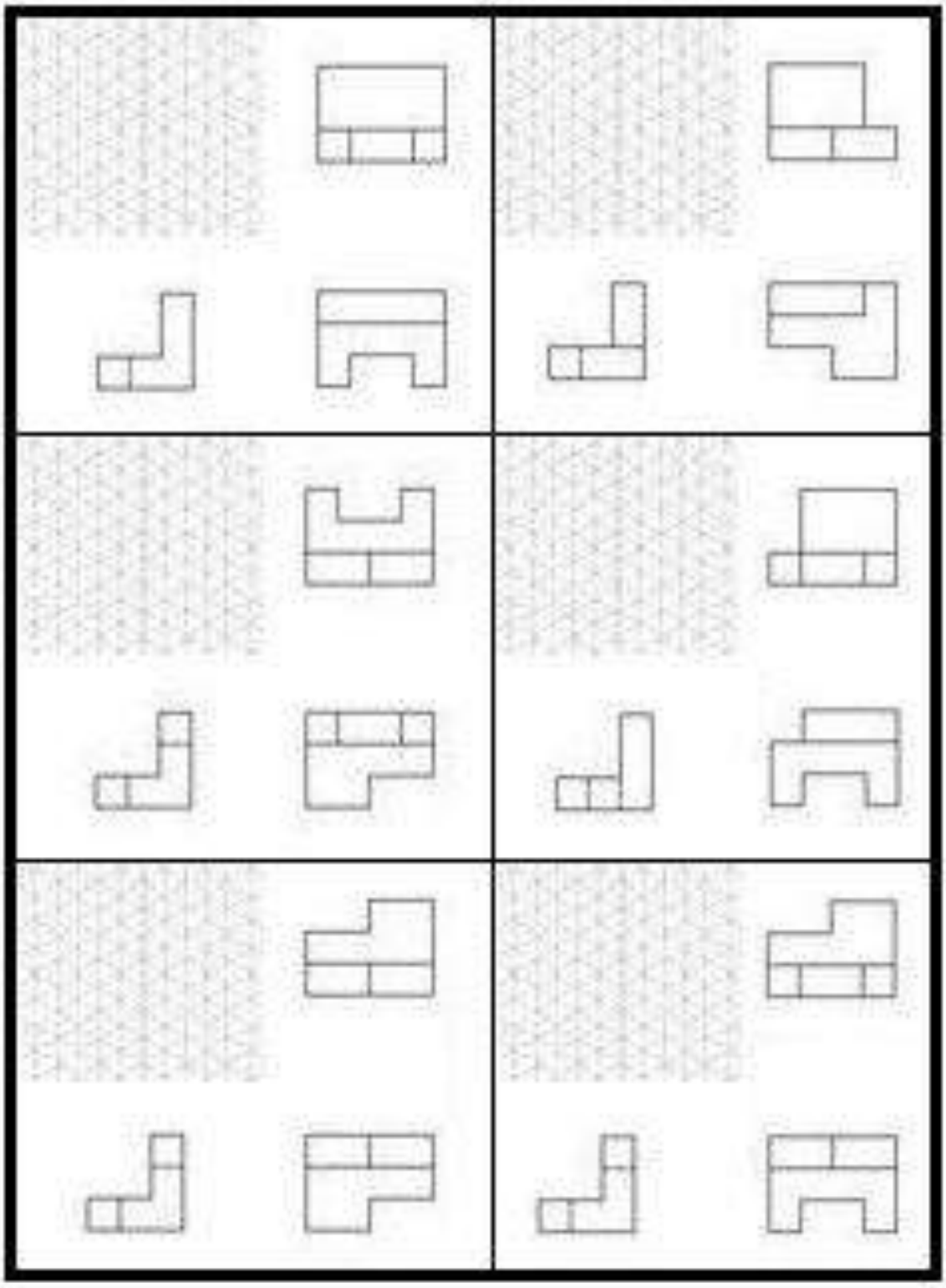




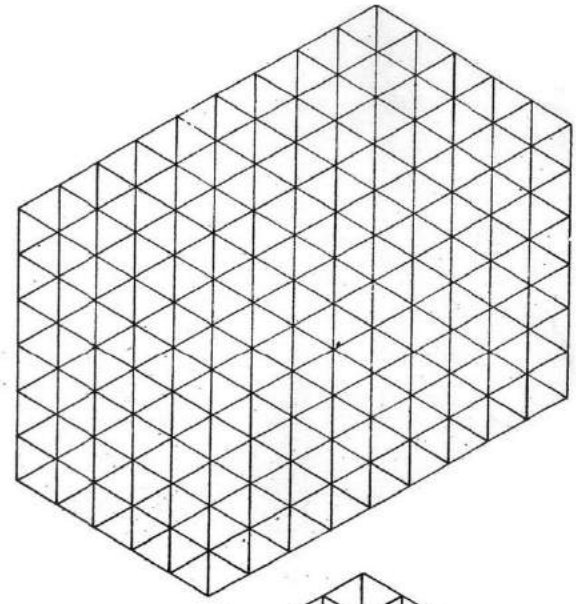


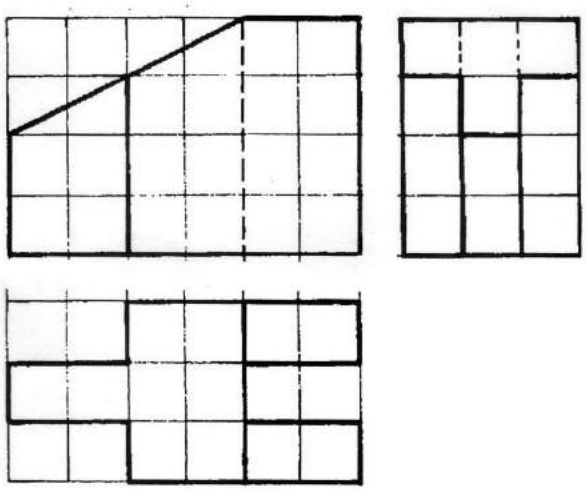


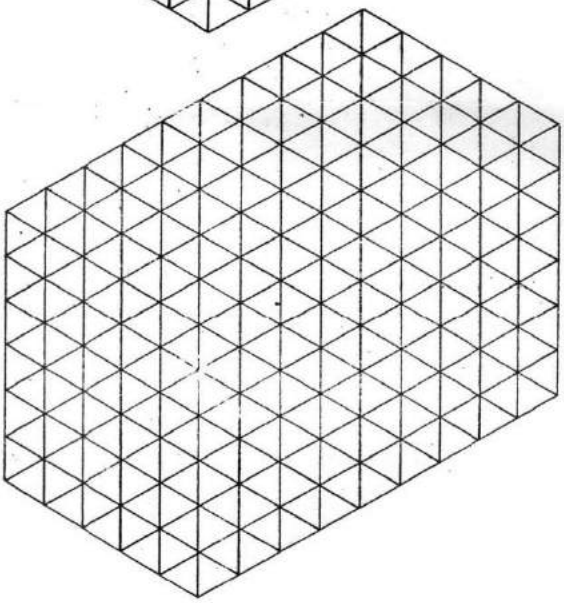


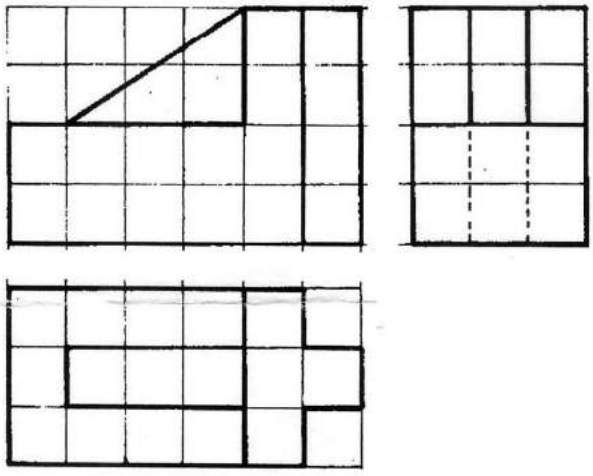


Draw the Isometric for the given 2 models

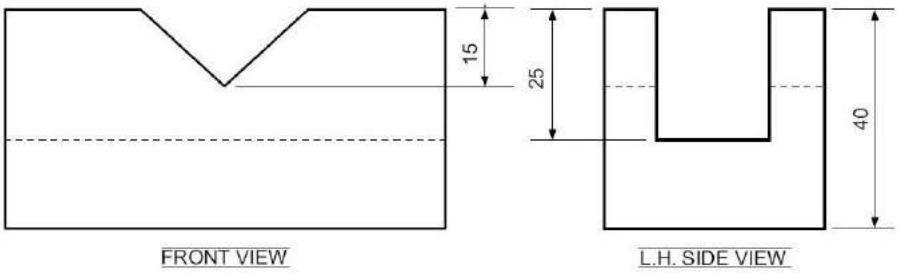


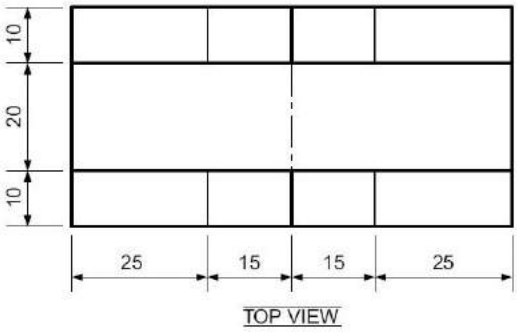


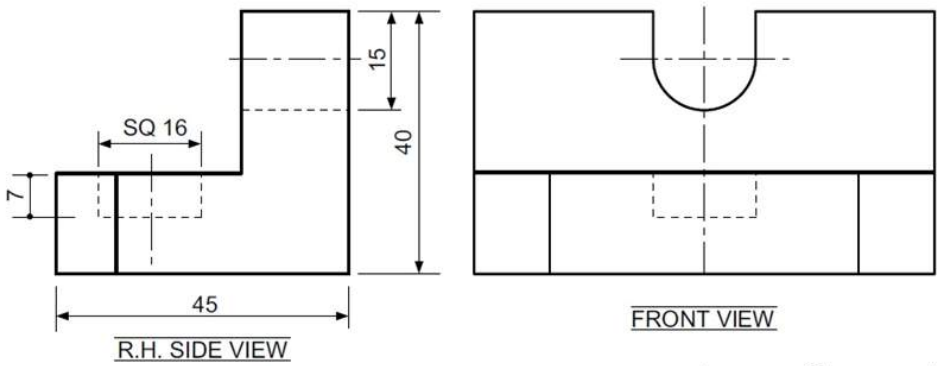


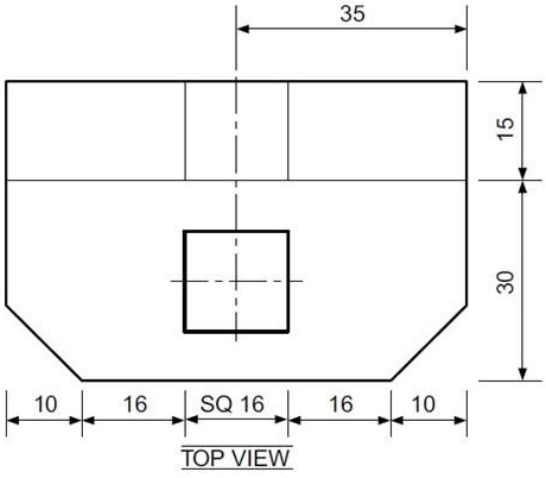


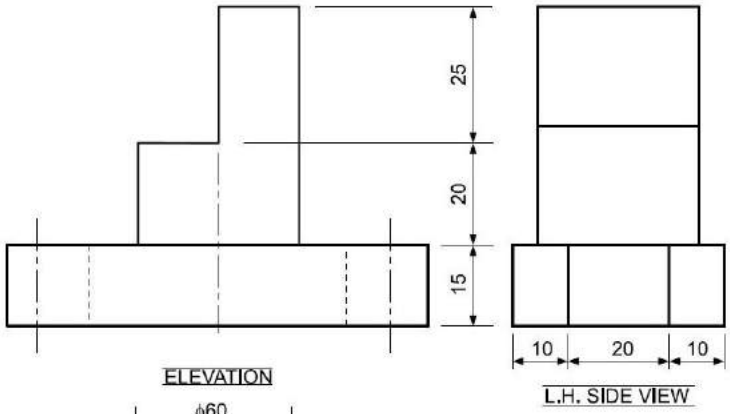
Draw the Isometric and Oblique of the following models

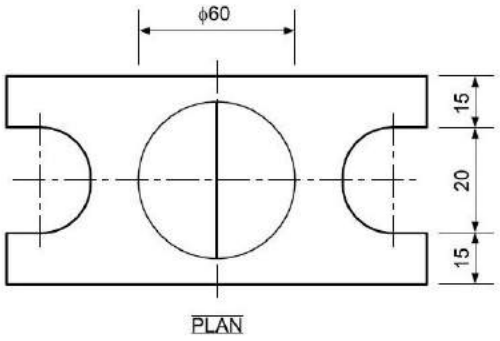


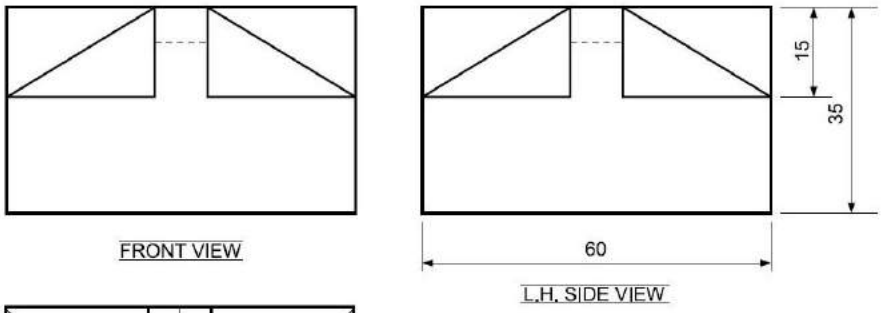


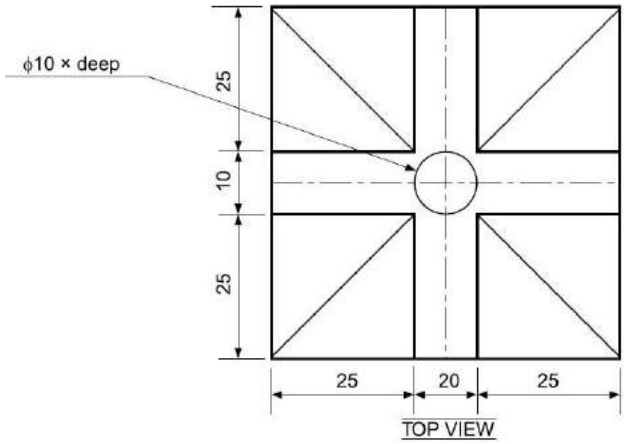


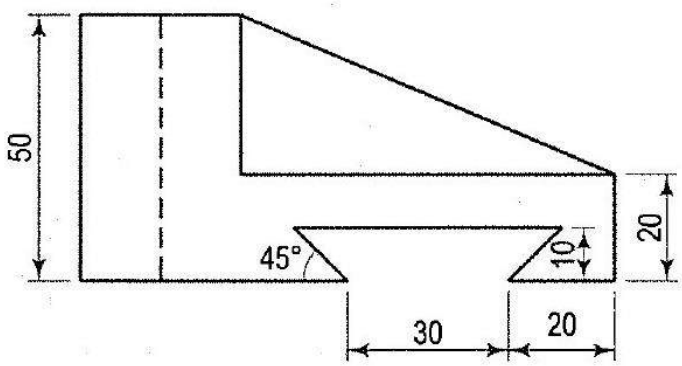


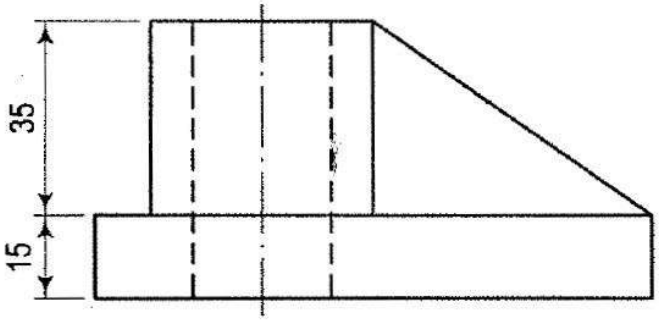

25
20
15
10
20
10
ELEVATION
L.H. SIDE VIEW

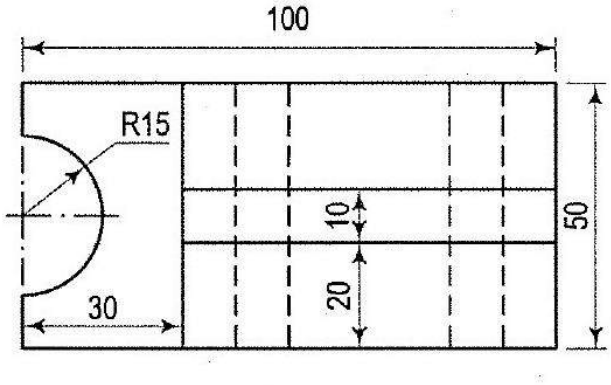

ϕ60
15
20
15
PLAN

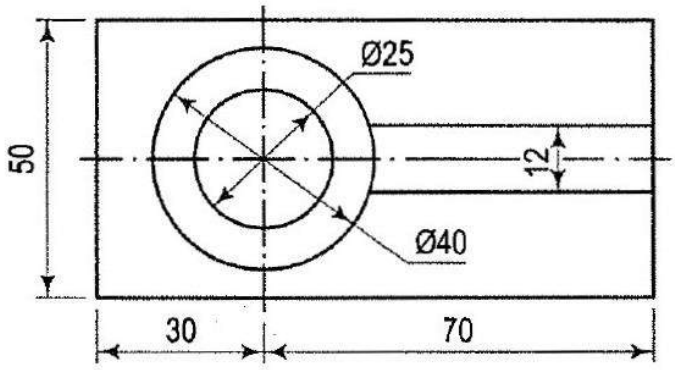

15
35
60
FRONT VIEW
L.H. SIDE VIEW

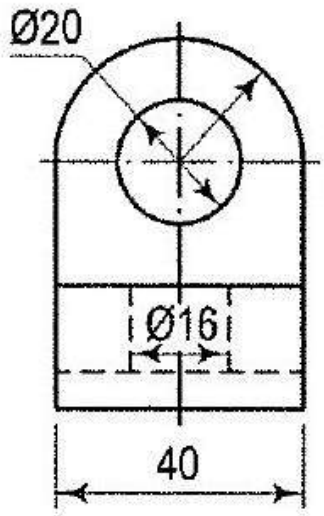

ϕ10 × deep
25
10
25
25
20
25
TOP VIEW

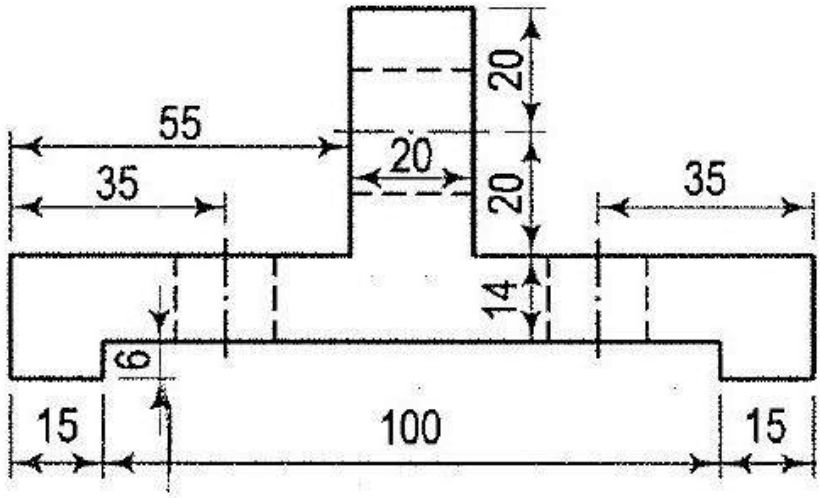

50
45°
10
20
30
20


35
15


100
R15
10
20
50
30


Ø25
12
50
Ø40
30
70


Ø20
Ø16
40

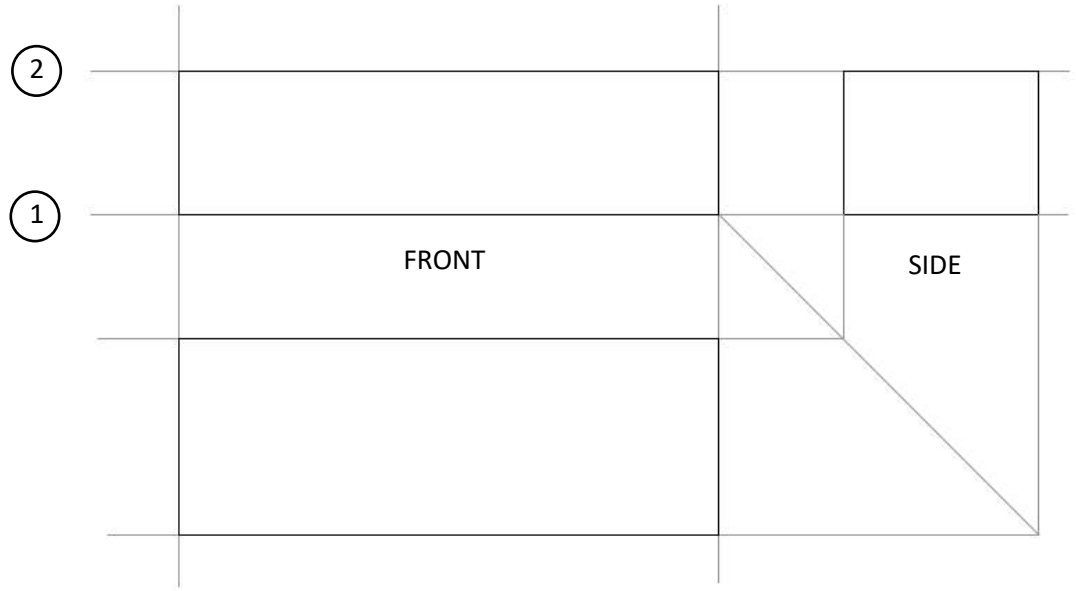

55
35
20
20
20
35
14
6
15
100
15

The concept of missing view is coming from the orthographic projection. The new part here is that the 3D model is not given, only 2 views are given and it is required to find the 3rd one. The front view is always given since it is the main view then the problem of missing view is always to find the top view or the side view.

No new rules are applied in missing view while using the 45° line is a must in order to transform lines from the side to the top view or from the top to the side.

A very simple rule is valid if the missing view is the top.

Given the front and the side, in order to get the top, let us apply the rule of ***levels:***

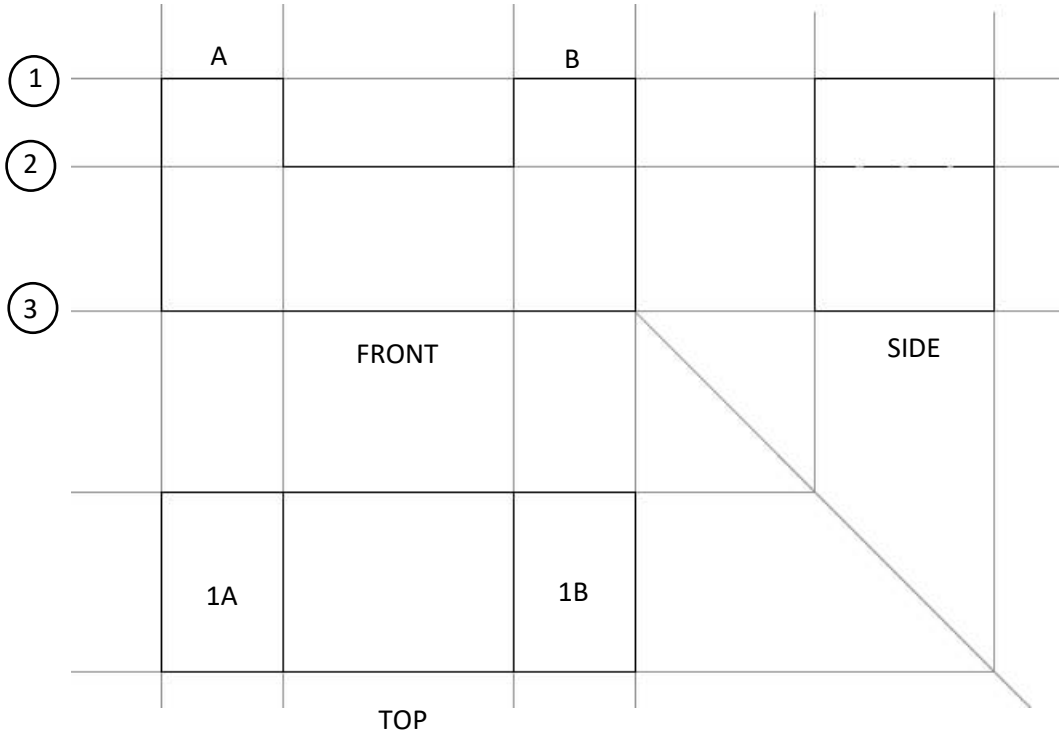


Considering level 1 as shown in the figure above, at this level, we have a line in the front view and a line at the same level in the side view. The intersection of the construction lines formed from these lines forms a rectangle.

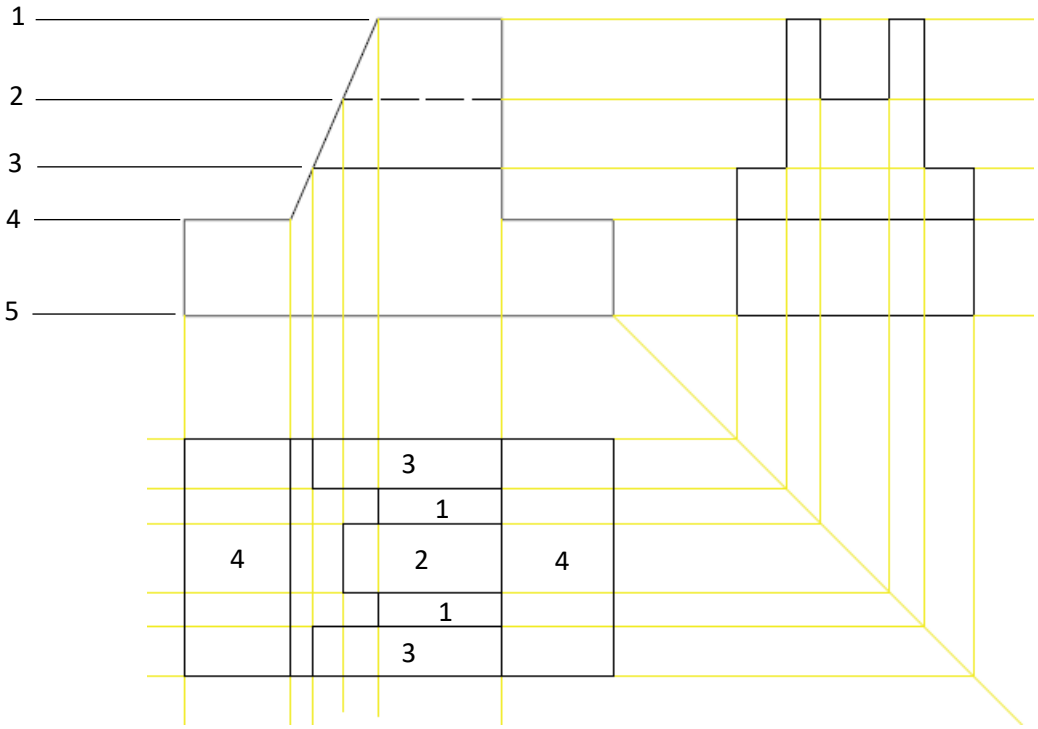
Note that at level 2, the intersection forms a rectangle identical to the first one.

In figure 2, we are adding one more level as shown,

1. Level 1: we have 2 lines in the front, and at the same level we have 1 line in the side view.
   What do you expect in the top view? We should expect n rectangles, $n = 2 \times 1 = 2$ rectangles. The 2 rectangles are 1A and 1B
2. Level 2: we have 1 line in the front and 1 line in the side, $n = 1 \times 1 = 1$ rectangles.
3. Level 3: we have 1 line in the front and 1 line in the side, $n = 1 \times 1 = 1$ rectangles.



Let us now apply this rule to solve the following problem



Now, we found the top view without trying to imagine the part but, Is this the case all the time? The answer is *No.* This is a special model without any hidden lines in the top view.

The main disadvantage of using this rule is:

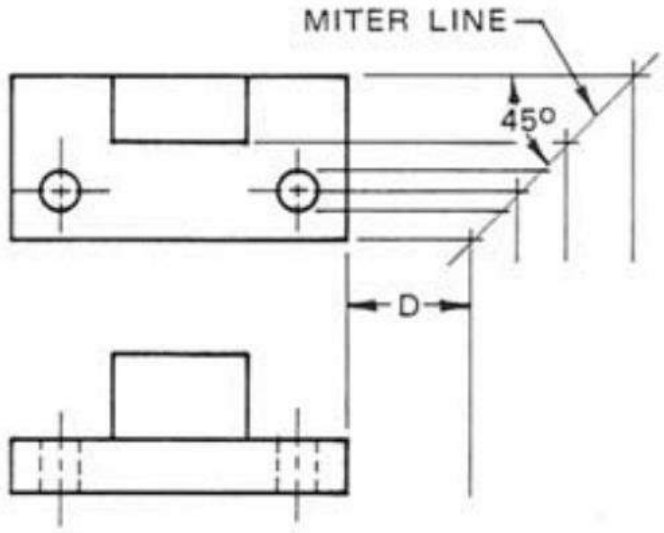
- Will not give the decision about the type of lines in the top, hidden or solid? It is your decision.
- Not valid for curved surfaces, you need to imagine.

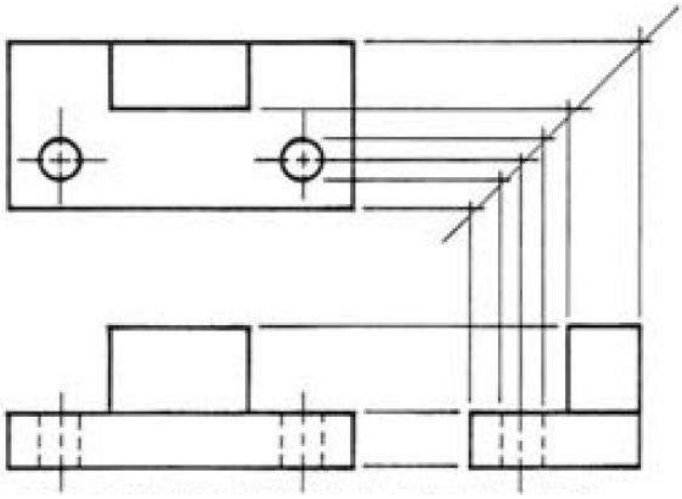
As a conclusion, we have a good tool in hand but we need to imagine also.

Solved Problems using *Milter line*

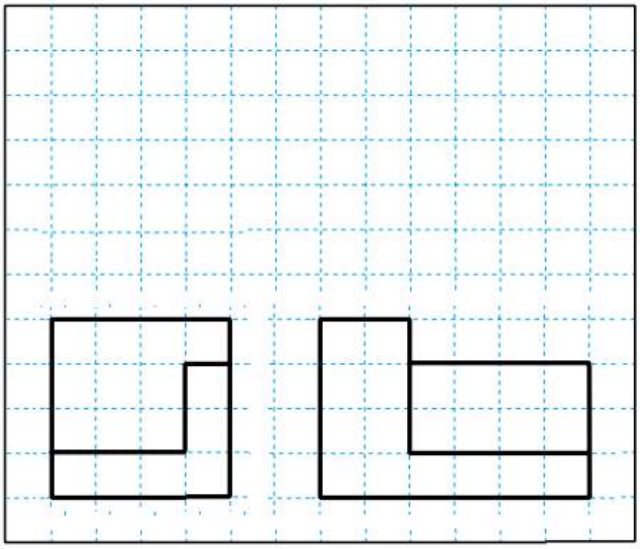
Using a milter line to construct the side view, given the front and top views.

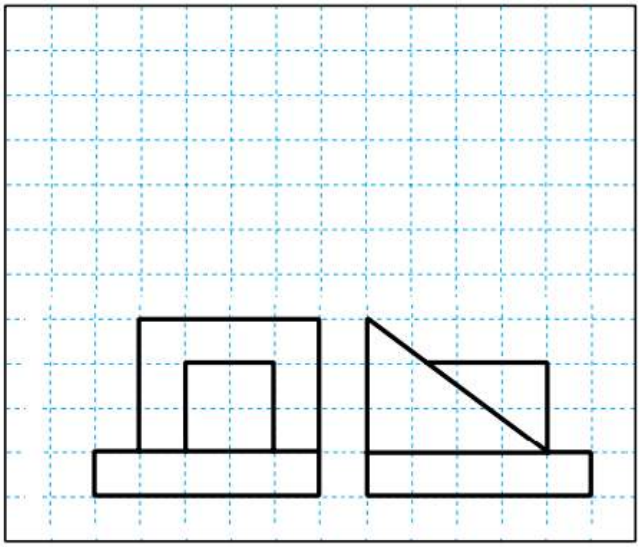
- Given the front and top views, project lines to the right of the top view.
- Establish how far from the front view the side view is to be drawn (distance D).
- Construct the milter line at 45° to the horizontal line.
- Where the horizontal projection lines of the top view intersect the milter line, drop vertical projection lines.
- Project horizontal lines to the right of the front view and complete the side view from the intersections.

