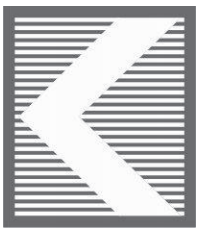
ENGINEERING GRAPHICS

Sharad K. Pradhan
K. K. Jain



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Engineering Graphics

by Sharad K. Pradhan, K.K. Jain

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FOREWORD

Engineering has played a very significant role in the progress and expansion of mankind and society for centuries. Engineering ideas that originated in the Indian subcontinent have had a thoughtful impact on the world.

All India Council for Technical Education (AICTE) had always been at the forefront of assisting Technical students in every possible manner since its inception in 1987. The goal of AICTE has been to promote quality Technical Education and thereby take the industry to a greater heights and ultimately turn our dear motherland India into a Modern Developed Nation. It will not be inept to mention here that Engineers are the backbone of the modern society - better the engineers, better the industry, and better the industry, better the country.

NEP 2020 envisages education in regional languages to all, thereby ensuring that each and every student becomes capable and competent enough and is in a position to contribute towards the national growth and development.

One of the spheres where AICTE had been relentlessly working from last few years was to provide high-quality moderately priced books of International standard prepared in various regional languages to all it's Engineering students. These books are not only prepared keeping in mind it's easy language, real life examples, rich contents and but also the industry needs in this everyday changing world. These books are as per AICTE Model Curriculum of Engineering & Technology – 2018.

Eminent Professors from all over India with great knowledge and experience have written these books for the benefit of academic fraternity. AICTE is confident that these books with their rich contents will help technical students master the subjects with greater ease and quality.

AICTE appreciates the hard work of the original authors, coordinators and the translators for their endeavour in making these Engineering subjects more lucid.

(**Anil D. Sahasrabudhe**)

Acknowledgement

The author(s) are grateful to AICTE for their meticulous planning and execution to publish the technical book for Diploma students.

We sincerely acknowledge the valuable contributions of the reviewer of the book Prof. T K Shrivastava, for making it students' friendly and giving a better shape in an artistic manner.

This book is an outcome of various suggestions of AICTE members, experts and authors who shared their opinion and thoughts to further develop the engineering education in our country.

It is also with great honour that we state that this book is aligned to the AICTE Model Curriculum and in line with the guidelines of National Education Policy (NEP) -2020. Towards promoting education in regional languages, this book is being translated in scheduled Indian regional languages.

Acknowledgements are due to the contributors and different workers in this field whose published books, review articles, papers, photographs, footnotes, references and other valuable information enriched us at the time of writing the book.

Finally, we like to express our sincere thanks to the publishing house, M/s. Khanna Book Publishing Company Private Limited, New Delhi, whose entire team was always ready to cooperate on all the aspects of publishing to make it a wonderful experience.

Sharad K. Pradhan
K.K. Jain

Preface

The book Engineering Graphics is intended to provide comprehensive and schematic knowledge to the students of diploma of engineering and technology courses related to conventional and digital engineering drawing practices. The book covers all the topics mentioned in the AICTE Model curriculum for Diploma in Engineering and Technology 2019 related to 'Engineering Graphics' course in a very systematic and orderly manner. All the topics are explained in a conceptual and step by step manner with proper illustrations.

The chapters matter is written in accordance with 'Engineering Drawing Practices for School and Colleges SP 46:2003', published by Bureau of Indian Standards, Government of India, Third Reprint, October 1998; ISBN:. 81-7061-091-2, Manak Bhavan. Different standard books and available literature on the topic are also referred to make the book complete in all aspects. Care has been taken to include illustrations and figures to make each topic easy to understand. Solved examples and videos are included to support the learning of the students related to difficult to understand topics.

The book comprises of six units. First four units are on conventional engineering drawing practices using manual drawing instruments while the last two chapters are on digital engineering drawing using computer hardware and drafting software (AutoCAD). Apart from essential information a 'Know More' section is also provided in each unit to extend the learning beyond syllabus.

