የሥራ እና ክህሎት ሚኒስቴር
MINISTRY OF LABOR AND SKILLS

Welding Level-II

Based on March 2022, Curriculum Version 1

Module Title: -Preparing Basic (2D) Engineering Drawing Using CAD

Module code: IND WLD2 M01 0322

Nominal duration: 100 Hour

Prepared by: Ministry of Labor and Skill

August, 2022

Addis Ababa, Ethiopia

Table of content

Acknowledgment

Ministry of Labor and Skills wish to extend thanks and appreciation to the many representatives of TVET instructors and respective industry experts who donated their time and expertise to the development of this Teaching, Training and Learning Materials (TTLM).

Acronym

ASME	American Society for Mechanical Engineering
BIS	British Standard
CAD	Computer Aided Drawing
CADD	Computer Aided Drawing and Design
DWG	Drawing
2D	Two - Dimensional
3D	Three -Dimensional
ISO	International Standard Organizational

List of Figures

List of Tables

Introduction to the Module

In welding field; preparing basic 2D engineering drawing using CAD projects helps to know drawing instrument and requirements, to perform manual drafting, to produce simple Two Dimensional (2D) metal engineering drawings, part and material lists, to Identify key features of CAD software and to produce computer aided drafting Supplied for welding field.

This module is designed to meet the industry requirement under the welding occupational standard, particularly for the unit of competency: **Preparing Basic (2D) Engineering Drawing Using CAD.**

This module covers the units:

- Drawing instrument and requirements
- Engineering drawing
- Engineering parts list
- Key features of CAD software
- Basic drawing elements and 2D drawings
- Approved drawing

Learning Objective of the Module

- Select drawing instrument and requirements
- Prepare a drawing
- Prepare engineering parts list
- Identify key features of CAD software
- Create basic drawing elements and 2D drawings
- Check approved drawing

Module Instruction

For effective use this modules trainees are expected to follow the following module instruction:

1. Read the information written in each unit
2. Accomplish the Self-checks at the end of each unit
3. Perform Operation Sheets which were provided at the end of units
4. Do the “LAP test” giver at the end of each unit and
5. Read the identified reference book for Examples and exercise

Unit One: Drawing Instrument and Requirements

This unit is developed to provide you the necessary information regarding the following content coverage and topics:

- Specifications and engineering drawing data
- Drafting and measuring instruments and tools
- Drawing requirements/criteria for conformance

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Identify engineering drawing specification and Standards
- Identify types and uses of drafting equipment and drawing instruments
- Apply drawing requirements/criteria for conformance

1. Drawing Instrument and Requirements

1.1. Specifications and Engineering Drawing Data

1.1.1. Introduction

A technical person can use the graphic language as powerful means of communication with others for conveying ideas on technical matters. However, for effective exchange of ideas with others, the engineer must have proficiency in language, technical symbol sand the graphic language. Engineering drawing is a suitable graphic language from which any trained person can visualize the required object. As an engineering drawing displays the exact picture of an object, it obviously conveys the same ideas to every trained eye. Thus, the engineering drawing is the universal language of all engineers.

Engineering drawing is a two dimensional representation of three dimensional objects. In general, it provides necessary information about the shape, size, surface quality, material, manufacturing process, etc., of the object.

1.1.2. Engineering drawing data

Before starting any drawings we need always to identify the specification and engineering drawing information required from the customer order or other sources of data's like production drawing, machine drawing, part drawing and assembly drawing.

A. Machine drawing

It is pertaining to machine parts or components. It is presented through a number of orthographic views, so that the size and shape of the component is fully understood. Part drawings and assembly drawings belong to this classification. An example of a machine drawing is given in Fig. 1.1.

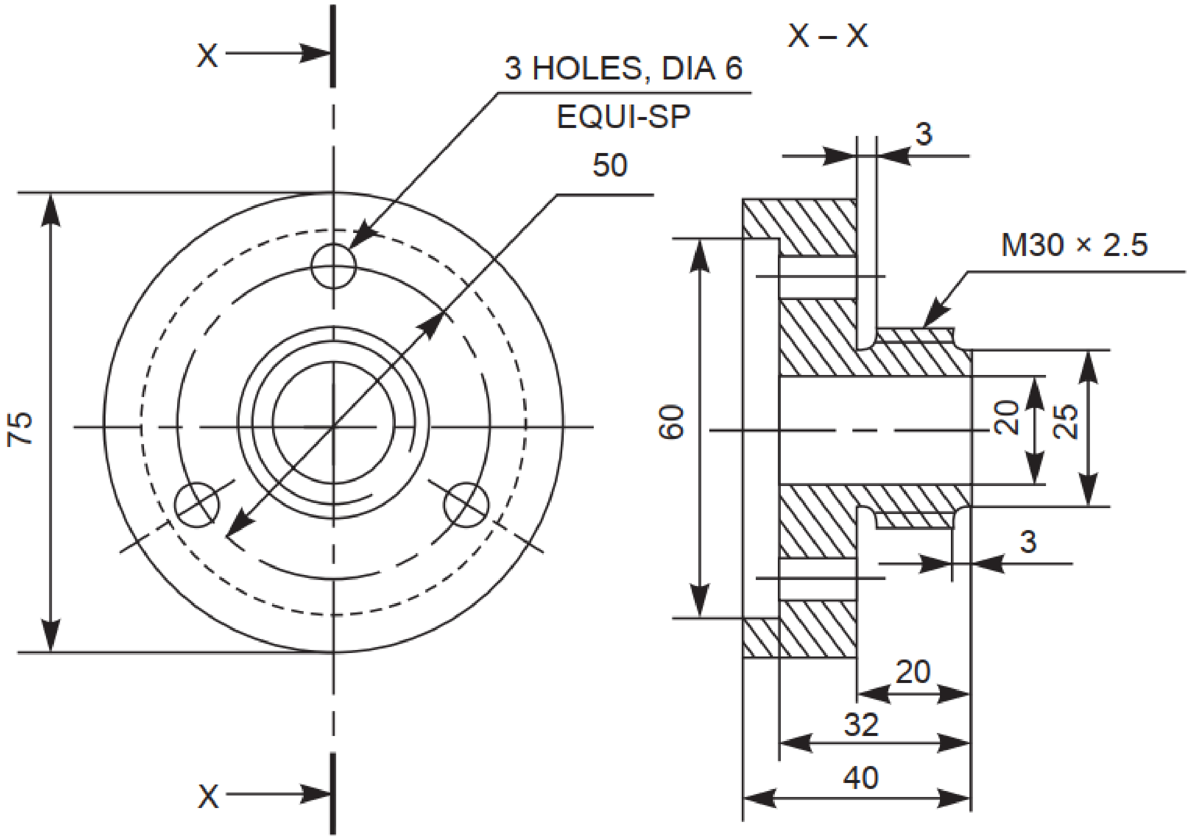


Figure 1.1 Part drawing of a machine component.

B. Production Drawing

A production drawing, also referred to as working drawing, should furnish all the dimensions, limits and special finishing processes such as heat treatment, honing, lapping, surface finish, etc., to guide the craftsman on the shop floor in producing the component. The title should also mention the material used for the product, number of parts required for the assembled unit, etc.

Since a craftsman will ordinarily make one component at a time, it is advisable to prepare the production drawing of each component on a separate sheet. However, in some cases the drawings of related components may be given on the same sheet. Figure 1.2 represents an example of a production drawing.

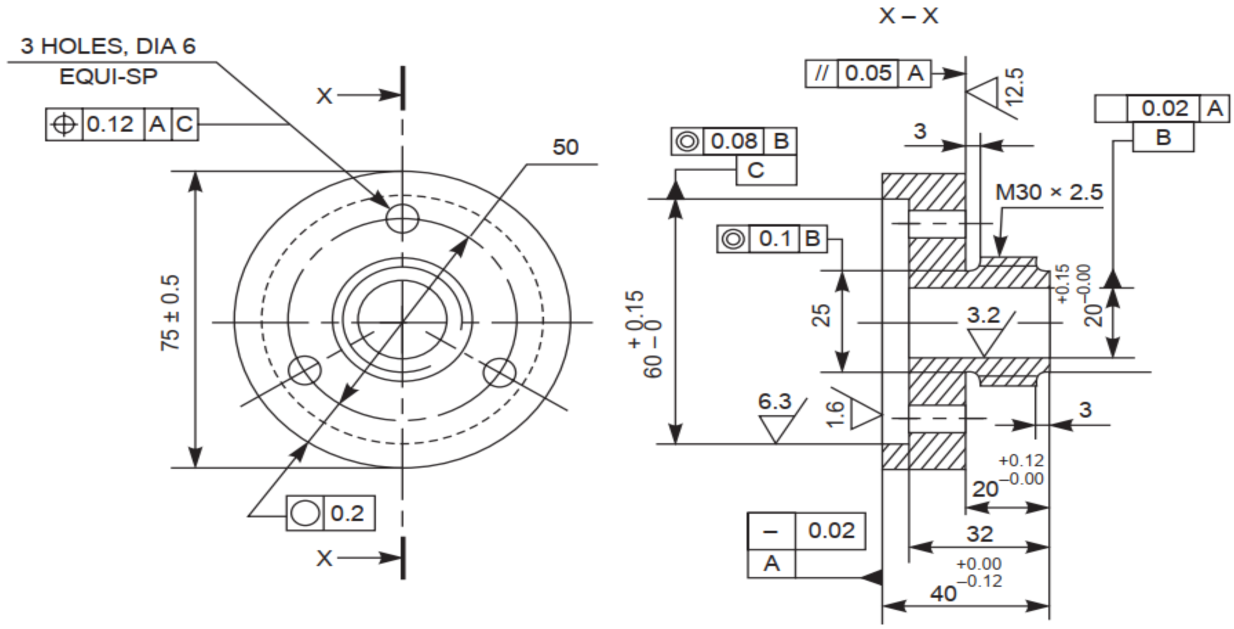


Figure 1.2 Production drawing of a machine component

C. Part drawing

Component or part drawing is a detailed drawing of a component to facilitate its manufacture. All the principles of orthographic projection and the technique of graphic representation must be followed to communicate the details in a part drawing. A part drawing with production a detail is rightly called as a production drawing or working drawing.

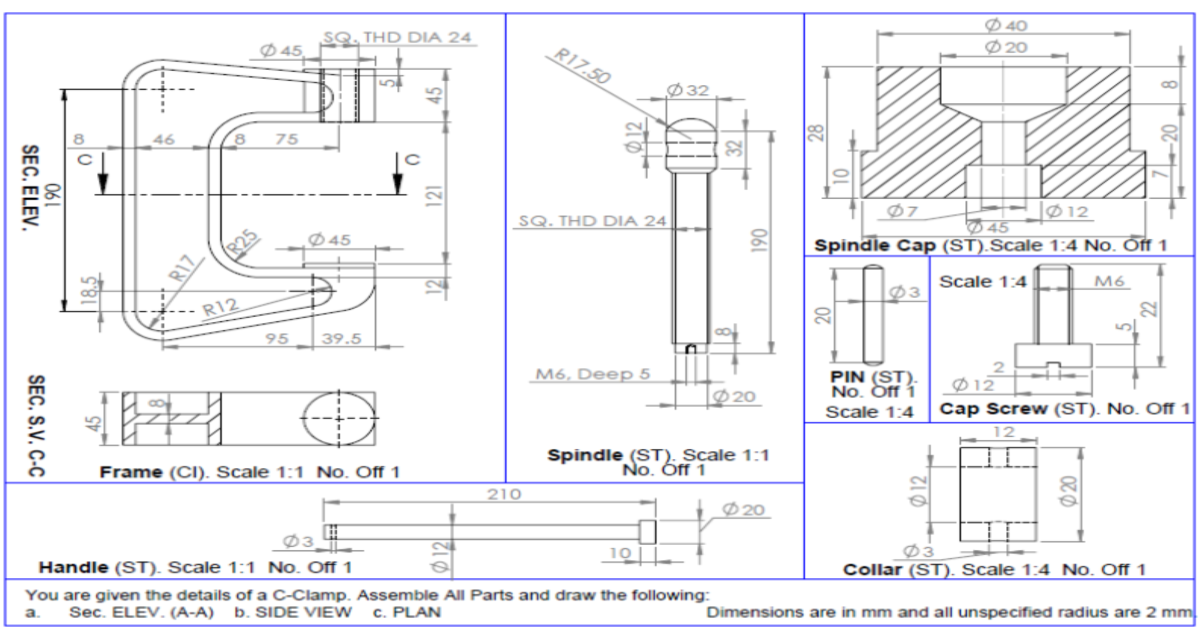


Figure 1.3 Part drawing

D. Assembly drawing

A drawing that shows the various parts of a machine in their correct working locations is an assembly drawing. Fig. 1.4 shows an example of an assembly drawing (Note that the drawing was drawn according to a different drawing standard and specification). There are several types of assembly drawings.

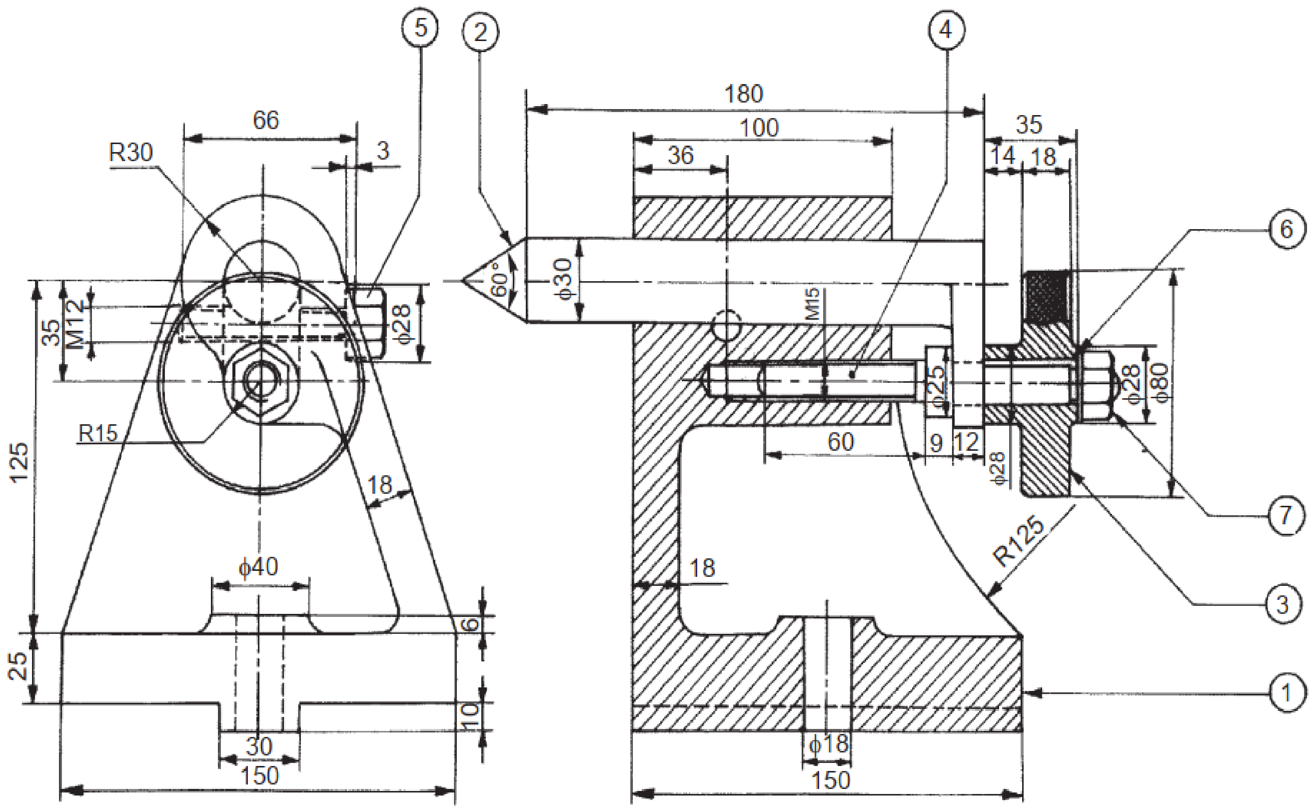


Figure 1.4 Assembly drawing of tail stock

Table 1.1Parts list of the Assembly drawing

Part No.	Name	Material	Qty.
1	Body	CI	1
2	Centre	Case hardened alloy steel	1
3	Hand wheel	Cast steel	1
4	Screw	MS	1
5	Screw	MS	1
6	Washer	MS	1
7	Nut	MS	1

1.2. Drafting and measuring instruments and tools

Drawing instruments are used to prepare drawings easily and accurately. The accuracy of the drawings depends largely on the quality of instruments. With instruments of good quality, desirable accuracy can be attained with ease. It is, therefore, essential to procure instruments of as superior quality as possible.

Below is the list of minimum drawing instruments and other drawing materials which every student must possess:

1. Drawing board
2. T-square
3. Set-squares - 45° and 30°- 60°
4. Drawing instrument box
5. Scales
6. Protractor
7. French curves
8. Drawing papers
9. Drawing pencils
10. Eraser (Rubber)
11. Drawing pins, clips or adhesive tapes
12. Drafting machine

We shall now describe each of the above in details with their uses:

1. Drawing board

Drawing board is rectangular in shape and is made of strips of well-seasoned softwood about 25 mm thick. One of the edges of the board is used as the working edge, on which the T-square is made to slide. It should, therefore, be perfectly straight. In some boards, this edge is grooved throughout its length and a perfectly straight ebony edge is fitted inside this groove. This provides a true and more durable guide for the T-square to slide on. Drawing board is made in various sizes. Its selection depends upon the sizeof the drawing paper to be used.

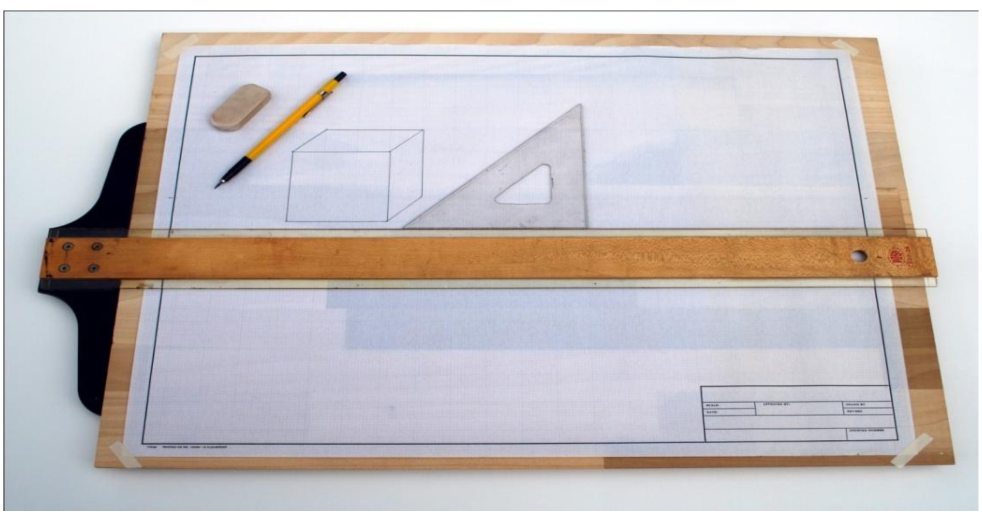


Figure 1.5Drawing Board

2. T-square

A T-square is made up of hard-quality wood. It consists of two parts - the stock and the blade - joined together at right angles to each other by means of screw sand pins. The stock is placed adjoining the working edge of the board and is made to slide on it as and when required. The blade lies on the surface of the board.

Its distant edge which is generally beveled is used as the working edge and hence, it should be perfectly straight.

Figure 1.6 Tee – square

3. Set-squares

The set-squares are made of wood, tin, celluloid or plastic. Those made of transparent celluloid or plastic are commonly used as they retain their shape and accuracy for longer time. Two forms of set-squares are in general use. A set-square is triangular in shape with one of the angle as right angle. The 30°-60° set-square of 250 mm length and 45° set-square of 200 mm length are convenient sizes for use in schools and colleges.

4. Drawing instrument box

The drawing instrument box contains the following as mentioned earlier:

(1) Large-size compass with interchangeable pencil and pen legs

(2) Lengthening bar (5) Small bow divider

(3) Small bow compass (6) Small bow ink-pen

(4) Large-size divider (7) Inking pen.

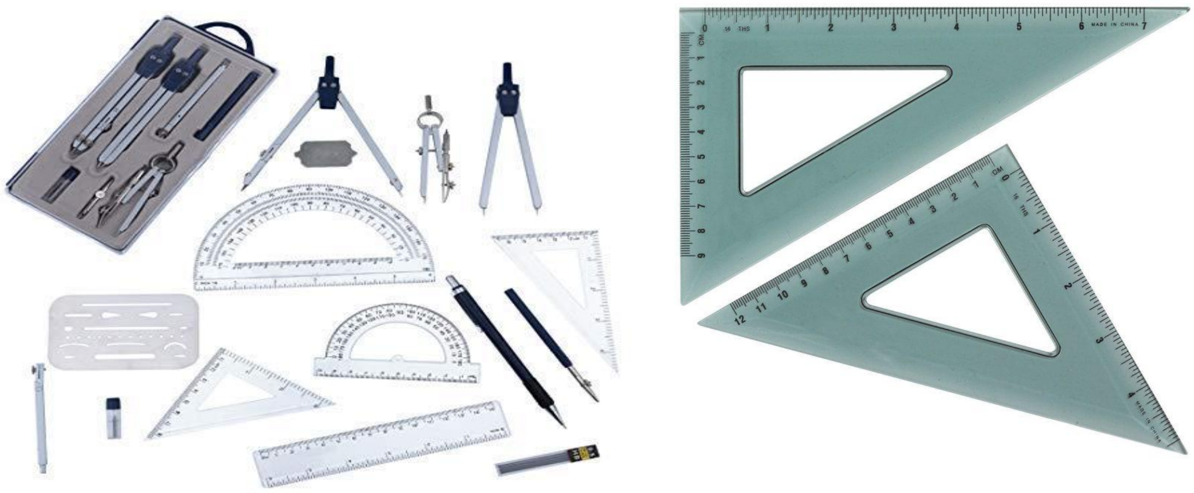


Figure 1.7Tri square and set of drawing instrument box

5. Scales

Scales are made of wood, steel, celluloid or plastic or card board. Stainless-steel scales are more durable. Scales may be flat or of triangular cross-section. 15 cm long and 2 cm wide or 30 cm long and 3 cm wide flat scales are in common use. They are usually about 1 mm thick. Scales of greater thickness have their longer edges beveled. This helps in marking measurements from the scale to the drawing paper accurately. Both the longer edges of the scales are marked with divisions of centimeters, which are sub-divided into millimeters.

6. Protractor

Protractor is made of wood, tin or celluloid. Protractors of transparent celluloid are in common use. They are flat and circular or semi-circular in shape. The commonest type of protractor is semi-circular and of about 100 mm diameter. Its circumferential edge is graduated to 1 ° divisions, is numbered at every 10° interval land is readable from both the ends. The diameter of the semi-circle (viz. straight line 0-180°) is called the base of the protractor and its centre O is marked by a line perpendicular to it.

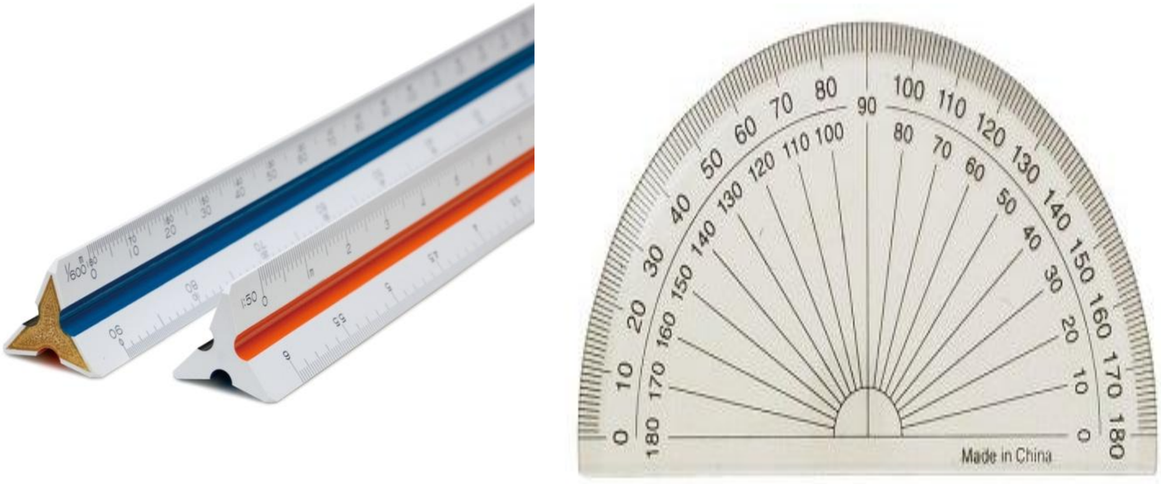


Figure 1.8 Scale and Protractor

7. French curves

French curves are made of wood, plastic or celluloid. They are made in various shapes, one of which is shown in fig. 1.9. Some set-squares also have these curves cut in their middle. French curves are used for drawing curves which cannot be drawn with a compass. Faint free hand curve is first drawn through the known points. Longest possible curves exactly coinciding with the hand curve are then found out from the french curve. Finally, neat continuous curve is drawn with the aid of the french curve. Care should be taken to see that no corner is formed anywhere within the drawn curve.

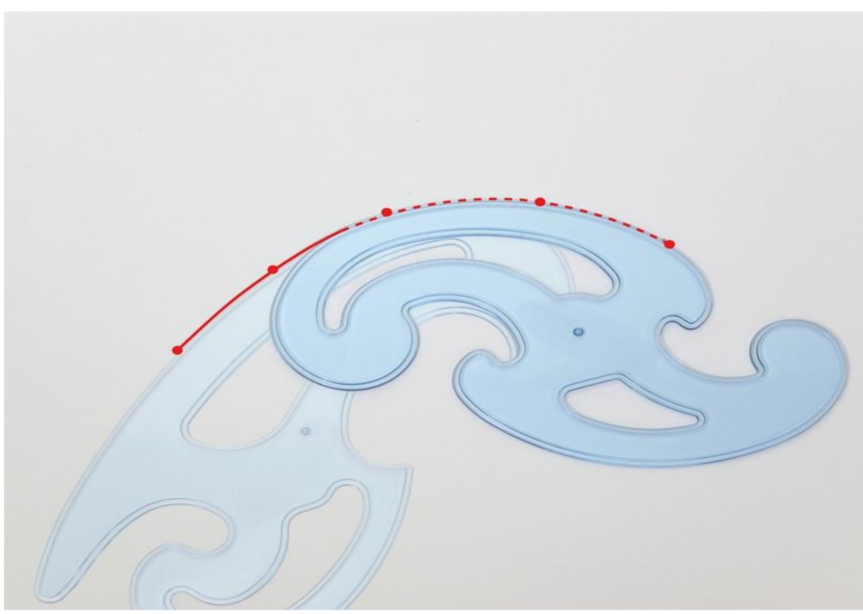


Figure 1.9 French Curve

8. Drawing papers

Drawing papers are available in many varieties. For ordinary pencil-drawings, the paper selected should be tough and strong. It should be uniform in thickness and as white as possible. When the rubber eraser is used on it, its fibers should not disintegrate. Good quality of paper with smooth surface should be selected for drawings which are to be inked and preserved for a long time. It should be such that the ink does not spread. Thin and cheap quality paper may be used for drawings from which tracings are to be prepared.

Surface area of AO size is one square meter. Successive format sizes (from AO to AS) are obtained by halving along the length or doubling along the width. The areas of the two subsequent sizes are in the ratio 1:2. See fig. 1-30.

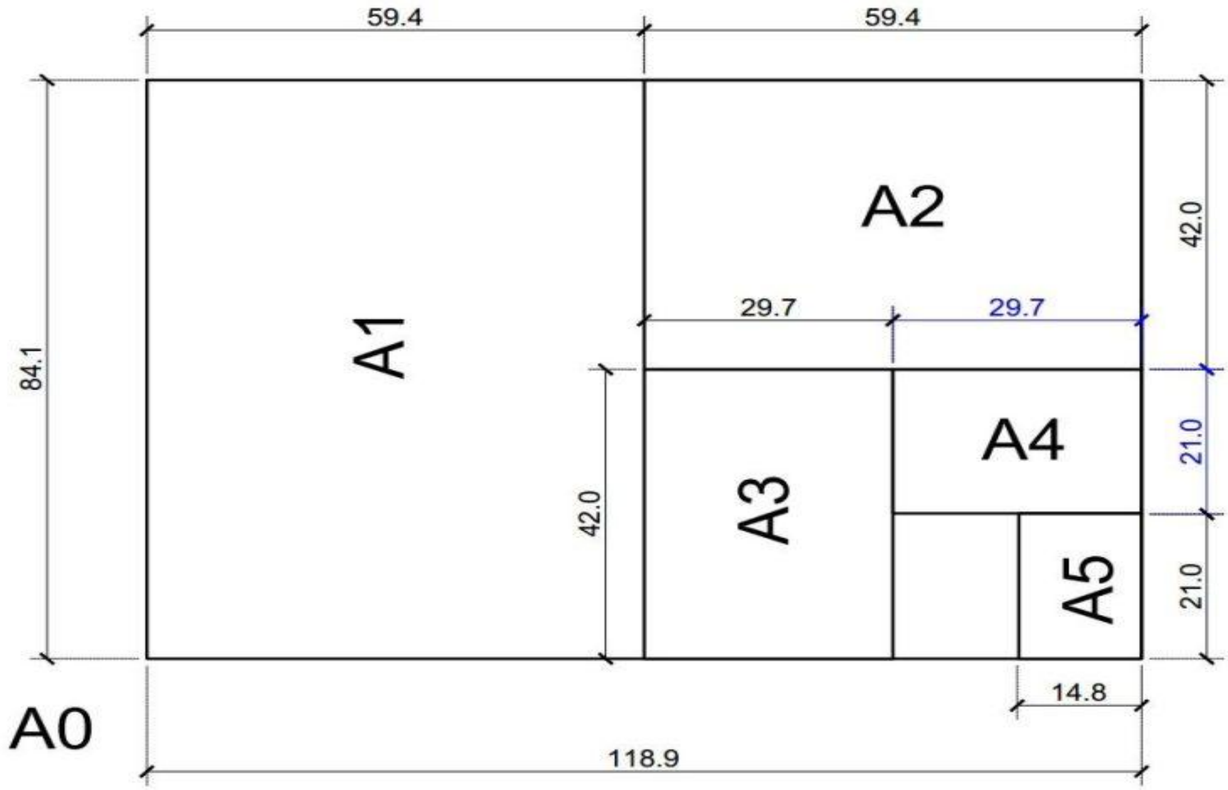


Figure 1.10 Drawing Paper and Size of Layout

9. Drawing Pencils

The accuracy and appearance of a drawing depend very largely on the quality of the pencils used. With cheap and low-quality pencils, it is very difficult to draw lines of uniform shade and thickness. The grade of a pencil lead is usually shown by figures and letters marked at one of its ends. Letters HB denote the medium grade.

The increase in hardness is shown by the value of the figure put in front of the letter H, viz. 2H, 3H, 4H etc. Similarly, the grade becomes softer according to the figure placed in front of the letter B, viz. 28, 38, 48 etc. Beginning of a drawing should be made with H or 2H pencil using it very lightly, so that the lines are faint, and unnecessary or extra lines can be easily erased. The final fair work may be done with harder pencils, e.g. 3H and upwards.

Lines of uniform thickness and darkness can be more easily drawn with hard grade pencils. H and HB pencils are more suitable for lettering and dimensioning. For freehand sketching, where considerable erasing is required to be done, soft-grade pencils such as HB should be used.

10. Eraser Rubber

Soft India-rubber is the most suitable kind of eraser for pencil drawings. It should be such as not to spoil the surface of the paper. Frequent use of rubber should be avoided by careful planning. Eraser is used to remove the lines or spots which drawn by mistake or with wrong measurements. The eraser used should be of good quality and soft. It should not damage the paper while erasing.

Figure 1.11 Drawing Pencil and Eraser Rubber

11. Drawing Tapes

These are used to fix the drawing paper on the drawing board. The needle part of the pin is generally made of steel, while the head may be of plated mild steel or brass. Pins of about 15 mm to 20 mm diameter and about 1 mm thick flatheads made of brass are quite convenient, as they do not rust. Pins should be so inserted that the heads sit on the surface of the paper. Clips or adhesive tapes are often used instead of the pins. (Refer fig. 1-32).

12. Drafting Machine

The uses and advantages of the T-square, set-squares, scales and the protractor are combined in the drafting machine. It’s one end is clamped by means of a screw, to the distant longer edge of the drawing board. At its other end, an adjustable head having protractor markings is fitted. Two blades of transparent celluloid accurately set at right angles to each other are attached to the head.

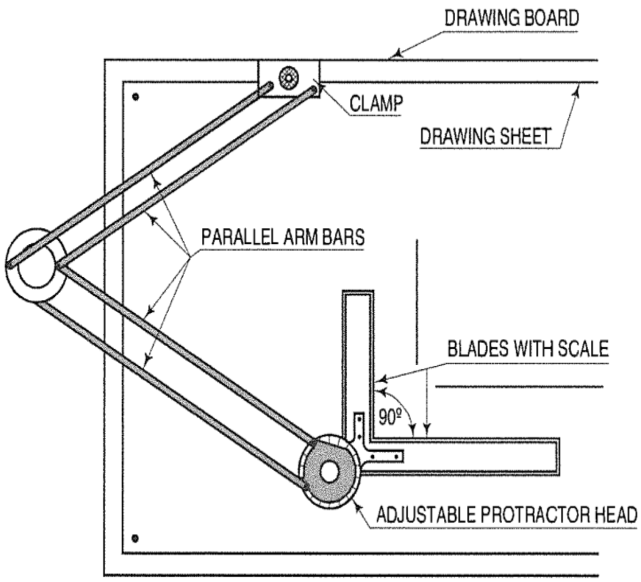


Figure 1.12Drafting Machine and Drawing tapes

13. Drawing Templates

Templates are nothing but plastic or wooden boards which contains spaces of several shapes or letters. Non-dimensional shapes or variety font letters are drawn by using templates which makes drawing easier and perfect.

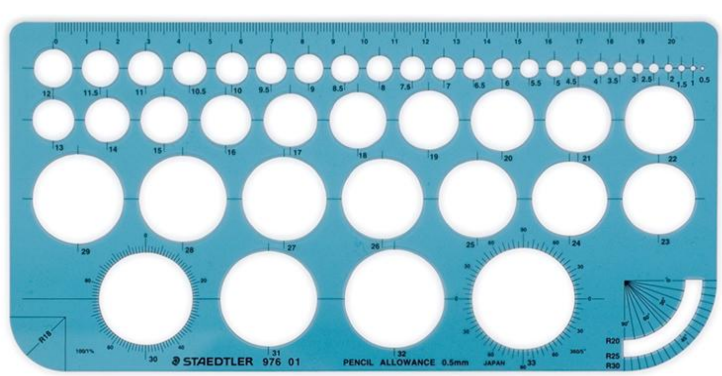


Figure 1.13 Circular Drawing Template

1.3. Drawing requirements/criteria for conformance

Drawing verification is an essential step in the development of any product. Also referred to as qualification testing, design verification ensures that the product as designed is the same as the product as intended. Unfortunately, many design projects do not complete thorough design qualification resulting in products that do not meet customer expectations and require costly design modifications. It is important to check and verify the drawing requirements by any relevant person depending on the standard of the production or project and customer specification.

The drawings prepared by any technical person must be clear, unmistakable in meaning and there should not be any scope for more than one interpretation, or else litigation may arise. In a number of dealings with contracts, the drawing is an official document and the success or failure of a structure depends on the clarity of details provided on the drawing. Thus, the drawings should not give any scope for misinterpretation even by accident. It would not have been possible to produce the machines/automobiles on a mass scale here a number of assemblies and sub-assemblies are involved, without clear, correct and accurate drawings.

To achieve this, the technical person must gain a thorough knowledge of both the principles and conventional practice of draughting. If these are not achieved and or practiced, the drawings prepared by one may convey different meaning to others, causing unnecessary delays and expenses in production shops.

Hence, an engineer should posses good knowledge, not only in preparing a correct drawing but also to read the drawing correctly. The module content of this learning guide is expected to meet these requirements. The study of product drawing mainly involves learning to sketch product parts and to make working and assembly drawings. This involves a study of those conventions in drawings that are widely adopted in engineering practice. The drawing must be checked for all drawing requirements by relevant personnel of the institute or industry before ordered for manufacturing.

The relevant personnel of the industries may include supervisors, Technical personnel, manufacturer, contractor and supplier.

Supervisor

- Communicating with Supervisors, Peers, or Subordinates
- Monitor Processes, Materials, or Surroundings

- Organizing, Planning, and Prioritizing Work
- Getting Information

Compliance adds requirements that transform document management into issue control issue. Document control is a regulatory requirement within accounting. Provide evidence of documents under control. Failing to comply could cause fines, the loss of business, or damage to your business reputation.

The basic requirements for document control require that you establish and document a procedure for:-

- Reviewing and approving documents prior to release
- Reviews and approvals
- Ensuring changes and revisions are clearly identified
- Ensuring that relevant versions of a “points of use”
- Ensuring that documents remain legible and identifiable
- Ensuring that external documents like customer supplied documents or supplier manuals are identified and controlled
- Preventing “unintended

Self check-1

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. It is a graphic representation of an idea a from which any trained person can visualize the required object..

   A. Machine Drawing
   B. Engineering Drawing
   C. Casting Drawing
   D. signals

2. It is a detailed drawing of a component to facilitate its manufacture.

   A. Component drawing
   B. Machine drawing
   C. Production Drawing
   D. Assembly drawing

3. What is the dimension of A1 size drawing sheet

   A. 1189mm x 841mm
   B. 594mm x 841mm
   C. 1230mm x 880mm
   D. 880mm x 625 mm

4. The _____ drawing shows how the components are added to their proportions

   A. Layout Drawing
   B. Assembly drawing
   C. Working Drawing
   D. Design Drawing

5. A common use is to specify necessary for construction of component and is called

   A. Drafting
   B. Assembly Drawing
   C. Detail Drawing
   D. Technical Drawing

6. Which of the following drawing tools is used to draw horizontal lines

   A. Compass
   B. Protractor
   C. French Curve
   D. T – square

7. _______ used to fix the drawing paper on the drawing board.

   A. Drawing Tapes
   B. Protractor
   C. French Curve
   D. T – square

8. ______ are used for drawing curves which cannot be drawn with a compass.

   A. Compass
   B. Protractor
   C. French Curve
   D. T – square

9. Which Drawing instrument is different according the General purpose

   A. Compass
   B. Protractor
   C. Drawing Paper
   D. T – square

Unit Two: Engineering Drawing Preparation

This unit to provide you the necessary information regarding the following content coverage and topics:

- ISO standard on Drafting Instruments
- Principles of Drafting as local & international standards
- Dimensions, notes and specifications in the drawing

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Use Drafting instruments and drawing device
- Apply the drafting principle and standard in the engineering drawing preparation
- Indicate dimensions, notes and specifications in the drawing
- Present the drawing with the standard operating procedure and time frames

2. Engineering Drawings Preparation

2.1. ISO standard on drafting Instruments

This unit provides an overview of engineering drawing from the standpoint of the relevant International Standards Organization (ISO) standards, where the emphasis is on producing engineering drawings of products for eventual manufacturing. The basic objective of engineering drawing is to communicate product design and manufacturing information in are liable and unambiguous manner because engineering drawing needs to be language independent so that a designer in one country can specify a product that is made in another country.

The language of engineering is defined by the rules that are embodied in the publications of standards organizations. Each country has its own standards organization, such as the United Kingdom has the British Standards Institution (BSI), the United States has the American national Standards Institute (ANSI), and Germany has the Deutsches Institutfür Normung (DIN).

2.1.1. Standard on Drawing Sheet

Engineering drawings are prepared on drawing sheets of standard sizes. The use of standard size sheet saves paper and facilitates convenient storage of drawings. The standard dealing with the sizes and layout of drawing sheets is called —ISO 5457:1999.The basic principles involved in arriving at the sizes of drawing sheets are:

(a) $X:Y = 1:\sqrt{2}$ (B) $XY = 1$, where X and Y are the sides of the sheet

For a reference size A0 (Table 2.1) having a surface area of1 m2, X = 841 mm and Y = 1189 mm. The successive format sizes are obtained either by halving along the length or doubling along the width, the areas beginning the ratio 1:2 (Fig. 2.1).

If hard copies of drawings are required, the first-choice standard sizes of drawings are the conventional “A1” sizes of drawing papers. The basic “A” size is the zero size or 0, known as “A0”.This has a surface area of 1 M2 but follows the 1:√2 ratio.

Designation of sheet size

The original drawing should be made on the smallest sheet, permitting the necessary clarity and resolution. The preferred sizes according to ISO-A series (First choice) of the drawing sheets are given in Table 2.1.

When sheets of greater length are needed, special elongated sizes (Second choice) are used (Table 2.2).These sizes are obtained by extending the shorter sides of format of the ISO-A series to lengths that are multiples of the shorter sides of the chosen basic format.

Table 2.1 preferred drawing sheet sizes (First choice) to ISO-A series

Designation	Dimensions(mm)
A0	841 x 1189
A1	594 x841
A2	420 x 594
A3	297 x 420
A4	210 x297

Table 2.2 Special elongated sizes (second choice) to ISO-A series

Designation	Dimensions(mm)
A3 x 3	420 x 891
A3 x 4	420 x 1188
A4 x 3	297 x 630
A4 x 4	297 x 840
A4 x 5	297 x 1050

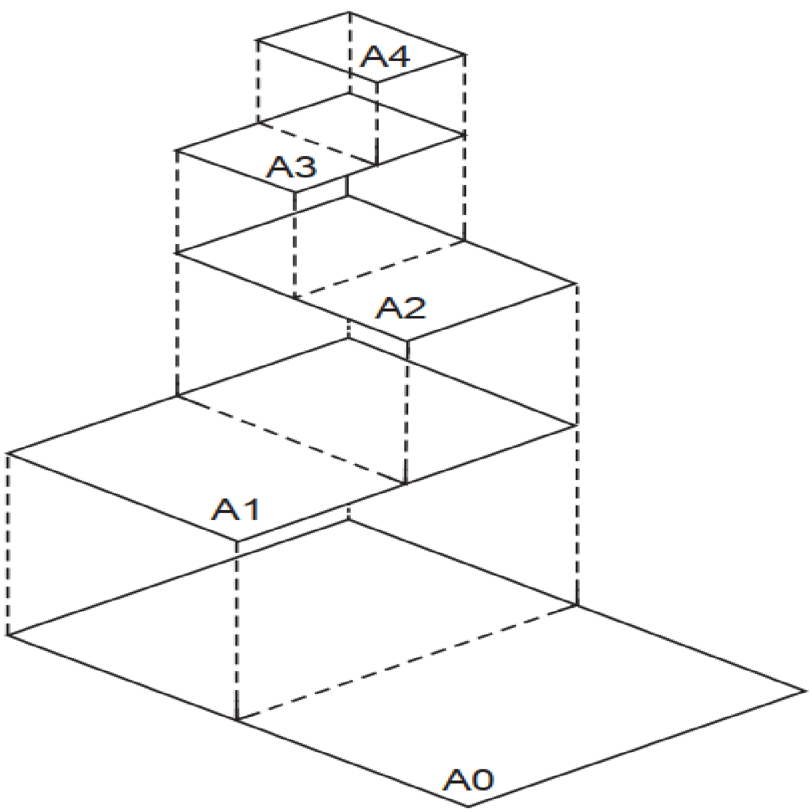


Figure 2.1Drawing sheet formats

2.1.2. Standard on Scales

Scale is the ratio of the linear dimension of an element of an object as represented in the drawing, to the real linear dimension of the same element of the object itself. Wherever possible, it is desirable to make full size drawings, so as to represent true shapes and sizes. If this is not practicable, the largest possible scale should be used. While drawing very small objects, such as watch components and other similar objects, it is advisable to use enlarging scales.