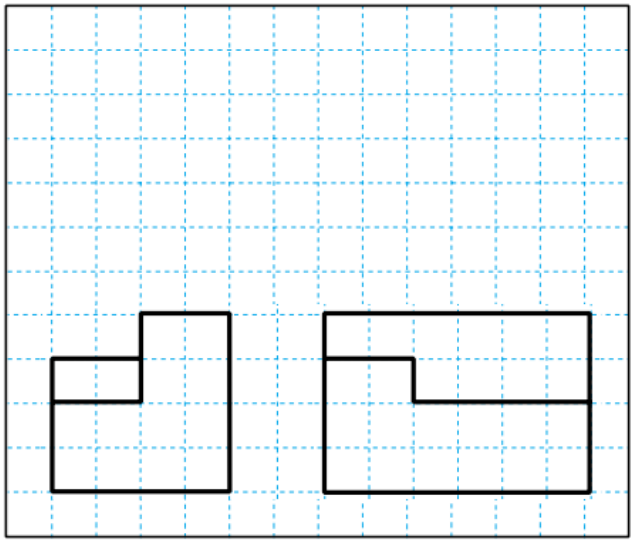


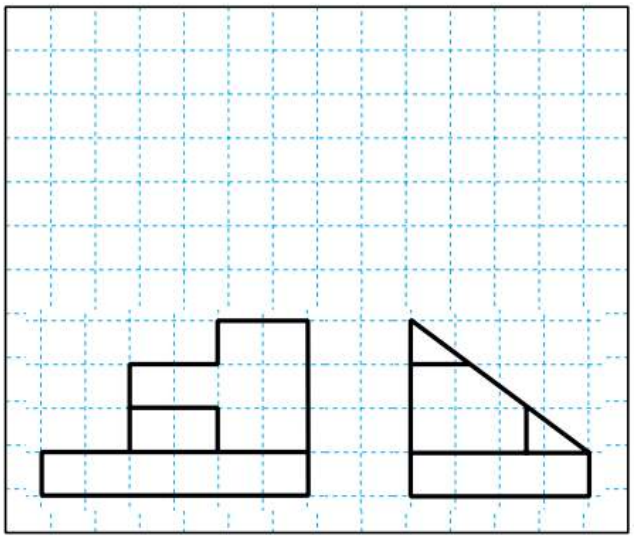


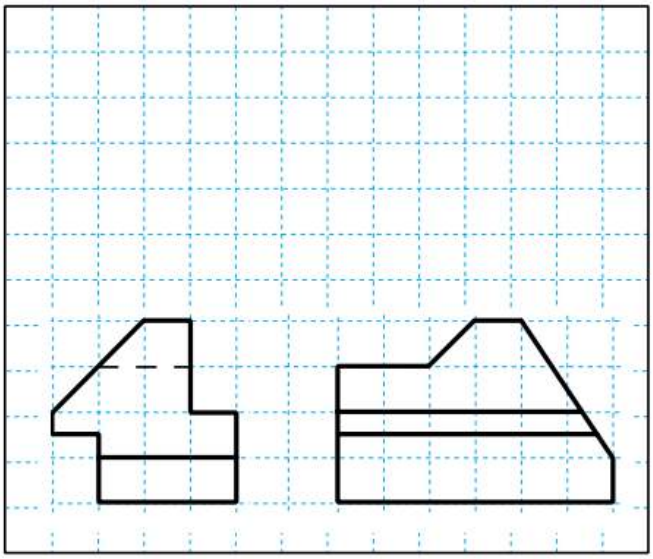
Problems

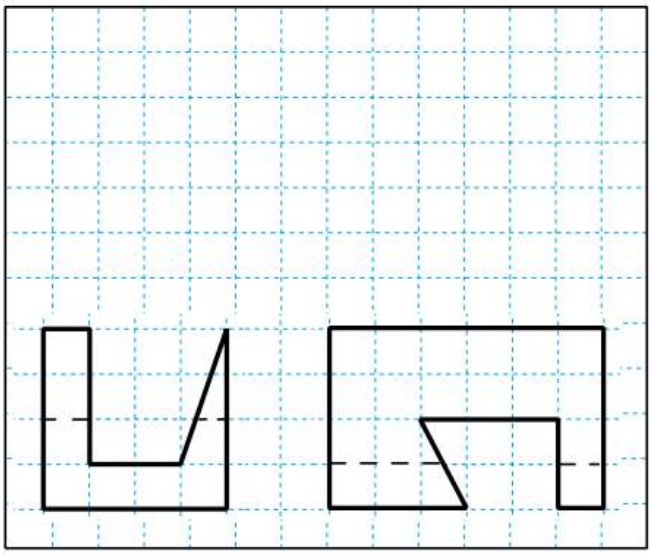


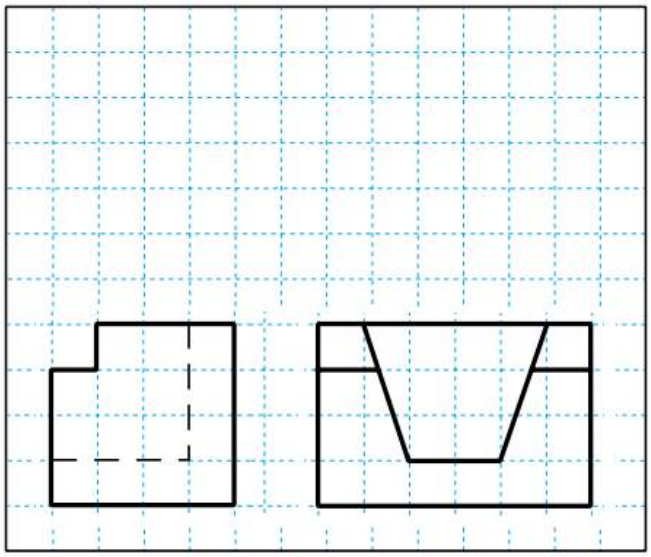


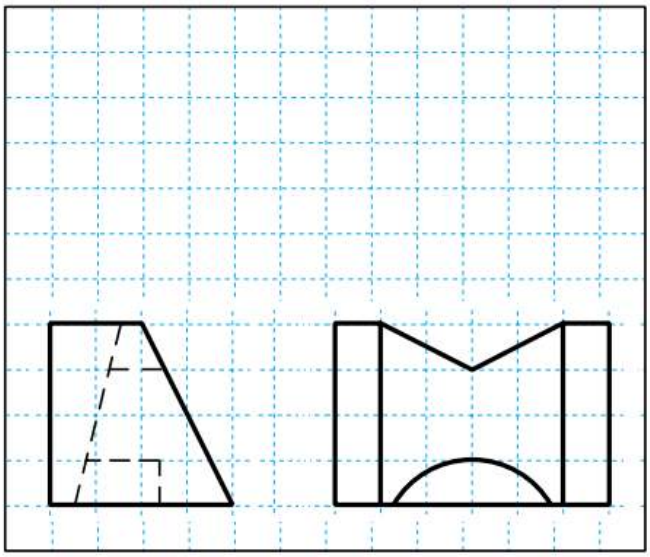


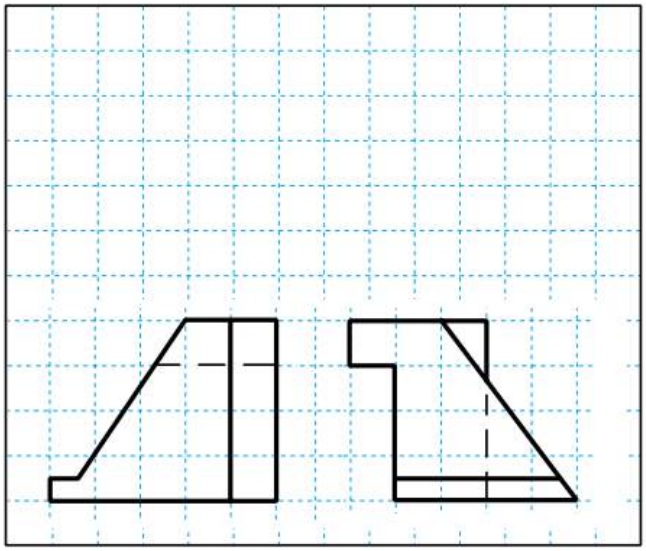


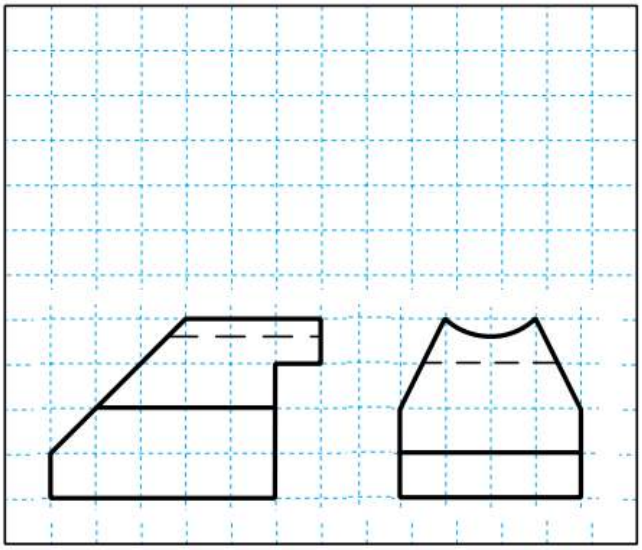


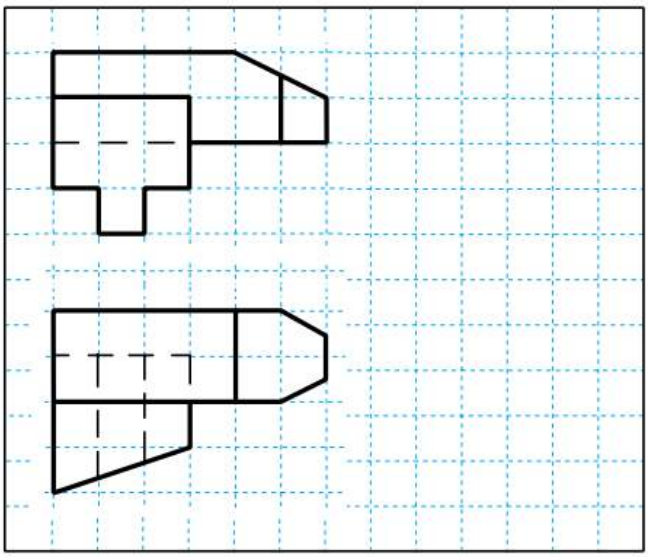


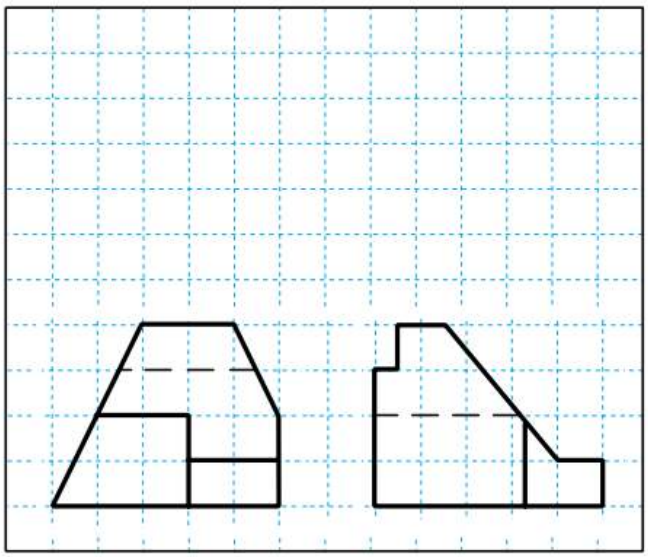


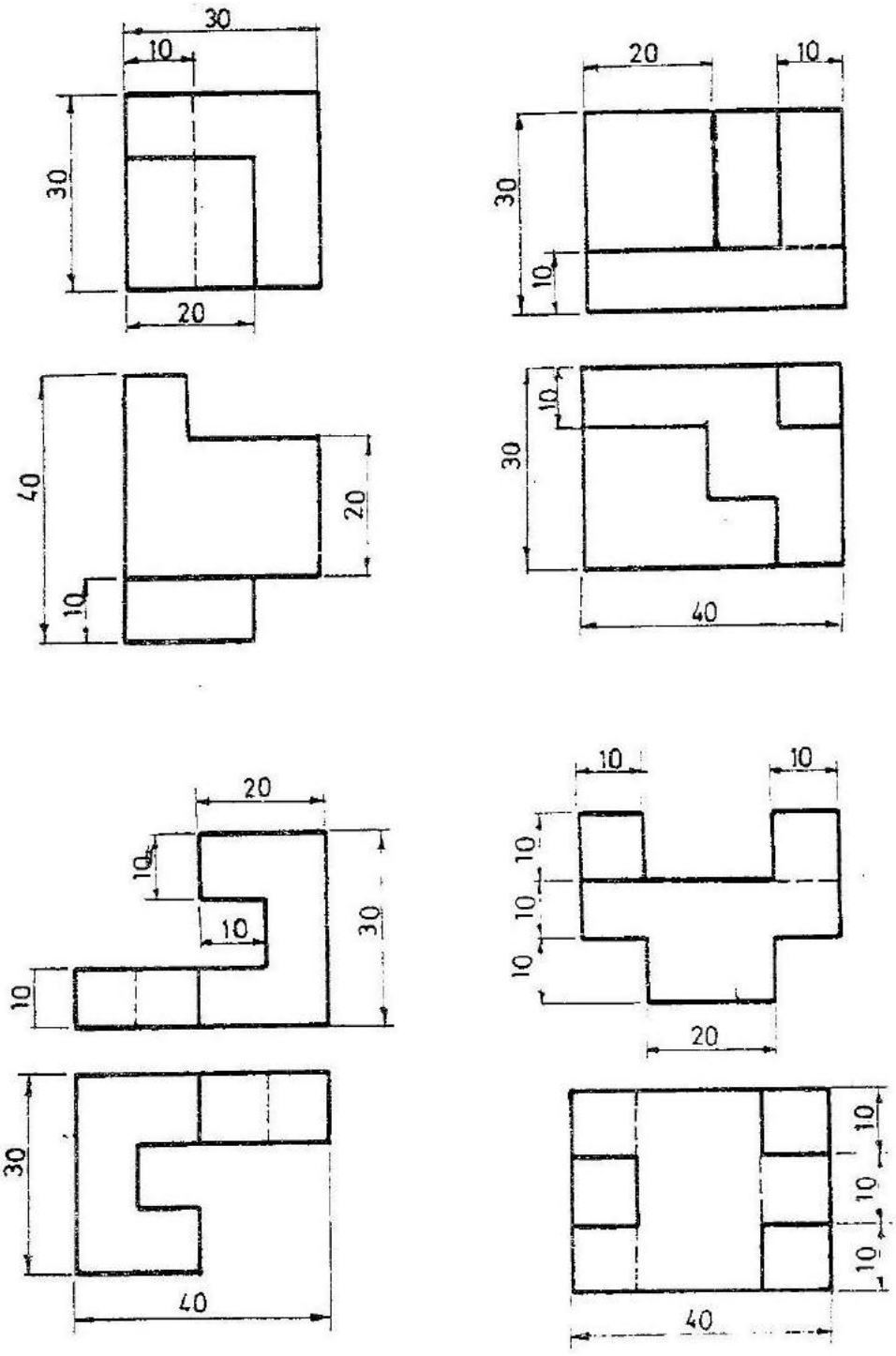








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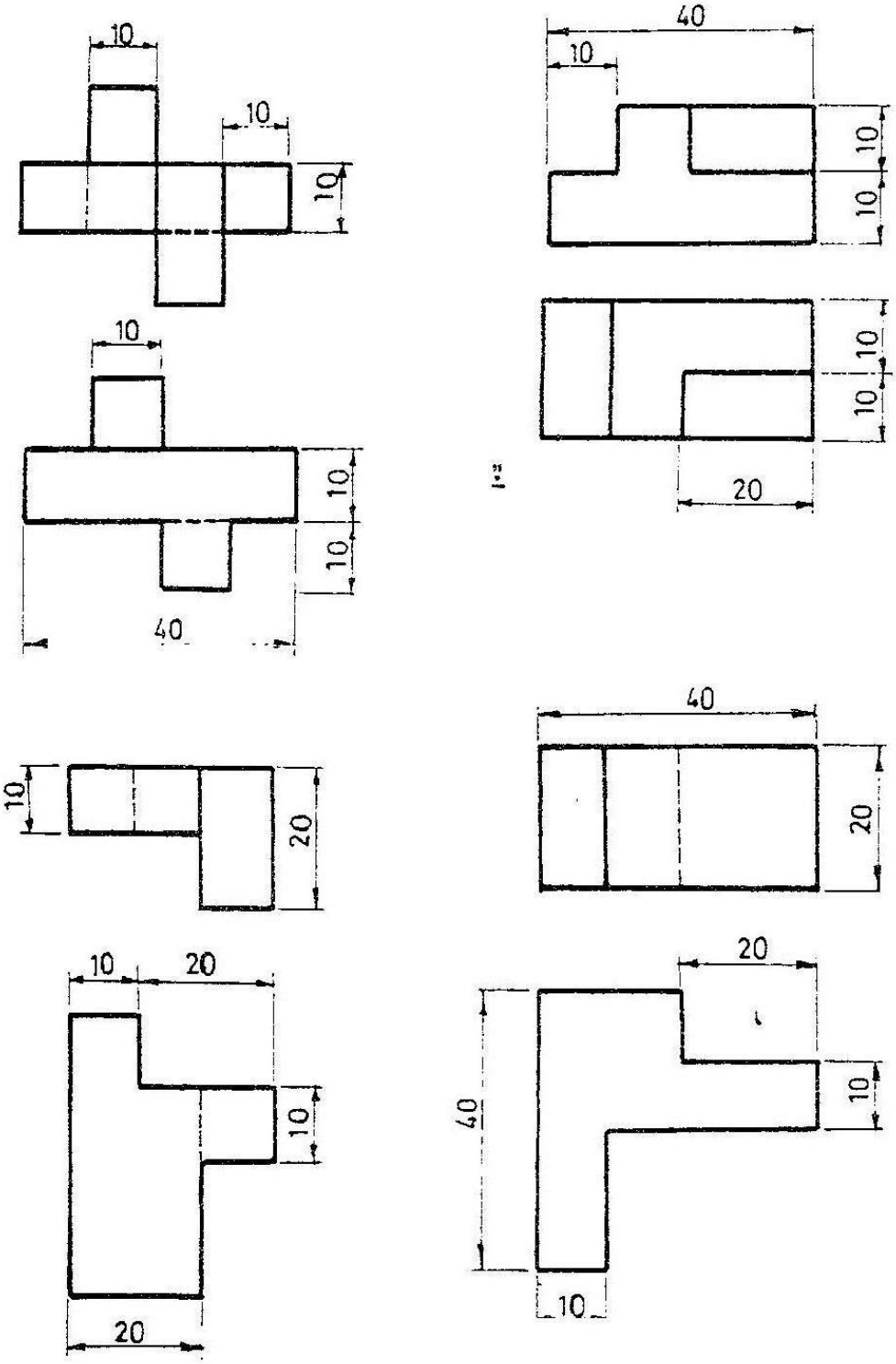
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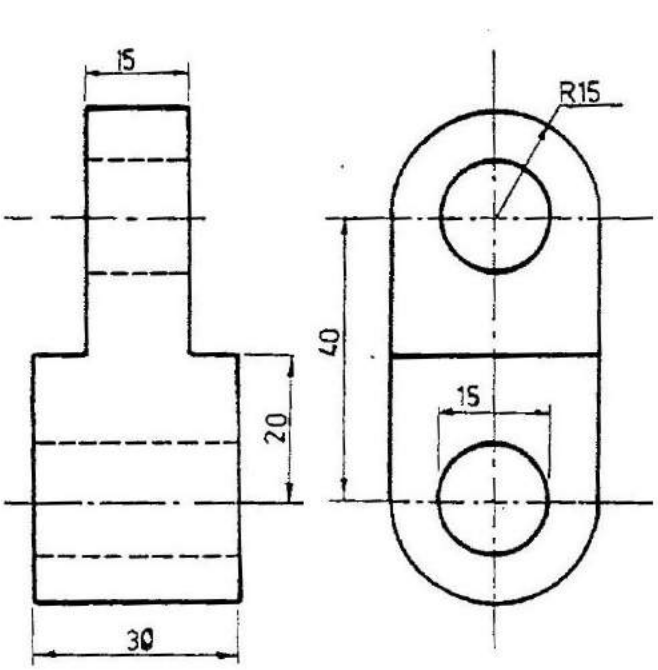
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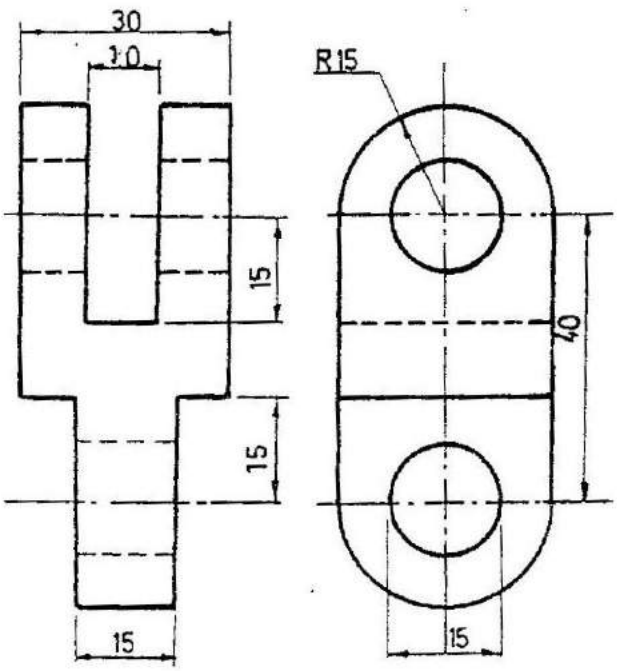
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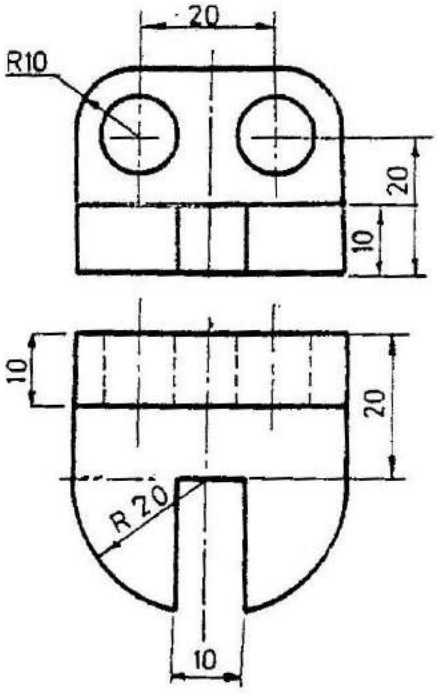
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10
20
R20
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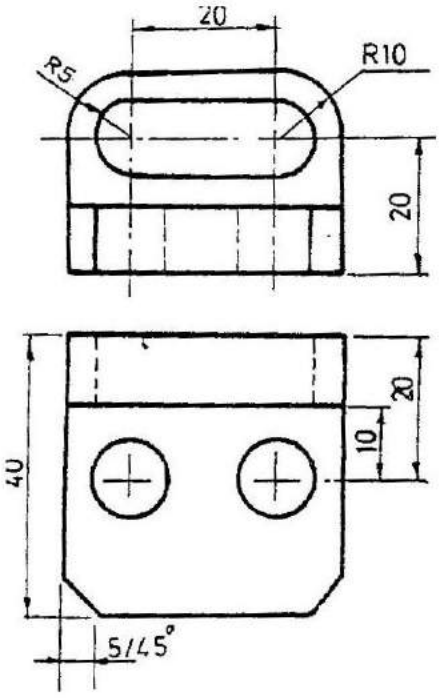
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R10
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40
10
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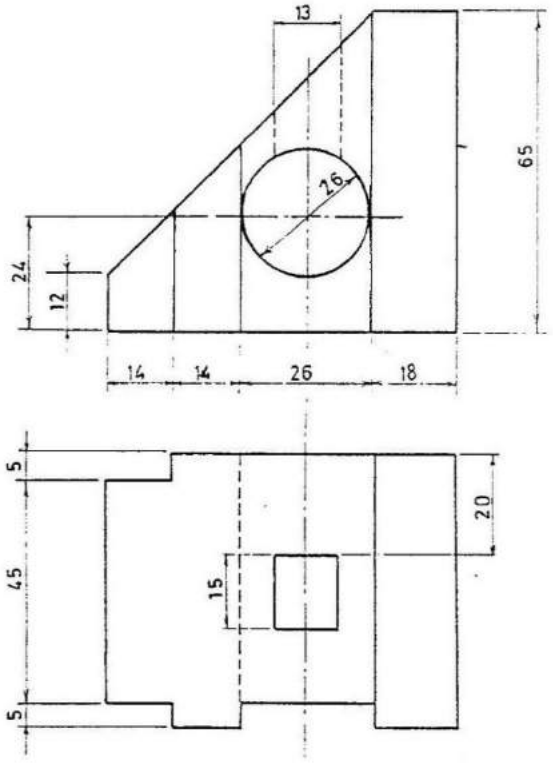
5/45°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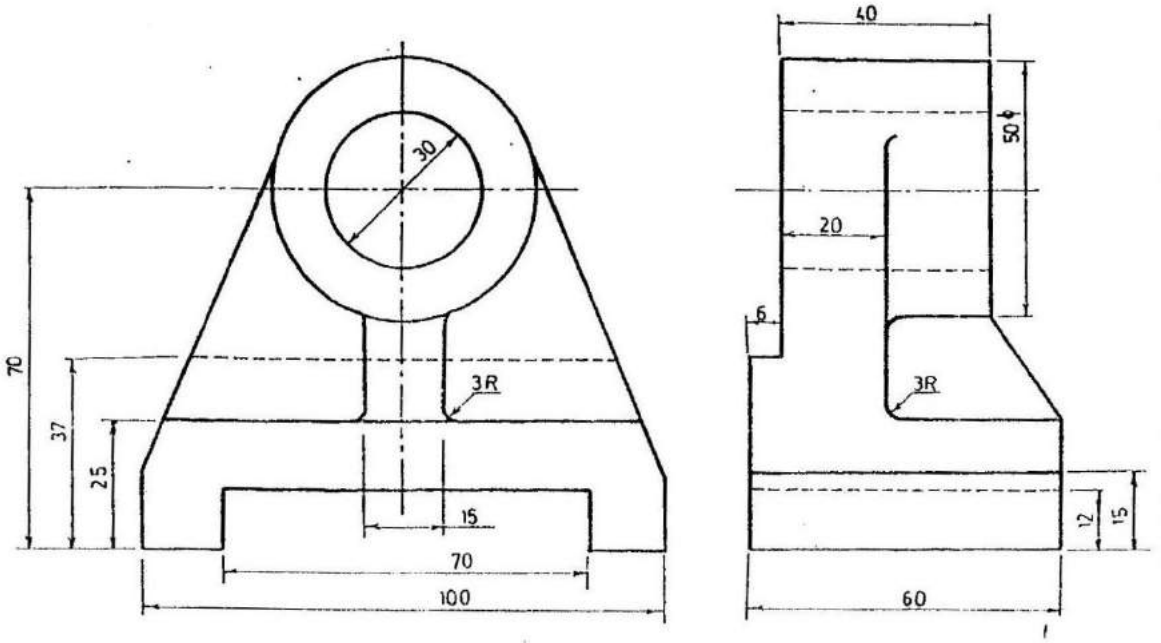
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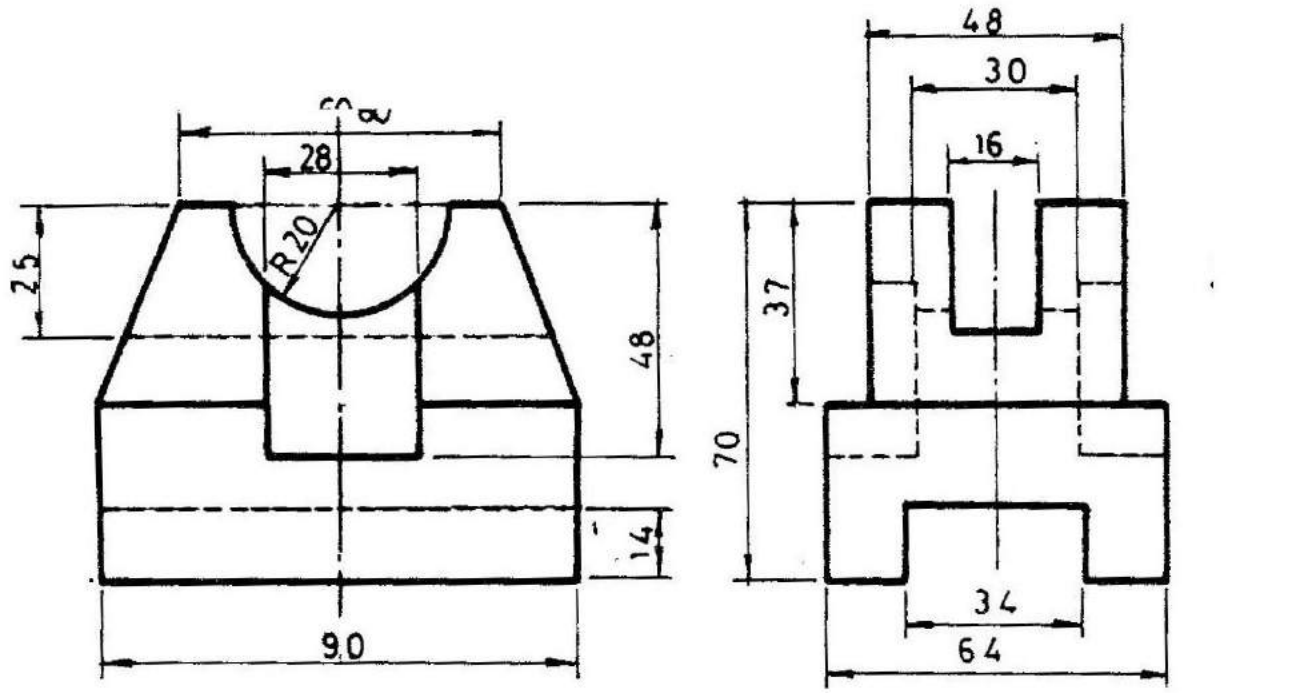
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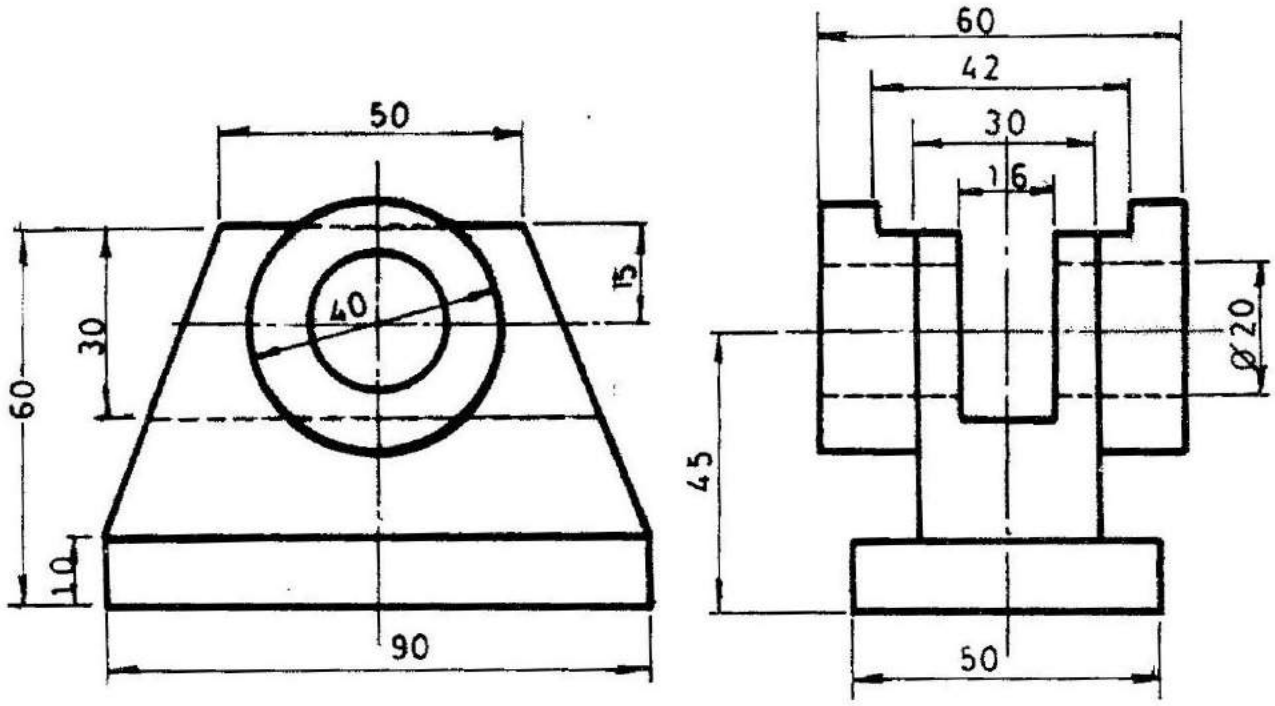
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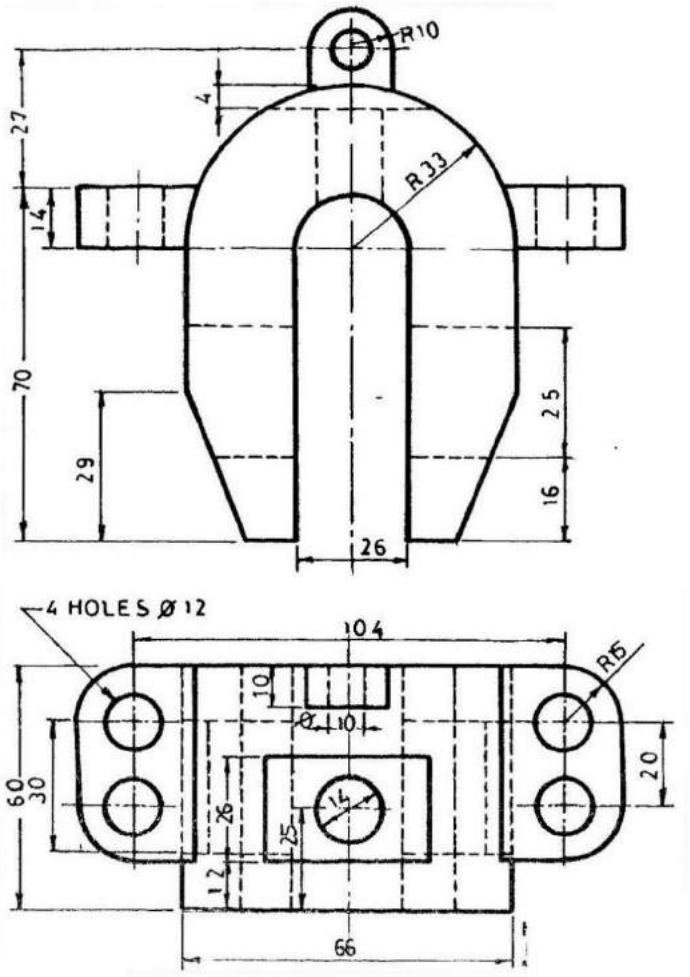
60
42
30
16
Ø20
45
50


R10
4
27
14
R33
70
29
25
16
26
4 HOLES Ø 12
104
R15
10
Ø10
60
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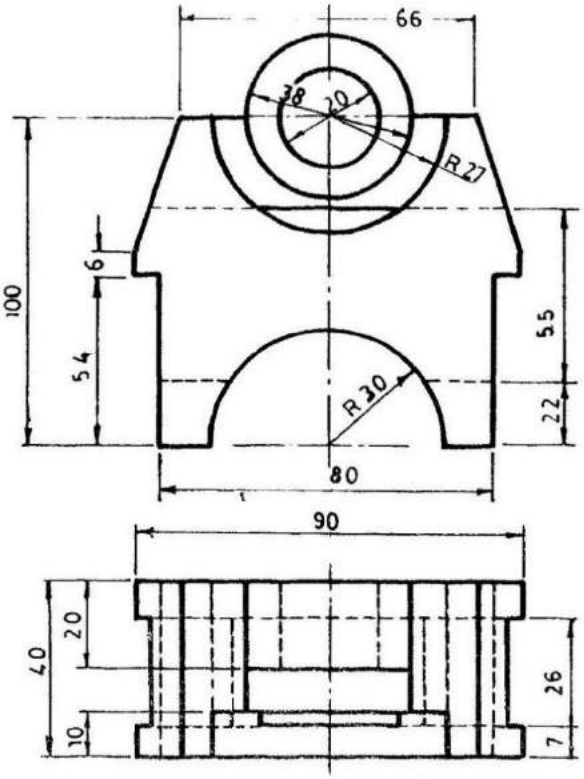
26
25
20
12
66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R27
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6
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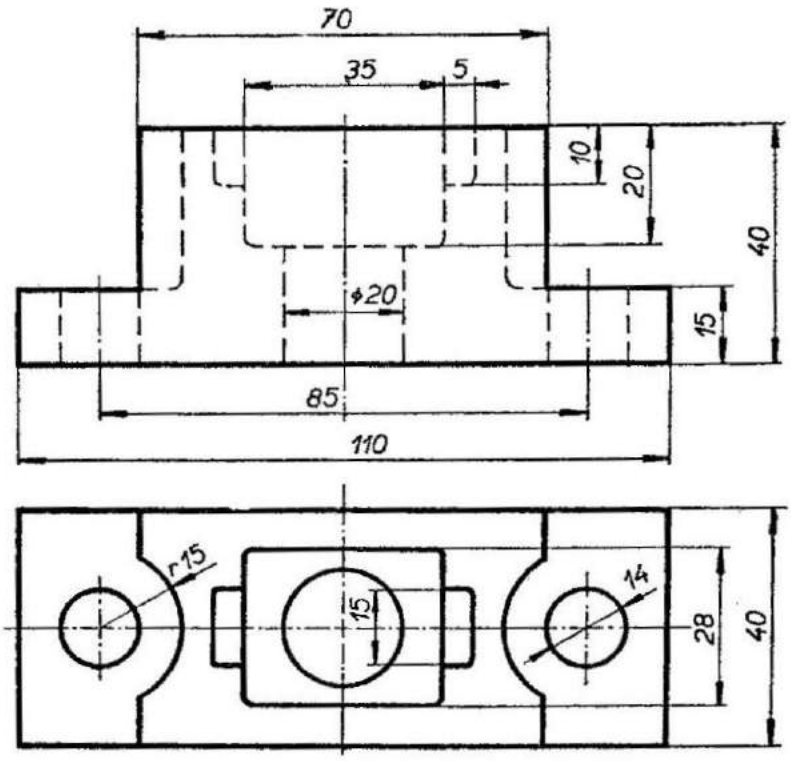
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35
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10
20
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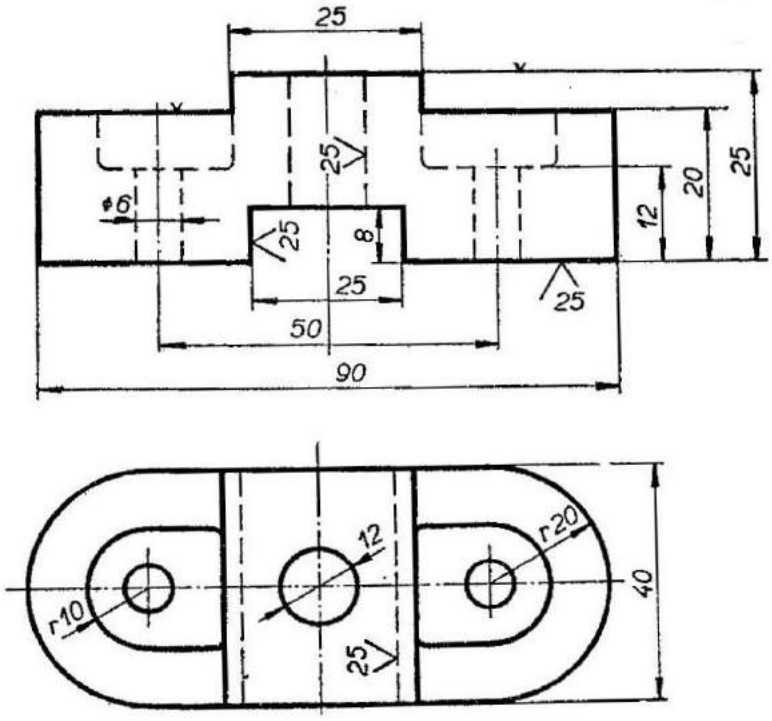
ϕ20
15
85
110
r15
15
14
28
40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ϕ6
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50
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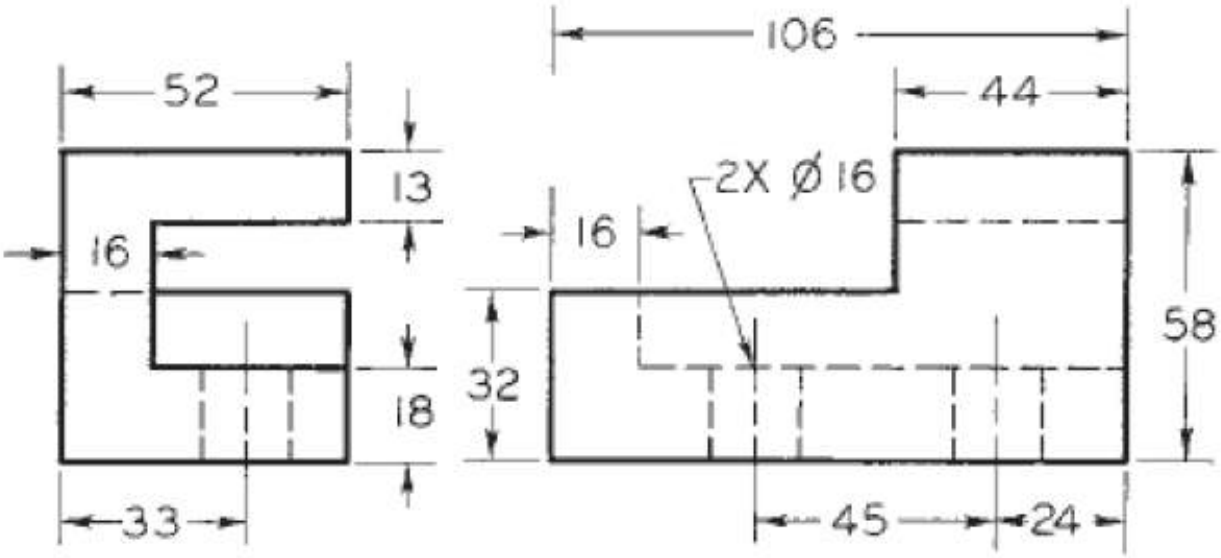
r10
12
r20
40
25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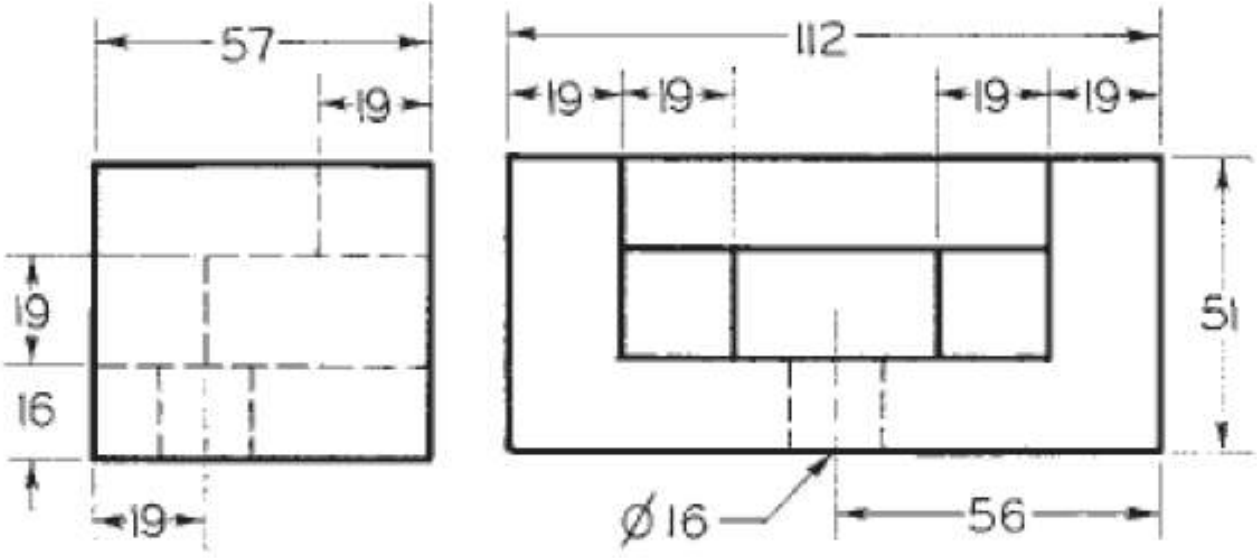
2X Ø 16
16
32
58
45
24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16
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112
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19
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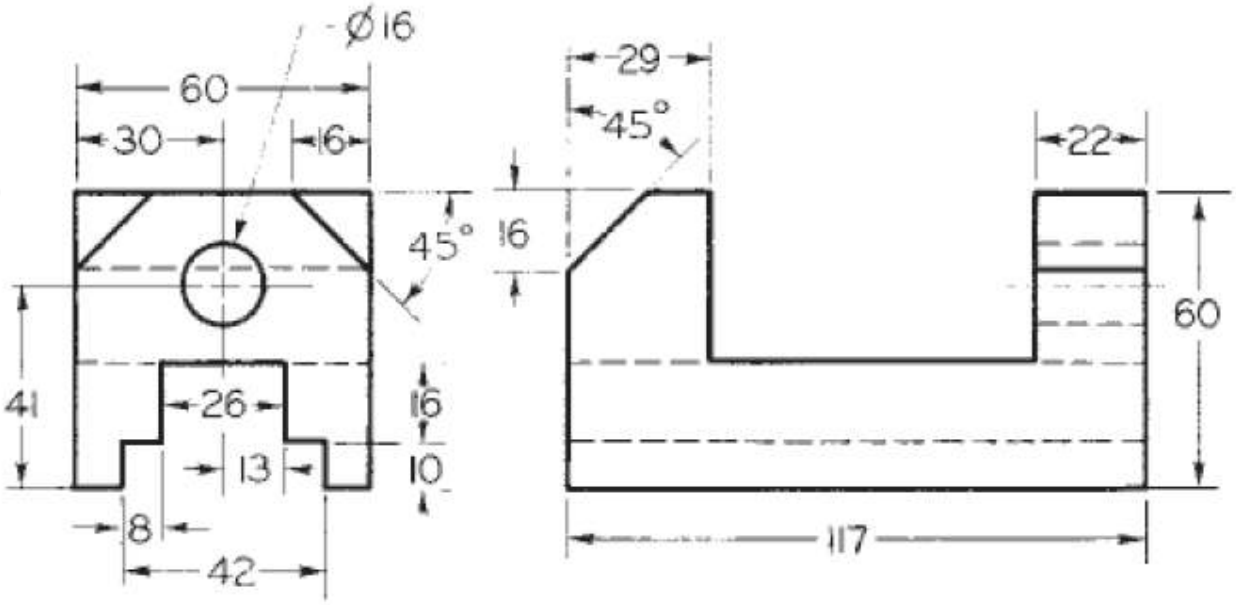
19
51
Ø 16
56