In start of each unit learning outcomes are listed to make the student understand what is expected out of him/her after completing that unit. Further, the unit outcome are mapped with the course outcomes mention in the initial pages of the book so that the student will always in a position to correlate outcomes achieved in each unit with the overall targeted course outcomes.

We are hopeful that the book will motivate the students to learn this difficult and skill based course 'Engineering Graphics' and make them able to draw the entities using both manual drawing instruments and computer aided drafting software. Authors are thankful to all suggestions made by reviewers, translators and stakeholders to make this book effective and beneficial to all concerned. It was a great learning experience for us and still the comments and suggestions are most welcome so that the coming version of the book can be improved for better learning of the course.

Sharad K. Pradhan
K.K. Jain

Outcome Based Education

Outcome based education is based on Outcome based curriculum, Outcome based Teaching-Learning Process and Outcome based Assessment to achieve targeted learning outcomes at each level. As per National Board of Accreditation, after completion of diploma program in engineering and technology the graduate will be able to:

PO1. Basic and Discipline specific knowledge: Apply knowledge of basic mathematics, science and engineering fundamentals and engineering specialization to solve the engineering problems.

PO2. Problem analysis: Identify and analyse well-defined engineering problems using codified standard methods.

PO3. Design/ development of solutions: Design solutions for well-defined technical problems and assist with the design of systems components or processes to meet specified needs.

PO4. Engineering Tools, Experimentation and Testing: Apply modern engineering tools and appropriate technique to conduct standard tests and measurements.

PO5. Engineering practices for society, sustainability and environment: Apply appropriate technology in context of society, sustainability, environment and ethical practices.

PO6. Project Management: Use engineering management principles individually, as a team member or a leader to manage projects and effectively communicate about well-defined engineering activities.

PO7. Life-long learning: Ability to analyse individual needs and engage in updating in the context of Technological changes.

Course Outcomes

After completion of the course the students will be able to:

CO-1. Draw geometrical figures and engineering scales.
CO-2. Use drawing codes, dimensioning, conventions and symbols as per IS SP-46(2003) in engineering drawing.
CO-3. Draw the views of given object using principles of orthographic projection.
CO-4. Draw isometric views of given component directly or from orthographic projections.
CO-5. Draw free hand sketches of given engineering elements, their orthographic view and isometric view.
CO-6. Use computer aided drafting software to draw 2D geometric entities.

Course Outcomes	EXPECTED MAPPING WITH COURSE OUTCOMES (1- Weak Correlation; 2- Medium Correlation; 3- Strong Correlation)						
	PO-1	PO-2	PO-3	PO-4	PO-5	PO-6	PO-7
CO-1	3	-	-	3	-	1	-
CO-2	3	-	-	3	2	1	-
CO-3	3	1	1	3	-	1	2
CO-4	3	1	1	3	-	1	2
CO-5	3	-	1	3	-	-	2
CO-6	3	-	1	3	-	-	2

Abbreviations and Symbols

List of Abbreviations

General terms			
Abbreviations	**Full form**	**Abbreviations**	**Full form**
CO	Course Outcome	TV	Top View
PO	Program Outcome	FV	Front View
UO	Unit Outcome	SV	Side View
LO	Learning Outcome	LSV	Left Side View
BIS	Bureau of Indian Standards	RSV	Right Side View
ISI	Indian Standards Institutions	MS	Microsoft
CAD	Computer Aided Design	CD	Compact Disc
CADr	Computer Aided Drafting	HD	Hard Disk/Drive
V.P.	Vertical reference Plane	PC	Personal Computer
H.P.	Horizontal reference Plane	I/O	Input/Output
RF	Representative fraction	CPU	Central Processing Unit
UCS	User Coordinate System	2D	Two Dimensional
WCS	World Coordinate System	3D	Three Dimensional
QR Code	Quick Response Code		
Unit used			
mm	millimeter	km	kilometers
cm	centimeter	GH	Gega Hertz
m	meters	GB	Gega Bite

List of Symbols

Abbreviations	Full form	Abbreviations	Full form
D, Ø	Diameter	M	Metric
R	Radius	L	Lenght
o	Degree	B, W	Width
⌒	Arc	H, h	Height
ɵ	Theta, Angle	t	Thickness
P	Pitch		