Designation of Scales

The complete designation of a scale should consist of the word Scale, followed by the indication of its ratio as:

SCALE 1 : 1 for full size,

SCALE × : 1 for enlarged scales,

SCALE 1 : × for reduced scales.

The designation of the scale used on the drawing should be shown in the title block.

Recommended Scale

The recommended scales for use on technical drawings are given in Table 2.3. The scale and the size of the object in turn, will decide the size of the drawing.

Table 2.3Recommended scales

Category	Recommended Scales		
Enlarged scales	50:1 5:1	20:1 2:1	10:1
Full size			1:1
Reduced scales	1:2 1:20 1:200 1:2000	1:5 1:50 1:500 1:5000	1:10 1:100 1:1000 1:10000

If all drawings are made to the same scale, the scale should be indicated in or near the title block. Where it is necessary to use more than one scale on a drawing, the main scale only should be shown in the title block and all the other scales, adjacent to the item reference number of the part concerned or near the drawings.

2.2. Principles of Drawing as local & international standards

2.2.1. Introduction

Engineering drawings are to be prepared on standard size drawing sheets and scales. The correct shape and size of the object can be visualized from the understanding of not only the views of it but also from the various types of lines used, dimensions, notes, scale, etc. To provide the correct information about the drawings to all the people concerned, the drawings must be prepared.

As Per drafting sheet standard, a blank drawing sheet should contain the title block, frame for limiting the drawing space, centering marks, orientation marks, metric reference graduation, grid reference system, and trimming marks. Before trying to communicate with such a language, one should learn its basics and principles, just like any other language. Those basics of engineering Drawing are: Lines, Signs, and Symbols. The process of producing engineering drawings, and the skill of producing them, is often referred to as technical drawing or drafting, although technical drawings are also required for disciplines that would not ordinarily be thought of as parts of engineering(such as Mechanical). More than just the drawing of pictures, engineering drawing it is also a language that communicates ideas and information from one being to another even if they do not understand each other languages. Moreover, it communicates all needed information from the engineer who designed a part to the workers who will manufacture it.

2.2.2. Local & international standards

Following the goal of **unambiguous communication**, engineering drawings are often made professionally and expected to follow certain national and international standards, such as ISO standards. Standardization also aids with internationalization, because people from different countries who speak different languages can share the common language of engineering drawing, and can thus communicate with each other quite well, at least as concerns the geometry of an object.

1. ISO Standards
2. ASME Standards
3. BS Standards
4. DIN Standards
5. JIS Standards

ISO Standards

Standardization is a dynamic and continuous process. The standards follow the development in engineering.ISO 128 is an international standard (ISO), about the general principles of presentation in technical drawings; specifically the graphical representation of objects on technical drawings.ISO 1101 represents the initial basis and describes the required fundamentals for geometrical tolerance. Nevertheless, it is advisable to consult the separate standards

Some ISO standard engineering drawings

- ISO 128 Technical drawings—General principles of presentation
- ISO 129 Technical drawings -- Indication of dimensions and tolerances
- ISO 1101 Geometrical tolerance
- ISO 2553 Symbolic representation on drawings -- Welded joints
- ISO 15786 Technical drawings -- Simplified representation and dimensioning of holes
- ISO 216 paper sizes, e.g. the A4 paper size
- ISO 406:1987 Technical drawings—Tolerance of linear and angular dimensions
- ISO 1660:1987 Technical drawings—Dimensioning and tolerance of profiles
- ISO 4172:1991 Technical drawings -- Construction drawings -- Drawings for the assembly of prefabricated structures
- ISO 5455:1979 Technical drawings—Scales
- ISO 5456 Technical drawings -- Projection methods
- ISO 5457:1999 Technical product documentation -- Sizes and layout of drawing sheets
- ISO 5459:1981 Technical drawings -- Geometrical tolerance -- Datums and datum-systems for geometrical tolerances
- ISO 5845-1:1995 Technical drawings—Simplified representation of the assembly of parts with fasteners—Part 1: General principles
- ISO 6411:1982 Technical drawings—Simplified representation of centre holes
- ISO 7200:2004 Technical drawings — Title blocks
- ISO 7083:1983 Technical drawings—Symbols for geometrical tolerance -- Proportions and dimensions
- ISO 7519:1991 Technical drawings -- Construction drawings -- General principles of presentation for general arrangement and assembly drawings
- ISO 8015:1985 Technical drawings—Fundamental tolerance principle
- ISO 8048:1984 Technical drawings -- Construction drawings -- Representation of views, sections and cuts
- ISO 8560:1986 Technical drawings -- Construction drawings -- Representation of modular sizes, lines and grids

- ISO 8560:1986 Technical drawings—Construction drawings—Representation of modular sizes, lines and grids
- ISO 13567 is an international Computer-aided design (CAD) layer standard.

2.2.3. Principles on Drawing Sheet

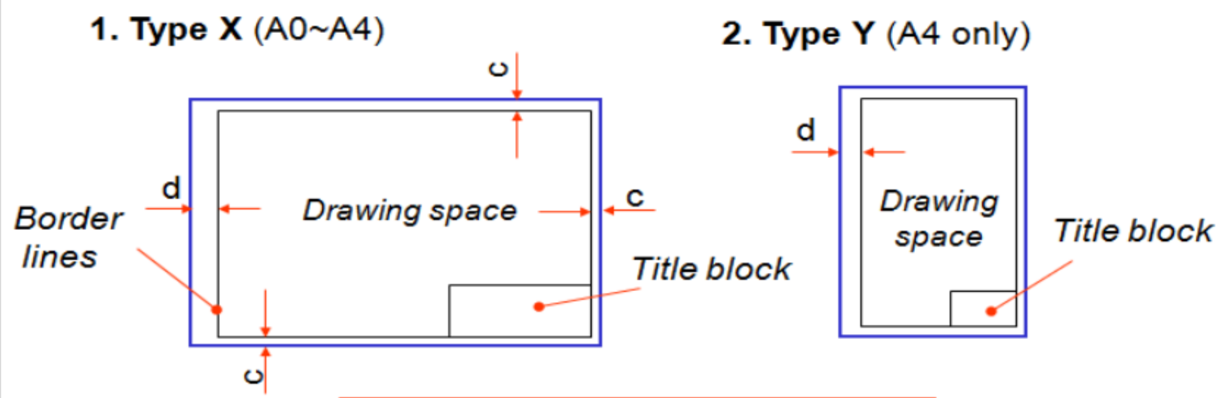
A. Title Block

The title block should lie within the drawing space such that, the location of it, containing the identification of the drawing, is at the bottom right hand corner. This must be followed, both for sheets positioned horizontally or vertically (Fig. 2.2).

The direction of viewing of the title block should correspond in general with that of the drawing. The title block can have a maximum length of 170 mm. Figure 2.3 shows a typical title block, providing the following information:

(i) Title of the drawing
(ii) Sheet number
(iii) Scale
(iv) Symbol, denoting the method of projection
(v) Name of the firm
(vi) Initials of staff drawn, checked and approved.

NOTE According to Bureau of Indian Standards, SP-46:1998, ‘‘Engineering Drawing Practice for Schools and Colleges’’, First angle projection is preferred.



Sheet size	c (min)	d (min)
A4	10	25
A3	10	25
A2	10	25
A1	20	25
A0	20	25

Figure 2.2Orientation of drawing sheet

B. Borders and frames

Borders enclosed by the edges of the trimmed sheet and the frame, limiting the drawing space, should be provided with all sheet sizes. It is recommended that these borders have a minimum width of 20 mm for the sizes A0 and A1 and a minimum width of 10 mm for the sizes A2, A3 and A4 (Fig. 2.4). A filing margin for taking perforations, may be provided on the edge, far left of the title block.

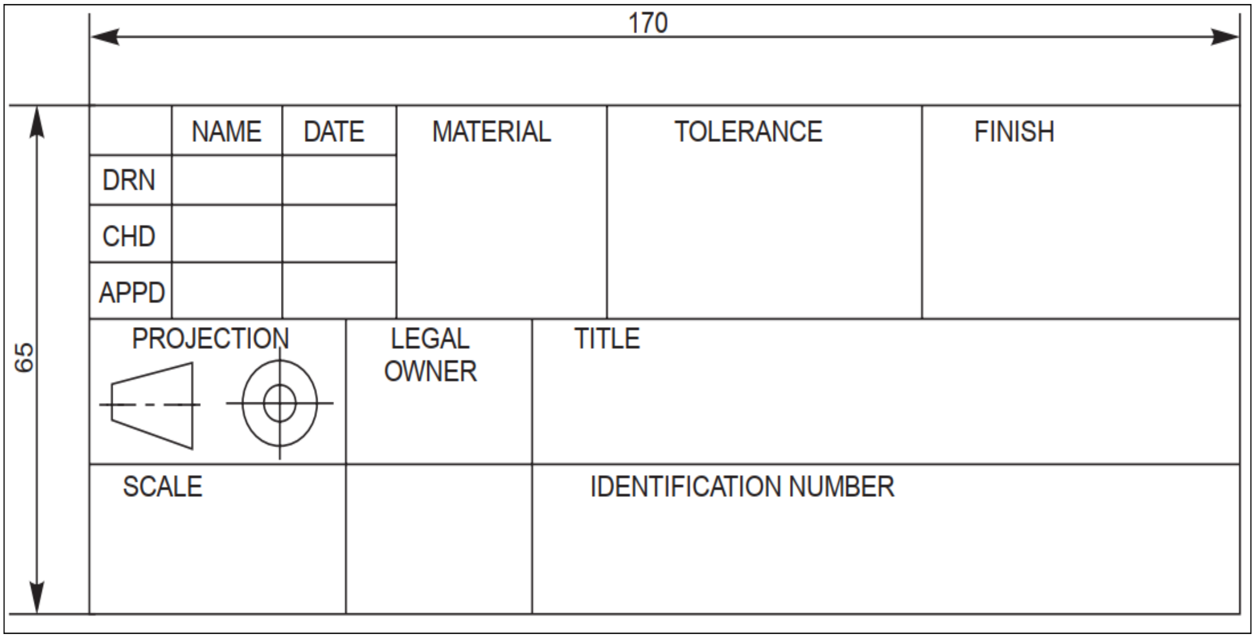


Figure 2.3Details in title block

C. Centering marks

Four centering marks may be provided, in order to facilitate positioning of the drawing when reproduced or microfilmed. Two orientation marks may be provided to indicate the orientation of the drawing sheet on the drawing board (Fig. 2.4).

D. Metric Reference graduation

It is recommended to provide a figure-less metric reference graduation, with a minimum length of 100 mm and divided into 10 intervals on all the drawing sheets (Fig. 2.4) which are intended to be microfilmed. The metric reference graduation may be disposed symmetrically about a centering mark, near the frame at the border, with a minimum width of 5 mm.

E. Grid Reference system (zoning)

The provision of a grid reference system is recommended for all the sizes, in order to permit easy location on the drawing of details, additions, modifications, etc. The number of divisions should be divisible by two and be chosen in relation to the complexity of the drawing. It is recommended that the length of any side of the grid should not be less than 25 mm and not more than 75 mm. The rectangles of the grid should be referenced by means of capital letters along one edge and numerals along the other edge, as shown in Fig. 2.4.

The numbering direction may start at the sheet corner opposite to the title block and be repeated on the opposite sides.

F. Trimming Marks

Trimming marks may be provided in the borders at the four corners of the sheet, in order to facilitate trimming. These marks may be in the form of right angled isosceles triangles or two short strokes at each corner (Fig. 2.4).

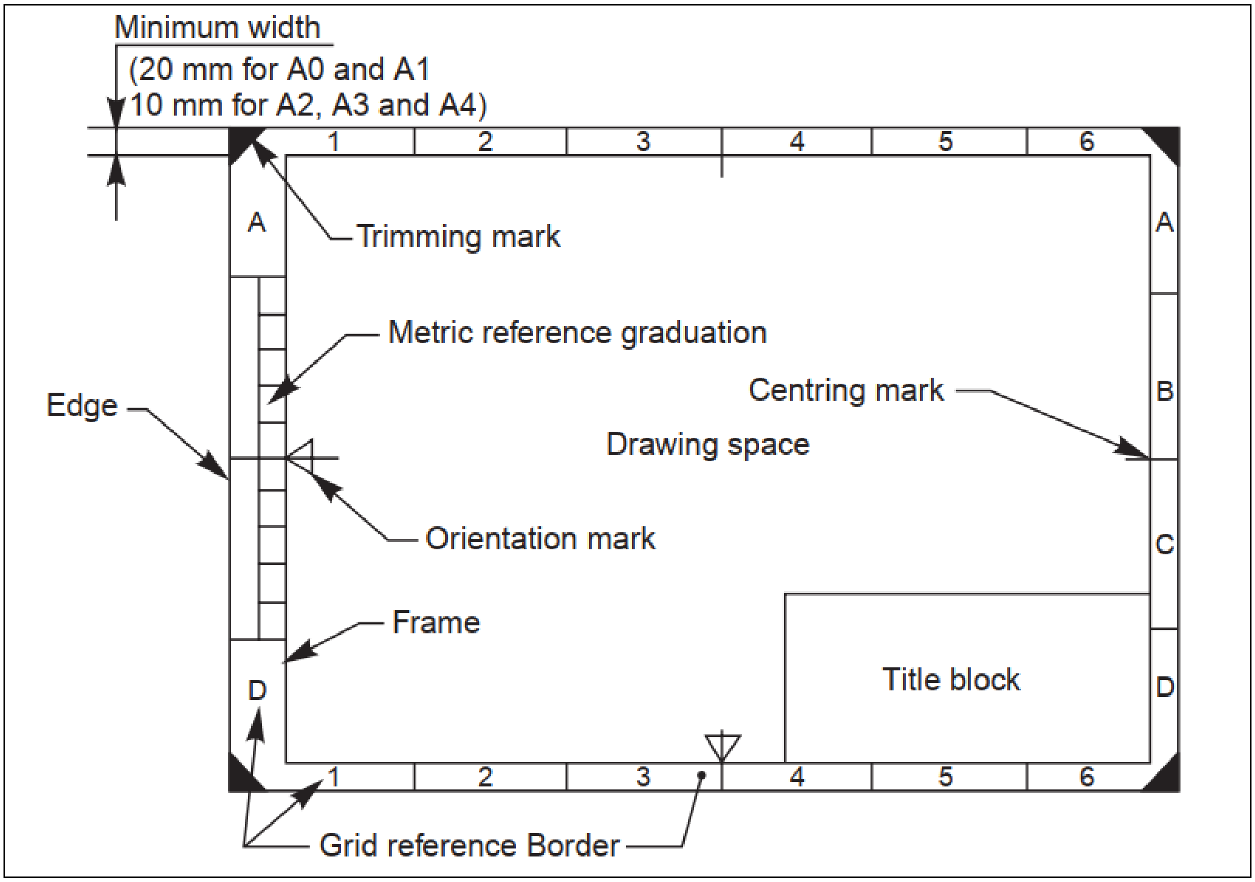


Figure 2.4Drawing Sheet Layouts

2.2.4. Principles on Drawing Lines

In Engineering Graphics, the details of various objects are drawn by different types of lines. Each line has a definite meaning and sense to convey. Lines of different types and thicknesses are used for graphical representation of objects. The types of lines and their applications are shown in Table 2.4. Typical applications of different types of lines are shown in Figure 2.5

A. Thickness of Lines

Two thicknesses of lines are used in draught practice. The ratio of the thick to thin line should not be less than 2:1. The thickness of lines should be chosen according to the size and type of the drawing from the following range:

0.18, 0.25, 0.35, 0.5, 0.7, 1, 1.4 and 2

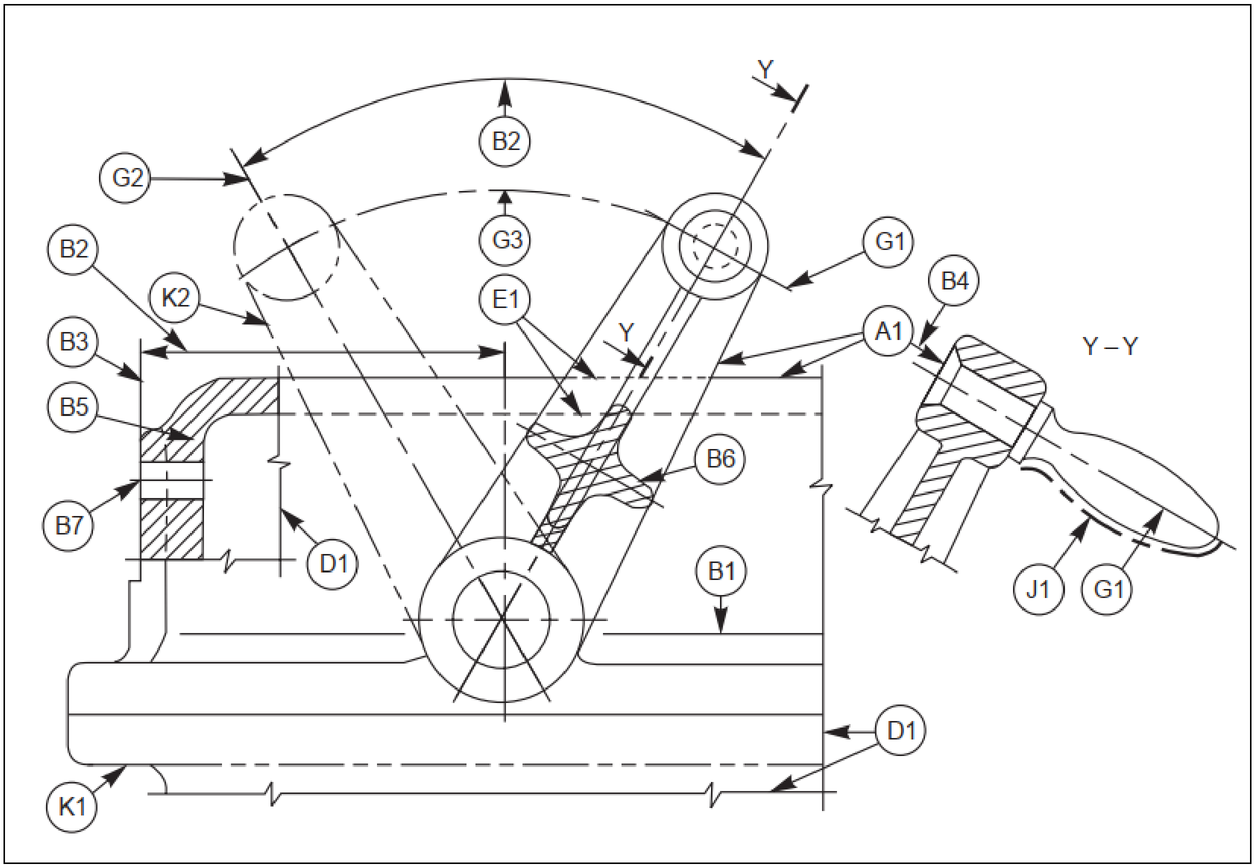
It is recommended that the space between two parallel lines, including hatching, should never be less than 0.7 mm.

Table 2.4 Typical applications of different types of lines

Line	*Description*	*General Applications*
A	Continuous thick	A1 Visible outlines
B	Continuous thin (straight or curved)	B1 Imaginary lines of intersection B2 Dimension lines B3 Projection lines B4 Leader lines B5 Hatching lines B6 Outlines of revolved sections in place B7 Short centre lines
C	Continuous thin, free-hand	C1 Limits of partial or interrupted views and sections, if the limit is not a chain thin
D	Continuous thin (straight) with zigzags	D1 Line (see Fig. 2.5)
E	Dashed thick	E1 Hidden outlines
G	Chain thin	G1 Centre lines G2 Lines of symmetry G3 Trajectories
H	Chain thin, thick at ends and changes of direction	H1 Cutting planes
J	Chain thick	J1 Indication of lines or surfaces to which a special requirement applies
K	Chain thin, double-dashed	K1 Outlines of adjacent parts K2 Alternative and extreme positions of movable parts K3 Centroidal lines

Specify the following types of lines and their applications:

- **Visible Outlines, Visible .Edges:** (Continuous wide lines) the lines drawn to represent the visible outlines/ visible edges / surface boundary lines of objects should be outstanding in appearance.
- **Dimension Lines:** (Continuous narrow Lines) Dimension Lines are drawn to mark dimension.
- **Extension Lines:** (Continuous narrow Lines) There are extended slightly beyond the respective dimension lines.
- **Construction Lines: (**Continuous narrow Lines**)** Construction Lines are drawn for constructing drawings and should not be erased after completion of the drawing.
- **Hatching / Section Lines: (**Continuous Narrow Lines**)** Hatching Lines are drawn for the sectioned portion of an object. These are drawn inclined at an angle of45° to the axis or to the main outline of the section.
- **Guide Lines: (Continuous Narrow Lines)** Guide Lines are drawn for lettering and should not be erased after lettering.
- **Break Lines: (Continuous Narrow Freehand Lines)** Wavy continuous narrow line drawn freehand is used to represent bre~ of an object.
- **Break Lines: (Continuous Narrow Lines with Zigzags)** Straight continuous ~arrow line with zigzags is used to represent break of an object.
- **Dashed Narrow Lines: (Dashed Narrow Lines)**Hidden edges / Hidden outlines of objects are shown by dashed lines of short dashes of equal lengths of about 3 mm, spaced at equal distances of about 1mm. the points of intersection of these lines with the outlines / another hidden line should be clearly shown.
- **Center Lines: (Long-Dashed Dotted Narrow Lines)** Center Lines are drawn at the center of the drawings symmetrical about an axis or both the axes. These are extended by a short distance beyond the outline of the drawing.
- **Cutting Plane Lines:** Cutting Plane Line is drawn to show the location of a cutting plane. It is long-dashed dotted narrow line, made wide at the ends, bends and change of direction. The direction of viewing is shown by means of arrows resting on the cutting plane line.
- **Border Lines** Border Lines are continuous wide lines of minimum thickness 0.7 mm



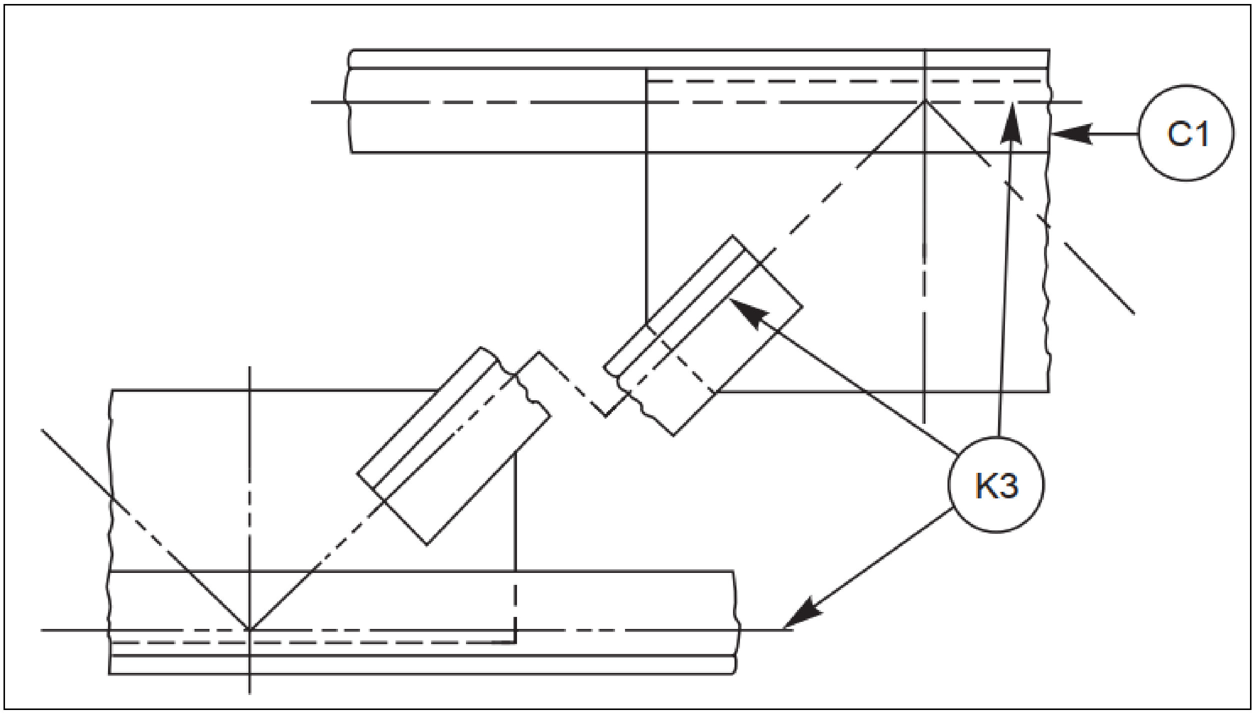


Figure 2.5Applications of Lines in engineering drawing

B. Order Of Priority Of Coinciding Lines

When two or more lines of different types coincide, the following order of priority should be observed:

i. Visible outlines and edges (Continuous thick lines, type A),

ii. Hidden outlines and edges (Dashed line, type E or F),

iii. Cutting planes (Chain thin, thick at ends and changes of cutting planes, type H),

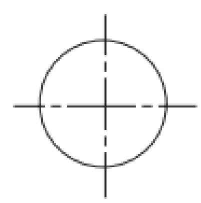
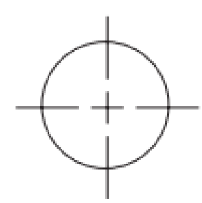
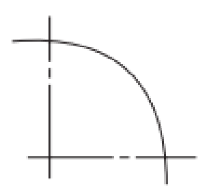
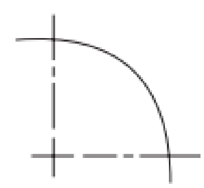
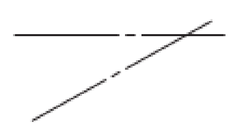
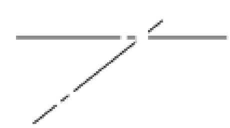
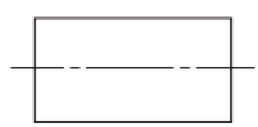
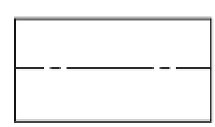
iv. Centre lines and lines of symmetry (Chain thin line, type G),

v. Centroidal lines (Chain thin double dashed line, type K),

vi. Projection lines (Continuous thin line, type B).

The invisible line technique and axis representation should be followed as per there commendations given in Table 2.5

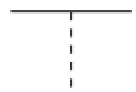
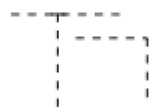
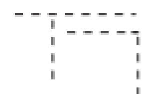
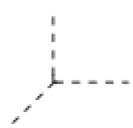
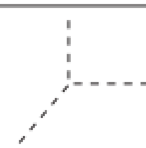
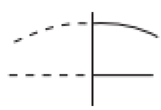
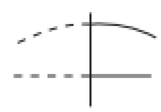
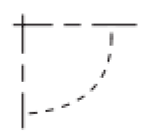
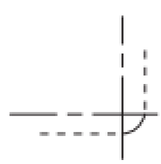
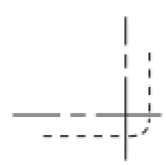
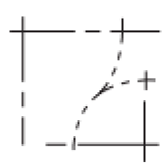
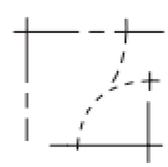
Table 2.5Axis Lines and its ways of techniques

Instructions	*Correct*	*Incorrect*
Axis line starts and ends with a longer dash		
		
Two axes intersect with longer dashes		
		
Axis extends the boundary with a longer dash		

Precedence of Lines

1. When a Visible Line coincides with a Hidden Line or Center Line, draw the Visible Line. Also, extend the Center Line beyond the outlines of the view.
2. When a Hidden Line coincides with a Center Line, draw the Hidden Line.
3. When a Visible Line coincides with a Cutting Plane, draw the Visible Line.
4. When a Center line coincides with a Cutting Plane, draw the Center Line and show the Cutting Plane line outside the outlines of the view at the ends of the Center Line by thick dashes.

Table 2.6Invisible Line and its technique

Instructions	*Correct*	*Incorrect*
Begin with a dash, not with a space		
Dashes intersect without a gap between them		
Three dashes meet at the intersection point		
As a continuation of a visible line/arc, begin with space		
Invisible arcs begin with a dash		
Small arcs may be made solid		
Two arcs meet at the point of tangency		

C. Termination Of Leader Lines

A leader is a line referring to a feature (dimension, object, outline, etc.).Leader lines should terminate (Fig. 2.6),

with a dot, if they end within the outlines of an object,

(a). with an arrow head, if they end on the outline of an object,

(b). Without dot or arrow head, if they end on a dimension line.

It is common practice to omit hidden lines in an assembled view, when their use tends to confuse an already complex drawing or when the feature is sufficiently clear in another view; but it is not advisable for a beginner to do the same and he will have to show the hidden lines in his drawing practice.

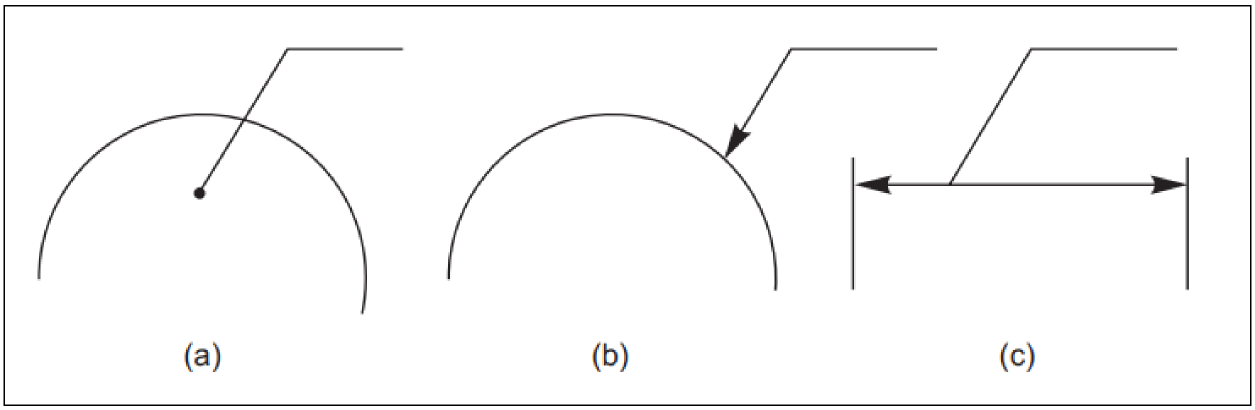


Figure 2.6Termination of leader lines

2.2.5. Principles on Drawing Lettering

Lettering is defined as writing of titles, sub-titles, dimensions, etc., on a drawing. Importance of lettering is to undertake production work of an engineering component as per the drawing; the size and other details are indicated on the drawing. This is done in the font of notes and dimensions.

The **essential features of lettering** on engineering drawings are legibility, uniformity and suitability for microfilming and other photographic reproductions. In order to meet these requirements, the characters are to be clearly distinguishable from each other in order to avoid any confusion between them, even in the case of slight mutilations. The reproductions require the distance between two adjacent lines or the space between letters to be at least equal to twice the line thickness (Fig. 2.7). The line thickness for lower case and capital letters shall be the same in order to facilitate lettering.

Use of drawing instruments for lettering consumes more time. Lettering should be done freehand with speed. Practice accompanied by continuous efforts would improve the lettering skill and style. Poor lettering mars the appearance of an otherwise good drawing.

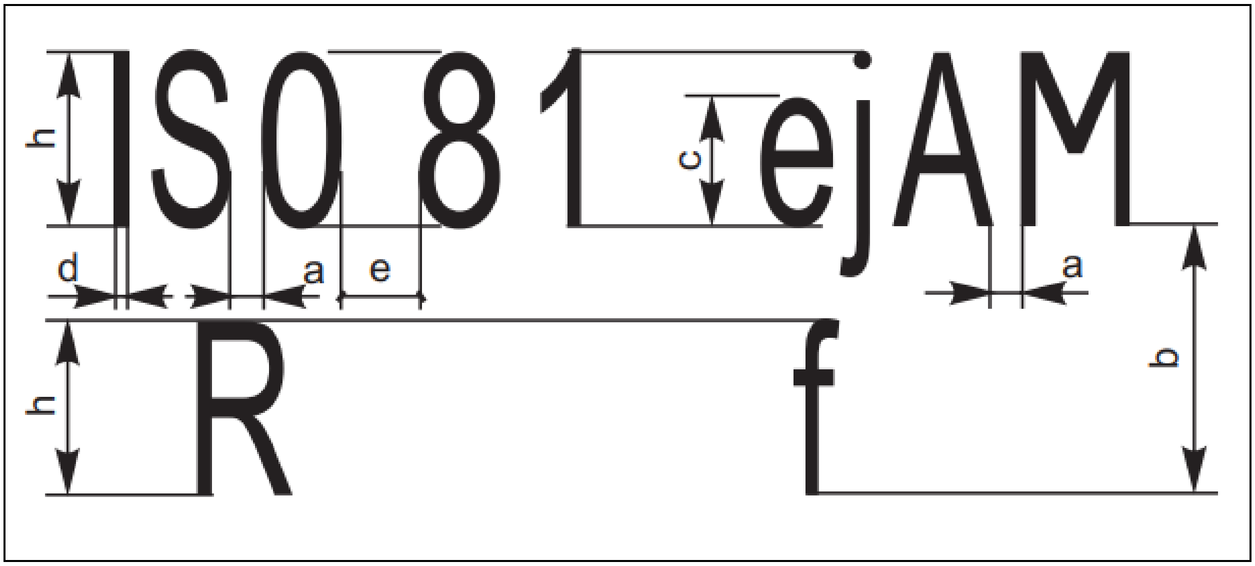


Figure 2.7 dimensioning on letter

A. Dimensions in lettering

The following specifications are given for the dimensions of letters and numerals:

(i). The height of capital letters is taken as the base of dimensioning (Tables 2.6 and 2.7).

(ii). (The two standard ratios for *d/h*, 1/14 and 1/10 are the most economical, as they result in a minimum number of line thicknesses.

(iii). The lettering may be inclined at 15° to the right, or may be vertical.

Table 2.7 Lettering A ($d = h/14$)

Characteristic	*Ratio*	*Dimensions*, (mm)						
Lettering height (Height of capitals)	*h* (14/14)*h*	2.5	3.5	5	7	10	14	20
Height of lower-case letters (without stem or tail)	*c* (10/14)*h*	—	2.5	3.5	5	7	10	14
Spacing between characters	*a (2/14)h*	0.35	0.5	0.7	1	1.4	2	2.8
Minimum spacing of base lines	*b (20/14)h*	3.5	5	7	10	14	20	28
Minimum spacing between words	*e (6/14)h*	1.05	1.5	2.1	3	4.2	6	8.4
Thickness of lines	*d (1/14)h*	0.18	0.25	0.35	0.5	0.7	1	1.4

NOTE The spacing between two characters may be reduced by half, if this gives a better visual effect as for example LA, TV; it then equals the line thickness.

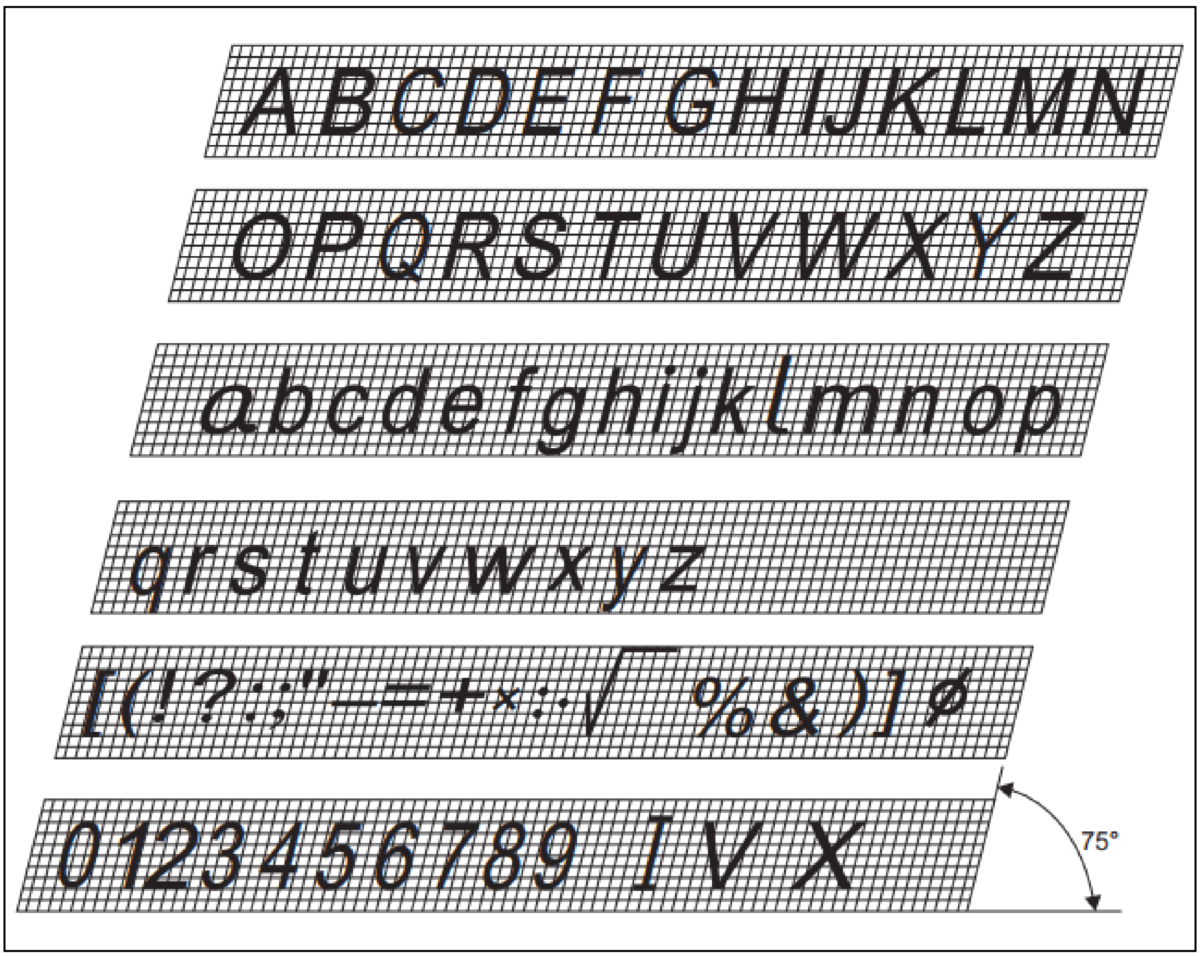


Figure 2.8inclined lettering

Table 2.8Lettering B ($d = h/10$)

Characteristic	Ratio	Dimensions, (mm)						
Lettering height (Height of capitals)	h (10/10)h	2.5	3.5	5	7	10	14	20
Height of lower-case letters (without stem or tail)	c (7/10)h	—	2.5	3.5	5	7	10	14
Spacing between characters	*a (2/10)h*	0.5	0.7	1	1.4	2	2.8	4
Minimum spacing of base lines	*b (14/10)h*	3.5	5	7	10	14	20	28
Minimum spacing between words	*e (6/14)h*	1.5	2.1	3	4.2	6	8.4	12
Thickness of lines	*d (1/10)h*	0.25	0.35	0.5	0.7	1	1.4	2

Figures 2.8 and 2.9 show the specimen letters of type A, inclined and vertical and are given only as a guide to illustrate the principles mentioned above.

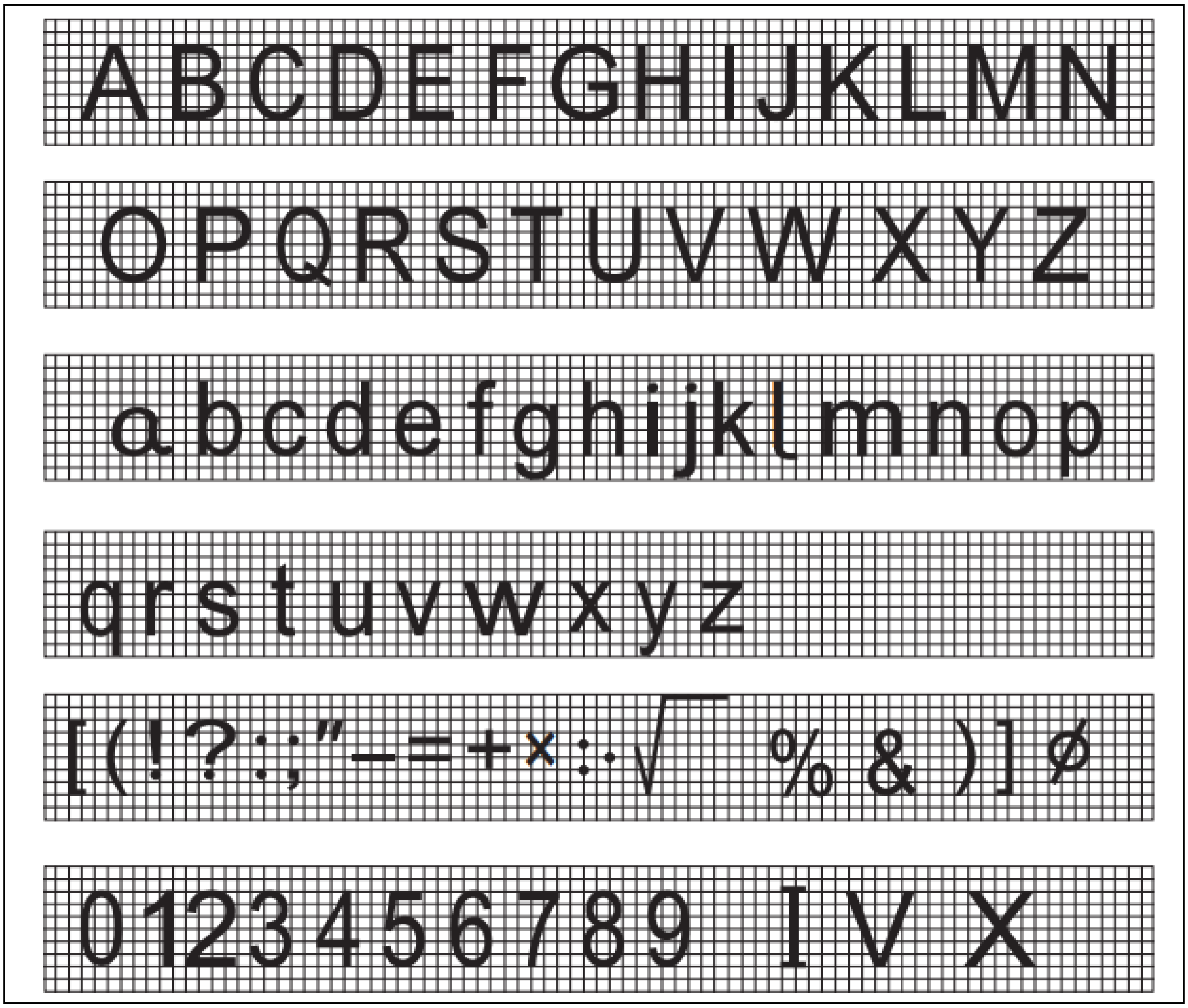


Figure 2.9vertical lettering

2.2.6. Principles on Drawing Sections

In order to show the inner details of a machine component, the object is imagined to be cut by a cutting plane and the section is viewed after the removal of cut portion. Sections are made by at cutting planes and are designated by capital letters and the direction of viewing is indicated by arrow marks.