Ø16
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30
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45°
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41
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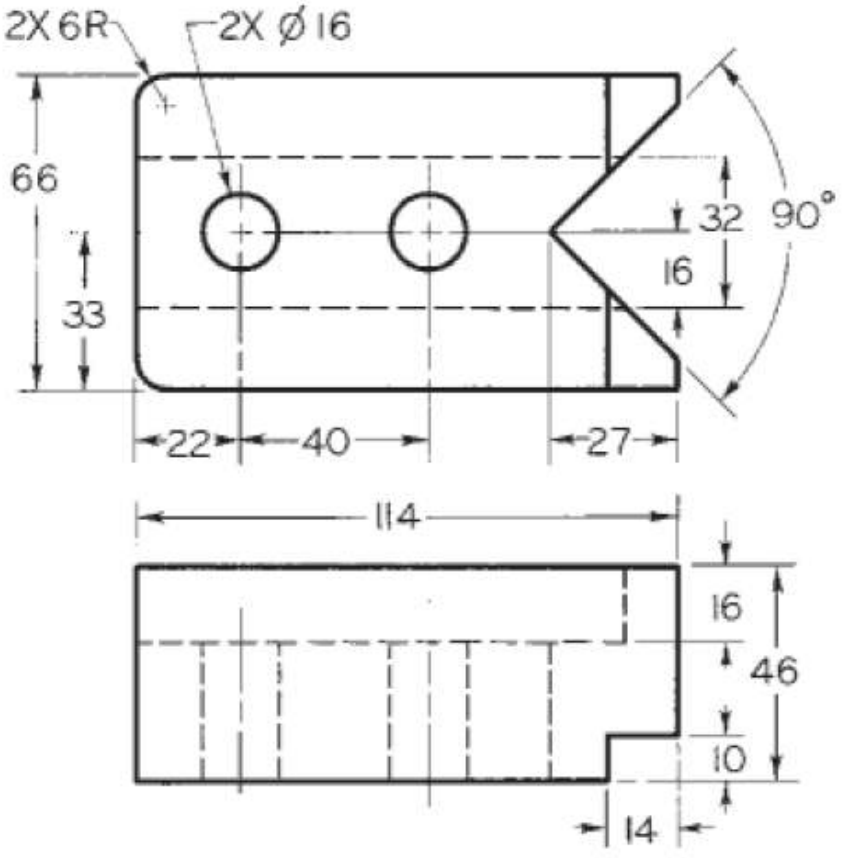
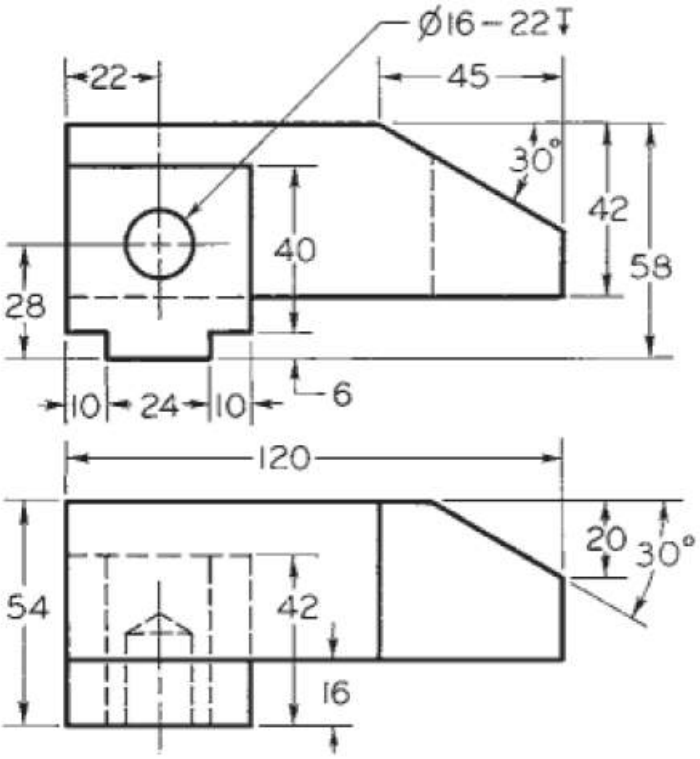
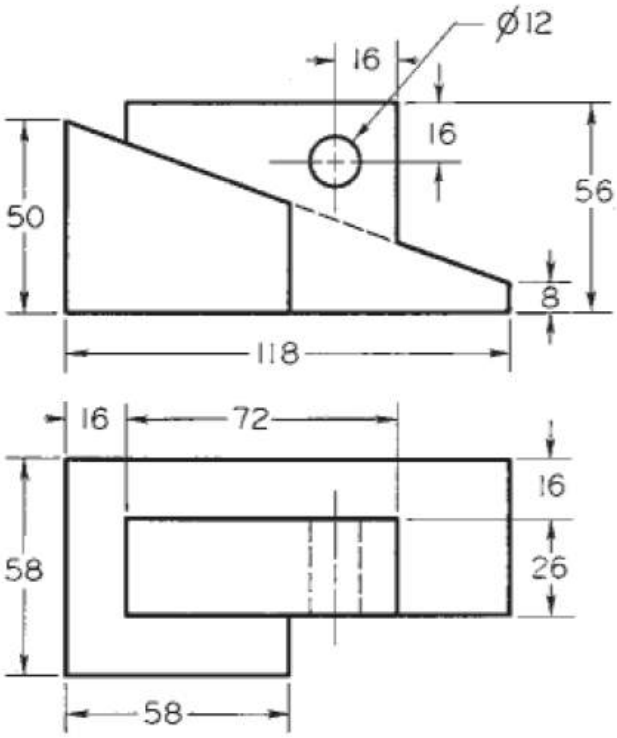
13
16
10
8
42
29
45°
22
60
117


2X 6R
2X Ø16
66
33
32
16
90°
22
40
27
114
16
46
10
14


Ø16 – 22↧
22
45
30°
42
58
40
28
10
24
10
6
120
20
30°
54
42
16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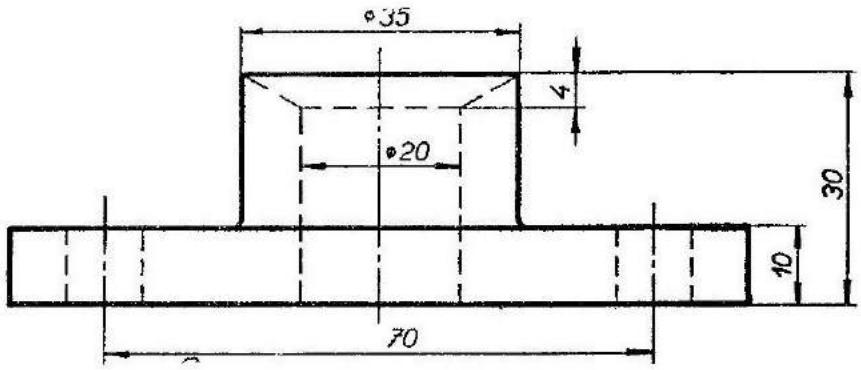
26
58


⌀35
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⌀20
30
10
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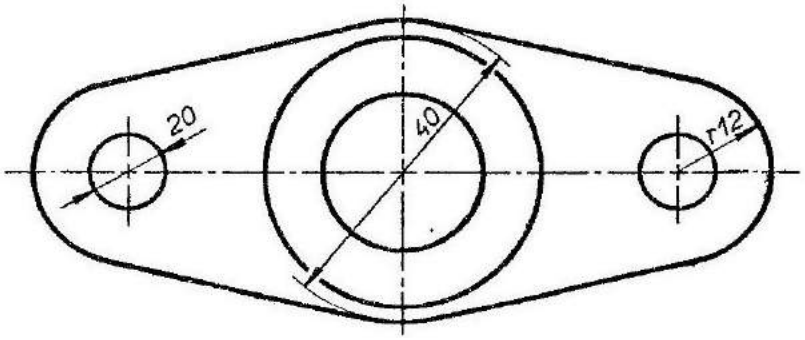
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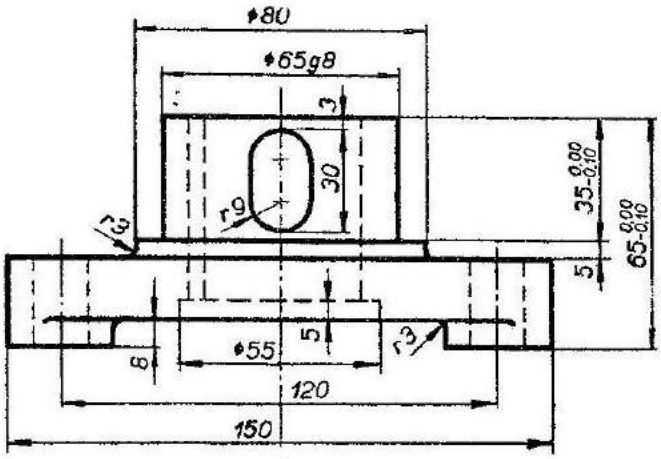
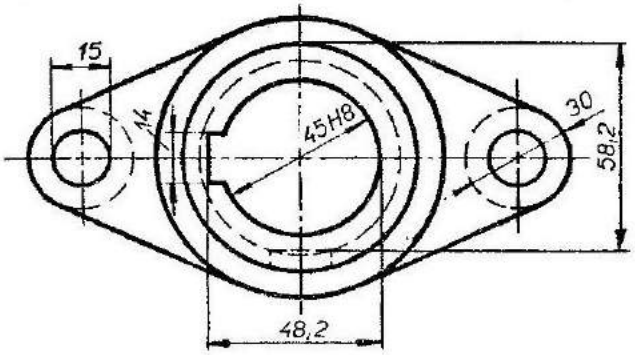
r12


⌀80
⌀65g8
3
30
r9
35 0 -0.10
65 0 -0.10
3
5
⌀55
5
r3
8
120
150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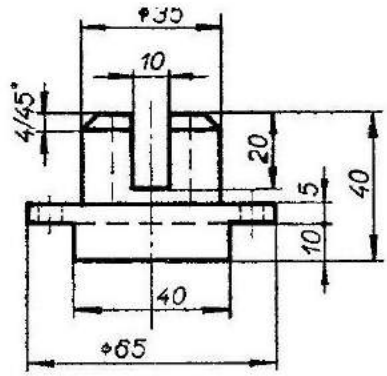
48,2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10
4/45°
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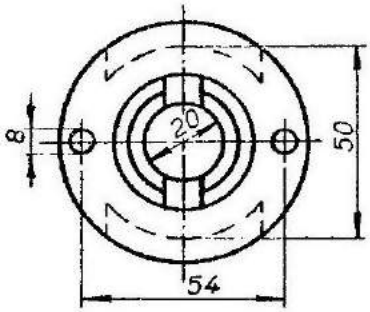
⌀65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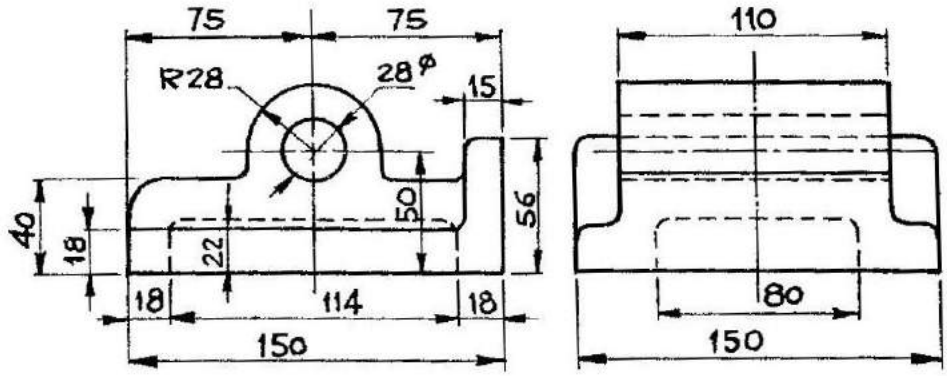
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75
R28
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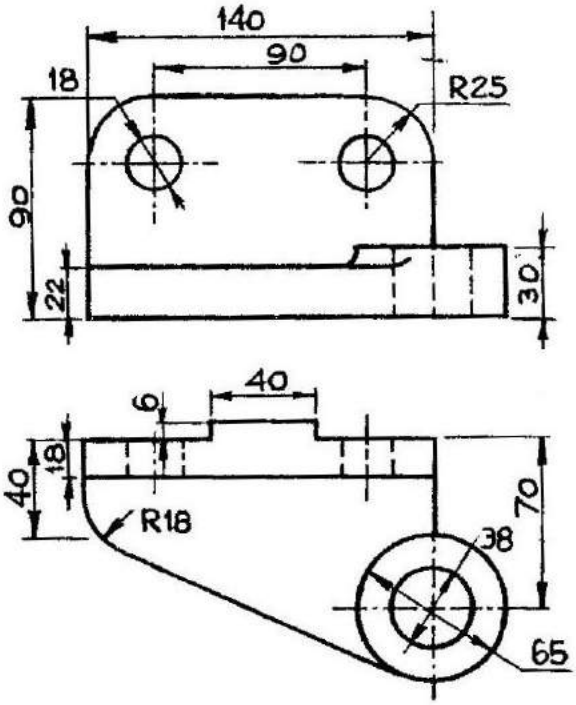
140
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R25
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6
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R18
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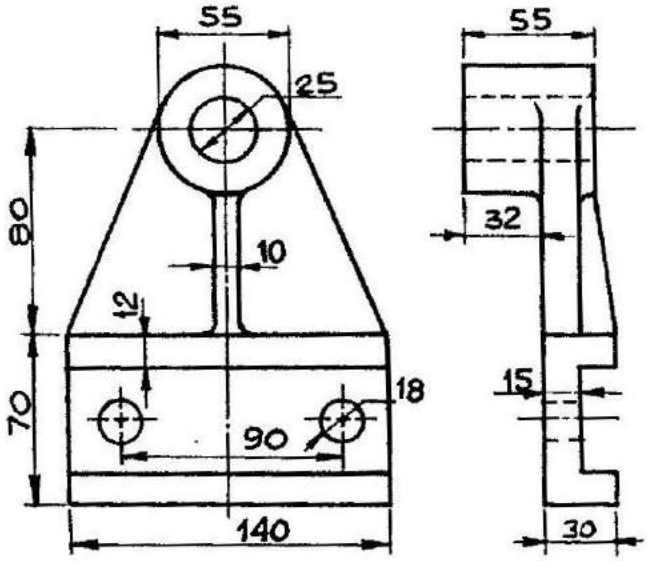
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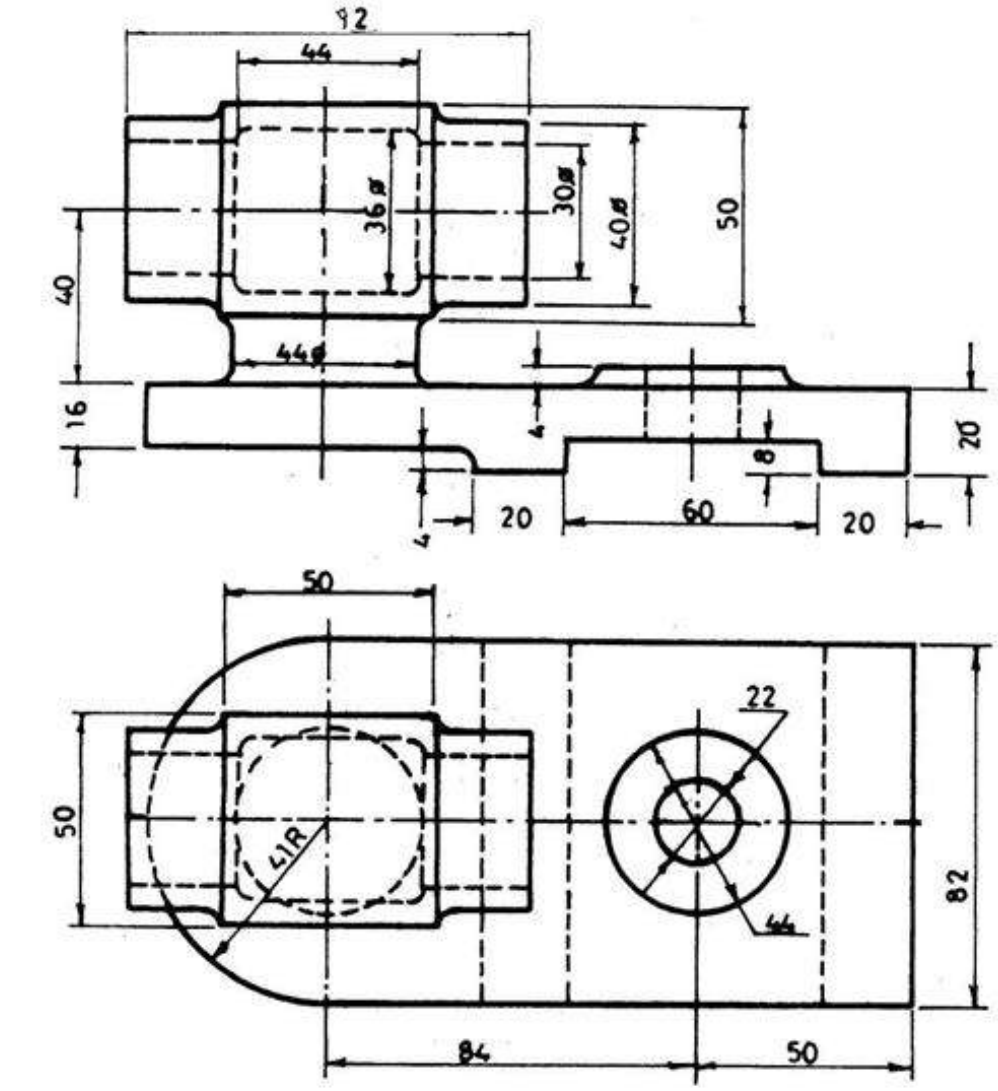
92
44
36ø
30ø
40ø
50
40
44ø
16
4
8
20
4
20
60
20
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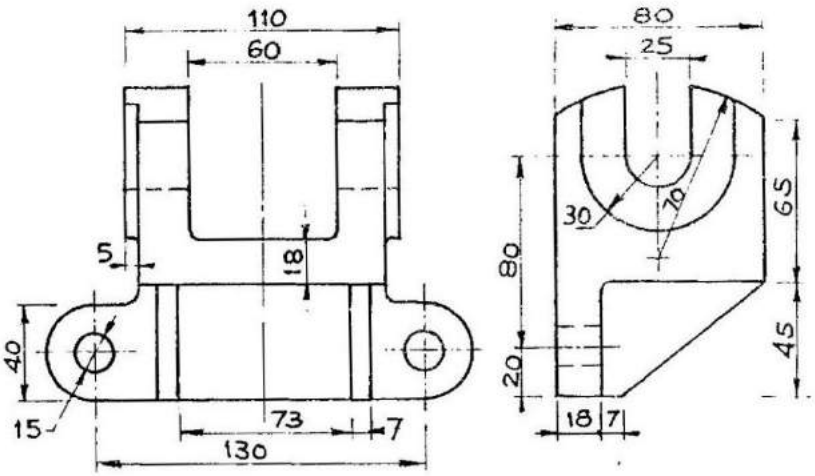
50
22
41R
44
82
84
50

FILLETS 2mm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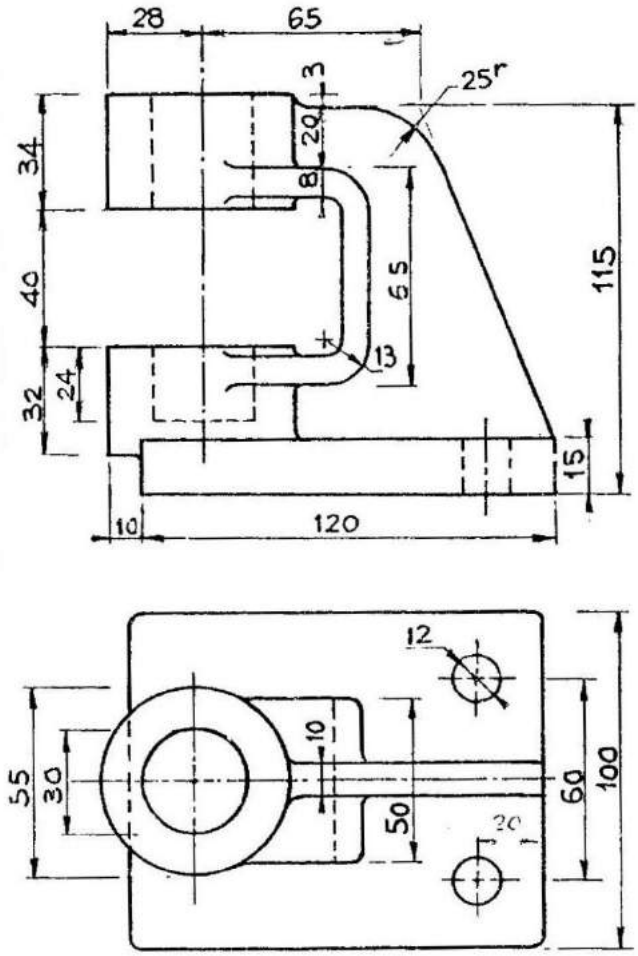
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65
3
25r
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100
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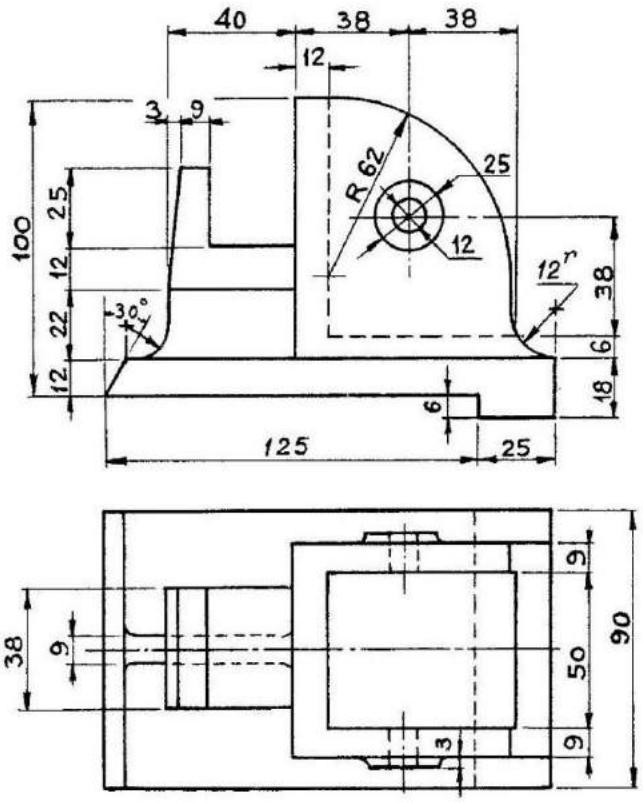
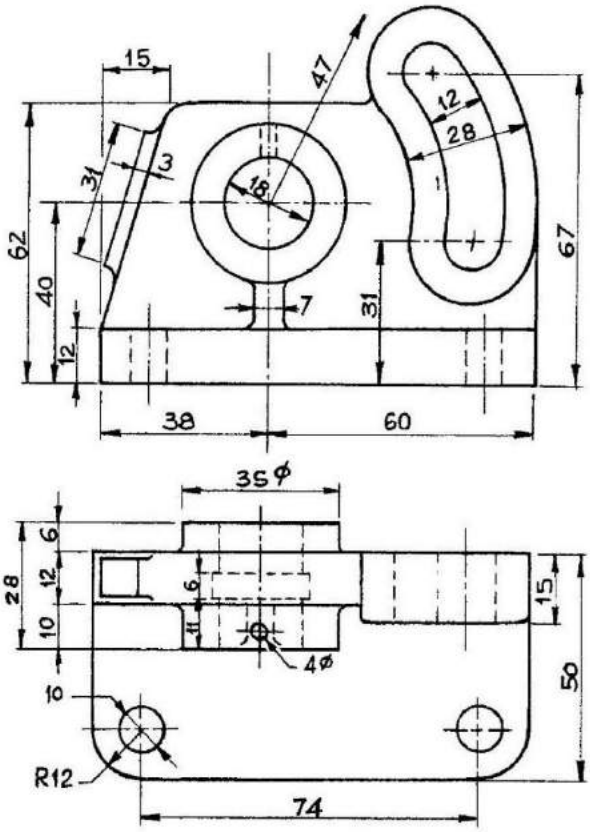

40
38
38
12
3
9
R62
25
12
12
38
6
18
100
25
12
22
12
30°
6
125
25
38
9
90
50
9
3


15
47
12
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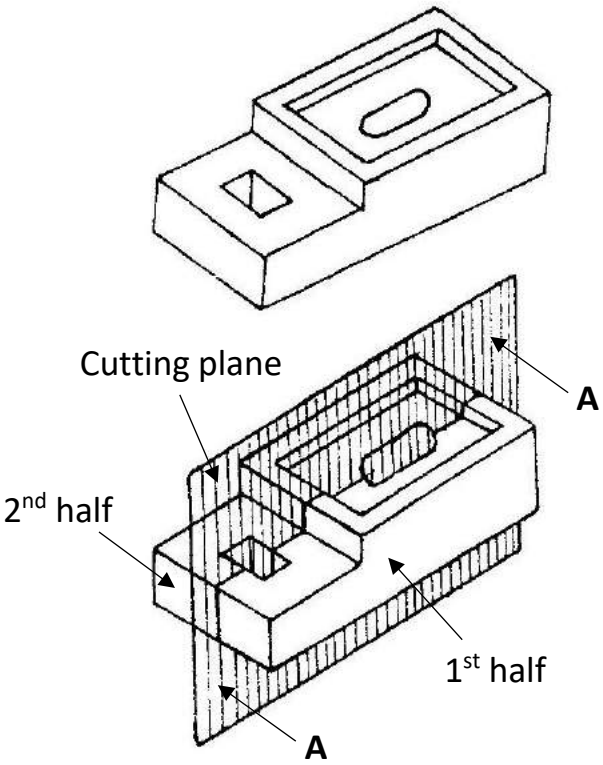
31
13
16
62
40
12
7
31
67
38
60
35ϕ
6
28
12
10
6
11
4ϕ
15
50
10
R12
74

The concept of sectioning is cutting the model with a cutting plane, see figure 1, and see what we have inside. After cutting the model a direction of looking is required, it is represented by A-A in the figure. In this case we remove the first part and we are looking to the second one. The cutting plane is shown in the top and side views while the effect (hatched view) is shown in the front view.

The Sectional views are also called sections, and the process of creating sections is referred to as sectioning. Sections are used to describe the interior portions of an object that are otherwise difficult to visualize. Interior features that are described using hidden lines are not as clear as if they are exposed for viewing as visible features. It is also a poor practice to dimension the hidden features. The sectional view allows you to expose the hidden features for dimensioning.



<u>There are 2 essential rules in sectioning</u>:

- No hidden line is section views.
- Do not hatch the web.

But how to identify the web? It is a feature in the part having: ***large surface area*** with ***small thickness without any internal details***.

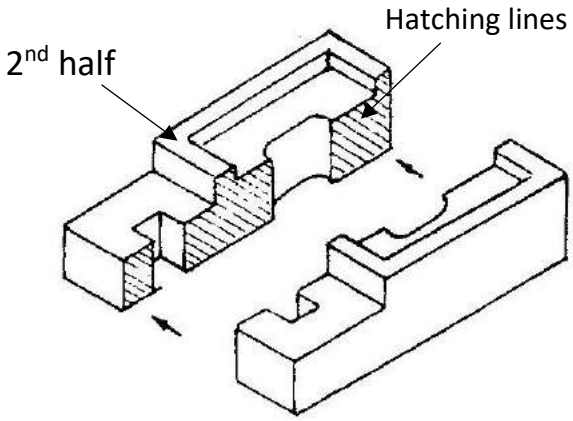


Figure 1

Section views are used for three main purposes:

- To document the design and manufacture of single parts that are manufactured as one piece.
- To document how multiple parts are to be assembled or built.
- To aid in visualizing the internal workings of a design.

CUTTING-PLANE LINES AND SECTIONAL VIEW IDENTIFICATION

The sectional view is created by placing an imaginary cutting plane through the object that cuts away the area to be exposed. The adjacent view becomes the sectional view by removing the portion of the object between the viewer and the cutting plan (see Figure 2).

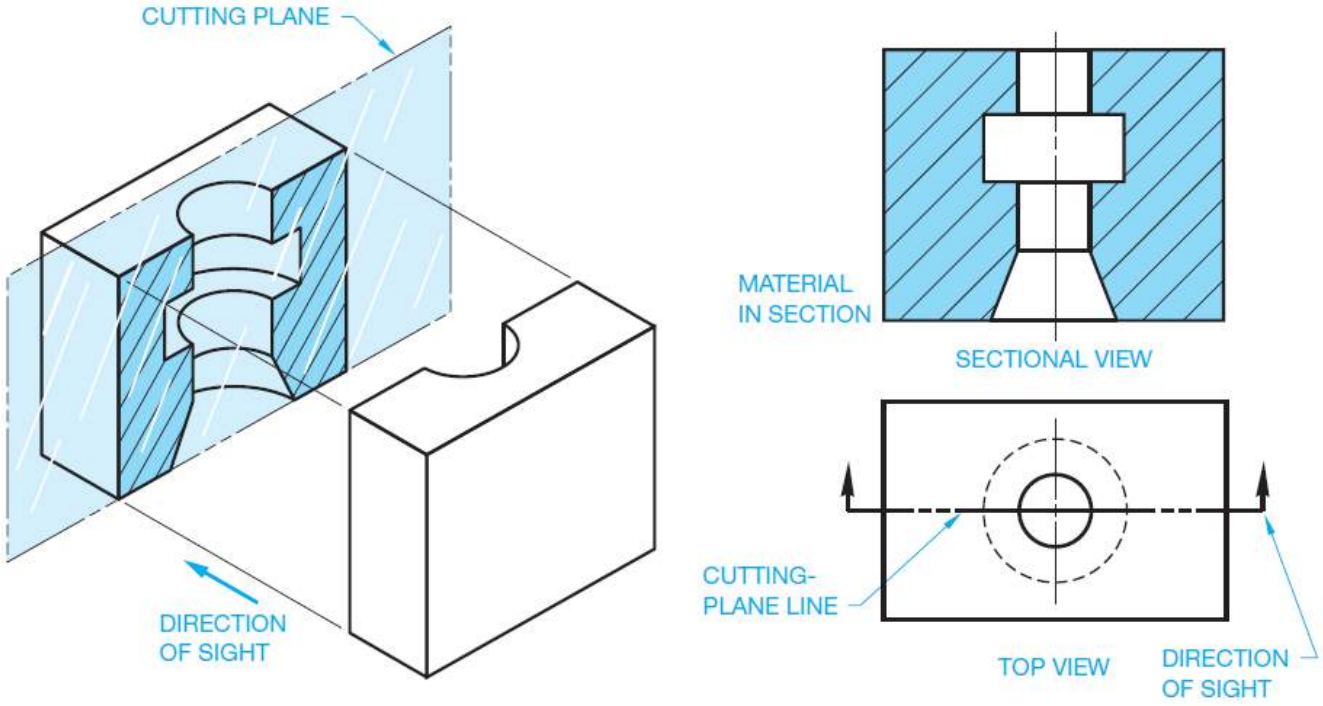


Figure 2 Cutting-plane and sectional view visualization.

The ***cutting-plane line*** is a thick line representing the cutting plane as shown in Figure 3. The cutting-plane line can be drawn using alternating long and two short dashes, or evenly spaced dashes. The long dashes can vary in length depending on the size of the drawing, but the short dashes are generally .25 in. (6 mm) in length. Figure 12.4 shows an example of the cutting plane line styles. The cutting-plane line is capped on the ends, with arrowheads showing the direction of sight of the sectional view.

The cutting-plane line arrowheads maintain the same 3:1 length-to-width ratio as dimension line arrowheads. Cutting plane line arrowheads are generally twice the size of dimension line arrowheads, so they show up better on the drawing. If the dimension line arrowheads are .125 (3 mm) long on your drawing, then make the cutting-plane line arrowheads .25 in. (6 mm) long. This depends on the size of the drawing and your school or company standards.

When the extent of the cutting plane is obvious, only the ends of the cutting-plane line can be used as shown in Figure 4. Such treatment of the cutting plane also helps keep the view clear of excess lines.

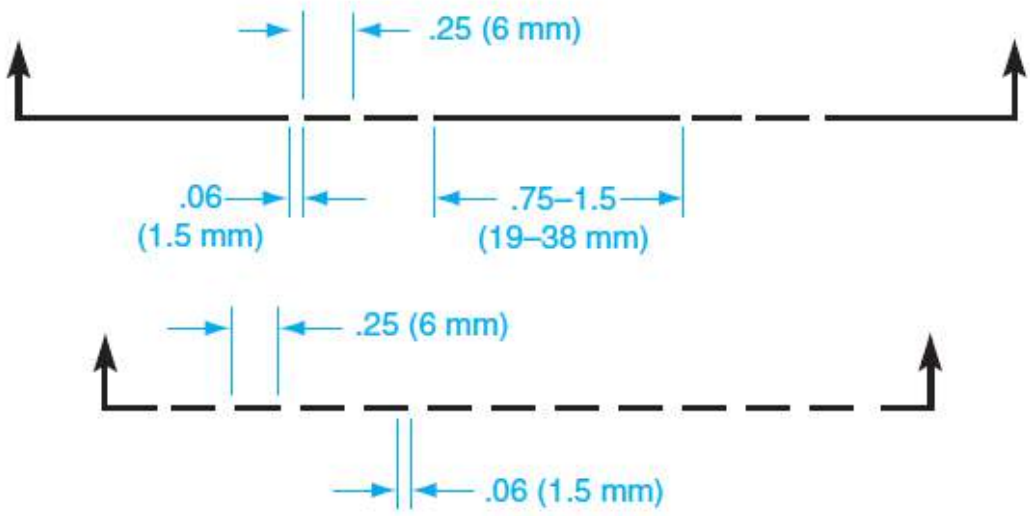


Figure 3 Cutting-Plane Line Options

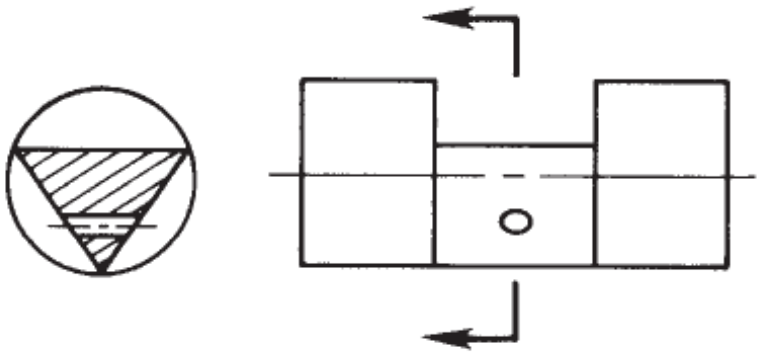


Figure 4 Simplified cutting-plane line showing only the ends of the cutting-plane line.

The sectional view should be projected from and perpendicular to the cutting-plane line and placed as one of the standard principal multiviews. If lack of space restricts the normal placement of a sectional view, the view can be placed in an alternate location. When this is done, the sectional view should not be rotated but should remain in the same orientation as if it is a direct projection from the cutting plane. The cutting planes and related sectional views should be labeled with letters beginning with AA as shown in Figure 5. The cutting-plane line labels should be placed near the arrowheads. When the cutting plane line has labels A on each end, then the sectional view has the related title SECTION A-A placed below the sectional view.

When there is more than one sectional view on a drawing, the additional cutting-plane lines and views are labeled BB, CC, and so on.

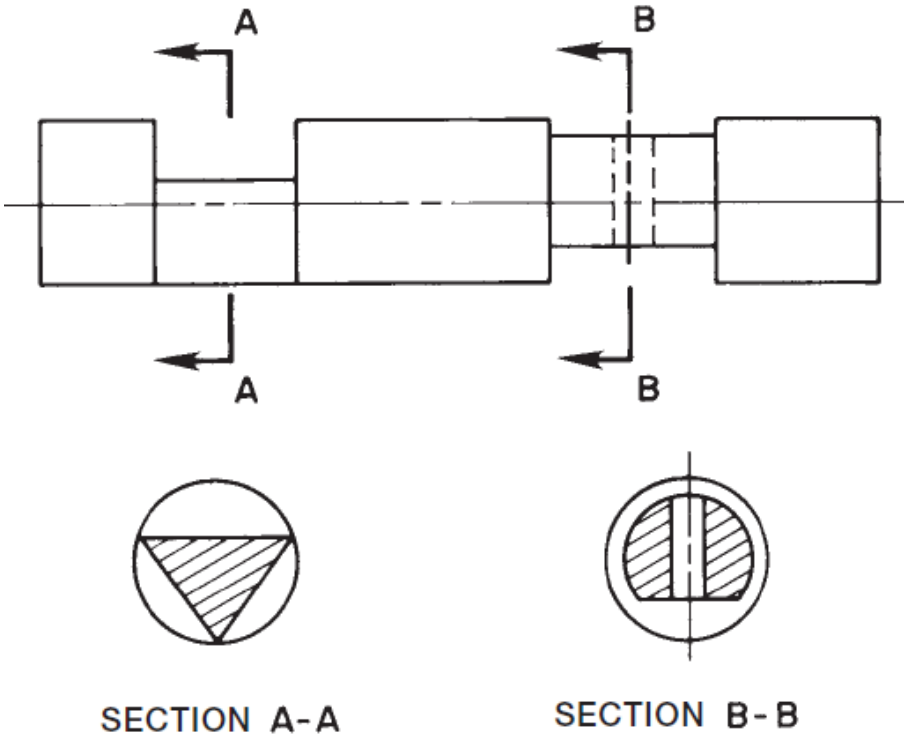


Figure 5 Labeled cutting-plane lines and related sectional view.

*The cutting-plane line **can** be omitted when the location of the cutting plane is obvious as shown in Figure 6*

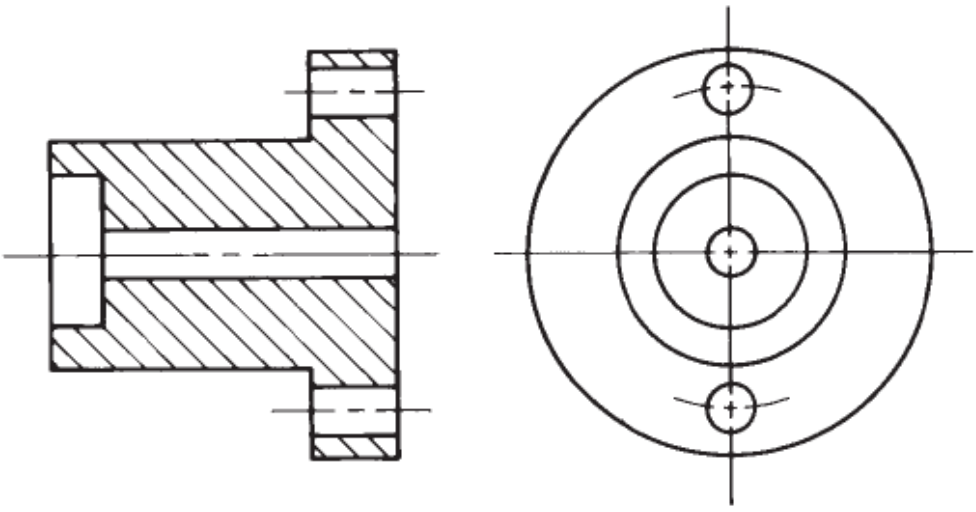


Figure 6 An obvious cutting-plane line can be omitted.

SECTION LINES (SECTION-LINNING)

Section lines are thin lines used in the view of the section to show where the cutting-plane line has cut through material (see Figure 7). Section lines are usually drawn equally spaced at 45° but cannot be parallel or perpendicular to any line of the object. Any convenient angle can be used to avoid placing section lines parallel or perpendicular to other lines of the object.

Angles of 30 ° and 60 ° are common. Section lines that are more than 75 ° or less than 15 ° from horizontal should be avoided. Section lines must never be drawn horizontally or vertically. Figure 8 shows some common errors in drawing section lines. Section lines should be drawn in opposite directions on adjacent parts; when several parts are adjacent, any suitable angle can be used to make the parts appear clearly separate.

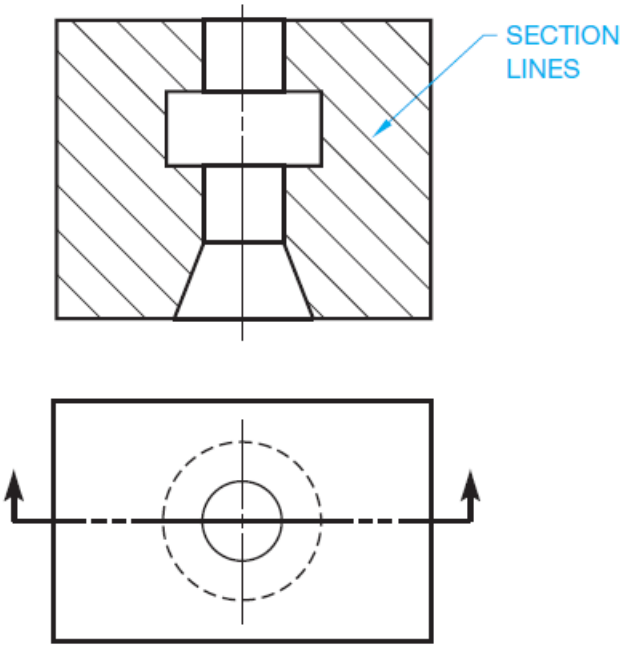


Figure 7 Section lines represent the material being cut by the cutting plane.

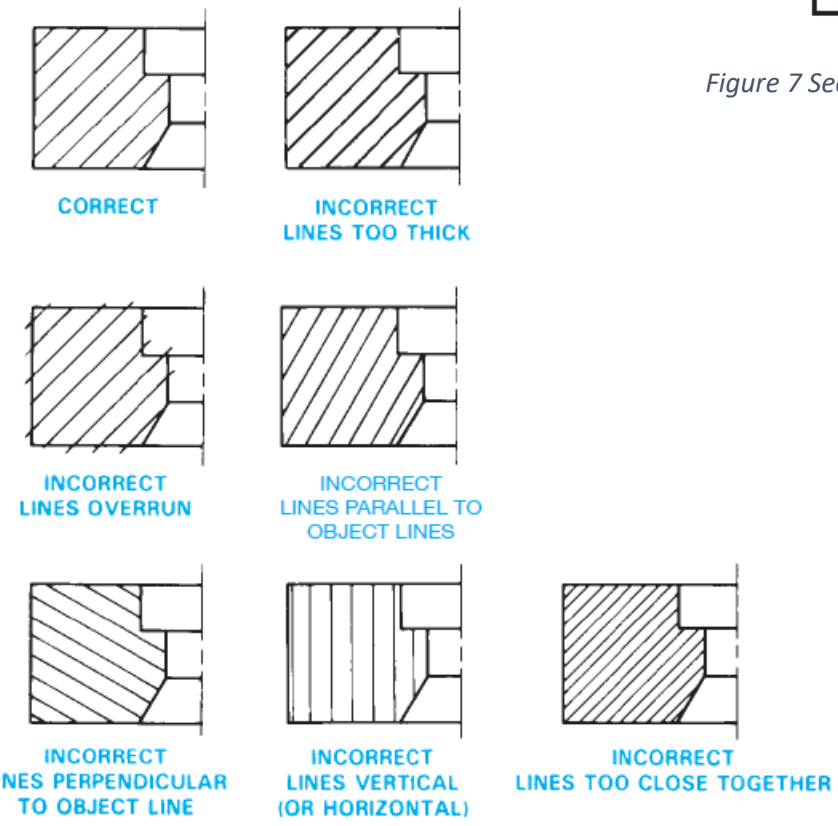


Figure 8 Common section-line errors.