List of Figures

Unit-1 Basic Elements of Drawing

Unit-2 Orthographic Projections

Unit-3 Isometric Projections

Unit-4 Free Hand Sketches of Engineering Elements

Unit-5 Computer Aided Drafting Interface

Unit-6 Computer Aided Drafting

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Guidelines for Teachers

To implement Outcome Based Education (OBE) knowledge level and skill set of the students should be enhanced. Teachers should take a major responsibility for the proper implementation of OBE. Some of the responsibilities (not limited to) for the teachers in OBE system may be as follows:

- Within reasonable constraint, they should manoeuvre time to the best advantage of all students.
- They should assess the students only upon certain defined criterion without considering any other potential ineligibility to discriminate them.
- They should try to grow the learning abilities of the students to a certain level before they leave the institute.
- They should try to ensure that all the students are equipped with the quality knowledge as well as competence after they finish their education.
- They should always encourage the students to develop their ultimate performance capabilities.
- They should facilitate and encourage group work and team work to consolidate newer approach.
- They should follow Blooms taxonomy in every part of the assessment. For assessment of practical work suggestive performance indicators are mentioned at the end of practical in each unit.

Bloom's Taxonomy

Level	Teacher should check	Student should be able to	Possible mode of assessment
Creating	Students ability to create	Design or Create	Mini project
Evaluating	Students ability to justify	Argue or Defend	Assignment
Analysing	Students ability to distinguish	Differentiate or Distinguish	Project/Lab Methodology
Applying	Students ability to use information	Operate or Demonstrate	Technical Presentation/ Demonstration
Understanding	Students ability to explain the ideas	Explain or Classify	Presentation/Seminar
Remembering	Students ability to recall (or remember)	Define or Recall	Quiz

Guidelines for Students

Students should take equal responsibility for implementing the OBE. Some of the responsibilities (not limited to) for the students in OBE system are as follows:

- Students should be well aware of each UO before the start of a unit in each and every course.
- Students should be well aware of each CO before the start of the course.
- Students should be well aware of each PO before the start of the programme.
- Students should think critically and reasonably with proper reflection and action.
- Learning of the students should be connected and integrated with practical and real life consequences.
- Students should be well aware of their competency at every level of OBE.

Contents

1 Basic Elements of Drawing

UNIT SPECIFICS

This unit elaborately discusses the following topics:

- Drawing Instruments and supporting materials
- Convention of lines and their applications
- Engineering scale
- Dimensioning techniques
- Geometrical and tangency constructions

The understanding of these topics will be developed after reading the contents, completing the solved problems, activities, performing exercises and viewing the ICT and web resources mentioned in this unit.

At the end of the unit, summary is provided to recapitulate the topics covered and applications are mentioned so that the learner can correlate the presented knowledge with real life and industrial situations. Few activities are mentioned to create inquisitiveness and curiosity in the learner. Subjective and objective questions are provided for the reinforcement of the knowledge and a list of references and suggested readings is also provided for further learning. Video resources along with QR codes are mentioned for getting more information on various topics of interest which can be surfed or scanned through mobile phone for viewing. Practical with details are provided to horn the drawing and drafting skills.

RATIONALE

Engineering drawing is a language of engineers that enables them to visualize the ideas converting to realization by drawing them. It is a graphical language that communicates all the information about an object from an engineer, who designed it, to an artisan who will make it. In engineering drawing, the figures are drawn by using straight and curved lines to represent the surfaces, edges and centres of objects. Symbols, dimensional values and text are added to these lines so that they collectively make the complete drawing description. The drafting is the preparation of these drawing manually or with the help of drawing instruments.

The aim of this unit is to introduce you with some of the most common drawing equipment that you need for your drawing work. Besides owing a right type of drawing equipment, it is also important to understand the correct techniques of using and handling the instruments, line types,

dimensioning methods and to prepare drawings to suitable engineering scales as recommended by the BIS.