A. Hatching of section

Hatching is generally used to show areas of sections. The simplest form of hatching is generally adequate for the purpose, and may be continuous thin lines (type B) at a convenient angle, preferably 45°, to the principal outlines or lines of symmetry of the sections (Fig. 2.10).

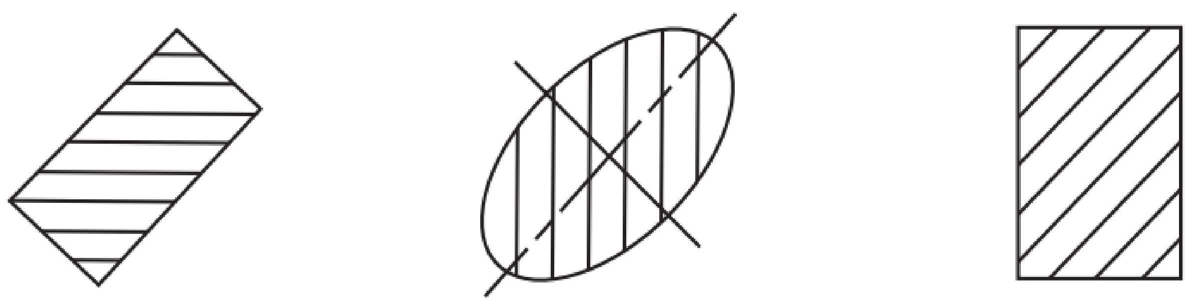


Figure 2.10Preferred hatching angles

Separate areas of a section of the same component shall be hatched in an identical manner. The hatching of adjacent components shall be carried out with different directions or spacing (Fig 2.11 a). In case of large areas, the hatching may be limited to a zone, following the contour of the hatched area (Fig. 2.11 b).

Where sections of the same part in parallel planes are shown side by side, the hatching shall be identical, but may be off-set along the dividing line between the sections (Fig. 2.12). Hatching should be interrupted when it is not possible to place inscriptions outside the hatched area (Fig. 2.13).

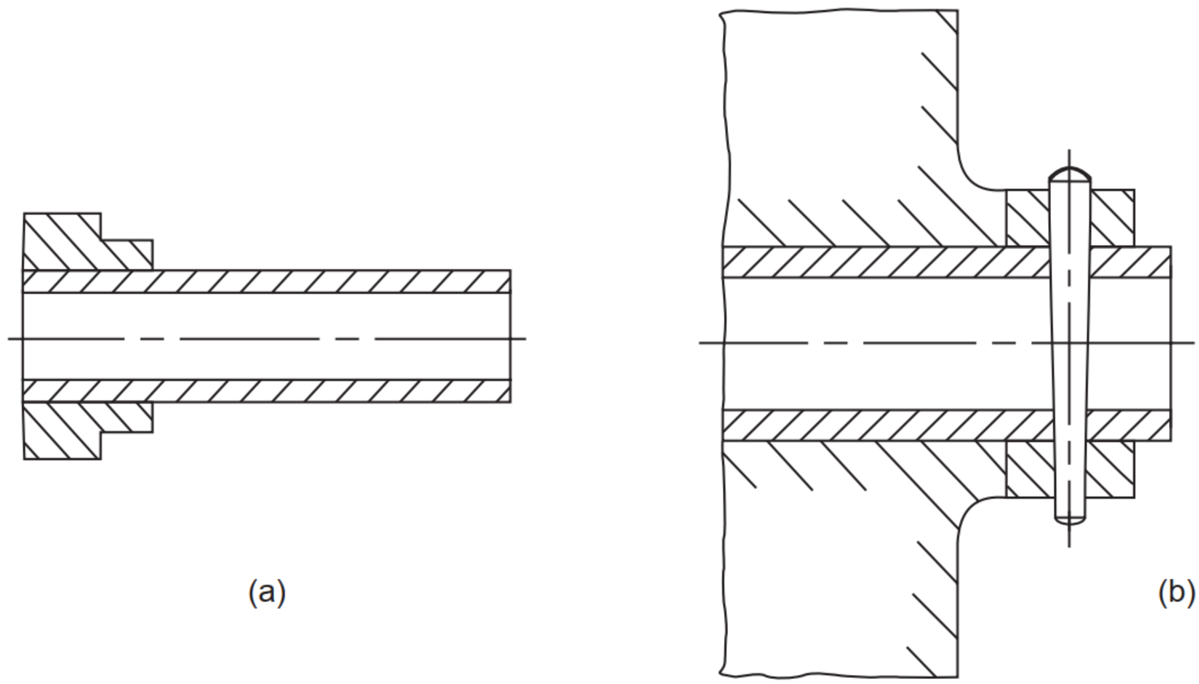
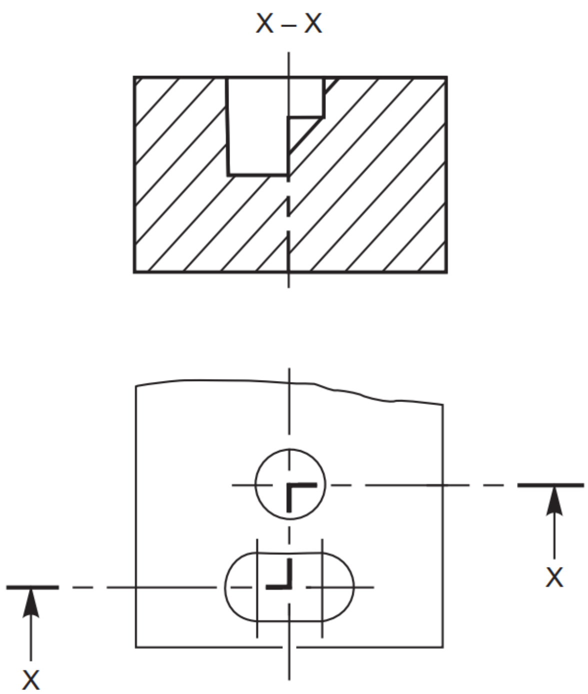


Figure 2.11Hatching of adjacent components



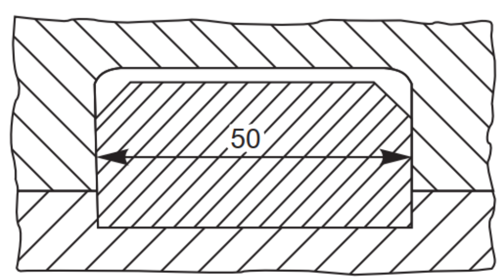


Figure 2.12 Sectioning along two parallel planes

Figure 2.13 Hatching interrupted for dimensioning

B. Cutting Planes

The cutting plane(s) should be indicated by means of type H line. The cutting plane should be identified by capital letters and the direction of viewing should be indicated by arrows. The section should be indicated by the relevant designation (Fig. 2.14).

In principle, ribs, fasteners, shafts, spokes of wheel sand the like are not cut in longitudinal sections and therefore should not be hatched (Fig. 2.15).

Figure 2.16 represents sectioning in two parallel plane sand Fig. 2.17, that of sectioning in three continuous planes.

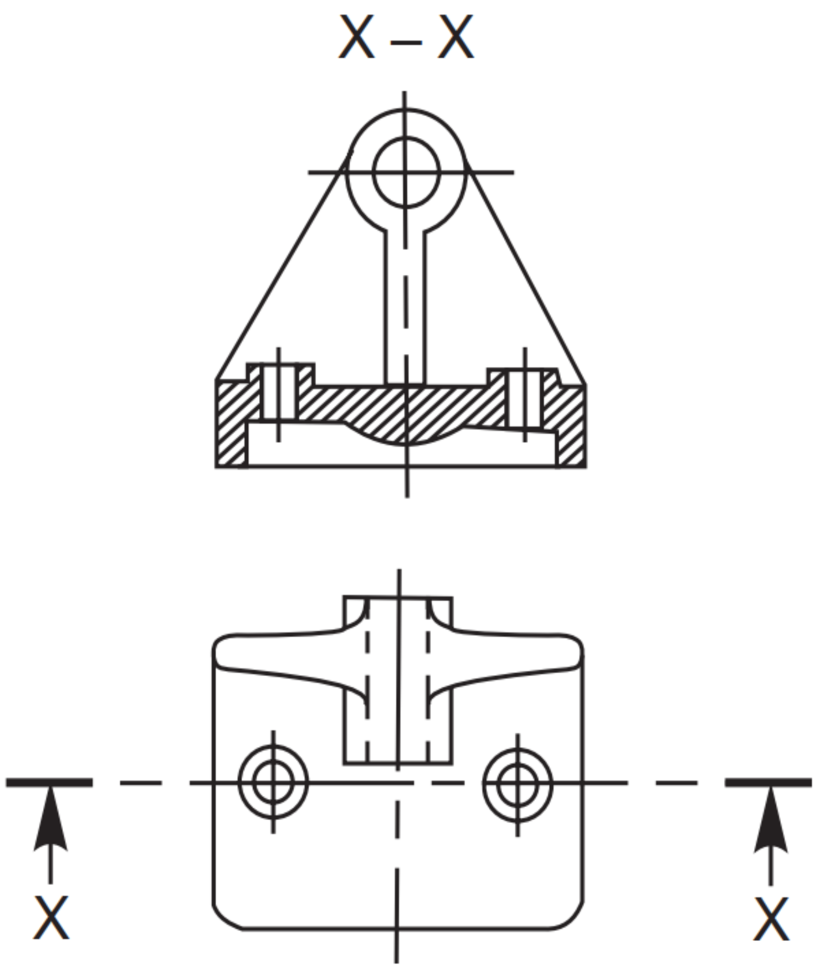


Figure 2.14Cutting plane indication

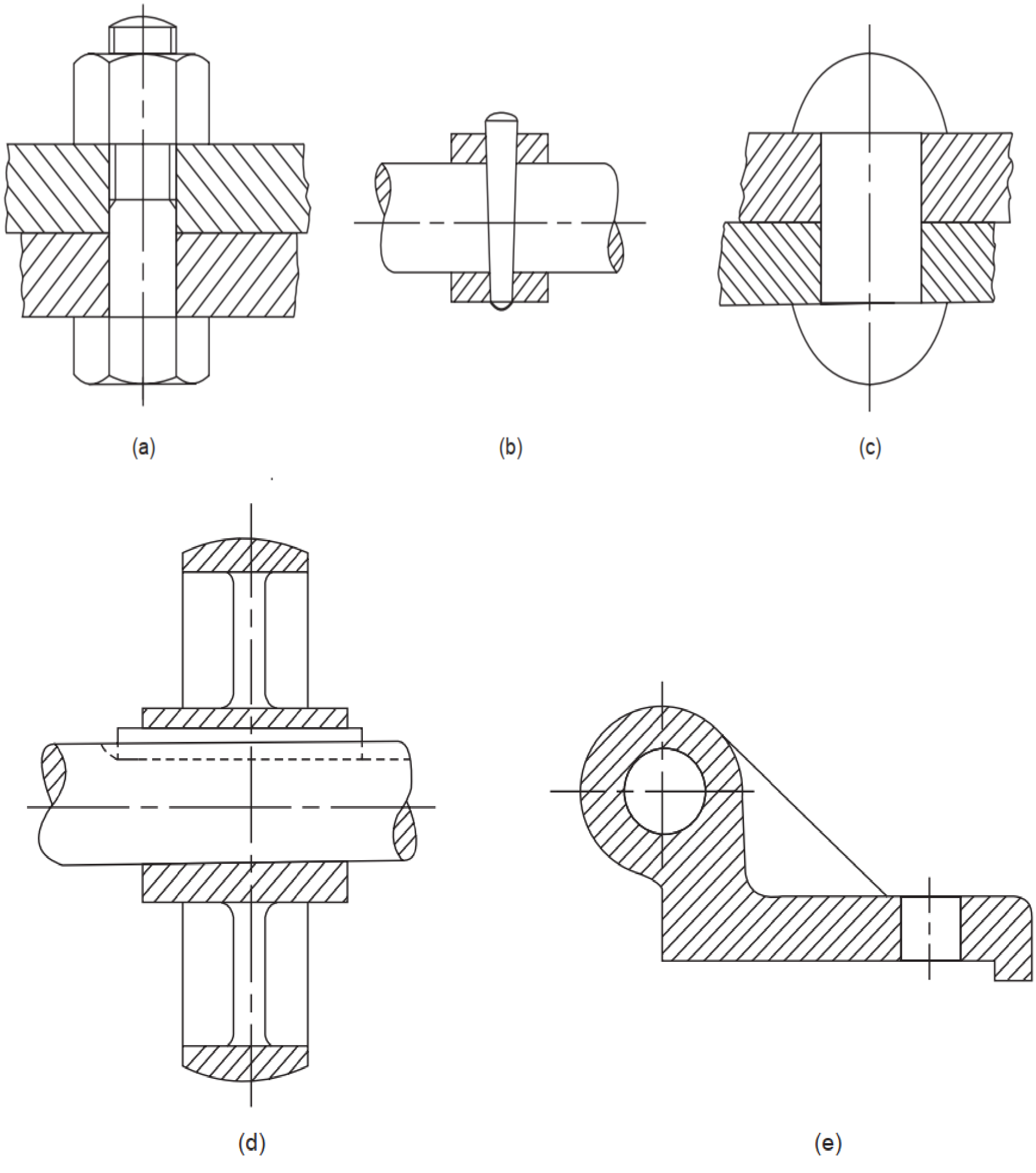


Figure 2.15Sections not to be hatched

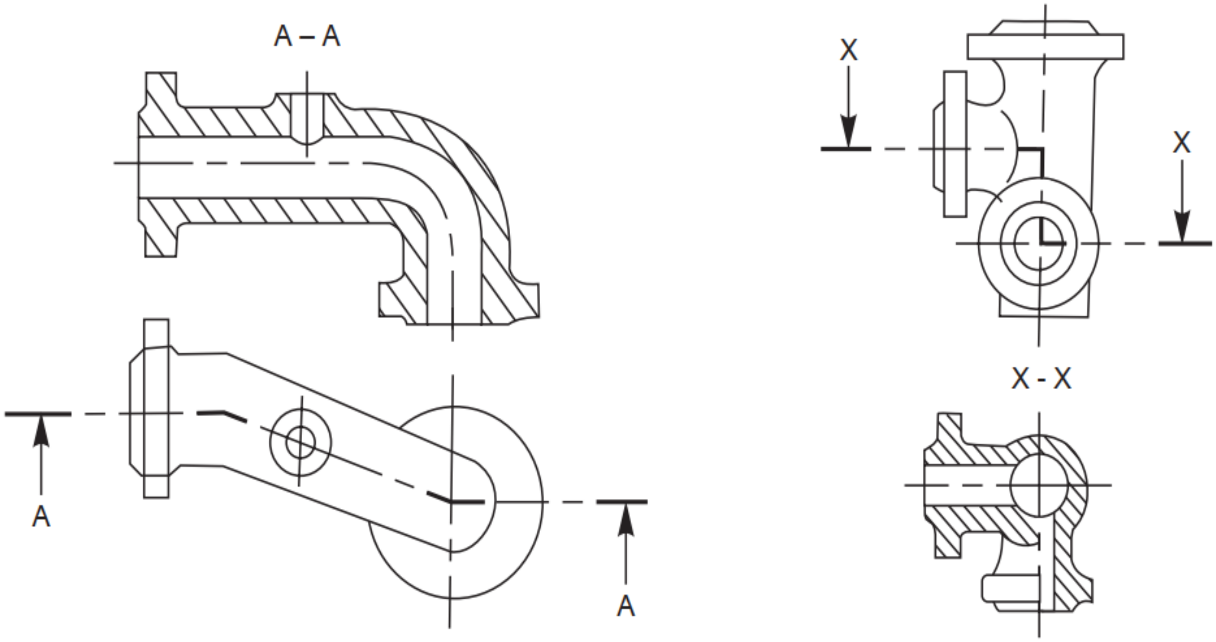


Figure 2.16 sectioning in three continuous planes.

Sectioning in two intersecting planes, in which one is shown revolved into plane of projection, as shown in Fig. 2.18.

In case of parts of revolution, containing regularly spaced details that require to be shown in section, but are not situated in the cutting plane; such details may be depicted by rotating them into the cutting plane (Fig. 2.19).

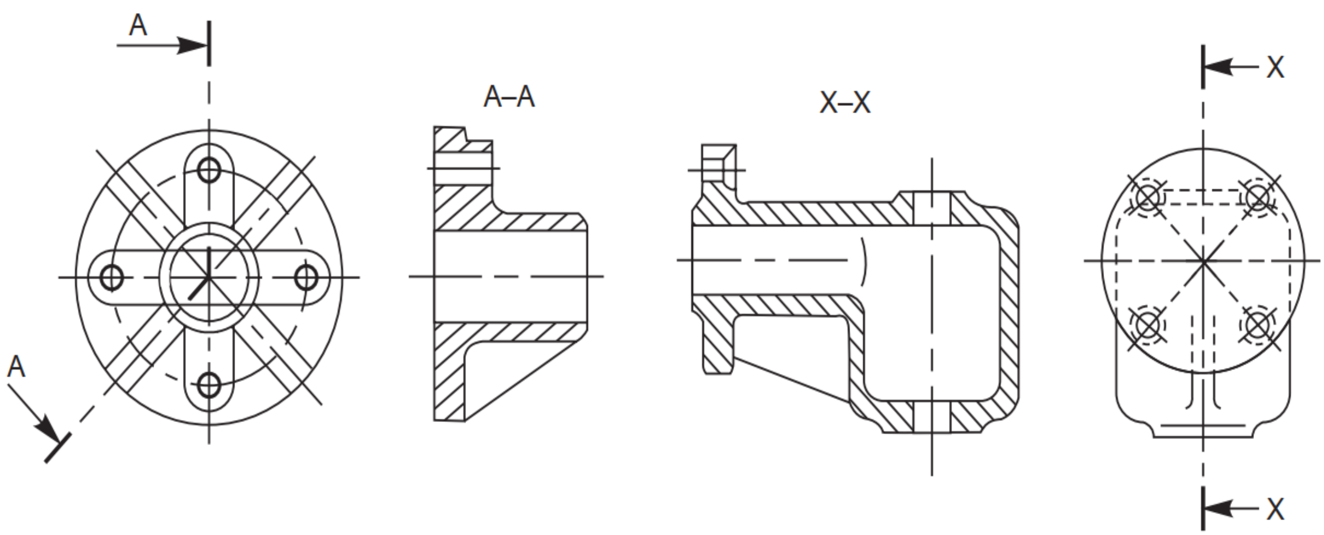


Figure 2.17Sectioning in two intersecting planes

Figure 2.18 rotating them into the cutting plane

C. Revolved or Removed Section

Cross sections may be resolved in the relevant view or removed. When revolved in the relevant view, the outline of the section should be shown with continuous thin lines (Fig. 2.19). When removed, the outline of the section should be drawn with continuous thick lines. The removed section may be placed near to and connected with the view by a chain thin line (Fig. 2.20*a*) or in a different position and identified in the conventional manner, as shown in Fig. 2.20*b*.

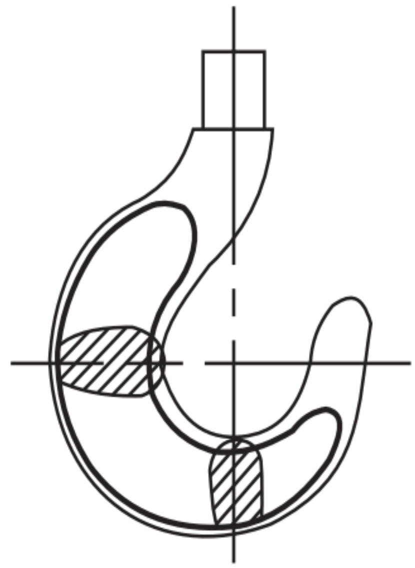


Figure 2.19Revolved section

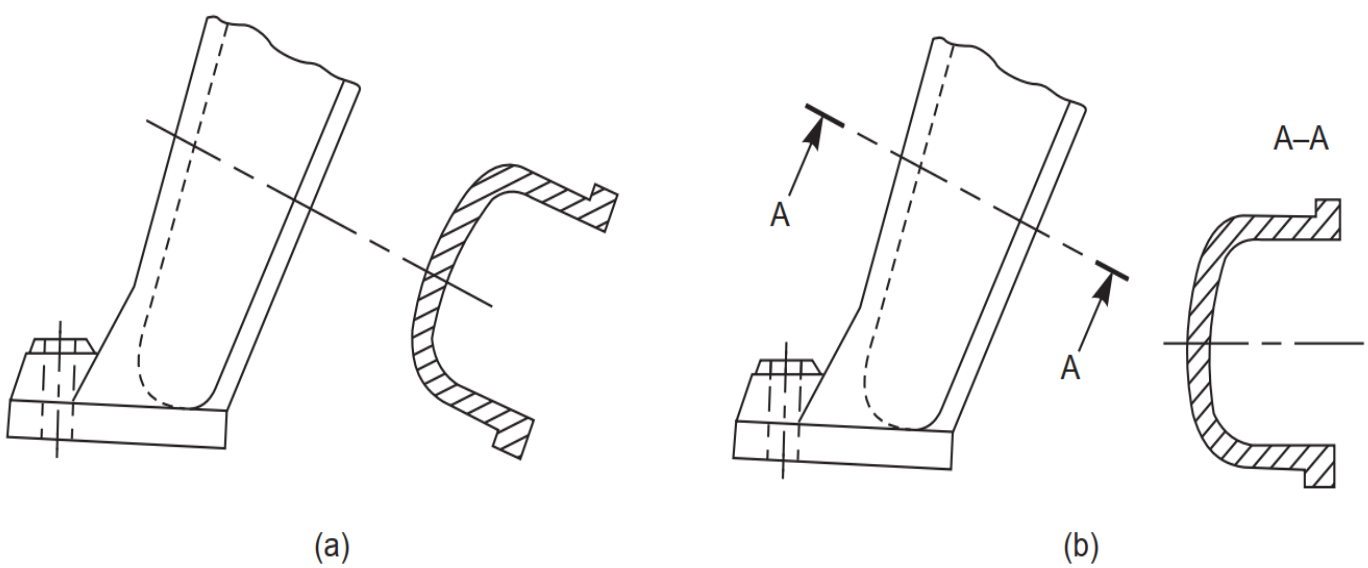


Figure 2.20Removed section

D. Half Section

Symmetrical parts may be drawn, half in plain view and half in section (Fig 2.21).

E. Local Section

A local section may be drawn if half or full section is not convenient. The local break may be shown by a continuous thin free hand line (Fig. 2.22).

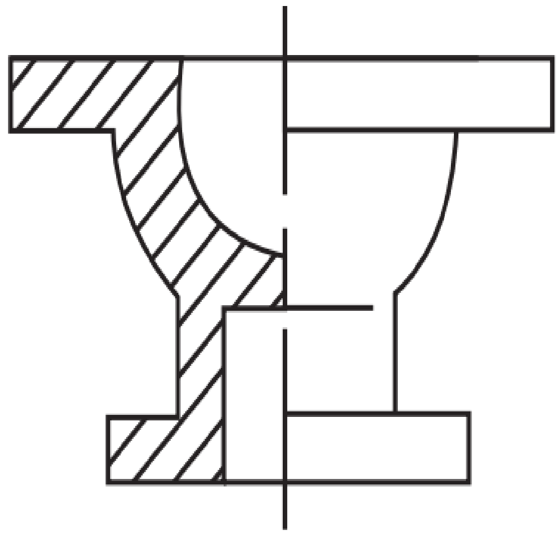


Figure 2.21 Half section

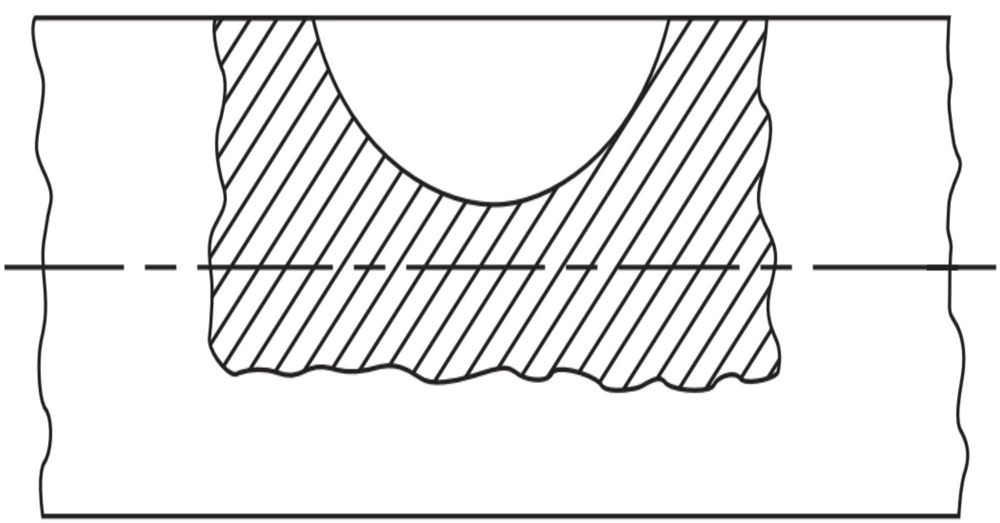


Figure 2.22Local section

F. Arrangement of Successive Section

Successive sections may be placed separately, with designations for both cutting planes and sections (Fig. 2.25) or may be arranged below the cutting planes.

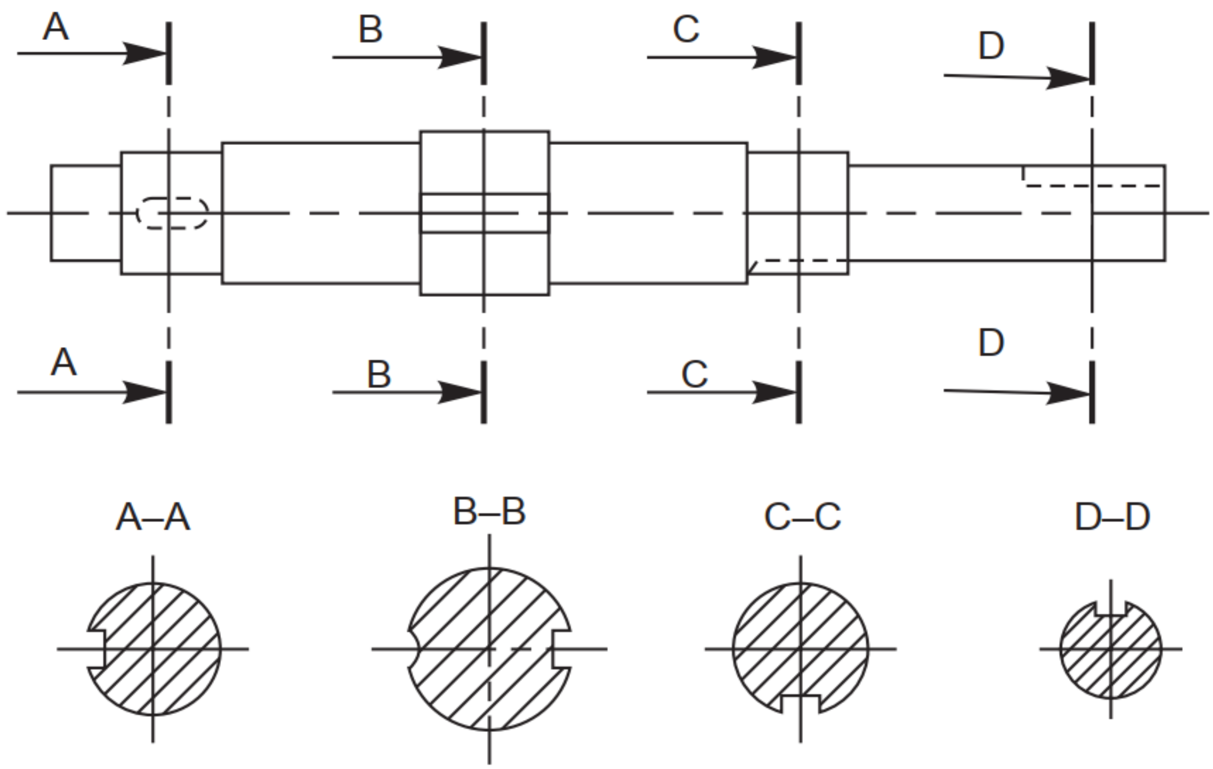


Figure 2.23 Successive sections

2.2.7. Principles on Conventional Representation

Certain draughting conventions are used to represent materials in section and machine elements in engineering drawings.

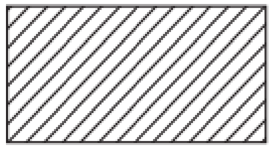
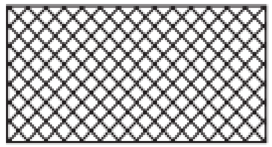
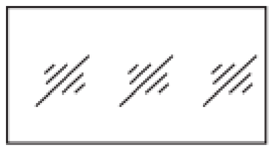
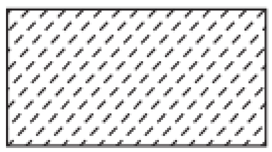
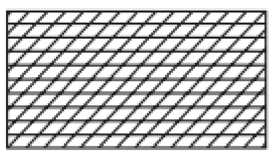
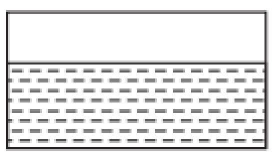
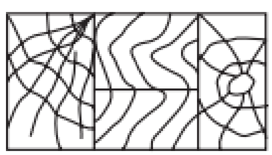
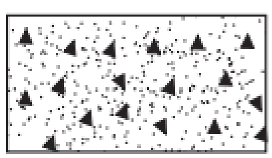
A. Materials

As a variety of materials are used for machine components in engineering applications, it is preferable to have different conventions of section lining to differentiate between various materials. The recommended conventions in use are shown in table 2.9.

B. Machine Component

When the drawing of a component in its true projection involves a lot of time, its convention may be used to represent the actual component. Table 2.9 shows typical examples of conventional representation of various machine components used in engineering drawing.

Table 2.9 Conventional representation of materials

Type	Convention	Material
Metals		Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.
		Lead, Zinc, Tin, White-metal, etc.
Glass		Glass
Packing and Insulating material		Porcelain, Stoneware, Marble, Slate, etc.
		Asbestos, Fibre, Felt, Synthetic resin products, Paper, Cork, Linoleum, Rubber, Leather, Wax, Insulating and Filling materials, etc.
Liquids		Water, Oil, Petrol, Kerosene, etc.
Wood		Wood, Plywood, etc.
Concrete		A mixture of Cement, Sand and Gravel

2.3. Dimensions, notes and specifications in the drawing

2.3.1. Dimensioning

A. General principle

A drawing of a component, in addition to providing complete shape description, must also furnish information regarding the size description. These are provided through the distances between the surfaces, location of holes, nature of surface finish, type of material, etc. The expression of these features on a drawing, using lines, symbols, figures and notes is called dimensioning.

Dimension is a numerical value expressed in appropriate units of measurement and indicated on drawings, using lines, symbols, notes, etc., so that all features are completely defined.

1. As far as possible, dimensions should be placed outside the view.
2. Dimensions should be taken from visible outlines rather than from hidden lines.
3. Dimensioning to a centre line should be avoided except when the centre line passes through the centre of a hole.
4. Each feature should be dimensioned once only on a drawing.
5. Dimensions should be placed on the view or section that relates most clearly to the corresponding features.
6. Each drawing should use the same unit for all dimensions, but without showing the unit symbol.
7. No more dimensions than are necessary to define a part should be shown on a drawing.
8. No features of a part should be defined by more than one dimension in any one direction.

B. Method of Execution

The elements of dimensioning include the projection line, dimension line, leader line, dimension line termination, the origin indication and the dimension itself. The various elements of dimensioning are shown in Figs. 2.24. The following are some of the principles to be adopted during execution of dimensioning:

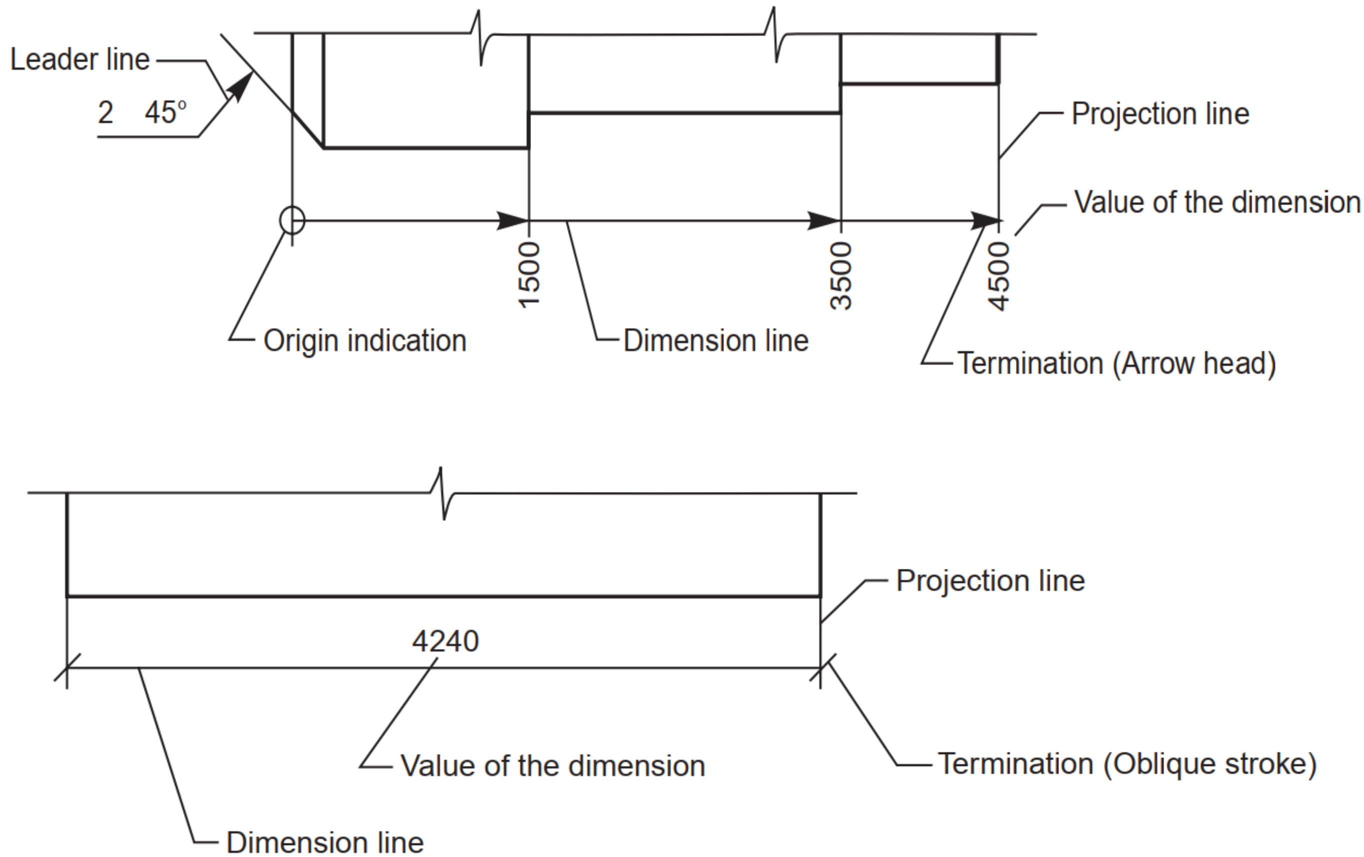
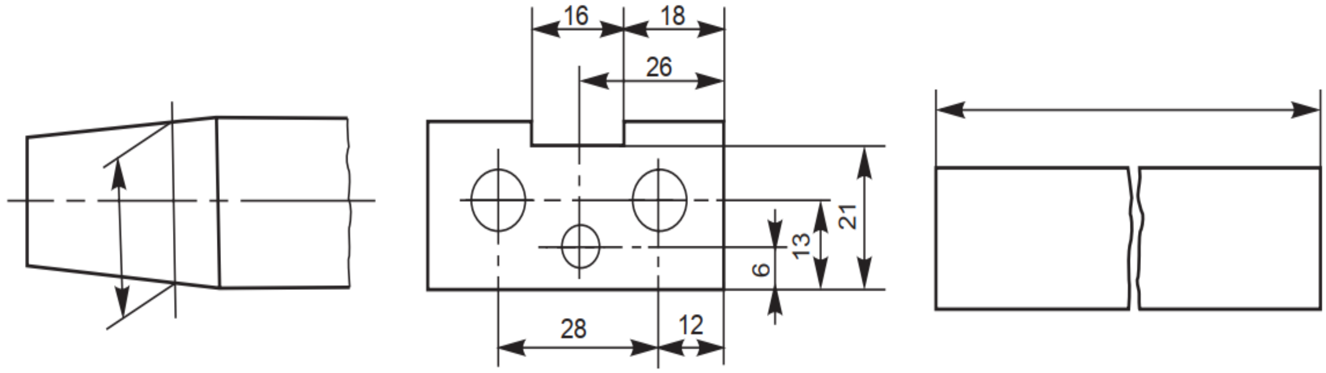


Figure 2.24 Elements of dimensioning

1. Projection and dimension lines should be drawn as thin continuous lines.
2. Projection lines should extend slightly beyond the respective dimension lines.
3. Projection lines should be drawn perpendicular to the feature being dimensioned. Where necessary, they may be drawn obliquely, but parallel to each other (Fig. 2.30). However, they must be in contact with the feature.
4. Projection lines and dimension lines should not cross each other, unless it is unavoidable (Fig. 2.25 a).
5. A dimension line should be shown unbroken, even where the feature to which it refers, is shown broken (Fig. 2.25 b).
6. A centre line or the outline of a part should not be used as a dimension line, but may be used in place of projection line (Fig. 2.25 c).



(a) (b) (c)

Figure 2.25 Principle of Projection and dimension

Termination and Original indication

Dimension lines should show distinct termination, in the form of arrow heads or oblique strokes or where applicable, an origin indication. Two dimension line terminations and an origin indication are shown in Fig. 2.26. In this,

1. The arrow head is drawn as short lines, having an included angle of 15°, which is closed and filled-in.
2. The oblique stroke is drawn as a short line, inclined at 45°.
3. The origin indication is drawn as a small open circle of approximately 3 mm in diameter.

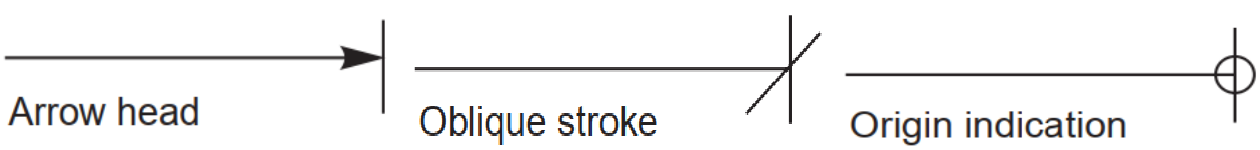
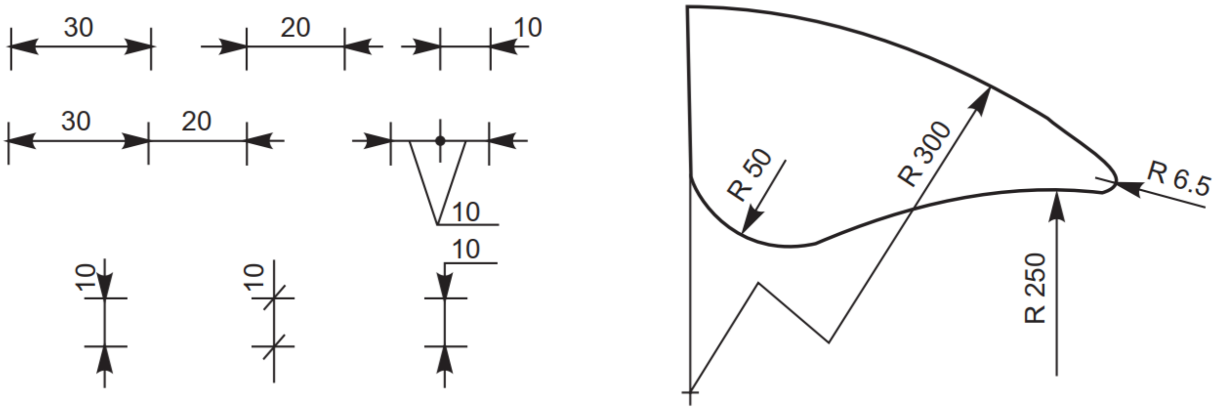


Figure 2.26 line terminations and an origin indication

The size of the terminations should be proportionate to the size of the drawing on which they are used. Where space is limited, arrow head termination may be shown outside the intended limits of the dimension line that is extended for that purpose. In certain other cases, an oblique stroke or a dot may be substituted (Fig. 2.27 a).

Where a radius is dimensioned, only one arrow head termination, with its point on the arc end of the dimension line, should be used (Fig. 2.27 b). However, the arrow head termination may be either on the inside or outside of the feature outline, depending upon the size of feature.