FULL SECTION

A full section is drawn when the cutting plane extends completely through the object, usually along a center plane as shown in Figure 9. The object shown in Figure 9 could have used two full sections to further clarify hidden features. In such a case, the cutting planes and related views are labeled (see Figure 10). The cutting-plane line can be omitted when the relationship between views is obvious. Confirm this practice with your instructor or employer. It is normally best to show the cutting-plane line for clarity.

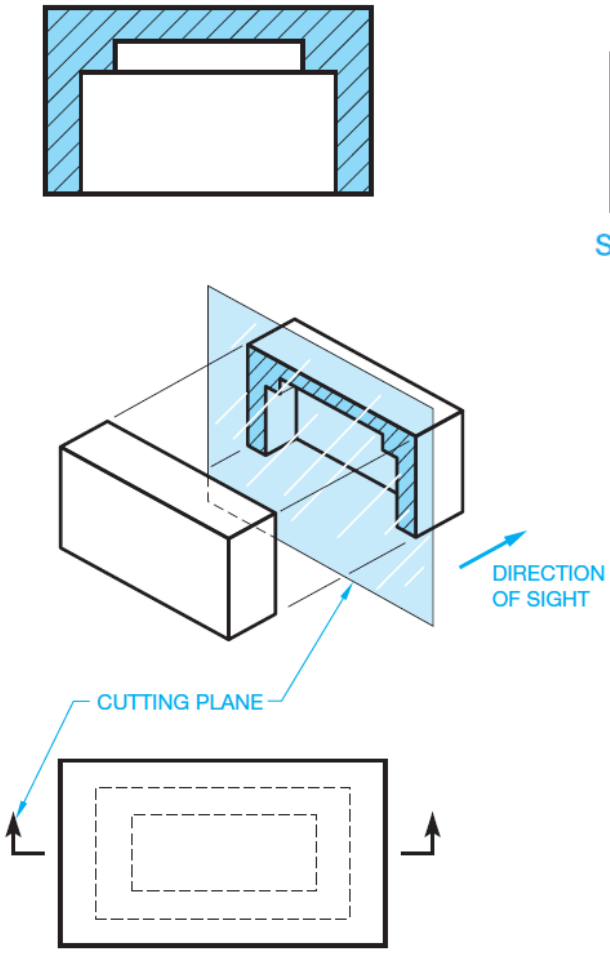


Figure 9 Full section pictorial visualization and related views.

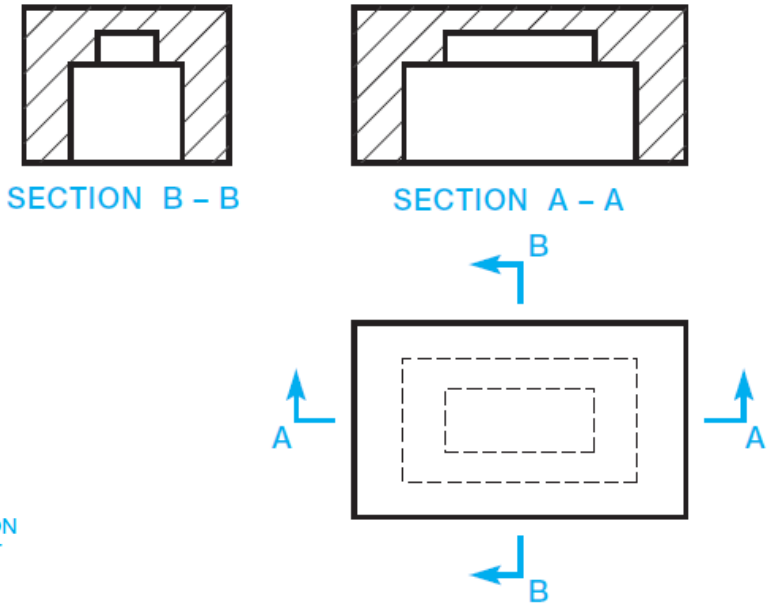


Figure 10 Two full sections drawn for one part. The cutting-plane lines are labeled, and the sectional views have correlated titles.

VISUALIZING A FULL SECTION

Choose a Cutting Plane

1. This illustration shows a collar to be sectioned. To produce a clear section showing both the counter bored recess and the smaller hole near the top of the object, choose a cutting plane that will pass through the vertical centerline in the front view, and imagine the right half of the object removed. It is normally best to show the cutting-plane line for clarity.

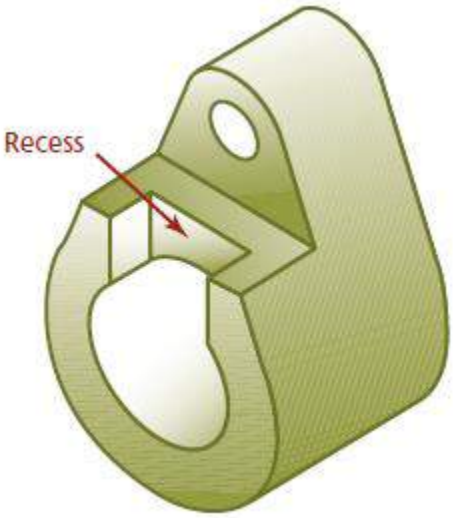


Figure 11

Identify the Surfaces

3. The pictorial drawing of the remaining half is shown at right. The first step in projecting the section view is making sure that you interpret the object correctly. Identifying the surfaces on the object can help. Pass through the vertical centerline in the front view, and imagine the right half of the object removed. It is normally best to show the cutting-plane line for clarity.

Surfaces R, S, T, U, and V have been labeled on the given views and the pictorial view.

- Which surface is R in the front view?
- Which surface is U in the top view?
- Are they normal, inclined, or oblique surfaces?
- Can you identify the counter bored recess in each view?

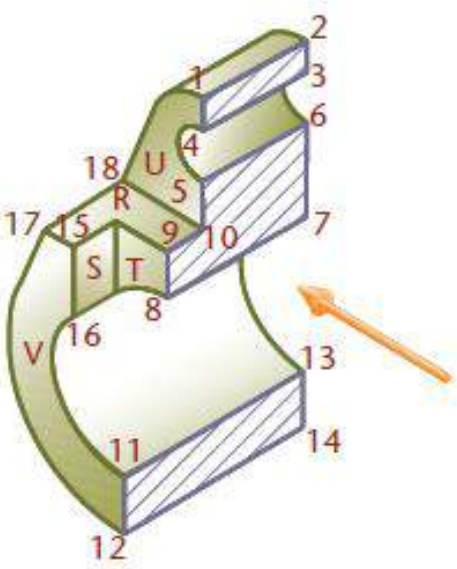


Figure 12

Draw the Section View

2. To draw the section view, omit the portion of the object in front of the cutting plane. You will be drawing only the portion that remains.

Determine which solid parts of the object the cutting plane will pass through. *Hint: The outside of an object can never be a hole; it must be solid, unless the cutting plan passes through a slot to the exterior.*

The points that will be projected to create the section view have been identified for you in the example shown.

The three surfaces produced by the cutting plane are bounded by points 1-2-3-4 and 5-6-7-8-9-10 and 13-14- 12-11. These are shown hatched.

Each sectioned area is completely enclosed by a boundary of visible lines. In addition to the cut surfaces, the section view shows all visible parts behind the cutting plane.

No hidden lines are shown. However, the corresponding section shown in this step is incomplete, because visible lines are missing.

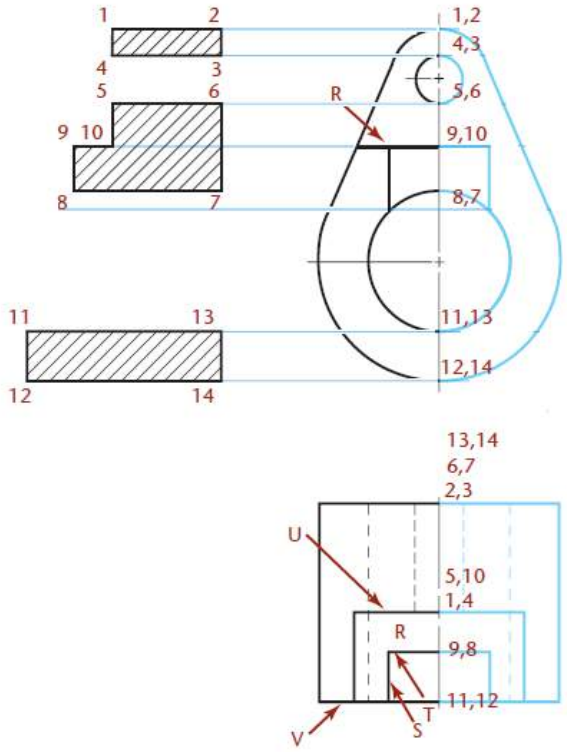


Figure 13

Draw the Section View

4. To From the direction the section is viewed, the top surface (V) of the object appears in the section as a visible line (12-11-16-15).

The bottom surface of the object appears similarly as 14-13-7-6-3-2. The bottom surface of the counter bored recess appears in the section as line 19-20.

Also, the back half of the counter bored recess and the drilled hole appear as rectangles in the section at 19-20 15-16 and 3-4-5-6. These points must also be projected. The finished view is shown at right.

Notice that since all cut surfaces are part of the same object, the hatching must all run in the same direction.

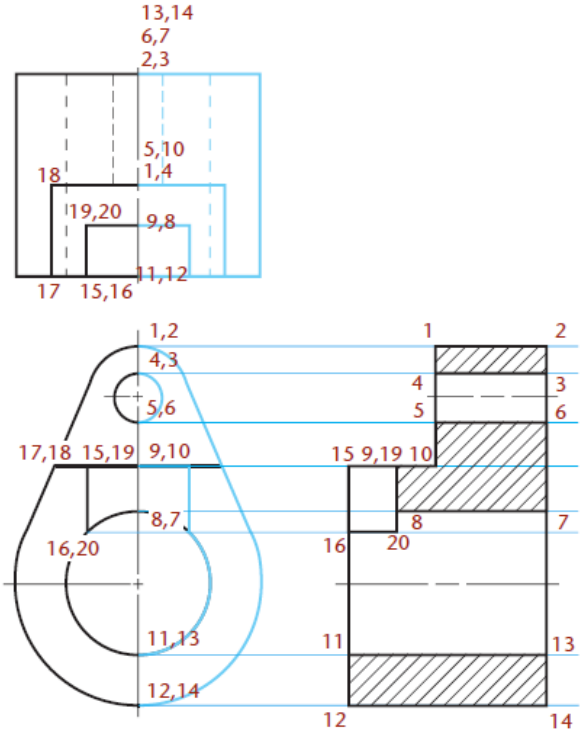
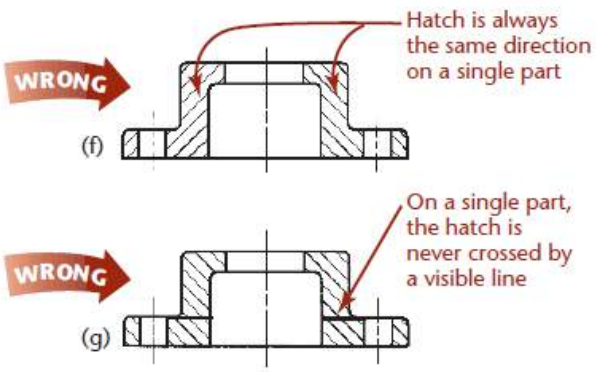


Figure 14

RULES FOR LINES IN SECTION VIEWS

- Show edges and contours that are now visible behind the cutting plane; otherwise a section will appear to be made up of disconnected and unrelated parts. (Occasionally, visible lines behind the cutting plane may be omitted, particularly from those generated from 3D models.)
- Omit hidden lines in section views. Section views are used to show interior detail without a confusion of hidden lines, so add them only if necessary to understand the part.
- Sometimes hidden lines are necessary for clarity and should be used in such cases, especially if their use will make it possible to omit a view (Figure d).
- A sectioned area is always completely bounded by a visible outline—never by a hidden line, because in every case the cut surfaces will be the closest surface in the section view and therefore their boundary lines will be visible (Figure e).
- In a section view of an object, the section lines in all hatched areas for that object must be parallel, not as shown in Figure f. The use of section lining in opposite directions is an indication of different parts, as when two or more parts are adjacent in an assembly drawing.
- A visible line can never cross a sectioned area in a view of a single part. This would be impossible on the full section of a single part because the section lines are all in the same plane. A line across it would indicate a change of plane (Figure g).



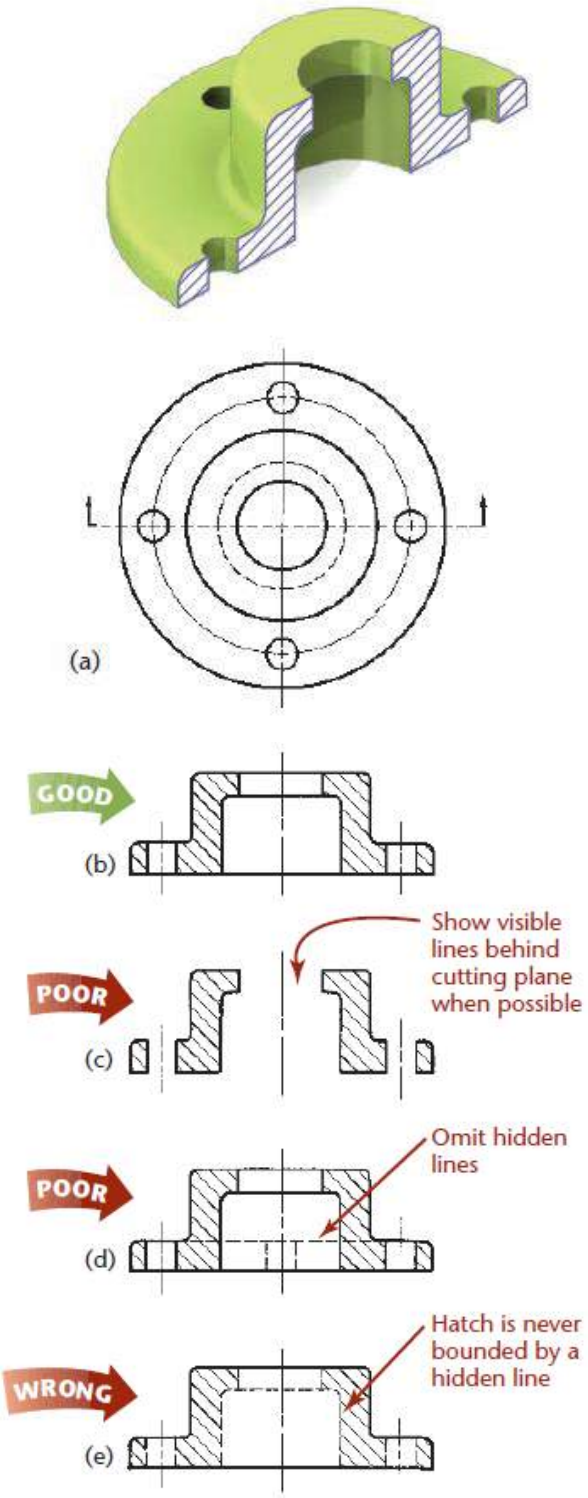


Figure 15

SECTION-LINING TECHNIQUE

The correct method of drawing section lines is shown in Figure 16. When drawing by hand, use a sharp, medium grade pencil (H or 2H) to draw uniformly thin section lines, or hatching (a term meaning closely spaced parallel lines). There should be a marked contrast between the thin section lines and the thick visible outlines of the part.

- Draw section lines at 45° from horizontal unless they would be parallel or perpendicular to major edges of the part, in which case use a different angle. Figure 17 shows an example of section lines drawn at a different angle to prevent them from being *parallel or perpendicular to visible outlines.*
- Space the lines as evenly as possible by eye (for most drawings, about 2.5 mm apart). The spacing interval depends on the size of the drawing or of the sectioned area, with larger drawings having wider spacing. In a smaller drawing the spacing interval may be as small as 1.5 mm; in a large drawing, it may be 3 mm.
- Keep extension lines and dimension values off sectioned areas. If there is no alternative, omit the section lines behind the dimensions (Figure 18).

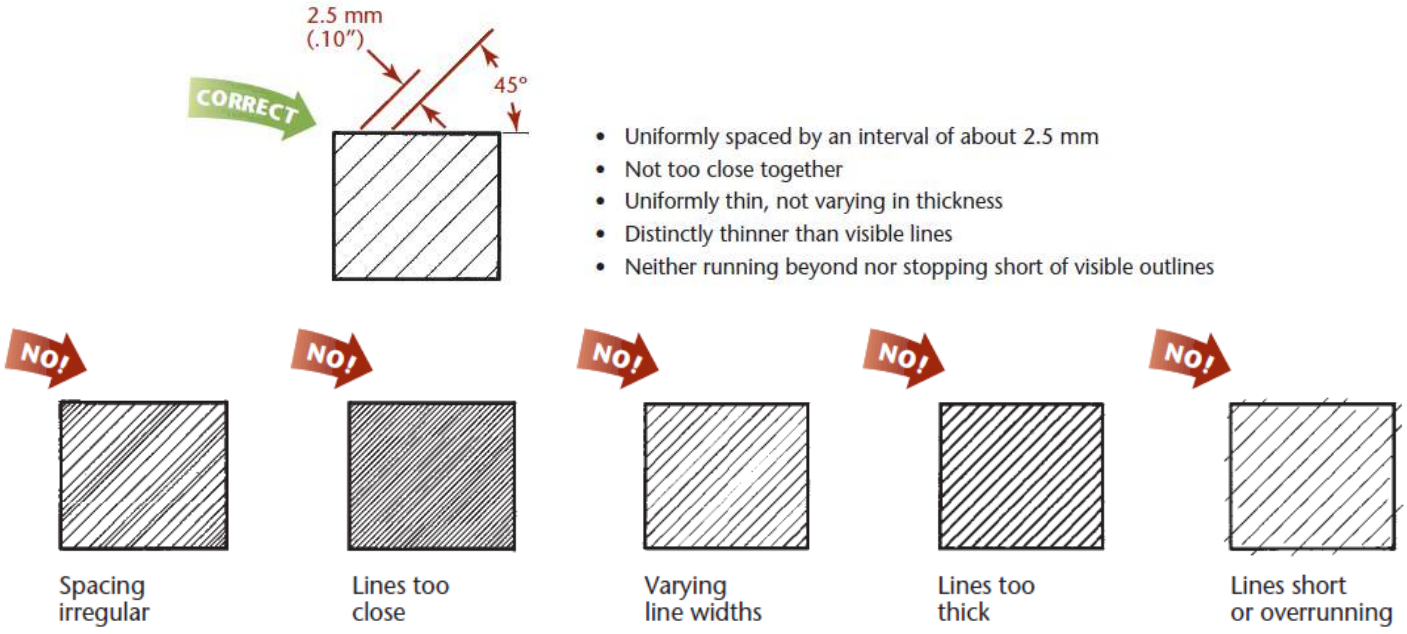


Figure 16 Correctly drawn section lines

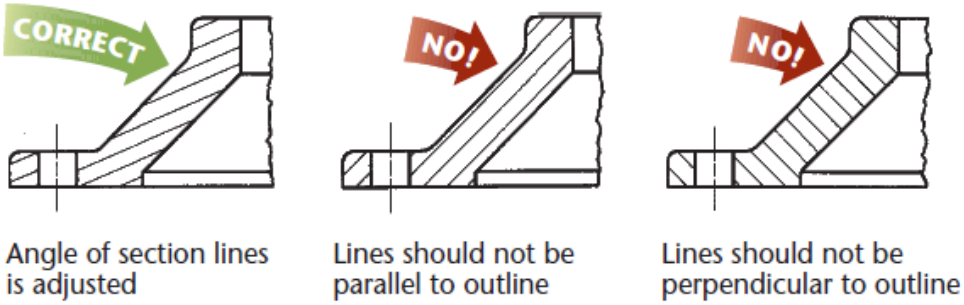


Figure 17 Direction of Section Lines

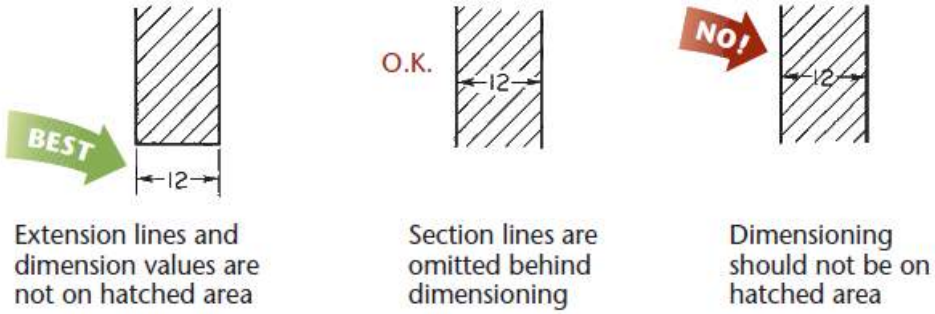


Figure 18 Keep extension lines and values for dimensions off crosshatched areas, but when this is unavoidable, the cross-hatching should be omitted where the dimension value is placed.

SECTION-LINING SYMBOLS

Section-lining symbols (Figure 19) may be used to indicate specific materials. These symbols represent general material types only, such as cast iron, brass, and steel. Because there are so many different types of materials (there are hundreds of types of steel, for example), a general name or symbol is not enough. A detailed specification listing the material must be lettered in the form of a note or in the title strip.

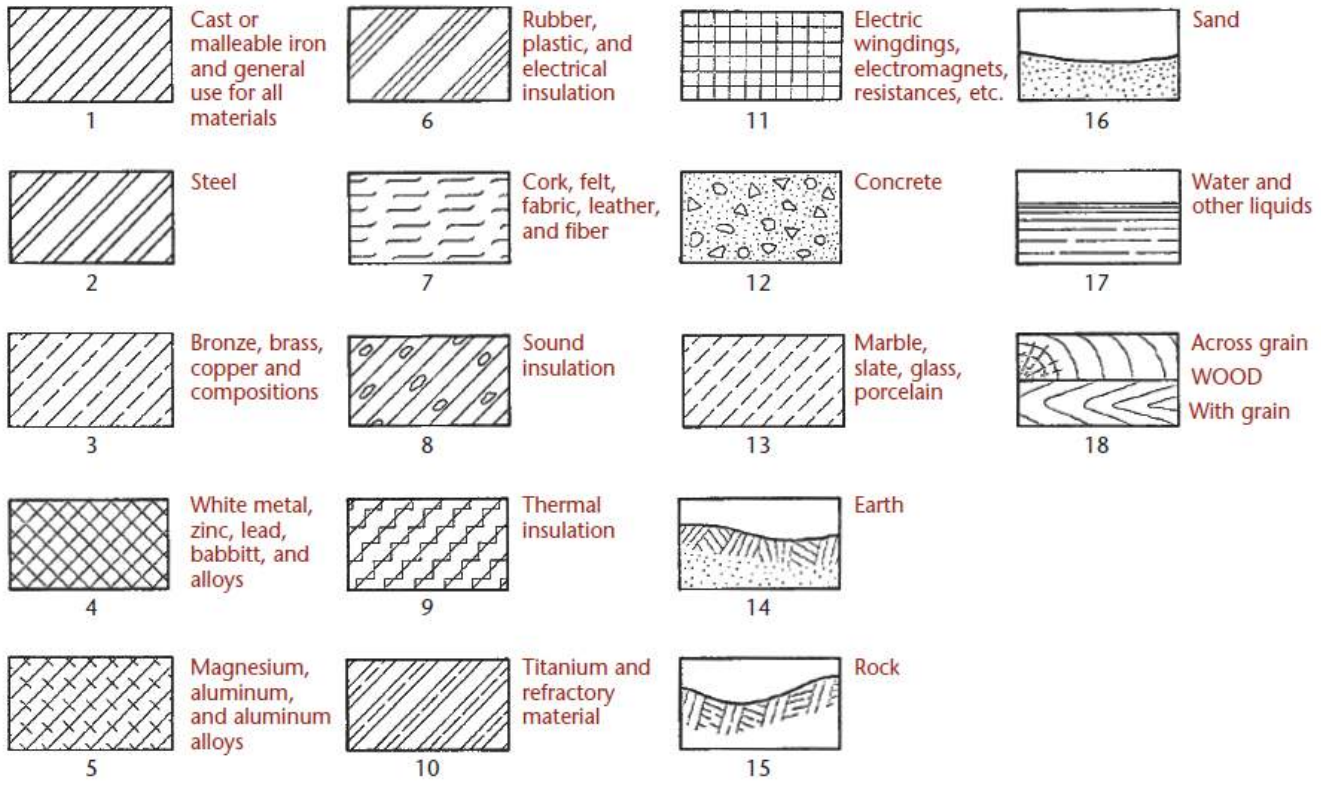


Figure 19 Symbols for Section Lining

HALF SECTIONS

Symmetrical objects can be shown effectively using a special type of section view called a ***half section*** (Figure 20). A half section exposes the interior of half of the object and the exterior of the other half. This is done by removing one quarter of the object. Half sections are not widely used to create detail drawings showing how to make a single part because it can be difficult to show all the dimensions clearly when some internal features are only partly shown in the sectioned half (Figure 21). In general,

- Omit hidden lines from both halves of a half section, whenever possible.
- Use a centerline to divide the sectioned half and the unsectioned half, as shown in Figure 21.

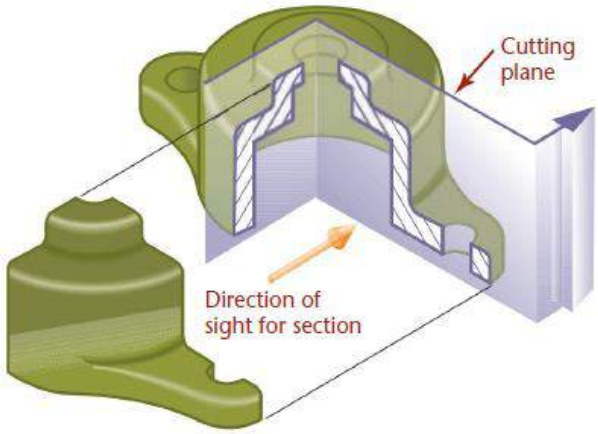


Figure 20

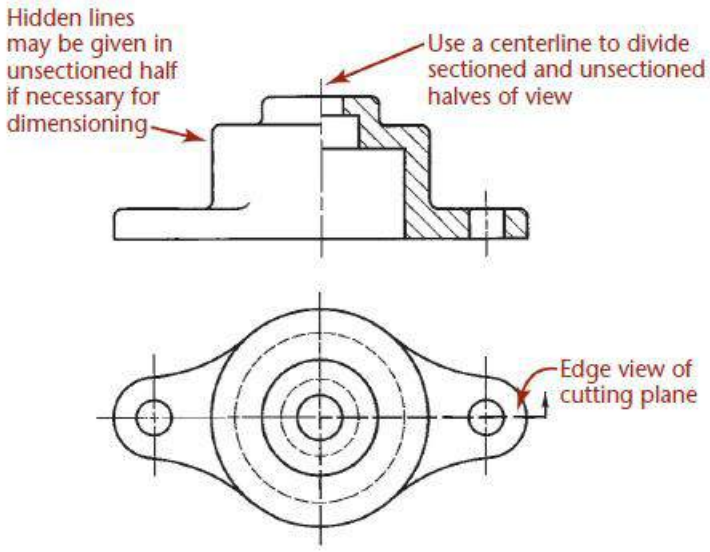


Figure 21

OFFSET SECTIONS

Staggered interior features of an object are sectioned by allowing the cutting-plane line to offset through the features creating an ***offset section*** as shown in Figure 22.

The cutting-plane line for an offset section is generally drawn using 90 ^{O} turns where it offsets through the staggered features as shown in Figure 22. Notice in Figure 22 that there is no line in the sectional view indicating a change in direction of the cutting-plane line. Normally, the cutting-plane line in an offset section extends completely through the object to display the location of the section clearly. A cutting plane line is always used when the cutting plane is bent or offset or when the sectional view is nonsymmetrical.

Figure 23 shows how the segments of an offset cutting plane project from and are aligned with the center when used on a circular-shaped object. The portion of the cutting-plane line between offsets is drawn as an arc, with the arc center at the center of the object.

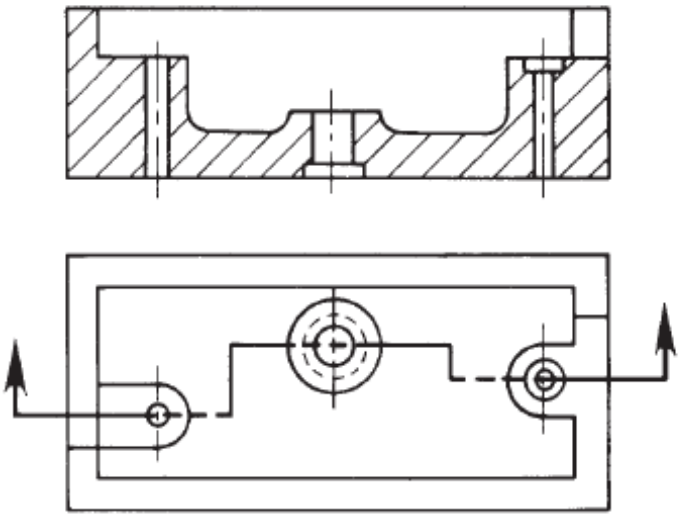


Figure 22 The cutting-plane line offsets using 90^O turns to go through offset features on the part.

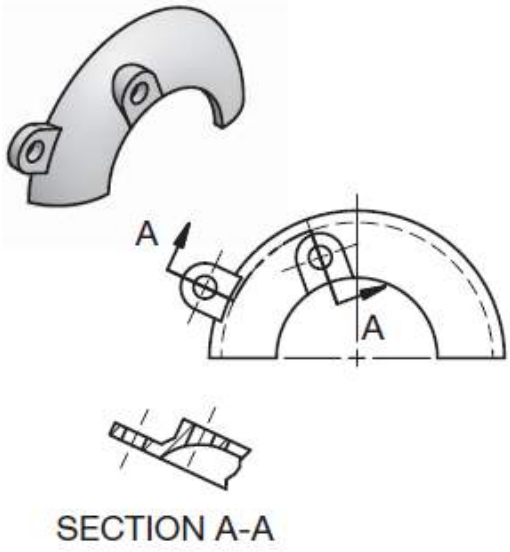


Figure 23

WEBS IS SECTIONS

To avoid giving a false impression of thickness and solidity, webs, webs, gear teeth, and other similar flat features are not hatched with section lining even though the cutting plane slices them. For example, in Figure 24, the cutting plane A–A slices through the center of the vertical web, or rib, and the web is not sectioned (Figure 24b). Thin features are not hatched even though the cutting plane passes lengthwise through them. The incorrect section is shown in Figure 24c. Note the false impression of thickness or solidity resulting from section lining the web.

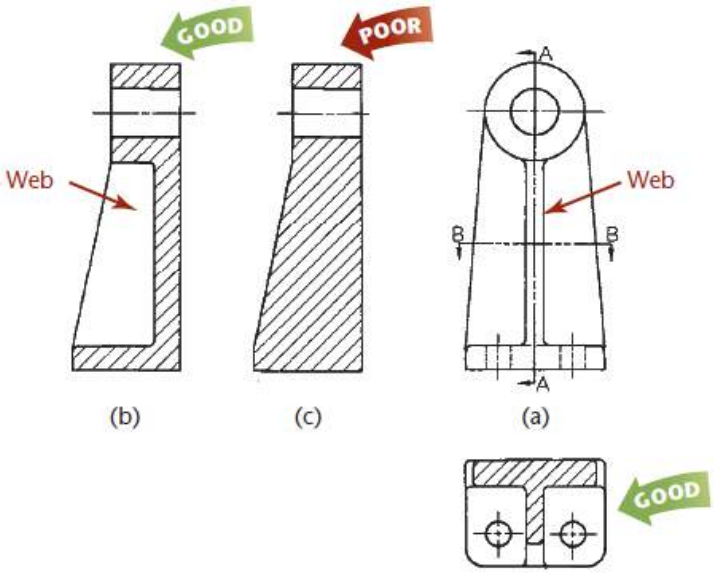
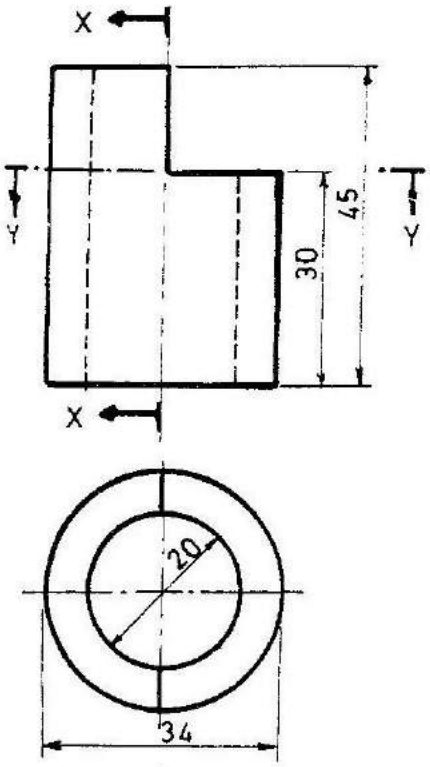