The Bureau of Indian Standards (BIS) is the national standards body of India working under the support of Ministry of Consumer Affairs, Food and Public Distribution, Government of India. It is established by the Bureau of Indian Standards Act,1986 which came into effect on 23 December 1986. It was formerly known as the Indian Standards Institutions(ISI). Bureau of Indian Standards "SP 46:2003 Engineering Drawing Practices for School and Colleges" provides you a set of rules and codes for engineering drawing. BIS has developed this standard code of practice to maintain uniform standard of engineering drawing throughout the country.

PRE-REQUISITES

Before reading this unit the student is advised to revisit following:

- Maths: Coordinate and Plane geometry

UNIT OUTCOMES

After reading the contents, completing the solved problems, activities, exercises and viewing the ICT & web resources mentioned in this unit, the students are expected to:

U1-O1: Use various drafting equipment and instruments for manual drafting

U1-O2: Use different types of lines in technical drawing as per BIS Codes

U1-O3: Use concept of engineering scale to solve engineering drawing problems

U1-O4: Apply dimensions to technical drawing in accordance with BIS standards

U1-O5: Construct the various geometrical shapes

Unit-1 Outcome	EXPECTED MAPPING WITH COURSE OUTCOMES (*1- Weak Correlation; 2- Medium Correlation; 3- Strong Correlation*)					
	CO-1	CO-2	CO-3	CO-4	CO-5	CO-6
U1-O1	3	2	3	3	2	-
U1-O2	3	3	3	2	3	2
U1-O3	3	1	2	2	2	1
U1-O4	2	3	3	2	3	2
U1-O5	3	2	2	2	2	2

1.1 DRAWING INSTRUMENTS AND SUPPORTING MATERIALS

1.1.1 Introduction

Drawing is a graphic language of engineers and technicians. Therefore, it should be taught as any other

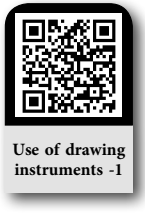
language. Every language has its own rules. Similarly, in case of engineering drawing, it is necessary to use instruments correctly in order to get the required degree of accuracy.

The aim of this section is to introduce you with some of the most common drawing equipment that you need for your drawing work. Besides owing a right type of drawing equipment, it is also important to know the correct techniques of using and handling the instruments. The Bureau of Indian Standards (BIS) is the national standards body of India provides you a set of rules and standard code of practice to maintain the uniform standard of engineering drawing throughout the country.

1.1.2 Drawing Equipment

Engineering drawing is a graphical language wherein information is recorded with the help of drawing equipment and instruments. A student has to be conversant with all drawing instruments, to enable him to select and use suitable instruments as per the situations. It is also important to follow certain rules and a code of practice for making a presentable and accurate drawing. Accuracy of drawings largely depend on the quality of instruments. Following is the list of common drawing equipment and instruments for preparing drawings.

1. Drawing board
2. T-square
3. Mini-drafter
4. Pairs of set squares/triangles
5. Drawing instrument box containing: Divider; Compass with interchangeable pencil and pen legs; Bow pencil; Bow pen; Bow divider; Lengthening bar; Inking pen; Box of leads; Screw driver
6. Protractor (circular or semi-circular)
7. Drawing pencil
8. Drawing paper
9. Pencil pointer
10. Drawing clips
11. Dusting cloth
12. Rubber/Eraser
13. Erasing shield
14. Roll-n-draw


Use of drawing instruments -1

Drawing board

Drawing Board is a flat, rectangular surface used for sketching, drawing, writing, etc. The surface of the board must be smooth and soft so that pencils can easily draw lines. The drawing board as shown in Fig.1.1 is required to prepared from best seasoned, good quality wood (white pine, masonite etc.) and are fitted with battens at back with brass round slots for expansion and contraction. The ebony edges are fitted on one edge and are saw cut to allow expansion and contraction. The left hand edge of the board is called the working edge because the T-square head slides against it, as shown in Fig.1.2. It is generally placed on drawing table at an angle of 20^0. The standard sizes of the drawing boards as recommended by BIS are shown in Table 1.1.

The length of drawing boards varies from 1500 mm - 500 mm, width ranges from

1000 mm – 350 mm and thickness ranges from 25 mm – 15 mm. The drawing boards designated by D0 and D1 are used for drawing offices where as D2 is recommended for the engineering students.