(a) (b)

Figure 2.27 size of the terminations

C. Method of Indicating dimensions

Dimensions should be shown on drawings in characters of sufficient size, to ensure complete legibility. They should be placed in such a way that they are not crossed or separated by any other line on the drawing. Dimensions should be indicated on a drawing, according to one of the following two methods. However, only one method should be used on any one drawing.

METHOD–1 (Aligned System)

Dimensions should be placed parallel to their dimension lines and preferably near the middle, above and clear-off the dimension line (Fig. 2.36). An exception may be made where superimposed running dimensions are used (Fig. 2.44 b)

Dimensions may be written so that they can be read from the bottom or from the right side of the drawing. Dimensions on oblique dimension lines should be oriented as shown in Fig. 2.37. Angular dimensions may be oriented as shown in Fig. 2.38.

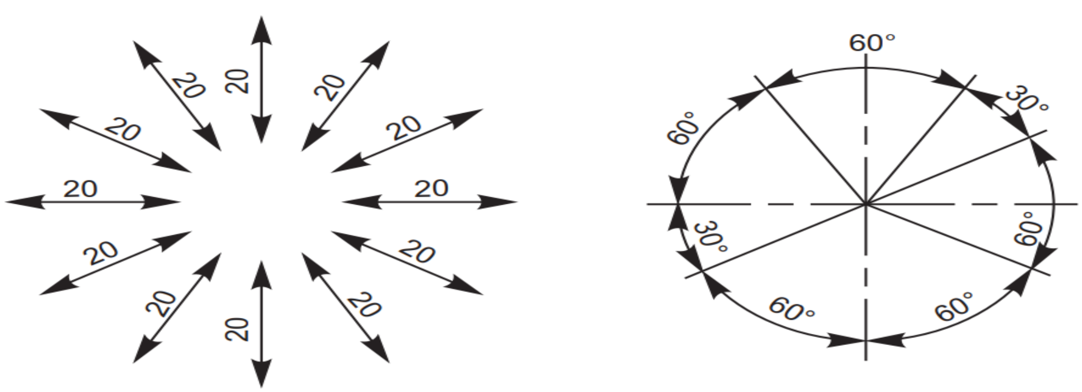


Figure 2.28Oblique dimensioning and Angular dimensioning

METHOD–2 (Uni-directional System)

Dimensions should be indicated so that they can be read from the bottom of the drawing only. Non-horizontal dimension lines are interrupted, preferably near the middle, for insertion of the dimension (Fig. 2.39).

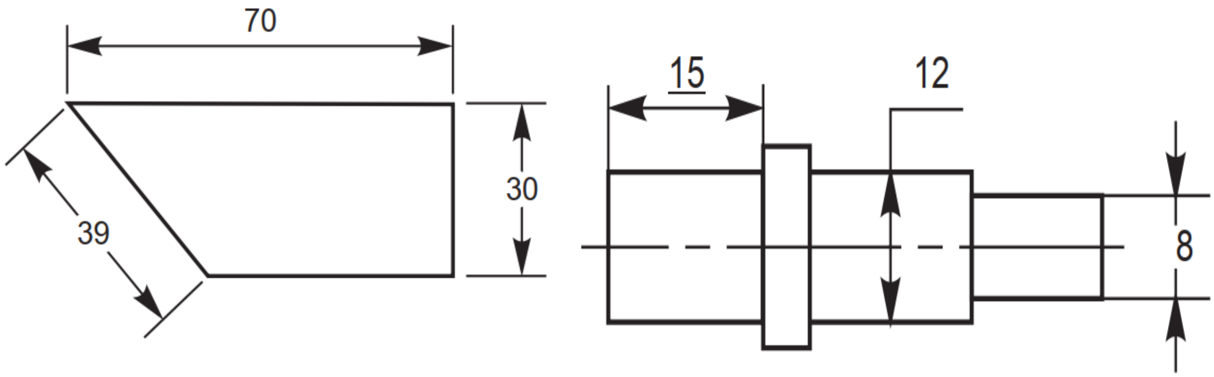


Figure 2.29 Non-horizontal dimension line

Dimensions can be,

(i) above the extension of the dimension line, beyond one of the terminations, where space is limited (Fig. 2.34) or

(ii) at the end of a leader line, which terminates on a dimension line, that is too short to permit normal dimension placement (Fig. 2.34) or

(iii) Above a horizontal extension of a dimension line, where space does not allow placement at the interruption of anon-horizontal dimension line (Fig. 2.41). Values of dimensions, out of scale (except where break lines are used) should be underlined as shown in Fig. 2.41.

The following indications (symbols) are used with dimensions to reveal the shape identification and to improve drawing interpretation. The symbol should precede the dimensions (Fig. 2.30).

φ : Diameter Sφ : Spherical diameter R : Radius SR : Spherical radius : □ Square

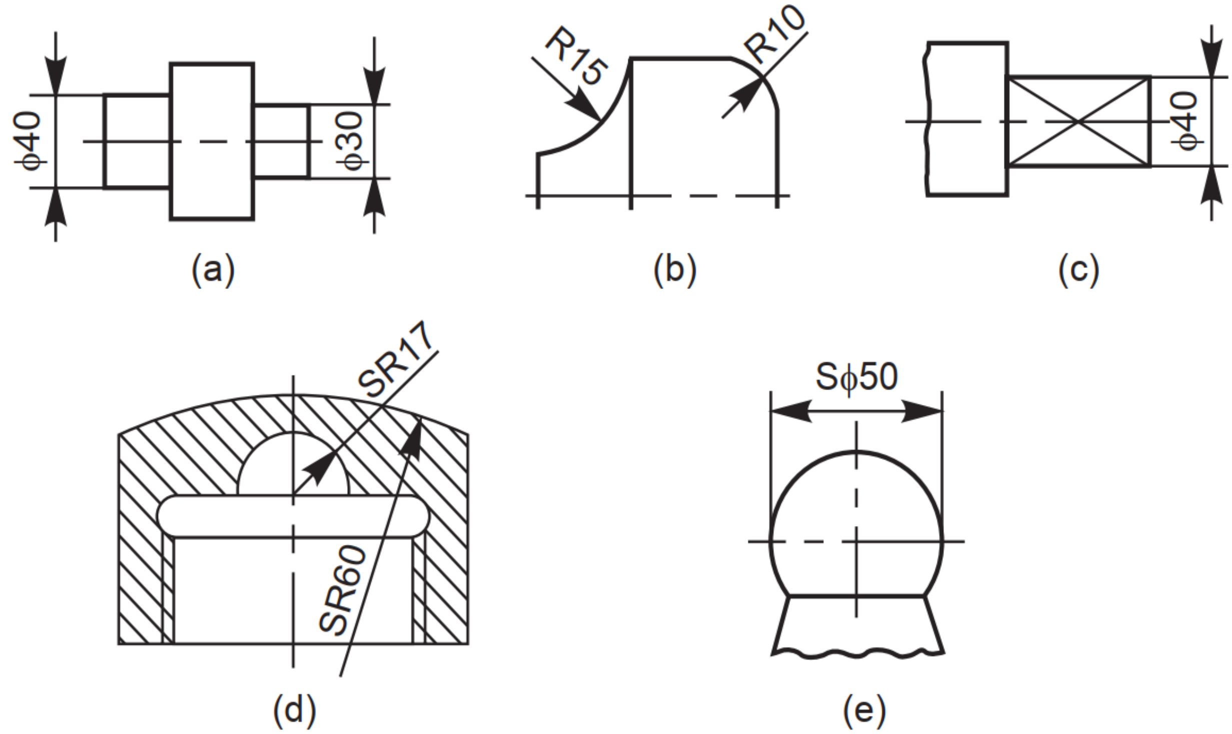


Figure 2.30 Shape identification symbols

D. Arrangement of Dimensions

The arrangement of dimensions on a drawing must indicate clearly the design purpose. The following are the ways of arranging the dimensions.

Chains dimensions

Chains of single dimensions should be used only where the possible accumulation of tolerances does not end anger the functional requirement of the part (Fig. 2.43)

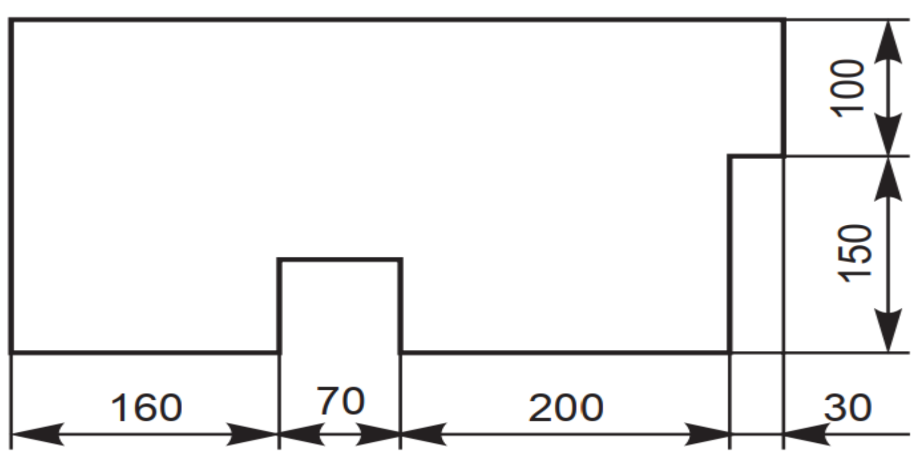


Figure 2.31 Chain dimensioning

Parallel dimension

In parallel dimensioning, a number of dimension lines, parallel to one another and spaced-out are used. This method is used where a number of dimensions have a common datum feature (Fig. 2.44 *a*).

Supper impose running dimensioning

These are simplified parallel dimensions and may be used where there are space limitations (Fig. 2.44 *b*).

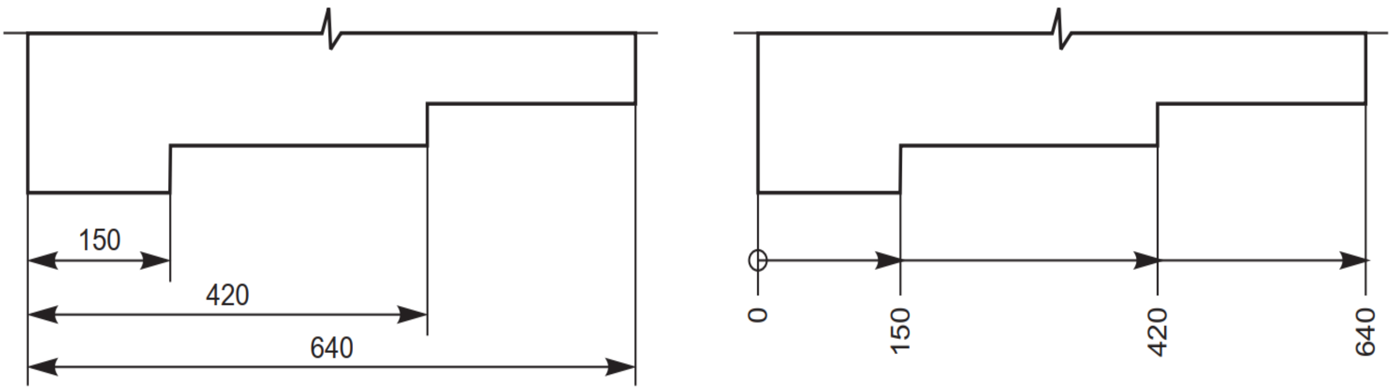


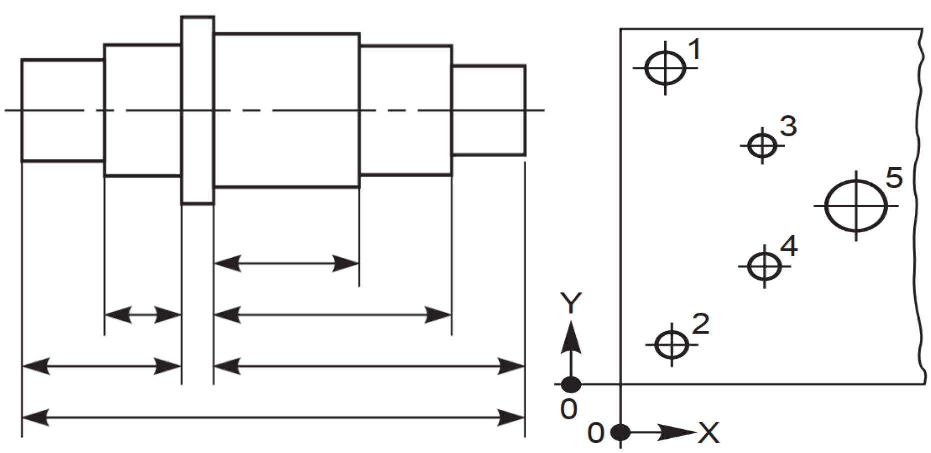
Figure 2.32 Parallel dimension and Supper impose running dimensioning

Combined Dimension

These are the result of simultaneous use of chain and parallel dimensions (Fig. 2.45).

Co-ordinate dimension

The sizes of the holes and their co-ordinates may be indicated directly on the drawing; or they may be conveniently presented in a tabular form, as shown in Fig. 2.46.



	X	Y	ϕ
1	20	160	15.5
2	20	20	13.5
3	60	120	11
4	60	60	13.5
5	100	90	26
6			
7			
8			
9			
10			

Figure 2.33 Combined dimensioning and Co-ordinate dimensioning

E. Special Indication

Diameter

Diameters should be dimensioned on the most appropriate view to ensure clarity. The dimension value should be preceded by φ. Figure 2.47 shows the method of dimensioning diameters.

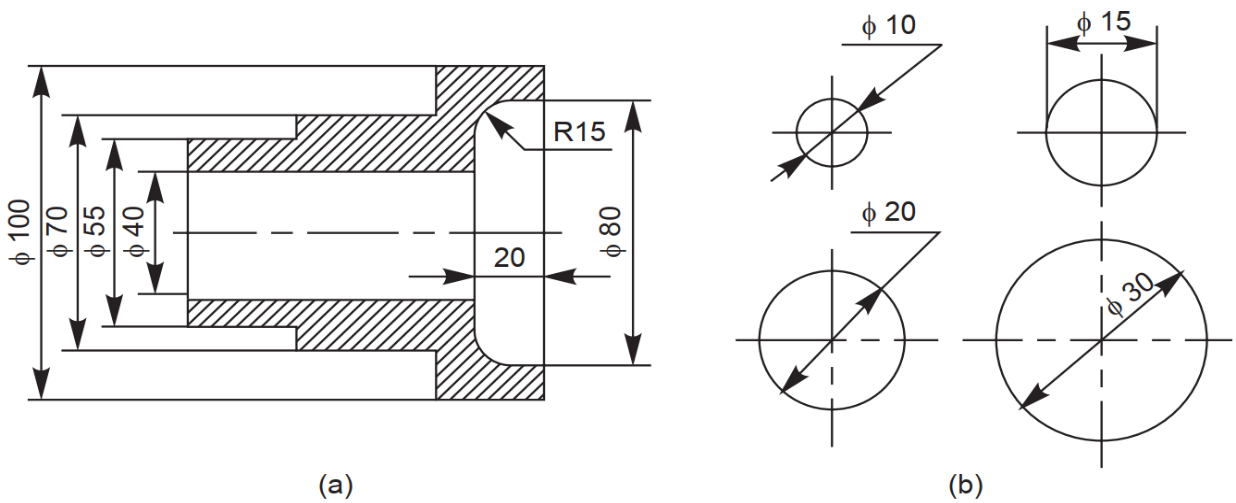


Fig. 2.47

Figure 2.34 Dimensioning of diameters

Chords, Arcs Angles and Radii

The dimensioning of chords, arcs and angles should be as shown in Fig. 2.48. Where the centre of an arc falls outside the limits of the space available, the dimension line of the radius should be broken or interrupted according to whether or not it is necessary to locate the centre (Fig. 2.35).

Where the size of the radius can be derived from other dimensions, it may be indicated by a radius arrow and the symbol R, without an indication of the value (Fig. 2.49).

Equi-distant Features

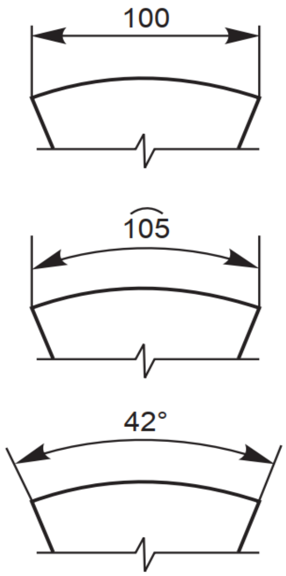
Linear spacing with equi-distant features may be dimensioned as shown in Fig. 2.50.

Chamfer and Counter sinks

Chamfers may be dimensioned as shown in Fig. 2.51 and counter sinks, as shown in Fig. 2.52.

Screw threads

Screw threads are always specified with proper designation. The nominal diameter is preceded by the letter M. The useful length of the threaded portion only should be dimensioned as shown in Fig. 2.53. While dimensioning the internal threads, the length of the drilled hole should also be dimensioned (Fig. 2.53).



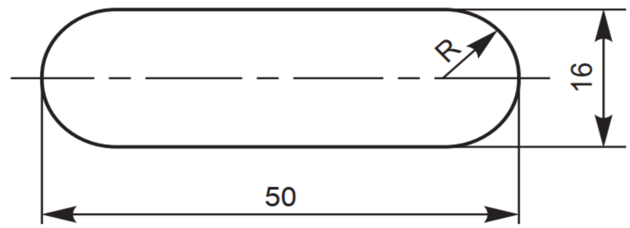


Fig. 2.49 Dimensioning of radius

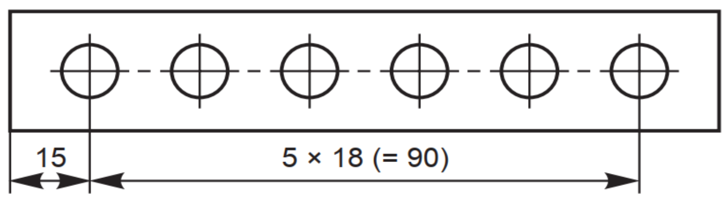


Figure 2.35 Dimensioning of chords, arcs and angle sand equi-distant features

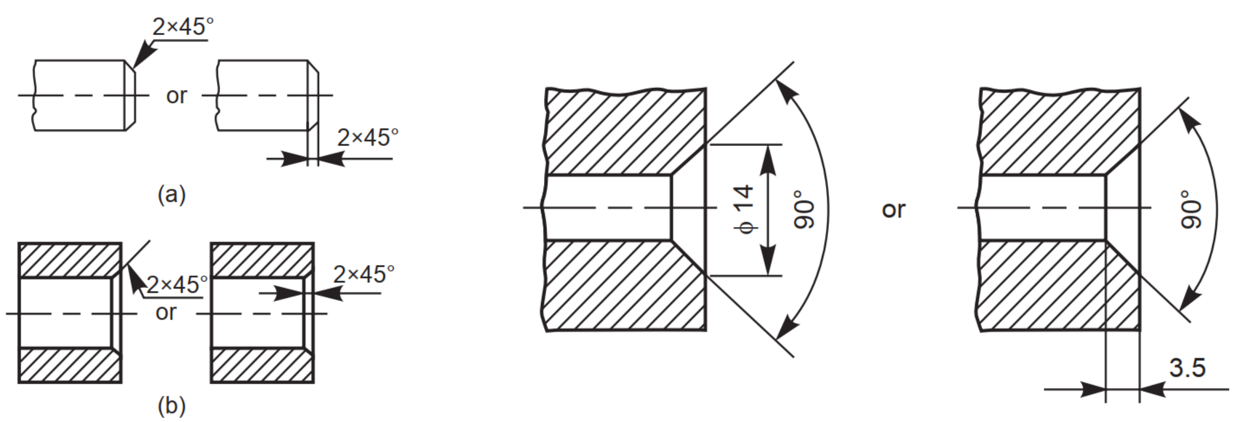


Figure 2.36 Dimensioning chamfers and Dimensioning counter sinks.

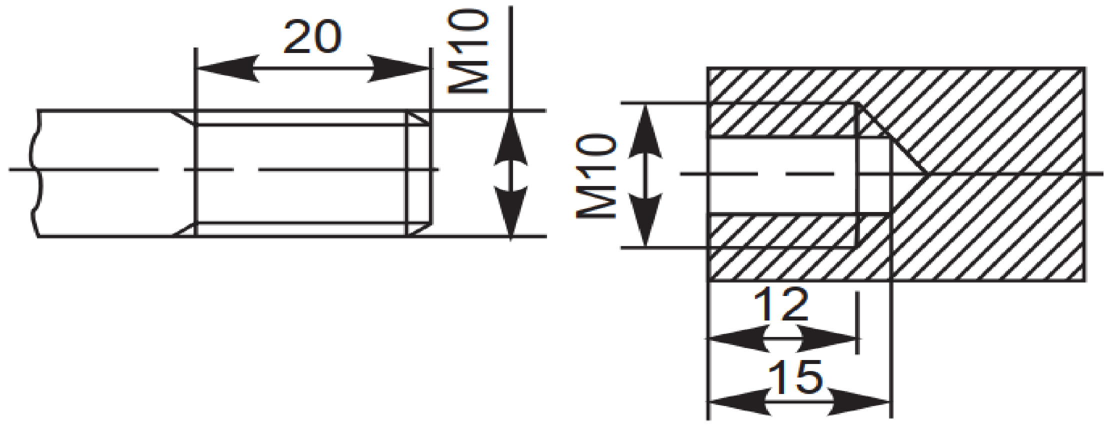


Figure 2.37 Dimensioning screw threads

Tapered features

Tapered features are dimensioned, either by specifying the diameters at either end and the length, or the length, one of the diameters and the taper or the taper angle (Fig. 2.54 *a*).A slope or flat taper is defined as the rise per unit length and is dimensioned by the ratio of the difference between the heights to its length (Fig. 2.54 *b*).

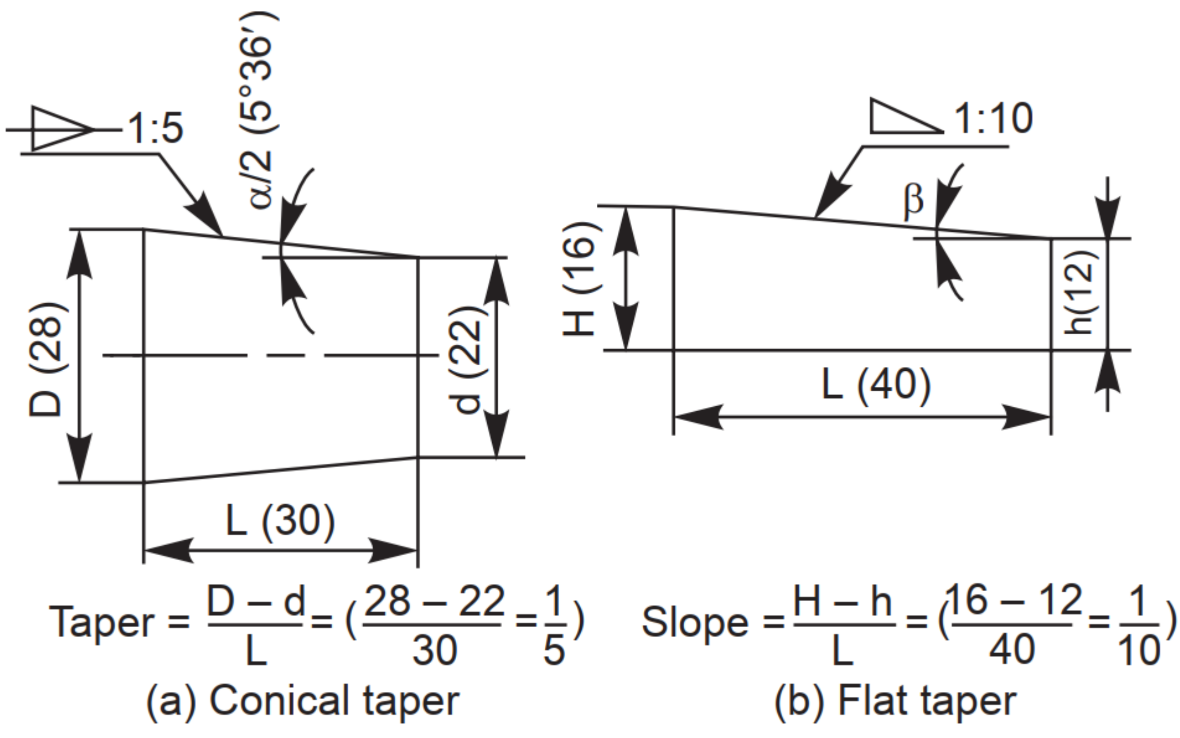


Figure 2.38 Dimensioning tapered features

2.3.2. Drawing notes

Notes should always be written horizontally in capital letters and begin above the leader line and may end below also. Further, notes should be brief and clear and the wording should be standard in form. The standard forms of notes and the method of indication, for typical cases is shown in Fig. 2.55. The meaning of the notes is given in Table 2.8.

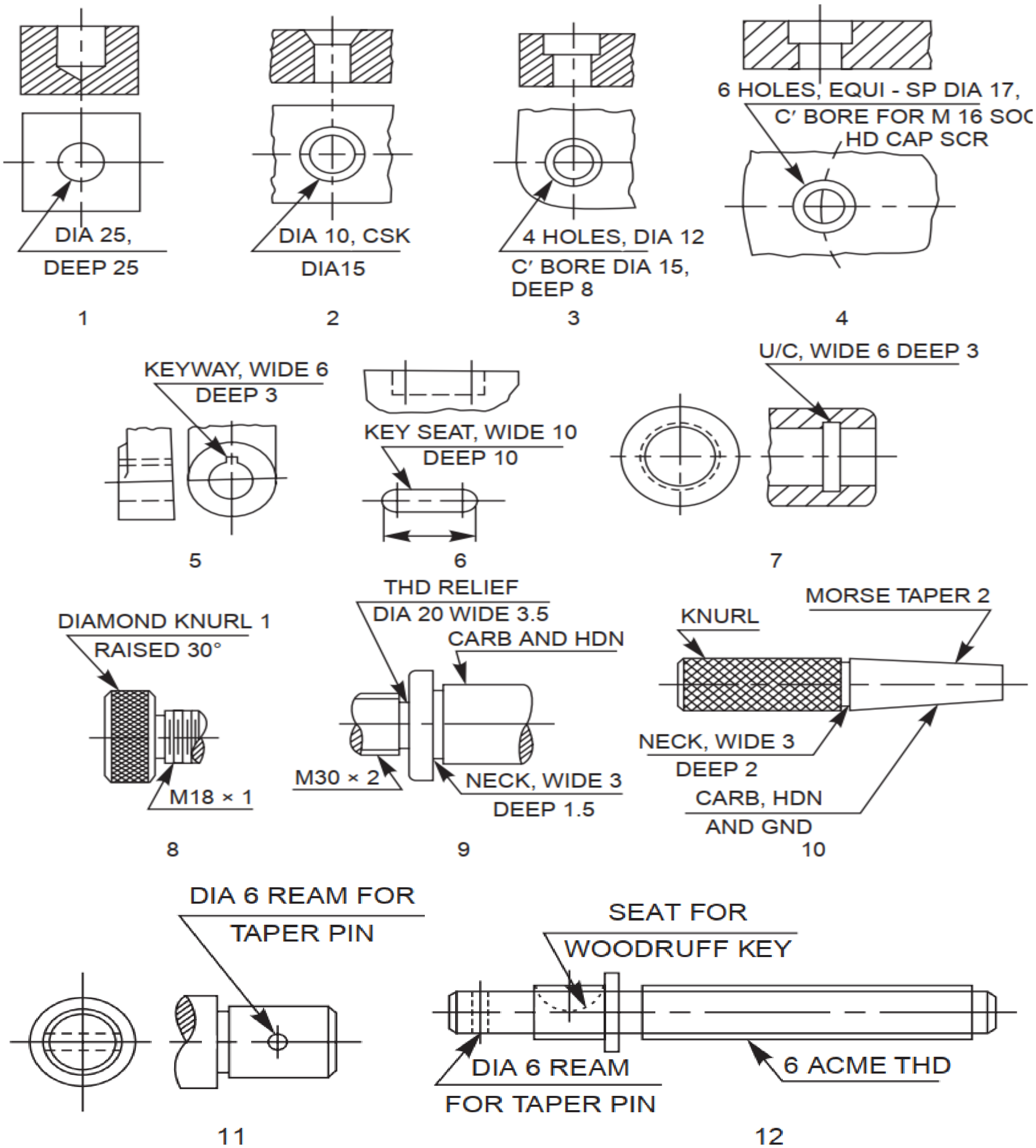


Figure 2.39Method of indicating notes

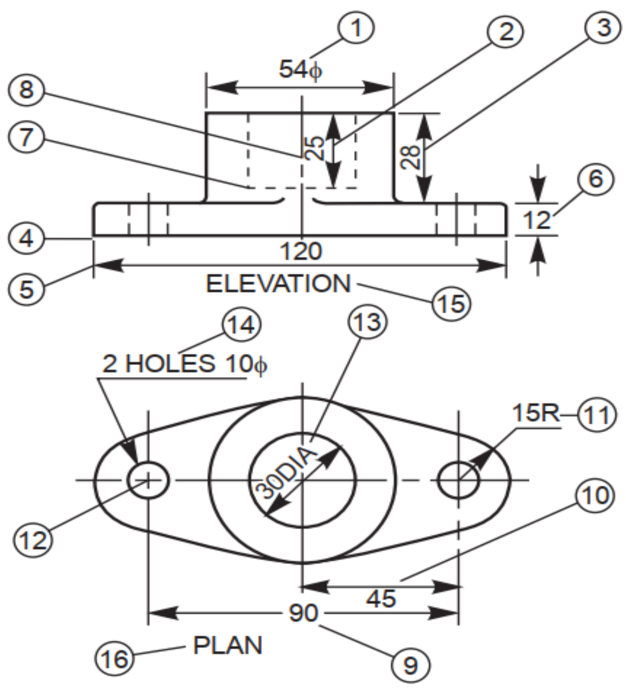
Table 2.10 Meaning of notes given in Fig. 2.39

S.No. Note	*Meaning/Instruction*
1. DIA 25 DEEP 25	Drill a hole of diameter 25 mm, to a depth of 25 mm.
2. DIA 10 CSK DIA 15	Drill a through hole of diameter 10 mm and countersink to get 15 mm on top.
3. 4 HOLES, DIA 12 C BORE DIA 15 DEEP 8	Drill through hole of φ 12 mm, counter bore to a depth of 8 mm, with a φ 15 mm, the number of such holes being four.
4. 6 HOLES, EQUI–SP DIA 17 C BORE FOR M 16 SOCKET HD CAP SCR	Drill a through hole of φ 17 and counter bore to insert a socket headed cap screw of M 16. Six holes are to be made equi-spaced on the circle.
5. KEYWAY, WIDE 6 DEEP 3	Cut a key way of 6 mm wide and 3 mm depth.
6. KEY SEAT, WIDE 10 DEEP 10	Cut a key seat of 10 mm wide and 10 mm deep to the length shown.
7. U/C, WIDE 6 DEEP 3	Machine an undercut of width 6 mm and depth 3 mm.
8. (a) DIAMOND KNURL 1 RAISED 30°	Make a diamond knurl with 1 mm pitch and end chamfer of 30°.
(b) M 18 × 1	Cut a metric thread of nominal diameter 18 mm and pitch 1 mm.
9. (a) THD RELIEF, DIA 20 WIDE 3.5	Cut a relief for thread with a diameter of 20.8 mm and width 3.5 mm.
(b) NECK, WIDE 3 DEEP 1.5	Turn an undercut of 3 mm width and 1.5 mm depth
(c) CARB AND HDN	Carburise and harden.
10. (a) CARB, HDN AND GND	Carburise, harden and grind.
(b) MORSE TAPER 2	Morse taper No. 1 to be obtained.
11. DIA 6 REAM FOR TAPER PIN	Drill and ream with taper reamer for a diameter of 6 mm to suit the pin specified.
12. 6 ACME THD	Cut an ACME thread of pitch 6 mm.

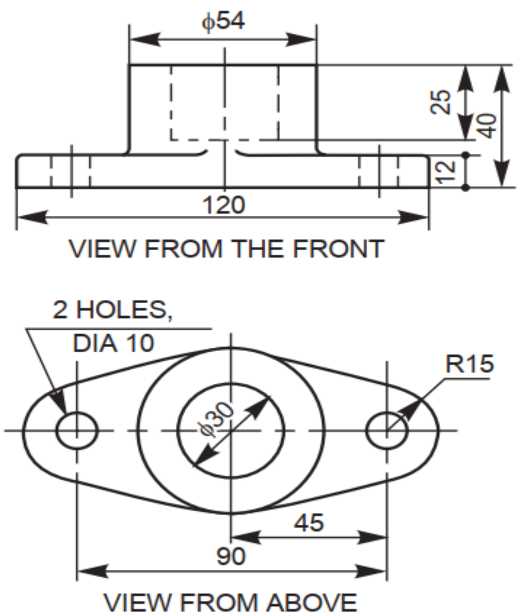
Operation Sheet 1

Violations of some of the principles of drawing are indicated in Fig. 2.56 *a*. The corrected version of the same as per the ISO Standard is given in Fig. 2.56 *b* and the reasons are given below:

1. Dimension should follow the shape symbol 2 and 3. As far as possible, features should not be used as extension lines for dimensioning.
4. Extension line should touch the feature.
5. Extension line should project beyond the dimension line.
6. Writing the dimension is not as per the aligned system.
7. Hidden lines should meet without a gap
8. Centre line representation is wrong. Dot should be replaced by a small dash.
9. Horizontal dimension line should not be broken to insert the value of the dimension
10. Dimension should be placed above the dimension line (Fig. 2.39).
11. Radius symbol should precede the dimension (Fig. 2.42)
12. Centre lines should cross at long dashes
13. Dimension should be written by symbol (not abbreviation) followed by its value
14. Note with dimensions should be written in capitals.
15. Elevation is not the correct usage.
16. Usage of the term ‘‘plan’’ is obsolete in graphic language.



a-Incorrect



b-Correct

Self – Check

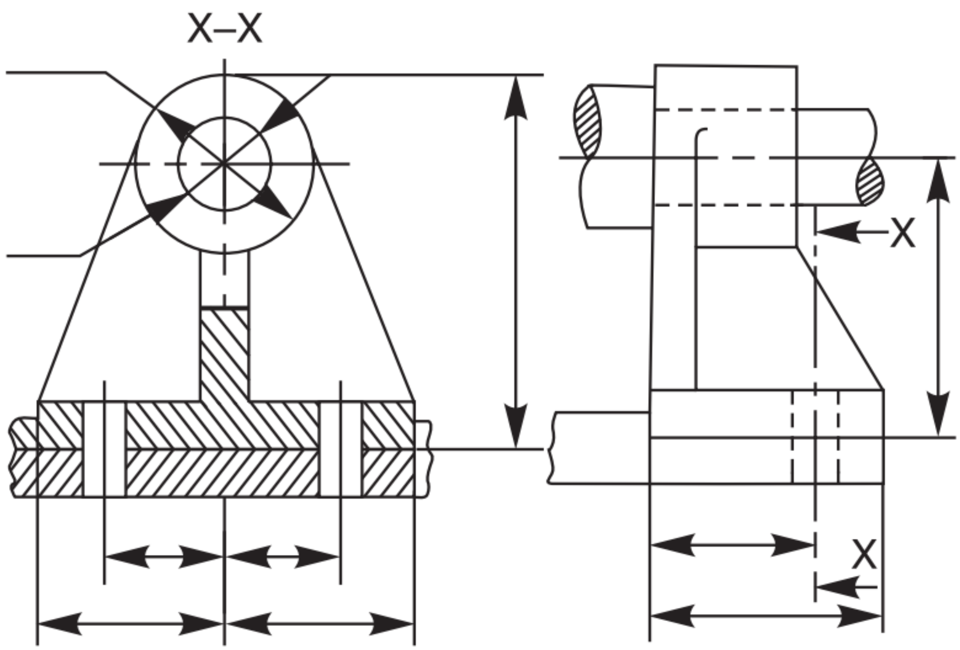
Give Clear and precise answer for the Question

1. What is the principle involved in fixing the sizes of the drawing sheets?
2. What is the information generally provided by the title block and what is its maximum length?
3. List out the standard thicknesses of lines that are used in machine drawing.
4. List out the elements of a dimension line.
5. List type of lines used in dimensioning.
6. List at list four principles of dimensioning.
7. List the methods of dimensioning.

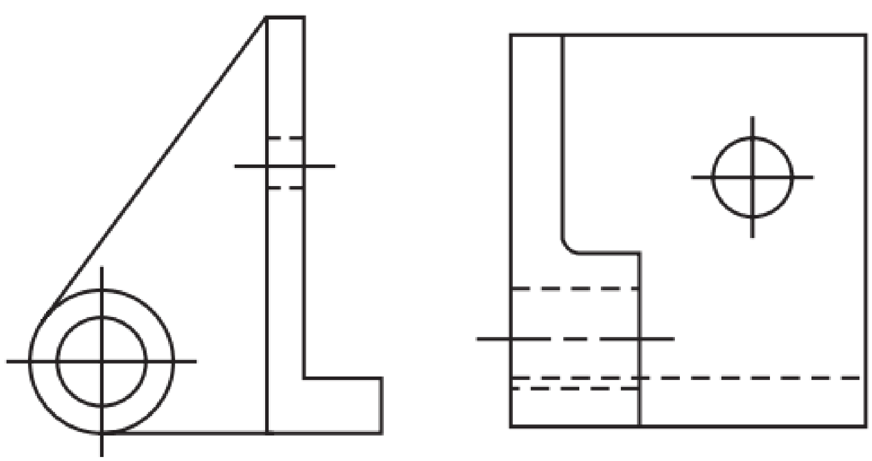
LAP TEST

Instructions: Given necessary drawing instruments, tools and materials you are required to perform the following tasks

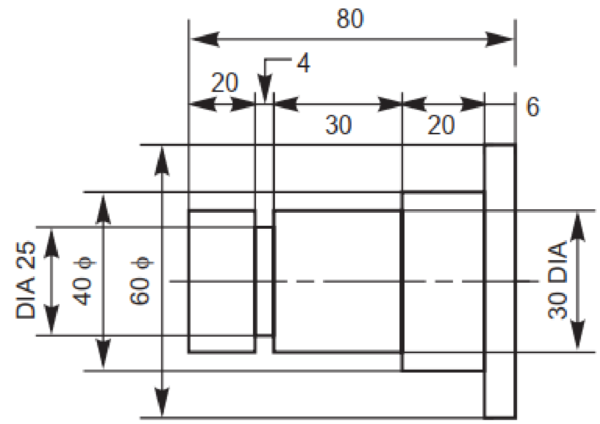
2. Identify (*i*) Functional, (*ii*) Non-functional and (*iii*) Auxiliary dimensions in Fig. below



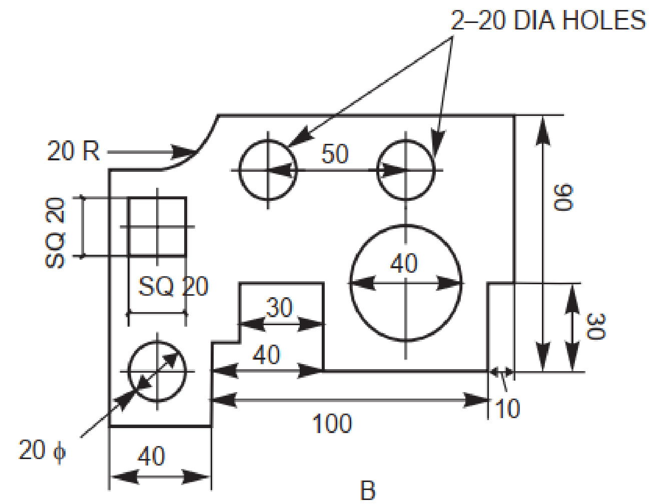
3. Identify the size and location dimensions in Figure shown below.



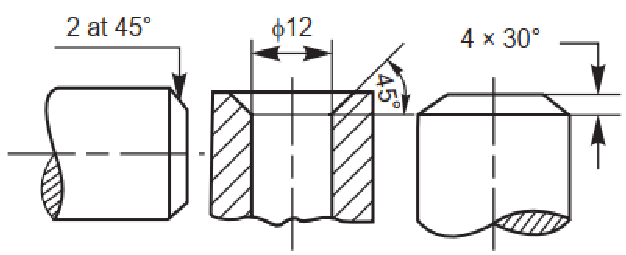
4. The drawings in Figure below are not dimensioned properly. Correct them according to standards.


80
20
4
30
20
6
DIA 25
40 ϕ
60 ϕ
30 DIA

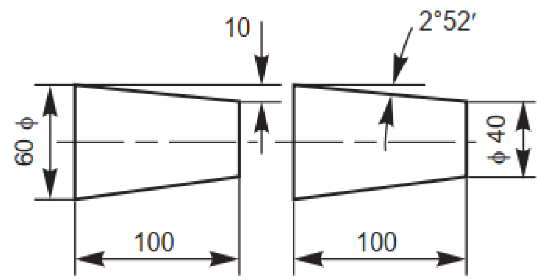
A


2–20 DIA HOLES
20 R
50
90
SQ 20
SQ 20
40
30
30
40
20 ϕ
10
100
40

B


2 at 45°
ϕ12
45°
4 × 30°

C


10
2°52′
60 ϕ
ϕ 40
100
100

D

Unit Three: Engineering Parts List

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics:

- Preparation of Engineering parts list
- Limit of accuracy of the measuring drawing instruments
- Method of records of parts lists (Cataloguing, Issuing security classifications

Filing and Preparing distribution lists

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Produce the component parts list with part name, description of part, material specification or part number, quantities and all other details
- handle of drafting and measuring instruments
- Record completed drawings and or parts lists

3. Engineering Parts List

3.1. Preparation of Engineering parts list

List of parts or the bill of materials: When drawings of a number of constituent parts of an Engineering object are drawn in a single drawing sheet, a list of these parts should be placed above or beside the title block in a tabular form. It should provide the following minimum particulars for each part: Part number, name or description, number. off i.e. quantity required, material and sometimes tock size of raw material, remarks.

Additional information such as job and order number, instructions regarding finish, heat-treatment, tolerances (general) and references pertaining to jigs, fixtures, tools, gauges etc. maybe, if necessary included in the title block or given separately in tabular form.

3.1.1. Elements of component List

Parts list, also known as a bill of materials (BOM) is a tabular list of the items used to make an assembly. Parts list is usually combined with the assembly drawing, but it is a separate and individual document and can be and provides a complete list of all parts needed to build the complete project.

The information associated with the parts list generally includes:

Item number: Item numbers are based on the assembly structure, that is, the order in which parts are displayed in assembly.

Part number:-Part number or drawing number which is a reference back to the detail drawing.

Description: Description is usually a part name or a complete description of purchase part or stock specification, including size and dimensions.