Figure 24 Web in section

If the cutting plane passes crosswise through a rib or any thin member, as in section B–B, section-line the feature in the usual manner, as in the top view of Figure 24a.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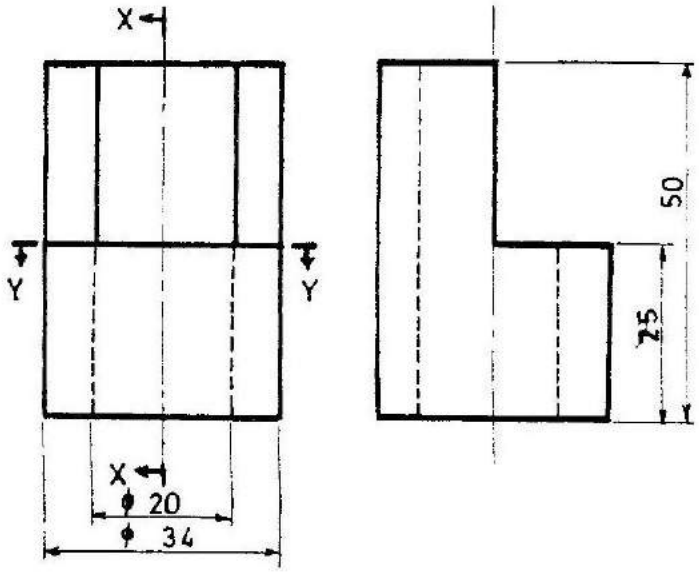
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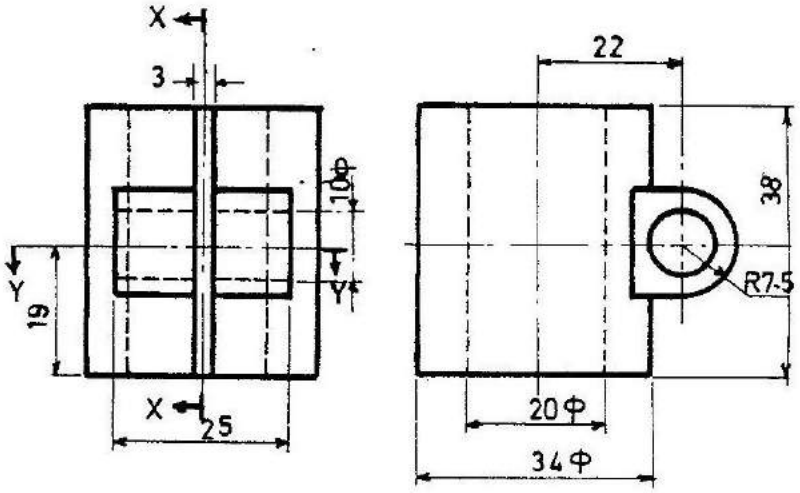
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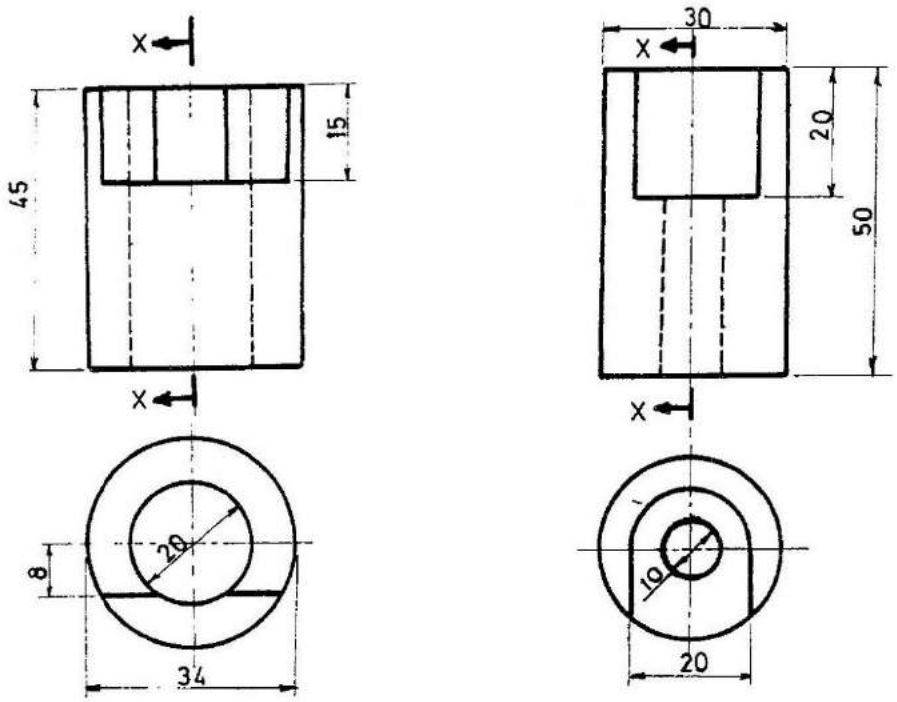
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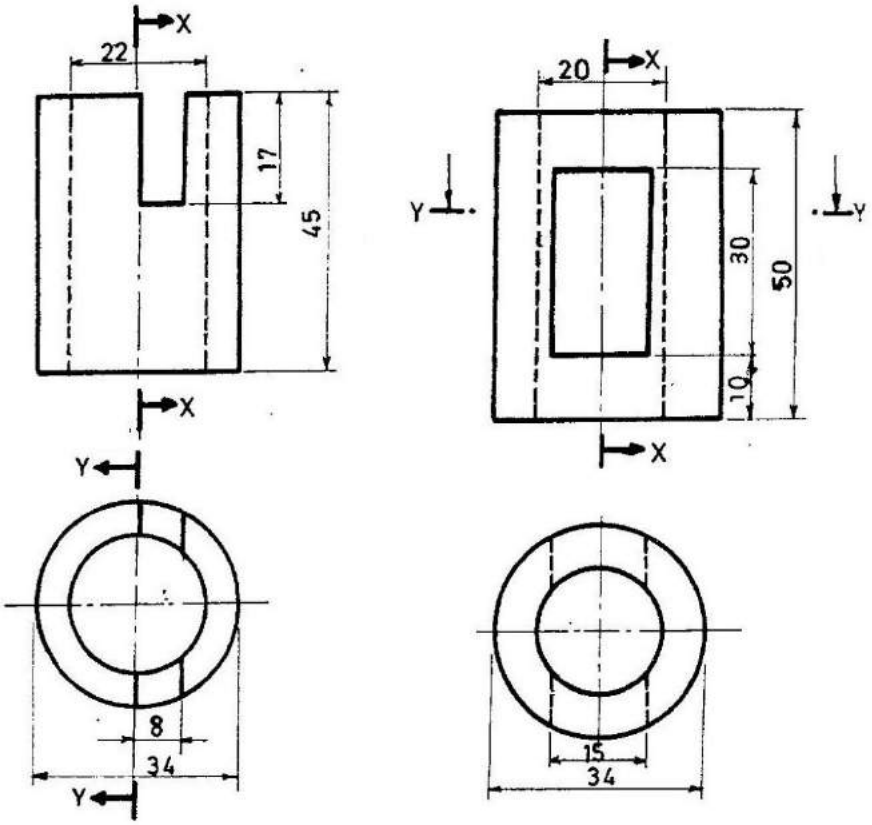
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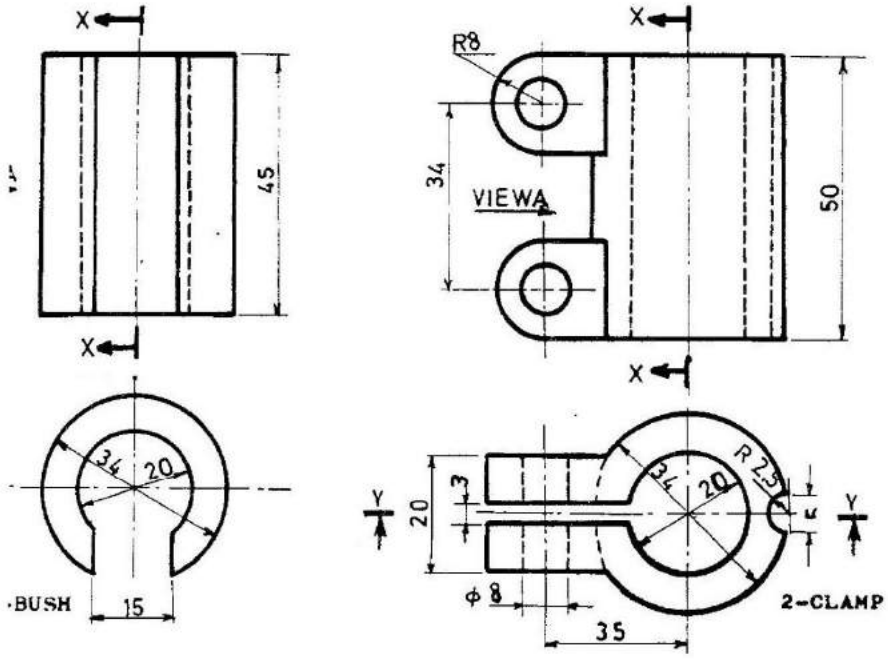
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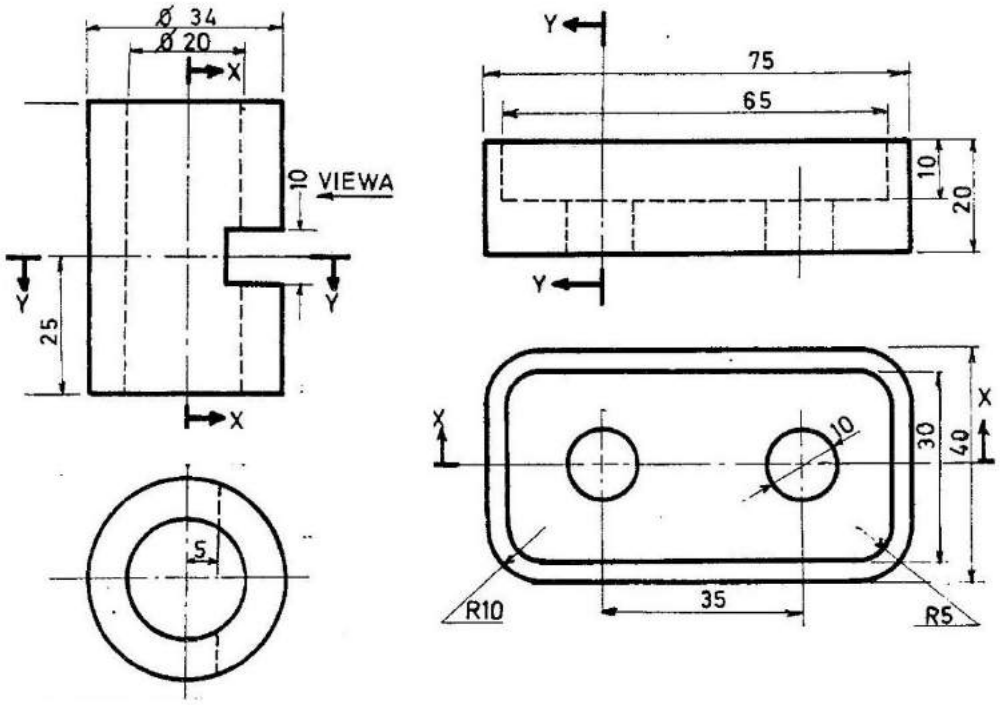
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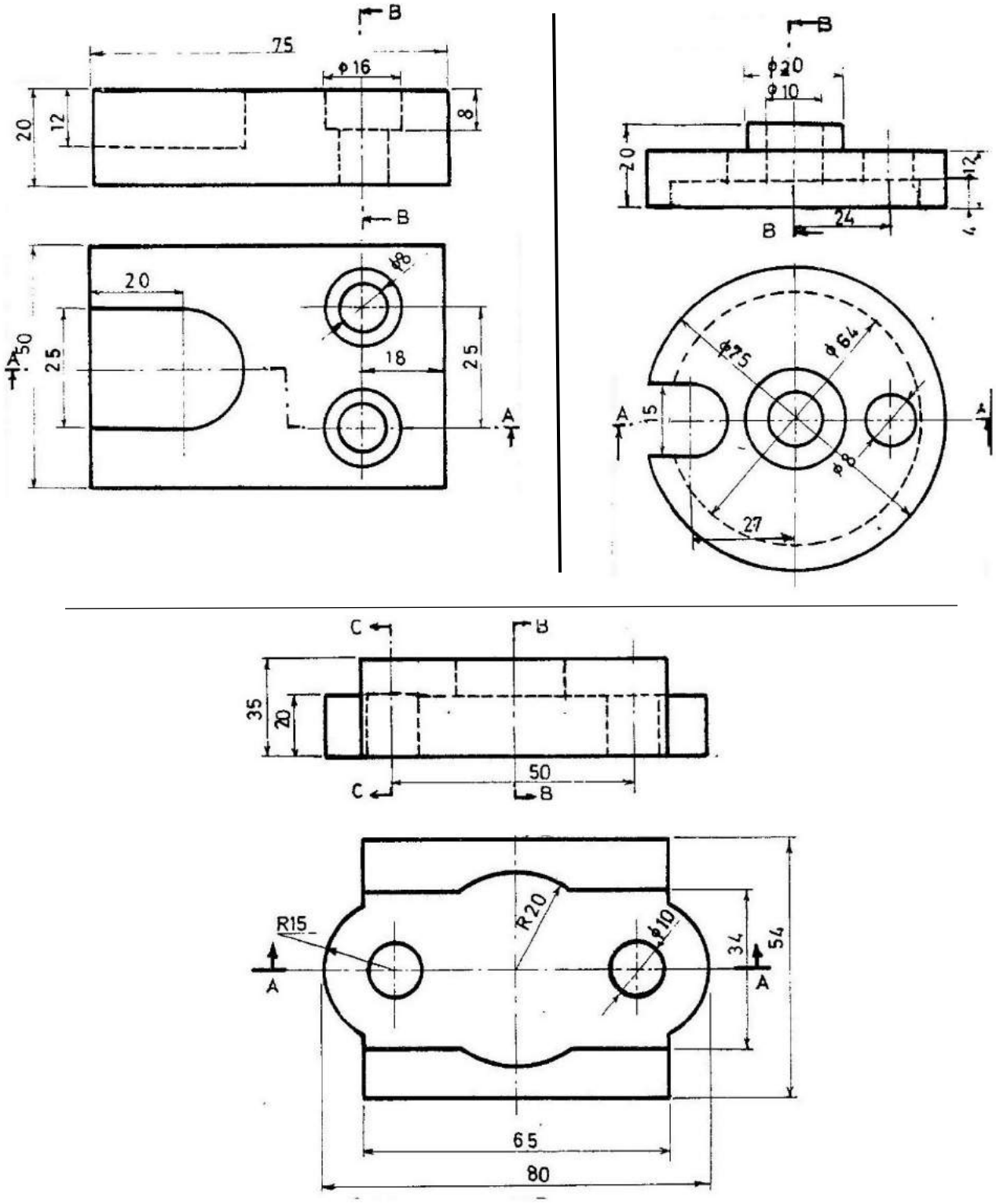
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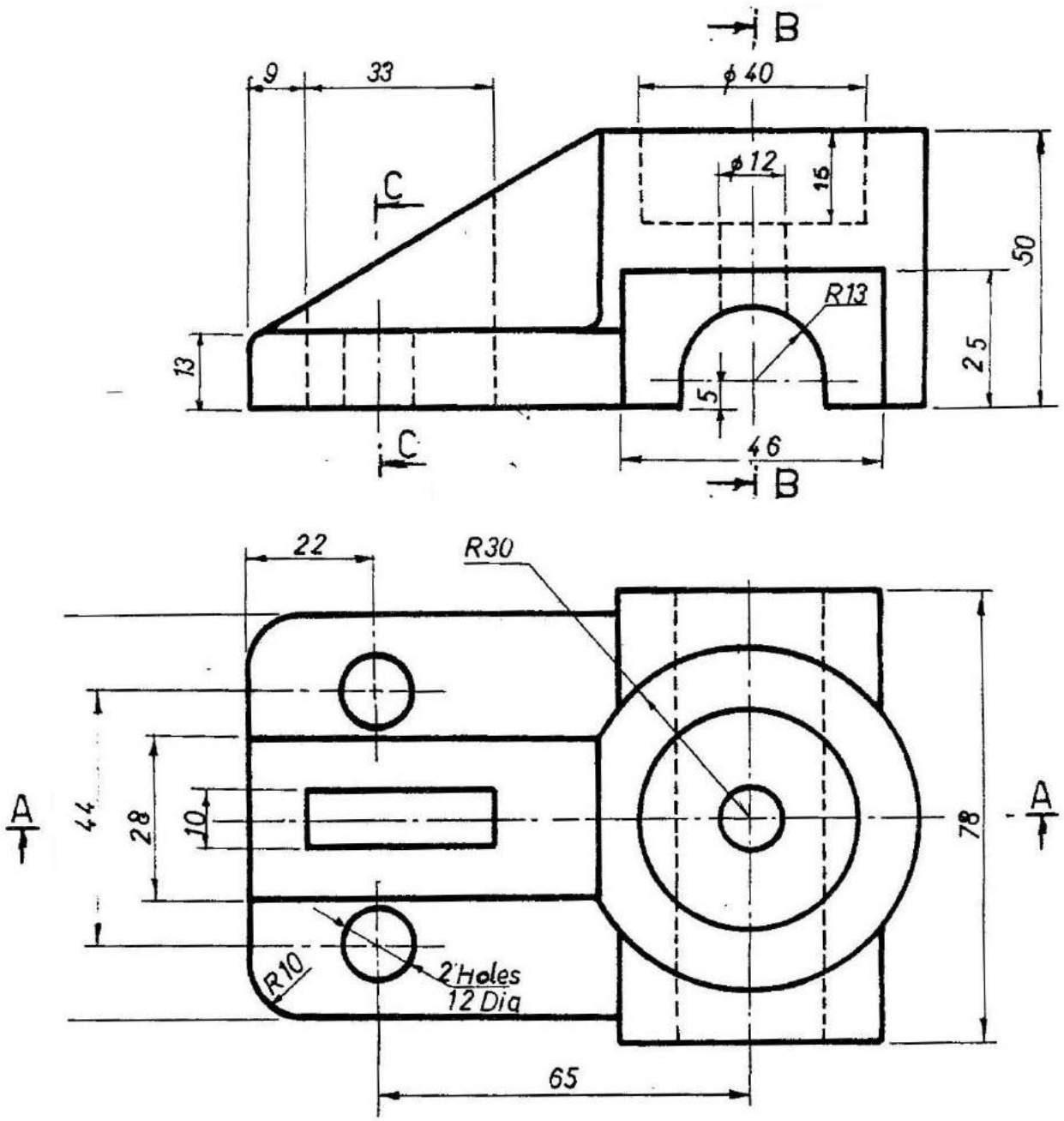
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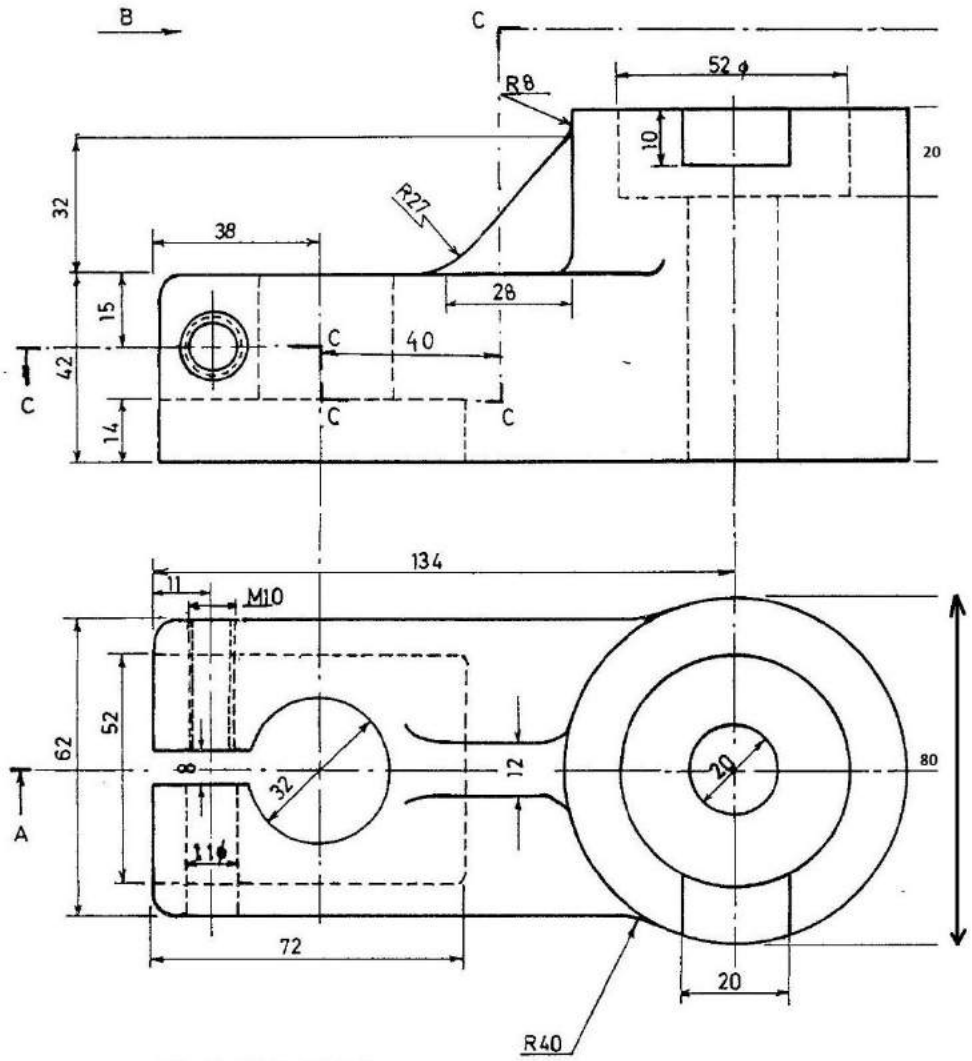
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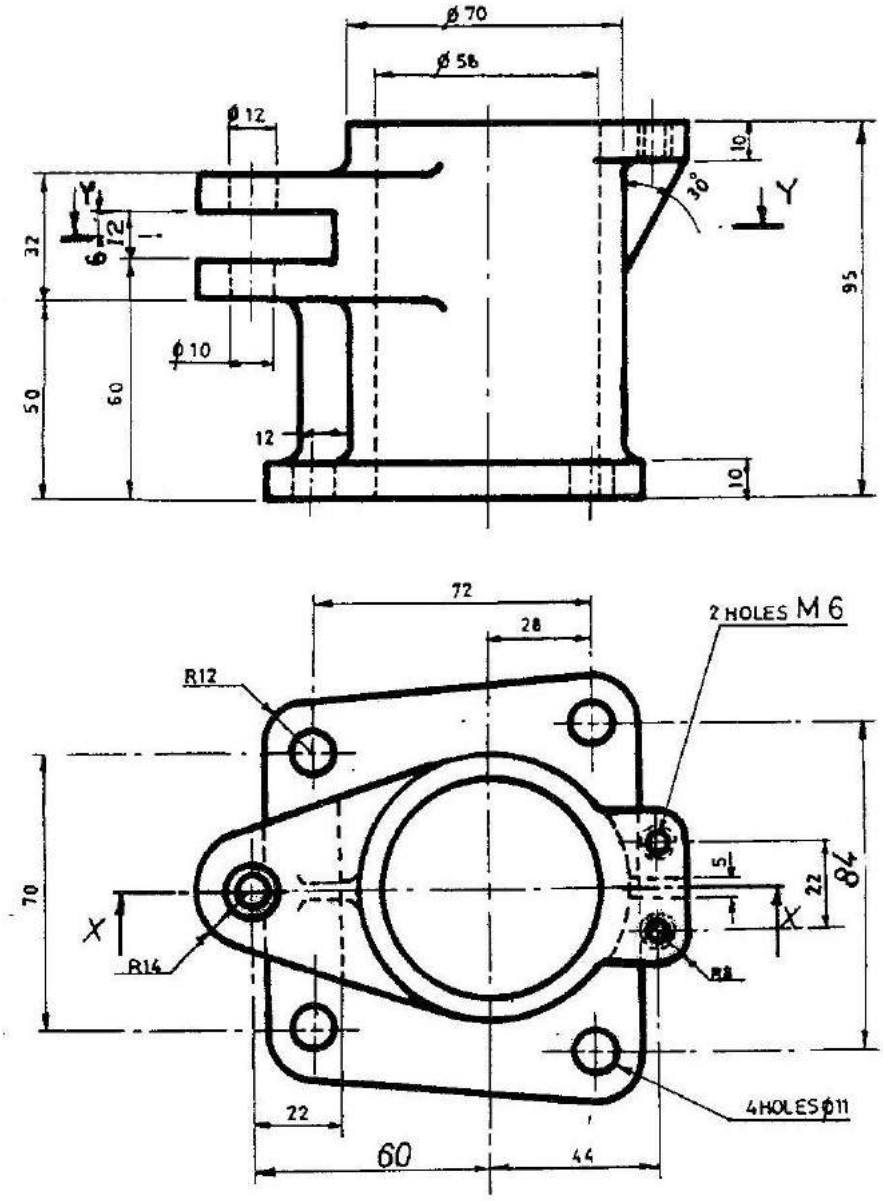
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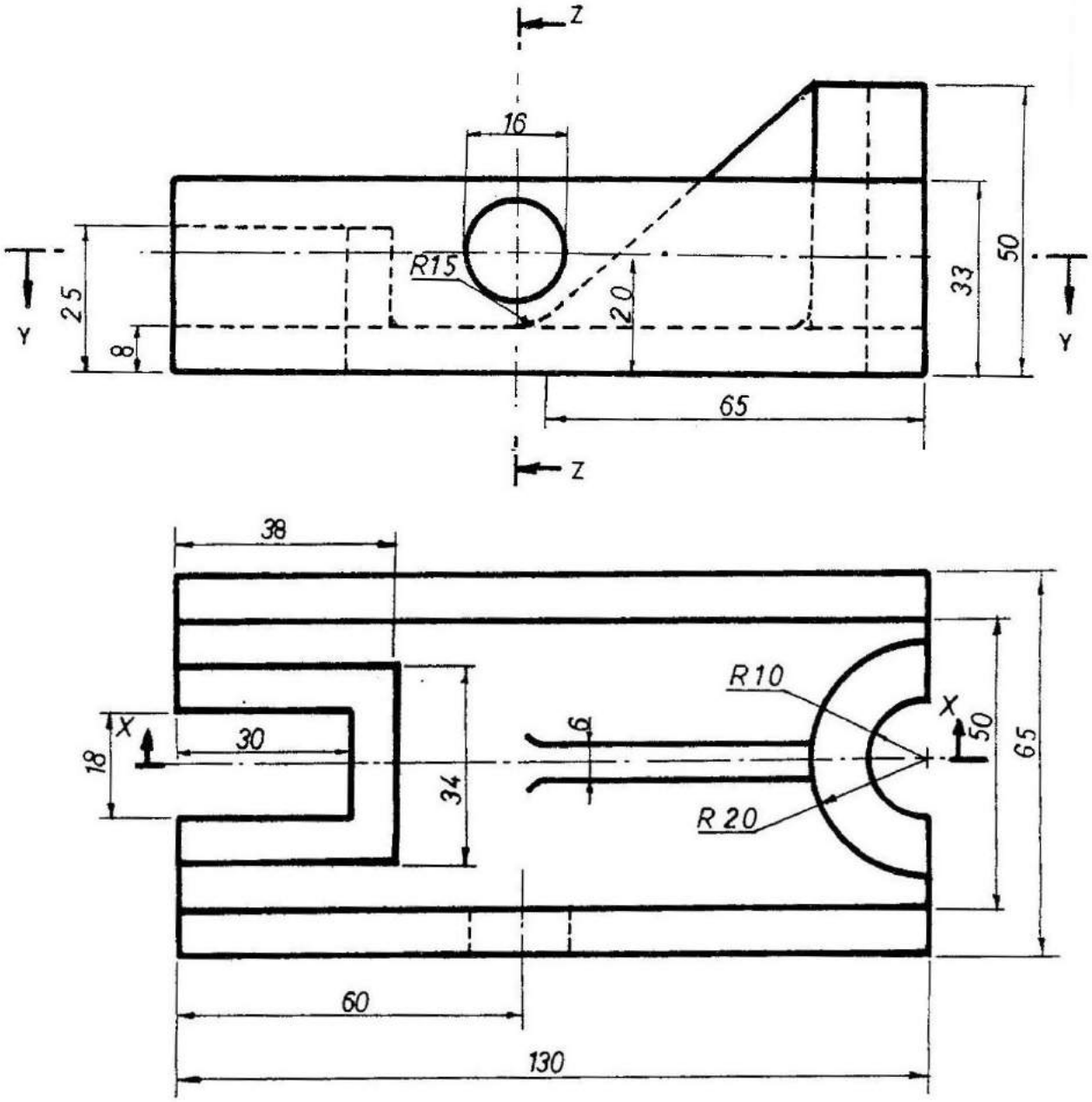
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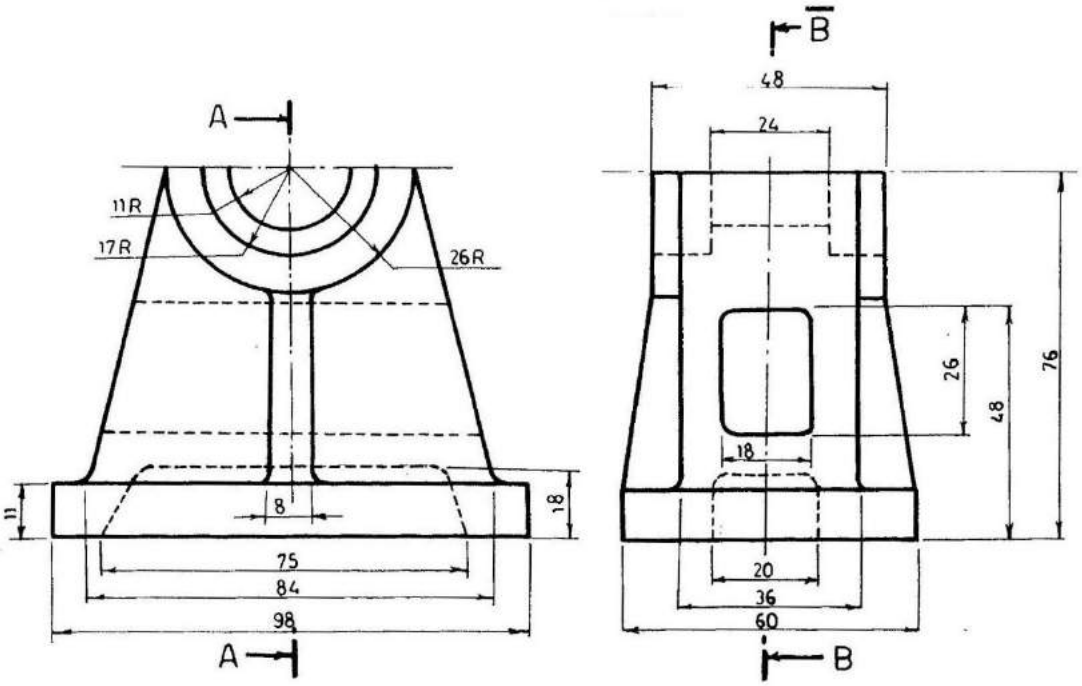
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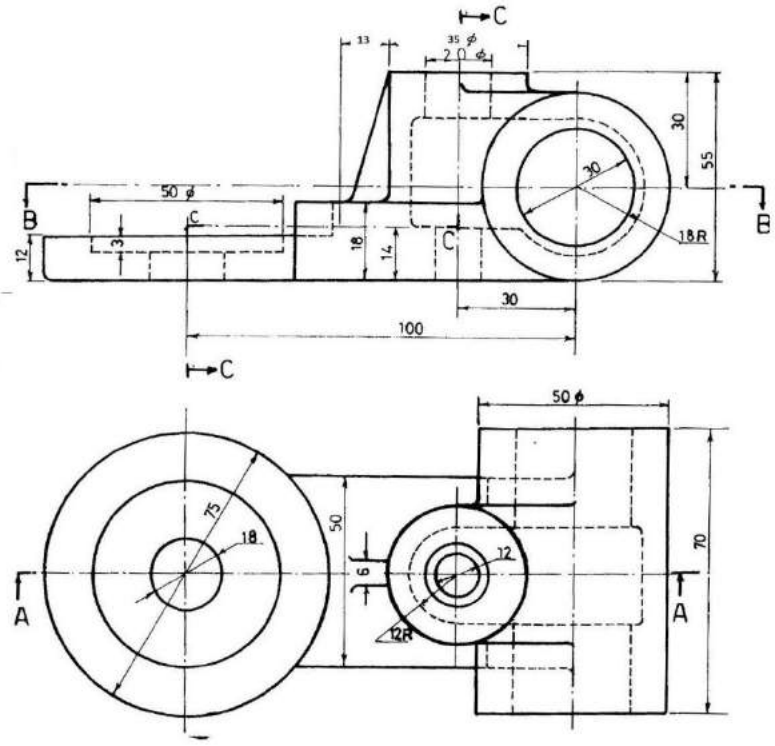
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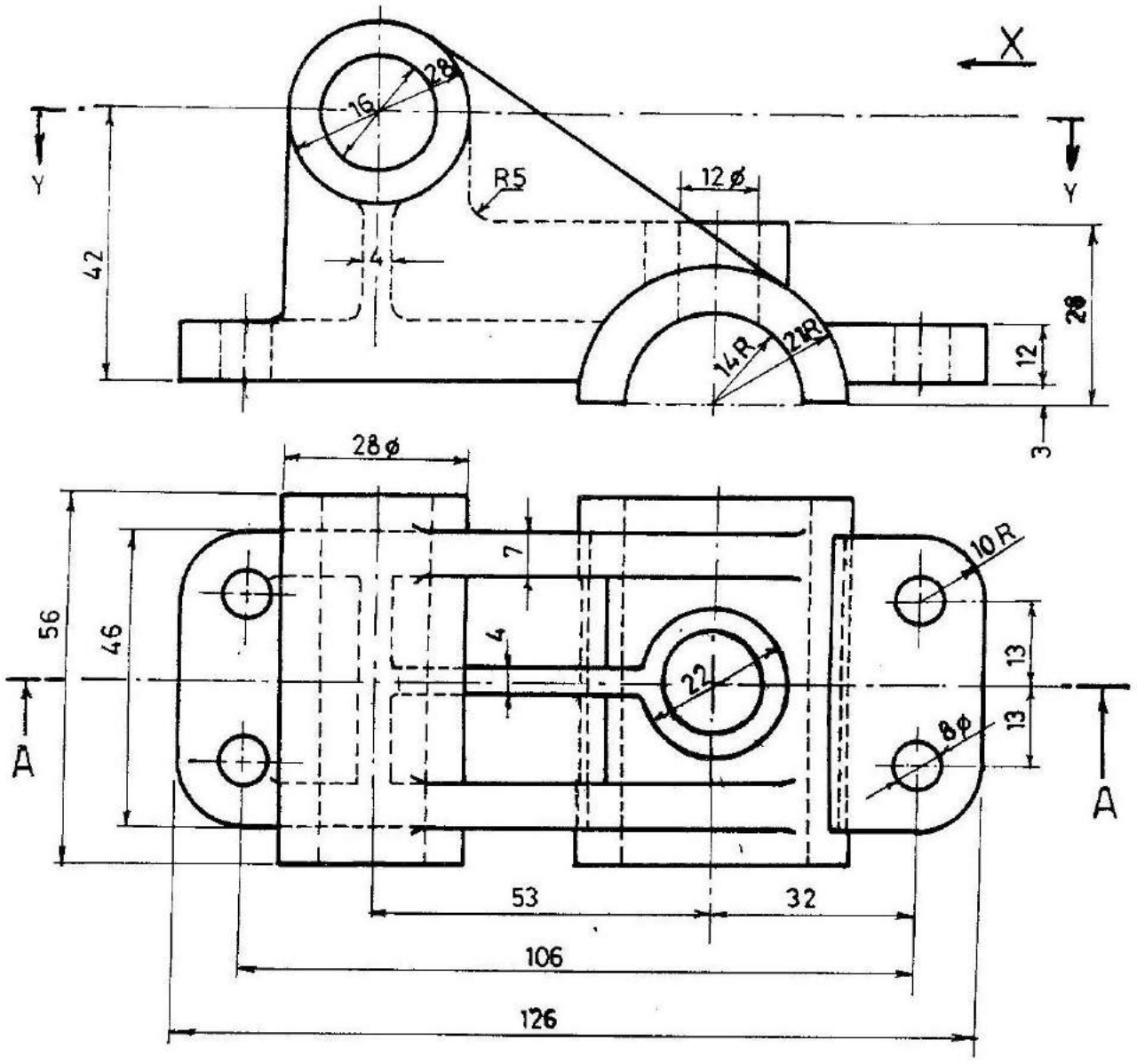
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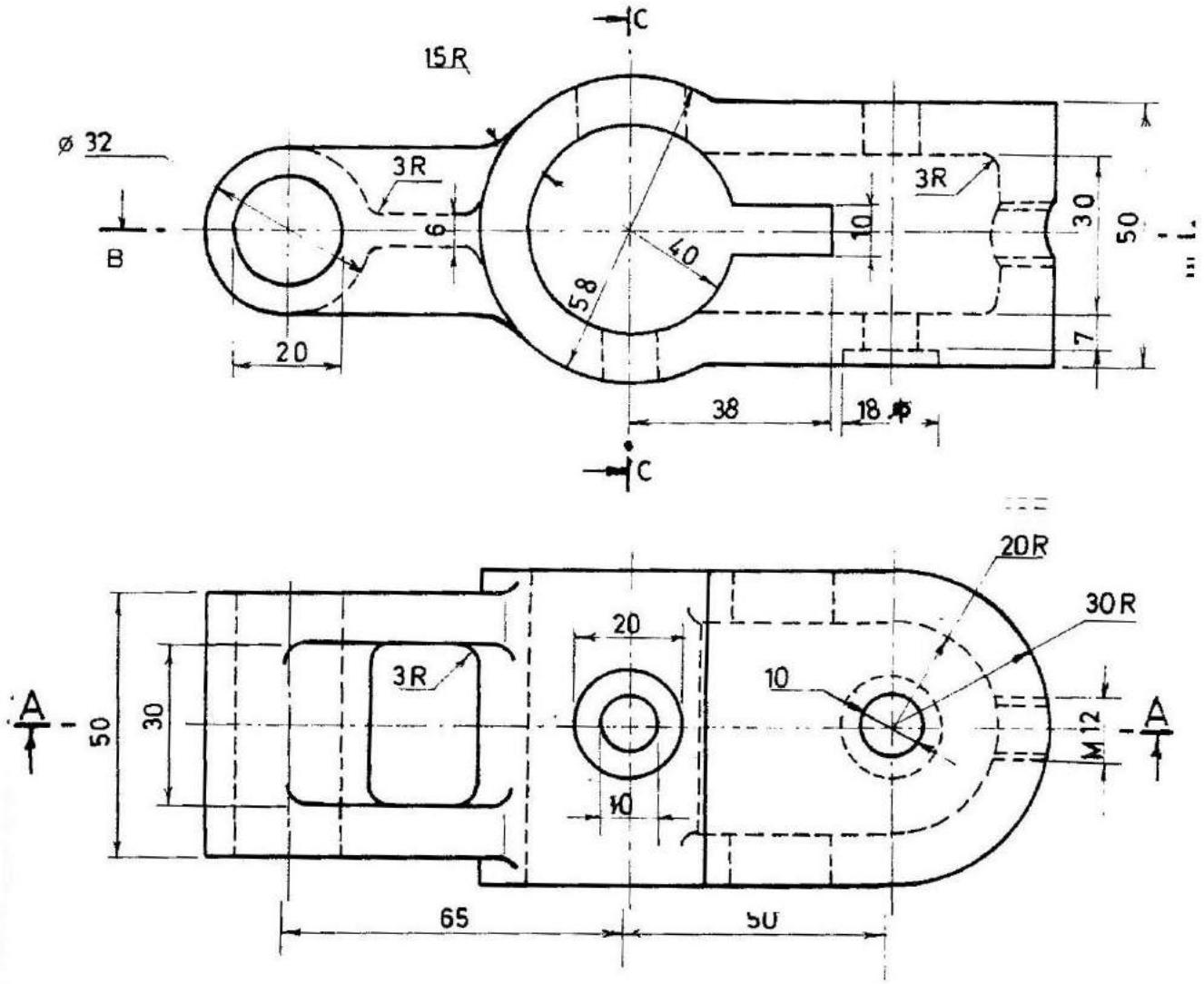
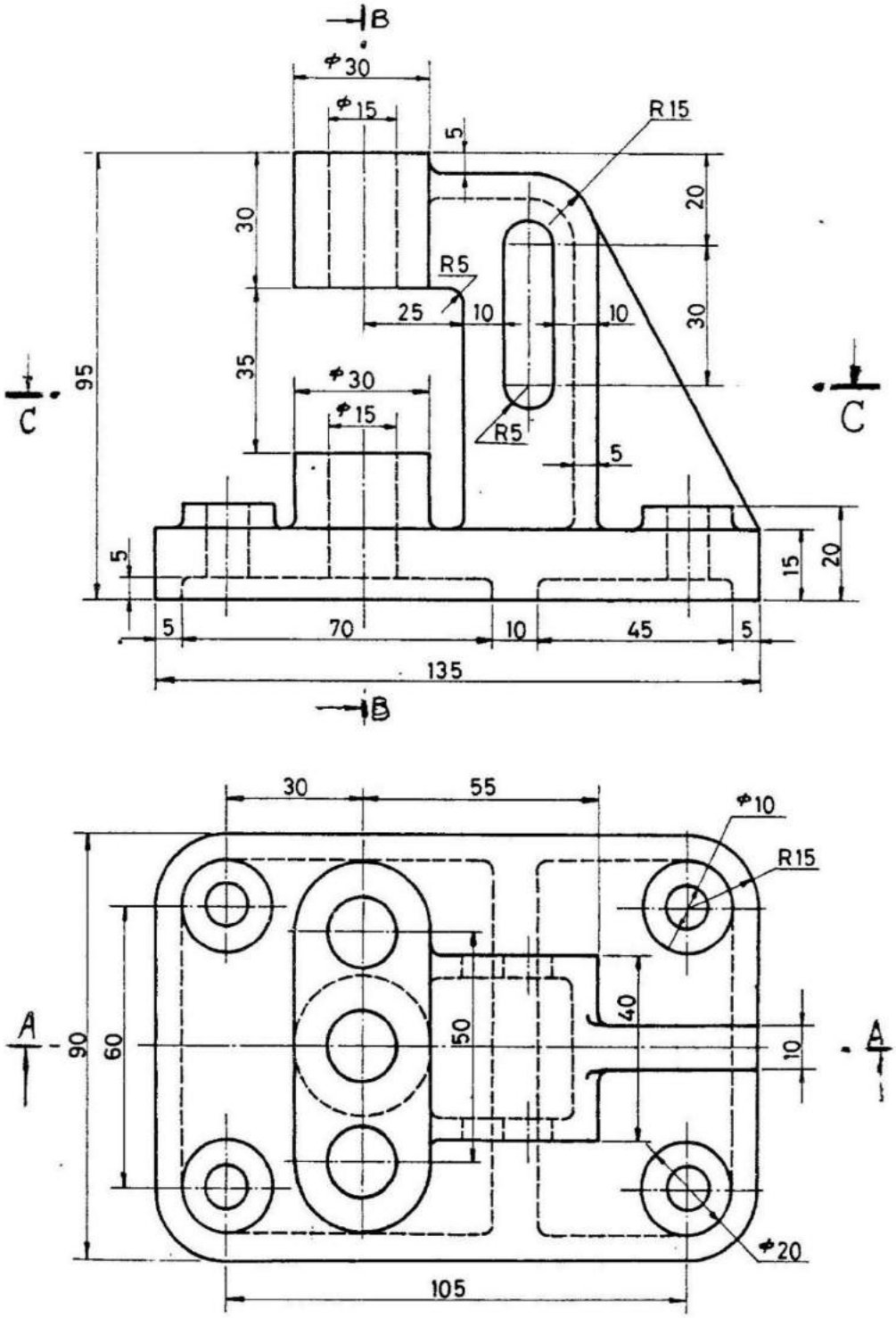
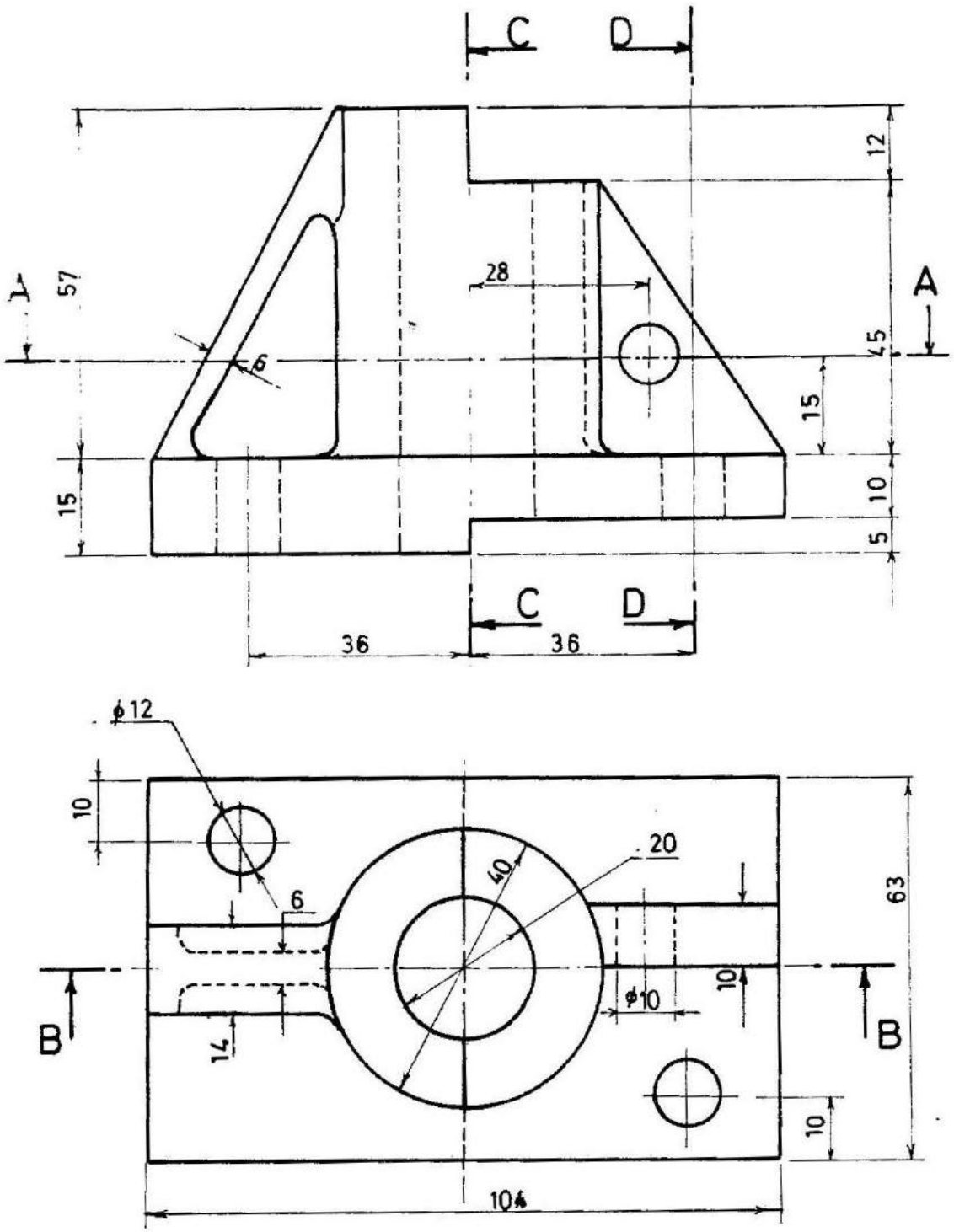
28
12
3
14R
21R
28 ø
7
4
56
46
22
10 R
13
13
8 ø
A
A
53
32
106
126


C
15R
ø 32
3R
B
6
3R
10
40
58
30
50
20
7
38
18
C
20R
30R
20
3R
10
A
50
30
M12
A
10
65
50


B
ϕ30
ϕ15
5
R15
20
30
R5
25
10
10
30
35
ϕ30
ϕ15
R5
5
C
95
C
5
15
20
5
70
10
45
5
135
B
30
55
ϕ10
R15
A
90
60
50
40
10
A
ϕ20
105