Table 1.1 Standard sizes of drawing boards

S. No.	Designation	Dimension in mm. (Length x Width x Thickness)	Recommended for use with drawing sheet
1	D0	1500 x 1000 x 25	A0
2	D1	1000 x 700 x 25	A1
3	D2	700 x 500 x 15	A2
4	D3	500 x 350 x 15	A3

T-square

The T-square shown in Fig.1.2 is composed of a long strip, called the blade, fastened rigidly at right angles to a shorter piece called the head. The upper edge of the blade and the inner edge of the head are working edges and must be straight. The blade should have transparent plastic edges and should be free of nicks along the working edge. Transparent edges are recommended, as they permit the draftsman to see the drawing in the vicinity of the lines being drawn. T-square are available from 600 mm. size to 1500 mm and above.

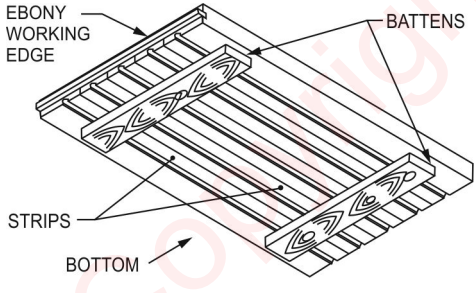


Fig.1.1: Drawing board

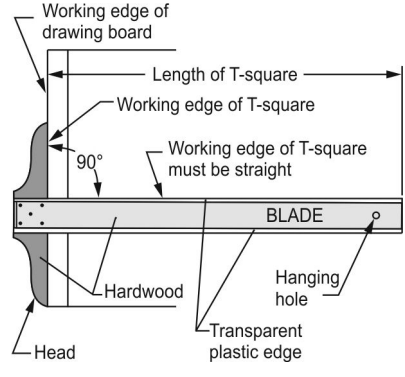


Fig.1.2: T-square

Mini-drafter

The mini-drafter shown in Fig.1.3 is a device that replaces the T-square, triangles, scales and protractor. It consists of a fixed clamp (use to fix the device with the drawing board) and two sets of parallelogram of bars connected by pivot plate. The free end of the parallelogram of bars has a set of graduated acrylic or metallic scales held at 90^0 and attached to a protractor head with an index plate and a locking screw knob. The mini drafter has

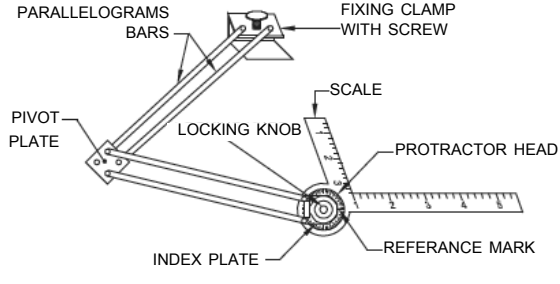


Fig 1.3: Mini-drafter

the mechanism which keeps the two perpendicular arms along with the graduated scale can swivel to any angle and can move to any location in the drawing sheet with the help of parallelogram of bars. Thus, a mini-drafter is used to draw horizontal, vertical or inclined parallel lines of desired length anywhere on the drawing sheet with considerable ease and saving time. The main purpose of the mini-drafter is to speed up the drawing work.

Set-squares

The set-squares used are of two types: 45° triangle and 30°- 60° triangle. They are also called triangles. The 45° set-square shown in Fig.1.4(a) is a right angled in which acute angles measure 45°. The 30°- 60° set-square shown in Fig.1.4(b) is a right angled in which acute angles measure 30° and 60°. Set- squares are made of transparent plastic so that lines of the drawing can be seen through them. All straight lines except the horizontal lines can be drawn with the help of set square. Vertical or upright lines are drawn with the T-square and the set square.