Quantity: The number of that particular part used on this assembly.

Material Specifications: means the physical properties and dimensions of all materials used or to be used or fabricated into pressure appurtenances on or in connection with any pipeline; Sample 1. Material Specifications

The four elements listed are the most common items and placed in the assembly drawing.

A. Parts list location

When placed on the assembly drawing, the parts list can be located in the upper-left or upper-right corner of the sheet, above or to the left of the title block, or in a convenient location. The location also depends on company standard.

Table 3.1 Parts list location

Parts list			
Item	**Part number**	**Description**	**Qty**
1	ABC123-01	Fuel-injection tube	2
2	ABC123-02	Delivery-valve holder	1
3	ABC123-03	Delivery valve	1
4	ABC123-04	Barrel	1
5	ABC123-05	Control rack	1
6	ABC123-06	Plunger	2
7	ABC123-07	Guide sleeve	4
8	ABC123-08	Retainer	2
9	ABC123-09	Nozzle needle	1

When the parts list is so extensive, the best location is the upper-right corner of the assembly sheet, in this case a new group of column is added to the right-side of the first one. If additional parts are added to the assembly, then space on the bottom of parts list is always available.

When the parts list is so extensive, the best location is the upper-right corner of the assembly sheet, in this case a new group of column is added to the right-side of the first one. If additional parts are added to the assembly, then space on the bottom of parts list is always available.

If the parts list is located in the bottom of page, then each row of data is provided from the bottom of the sheet upward, that is less natural.

Most companies prepares parts list in database software, so information can be retrieved and edited easily. When the set of working drawings is prepared on a CADD system, the parts list can be parametrically associated with the details and assembly, so changes to any are automatically updated on all.

A. Parts list in separate sheet

The parts list is not always placed on the assembly drawing. Some companies prefer to prepare parts list on separate sheets. This method allows the parts list to be computer generated separately from the drawings. Separate parts list are usually prepared on a computer so information can be edited conveniently.

Table 3.2 Parts list in separate sheet

GD&T Training Center Inc.				
Assembly name:			Assembly number:	
Number of parts:			Date:	
Parts list				
Item	**Part number**	**Description**	**Qty**	**Notes or Remarks**
1	ABC123-01	Fuel-injection tube	2	
2	ABC123-02	Delivery-valve holder	1	
3	ABC123-03	Delivery valve	1	
4	ABC123-04	Barrel	1	
5	ABC123-05	Control rack	1	
6	ABC123-06	Plunger	2	
7	ABC123-07	Guide sleeve	4	
8	ABC123-08	Retainer	2	
9	ABC123-09	Nozzle needle	1	
10	ABC123-10	Camshaft	1	
11	ABC123-11	Sliding bolt	2	
12	ABC123-12	Control level	2	

A. Removed parts

An engineering change request (ECR) can require to removal of a component from the parts list. When this happens, the balloon and the parts list numbers are not usually renumbered, because other documents may reference the balloons the parts list numbers. Instead, the word REMOVED is placed in the parts list next to the balloon number that has been deleted. Not all companies use this practice. Instead, they renumber the balloons and parts list and correlate all other related documents.

B. Purchase parts

Purchase parts, also refer to as standard parts, are parts that are manufactured and available for purchase from a supplier. Purchase parts are items such as fastener. pin, key, washer and other common products. It is generally more economical for a company to buy these items than to make them. Purchase parts do not require a detail drawing, because a written description completely describes the part. For this reason, the purchase parts found in an assembly must be described clearly and completely in the parts list. When the part description is too long and there is not enough space for putting that into the parts list, then using a Note Drawing can be a better way.

3.2. Limit of accuracy of the measuring drawing instruments

3.2.1. Introduction

The manufacture of interchangeable parts requires precision. Precision is the degree of accuracy to ensure the functioning of a part as intended. However, experience shows that it is impossible to make parts economically to the exact dimensions. This may be due to,

i. inaccuracies of machines and tools,

ii. inaccuracies in setting the work to the tool, and

iii. Error in measurement, etc.

The workman, therefore, has to be given some allowable margin so that he can produce a part, the dimensions of which will lie between two acceptable limits, a maximum and a minimum.

The system in which a variation is accepted is called the limit system and the allowable deviations are called tolerances. The relationships between the mating parts are called fits. The study of limits, tolerances and fits is a must for technologists involved in production.

The same must be reflected on production drawing, for guiding the craftsman on the shop floor.

3.2.2. Limit system

Tolerance: The permissible variation of a size is called tolerance. It is the difference between the maximum and minimum permissible limits of the given size. If the variation is provided on one side of the basic size, it is termed as unilateral tolerance. Similarly, if the variation is provided on both sides of the basic size, it is known as bilateral tolerance.

Limits: The two extreme permissible sizes between which the actual size is contained are called limits. The maximum size is called the upper limit and the minimum size is called the lower limit.

Deviation: It is the algebraic difference between a size (actual, maximum, etc.) and the corresponding

basic size: Actual Deviation It is the algebraic difference between the actual size and the corresponding basic size.

Upper Deviation: It is the algebraic difference between the maximum limit of the size and the corresponding basic size.

Lower Deviation: It is the algebraic difference between the minimum limit of the size and the corresponding basic size.

Allowance: It is the dimensional difference between the maximum material limits of the mating parts, intentionally provided to obtain the desired class of fit. If the allowance is positive, it will result in minimum clearance between the mating parts and if the allowance is negative, it will result in maximum interference.

Basic Size: It is determined solely from design calculations. If the strength and stiffness requirements need a 50mm diameter shaft, then 50mm is the basic shaft size. If it has to fit into a hole, then50 mm is the basic size of the hole.

Design Size: It is that size, from which the limits of size are derived by the application of tolerances. If there is no allowance, the design size is the same as the basic size. If an allowance of 0.05 mm for clearance is applied, say to a shaft of 50 mm diameter, then its design size is (50 – 0.05) = 49.95mm. A tolerance is then applied to this dimension.

Actual Size: It is the size obtained after manufacture.

Limits of accuracy or Upper and Lower bound are the accuracy of a measurement is how close that measurement is to the true value. This is restricted or limited by the accuracy of the measuring instrument. The ruler is marked in millimeters, so any length measured with it can only be given to the nearest millimeters,

In Limits of Accuracy, Nothing that is measured can be 100% accurate. Whether you are using a ruler, a protractor, a V-caliper or a set of machine scales, there will always be an error of ± half the unit of accuracy used.

Limit of Accuracy for each of these measuring scales, state the size of one unit on the scale and state the limit of accuracy.

(a). The size of one unit is 1 mm. The limits of accuracy are $\pm 0.5 \times 1$ mm = ± 0.5 mm.

(b). The size of one unit is 5 Kg. The limits of accuracy are $\pm 0.5 \times 5$ Kg = ± 2.5 Kg.

Table3.2. Principle of Limits of accuracy or Upper and Lower bound

No	Quantity given to the nearest...	Minimum value	Maximum Value
1	0.1 (to 1 decimal place)	Given value – 0.05	Given value
2	Whole Number	Given value – 0.5	Given value + 0.5
3	Ten	Given value – 5	Given value + 5
4	Hundred	Given value – 50	Given value + 50
5	Thousand	Given value – 500	Given value + 500

Example 1: Limits of accuracy or Upper bound and Lower bound

The following numbers have been rounded to two significant figures. Find the limit of accuracy or upper and lower bounds for each value.

(a). 23
(b). 0.56
(c). 830
(d). 200
(e). 6.17
(f). 0.40
(g). 78
(h). 0.91
(i). 0.011
(j). 6000
(k). 23.55
(l). 0.82

Solution: based on the principle in the above table

(a).	23	**Upper Bound** = 23.5 **Lower Bound** = 22.5
(b).	0.56	Upper Bound =0.565 Lower Bound = 0.555
(c).	830	Upper Bound = 835 Lower Bound = 825
(d).	200	Upper Bound = 205 Lower Bound = 195
(e).	6.17	Upper Bound = 6.175 Lower Bound = 6.165
(f).	0.40	Upper Bound =0.405 Lower Bound = 0.395
(g).	78	Upper Bound = 78.5 Lower Bound = 77.5
(h).	0.91	Upper Bound =0.915 Lower Bound = 0.905
(i).	0.011	Upper Bound = 0.0115Lower Bound =0.0105
(j).	6000	Upper Bound = 6050 Lower Bound = 5950
(k).	23.55	Upper Bound = 23.555 Lower Bound =23.545
(l).	0.82	Upper Bound = 0.825 Lower Bound = 0.815

Example 2: Limits of accuracy or Upper bound and Lower bound

The radius of a circle is 6.5cm to one decimal place.

(a). Calculate the minimum perimeter of the circle.

(b). Calculate the maximum area of the circle.

Solution

Upper Bound 6.55cm Lower Bound 6.45cm

(a). Minimum diameter = 2 x 6.45cm = 12.9cmCircumference = π x 12.9 = 40.53cmMinimum perimeter = 40.53cm

(b). Maximum radius = 6.55cmMaximum Area = π x 6.55^2 = cm^2

3.3. Methods of Part lists records

Cataloguing or **Library Cataloging** is the process of creating and maintaining bibliographic and authority records in the library catalog, the database of books, serials, sound recordings, moving images, cartographic materials, computer files, e-resources etc. that are owned by a library. The catalog may be in tangible form, such as a card catalog or in electronic form, such as online public access catalog (OPAC). Relative to the movement of materials within technical services, cataloging usually follows the receipt of ordered books in acquisitions. The process of cataloging involves three major activities, namely, Descriptive Cataloging, Subject Cataloging, and Authority Control.

In libraries, metadata creation is often called cataloging[1].Cataloging is a subset of the larger field called *information organization*.

It can be defined as, "The process of creating metadata for resources by describing a resource, choosing name and title access points, conducting subject analysis, assigning subject headings and classification numbers, and maintaining the system through which the cataloging data is made available.

In other words, it is the process of creating metadata about library resources, which is included in the catalog.

A catalog is an organized compilation of bibliographic metadata that represents the holdings of a particular institution or a library network and/or resources accessible in a particular location.

Filing (File Management)

File management organizes parts, assemblies and drawings. File management is utilized to organize Drawing Templates and Sheet Formats.. A top level assembly necessitates hundreds or even thousands of drawings to document its parts and sub-assemblies. Drawings utilize various Drawing Templates and Sheet Formats. Parts, assemblies and drawings are distributed between team members to conserve development time. Design changes occur frequently in the development process. How do you manage and control changes? Answer: Through file management. File management is a very important tool in the development process. Utilize file folders to organize projects, vendor components, templates and libraries. The documents required to complete the projects in Drawing and Detailing with CAD

Self – Check 3

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. _______ a is a tabular list of the items used to make an assembly.
   A. Parts list
   B. bill of materials (BOM)
   C. Part Drawing
   D. A and B
2. Which is an Elements of BOM that describe the physical properties and dimensions of all materials used or to be used or fabricated
   A. Specifications
   B. Item number:
   C. Part number
   D. Quantity:
3. __________is the degree of accuracy to ensure the functioning of a part as intended.
   A. Limit
   B. Precision
   C. Tolerance
   D. Allowance
4. The two extreme permissible sizes between which the actual size is contained are
   A. Limit
   B. Precision
   C. Tolerance
   D. Allowance
5. It is size from which the limits of size are derived by the application of tolerances.
   A. Actual Size
   B. Design Size
   C. Basic Size
   D. Deviation:

Why do you require file management?

LAP TEST 3

The following numbers have been rounded to two significant figures. Find the limit of accuracy or upper and lower bounds for each value.

(a).	56	(f).	0.91
(b).	0.056	(g).	0.011
(c).	960	(h).	6000
(d).	6.17	(i).	23.55
(e).	0.40	(j).	0.82

Unit Four: Key features of CAD software

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics:

- CAD software for detail drafting
- Types of CAD software (AutoCAD, CATIA and Solid works)
- 2-D & 3-D Geometrical modelling,
- Orthographic/isometric/perspectives/schematics Drawing
- Compatibility of CAD software

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Describe Types of CAD software
- identify key features and suitability CAD software for detail drafting
- Identify Differences in CAD process to generate 2D drawings and 3D models
- Know and confirm CAD software compatibility

4. Key features of CAD software system

4.1. CAD software for detail drafting

4.1.1. Introduction

The heart of any CAD model is the component database. This includes the graphics entities like points, lines, arcs, circles etc. and the co-ordinate points, which define the location of these entities. This geometric data is used in all downstream applications of CAD, which include finite element modeling and analysis, process planning, estimation, CNC programming, robot programming, programming of co-ordinate measuring machines, ERP system programming and simulation.

In order to achieve at least a reasonably high level of integration between CAD, analysis and manufacturing operations, the component database must contain:

i. Shapes of the components (based on solid models)
ii. Bill of materials (BOM), of the assembly in which the components are used.
iii. Material of the components
iv. Manufacturing, test and assembly procedures to be carried out to produce a

Component so that it is capable of functioning as per the requirements of design. In designing a data structure for CAD database the following factors are to be considered:

i. The data must be neutral
ii. The data structure must be user-friendly
iii. The data must be portable.

In order to achieve the above requirements, some type of standardization has to be followed by the CAD software designers. The basic elements associated with a CAD system are:

- Operator (user)
- Graphics support system
- Other user interface support system
- Application functions
- Database

A diagrammatic presentation of these elements is given in Figure 17.1

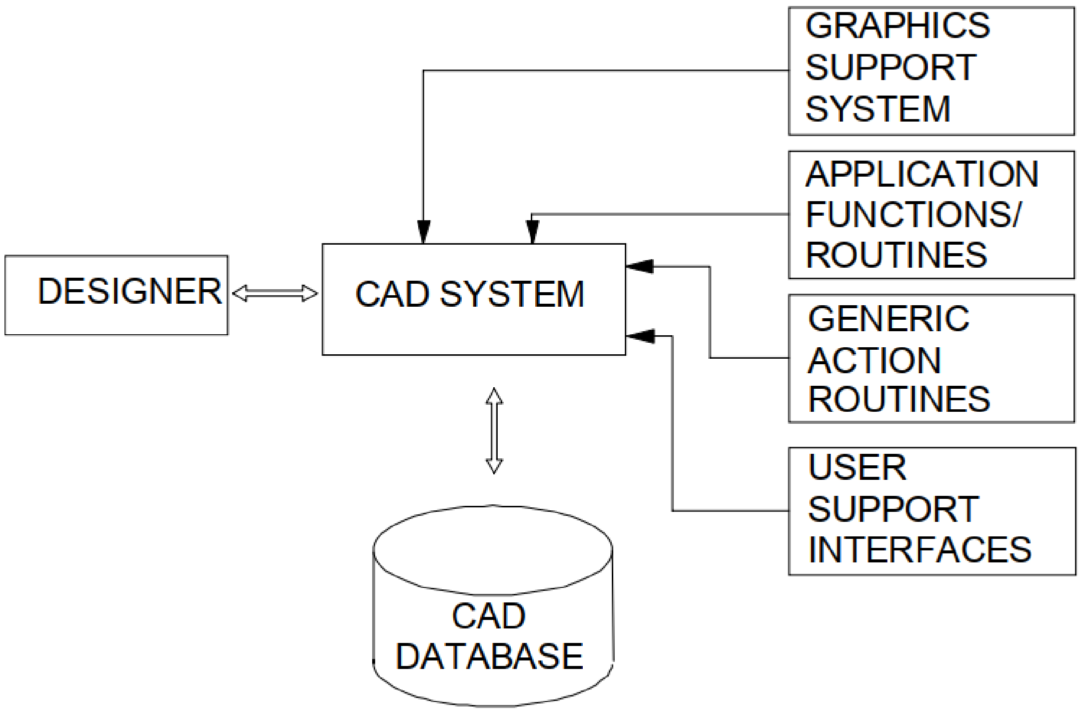


Figure 4.1CAD System

The reasons for evolving a graphic standard thus include:

- Need for exchanging graphic data between different computer systems.
- Need for a clear distinction between modeling and reviewing aspects.

4.1.2. Key Features of CAD Software

Designs break the barriers of the third dimension through Computer-Aided Design (CAD) software. These are computer programs that let you design and document design processes virtually. It replaces the traditional pen and paper drafting by letting you visualize, build, and render your designs in your computers, both in 2D and 3D.

Now is the perfect time to begin exploring examples of popular CAD software solutions out there. To help you with your search, here are the key features of CAD software

A. A modern and ergonomic user interface

The first essential feature of the CAD software: its user interface. The way in which the tool is controlled and operated has a direct impact on the final productivity. The easier and faster the software is to use, the more productive the user becomes.

- The commands must be designed according to modern conventions: copy and paste, drag & drop, contextual menus, etc. to allow the user to immediately adapt to the new tool.
- Access to the commands should be easy, fast and intuitive; in order to optimize the user's working time.

Choose CAD software with a modern and ergonomic user interface: you reduce the learning curve and you can work faster.

B. Suitability, a guarantee of reliability

Suitability automates the updating of one element with respect to another. The designer can therefore easily modify his construction history: the associated operations are automatically updated. Illustrations:

- Moving a screw automatically moves the holes.
- Changing a chassis length automatically changes the positioning of the associated components.

With Suitability, a key feature of good CAD software, you can drastically reduce the risk of errors or omissions when updating an element. And therefore, you can save time in the design process. So be sure to choose a tool that allows associative or parametric design.

C. Choice in design methods

A good CAD software offers three design methods.

- **"Bottom-up" design**: this is the simplest method which consists of first designing the parts individually, and then assembling them. The assembly is then a succession of predefined part positioning.
- **"Top-down" design**: conversely, the designer starts from the constraint, and then deduces the necessary assemblies and parts.
- **"In-place" design**: a variation of top-down design where parts are designed in their assembly context. The designer then has the advantage of simplicity for the creation of connecting parts, in particular.

The relevance of which method to be used varies from one profession to another. A manufacturer of highly standardized products will favor the Bottom-up method. In any case, the three methods are complementary. To take advantage of your CAD software, make sure that the tool offers these three powerful design methods: maximum flexibility in choice and the precious opportunity of complementary methods.

D. Smart components

User productivity increases as good standard components are created. For this purpose, your CAD system must provide the appropriate features. Unique smart components, process management, installation wizards, etc. allow you to store user know-how. ***Ultimately,*** the design is accelerated: the **productivity gain is obvious**.

E. Design documentation tools

Once completed, the design must be documented. This includes generating exploded views, assembly instructions, bills of materials, etc. These features are essential to get the most out of your CAD software. The tool must therefore make it possible to easily generate all useful production documents from an assembly.

F. A detailed and standardized drawing

Despite the increasing use of 3D, **detailed and standardized drawings of the designs are still often necessary**, "the drawing is authentic!" Your CAD software must therefore integrate a powerful automated drawing module, to perform this task quickly when necessary.

G. Change management tools

When choosing your CAD software, **the extent and performance of the modeling features are essential criteria**, but are not sufficient! The designer also has to deal with management issues. Changes to the specifications during the design process are common practice, so the tool must offer intuitive and powerful features for this purpose. To manage all types of modifications, without wasting time or reliability, choosing a PDM is then crucial. Therefore, you should choose a fully integrated PDM, which will allow you to track changes during the design process. From simple part number change to more complex modifications (interchanging parts, for example), pay attention to the extent of the PDM capabilities of your CAD software.

Top Solid is designed to address all the current challenges faced by professionals when designing parts and machines. **Top Solid integrates the essential features of good CAD software**. In addition, you benefit from close support for the deployment of your new solution. Beforehand, evaluating your needs allows us to define the features of your tool according to your working methods. Afterward and in the long term, our technical support, everywhere in France and around the world, is available and responsive for beneficial proximity.

4.2. Types of CAD software

There are a variety of engineering and design works that use CAD (computer-aided design software).Most common types of CAD software use include AutoCAD, Solid Works, CATIA,

4.2.1. AutoCAD

One of the oldest and most used CAD software for 2D / 3D drafting & design. It has the ability to create blueprints, equipment layouts, section planes, model documentation, and more. Learning AutoCAD and additional 3D CAD software is a recipe for success.

Some features including:

- 2D drawing, drafting, and annotation
- Extended workflows
- Xrefs and block palettes
- Dynamic blocks
- 3D Mesh, solid and surface modeling

4.2.2. CATIA

CAD software used by OEM's (original equipment manufacturers) for surface modeling, design, and visualization Want an engineering or design job in the Automotive or Aerospace industries Put this one on your list.

Some features including:

- 3D Modeling and Digital prototyping
- Highly detailed surface modeling and shape manipulation
- Virtually create and analyze products during development

4.2.3. Solid Works

As popular mechanical design software with a large community of users, employers are frequently seeking job seekers with Solid Works experience. This 3D modeling CAD software is used heavily in the mechanical engineering and design industries. It is the industry standard for product development.

Some features including:

- Product design simulation and 3D solid modeling
- Product data management and Cost estimation, manufacturability checks
- Created with ease of use in mind
- Used by mechanical and electrical engineering departments

4.3. 2-D & 3-D Geometrical model

4.3.1. Types of Geometrical model

Computer representation of the geometry of a component using software is called a geometric model. Geometric modeling provides a means of representing part geometry in graphical representation. This constitutes the most important and complex part, in many software packages. There are a variety of modeling methods available in the industry for the variety of functions. They are:

A. 2-D Wire frame/Geometrical model

This method consists of a range of 2D shapes which can be used to develop basically the outline of a part, which in most of the cases is composed of lines and circles (Fig. 21.12), this is the easiest and most popular way to model simple parts. They are easy to understand.

B. 3-D Wire frame/Geometrical model

This is similar to its 2D counter-part, except that it is drawn in 3 dimensions. This is used in low cost designing systems. The complete object is represented by a number of lines with their end point co-ordinates (x, y, z) and their connectivity relationships. It is difficult to understand the outside of the solid, represented by the wire frame model. Thus, the wire frame model is inadequate for representing the more complex solids (Fig. 21.13).

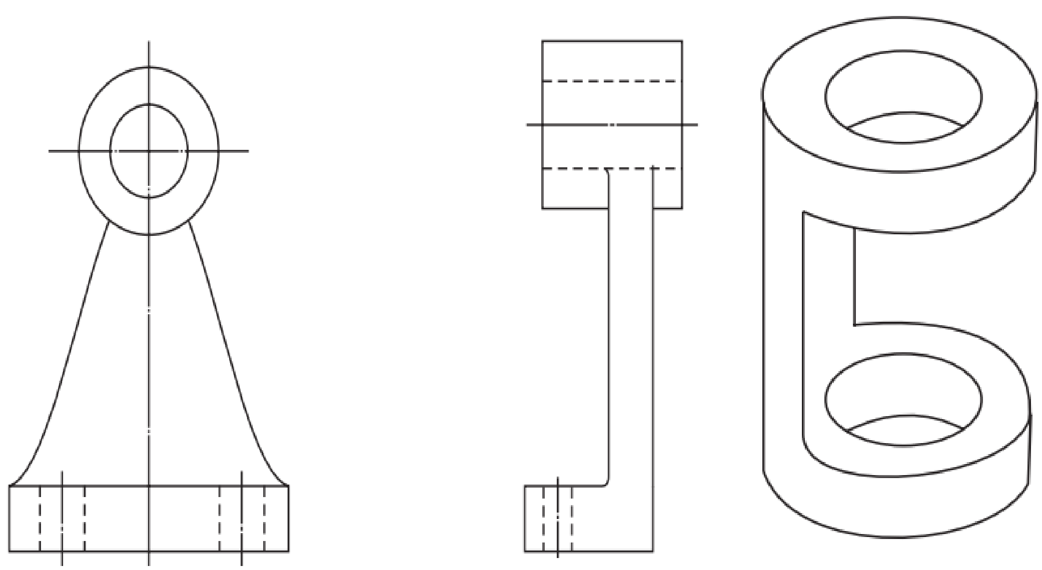


Figure 4.2 2D geometrical and 3D geometrical model

Table 4.1Differences in CAD process to generate 2-D drawings and 3-D models,

2 - D Models	3-D Wire Frame Models
Ends (vertices) of lines are represented by their X and Y coordinates	Ends of lines are represented by their X, Y and Z coordinates.
Curved edges are represented by circles, ellipses, splices etc. Additional views and sectional views are necessary to represent a complex object with clarity.	Curved surfaces are represented by suitably spaced generators. Hidden line or hidden surface elimination is a must to interpret complex components correctly.
3-D image reconstruction is tedious.	2-D views as well as various pictorial views can be generated easily.
Uses only one global coordinate system	May require the use of several user coordinate systems to create features on different faces of the component.

Functions of Geometric Modelling

In Drafting

- Automatic planar cross sectioning, hidden line and surface removal.
- Automatic production of shaded images.
- Automatic dimensioning.
- Automatic creation of exploded views for technical illustrations.

Manufacturing

- Parts classification and Process planning.
- Numerical control data generation and verification.
- Robot program generation

Production Engineering

- Bill of materials and – Material requirement.
- Manufacturing resource requirement and Scheduling.
- Inspection and Quality Control:
- Program generation for inspection machines.
- Comparison of produced part with design.

4.4. Orthographic/isometric/perspectives/schematics drawing

4.4.1. Isometric Drawing

In isometric projection all vertical lines on an object remain vertical while horizontal lines are drawn at 30° to the horizontal. Isometric drawings are usually produced with drawing equipment to ensure accuracy. Isometric projection distorts shapes slightly in order to keep all upright lines vertical (and because perspective is ignored). Their advantage is that they show the object's dimensions accurately and in correct proportion to each other, making it easy to draw the projection correctly to scale from a plan view.

Drawings of objects are seldom drawn in true isometric projections, as the use of an isometric scaleis inconvenient. Instead, a convenient method in which the foreshortening of lengths is ignored and actual or true lengths are used to obtain the projections, called isometric drawing or isometric view is normally used. This is advantageous because the measurement may be made directly from a drawing. The isometric drawing of figure is slightly larger (approximately 22%) than the isometric projection. As the proportions are the same, the increased size does not affect the pictorial value of their presentation and at the same time, it may be done quickly. Figure 9.5 shows the difference between the isometric drawing and isometric projection.

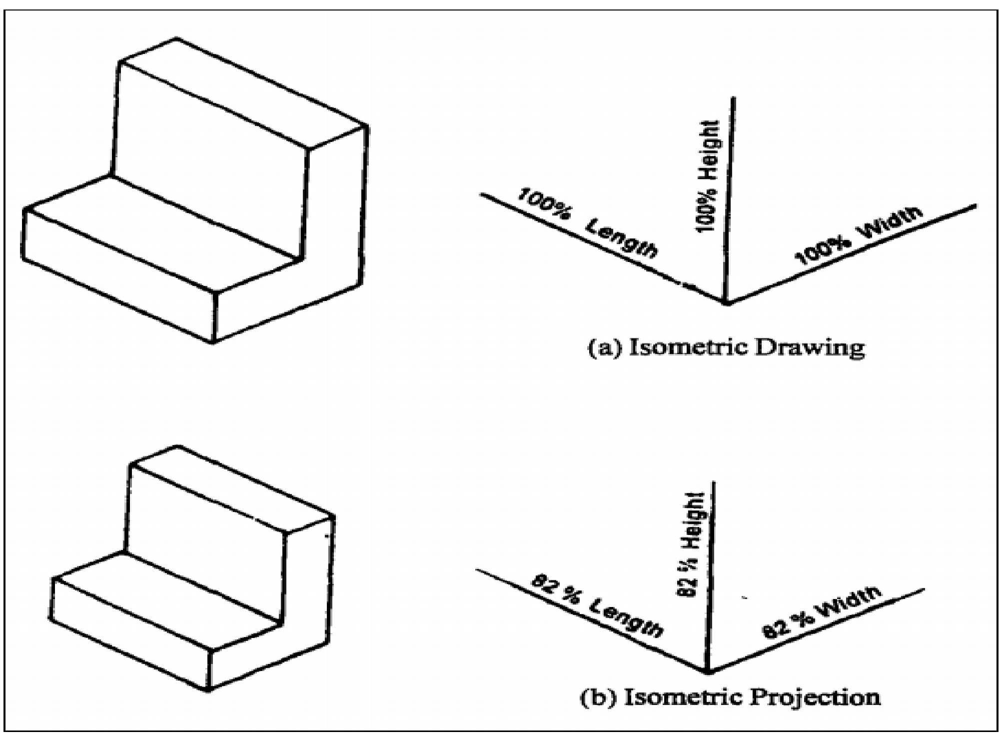


Figure 4.3 Graphical representation of Isometric Drawing and projection

4.4.2. Orthographic Projection

Orthographic projection shows complex objects by doing a 2D drawing of each side to show the main features. Orthographic drawings usually consist of a front view, a side view and a plan, but more views may be shown for complex objects with lots of detail. A drawing board and parallel motion or T-square is used to project one view from another. Orthographic drawing may be done using first angle projection or third angle projection. This graphic drawing shows the differences between the two.

An Orthographic projection is a 2-D representation of a 3-D object. The 2-D drawing represents different sides of an object.

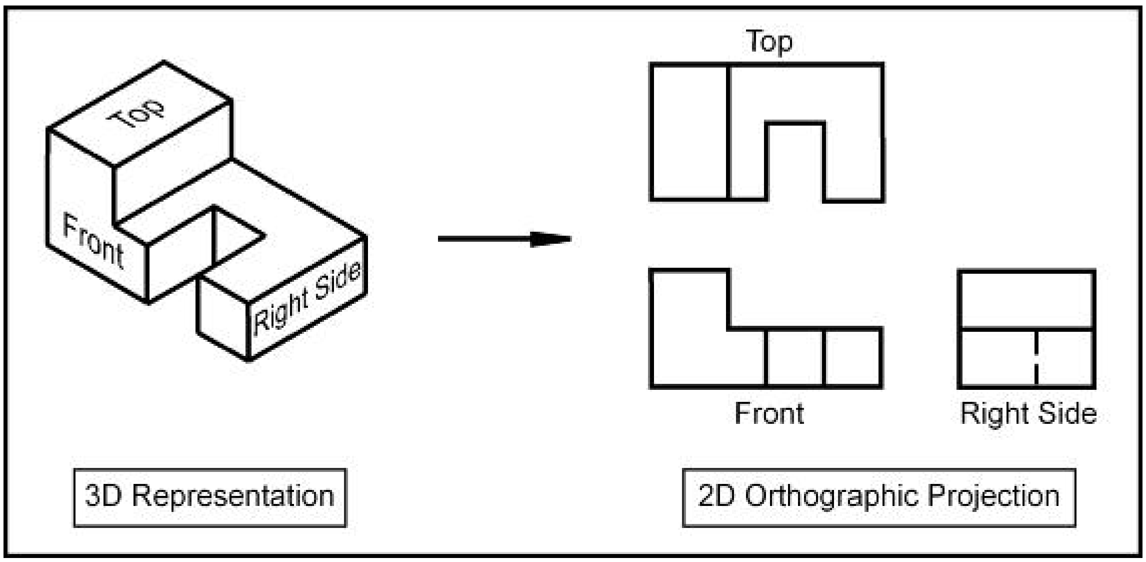


Figure 4.4 3D representation and 2D orthographic projection

4.4.3. Perspective Drawing

In perspective drawing the forward face or edge of the object is drawn first, with the other lines receding away from the viewer and gradually approaching each other - just as they appear to do when you look at a real object. If the receding lines are extended they will meet at points that are called vanishing points. Perspective drawing can be done using drawing equipment or freehand. Perspective drawing can use one, two or three vanishing points. One-point perspective is often used for room interiors. Two-point perspective has many applications for developing ideas in 3D. Three-point perspective is often used for drawings of tall buildings. The advantage of perspective drawing is that it makes objects appear more realistic, as objects appear to get smaller as their distance from the observer increases.

Perspective projection is a method of graphic representation of an object on a single plane called picture plane as seen by an observer stationed at a particular position relative to the object. As the object is placed behind the picture plane and the observer is stationed in front of the picture plane, visual rays from the eye of the observer to the object are cut by the picture plane. The visual rays locate the position of the object on the picture plane. This type of projection is called perspective projection. This is also known as sceno graphic projection or convergent projection

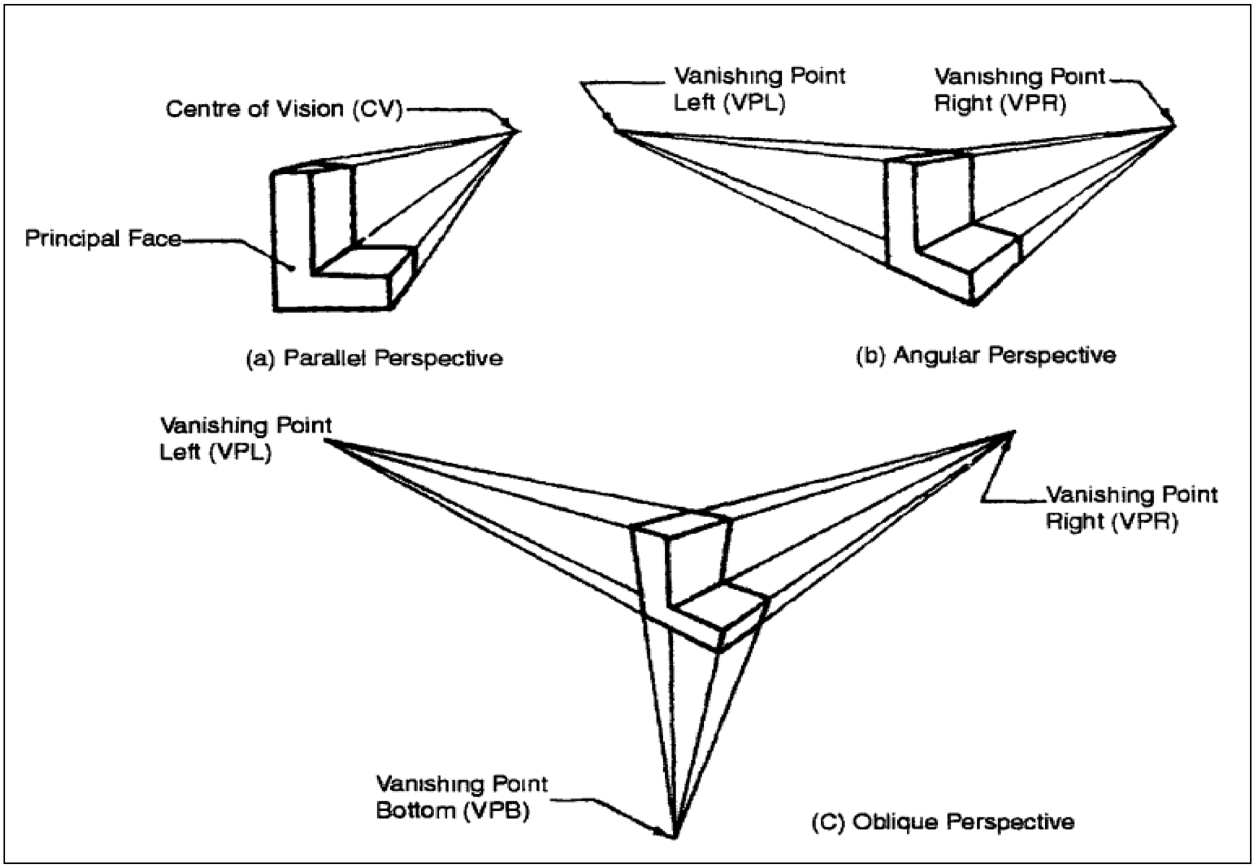


Figure 4.5 Graphical representation Perspective projection

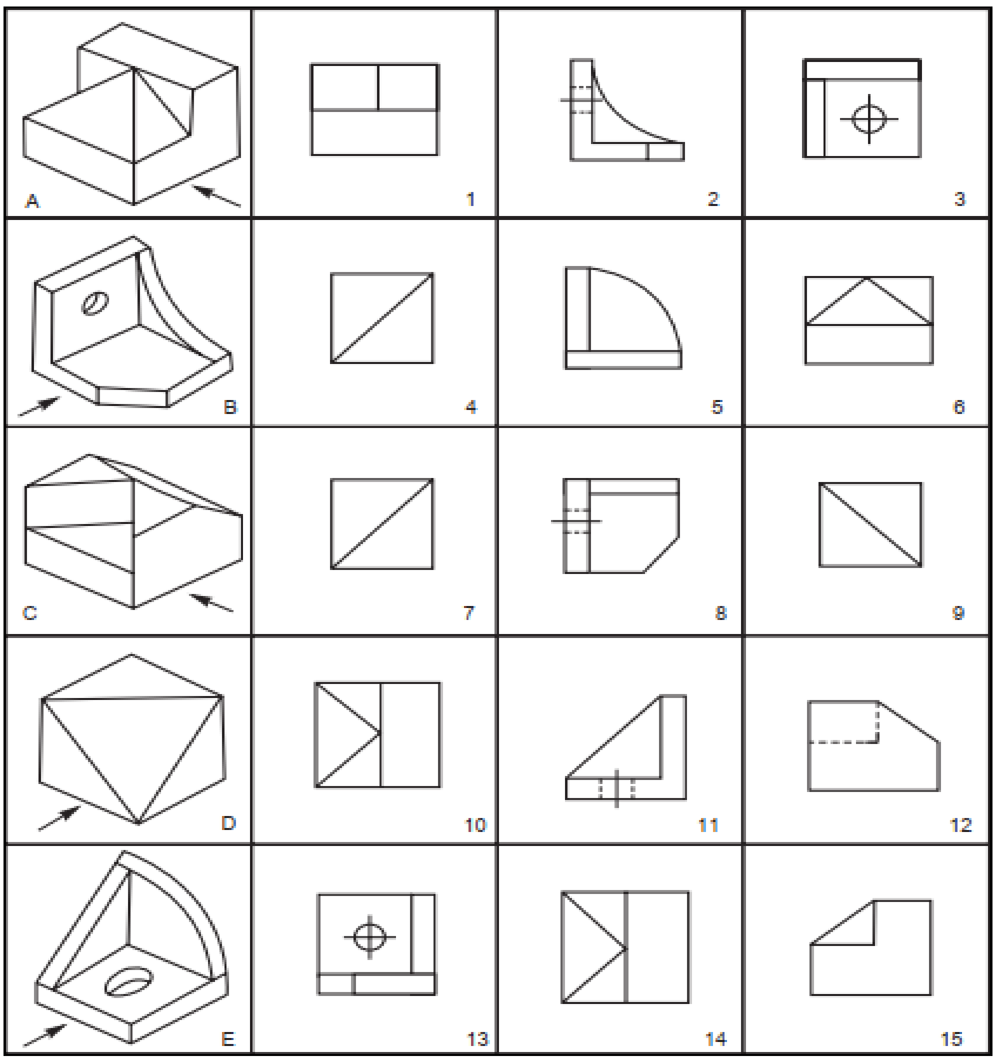
4.5. Compatibility of CAD software

Capabilities and versatilities of the drafting system vary depending on the system on which they are implemented. AutoCAD, Versa CAD, CAD Key, Design CAD, ZWCAD, etc. are few popular commercially available drafting systems in use. These systems provide a variety of features required for producing engineering drawings. As an example of a popular, low-cost CAD software, we describe here some of the fundamental capabilities of AutoCAD. All these or similar facilities are very much desired in any CAD software for it to be useful.

Self – Check

Give Clear and precise answer for the Question

1. List down the Key Features of CAD Software
2. What are the Most Common types of CAD software used for engineering drawing
3. List down the Key Features of AUTOCAD Software
4. List and describe the types of Geometrical model with drawing sketch
5. Differences in CAD process to generate 2-D drawings and 3-D models,
6. What are the Functions of Geometric Modeling in the field of Engineering
7. Identification of views by matching A – E in the left with 1 – 15 in the right.



Unit Five: Basic drawing elements and 2D drawings

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics:

- CAD functions for basic drawing elements
- Editing and transfer tools and methods
- Dimensions, text and symbols to drawing elements
- Perform Drawings using commands.
- Perform Importing and exporting Files

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Apply CAD functions
- produce basic drawing elements
- modify drawing elements usingEditing and transfer tools and commands
- applyDimensions, text and symbols to drawing elements
- import and exportFiles into/out of workingspace
- generate/makeDifferent views and perspectives drawing in CAD software
- createDrawings and Detailed views using various scales and the full capability of theCAD

5. Basic drawing elements and 2D drawings

5.1. CAD functions for basic drawing elements

5.1.1. Functionsand Advantages of CAD

The major functions to be performed by a computer aided drafting system are:

(a). Basic set-up of a drawing
(b). Drawing the objects
(c). Changing the object properties
(d). Translating the objects
(e). Scaling the objects
(f). Clipping the objects to fit the image to the screen
(g). Creating symbol libraries for frequently used objects
(h). Text insertion
(i). Dimensioning
(j). Creates various layers (Transparent sheets)
(k). Allows zoom-in and zoom-out of any componentsof drawing or completedrawing
(l). Creates different numbers of print/plot layouts.

Some of the features of CAD systems are:

1) **Modelling and Drafting:** The majority of systems provide 2D and 3D modellingcapabilities. Some low cost CAD systems are dedicated to 2D drafting only.
2) **Ease of use:** The users find CAD systems very easy to learn and use.
3) **Flexibility:** Popular CAD systems provide greater flexibility when configuringthe available hardware. Hundreds of computers, display devices, expansion boards,input and output devices are compatible and configurable with popular softwares.
4) **Modularity:** Standard input and output devices are attached to standard connectorsthereby making the system modular in nature.
5) **low maintenance cost:** Little maintenance is needed to keep the system functional.