C
D
12
57
28
A
6
45
15
15
10
5
C
D
36
36
ϕ12
10
6
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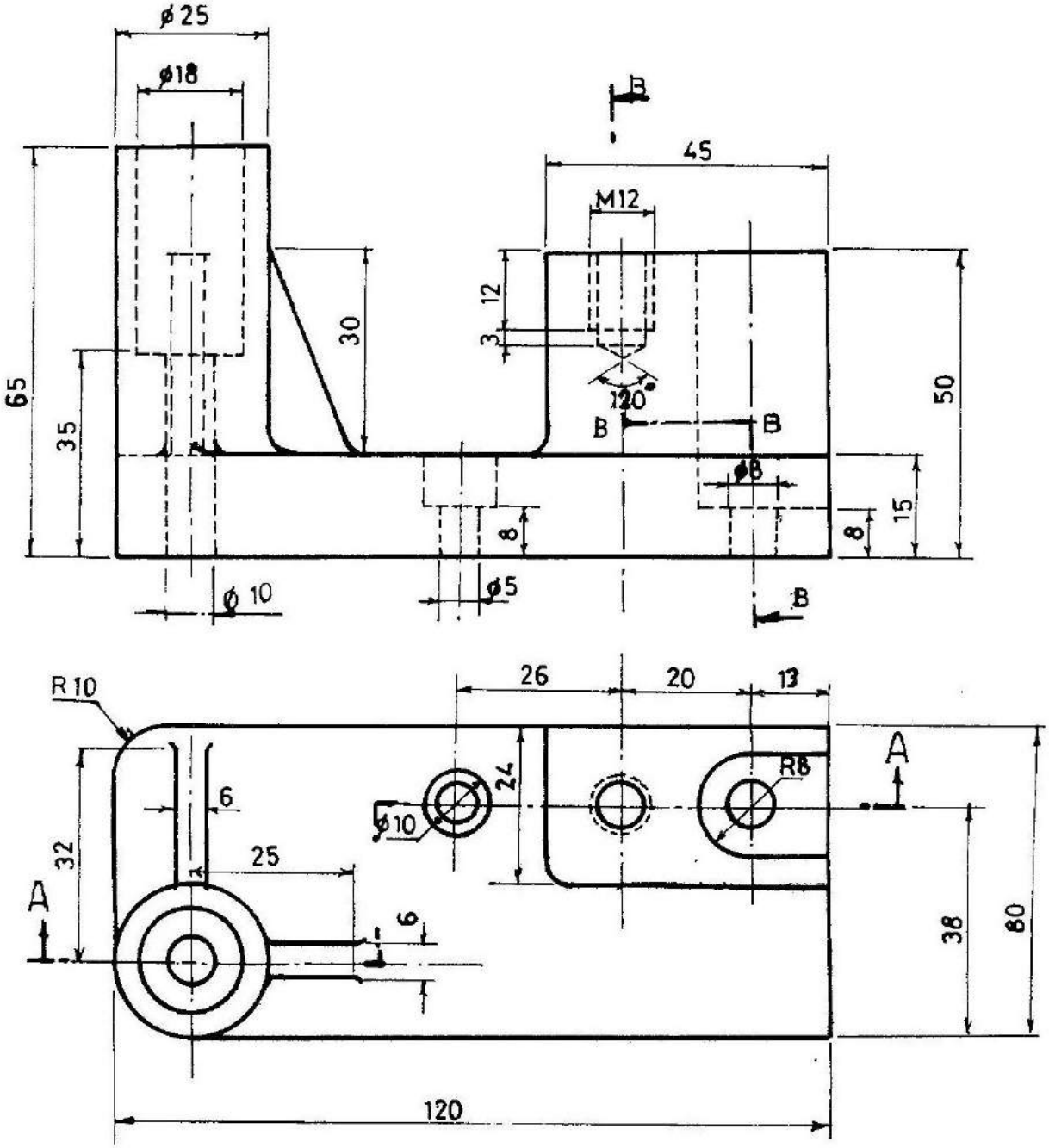
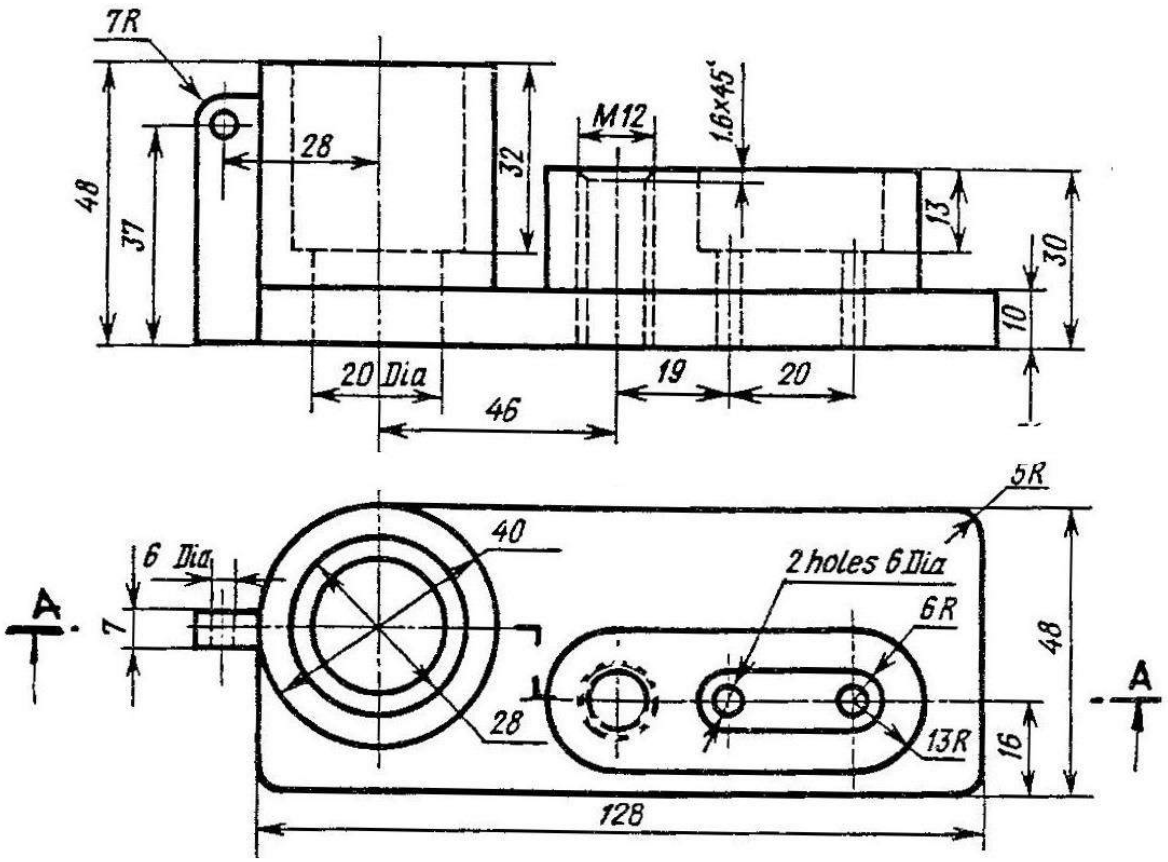
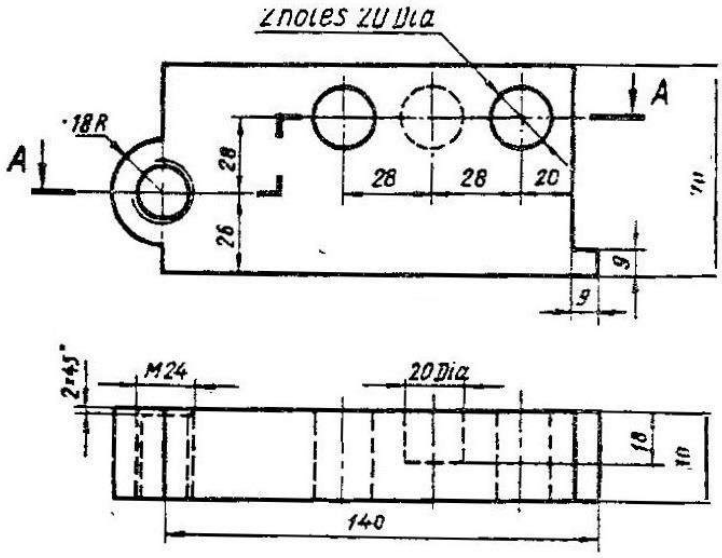
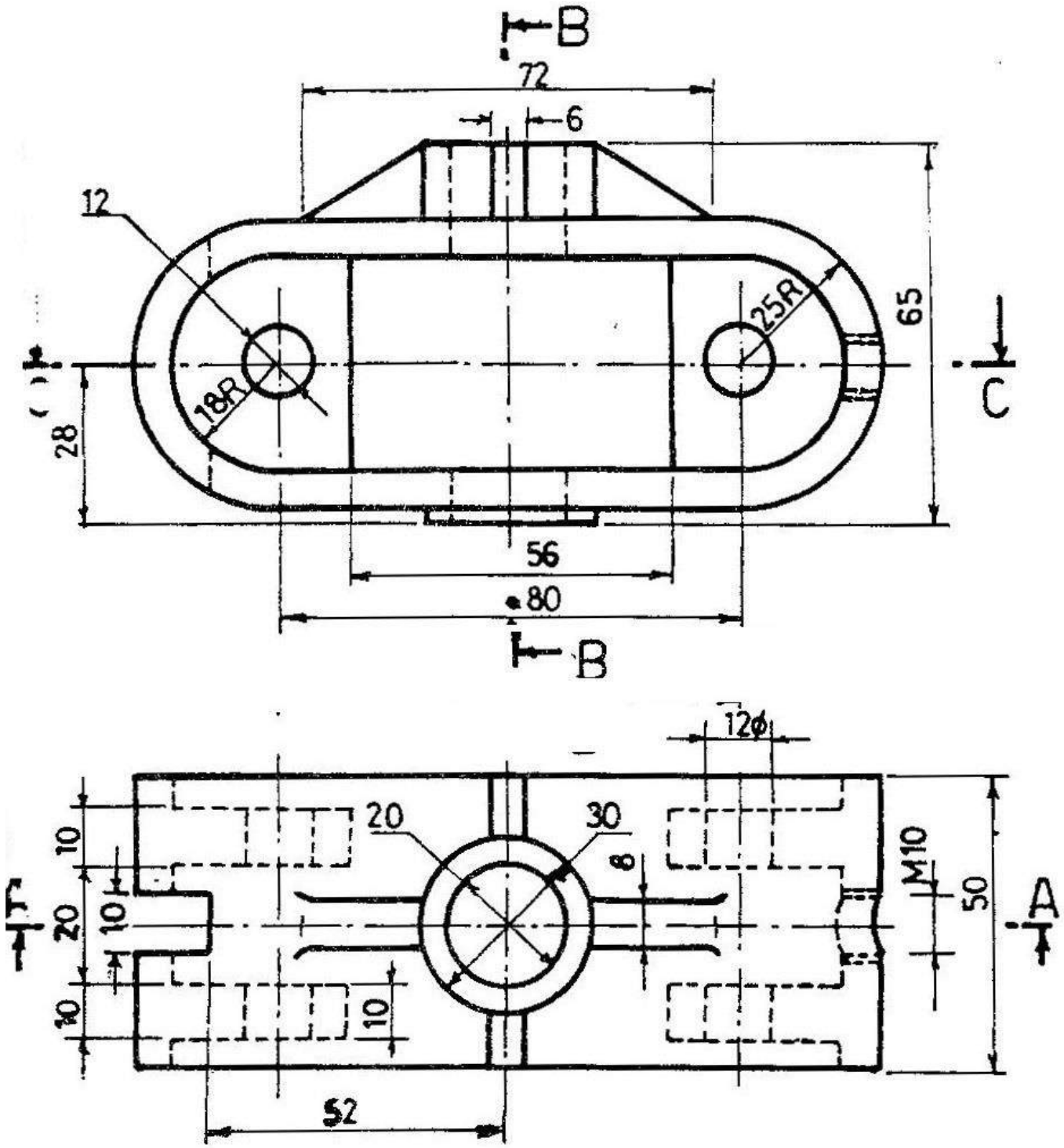
40
63
B
B
ϕ10
10
14
10
104


ø25
ø18
B
45
M12
12
3
30
65
35
120°
B
B
50
ø8
8
15
8
ø10
ø5
B
R10
26
20
13
A
R8
24
ø10
6
32
25
A
6
38
80
120


7R
48
37
28
32
M12
16x45°
13
30
10
20 Dia
46
19
20
5R
6 Dia
40
2 holes 6 Dia
6R
A
7
28
48
13R
16
A
128


2 holes 20 Dia
A
18R
A
28
26
28
28
20
9
9
2x45°
M24
20 Dia
18
10
140

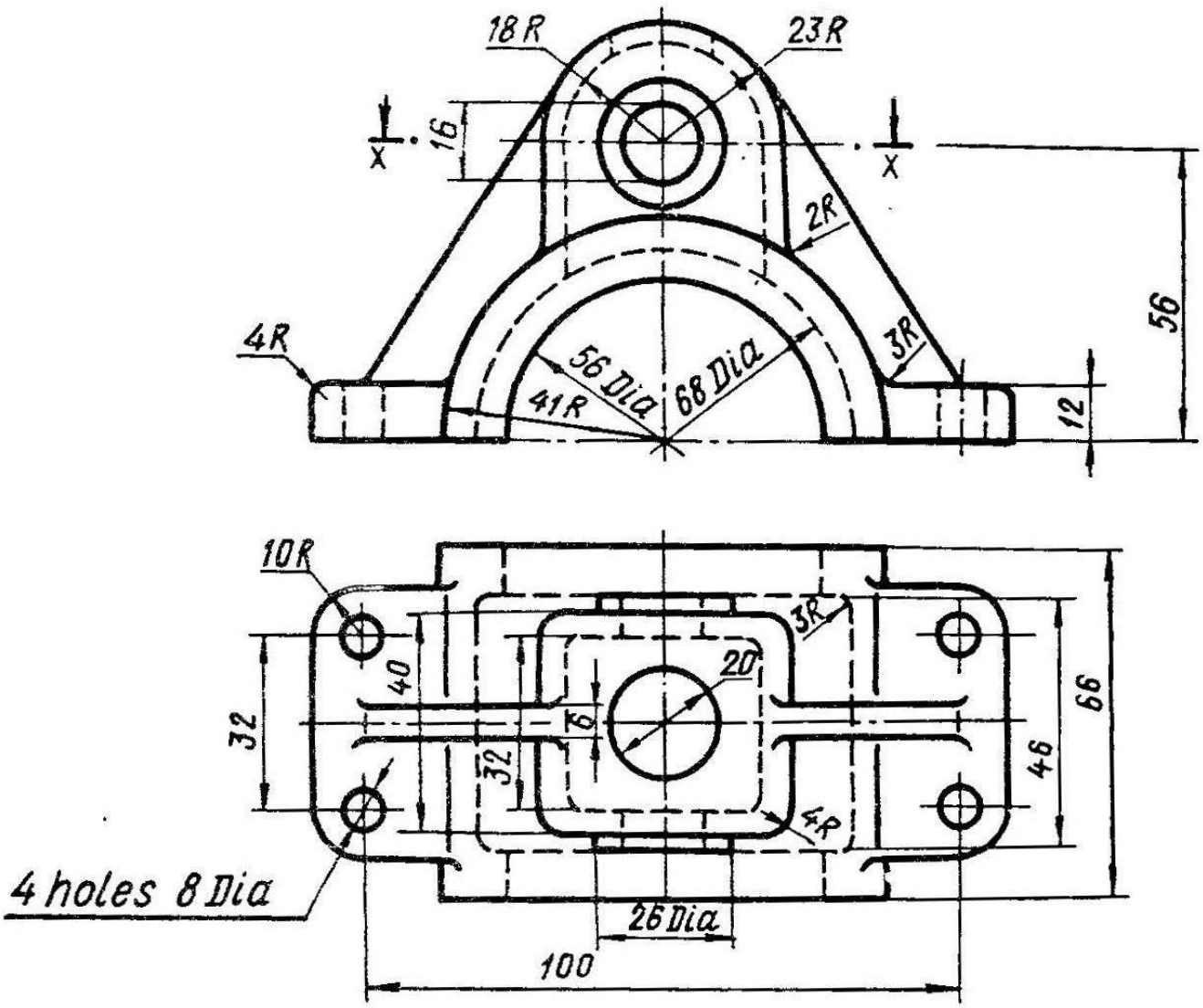

B
72
6
12
25R
65
18R
28
C
56
80
B
12ϕ
20
30
6
10
10
20
10
10
10
M10
50
A
52

DRAW TO SCALE 1:1

1-HALF SEC. FRONT VIEW

2-HALF SEC. SIDE VIEW.

3. SEC. X-X



Learning to use a CAD system is similar to learning a new language. It is necessary to begin with the basic alphabet and learn how to use it correctly and effectively through practice. This will require learning some new concepts and skills as well as learning a different vocabulary. Today, the majority of the Mechanical CAD systems are capable of creating three-dimensional solid models. Nonetheless, all CAD systems create designs using basic geometric entities and many of the constructions used in technical designs are based upon two-dimensional planar geometry. The method and number of operations that are required to accomplish the basic planar constructions are different from one system to another.

In order to become effective and efficient in using a CAD system, we must learn to create geometric entities quickly and accurately. In learning to use a CAD system, **lines** and **circles** are the first two, and perhaps the most important two, geometric entities that one should master the skills of creating and modifying. Straight lines and circles are used in almost all technical designs. In examining the different types of planar geometric entities, the importance of lines and circles becomes obvious. Triangles and polygons are planar figures bounded by straight lines. Ellipses and splines can be constructed by connecting arcs with different radii. As one gains some experience in creating lines and circles, similar procedures can be applied to create other geometric entities. In this chapter, the different ways of creating lines and circles in **AutoCAD 2020** are examined.

Starting up AutoCAD 2020

1. Select the **AutoCAD 2020** option on the Program menu or select the **AutoCAD 2020 icon** on the Desktop. Click Start Drawing to start a new drawing.
   Once the program is loaded into memory, the AutoCAD 2020 main drawing screen will appear on the screen.

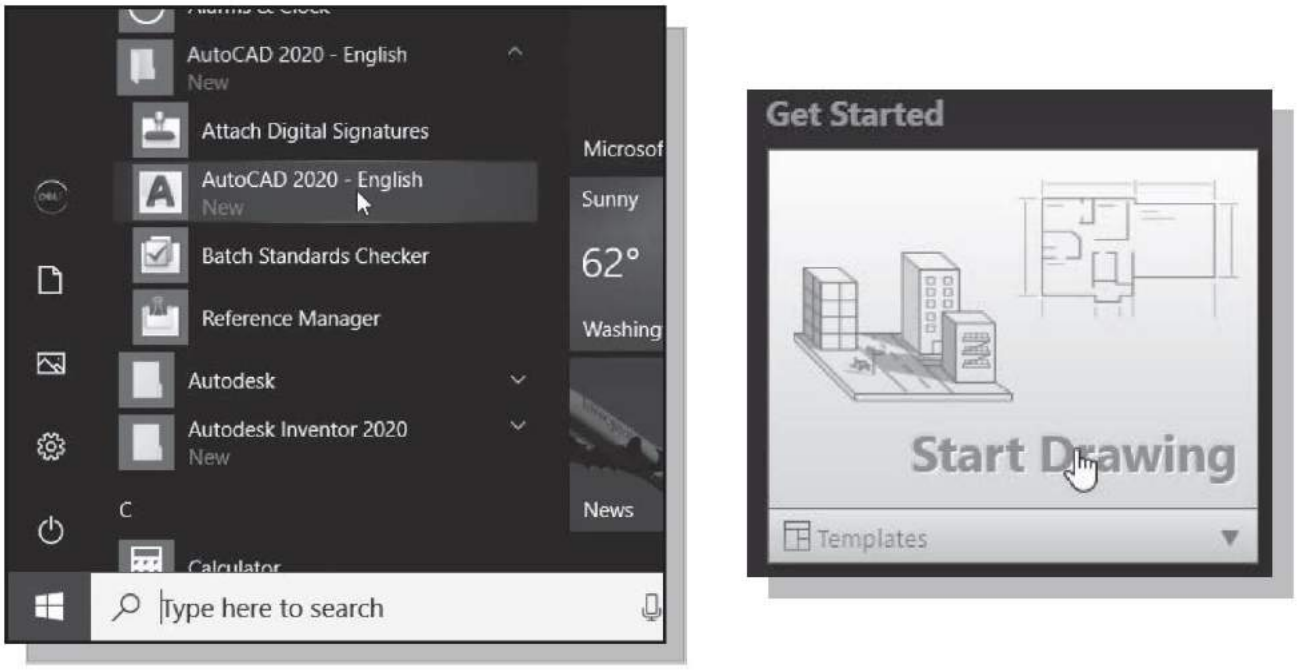


Figure 1

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

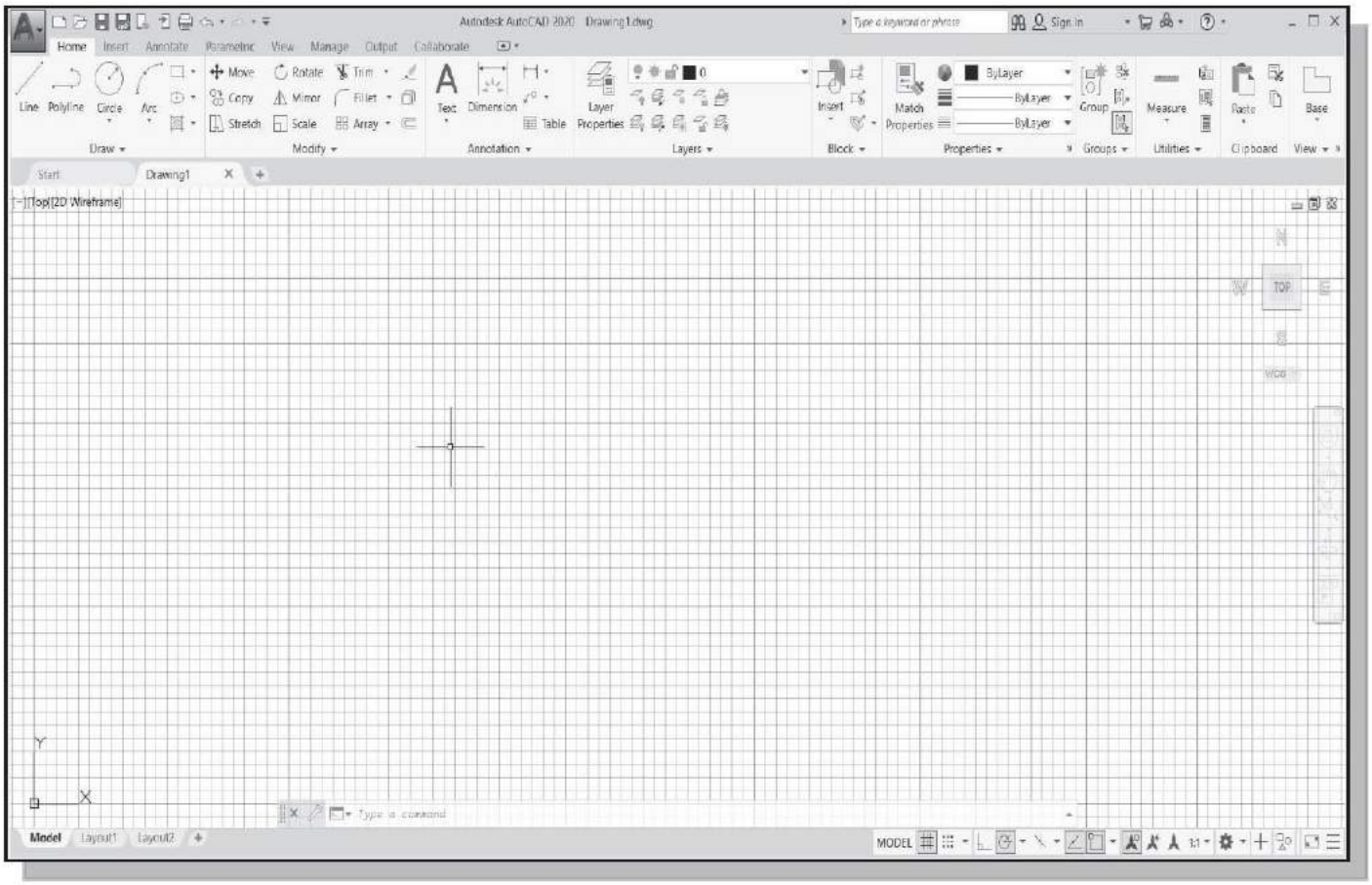


Figure 2

2. If necessary, click on the down-arrow in the *Quick Access bar* and select **Show Menu Bar** to display the AutoCAD **Menu Bar**. *The Menu Bar provides access to all AutoCAD commands.*

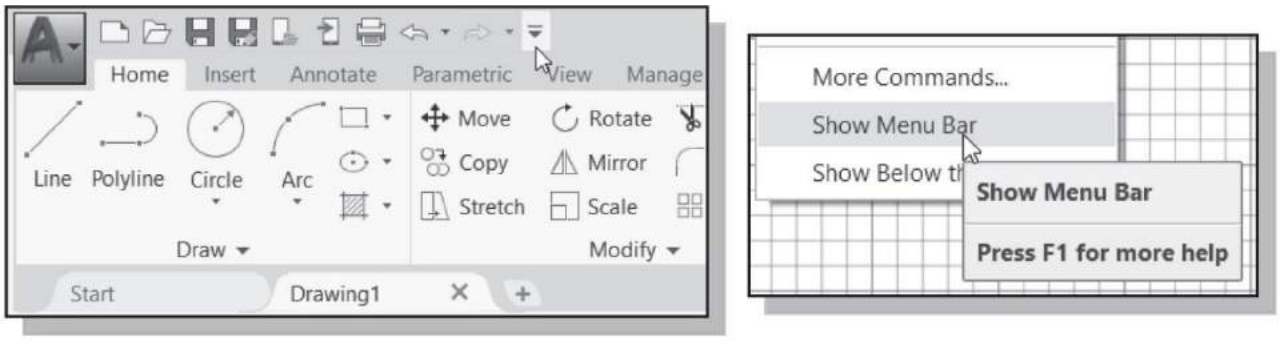


Figure 3

3. To switch on the **AutoCAD Coordinates Display**, use the Customization option at the bottom right corner.

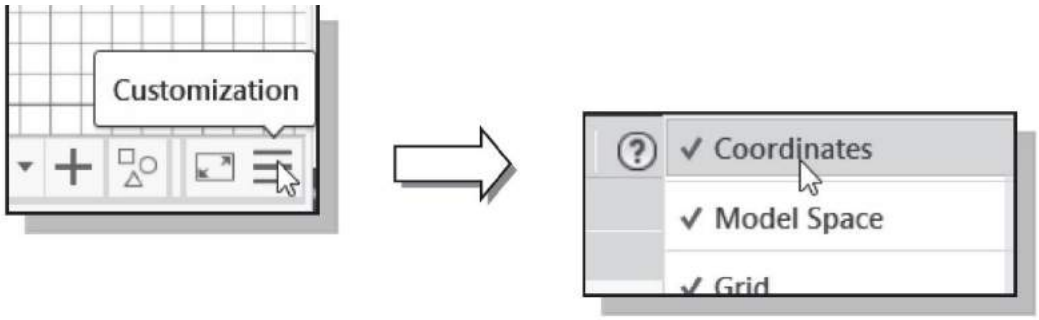


Figure 4

Setting up Drawing Units

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

In order to setup the Drawing units, follow the following steps:

1. In the **Menu Bar**, select **Format > Units** (Figure 5).
2. Click on the *Length Type* option to display the different types of length units available. Confirm the *Length Type* is set to **Decimal** (see Figure 6).

Note: The AutoCAD Menu Bar contains multiple pulldown menus where all of the AutoCAD commands can be accessed. Note that many of the menu items listed in the pull-down menus can also be accessed through the Quick Access toolbar and/or Ribbon panels.

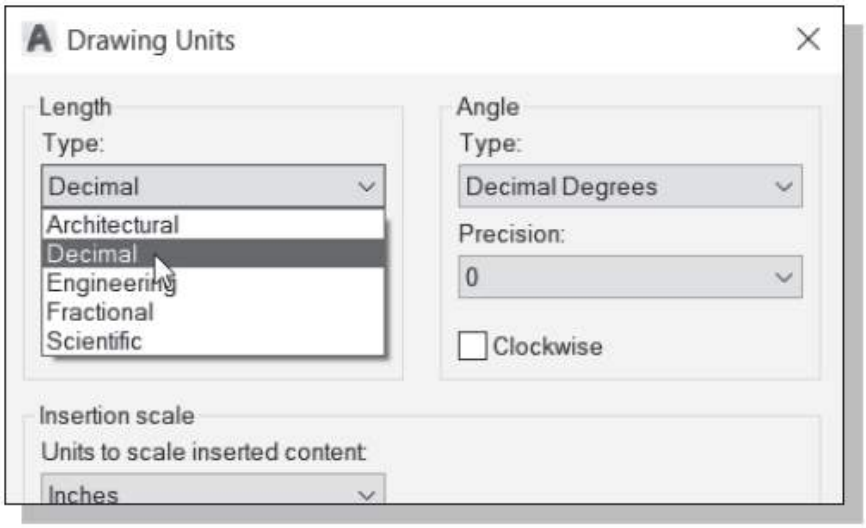


Figure 6

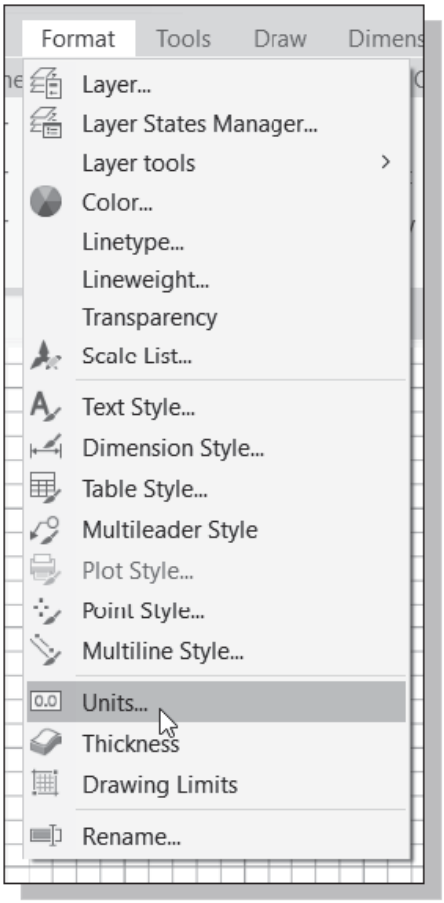


Figure 5

3. Set the **Precision** to two digits after the decimal point as shown in Figure 7.
4. Own your own, examine the other settings that are available.
5. Pick OK to exit the Drawing Units dialog box.

Also note the Insertion Scale section will show the default measurement system, such as the English units, inches.

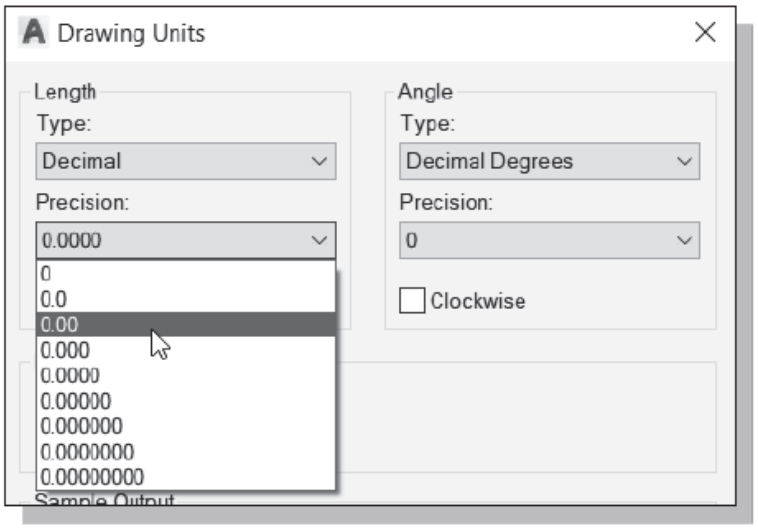


Figure 7

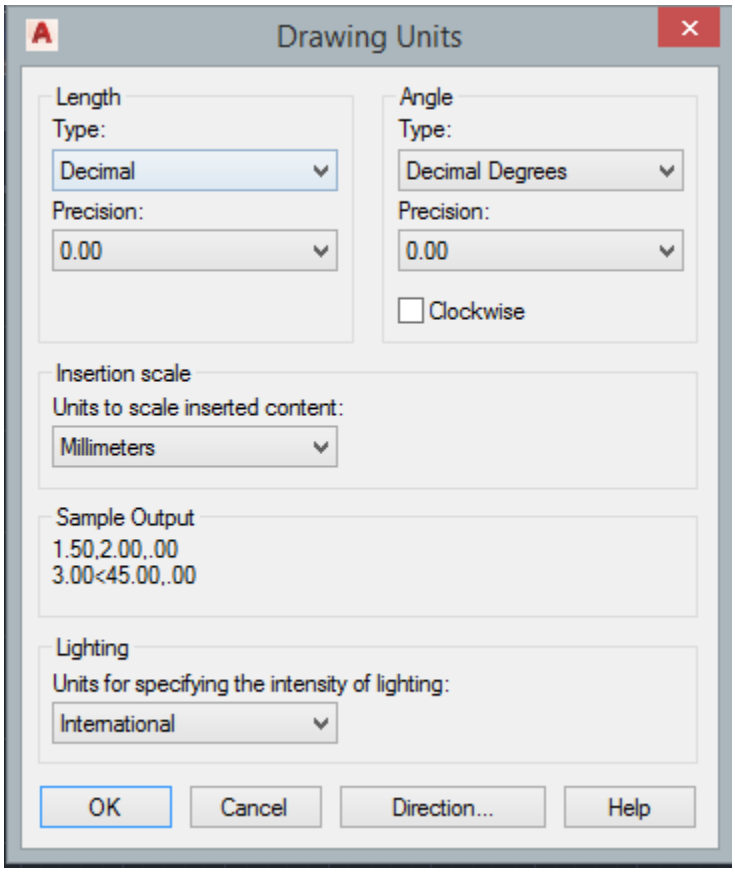


Figure 8

Drawing Area Setup

Next, we will set up the **Drawing Limits** by entering a command in the command prompt area. Setting the Drawing Limits controls the extents of the display of the grid. It also serves as a visual reference that marks the working area. It can also be used to prevent construction outside the grid limits and as a plot option that defines an area to be plotted and/or printed. Note that this setting does not limit the region for geometry construction.

1. In the Menu Bar select: **[Format] > [Drawing Limits]** (see figure 9)
2. In the command prompt area, the message "*Reset Model Space Limits*: *Specify lower left corner or [On/Off] <0.00, 0.00>:"* is displayed. Press the **ENTER** key once to accept the default coordinates <0.00, 0.00>.

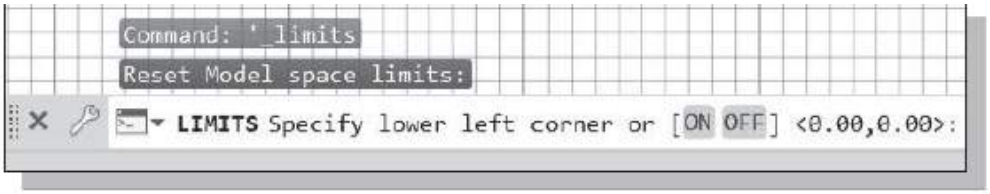


Figure 9

3. In the command prompt area, the message "Specify upper right corner <12.00, 9.00>:" is displayed. Press the ENTER key again to accept the default coordinates <12.00, 9.00> or put in your values.

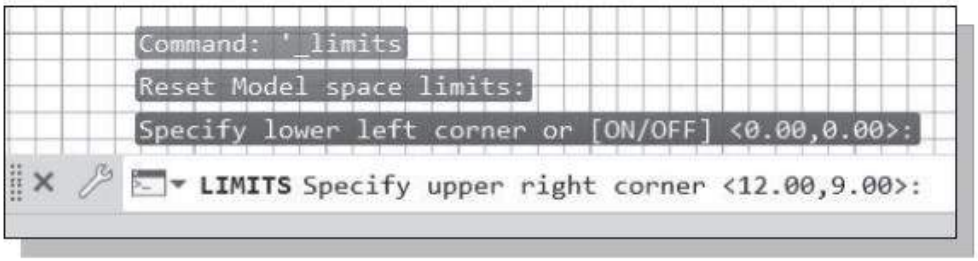


Figure 10

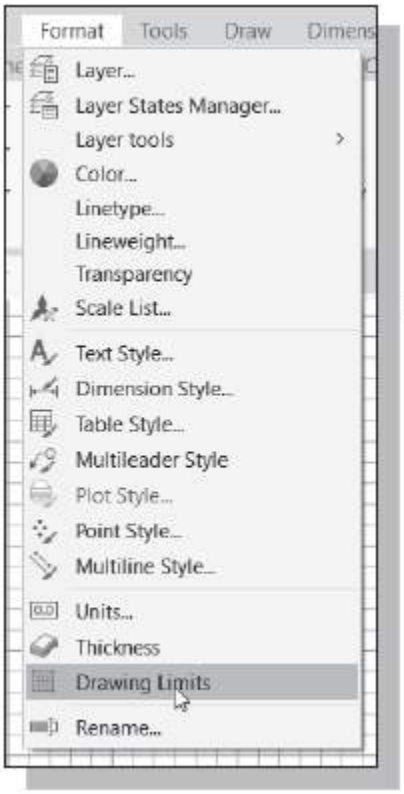


Figure 11

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

5. Inside the Menu Bar area select: **[View] > [Zoom] > [All]**
   The **Zoom All** command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.
6. Move the graphics cursor near the upper-right corner inside the drawing area and note that the display area is updated.
7. Hit the function key [F7] once to turn off the display of the Grid lines.

Note that function key [F7] is a quick key, which can be used to quickly toggle on/off the grid display. Also, note the command prompt area can be positioned to dock below the drawing area or float inside the drawing area.

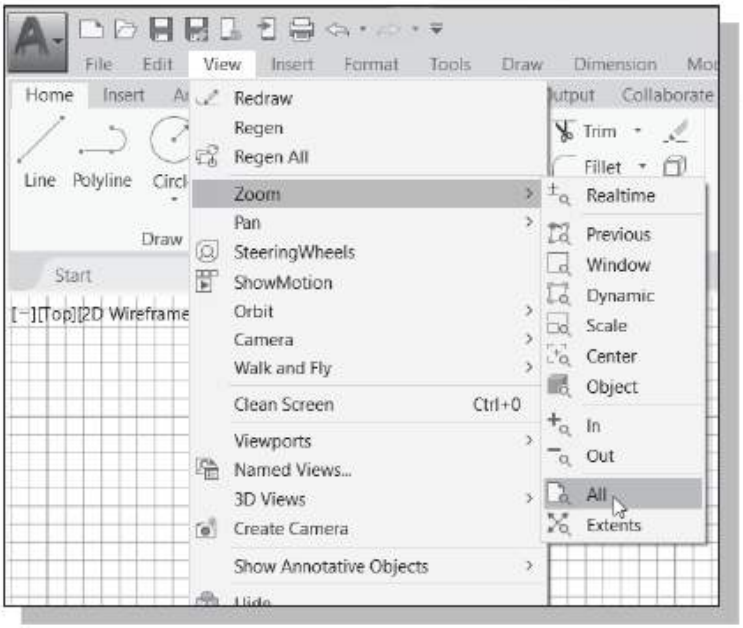


Figure 12

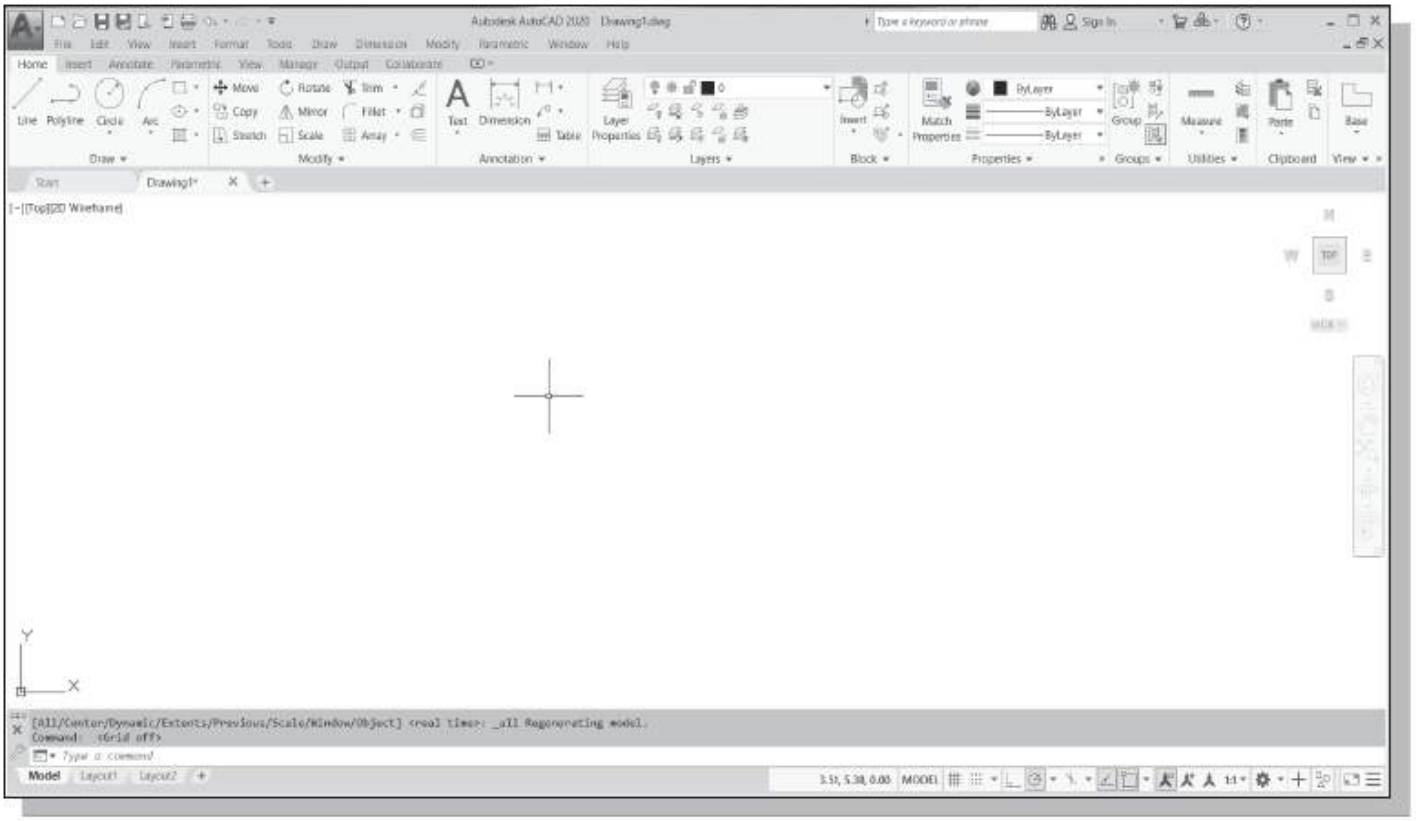


Figure 13

As mentioned in the beginning of the chapter, we must learn to create geometric entities quickly and accurately in order to be effective and efficient in using a CAD system (AutoCAD 2020 here). We are going to examine the most common geometric entities in this software.