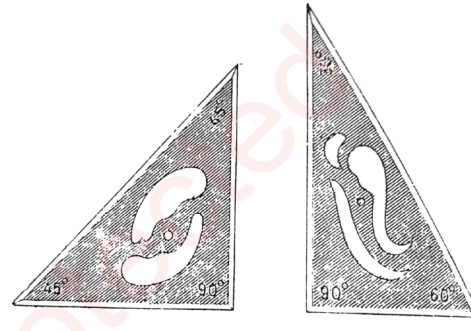


Fig.1.4: (a) and (b) Set-square

Drawing instrument box

In technical drawing, accuracy, neatness and speed are essential. These objectives are not likely to be obtained with cheap or inferior drawing instruments. Therefore, student should get an advice from his drafting instructor or an experienced drafter. Nickel, Silver or chromium plated instruments are costly. The cost of instruments may be justified by its good quality of work. A typical drawing instrument set is shown in Fig.1.5.

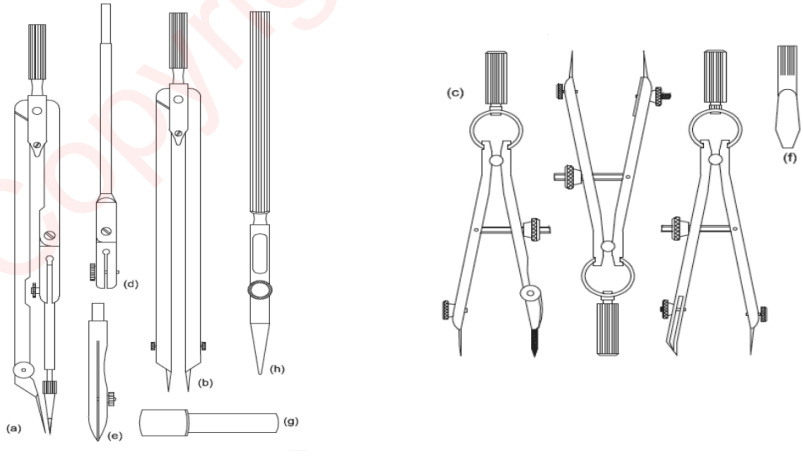
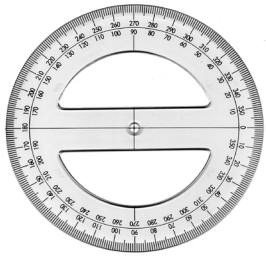


Fig.1.5: Drawing instrument set

Protractors

Protractors are used for measuring various angles. They are available in different shapes-circular or semi-circular. They are made of nickel silver, plastic or sheet metal. Fig.1.6 (a & b) shows a transparent

circular and semi-circular protractor reading up to half a degree.



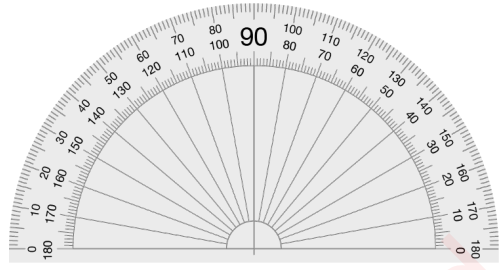


Fig: 1.6: (a) Circular and **Fig 1.6:** (b) Semi-circular protractor

Pencils

The basic instrument is the graphite lead pencil. These are available in various hardness and grades. Good quality pencils should always be used as they contribute to a very large extent towards accuracy and appearance of a drawing work. Generally, pencils are available in eighteen grades as follows: -

Type	Designation	Purpose
Hard grade	9H,8H,7H,6H,5H,4H	Fine line work
Medium grade	i) 3H, 2H, H ii) F, HB, B	Line work, lettering and arrowheads
Soft grade	2B, 3B, 4B, 5B, 6B, 7B	Mechanical drafting, Architecture and art work

For good and accurate work, proper mending of pencil and sharpening of lead is very essential. The lead of 3H and 2H pencil should be sharpened to a chisel point as shown in Fig.1.7. This enables to draw long thin lines of uniform thickness. The lead of HB pencil should be sharpened to a conical point as shown in Fig.1.8, which is used for lettering.

For sharpening a pencil, a small piece of sand-paper of zero grade, pasted upon a piece of wood, will be very useful for keeping the point in good condition. Pull and roll the pencil point on a sand-paper block shown in Fig.1.9 to sharpen the pencil point.