Some of the Advantages of CAD systems are:

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

5.1.2. Auto Cad Main Window

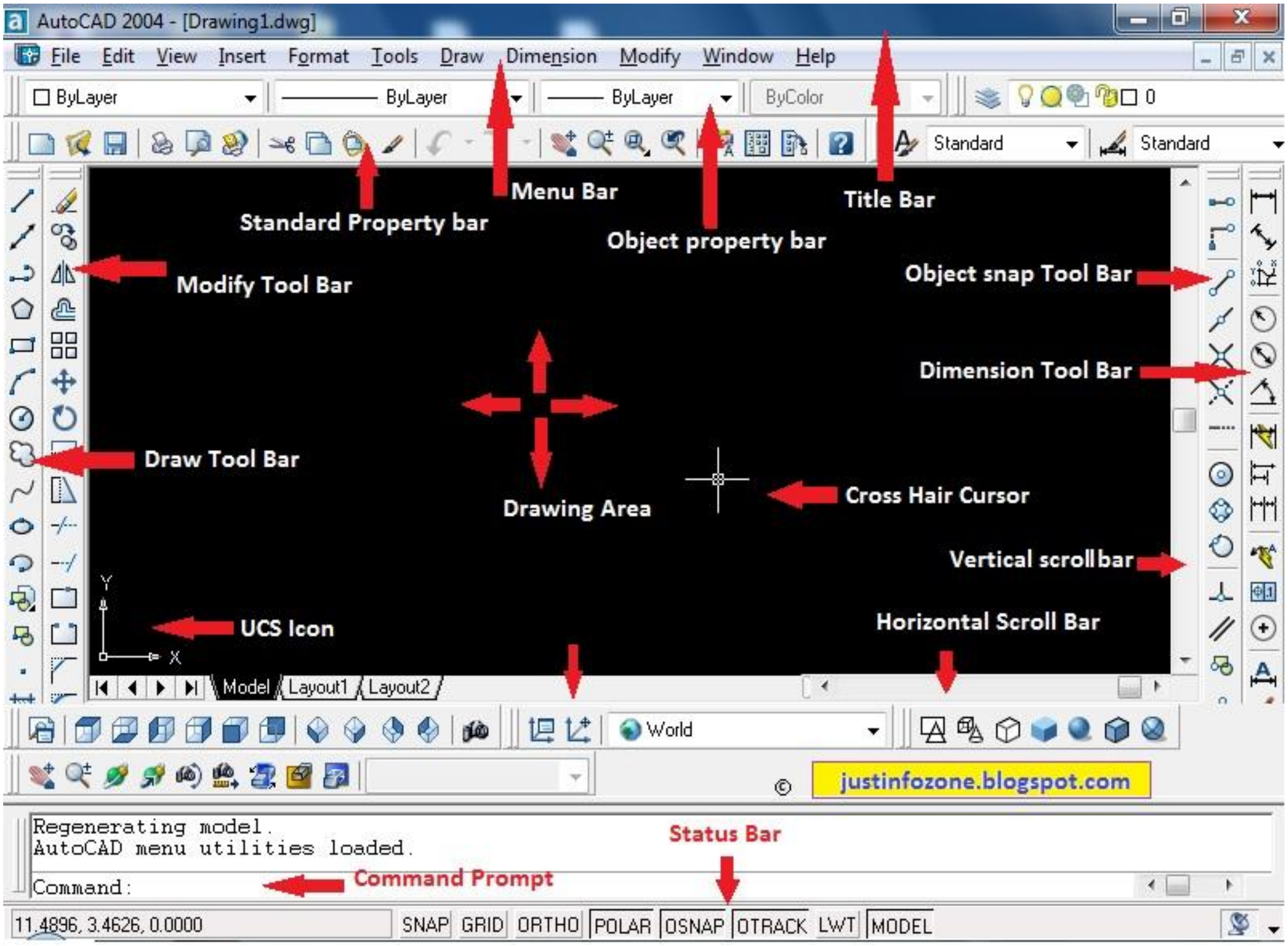


Figure 5.1Auto CAD Main Window

5.1.3. Plotting of Drawing

AutoCAD drawing may be plotted either by a pen plotter or a printer plotter. Pen plotters are very accurate and multiple colours may be obtained. Printer plotters have limited resolution and smaller paper sizes and produce monochrome output. However, printer plotters are usually faster than pen plotters. For pen plotter, PLOT command is used, whereas for a printer plotter, PRPLOT command is used.

While beginning a plot from the main menu, tell AutoCAD, which portion of the drawing to be plotted. Specify the part of the drawing to be plotted by entering: Display, Extents, Limits, View or window: the response specifies a rectangular area of the drawing. By Specifying:

D (Display)	—	this option plots the view that was displayed in the current view port just prior to the last SAVE or END command for that drawing.
E (Extents)	—	this option is similar to ZOOM extents. The extents are updated automatically as one draws new entities.
L (Limits)	—	plots the entire drawing area as defined by the drawing limits.
V (View)	—	plots a view that was previously saved, using the drawing editor's view command.
W (Window)	—	plots any portion of the drawing. Specify the lower left corner and upper right corner of the area to be plotted.

5.1.4. Starting in CAD

Once AutoCAD 2004/05 software is located on to the computer and the operating system is available, one can start using the facility. Soon the computer is turned on, the operating system is automatically loaded. Various application icons appear on the windows screen. AutoCAD can be started by double-clicking on the AutoCAD icon available on the desktop of the computer. The various components of the initial AutoCAD screen are as shown in Fig. 21.1 and Fig. 21.2 consisting of:

A. **Drawing Area** The drawing area covers a major portion of the screen. Various objects can be drawn in this region by the use of AutoCAD commands. The position of the pointing device is represented on the screen by the cursor. On the lower left corner, a coordinate system icon is present. On the top right corner, standard windows buttons are also available.
B. **Command Window** At the bottom of the drawings area, command window is present and commands can be entered by keyboard.
C. **Status Bar** At the bottom of the screen, status bar is displayed, which will make it easy to change the status of some AutoCAD functions by proper selection.
D. **Standard Tool Bar** Standard tool bar displays coordinates and they will change only when a point is specified. The absolute coordinates of the cursor will be specified with respect to the origin.
E. **Snap** Snap mode allows the cursor to be moved in specified/fixed increments.

F. **Grid** By choosing this button, grid lines are displayed on the screen and can be used asreference lines to draw AutoCAD objects.

G. **Ortho** By selecting the orthomode, lines can be drawn only at right angles on the screen.

H. **Polar** The movement of the cursor is restricted along a path based on the angle set asthe polar angle. One can use either polar mode or orthomode only at a particular time.

One can also use function keys for quick access to certain commands. Only important functions defined by AutoCAD 2004 are given below:

Function Key	Function
F1	Online help
F2	Toggles between command window on and off
F5	Switches among Isoplanes Top, right and left
F6	Toggles between coordinates on and off
F7	Toggles between grid on and off
F8	Toggles between orthomode on and off
F9	Toggles between snap mode on and off
F10	Toggles between Polar tracking on and off

After starting AutoCAD and when the cursor is in the drawing area, to perform an operation, commands must be invoked. The following methods are provided to invoke the commands.

B. Keyboard Using keyboard, command name can be typed at the command prompt and by pressing ENTER or SPACE BAR, the command can be invoked.

C. Menu The menu bar is at the top of the screen which displays the menu bar titles. As the cursor is moved over this, various titles are highlighted and by means of pick button, a desired item can be chosen. Once it is selected, the corresponding menu is displayed directly under the title. A command can be invoked by picking from this (Fig. 21.3*a*).

D. Draw Toolbar This is an easy and convenient way to invoke a command. This is displayed on the left extreme of the initial AutoCAD screen (Fig. 21.3*b*) and very easy to choose by picking.

E. Tool Palettes These are shown on the right side of the monitor screen (Fig. 21.2). An easy and convenient way of placing blocks/patterns of hatching in the present drawing.

By default, AutoCAD displays the tool palettes on the right of the drawing area. Various hatching patterns also can be selected from this.

The default AutoCAD 2010 drawing screen can be divided into four areas such as drawing area, command area, menu bar area and tool bar area. The various component of Graphical User Interface (GUI) of AutoCAD 2010 is shown in fig. 26-3.

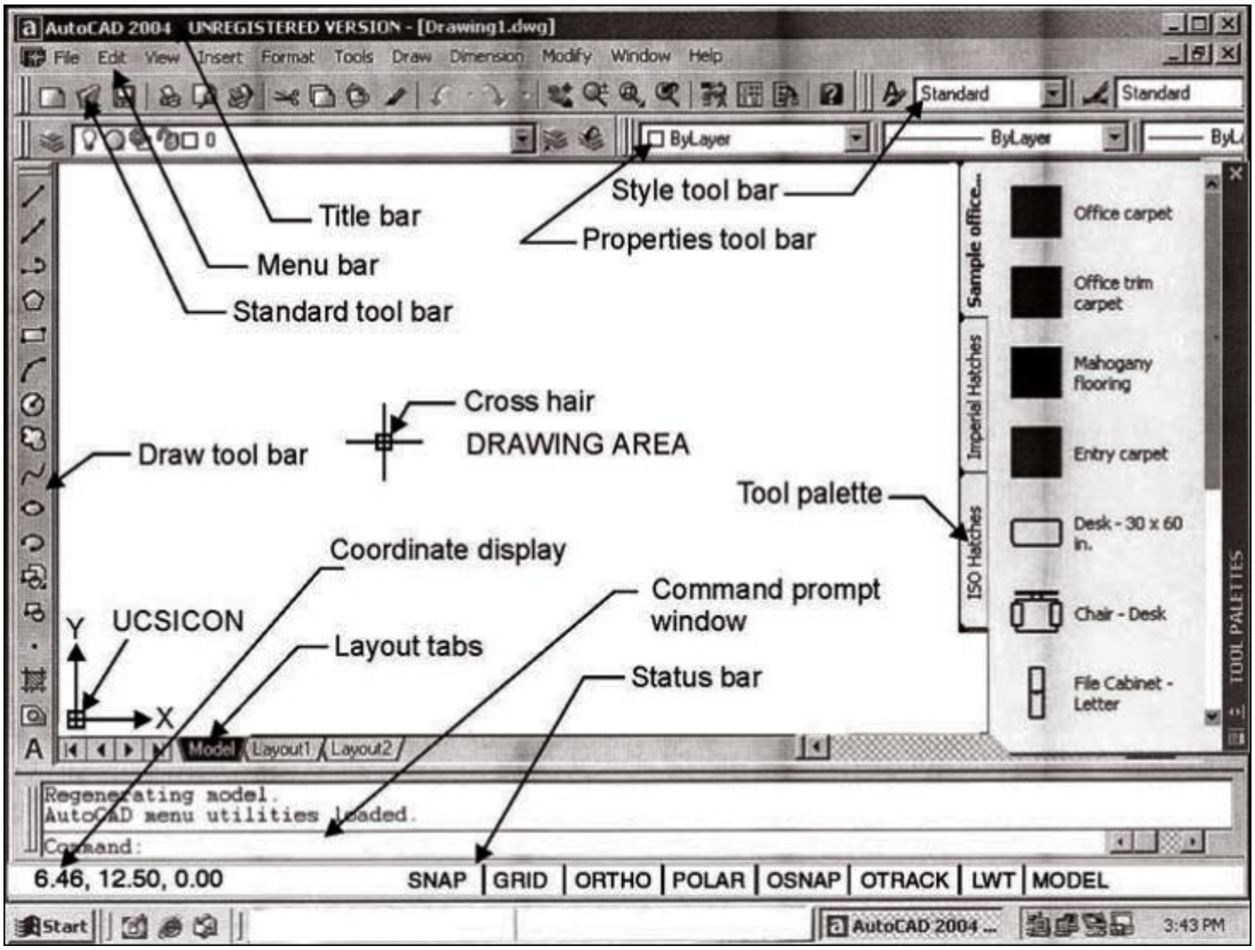


Figure 5.2Components of AutoCAD Classic screen of AutoCAD 2010

Setting drawing limits

It is normal when using AutoCAD to draw objects full size, so it's usually necessary to reset the drawing limits to (about) the size ofthe object being drawn. Move the cursor to the bottom left ofthe screen, you can notice Command box. We can fix required paper size like AO, AI, A2, A3, A4 etc. from the Command box.

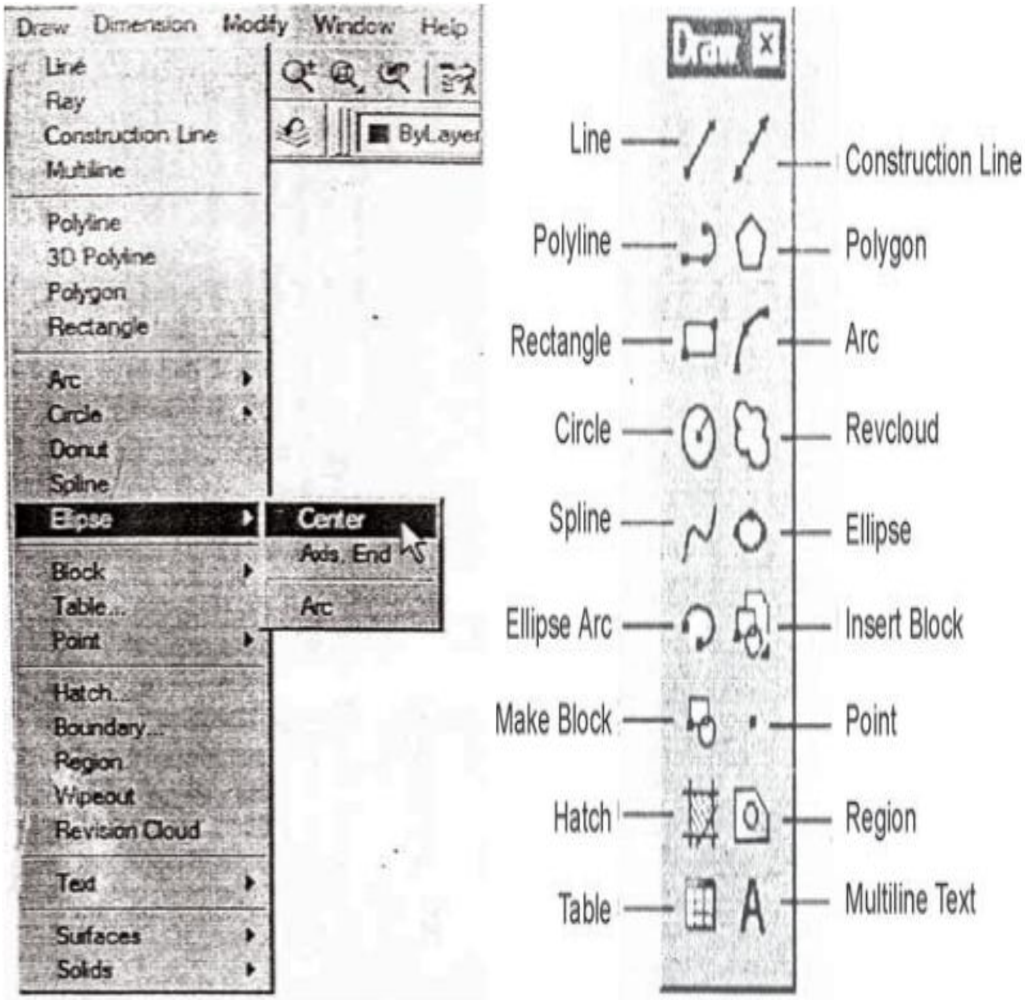


Figure 5.3 Invoking the ELLIPSE and The Draw toolbar

The layering concept is similar to the transparent overlays used in many draughting applications. This allows the user to view and plot-related aspects of a drawing separately or in any combination. In drawing insertion, a drawing can be stored in a drawing file and this may be inserted in subsequent drawings for any number of copies. To refer to geometric features of existing objects when entering points, the object snap may be used. The visual image of the drawing on the screen may be magnified or shrunk by zooming. Whereas, panning allows viewing different portions of the drawing, without changing the magnification. In plan view, the construction plane of the current user co-ordinate system is parallel to the screen. The drawing may also be viewed from any point in space (even from inside an object). The graphics area of the screen can be divided into several view ports, each displaying a different view of the drawing.

Physical resolution refers to the amount of detail that can be represented. This resolution can be changed at any time. The editing facilities of AutoCAD make it easy to correct or revise a drawing. Multiple copies of an object, arranged in rectangular or circular patterns are easy to create.

While planning a drawing in Auto CAD, one has to organize some of the information such as choosing the units, co-ordinates, etc

Coordinate system: The system used by all the CAD packages is generally the rectangular cartesian co-ordinatesystem designated as x, y and z axes. The positive direction of these axes follows the right handrule. Any point in space can therefore be designated by the co-ordinate values of these 3 axes.viz., x, y and z.

The co-ordinates can be input into the system by:

(i). The direct input of co-ordinate values in the respective order of *x*, *y* and *z*. If *z* coordinate is not mentioned, then the values are assumed to be at a single given level.

(ii). Specifying the co-ordinates in an incremental format from the current cursor position in the drawing area. The distance is specified by using @ parameter before the actual values. The incremental values apply to all the ordinates.

(iii). Point co-ordinates may also be specified using the polar co-ordinate format. It can also be an extension of the incremental format.

(iv). Using the mouse button, the cursor may be taken to the required position and the button is clicked.

It is generally necessary to specify the limits of the drawing with the help of the command LIMITS, where the user will be asked to specify the lower left corner and upper right corner of the drawing sheet size. This establishes the size of the drawing

5.1.5. Basic drawing elements (Geometric commands)

By way of choosing the basic commands in AutoCAD, one can make simple drawings. The various entities that can be used for making an AutoCAD drawing in 2D are: point, line, ellipse, polygon, rectangle, arc, circle, etc.

Generally AutoCAD provides a default option as <> in each of the command response. The value shown in the angle brackets is the most recently set value. To have the same value, one has to simply press the <Enter> key. The various options available for each command are shown in the command window. But the user need to respond by choosing one letter in most cases, which makes the AutoCAD choose the right option.

The basic drawing entities are lines, polylines of any width,circles, arcs, ellipses and solids. There are many ways of defining a drawing entity, andthe software always prompts the user for all options. Each drawing entity has anassociated line-type, colour, layer and thickness. The thicknessis a property associated with 3D entities.

A. Drawing Entity – POINT

The point command locates a point in the drawing.

Command: POINT (one has to give the location)

POINT: 25, 45 location of the point. Thus, a point is placed at the given location (25, 45).

After setting the limits of the drawing, the following drawing aids/tools may be used tolocate specific points on the screen (electronic drawing sheet).

ORTHO Command—this is orthogonal drawing mode. This command constrains the lines drawn in horizontal and vertical direction only.

Command: ORTHO

ON/OFF <current>:

SNAP Command—this command is used to set increments for cursor movement.

If thescreen is on SNAP mode, the cursor jumps from point to point only. The cursor movement canbe effectively controlled using the SNAP command. This is useful for inputting the data throughdigitizer/mouse.

Command: SNAP

Snap spacing or ON/OFF/Aspect/Locate/Style <current>: 0.1 (default)

GRID Command-working on a plain drawing area is difficult since there is no means for the user to understand or correlate the relative positions or straightness of the various objects made in the drawing. The command enables to draw dotted lines on the screen at pre-defined spacing. These lines will act as graph for reference lines in the drawing. The grid spacing can be changed at will. The grid dots do not become part of the drawing.

Command: GRID

Grid spacing or ON/OFF/Snap/Aspect <0>: 0.5 (default)

Function keys may create drawing aids/tools also. The function keys F7, F8 and F9 act as toggle keys for turning ON or OFF of GRID, ORTHO and SNAP tools respectively.

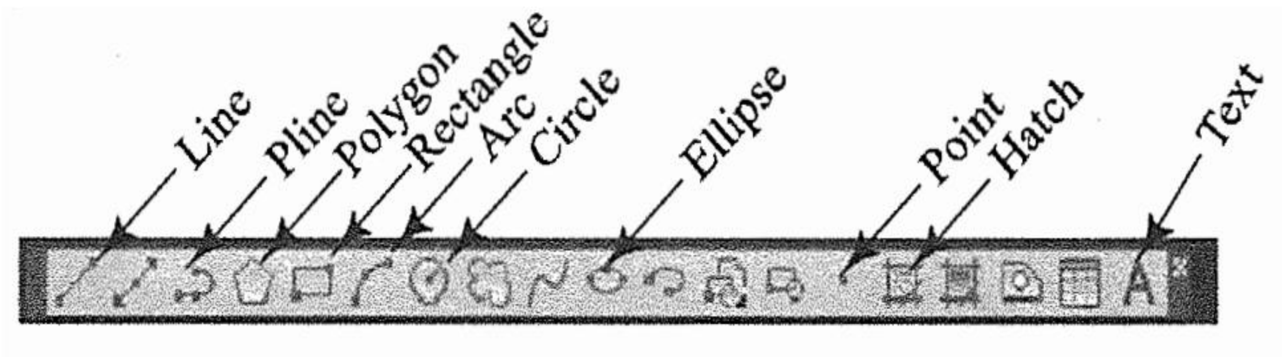
HELP Command—AutoCAD provides with complete help at any point of working in the program. HELP can be obtained for any of the individual commands. Most of the information required by the user is generally provided by the help which is always instantaneous.

SAVE Command—AutoCAD provides the following commands to save the work/drawing on the hard disk/floppy diskette:

SAVE SAVEAS QSAVE

Command: SAVE

Save drawing as <current name>: KLNI



B. Drawing Entity – LINE

A line is specified by giving itstwo endpoints. The LINE command can be used to draw a single line or a series of lines with the end-point of one being the startpoint of the next. When a seriesof such lines is created, eachline is treated as a separate entity. To create a closed polygon,the user has to type in C (close option) for the *To point:* prompt.

This causes the last and the first points to be joined by a line andthus cerating a closed boundary. Lines can be constrained to horizontal/vertical by the ORTHOcommands. CLOSE option uses the starting point of the firstline segment in the current LINE command as the next point.

(i). Lines can be drawn using co-ordinate system (rectangular cartesian co-ordinates). To draw arectangle (Fig. 21.4a):

Command: LINE

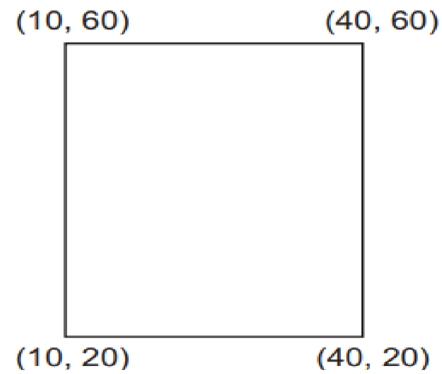
From point: 10, 20 ↵

To point: 40, 20 ↵

To point: 40, 60 ↵

To point: 10, 60 ↵

To point: ↵



(ii). It is also possible to specify the co-ordinates in the incremental format as the distances from the current cursor position in the drawing area. The distance is specified by using the @ parameter before the actual value. To construct a triangle of given altitude (30) and base (40) (Fig. 21.4b):

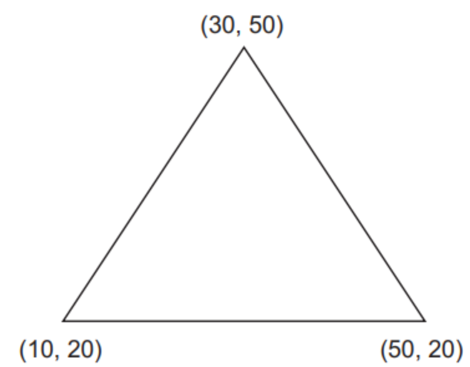
Command: LINE

From Point: 10, 20 ↵

To point: @ 40, 0 ↵

To point: @ – 20, 30 ↵

To point: ↵



(iii). It is also possible to specify the point co-ordinate using the ploar co-ordinate format. To construct a hexagon (Fig. 21.4c) of side 30:

Command: LINE

From point: 10, 20 ↵ (A)

To point: @ 30<0 ↵ (B)

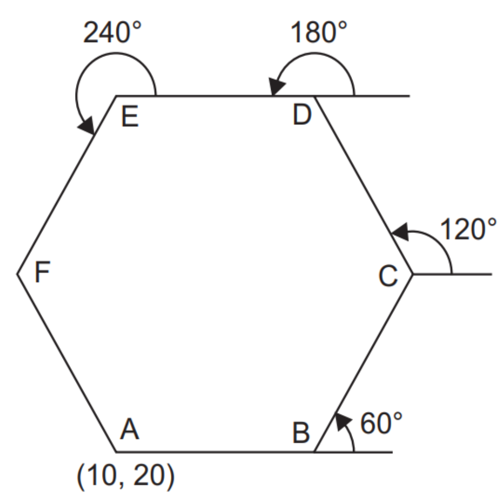
To point: @ 30<60 ↵ (C)

To point: @ 30<120 ↵ (D)

To point: @ 30<180 ↵ (E)

To point: @ 30<240 ↵ (F)

To point: close



C. Drawing Entity - POLYLINE

Polylines are interestingdrawing entities. Polylines can include both lines and arcs connectedat end-points. Thus, a polyline is a single entity with multiplesegments. The polylines can be straight or curved, can be wide(like a TRACE) or tapered. Fillets and chamfers can be addedwhere needed on a polyline. Curve fitting and hatching can easilybe performed on a polyline.

Polyline is basically a composite curve which is a combination of linear and arc segments in AutoCAD. The other property that can be varied is the thickness of the line drawn. Special properties of the polyline are:

(a). All the connected segments are treated as single entity.

(b). Width of line of any or all segments can be varied.

(c). It can also be a closed curve.

(d). Line type can be varied as required along various segments of the line.

Command: PLINE

Front point: 15, 25 (starting point). Once the starting point is selected, the computer willindicate current line width; 0,0 (default value)

When entered, the prompt responds:

Arc/Close/Half width/Length/Undo/Widt/<end point of line>:

By choosing (Fig. 21.22):

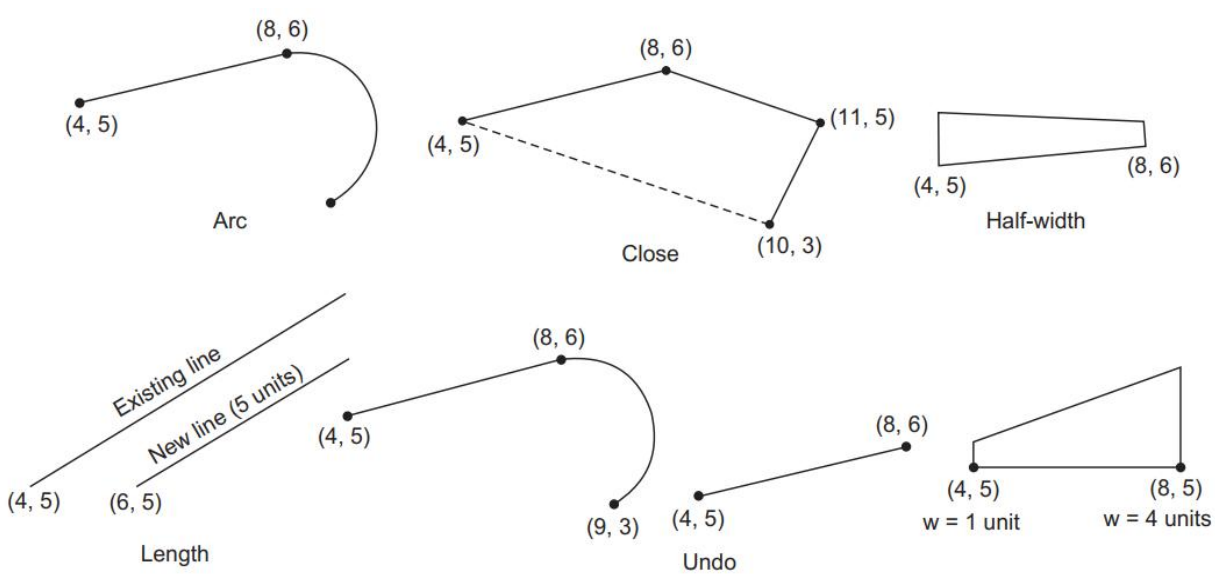


Fig. 21.22

Arc	— one can draw an arc with proper choice or selection
Close	— from the present position, the line joins with the starting point
Half width	— in case the starting width is chosen earlier; from this point, the width of the line will be half of it
Length	— length of the line may be given or co-ordinates may be given to draw the line
Undo	— the previous operation is reversed
Width	— one can specify the width at this point and also have a tapered line drawn by suitably instructing both starting width and ending width
<End of line>	— (default) by just entering, PLINE stops at the above point

NOTE PLINE may be used similar to LINE command with added advantages.

D. Drawing Entity –ELLPSE

This command allows one to draw ellipses or egg shaped objects. From Release 13 onwards, ellipse is treated as a separate entity. The methods available for making ellipses are:

1. By means of axis end points: (Fig. 21.5a)

Command: ELLIPSE <axis end point 1>/ center: point ↵

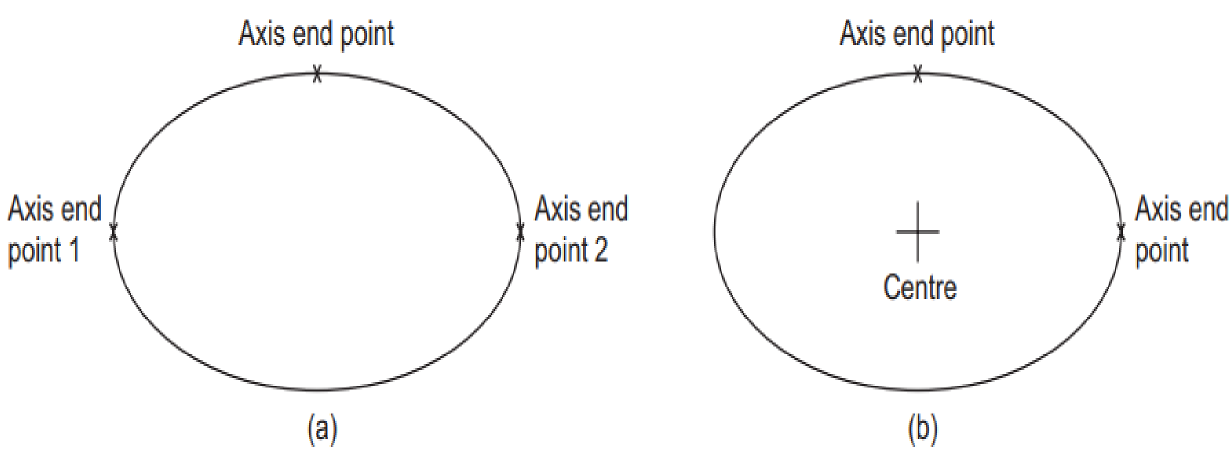
<other axis distance>/ Rotation:

Now, if the distance is entered, AutoCAD interprets it as half the length of the other axis.

2. By means of centre, axis end points (Fig. 21.5b)

Command: ELLIPSE <axis end point 1>/ centre: C ↵

Centre point and one end point of each axis should be provided for the response of the AutoCAD.



E. Drawing Entity – POLYGON

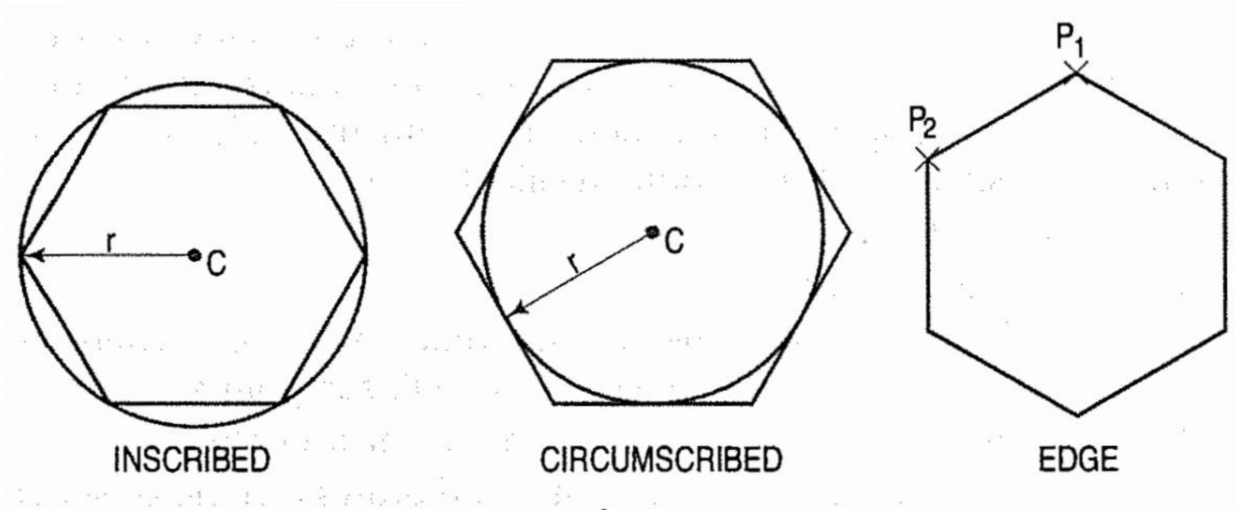
A polygonis also a polyline with equal length of sides. The regular polygoncan either be inscribed in a circle or circumscribed about thecircle. The polygon may also be constructed by specifying thelength of one side and the number of sides of polygon (callededges). In this method a polygon is constructed in anti-clockwisedirection from the two edge end-points that have been specified.

This option permits to make/draw polygons from 3 to 24 sides in a number of ways:
1. For making inscribed/circumscribed polygon with a side parallel to*x*-axis: (Fig. 21.6a, b)

Command: POLYGON
Number of sides: 6
Edge/ <centre of polygon>: 100, 200 ↵
Inscribed / circumscribed about a circle (I/C): I or C ↵
Radius of circle: 80

With edge option, specifying the size of the edge andorientation: (Fig. 21.7)

Command: POLYGON
Number of sides: 6
Edge/<center of polygon>: E ↵
First end point of edge: P1 15, 15 ↵
Second end point of edge: P2 15, 30 ↵



The above and various other entities that can be usedfor making an AutoCAD drawing may also be selectedfrom the tool bar.

F. Drawing Entity – RECTANGLE

A rectangle is a polygon based on two opposite cornerpoints, known as diagonal points (Fig. 21. 8).Rectangles are drawn by three methods

(i). By specifying the co-ordinates at the "specify first corner point" prompt and at the specify other corner point" prompt.

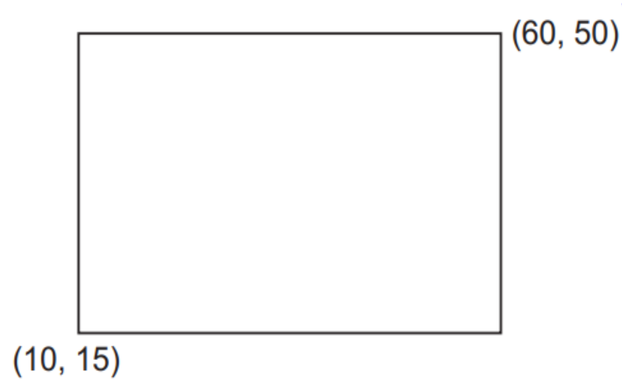
(ii). By entering area of rectangle in current units and specifying length or width of rectangle. For this select "Area" option of rectangle command.

(iii). By specifying length distance and width distance of rectangle. For this select ‘Dimensions option of rectangle command.

Command: RECTANGLE

First corner: 10, 15 ↵

Second corner: 60, 50 ↵

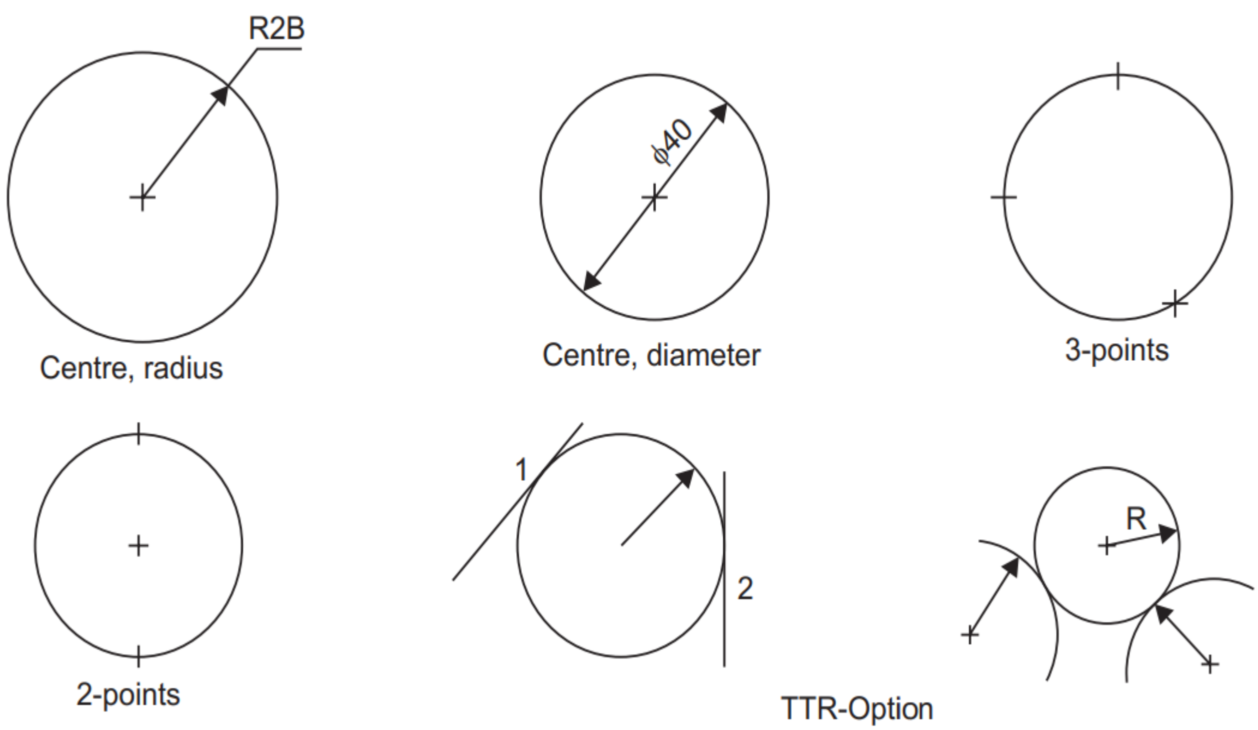


Or from the tool bar menu icon, the pointing device can drag the rectangle and therectangle can be completed.

G. Drawing Entity – CIRCLE

There are many ways of drawing acircle, the default being the centre point of circle and radius. Either on typing thecommand CIRCLE or selecting it from a menu bar with the help of mouse, all circledrawing options are displayed. The options available are:

1. Command: CIRCLE3P/ 2P/ TTR/ <centre point>: Pick a centre point or enter an option
2. Diameter/ <Radius><current default>: select D or RmeansCenter point and RadiusCenter point and Diameter.
3. 3P (3 point) option: one is prompted for a first, second and third point. The circle will be drawn to pass through these points.This specifies 3 points on the circumference of a circle. There is a uniquecircle passing through three given non-collinear points.
4. 2p (2 point) option: one is prompted for the selection of two points which form the opposite ends of the diameter.2P: This specifies the end-points of diameter of a circle
5. TTR option: allows one to define a circle based on two tangent points and a radius. The tangent points can be on lines, arcs or circles.
6. TTT (Tangent Tangent Tangent): This command draws a circle tangent to three entities.



H. Drawing Entity – ARC

This command is used todraw an arc accurately. Usually there are *three* parameters required for drawing an arc.Different ways of drawing circular arcs are:

3 point arc:The arc is drawn by specifying three points on the chord of arc.The first and third points define the start and end-points of an arc respectively.

Start, Center*:* This option needs start point and center point of an arc. Thethird parameter may either be an end-point, included angle, or length of chord.

Start, End:This option asks the user to enter the start and end-points of an arc.The arc iscompleted by either specifying radius or included angle or center point.

Arc command permits to draw an arc, using a variety of methods.

Command: ARC

Centre/ <start point>: pick a start point using mouse or select C for more options.

Centre/End/ <second point>: pick a second point of the arc or select C, if option is C.

Angle/length of chord/end point: pick end point of the arc, if option is E.

Angle/Direction/Radius/ <centre point>: pick end point of the arc or specify the option.

Options (Fig. 21. 10.)

Angle	— "included angle" prompt appears, to enter the value.
Centre	— enter the location of an arc's centre point-at the prompt centre-pick apoint,
Direction	— enter a tangent direction from the starting point of an arc. At this prompt, pick a point with cursor.
End	— at this prompt, pick the end point of the arc.
Length	— enter the length of a arc's chord. At this prompt, enter a length or dragand pick a length with cursor
Radius	— at the prompt "radius", enter a radius value
Start point	— enter the beginning point of an arc.

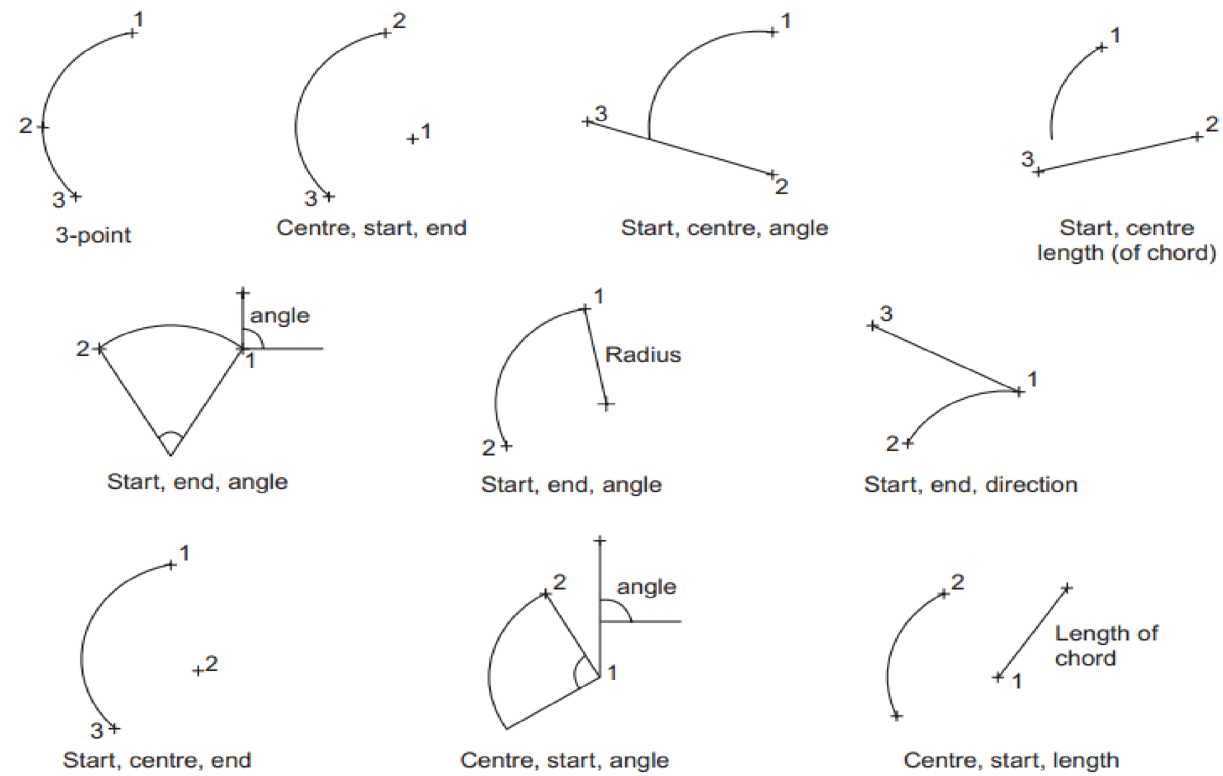


FIG. 26-5 Arc definitions

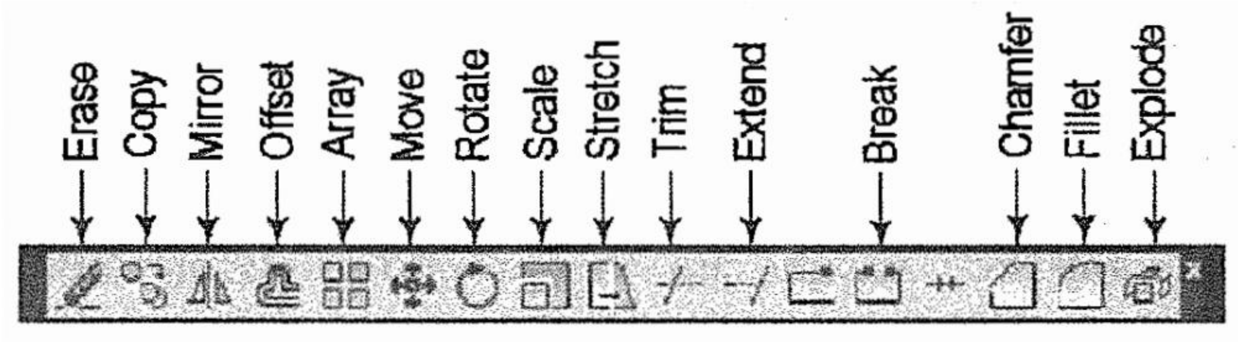
5.2. Editing and transfer tools methods (Modify Commands)

5.2.1. Introduction

Editing capabilities are the most useful part of AutoCAD system, by making use of the already existing drawing. For the purpose of editing an object, it is necessary to make selection of the objects in the drawing. There are various options available for the selection of an object:

1. **Pick box**—the cursor is converted to a small box/square, called pick box. By pressing the left button of the mouse when the pick box touches an entity, the object can be selected for editing.
2. **Window option**—a single or group entities can be selected by bringing them fully inside a rectangular window. Entities, which lie only partially inside the boundaries of window, will not be selected. Rectangular window may be created by picking the first corner, by pressing the left button and then moving the mouse to the desired position of diagonally opposite corner. Selection of the object is complete, by pressing the button again.