Examine the following figure carefully and see the main ribbons which are: Draw, Construction, Detail, modify and Layers. And then see the commands in each ribbon which are going to be presented here

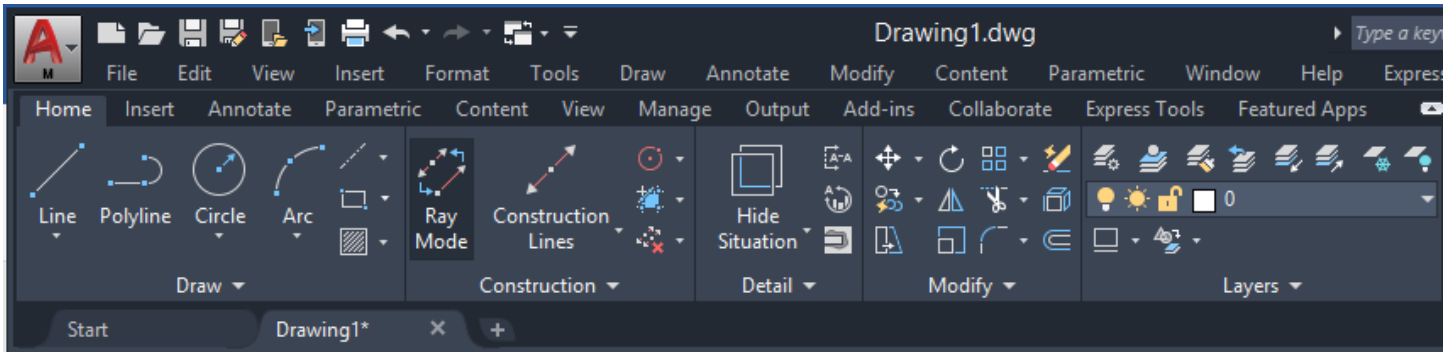
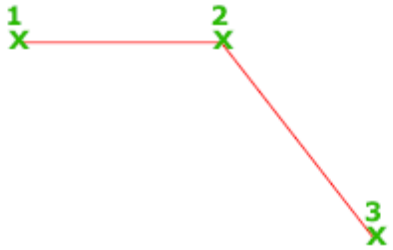


Figure 14

LINE (Command)

Create a series of contiguous line segments. Each segment is a line object that can be edited separately. The following prompts are displayed:



- *Specify first point:* Sets the starting point for the line. Click a point location. With object snaps or grid snap turned on, the points will be placed precisely. You can also enter coordinates. If instead, you press Enter at the prompt, a new line starts from the endpoint of the most recently created line, polyline, or arc. If the most recently created object is an arc, its endpoint defines the starting point of the line. The line is tangent to the arc.

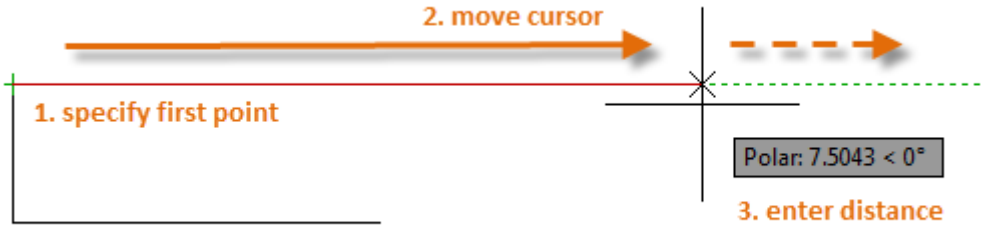


Figure 15

- *Specify next point:* Specifies the endpoint of the line segment. You can also use polar and object snap tracking together with direct distance entry.

Using a Specific Coordinate:

- If dynamic input is on: Type the pound sign (#) followed by the X-value, a comma, then the Y-value, for example #40, 60.
- If dynamic input is off: Type the X value, a comma, then the Y value, for example 40, 60.

Note: When dynamic input is on, relative coordinates are the default. When dynamic input is off, absolute coordinates are the default. Press F12 to turn dynamic input on or off.

Using a Relative Coordinate:

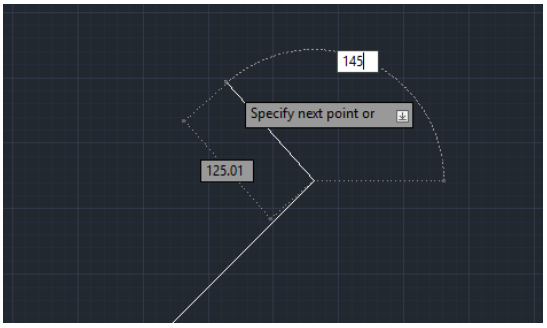
A relative coordinate specifies the distance and direction from the previous coordinate.

- If dynamic input is on: Type the X-value, a comma, then the Y-value, for example 40, 60.
- If dynamic input is off: Type the sign (@) followed by the X-value, a comma, then the Y-value, for example @40, 60.

While drawing lines using the line command, there are few important tricks, after specifying the first point, you can make perfectly horizontal or vertical lines using the ***Orthomode***, see figure 16, it is the icon with perpendicular lines on it, the one just next to the MODEL word

F8 is the shortcut of activating or deactivating this mode.

If you need to draw an inclined line, the Orthomode has to be disabled and after putting the first point, tap SHIFT key and type in your angle measured from the horizontal line and counterclockwise



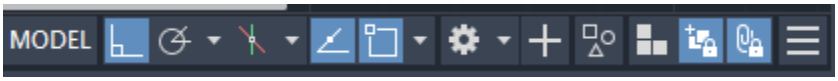


Figure 16

Polyline (PLINE Command)

Create a 2D polyline, a single object that is composed of line and arc segments. The following prompts are displayed:

- *Specify start point:* Sets the starting point for the polyline. A temporary plus-shaped marker displays at the first point. And pressing Enter starts a new polyline from the last endpoint specified in creating a polyline, line, or arc.

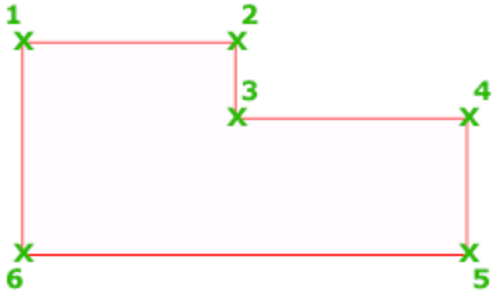


Figure 17

- *Specify next point:* If you specify a second point, you create straight segments. Or you can choose some other entities from the command window below

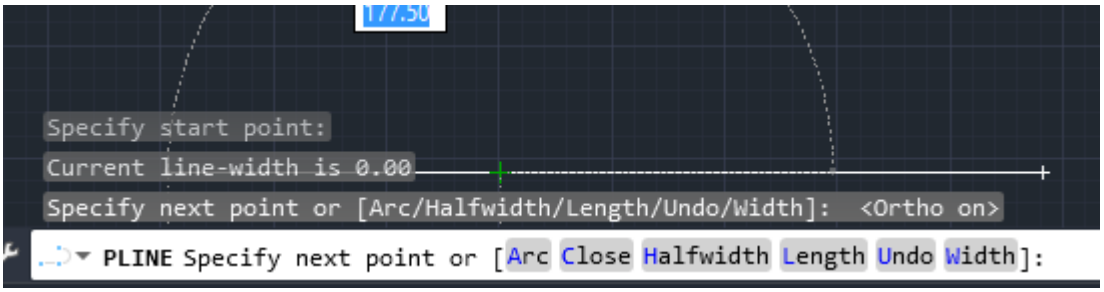


Figure 18

Prompts Common to Line and Arc Segments

1. Arc: begins creating arc segments tangent to the previous segment.
2. Close: connects the first and last segments to create a closed polyline.
3. Halfwidth: specifies the width from the center of a wide segment to an edge.
4. Length: creates a segment of a specified length at the same angle as the previous segment. If the previous segment is an arc, the new line segment is tangent to that arc segment.
5. Undo: removes the most recently added segment.
6. Width: specifies the width of the next segment.

Some things to keep in mind when defining the half-width or width of a polyline.

- The starting width becomes the default ending width.
- The ending width becomes the uniform width for all subsequent segments until you change the width again.
- The starting and ending points of wide line segments are at the centerline of the segment.

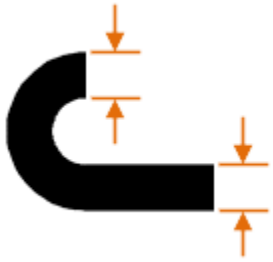


Figure 19 polyline width

There are some prompts that are shown only when arcs are being drawn:

- Endpoint of arc: completes an arc segment. The arc segment is tangent to the previous segment of the polyline.
- Angle: specifies the included angle of the arc segment from the start point. Entering a positive number creates counterclockwise arc segments. Entering a negative number creates clockwise arc segments.
- Line: switches from drawing arc segments to drawing straight segments.

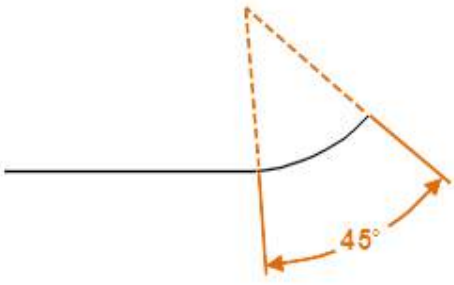


Figure 20

Circles (CIRCLE Command)

To create circles, you can specify various combinations of center, radius, diameter, points on the circumference, and points on other objects. You can create circles in several ways. The default method is to specify the center and the radius. There are more 5 ways of creating circles, these are presented here.

1. ***Radius:*** Creates a circle based on a center point and a radius. Enter a value, or specify a point (see Fig. 21)
2. ***Diameter:*** Creates a circle based on a center point and a diameter. Enter a value, or specify a point (see Fig. 22)
3. ***2P (Two Points):*** Creates a circle based on two endpoints of the diameter (Fig. 23).
4. ***3P (Three Points):*** Creates a circle based on three points on the circumference (Fig. 24).
5. ***TTR (Tangent, Tangent, Radius):*** Creates a circle with a specified radius and tangent to two objects (Fig. 25). Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.
6. ***Tan, Tan, Tan:*** Creates a circle tangent to three objects (Fig. 26).

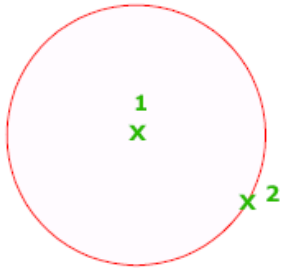


Figure 21

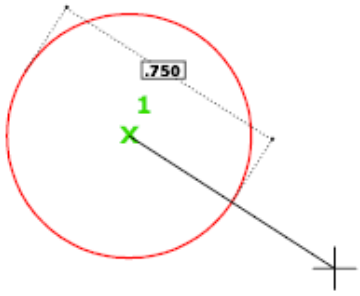


Figure 22

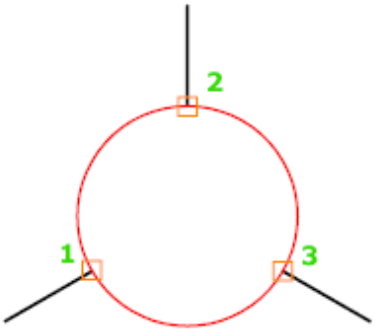


Figure 23

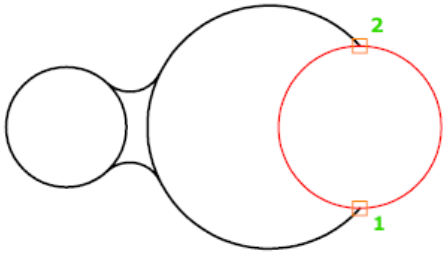


Figure 24

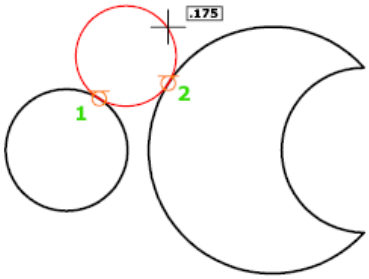


Figure 25

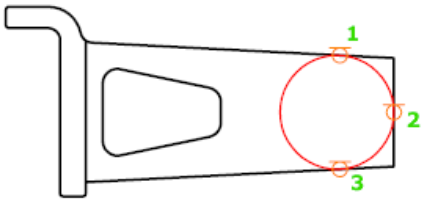


Figure 26

Arcs (ARC Command)

Create arcs by specifying various combinations of center, endpoint, start point, radius, angle, chord length, and direction values. Arcs are drawn in a counterclockwise direction by default. Hold down the Ctrl key as you drag to draw in a clockwise direction.

1. Draw Arcs by Specifying Three Points: You can create an arc by specifying three points. In the following example, the start point of the arc snaps to the endpoint of a line. The second point of the arc snaps to the middle circle in the illustration (Fig. 27).
2. Draw Arcs by Specifying Start, Center, End: You can create an arc using a start point, center, and a third point that determines the endpoint.
   The distance between the start point and the center determines the radius. The endpoint is determined by a line from the center that passes through the third point, see Fig. 28.
   Using different options, you can specify either the start point first or the center point first.

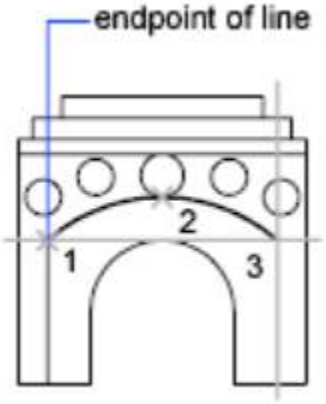


Figure 27

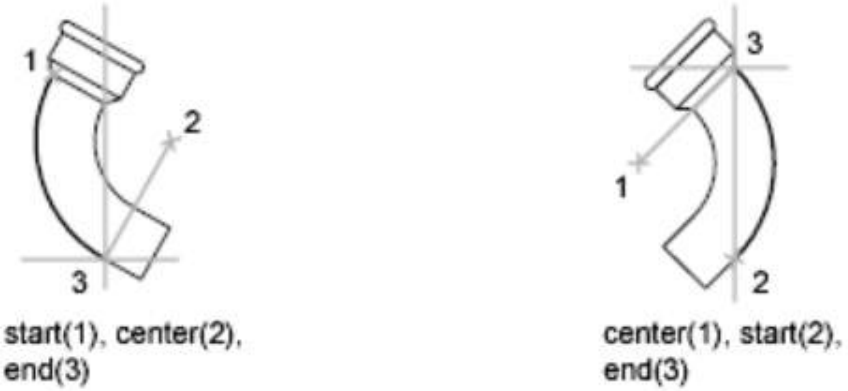


Figure 28

3. Draw Arcs by Specifying Start, Center, Angle: You can create an arc using a start point, center, and an included angle. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying an included angle that uses the center of the arc as the vertex.
   Using different options, you can specify either the start point first or the center point first.

There are more ways to draw arcs in AutoCAD, study them on yourself.

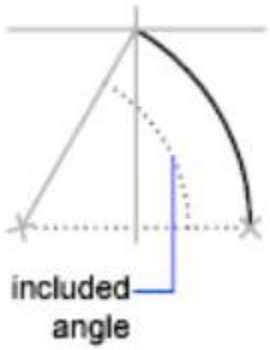


Figure 29

Rectangles (RECTANG Command)

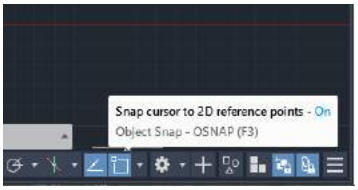
You can create rectangles by various ways:

- Using corners.
- Using a corner with the midpoint of the base.
- Using a corner with the midpoint of the height.
- Using a corner and the center of the rectangle.

You can simply go to the DRAW ribbon and put the cursor on the RECTSNGLE icon, and all these ways will show up in the drop-down menu and how to exactly use every way with the required dimensions. Also, you can see that you can draw squares using the same icon.

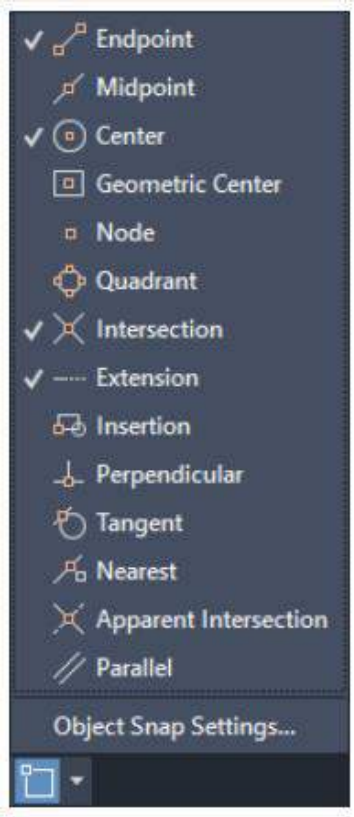
These were the main and commonly used **DRAW** commands in 2D, in the next section we are going to present the most common **MODIFY** commands which are: ***Move, Mirror, Array, Trim, Fillet, Explode, Offset*** and ***Scale.***

Before explaining the MODIFY commands, let us explain a very powerful feature in AutoCAD, ***Object snap.***



Object snaps provide a way to specify precise locations on objects whenever you are prompted for a point within a command. For example, you can use object snaps to create a line from the center of a circle to the midpoint of another line (Fig. 30).

If you need to use one or more object snaps repeatedly, you can turn on running object snaps, which persist through all subsequent commands. For example, you might set Endpoint, Midpoint, and Center as running object snaps.



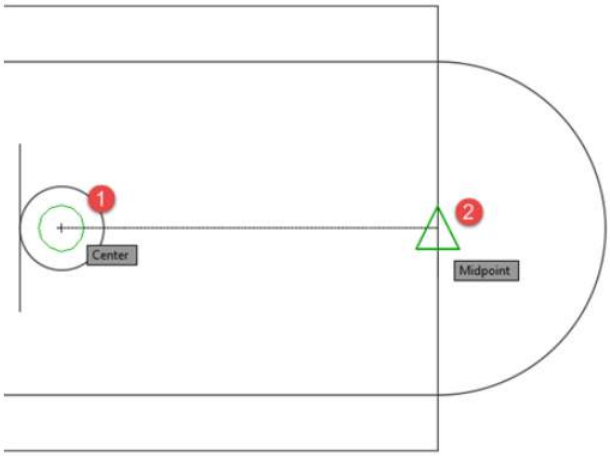


Figure 30

Move (MOVE Command)

You can move objects at a specified distance and direction from the originals. Use coordinates, grid snap, object snaps, and other tools to move objects with precision.

Specify Distance with Two Points: Move an object using the distance and direction specified by a base point followed by a second point. In this example, you move the block representing a window. Select the object to be moved (1). Specify the base point for the move (2) followed by a second point (3). The object is moved the distance and direction of point 2 to point 3.

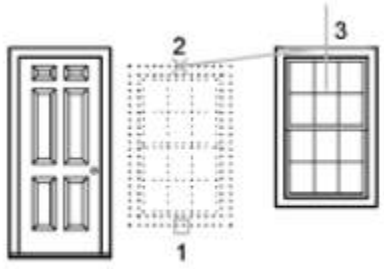


Figure 31

To Move an Object Using a Displacement

1. Click Home tab > Modify panel > Move
2. Select the object to move.
3. Enter the displacement in the form of a Cartesian, polar, cylindrical, or spherical coordinate value. Do not include the @ sign, because a relative coordinate is assumed.
4. At the prompt for the second point, press Enter.

The coordinate values are used as a relative displacement rather than the location of a base point. The selected objects are moved to a new location determined by the relative coordinate values you enter.

Mirror (MIRROR Command)

Creates a mirrored copy of selected objects. You can create objects that represent half of a drawing, select them, and mirror them across a specified line to create the other half.

To Mirror Objects in 2D

1. Click Home tab > Modify panel > Mirror
2. Select the objects to mirror.
3. Specify the first point of the mirror line.
4. Specify the second point.
5. Press Enter to retain the original objects, or enter y to erase them.

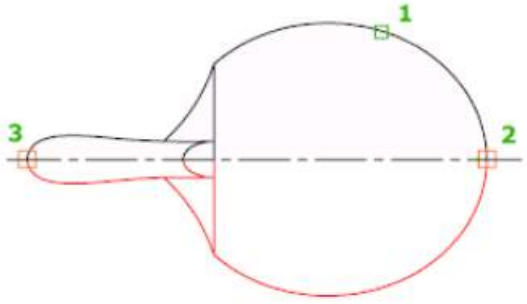
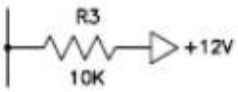
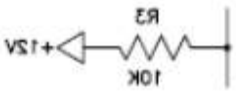


Figure 32

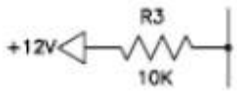
By default, when you mirror a text object, the direction of the text is not changed. Set the MIRRTEXT system variable to 1 if you do want the text to be reversed.



before mirroring



after mirroring
MIRRTEXT=1



after mirroring
MIRRTEXT=0

Array (ARRAY Command)

You can create copies of selected objects to be arranged in a pattern called an array. After you select the objects that you want to duplicate, which are called the source objects, you choose the arrangement pattern. There are three types of arrays: Rectangular, Path and Polar.

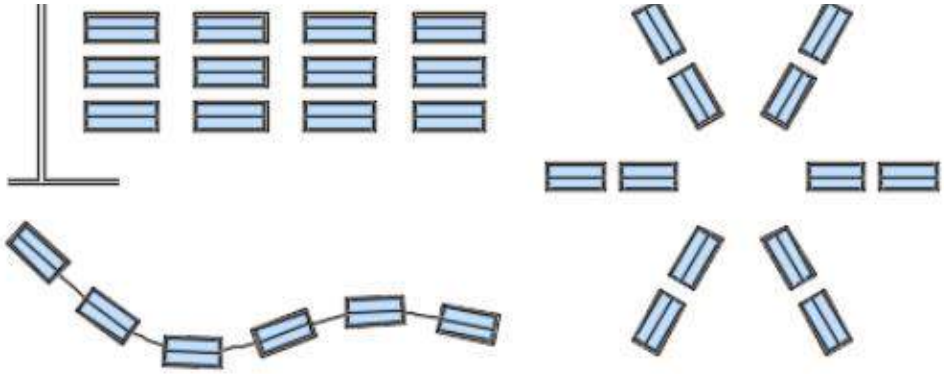


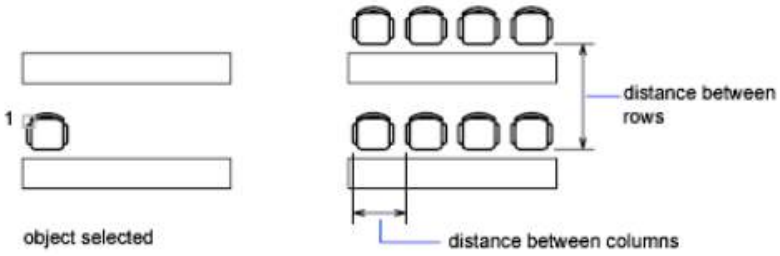
Figure 33

Each element of the array is called an array item, which can be composed of several objects. You can also specify a block to be the source object of an array.

Note: With a path array, you also need a line, polyline, 3D polyline, spline, helix, arc, circle, or ellipse to serve as the path.

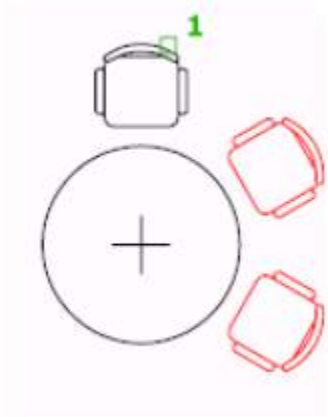
1. **Rectangular Array:**

Creates an array of rows and columns of copies of the selected objects. The selected object, or cornerstone element, is assumed to be in the lower-left corner, and generates the array up and to the right.



- Number of rows (---): Specifies the number of rows using a nonzero integer. If you specify one row, you must specify more than one column and vice versa.
- Number of columns (|||): Specifies the number of columns.
- Distance between rows or specify unit cell: Specifies the distance between rows, including the length of the object to be arrayed. To add rows downward, specify a negative value for the distance between rows. To specify the distance between rows and columns at the same time, specify two sets of coordinates or drawing locations that represent the opposite corners of a rectangle.
- Distance between columns (|||): Specifies the distance between columns. To add columns to the left, specify a negative value for the distance between columns.

2. Polar Array:



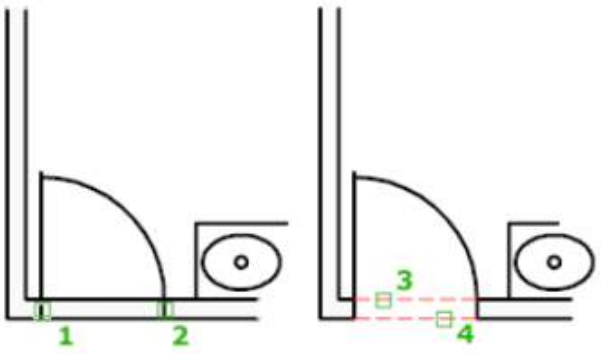
Creates an array by copying the selected objects around a specified center point. In a polar array, the reference point of the last object in the selection set is used for all objects. If you defined the selection set by using window or crossing selection, the last object in the selection set is arbitrary. Removing an object from the selection set and adding it back forces that object to be the last object selected. You can also make the selection set into a block and replicate it.

The prompt values you enter determine whether the array configuration is based on the number of items or the angle that is filled by the arrayed items.

- Center point of array: Specifies the central location for the array.
- Base: Specifies a new reference (base) point relative to the selected objects that will remain at a constant distance from the center point of the array as the objects are arrayed.
- Number of items in the array: Specifies the total number of items to be arrayed around the center point. If you do not enter a value, the array is based on the Angle to fill and Angle Between Items values.
- Angle to fill (+=ccw, -=cw) (+=ccw, -=cw): Specifies the included angle between the base points of the first and last arrayed objects. Enter a positive number for a counterclockwise rotation or a negative number for a clockwise rotation.
- Angle between items (+=ccw, -=cw) (+=ccw, -=cw): Specifies the included angle between objects, based on the center point of the array and the base points of the arrayed objects. Enter a positive number for a counterclockwise rotation or a negative number for a clockwise rotation.

Note: To display this option, press Enter at the Number of items prompt or enter 0 at the Angle to fill prompt.

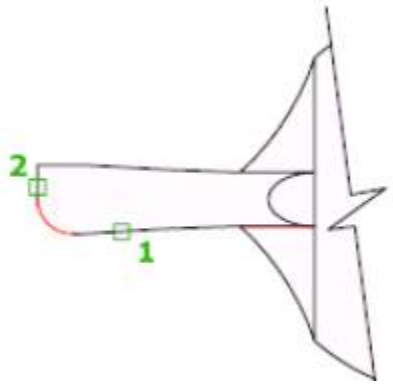
Trim (TRIM Command)



Trims objects to meet the edges of other objects. To trim objects, select the boundaries and press Enter. Then select the objects that you want to trim. To use all objects as boundaries, press Enter at the first Select Objects prompt.

Fillet (FILLET Command)

Rounds or fillets the edges of two 2D objects or the adjacent faces of a 3D solid. A round or fillet is an arc that is created tangent between two 2D objects. In this example, an arc is created tangent to the selected lines, which are trimmed to meet the endpoints of the arc.



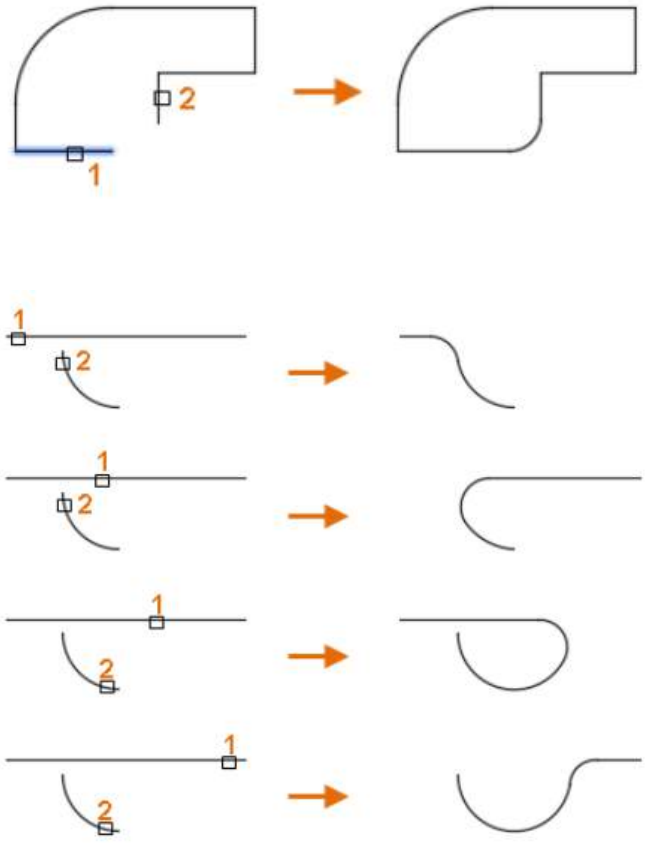
- **Create 2D Fillets:** A round or fillet can be created between two objects of the same or different object types: 2D polylines, arcs, circles, ellipses, elliptical arcs, lines, rays, splines, and x-lines. If the two selected objects are on the same layer, the arc defined is created on that layer. Otherwise, the arc is created on the current layer. The layer affects object properties including color and line type. The following prompts are displayed when creating a 2D fillet.
    1. First Object: Select the first of two objects or the first line segment of a 2D polyline to define the fillet.
    2. Second object or shift-select to apply corner: Select the second object or line segment of a 2D polyline to define the fillet.

You can also hold down the Shift key before selecting the second object or line segment of a 2D polyline to extend or trim the selected objects to form a sharp corner. While Shift is held down, a temporary value of zero is assigned to the current fillet radius value.

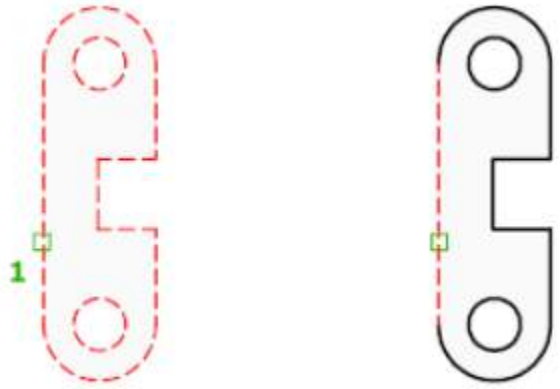
If the selected objects are straight line segments of a 2D polyline, the line segments can be adjacent to each other or separated by one other segment. When the selected segments are separated by a segment, the segment that separates them is removed and replaced with the fillet.

The direction and length of the arc created is determined by the points picked to select the objects. Always select an object closest to where you want the endpoints of the fillet to be drawn.

When a circle is selected, the circle is not trimmed; the fillet drawn meets the circle smoothly.



Explode (EXPLODE Command)



Have you wondered about the difference of making a rectangle using LINE, RECTANGLE or POLYLINE commands??

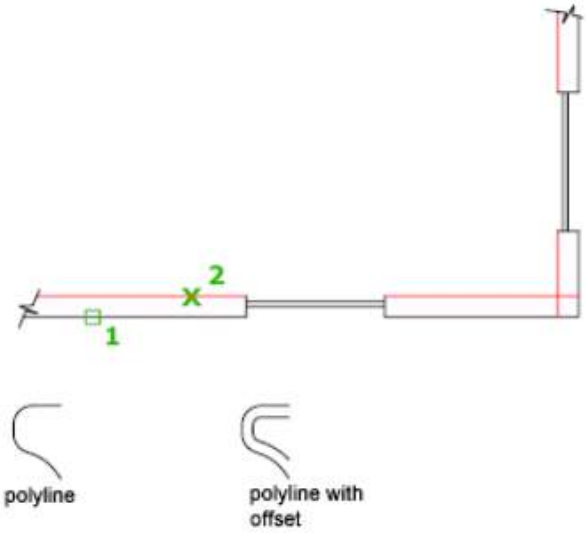
The difference or how the rectangle is treated, if it was drawn using lines, each line can treated independently but if it was drawn by the other 2 ways, the hole rectangle is treated as one unit.

Here come the benefit of EXPLODE, it breaks a compound object into its component objects. Explodes a compound object when you want to modify its components separately. Objects that can be exploded include blocks, polylines, and regions, among others.

The color, line-type, and line-weight of any exploded object might change. Other results differ depending on the type of compound object you're exploding. See the following list of objects that can be exploded and the results for each.

To explode objects and change their properties at the same time, use XPLODE.

Offset (OFFSET Command)

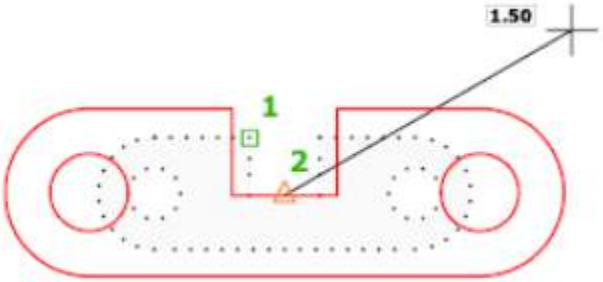


Creates concentric circles, parallel lines, and parallel curves. You can offset an object at a specified distance or through a point. After you offset objects, you can trim and extend them as an efficient method to create drawings containing many parallel lines and curves.

- **Specify an Offset Distance**
  1. Click Home tab > Modify panel > Offset.
  2. Specify the offset distance. You can either enter a value or use the pointing device to determine a distance with two points.
  3. Select the object to offset.
  4. Specify a point to indicate whether the object is to be offset inside or outside of the original object.
- **Specify a Pass-Through Point**
  1. Click Home tab > Modify panel > Offset.
  2. Enter t (Through).
  3. Select the object to offset.
  4. Specify a point through which the offset object will pass.

Scale (SCALE Command)

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling. To scale an object, specify a base point and a scale factor. The base point acts as the center of the scaling operation and remains stationary. A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.

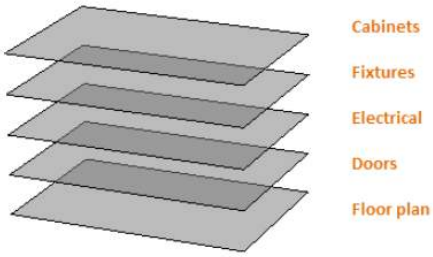


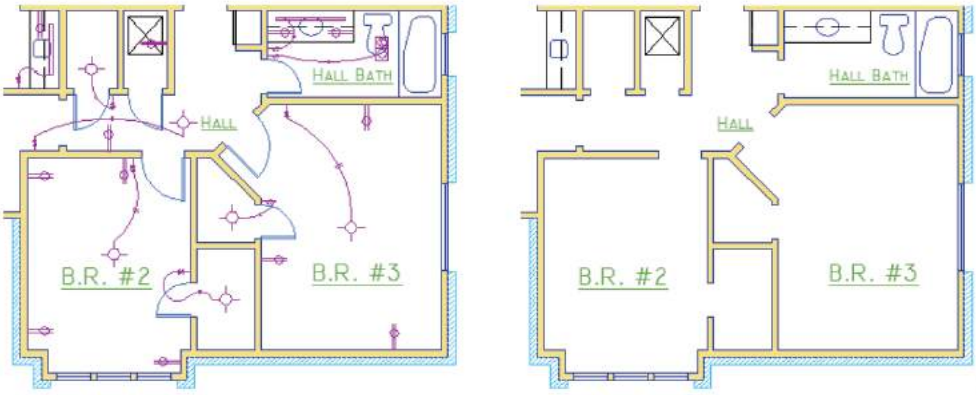
The following prompts are displayed:

- Select objects: Specifies which objects you want to resize.
- Base point: Specify a base point for the scale operation.
  The base point you specify identifies the point that remains in the same location as the selected objects change size (and thus move away from the stationary base point).
  Note: When you use the SCALE command with annotative objects, the position or location of the object is scaled relative to the base point of the scale operation, but the size of the object is not changed.
- Scale Factor: Multiplies the dimensions of the selected objects by the specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects. You can also drag the cursor to make the object larger or smaller.
- Copy: Creates a copy of the selected objects for scaling.
- Reference: Scales the selected objects based on a reference length and a specified new length.

Layers

Organize your drawing by assigning objects to layers. When a drawing becomes visually complex, you can hide objects that you currently do not need to see. In the drawing below, the doors and electrical wiring were temporarily hidden by hiding their layers.



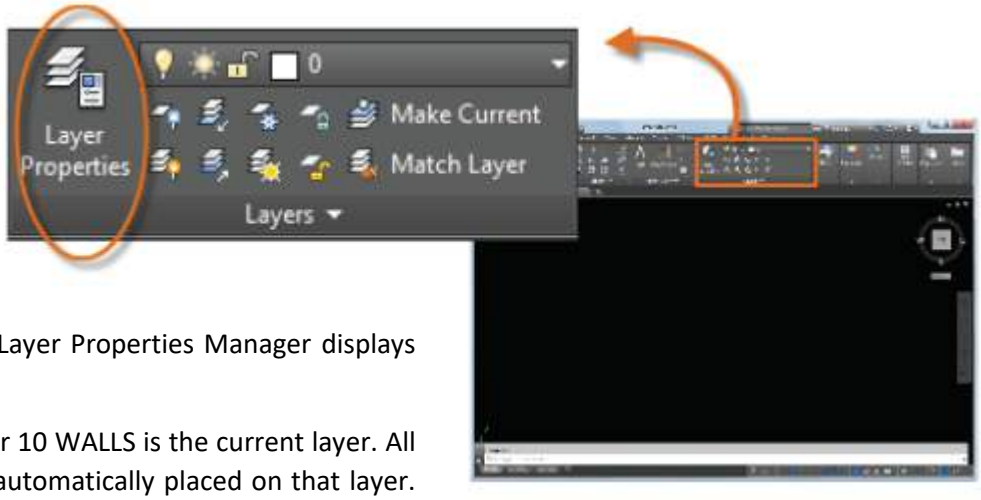


You gain this level of control by organizing the objects in your drawing on layers that are associated with a specific function or purpose. With layers, you can:

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

Layer Controls

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

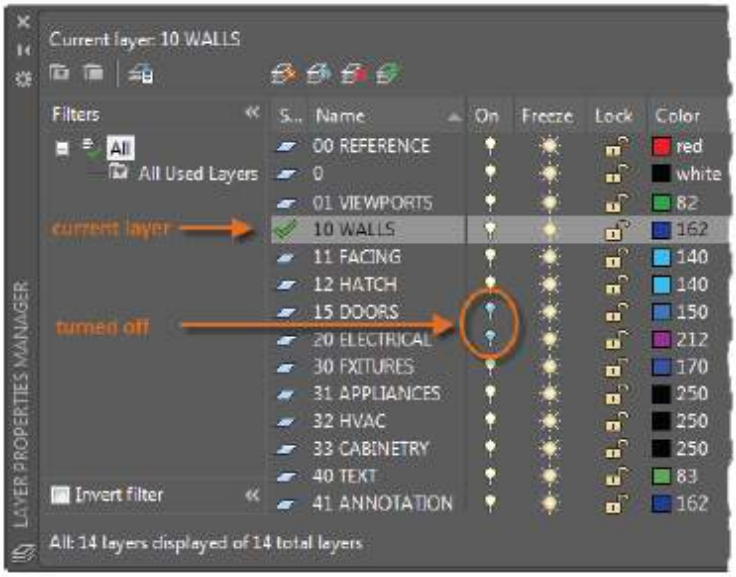


Here's what the Layer Properties Manager displays for this drawing.

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

In the column labeled on, notice that the light bulb icons for two layers are dark. This indicates that these layers were turned off to hide the doors and electrical wiring in the floor plan. Notice that each layer name starts with a two-digit number. This convention makes it easy to control the order of the layers because their order does not depend on the alphabet.

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



Practical Recommendations

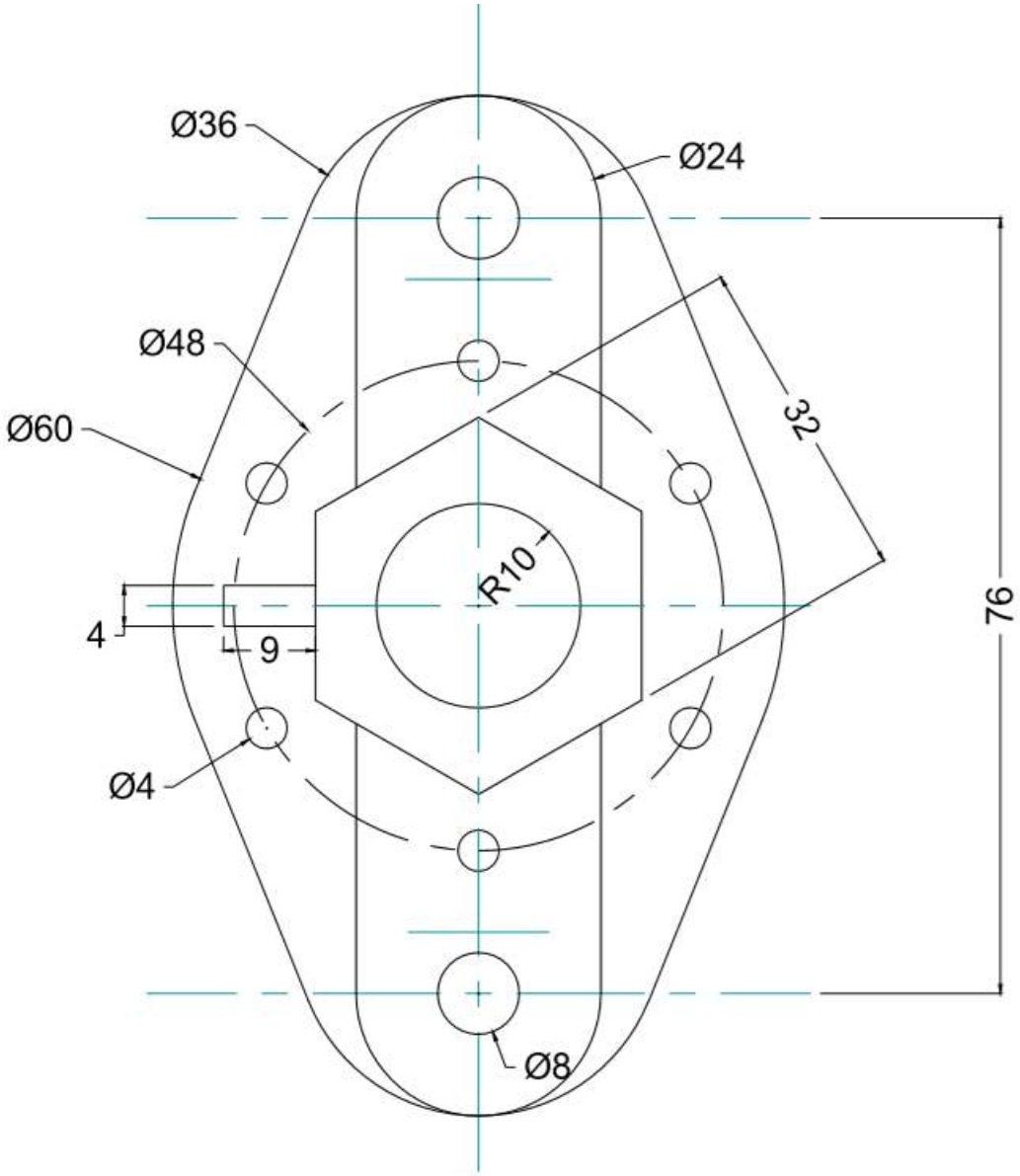
- Layer 0 is the default layer that exists in all drawings and has some esoteric properties. Instead of using this layer, it's best to create your own layers with meaningful names.
- Any drawing that contains at least one dimension object automatically includes a reserved layer named Defpoints.
- Create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

Layer Settings

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

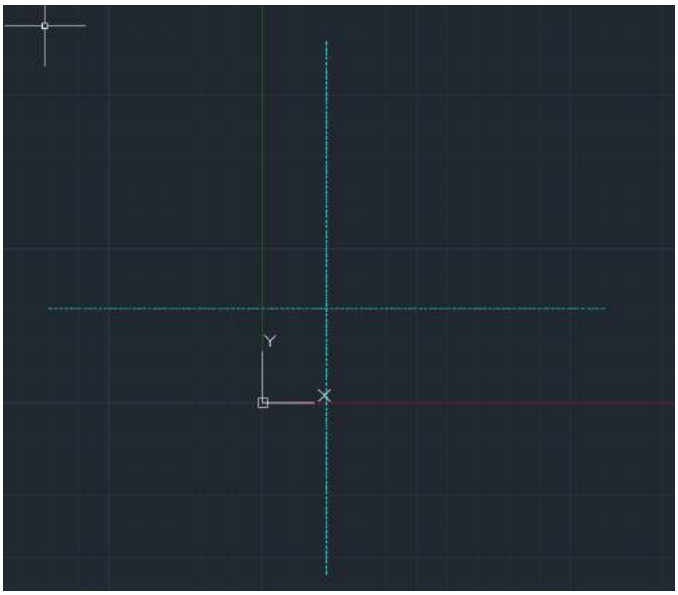
COMMONLY USER LAYER SETTINGS		
	Turn Off Layers: This will help reduce the visual complexity of your drawing while you work.	
	Freeze Layers: Freeze layers that you do not need to access for a while. Freezing layers is similar to turning them off, but improves performance in very large drawings.	
	Lock Layers: Locking layers prevents accidental changes to the objects on those layers. Also, the objects on locked layers appear faded, which helps reduce the visual complexity of your drawing.	
	Set Default Properties: You can set the default properties for each layer, including color, line-type, line weight, and transparency. New objects that you create will use these properties unless you override them. Overriding layer properties is explained later in this topic.	

Solved Example

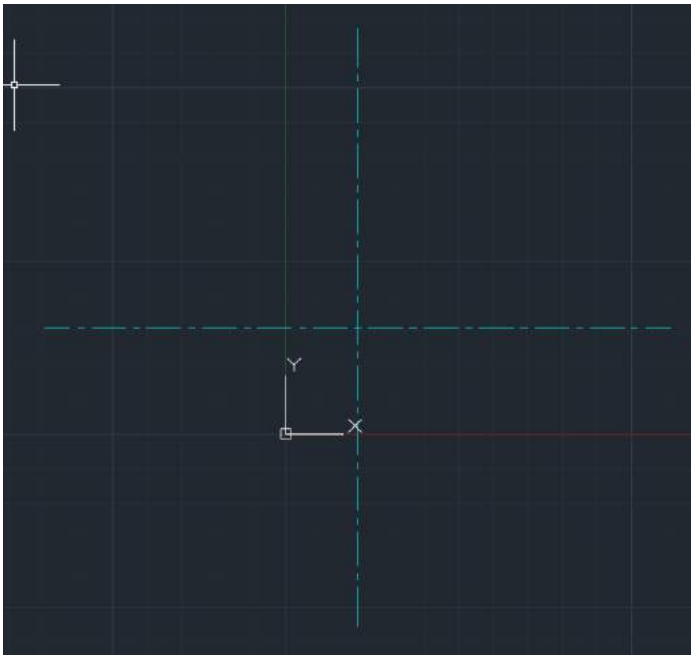


When you are trying to use any CAD software like AutoCAD, Solidworks or any other software, there is now optimum way to draw your models or designs as everyone has his unique way of thinking, it is like programming. However, there are some basics to follow: start your drawings with centerlines and circles, start from the features in the center of the drawing.

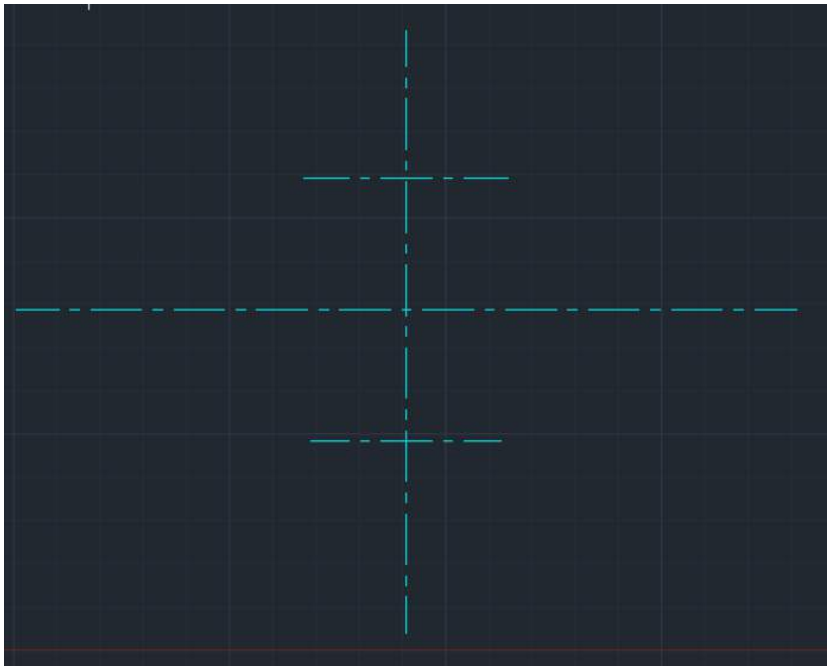
1. Start with the vertical centerline in the middle of the drawing, and the horizontal centerline.



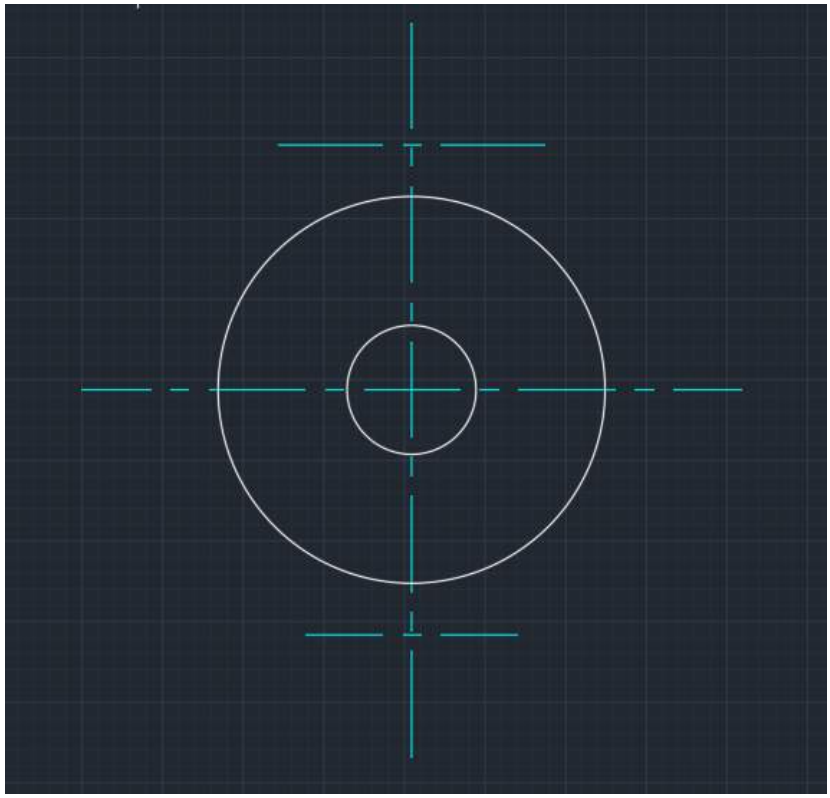
When the centerlines looks like they are dotted, modify the centerlines layer to make the centerlines more realistic. Type in the commands box (**LTSCALE**) and type (**10**) for example and see the difference.



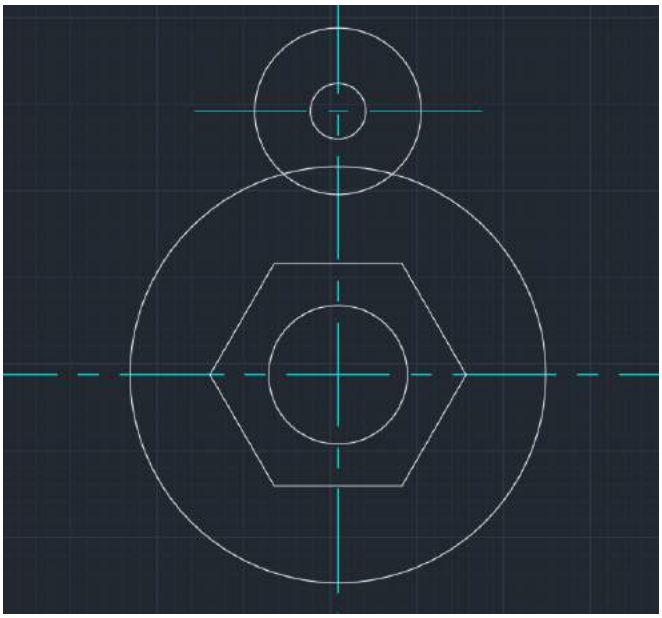
2. Use Offset tool to make the other 2 horizontal lines at distances 38 mm above and below the middle one.



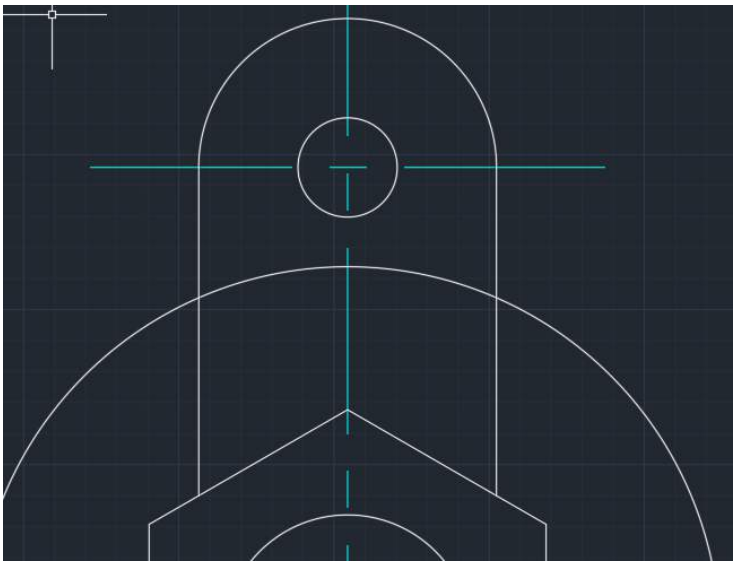
3. Now, return to your 0 layer, Draw the R10 and the D60 circles from the middle center



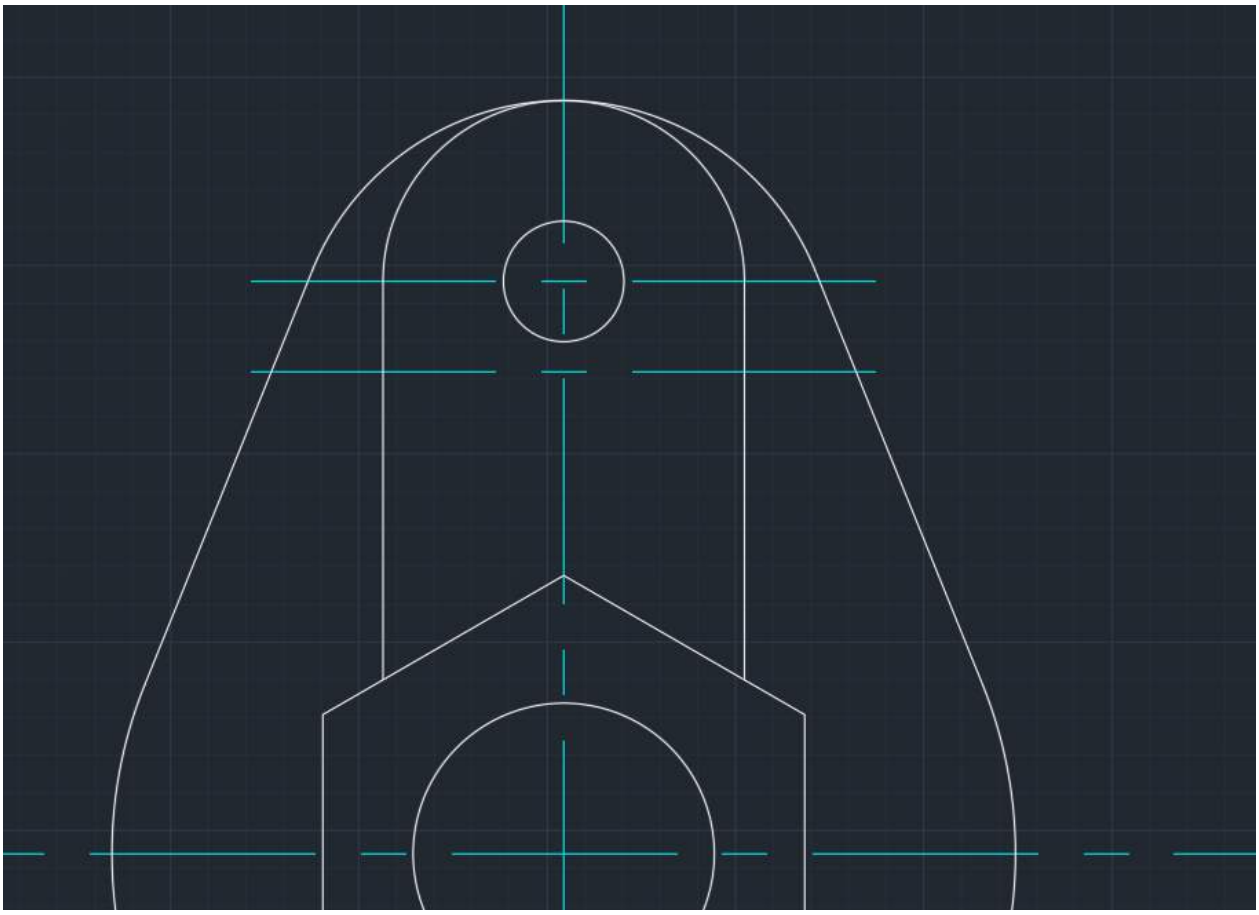
4. Go to the upper center point and draw the R12 and D8 circles. And return to the middle center point and draw the Hexagon with polygon tools and set the number of sides to 6 and choose the option of circumscribed about circle with R = 16 mm. Note that when you press Enter it will make the Hexagon as in the following fig.



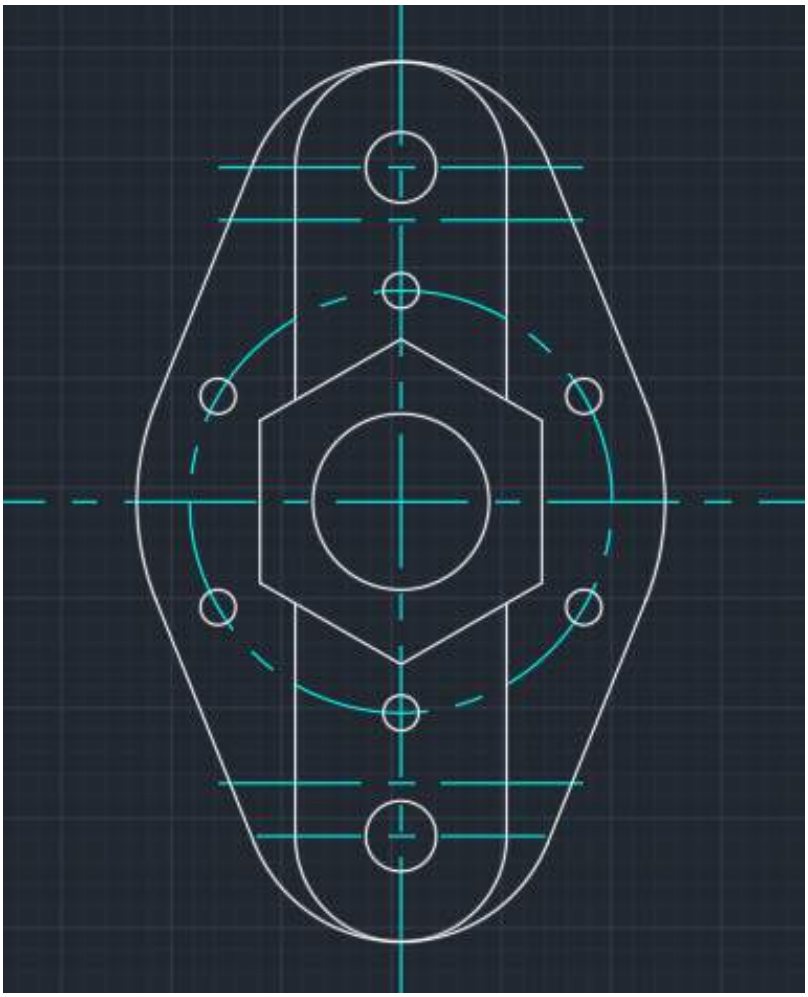
5. Then we have to ROTATE the Hexagon in order to have it like in the drawing. Use Rotate tool, select the Hexagon, press Enter, and select a base point (a point that will remain in its place) by clicking on the center point of the Hexagon. And then press on the vertical centerline.
6. Draw the vertical lines from the R12 circles to the intersection with the Hexagon sides, make the Orthomode active in this step to have them vertical.
7. Use trim tool to remove the unneeded half circle.



8. Draw the R18 circle, it is not centered with the R12 circle. Figure it out on your own.
9. Draw tangent lines, tangent to the R18 circle and the D60 circle. There are 2 ways to draw a line tangent to 2 circles. Open Line command, type TAN and press Enter, select the first circle and Type TAN again and press Enter then select the other circle. Or instead of typing after choosing the line command, press Ctrl with right click of the mouse and choose tangent.
10. Remove the unneeded lines.

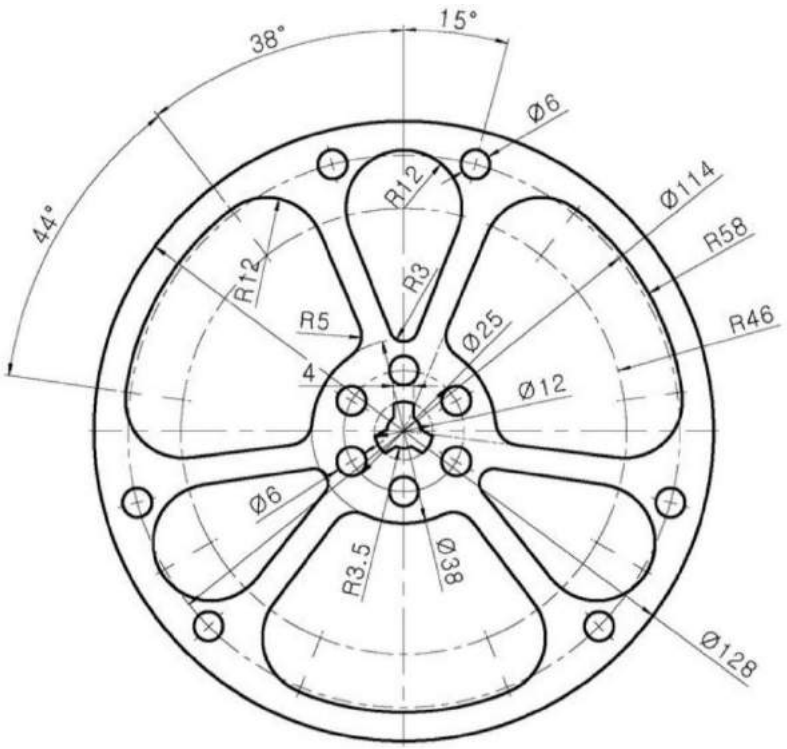


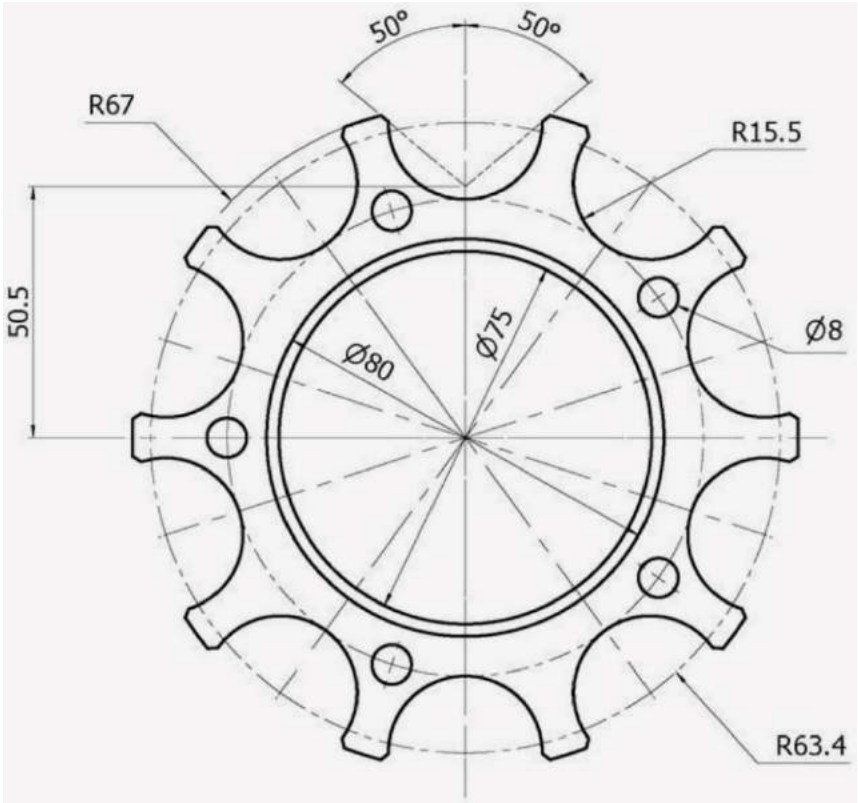
11. Use Mirror to copy these features to the other side.
12. Now go to the Centerlines Layer and draw the D48 circle, and draw a D4 circle on Layer 0 (the one on the vertical CL)
13. Use Polar Array to draw the other circles. Select the object and press Enter, then select the center point of the dashed circle. Make the number of items 6 and make sure it fills 360 degrees.

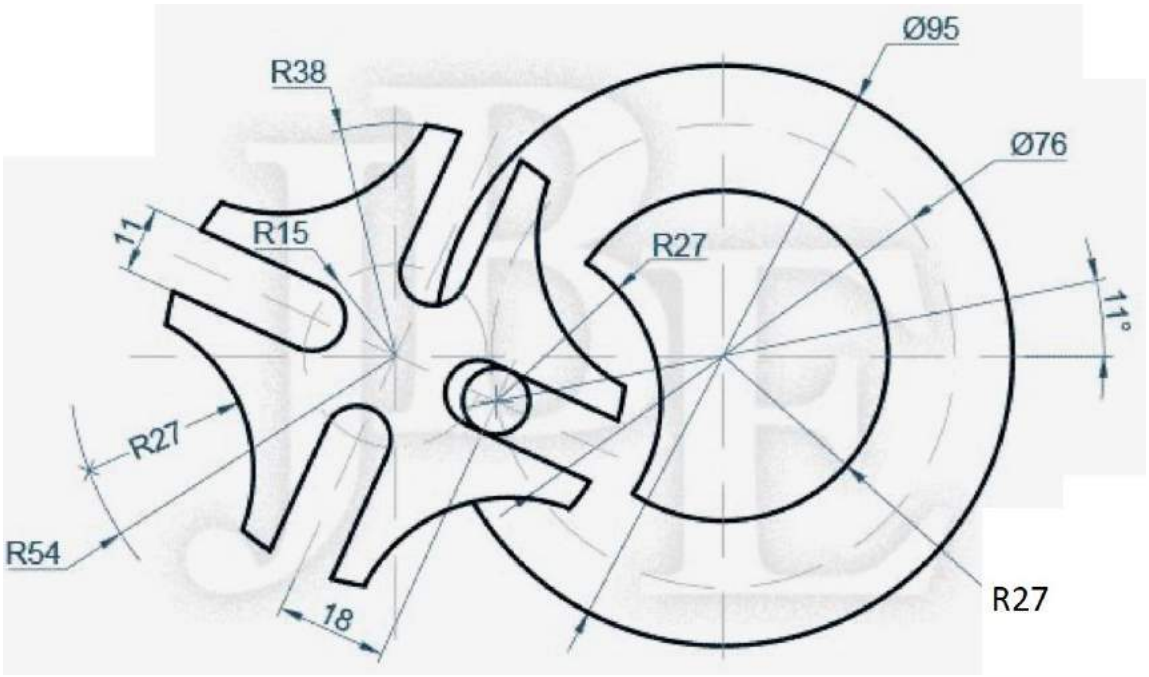


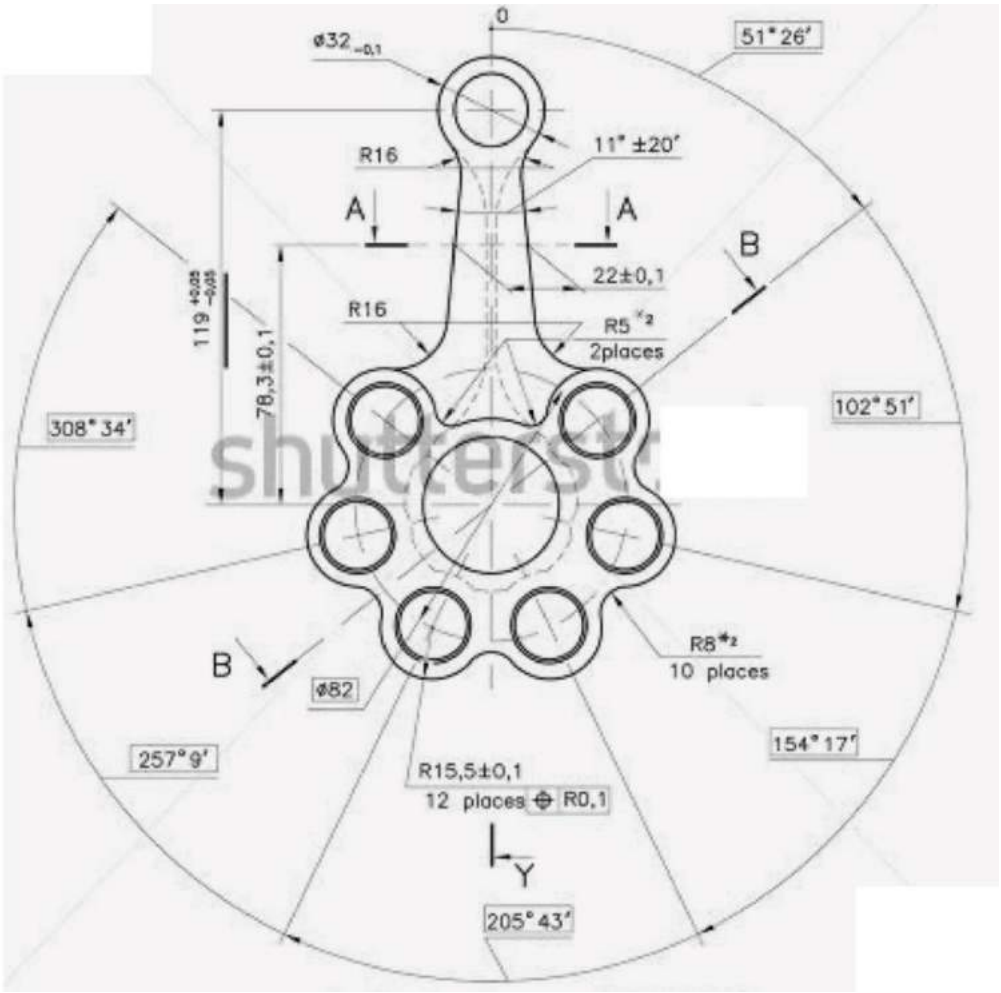
14. Now, we have only on step left, the rectangle with dimensions of 9*4 mm. Use the Midpoint of height and corner rectangle and type 9 mm for the length and Tab key, then 2 mm for the half of the height.
15. Save the file in the format you want.

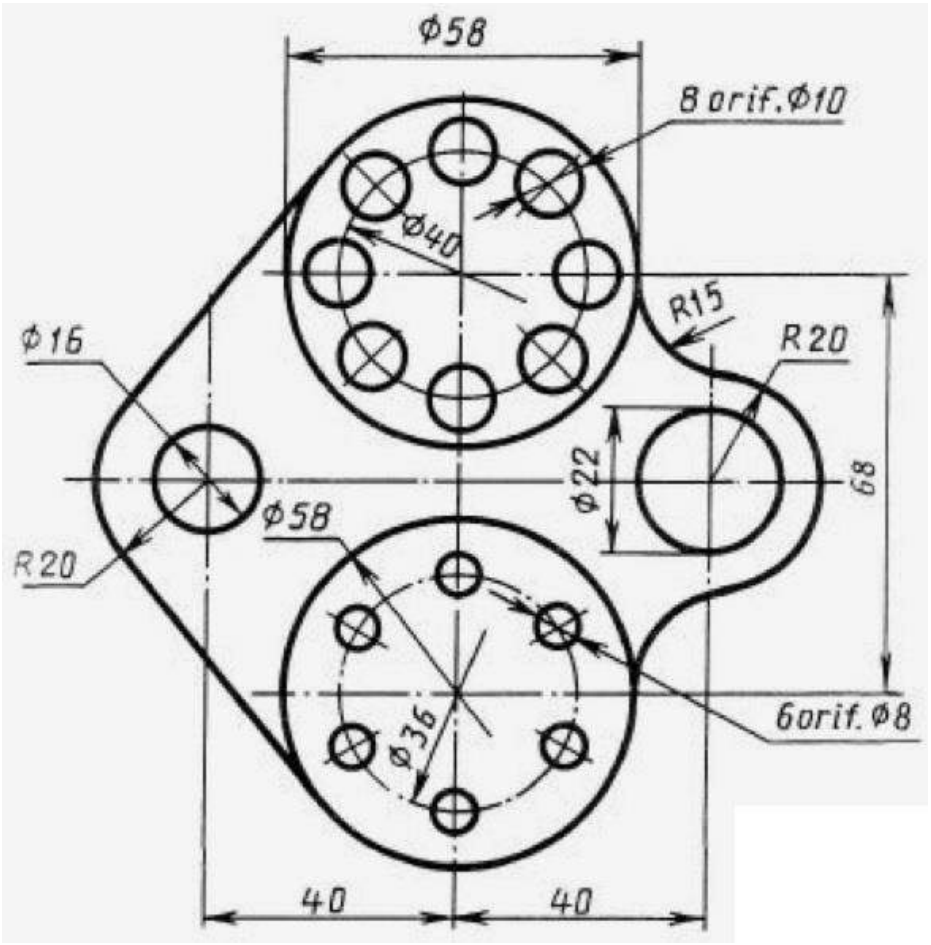
Problems

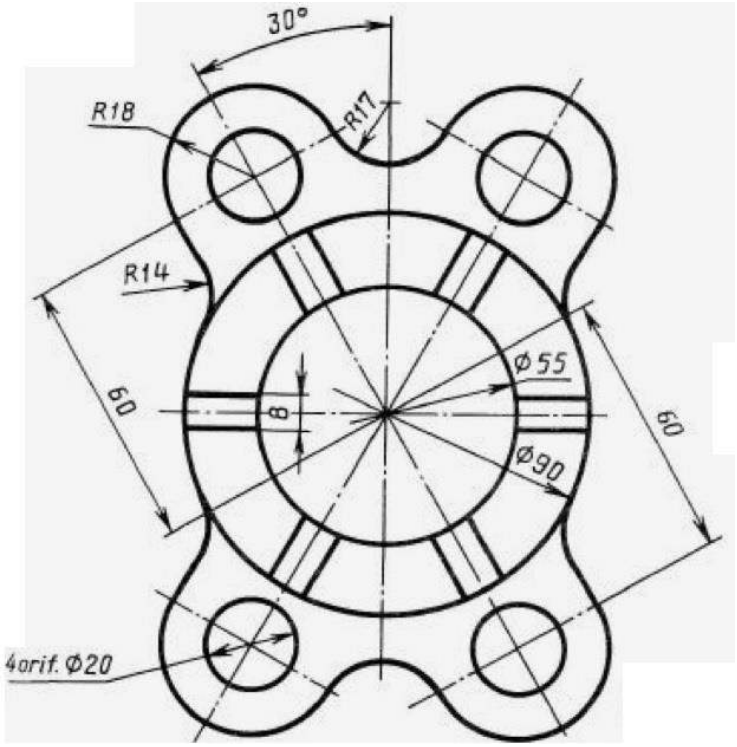

38°
15°
Ø6
44°
R12
Ø114
R58
R12
R3
R5
Ø25
R46
4
Ø12
Ø6
R3.5
Ø38
Ø128

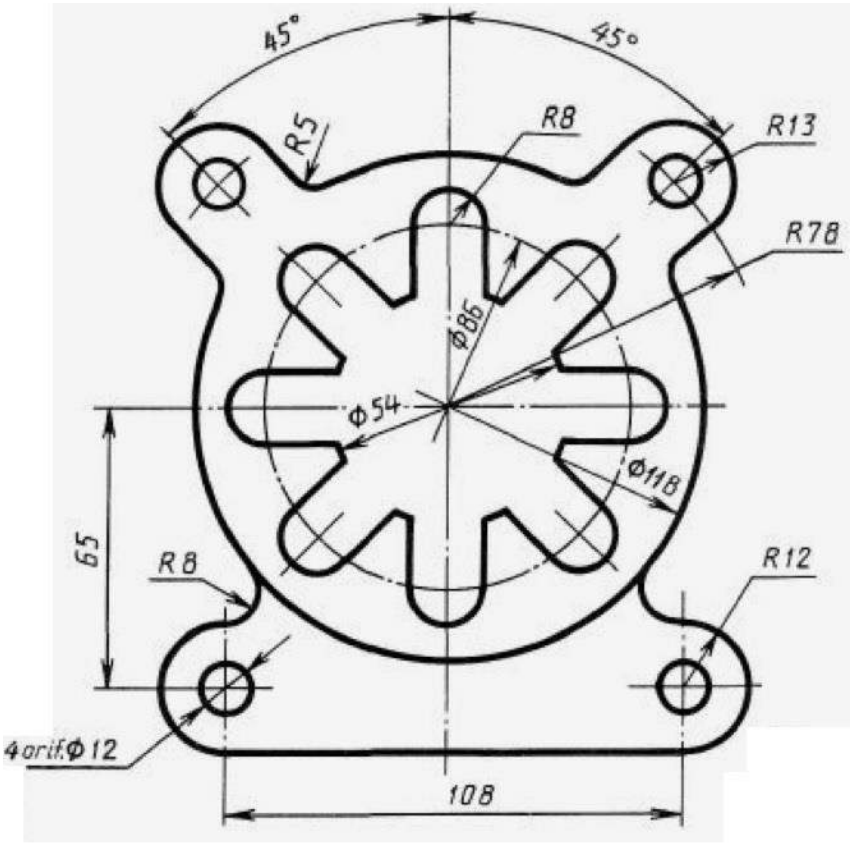

50°
50°
R67
R15.5
50.5
Ø80
Ø75
Ø8
R63.4

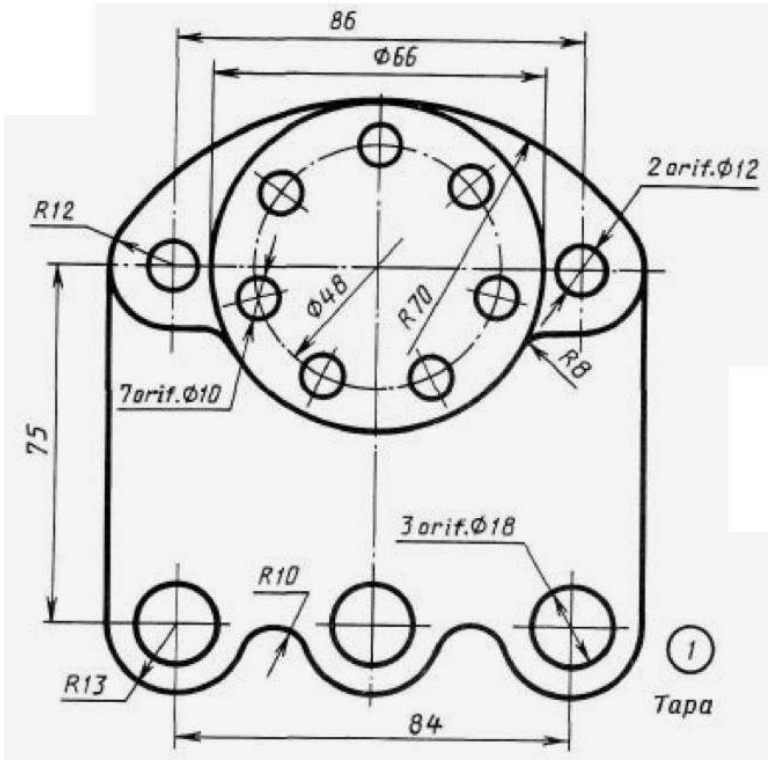

Ø95
R38
Ø76
11
R15
R27
11°
R27
R54
18
R27


0
ø32 -0,1
51°26'
R16
11° ±20'
A
A
B
22±0,1
R16
R5 *2
2places
119 +0,05 -0,05
78,3±0,1
308°34'
102°51'
B
ø82
R8 *2
10 places
257°9'
154°17'
R15,5±0,1
12 places ⌖ R0,1
Y
205°43'


Φ58
8 orif.Φ10
Φ40
R15
R20
Φ16
Φ22
68
Φ58
R20
Φ36
6orif.Φ8
40
40


30°
R18
R17
R14
60
8
Φ55
Φ90
60
4orif.Φ20


45°
45°
R5
R8
R13
R78
Φ85
Φ54
Φ118
65
R8
R12
4 orif.Φ12
108


86
Φ66
2 orif.Φ12
R12
Φ48
R70
R8
7 orif.Φ10
75
3 orif.Φ18
R10
R13
84
1
Tapa

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