5.2.2. Modify Commands



The fundamental commands to edit a drawing are:

1) **MOVE**[fig. 26-8(i)]: Moves selected objects to anotherlocation about a base point.
2) **ROTATE** [fig. 26-8(ii)J (Refer module 26-'12): Rotates selectedobjects through a specified angle about a base point.
3) **COPY** [fig. 26-8(iii)] (Refer module 26-6): Creates one or more copies of selectedobjects at another location. The function of COPY command is similar to the MOVEcommand except that it preserves a copy of the objects selected at the original location.
4) **MIRROR** [fig. 26-8(iv)J (Refer module 26-8): Creates a mirror image of the selectedobjects about a specified line.
5) **ARRAY** (Refer module 26-10, 26-11): This command creates multiple copies ofselected objects in rectangular or polar form. This is a form of COPY command.
6) **ERASE** (Refer module 26-8): This command deletes the selected entities. A recordof entities erased is always maintained. The most recent entity can be *unerased* byOOPS command.
7) **OOPS** (Refer module 26-14): This command retrieves all objects erased by thelast Erase and after executing Block or Wblock command.
8) **BREAK:** This command erases a portion of line, arc, circle or a 20 polylinebetween two selected points
9) **FILLET** (Refermodule26-5, 26-13): This command is usedtocreatearoundcornerbetweentwo lines. The lines are shortened or extended to fit atangent arc of specified radius. FILLETworks on any combination of two lines, arcs, circles, non-parallel lines, or a single polyline.
10) **CHAMFER** (Refer module 26-8): This command works on two lines or a singlepolyline to create a bevelled edge.

11) **EXTEND** (Refer module 26-11): This command extends the lines, polylines andarcs to a boundary edge which can be a line, polyline, arc or circle. A closed polygoncannot be extended. When you invoke this command, you will be prompted to selectthe boundary edges. These edges can be lines, polylines, circles, arcs,, ellipse, xllines,rays,splines etc. After the boundary edges are selected, you must select each object tobe extended. An object can be both a boundary edge and an object to be extended.
12) **OFFSET** (Refer module 26-5): This command creates a parallel single copy ofline, arc, circle, rectangle, polygon, or 20 polyline at a given offset distance. Eachoffset creates a new entity with the same linetype, color and layer settings.
13) **STRETCH** (Refer module 26-12): The STRETCH command can either lengthenentities or shorten them, and thus alter their shapes. The centre points of arcs orpolyline arcs are adjusted accordingly.
14) **TRIM** (Refer module 26-5, 26-13): This command trims the objects that extendbeyondarequired pointofintersection.When you invokethiscommand, you will bepromptedto selectthe cutting edges. These edges can be lines, polylines, circles, arcs, ellipse, xlines, rays,splines, text, blocks or even viewports. After the cutting edges are selected, you must selecteach object to be trimmed. An object can be both acutting edge and an object to be trimmed.
15) **SCALE** (Refer module 26-12): The SCALE command allows to shrink or enlargethe already existing drawing objects about a base point by specifying a scale factor.
16) **PEDIT:** A polyline is a single entity which is made up of a continuous series of lineand arc segments. The PEDIT command is exclusively used for editing of polyline properties.The selected line, arc and polyline can be added to an existing polyline by a JOIN option.A smooth curve passing through all vertices of a polyline can be created by using FIT option.Similarly, a spline can also be constructed by using SPLINE option.
17) **EXPLODE** (Refer module 26-6): This command breaks a polyline into its individualsegments. These segments can then individually be edited, and rejoined again to form
    an edited polyline.
18) **U** (Refer module 26-8): The U command reverses the effects of a series ofpreviously used commands and hence allows back-stepping. The REDO command wipesout the effect of U t:Command.

The advanced features of AutoCAD are:

- Built-in programming language AutoLISP provides programming environmentso that AutoCAD commands can be called along with programs written forcomputations. This is very useful for parametric design and drawing.
- Drawing Exchange Files (DXF) and script files can be used to interface AutoCADwith programs written in any other higher level language such as C. DXF and IGESfile formats allow the exchange of drawing files among various drafting softwares.
- AutoCAD provides techniques to define and extract attributes of entities. Thisfeature is used in extracting information from a drawing for processing byother programs or to transfer it to a database.

5.3. Dimensions, text and symbols to drawing elements

In many applications, a drawing should contain annotations showing lengths or distances orangles between objects to convey the desired information. Dimensioning is the process of adding

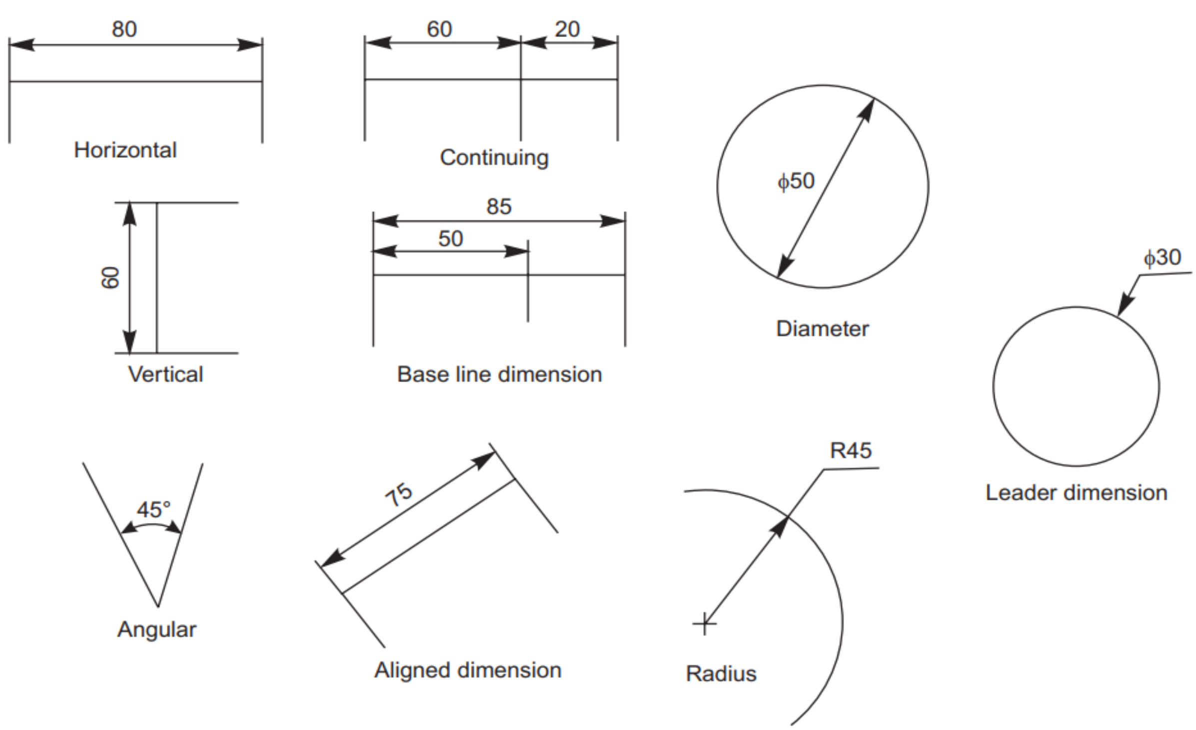
these annotations to a drawing. AutoCAD provides four basic types of dimensioning; linear, angular, diameter and radius.

DIM and DIMI Commands—DIMI command allows executing one dimensioning command and then returns to the normal command mode. If several dimensioning commands are to be executed, DIM command should be used. In this mode, the normal set of AutoCAD commands is replaced by a special set of dimensioning commands. To end the process of dimensioning, EXIT command has to be used.

The dimensioning commands can be grouped into six categories:

1) **Linear** is done with a horizontal, vertical, aligned and rotated command.However, rotated command requires specifying the dimension lineangle explicitly.
2) **Angular**is used to dimension angles. Here, one has to select two non-parallellines to introduce the angular dimension.
3) **Diameter** this can be invoked for dimensioning arcs and circles.
4) **Radius**it is almost identical to diameter dimensioning, except that only aradius line is drawn. This line has only one arrow.
5) **Associative** used to make various changes to associative dimension entities.
6) **Dimensioning utility commands**to draw a centre line or centre mark for a circle/arc, this command isused.

AutoCAD generally uses same type of dimensions and dimension label components as standard draughting. Figure 21.19 gives examples of types of dimensions possible: linear, angular, diametric, radial and aligned. A number of variables such as extension lines, text location, tolerance specifications, arrow styles and sizes, etc., actually control the way in which the dimensions may appear in the drawings.



5.3.1. Dimensioning Fundamentals

The student is already exposed to some definitions of fundamentals. However, the following are specific for AutoCAD:

(i). Base line dimension-a series of dimension lines, all starting at thesame extension line, that measure successive linear distances.

(ii). Continuing dimension aseries of dimension lines that follow one another alongsuccessive linear distances.

5.3.2. Dimensioning Methods

The procedure to be followed for dimensioning in AutoCAD is as follows:

1) Set-up the basic parameters for dimensioning. They are,
   (a). Arrow head size,
   (b). Arrow head type,
   (c). Extension line offset, and
   (d). Placement of dimension text.
2) Identify what to measure–pick the end points, lines, arcs or circles or other points ofexisting drawing entities using OSNAP if neccessary.
3) Specify where the dimension line and text are to be located.
4) Approve AutoCAD’s measurements or can type one’s own text.

Linear dimension

Command: DIM

Dim: hor/ver/ali/cont/ang/diam/rad/leader

Chose any one based on the requirement (Fig. 21.19)

First extension line origin or return to select: ↵

Select line, arc, or circle: pick

Dimension line location: pick

Dimension text <value>: ↵

Dim: exit

Continuing Linear dimension

Often a series of related dimensions must be drawn, sometimes several dimensions are measuredfrom the same base line; other times one long dimension is broken into shorter segments thatadd upto the total measurement. The base line and continue commands are provided to simplifythese operations. Draw the first dimension, using horizontal, vertical, aligned or rotatedcommands. Then enter base line or continue. AutoCAD proceeds directly to the "second extensionline origin" prompt, and then asks for the dimension text. The dimension line is placed at thesame angle as the previous dimension.

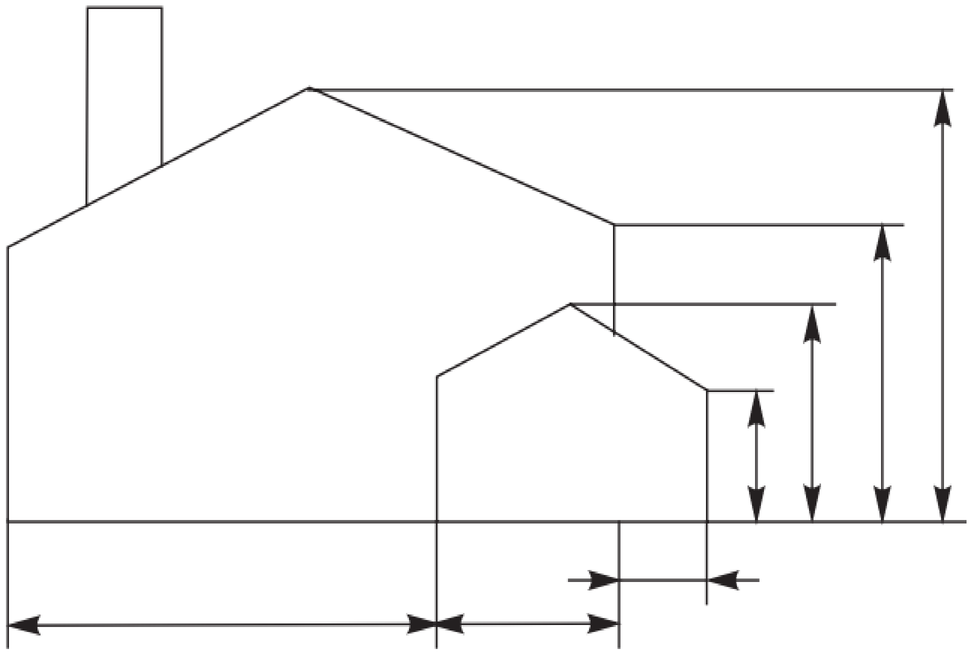


Figure 5.4the horizontal dimensions using continue command and the vertical dimensions using base line command.

When the base line command is used, AutoCAD offsets each new dimension line by an amount to avoid overlaying the previous dimension line. The first dimension line is extended accordingly. Dimension line off-setting can also occur with the continue command if either the new or previous dimension has its arrows outside the extension lines. In Fig. 21.20, the horizontal;dimensions are drawn using continue command and the vertical dimensions are drawn usingbase line command.

5.3.3. Text to drawing elements

TEXT Command — text may be added to a drawing by means of the TEXT command.

Text entities can be drawn with a variety of character patterns or fonts and can be stretched, compressed or drawn in a vertical column by applying a style to the font.

Command: TEXT

Start point or Align/Centre/Fit/Middle/Right/Style:

By choosing.

Start point	— Left justifies the text base line at the designated point
A (Align)	— prompts for two end points of the base line and adjusts overall Charactersize so that text just fits between these points.
C (Centre).	— asks for a point and centers the text base line at that point
F (Fit)	— similar to ‘align’, but uses a specified fixed height.
M (Middle)	— like ‘centre’, but centers the text both horizontally and vertically at the designated middle point.
R (Right)	— right justifies the text base line at that point.
S (Style)	— asks for a new text style
Null reply	— places the new text directly below the highlighted text

5.4. Perform Drawings using commands.

Operation Sheet 1.

Draw a line diagram as shown in fig. 26-9. Use Absolute CoordinateMethod.To draw a line diargram using Absolute Coordinate Method, follow the stepsmentioned below.

(1) Command: **LIMITS↩**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: ↩

Specify upper right corner <12.0000,9.0000>: **120,90 ↩**

(2) Command: **ZOOM** ↩

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **ALL ↩**

Regenerating model.

(3) Command: **LINE↩**

Specify first point: **20,20 ↩**

Specify next point or [Undo]: **100,20↩**

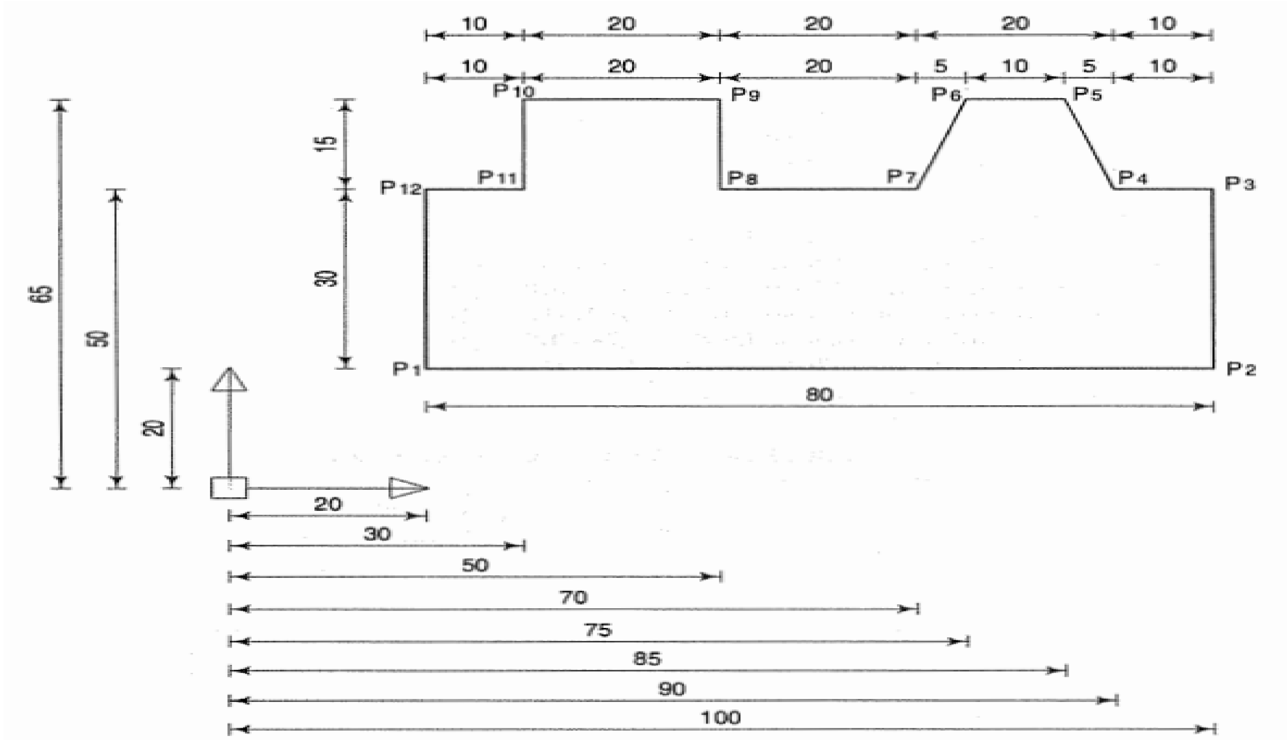
Specify next point or [Undo]: **100,50 ↩**

Specify next point or [Close/Undo]: **90,50 ↩**

Specify next point or [Close/Undo]: **85,65** ↩
Specify next point or [Close/Undo]: **75,65** ↩
Specify next point or [Close/Undo]: **70,50** ↩
Specify next point or [Close/Undo]: **50,50** ↩
Specify next point or [Close/Undo]: **50,65** ↩
Specify next point or [Close/Undo]: **30,65** ↩
Specify next point or [Close/Undo]: **30,50** ↩
Specify next point or [Close/Undo]: **20,50**↩
Specify next point or [Close/Undo]: **C** ↩

(4) Save This File As **Module 26-1.DWG**

Output of Module 26-1 (fig. 26-9):



Operation Sheet 2

Draw a line diagram as shown in fig. 26-10. Use Relative CoordinateMethod. To draw a line diargram using Relative Coordinate Method, follow the steps mentioned below.

(1) Command: **LIMITS** ↩

Reset Model space limits:

Specify lower left corner or [ON/OFF) <0.0000,0.0000>: ↩

Specify upper right corner <12.0000,9.0000>: **120,90** ↩

(2) Command: **ZOOM** ↩

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **All** ↩

Regenerating model.

(3) Command: **LINE ↩**

Specify first point: **20,20 ↩**

Specify next point or [Undo): **@80,0 ↩**

Specify next point or [Undo]: **@0,30 ↩**

Specify next point or [Close/Undo]: **@-10,0↩**

Specify next point or [Close/Undo): **@-5,15 ↩**

Specify next point or [Close/Undo]: **@-10,0↩**

Specify next point or [Close/Undo]: **@-5,-15 ↩**

Specify next point or [Close/Undo]: **@-20,0 ↩**

Specify next point or [Close/Undo]: **@0,15 ↩**

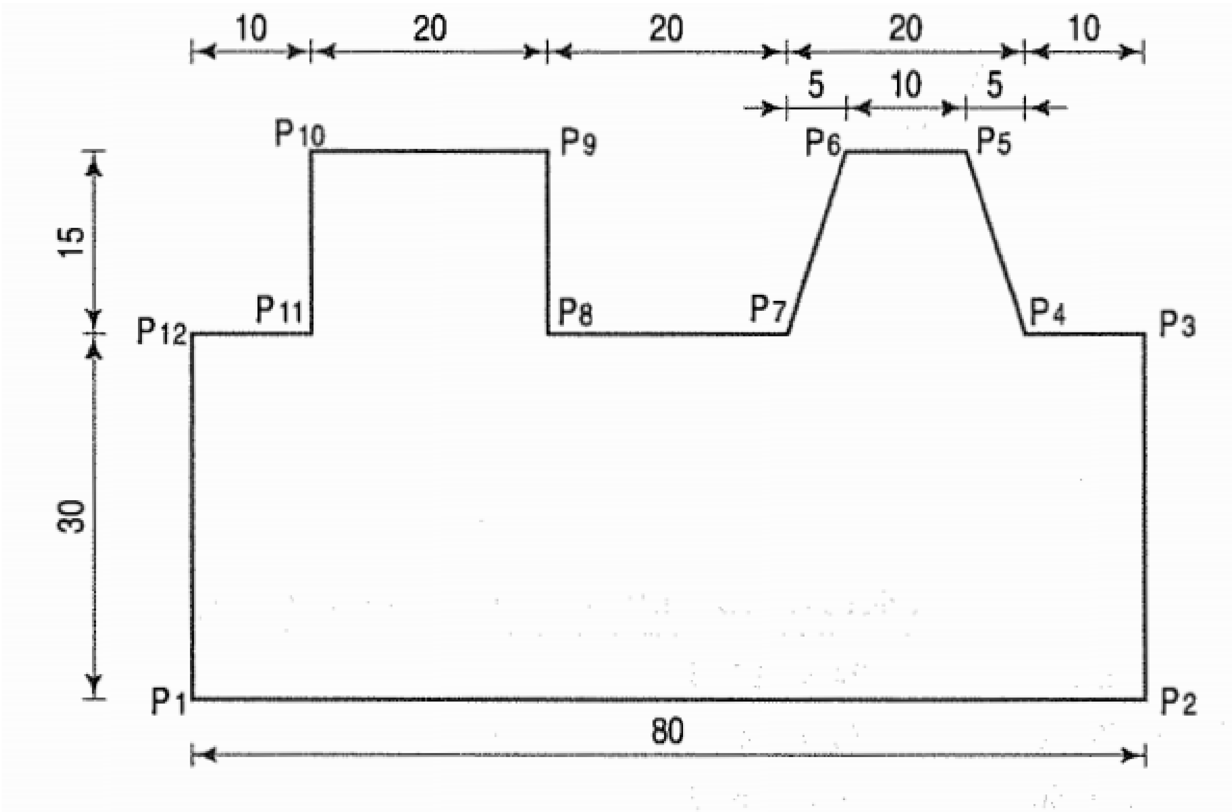
Specify next point or [Close/Undo): **@-20,0 ↩**

Specify next point or [Close/Undo]: **@0,-15 ↩**

Specify next point or [Close/Undo]: **@-10,0 ↩**

Specify next point or [Close/Undo]: **C ↩**

(4) Save This File As **operation 1 DWG**


10
20
20
20
10
5
10
5
P10
P9
P6
P5
15
P12
P11
P8
P7
P4
P3
30
P1
P2
80

Operation sheet 3.

Draw a line diagram as shown in figure below. Use Circle, Offset Trim and Fillet commands. To draw a line diagram using circle, offset, trim and fillet commands, followthe steps mentioned below. After executing the commands in sequence, we will getthe output as shown in figure below

1) {Switch ON POLAR switch by Mouse Click or press **F10** key}
2) Command: **LIMITS** ↩

   Reset Model space limits:

   Specify lower left corner or [ON/OFF] <0.0000,0.0000>: ↩

   Specify upper right corner <12.0000,9.0000>: **120,90** ↩
3) Command: **ZOOM** ↩

   Specify corner of window, enter a scale factor (nX or nXP), or

   [All/Center/Dynamic/Extents/Previous/Scale/Window] < real time>: **ALL** ↩

4) Command: **LINE** ↩

   Specify first point: **20,20** ↩

   Specify next point or [Undo]: **90** ↩

   (When **0**° POLAR is ON)

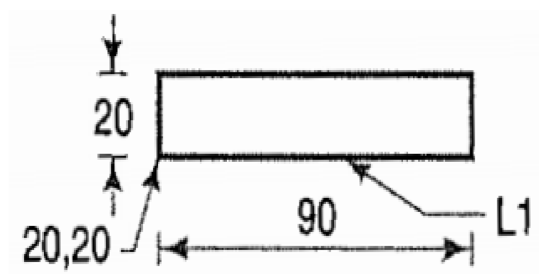
   Specify next point or [Undo]: **20** ↩

   (When **90**° POLAR is ON)

   Specify next point or [Close/Undo]: **90**↩

   (When **180**° POLAR is ON)

   Specify next point or [Close/Undo]: **C** ↩



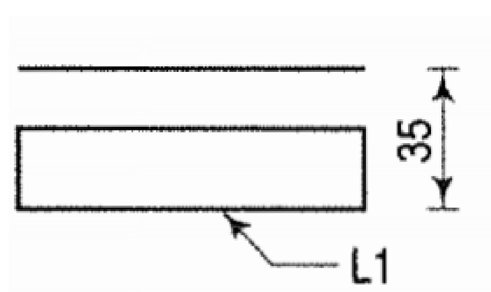
5) Command: **OFFSET** ↩

   Specify offset distance or [Through] <Through>: **35** ↩

   Select object to offset or <exit>: (Select Line **L1)**

   Specify point on side to offset: (Click Mouse Above Line **L1)**

   Select object to offset or <exit>: ↩



6) Command: **CIRCLE ↩**

   Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (Select MIDPOINT magnet of **L3)**

   Specify radius of circle or [Diameter]: **D ↩**

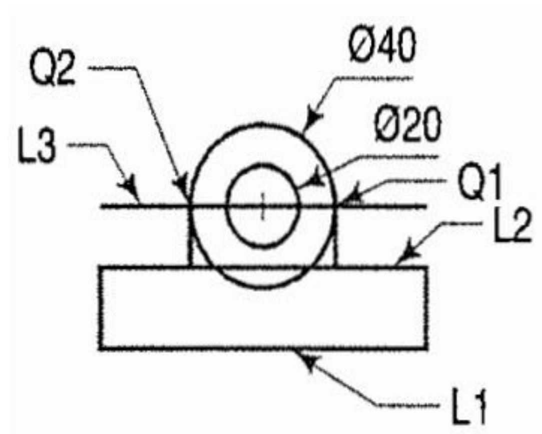
   Specify diameter of circle: **40 ↩**

7) Command: **CIRCLE ↩**

   Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (Select MIDPOINT magnet of **L3)**

   Specify radius of circle or [Diameter] <20.0000>: D↩

   Specify diameter of circle <40.0000>: 0 ↩



8) Command: **LINE ↩**

   Specify first point:(Select **0°** QUADRANT magnet **Q1 of 040** Circle)

   Specify next point or [Undo]:(Select PERPENDICULAR magnet on Line **L2)**

   Specify next point or [Undo]: ↩

9) Command: **LINE ↩**

   Specify first point:(Select **180°** QUADRANT magnet **Q2 of 040** Circle)

   Specify next point or [Undo]:(Select PERPENDICULAR magnet on Line **L2)**

   Specify next point or [Undo]: ↩

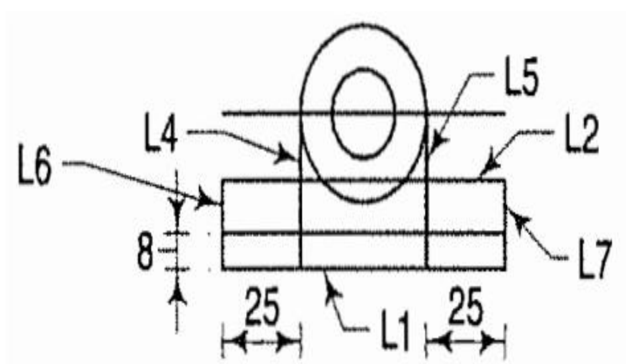
10) Command: **OFFSET ↩**

   Specify offset distance or [Through] <35.0000>: **8↩**

   Select object to offset or <exit>: (Select Line **L1)**

   Specify point on side to offset: (Click Mouse Above Line **L1)**

   Select object to offset or <exit>: ↩



11) Command: **OFFSET ↩**

Specify offset distance or [Through] <8.0000>: **25 ↩**

Select object to offset or <exit>: (Select Line **L6)**

Specify point on side to offset: (Click Mouse on Right Side of Line **L6)**

Select object to offset or <exit>: (Select Line **L7)**

Specify point on side to offset: (Click Mouse on Left Side of Line **L7)**

Select object to offset or <exit>: ↩

12) Command: **TRIM ↩**

Current settings: Projection=UCS, Edge=None

Select cutting edges ...

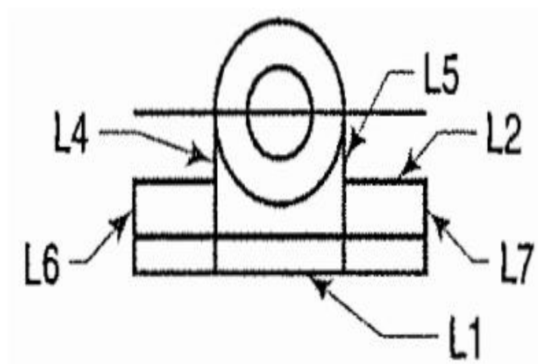
Select objects: (Select Line **L4)**

Select objects: (Select Line **LS)**

Select objects: ↩

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select Line **L2**from Circle Portion)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: ↩



13) Command: **TRIM ↩**

Current settings: Projection=UCS, Edge=None

Select cutting edges ...

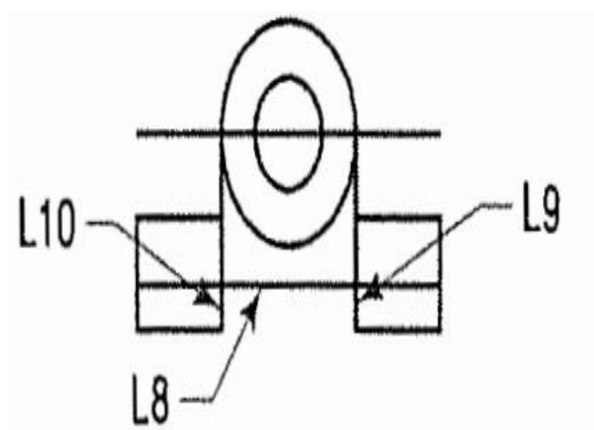
Select objects: (Select Line **L9)**

Select objects: (Select Line **L10)**

Select objects: ↩

14) Select object to trim or shift-select to extend or [Project/ Edge/Undo]: (Select Line L1 from Arrow Head)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: ↩



15) Command: **FILLET** ↩

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Polyline/Radiu?/Trim]: R ↩

Specify fillet radius <0.0000>: 2.5 ↩

Select first object or [Polyline/Radius/Trim]: (Select LineL8 from Arrow Head)

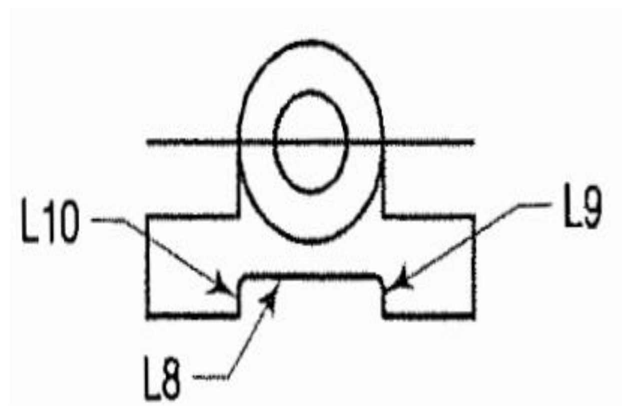
Select second object: (Select Line L10 from Arrow Head)

16) Command: **FILLET** ↩

Current settings: Mode = TRIM, Radius = 2.5000

Select first object or [Poly line/Radius/Trim]: (Select Line L8 from Arrow Head)

Select second object: (Select Line L9 from Arrow Head)



17) Command: **FILLET** ↩

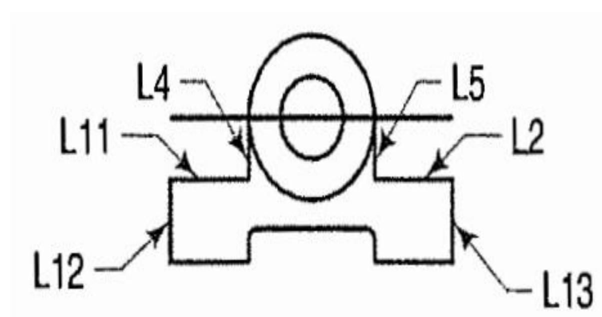
Current settings: Mode = TRIM, Radius = 2.5000

Select first object or [Poly line/Radius/Trim]: R ↩

Specify fillet radius <2.5000>: ↩

Select first object or [Poly line/Radius/Trim]: (Select Line L4 from Arrow Head)

Select second object: (Select Line L11 from ArrowHead)

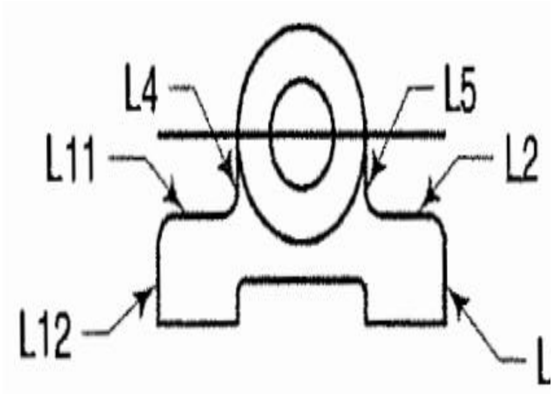


18) Command: **FILLET**↩

Current settings: Mode = TRIM, Radius = 5.0000

Select first object or [Poly line/Radius/Trim]: (Select Line LS from Arrow Head)

Select second object: (Select Line L2 from Arrow Head)



19) Command: **FILLET ↩**

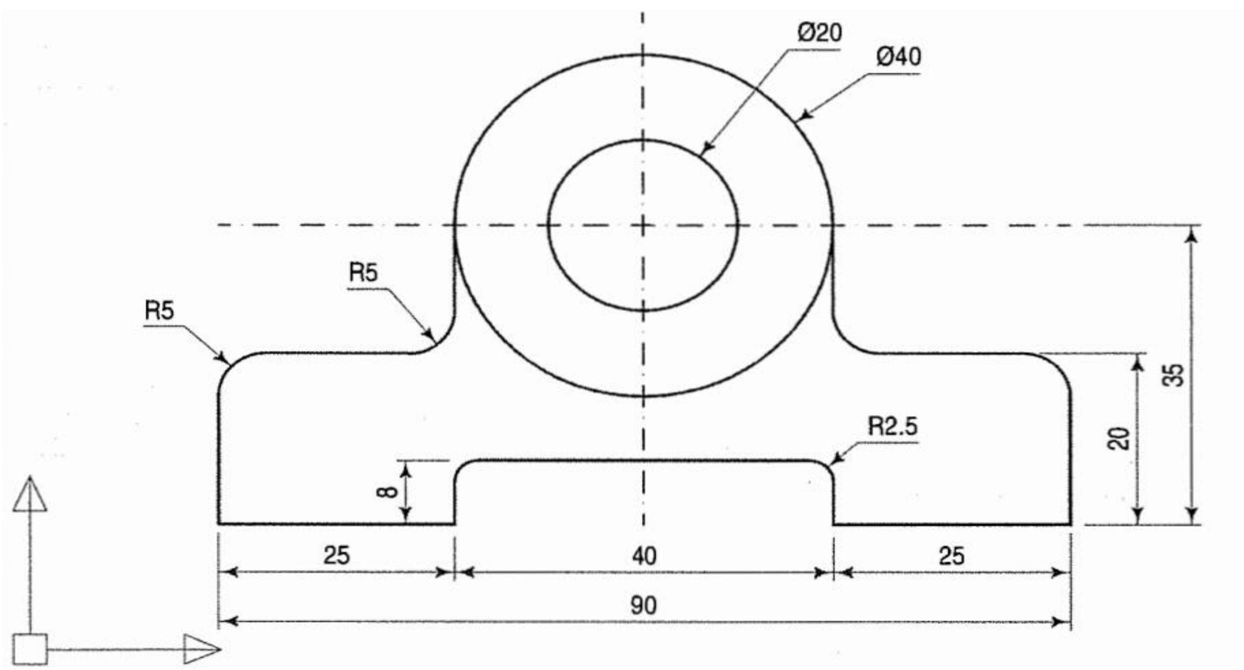
Current settings: Mode = TRIM, Radius = 5.0000

Select first object or [Poly line/Radius/Trim]: (Select

Line L11 from Arrow Head)

Select second object: (Select Line L12 from Arrow Head)

20) Save This File As Module 26-5.DWG



Operation sheet 4

Draw a line as shown in figure below. Use Polygon1 Chamfer Arc and mirror. To draw a line diagram using Polygon, Chamfer, Arc and Mirror commands, follow the steps mentioned below. After executing the commands in sequence, we will get the output as shown in fig, below.

(1)Command: **LIMITS ↩**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: ↩

Specify upper right corner <12.0000,9.0000>: **120,90 ↩**

(2) Command: **ZOOM ↩**

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] < real time>: **ALL ↩**

Regenerating model.

(3) Command: **LINE ↩**

Specify first point: **30,65 ↩**

Specify next point or [Undo]: **20 ↩**(When **0°** POLAR is ON)

Specify next point or [Undo): **10 ↩**(When **270°** POLAR is ON)

Specify next point or [Close/Undo): **20 ↩**(When **0°** POLAR is ON)

Specify next point or [Close/Undo): **10 ↩**(When **90°** POLAR is ON)

Specify next point or [Close/Undo): **20 ↩)** (When **0°** POLAR is ON)

Specify next point or [Close/Undo]: ↩

(4) Command: **ARC ↩**

Specify start point of arc or [Center): (Click Point **Pl** When ENDPOINT magnet gets ON)

Specify second point of arc or [Center/End): **E ↩**

Specify end point of arc: **@0,-40** ↩(Point **P2)**

Specify center point of arc or [Angle/Direction/Radius]:**A** ↩

Specify included angle: **180** ↩

(5)Command: **CHAMFER** ↩

(TRIM mode) Current chamfer Dist1 = 0.5000, Dist2= 0.5000

Select first line or [Poly line/Distance/Angle/Trim/Method):**D** ↩

Specify first chamfer distance <0.5000>: **4** ↩

Specify second chamfer distance <4.0000>: **4** ↩

Select first line or [Poly line/Distance/Angle/Trim/Method): (Select Line **L2)**

Select second line: (Select Line **L3)**

(6) Command: **CHAMFER** ↩

(TRIM mode) Current chamfer Dist1 = 4.0000, Dist2 = 4.0000

Select first line or [Poly line/Distance/Angle/Trim/Method): (Select Line **L3)**

Select second line: (Select Line **L4)**

(7) Command: **MIRROR** ↩

Select objects: (Select Arc **A1)**

Select objects: ↩

Specify first point of mirror line: (Click Midpoint magnet of Line **L3)**

Specify second point of mirror line: (Click When**270°** POLAR is ON)

Delete source objects? [Yes/No] <N>: **N** ↩

(8) Command: **MIRROR** ↩

Select objects: (Select All Objects in Dotted BOX)

Select objects: ↩

Specify first point of mirror line: (Click MIDPOINT of Arc **Al)**

Specify second point of mirror line: (Click MIDPOINT of Arc **A2)**

Delete source objects? [Yes/No] <N>: **N** ↩

(9) Command: **POLYGON** ↩

Enter number of sides <4 >: **6** ↩

Specify center of polygon or [Edge): (Select Center magnet of Arc **A1)**

Enter an option [Inscribed in circle/Circumscribe about circle) <I>: I ↩

Specify radius of circle: **8** ↩

(10) Command: **MIRROR** ↩

Select objects: (Select Hexagon **H1)**

Select objects: ↩

Specify first point of mirror line: (Click MIDPOINT of **L3)**

Specify second point of mirror line: (Click When **270°** POLAR is ON)

Delete source objects? [Yes/No] <N>: **N** ↩

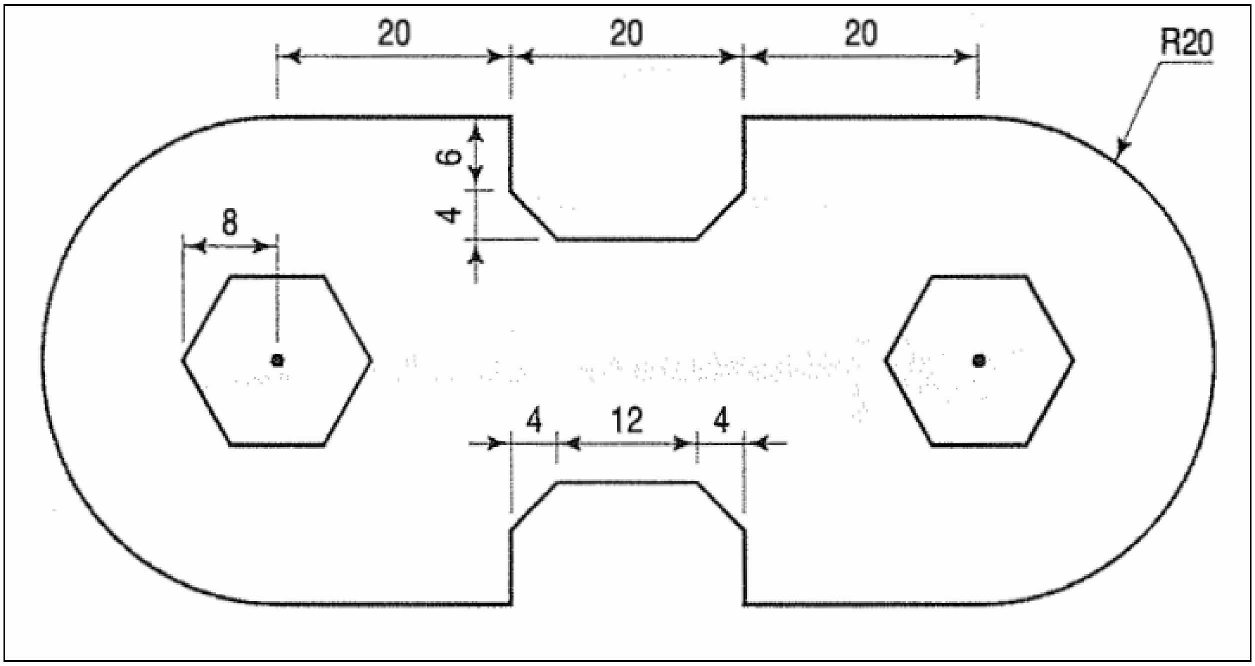
(11) Command: **ERASE** ↩

Select objects: **ALL** ↩

Select objects: ↩

(12) Command: **U** ↩

(13) Save This File As **Operation.DWG**



Operation Sheet 5

Draw a line diagram as shown in figure below. Use Donut and Rectangular Array, commands. To draw a line diagram using Donut and Rectangular Array commands, follow the steps mentioned below. After executing the commands in sequence, we will get the output as shown in figure below

(1) Command: **LIMITS** ↩

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: ↩

Specify upper right corner <12.0000,9.0000>: **120,90** ↩

(2) Command: **ZOOM** ↩

Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **ALL** ↩

Regenerating model.

(3) Command: **RECTANGLE**↩

Specify first corner point ↩or [Chamfer/Elevation/Fillet/Thickness/Width]: **10, 20** ↩

Specify other corner point or [Dimensions]:**@100,50** ↩

(4) Command: **DONUT** ↩

Specify inside diameter of donut <0.5000>: **4** ↩

Specify outside diameter of donut <1 .0000>: **6** ↩

Specify center of donut or <exit>: **FROM** ↩

Base point: (Select **R1** Point by ENDPOINT magnet) <Offset>:**@10,10** ↩

Specify center of donut or <exit>: ↩

(5) Command: **ARRAY** ↩

Select objects: (Select Donut **D1)**

Select objects: ↩

(1) Select Rectangular area

(2) Click here and select object i.e. donut

(3) Enter No. of columns= *5*

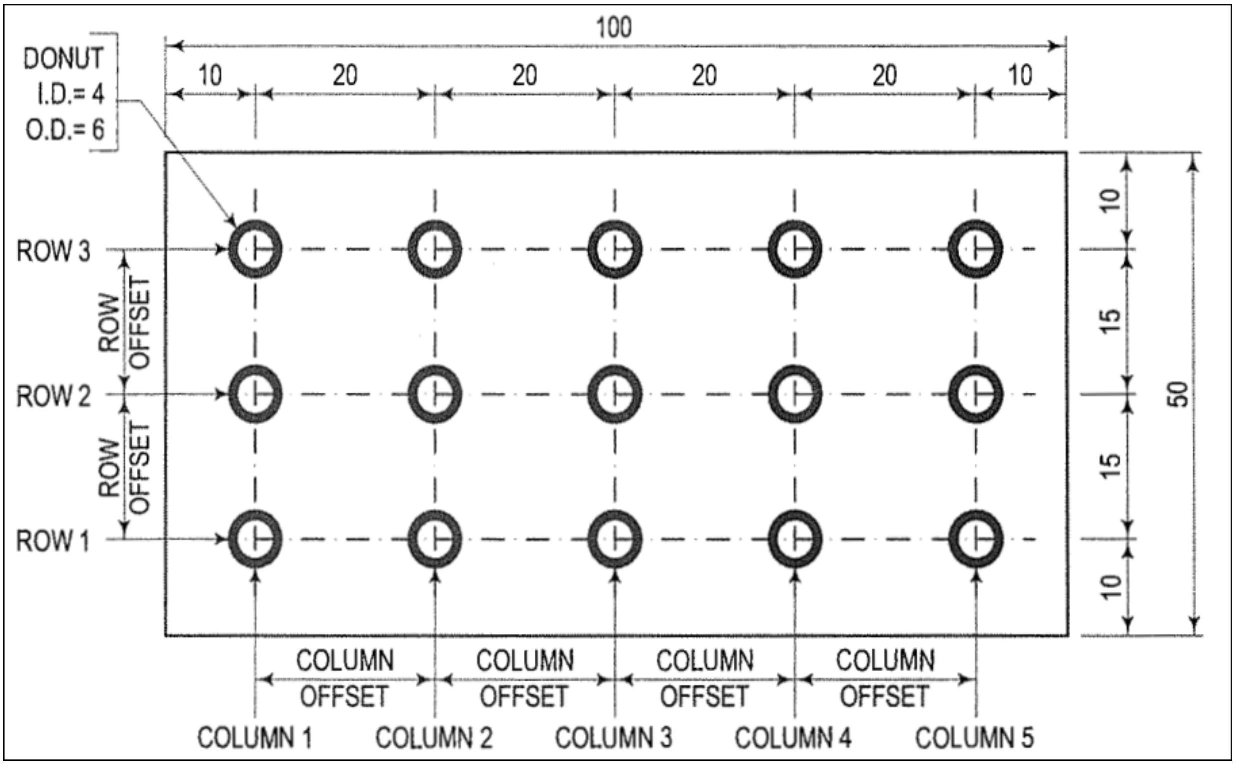
(4) Enter No. of rows= 3

(5) Enter row distance= 15

(6) Enter column distance= 20

(7) Finally click OK

(6) Save this File As Operation 5 **DWG**


100
10
20
20
20
20
10
DONUT
I.D.= 4
O.D.= 6
ROW 3
ROW 2
ROW 1
ROW OFFSET
ROW OFFSET
10
15
15
10
50
COLUMN OFFSET
COLUMN OFFSET
COLUMN OFFSET
COLUMN OFFSET
COLUMN 1
COLUMN 2
COLUMN 3
COLUMN 4
COLUMN 5

Operation Sheet 6

Draw a diagram as shown figure below. In Array commands, Use Ellipse and Polar. To draw a diagram using Ellipse and Polar Array commands, follow the steps mentioned below. After executing the commands in sequence, we will get the output as shown in figure below

(1) Command: **LIMITS** ↩

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: ↩

Specify upper right corner <12.0000,9.0000>: **120,90** ↩

(2) Command: **ZOOM** ↩

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **ALL** ↩Regenerating model.

(3) Command: **CIRCLE** ↩

Specify center point for circle or [3P/2P/Ttr (tantan radius)]: **60,45** ↩

Specify radius of circle or [Diameter]: **D** ↩

Specify diameter of circle: **80** ↩

(4) Command: **CIRCLE** ↩

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (Select CENTER magnet **C1** of **Φ80**Circle

Specify radius of circle or [Diameter] <40.0000>:**D** ↩

Specify diameter of circle <80.0000>: **60** ↩

(5) Command: **ELLIPSE** ↩

Specify axis endpoint of ellipse or [Arc/Center]: **C**↩

Specify center of ellipse: (Select **90°** QUADRANT**Q1** magnet of **060** Circle)

Specify endpoint of axis: 6 ↩(when **90°** POLAR is ON)

Specify distance to other axis or [Rotation]: **3** ↩

(6) Command: **ARRAY** ↩

Specify center point of array: (Select CENTER magnet**C1)**

Select objects: (Select Ellipse **E1)**

Select objects:

(1) Click Polar Array

(2) Click here and select objectas Ellipse

(3) Click here and select Center Cl

(4) Enter .\Jo. Of Items i.e. 8

(5) finally click OK

(7)Command: **DIMSCALE ↩**

Enter new value for DIMSCALE

<1.0000>: **10 ↩**

(8) Command: **DIMCENTER ↩**

Select arc or circle:(Select Circle **CR1)**

(9) Command: **EXTEND**

Current settings: Projection=UCS, Edge=None

Select boundary edges ...

Select objects or < select all>: (Select Circle **CR1**)

Select objects↩

Select object to extend or shift-select to trim or[Fence/Crossing/Project/Edge/Undo]:(Click Right End of Line **L1)**

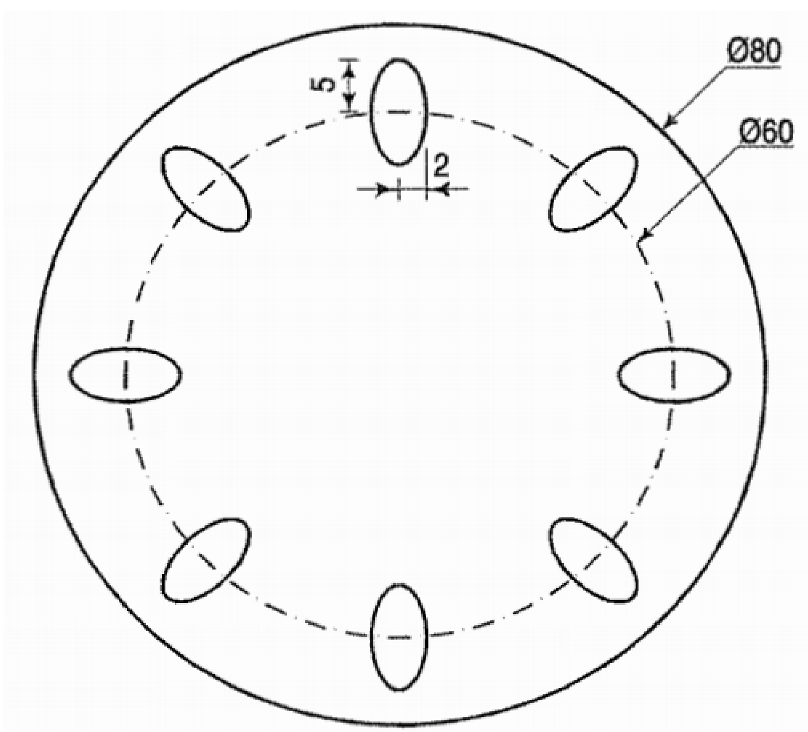
Select object to extend or shift-select to trim or[Fence/Crossing/Project/Edge/Undo]: (Click Left End of Line **L1)**

Select object to extend or shift-select to trim or[Fence/Crossing/Project/Edge/Undo]: (Click Top End of Line **L2)**

Select object to extend or shift-select to trim or[Fence/Crossing/Project/Edge/Undo]: (Click Bottom End of Line **L2)**

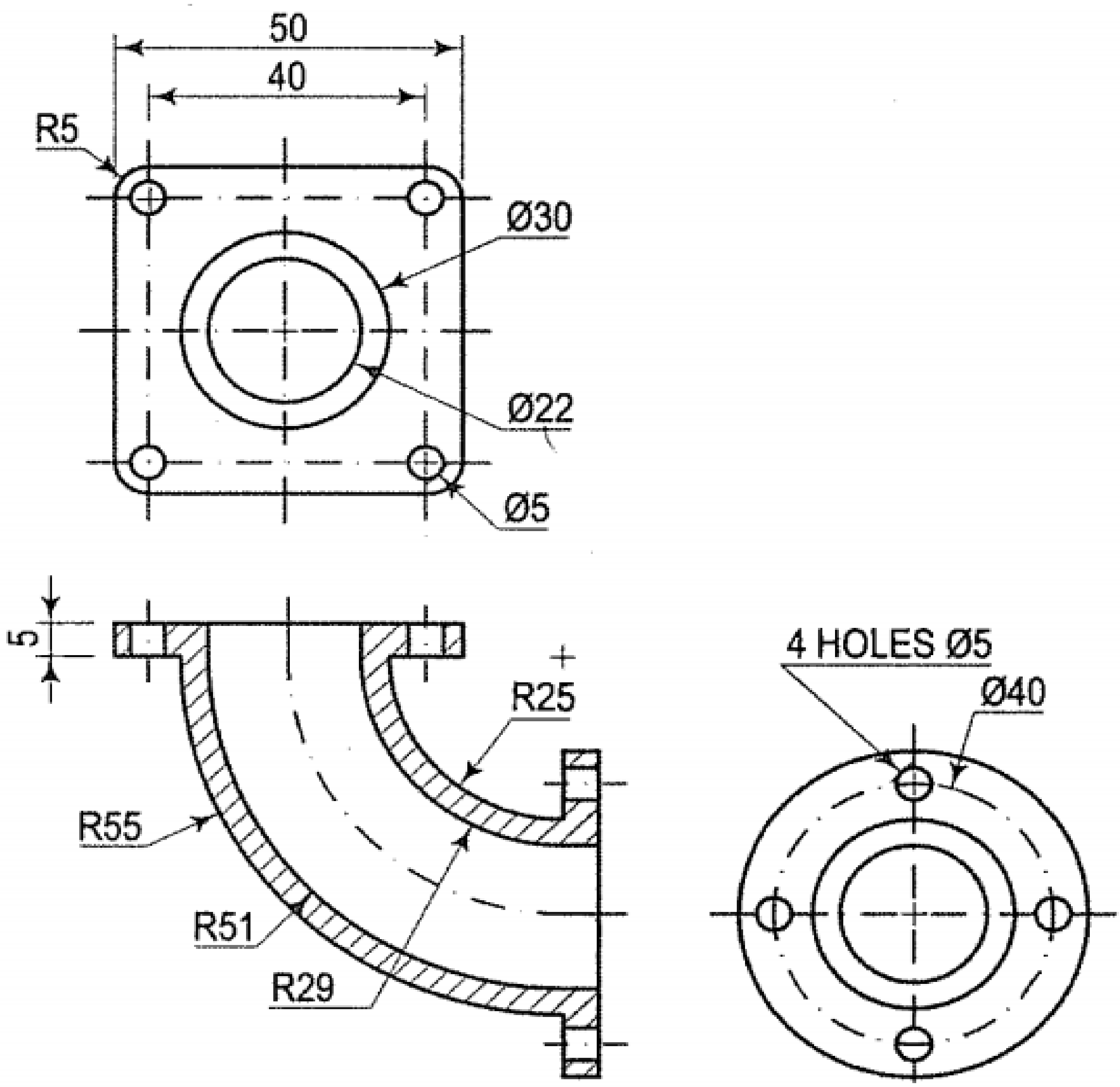
Select object to extend or shift-select to trim or[Fence/Crossing/Project/Edge/Undo]: **(.J)**

(10) Save This File As **operation 6.DWG**



LAP TEST

Draw figure shown in below using line, circle, rectangle, fillet, rectangular array, polar array, dimension, hatch, layer, hidden line, and center line etc., commands.



5.5. Perform Importing and exporting Files

5.5.1. Preparing a CAD file for import

A few steps to check and prepare your CAD file before import, you can make sure all the data you need imports into Sketch Up and streamline the import process. The following sections outline needs to look for and how to resolve any potential conflicts between CAD elements and Sketch Up.

- Change unsupported elements
- Delete unnecessary layers
- Move geometry close to the origin
- Reduce the file size if needed
- Check the unit of measure
- Save in a Sketch Up-compatible CAD file format

5.5.2. Importing a CAD file into Sketch Up

After checking and preparing the CAD file, you're ready to import it into Sketch Up. The following steps walk you through the import process for your operating system:

1. In Sketch Up, open the Sketch Up model into which you want to import your dwg or .dxf file.
2. Select **File > Import**. An Import dialog box appears.
3. Navigate the place your hard drive where your CAD file is saved.
4. From the **Files of Type** drop-down list, select **AutoCAD Files (*.dwg, *.dxf)**.
5. Select the file you want to import.
6. Click the **Options** button. The Import AutoCAD DWG/DXF Options dialog box appears, as shown in the following figure.
7. (Optional) In the Geometry area, select your preferences for the following options:
    - Select **Merge Coplanar Faces** to tell Sketch Up to automatically remove triangulated lines from planes.
    - Selecting the **Orient Faces Consistently** box tells Sketch Up to analyze the direction of imported faces and orient the faces so that their direction is uniform.
8. (Optional) Select the Preserve Drawing Origin checkbox to place the imported geometry at the origin defined in the .dwg or .dxf file. Leave the checkbox deselected if you want to place the imported geometry near the Sketch Up origin.

9. (Optional) To import geometry at the correct scale, select the unit used in your CAD file from the **Units** drop-down list. Your options are Model Units, Inches, Feet, Yards, Miles, Millimeters, Centimeters, Meters and Kilometers.

Note: If you don't know the units used in the original file, select Model Units so that SketchUp turns one CAD unit into 1 inch in Sketch Up. Or use a large unit type, such as feet or meters. Then you can resize the model as necessary after it's imported.

Warning: If you select a small unit, such as millimeters, but the model was originally intended to display in feet, you can unintentionally create itty-bitty faces that will be lost on import. For Sketch Up to recognize a face, the face must be .001 square inches or larger.

10. Click **OK** in the Import AutoCAD DWG/DXF Options dialog box.
11. Back in the Import dialog box, click the **Import** button, and Sketch up Pro begins importing your CAD file.

Note: When importing a Lay Out file created using the “Export for Sketch Up” feature; use the Merge Coplanar Faces option for clean imported faces.

Note: You can also drag and drop importable files into the drawing area. After you drop the file, any relevant Import dialog box opens so you can select your desired options.

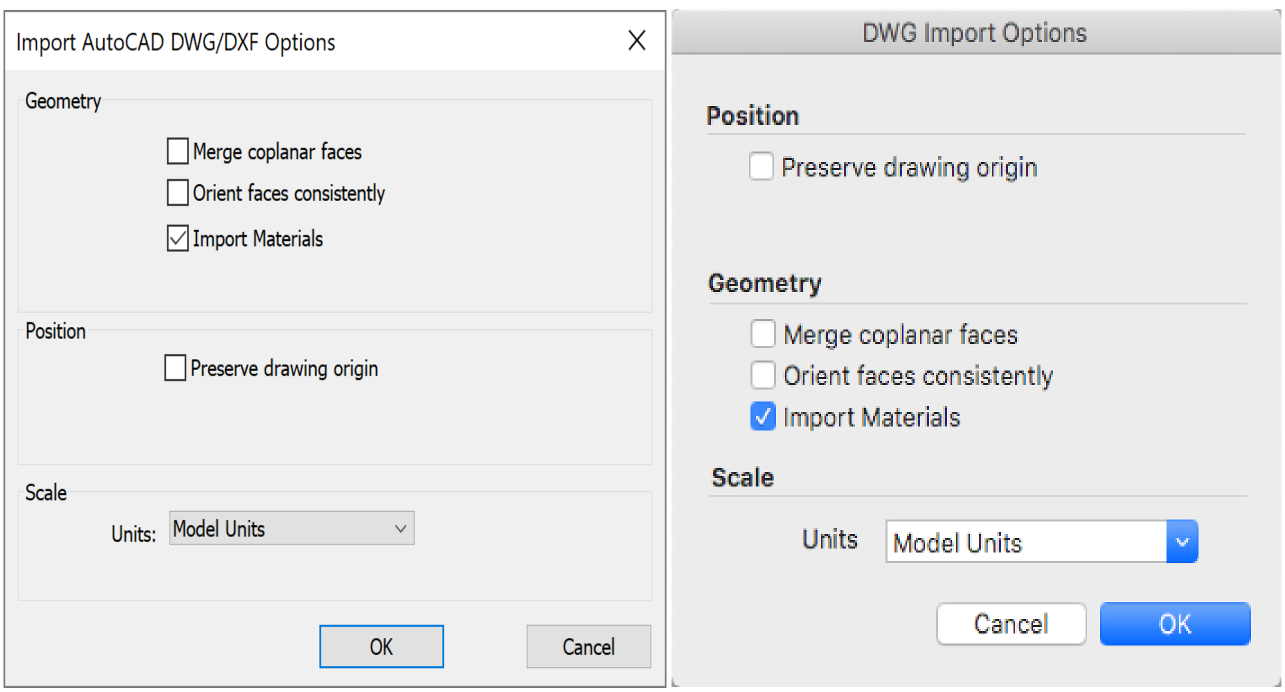


Figure 0.1 Import AutoCAD DWG/DXF and DWG import options window

5.5.3. Understanding how Sketch Up data is exported to CAD format

Before you export a Sketch Up model to a CAD file format (.dwg or .dxf), you may find it helpful to understand how Sketch Up data is translated into the CAD format that you choose. Here's what you need to know:

- Sketch Up faces are exported as a triangulated poly face mesh with interior splframe hidden lines (if applicable). This conversion helps to simulate the appearance of your Sketch Up file, even when all exported faces are triangular.
- Sketch Up uses the current units set in the Units pane of the Model Info dialog box as a reference for translation to a .dwg or .dxf file. For example, if the current Model Info unit setting is Decimal and Meters, then AutoCAD must be set to decimal for the units to translate correctly as meters when you open the exported file in AutoCAD.
- Duplicate line entities aren't created on top of a p-line entity.

5.5.4. Exporting a Sketch Up Model as a 2D CAD file

To export a model view as a 2D CAD file, choose the scale and a number of line options. The result is a 2D vector file in .dwg or .dxf format that you can open in CAD program.

Follow these steps to export a 2D CAD file:

1. To maintain your model's scale in the exported file, in Sketch Up, select **Camera** > **Parallel Projection**. Then set your view to one of Sketch Up's standard views by selecting **Camera** > **Standard Views** and selecting an option from the submenu that appears. (See Viewing a Model for details about the model view options.)
2. Select **File** > **Export** > **2D Graphic**.
3. Navigate to the place where you want to save your exported file.
4. (Optional) Change the file name if you'd like to name the exported file something other than the Sketch Up file's current name.
5. Select either the .dwg or .dxf file type. In Microsoft Windows, select this option from the **Save As Type** drop-down list. On a Mac, use the **Format** drop-down list.
6. Click the **Options** button to set the scale and line options, which are explained in detail in the upcoming list. When you're done, click **OK** in the DWG/DXF Hidden Line Options dialog box (Microsoft Windows) or the Export Options dialog box (Mac OS X).
7. Click **Export** and your CAD file is saved in your selected location.

In the following figure, you see the line options dialog box for your current operating system. The following list introduces your options so that the lines in your exported file meet your needs:

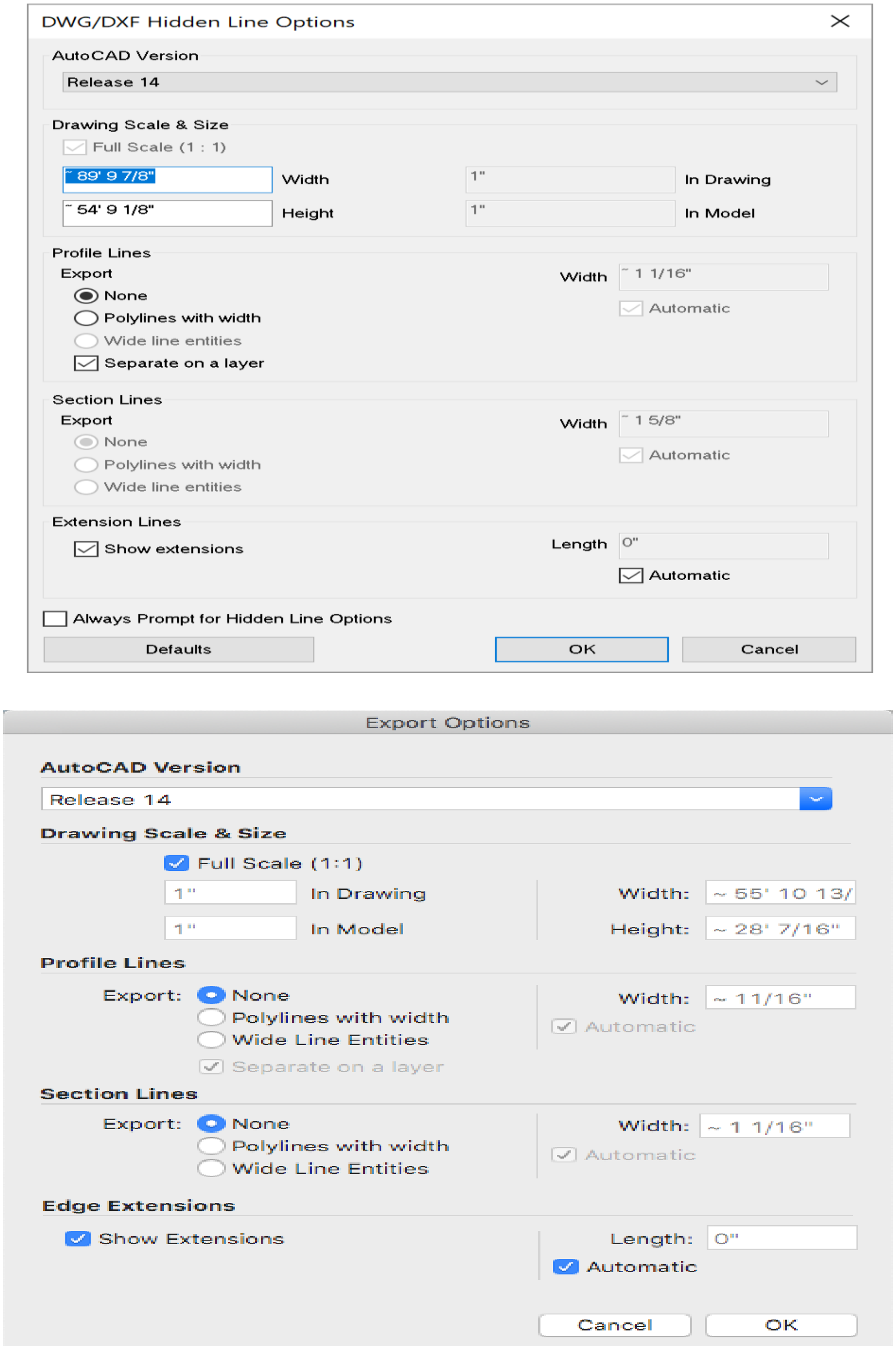


Figure 0.2 DXG/DXF Hidden line option and Export Option window

- **AutoCAD Version:** Select what version of AutoCAD you'd like to use to open the exported file.
- **Drawing Scale & Size:** The **Full Scale** checkbox is selected by default. However, if you deselect that checkbox, you can set a custom scale. In the **In Drawing** box, type the actual measurement you want to use to set your scale. In the **In Model** box, type the value for scaling your exported model. For example, for a scale of 1:4, type **1'** in the In Model box and **4'** in the In Drawing box.
- **Profile Lines:** Here, you can customize how profile lines appear in your exported file. You can set the line width as follows:
  - Select **None** to export profile lines at the standard width.
  - Select **Poly lines with Width** to export profile lines as AutoCAD poly lines. When you select this option, you can leave **Automatic** selected so that the exported lines match the profile line width. If you deselect Automatic, you can enter a custom width in the **Width** box.
  - Select **Wide Line Entities** to export profile lines as AutoCAD wide line entities.

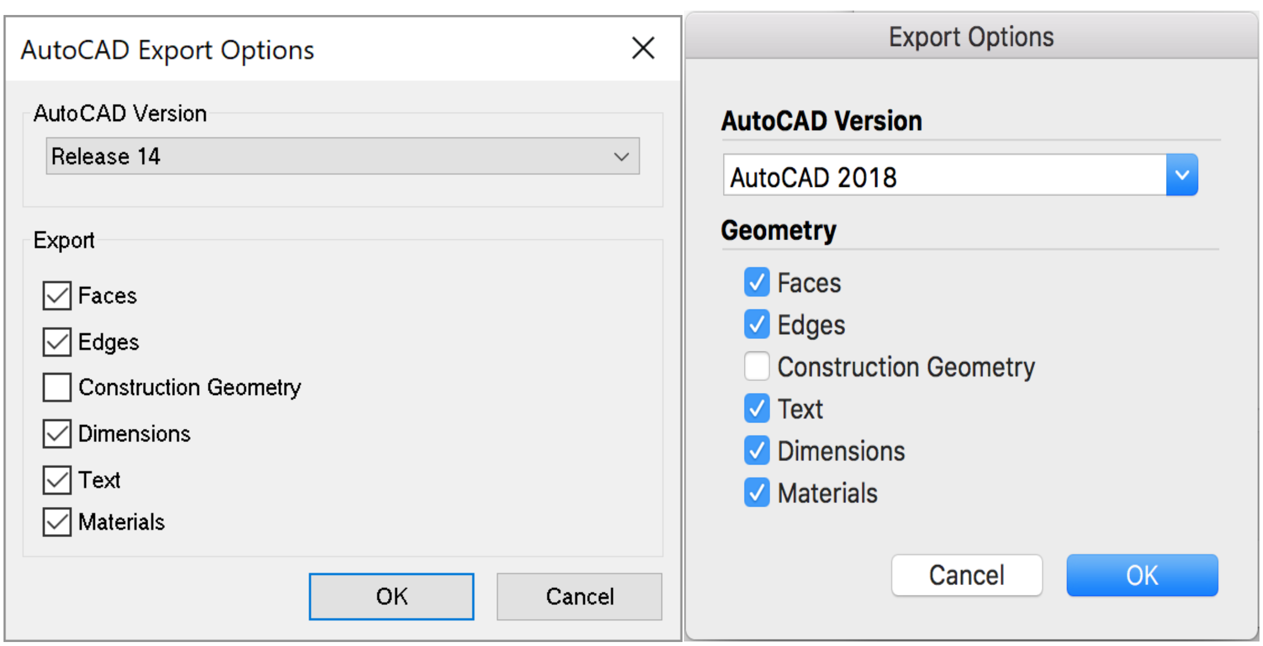
  The **Separate on a Layer** checkbox, which is selected by default, creates a layer for profile edges. If you've used Sketch Up layers to control visibility, note that Sketch Up layer assignments don't translate directly when you export a Sketch Up file to a 2D CAD file.
- **Section Lines:** If you export section lines or a section slice, these export options become available. See Slicing a Model to Peer Inside for details about sections and how to export them.
- **Edge Extensions:** Some CAD applications might have problems recognizing line endpoints and intersections when your model uses Sketch Up line extensions. Deselect the **Show Extensions** checkbox to toggle extensions off in your exported file. If you leave Show Extensions selected and deselect the **Automatic** checkbox, you can enter an exact length for line extensions in the **Length** box.

In Microsoft Windows, you can select the **Always Prompt for Hidden Line Options** if you'd like to set options in this dialog box anytime you export a 2D CAD file. You can also restore the default settings by clicking the **Defaults** button.

Exporting a Sketch up Model as a 3D CAD file

When you export a Sketch Up model as a 3D CAD file, you can select what entities are exported. To export your model, follow these steps:

1. In Sketch Up, select **File** > **Export** > **3D Model**. The Export Model dialog box appears.
2. Navigate to the location where you want to save your exported file.
3. (Optional) Change the file name if you like. By default, the exported file uses the same name as your Sketch Up file name.
4. Select either .dwg or .dxf as the file format for your exported file. In Microsoft Windows, select your file type from the **Save as Type** drop-down list. In Mac OS X, use the **Format** drop-down list.
5. Click the **Options** button to open the Export Options dialog box, shown in the following figure. From the **AutoCAD Version** drop-down list, select the version of AutoCAD you'd like to use to open the exported file. In the **Export** area, select the checkbox for each type of entity that you want to include in the exported file. Click **OK** when you're done.
6. Back in the Export Model dialog box, click **Export**, and your file appears in the location where you chose to save it.



Note: Export and Import with materials for better BIM interoperability and workflows using the .dwg format.

Figure 0.3 AutoCAD Export option window

Self – Check 5

Give a clear and precise answer for the following Questions

1) List some major functions to be performed by a computer aided drafting system CADS
2) What outline needs to look for and to resolve any potential conflicts between CAD elements and Sketch Up.?
3) What are the common features of CAD system
4) Identify the basic drawing entities in CAD
5) List down some Editing and Moving commands with their purpose in CAD
6) Mention steps in the import process of CAD Files in the operating system:
7) What are the steps in exporting a sketch up model as a 2D and 3D CAD file

Unit Six: Issue Approved Drawing

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics:

- Save and file Drawings
- Print and evaluate Drawing elements
- Process and form issue of the approved draw and parts lists

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Save and evaluate Drawing elements
- Perform Copying and issuing approved drawings and part lists
- Apply the Procedures in issuing approved drawings and part lists

6. Issue Approved Drawing

6.1. Save and file Drawings (Utility Command)

END Command — this command exits the drawing editor and returns to the main menu and updates the drawing file.

SAVE Command — this command saves the new/modified drawing and returns to the main menu. However, without exiting the drawing editor, if the changes are to be periodically saved, it is desirable to use this command. It protects the work from possible power failures, editing errors, etc.

Command: SAVE

File name <current>: return to save the current file

QUIT Command — this exits the drawing editor without saving the updated version of the current drawing and returns to the main menu. The AutoCAD checks up with the user for one more confirmation to avoid the accidental quitting since all the editing work done would be lost.

6.2. Print and evaluate Drawing elements

For a 2D drawing that has one view, you could create both the model and its annotation entirely in model space, without using a layout.

This method is simple but has several limitations, including

- It is suitable for 2D drawings only
- It does not support multiple views and view-dependent layer settings
- Scaling the annotation and title block requires computation unless you use annotative objects.

Process Overview

If you draw and plot from model space, you must determine and apply a scale factor to annotation objects before you plot. Follow the steps below.

1. Determine the unit of measurement (drawing units) for the drawing.
2. Specify the display style for the drawing unit.
3. Calculate and set the scale for dimensions, annotations, and blocks.

4. Insert the title block in model space, scaled inversely to intended the plotting scale.
5. Draw at full scale (1:1) in model space.
6. Create the dimensions, notes and labels, also scaled inversely to the intended plotting scale.
7. Plot the drawing at the predetermined scale.

Determine the Unit of Measurement: Before you begin drawing in model space, decide what each unit on the screen represents, such as an inch, a millimeter, or a kilometer. For example, if you are drawing a motor part, you might decide that one drawing unit equals a millimeter. If you are drawing a map, you might decide that one unit equals a kilometer.

Specify the Display Style of Drawing Units: Once you have determined a drawing unit for the drawing, you need to specify the style for displaying the drawing unit, which includes the unit type and precision. For example, a value of 14.5 can be displayed as 14.500, 14-1/2, or 1'2-1/2".

Specify the display style of drawing units with the UNITS command. The default drawing unit type is decimal.

Calculate and Set the Scale for Annotations and Blocks: Before you draw, you should set the scale for dimensions, annotations, and blocks in your drawings. Scaling these elements beforehand ensures that they are at the correct size when you plot the final drawing.

You should enter the scale for the following objects:

- **Text.** Set the text height as you create text or by setting a fixed text height in the text style (STYLE).
- **Dimensions.** Set the dimension scale in a dimension style (DIMSTYLE) or with the DIMSCALE system variable.
- **Line types.** Set the scale for non continuous line types with the CELTSCALE and LTSCALE system variables.
- **Hatch patterns.** Set the scale for hatch patterns in the Hatch and Gradient dialog box (HATCH) or with the HPSCALE system variable.
- **Blocks.** Specify the insertion scale for blocks either as you insert them, or set an insertion scale in the Insert dialog box (INSERT) or in Design Center (ADCENTER).

  The system variables used for inserting blocks are INSUNITS, INSUNITSDEFSOURCE, and INSUNITSDEFTARGET. This also applies to the border and title block of the drawing.

You can also use annotative objects if you want to scale annotations automatically.

Determine the Scale Factor for Plotting: To plot your drawing from the Model layout, you calculate the exact scale factor by converting the drawing scale to a ratio of 1:*n*. This ratio compares plotted units to drawing units that represent the actual size of the objects you are drawing.

For example, if you plan to plot at a scale of 1/4 inch = 1 foot, you would calculate the scale factor 48 as follows:

1/4" = 12"

1 = 12 x 4

1 (plotted unit) = 48 (drawing units)

Using the same calculation, the scale factor for 1 centimeter = 1 meter is 100, and the scale factor for 1 inch = 20 feet is 240.

Table 6.1 Sample Scale Ratios for Printing from Model Space

Scale	**Scale factor**	**To plot text size at**	**Set drawing text size to**
1 cm = 1 m	100	3 mm	30 cm
1/8" = 1'-0"	96	1/8"	12"
3/16" = 1'-0"	64	1/8"	8"
1/4" = 1'-0"	48	1/8"	6"
3/8" = 1'-0"	32	1/8"	4"
1/2" = 1'-0"	24	1/8"	3"
3/4" = 1'-0"	16	1/8"	2"
1" = 1'-0"	12	1/8"	1.5"
1 1/2" = 1'-0"	8	1/8"	1.0"

If you are working in metric units, you might have a sheet size of 210 x 297 mm (A4 size) and a scale factor of 20. You calculate grid limits as follows:

210 x 20 = 4200 mm

297 x 20 = 5900 mm

The command to output a drawing is PLOT and you can access it from the Quick Access toolbar. To control whether all of the options in the Plot dialog box are hidden or displayed, click the More Options button.

When all of the options are displayed, there are a lot of settings and options available for your use.

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

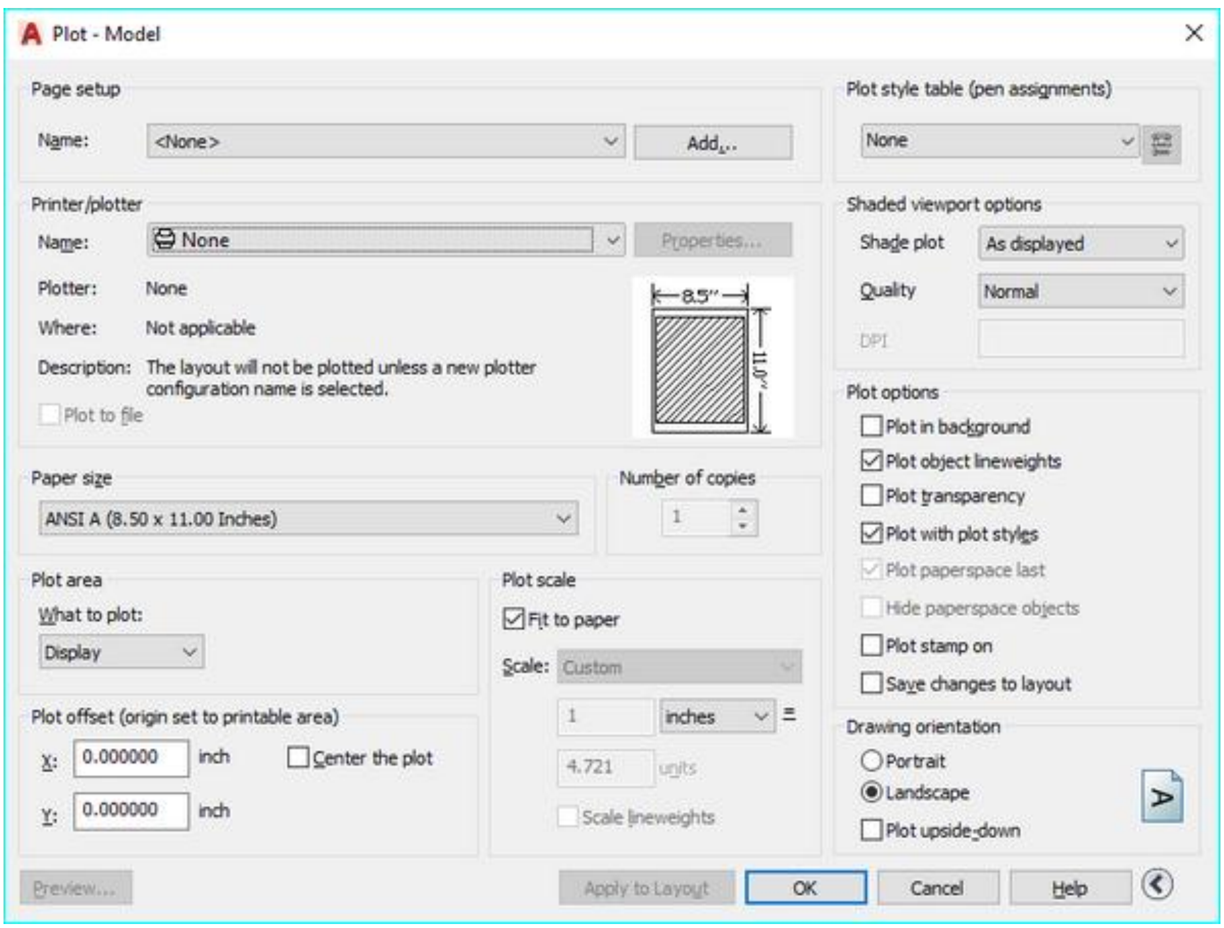


Figure 6.1 Plot window

6.3. Process and form issue of the approved draw and parts list

A complete drawing or a part of a drawing can be given a specific name and then stored as a block. **A** collection of such blocks form a library which is very useful for drawing purpose. The blocks, also referred to as symbols, can be scaled, rotated or mirrored if necessary and inserted into the drawing at the appropriate location e.g. bolt heads to be used at different locations in a drawing. Many suppliers of CAD add-on products offer symbol libraries of standard mechanical components, electronic components, architectural symbols, piping symbols, etc.

The **BLOCK** and **WBLOCK** command groups a number of selected entities together and treats them as a single object i.e. single block. Block commands saves the group of selected entities with block name and is inserted in the file in which it is created, but WBLOCK command saves the group of selected entities as drawing file and can be inserted in any drawing file.

The BLOCKS and WBLOCKS can be scaled, rotated, stretched or mirrored. The explode command separates the entities of BLOCK and WBLOCK.

INSERT command is used to insert blocks and WBLOCKS in current drawing. Blocks will be inserted only in the drawing file in which they created and WBLOCKS will be inserted in any drawing files.

Self check 6

Give a clear and precise answer for the following Questions

1) What are the most common Utility commands and processing steps in CAD drawing
2) Steps in Process overview to determine and apply a scale factor to annotation objects before you plot.
3) What are the Command that used to Process and form Issue the approved drawing and Parts list

Reference

Dr. K.I. Narayana, D. P. (2006). ***Machine Drawing, Third Edition***. New Dehli: New Age International P.LIM, Publishes.

N.D.BHATT, V. P. (2011).***Engineering Darwing*.** Gujarat, India: Charotar Publishing House PVT.LTD.

Reddy, K. V. (2008). ***Textbook of Engineering Drawing, second Edtion.*** India: BS Publication.

P. RADHAKAISHNAN, S. S. (2008). ***Computer Aided Drawing and /CAM/CIM***. New York: New Age International.

A.BOBOULOS, M. (2010). ***CAD - CAM and RAPID PROTOTYPING APPLICATION.*** Ventus Publishing aps.

Participants of This Module (Training Material) Preparation

No	Name	Qualification (Level)	Field of Study	Organization/ Institution	Mobile number	E-mail
1	Zeleke Amdine	A(MSC)	Manufacturing Tech.	Dilla PTC	0910411586	amdinezeleke@gmail.com
2	Eshetu Abate	B(BSc)	Manufacturing Tech.	A.A.T.P.T.C	0910767643	eshetuabate572@gmail.com
3	Tefere Demissie	A(MSC)	Manufacturing Tech.	Wolkite PTC	0912005022	teferad123@gmail.com
4	Abraham Mezemir	A(MSC)	Mechanical Engineering	CEE	0913678721	abrham678721@gmail.com

የሥራ እና ክህሎት ሚኒስቴር
MINISTRY OF LABOR
AND SKILLS