Fig.1.7: Chisel point pencil

Fig.1.8: Conical point pencil

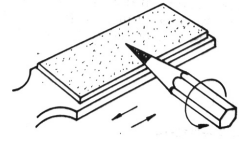


Fig.1.9: Sharpen the pencil point

Drawing paper

Various grades and qualities of drawing paper are available now-a-days. Cartridge or machine mode paper is largely used for drawing purpose. Drawing paper has two surfaces viz., rough and smooth.

The smooth surface is the proper side to be used. The standard sizes of drawing papers as recommended by BIS and its special publications SP 46-2003 are shown in Table 1.2.

Table 1.2: Standard sizes of drawing sheets (Main ISO-A Series - SP46-2003)

Sheet Designation	Trimmed Size (mm)	Untrimmed Size (mm)
A0	841 x1189	880 x 1230
A1	594 x 841	625 x 880
A2	420 x 594	450 x 625
A3	297 x 420	330 x 450
A4	210 x 297	240 x 330
A5	148 x 210	165 x 240

French curves

French curves serve as guide edges for irregular curves. They are available in various shapes and sizes. Fig.1.10 shows the common forms of French curves. The good ones are generally made of highly transparent plastic. Adjustable curves are also available. It has the same purpose as that of the French curve. The curve shown in Fig.1.11 consist of a core of lead, enclosed by a coil spring attached to a flexible strip. This can be bent for any desired curve. The curve edge may be set practically up to the line to be drawn, assuring great accuracy either with pencil or pen.

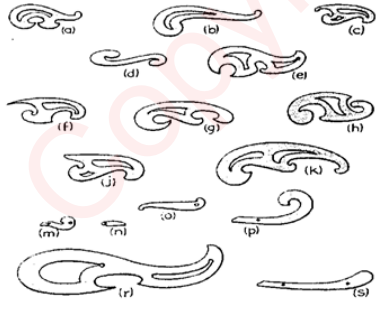


Fig 1.10: French or irregular curves

Fig 1.11: Flexible curve

Drawing clamps

If the drawing paper is of the same size as that of the drawing board, all the four drawing clamps can be used, otherwise, for smaller drawing papers drawing pins or cello-tape can be used. The use of drawing pins and cello-tape should be avoided as far as possible. Constant use of pins spoils the boards as well as drawing sheets.

Dusting cloth

A complete drawing sheet would look more attractive if it is not dirty. A clean piece of flannel or towel can be used as a duster for cleaning the sheet after the lines have been erased by rubber.

Rubber eraser

A soft rubber of good quality which will not spoil the surface of the sheet should be used. Erasing should be done very lightly so as not to leave any impression on the drawing sheet.

Erasing shields

Various sizes of metallic and plastic erasing shields are available. It consists of various holders and lines through which small portions of work can be erased without damage to other parts.

Roll-n-draw

It consists of a roller, protector and a graduated scale shown in Fig.1.12. It is a handy ruling drawing instrument used to draw a set of lines in horizontal, vertical and inclined direction. It is also used for drawing angles and circles.

Fig.1.12: Roll-n-Draw

Use of drawing instruments-2

1.1.3 Use of Various Drawing Equipment and Instruments

Use of T-square and set–square

1. Drawing horizontal lines

   Refer Fig.1.13 to draw a horizontal line, press the head of T-square firmly against the working edge of the board with the left hand. Then slide the left hand to the position shown at (A) so as to press the blade tightly against the paper. Place the pencil in the direction of the line at an angle of approximately 60° with the paper and draw the line from left to right.

2. Drawing vertical lines

   Use either the 45° triangle or the 30°-60° triangles to draw vertical lines shown in Fig.1.13. Place the triangle on the T-square with the vertical edge on the left shown in (B). With the left hand, press the head of the T-square against the board. Now slide the hand to the position shown where it holds both the T-square and the triangle firmly in position. Then draw the line upward, rotating pencil slowly between the thumb and forefinger.

3. Drawing parallel lines

   Refer Fig.1.14 to draw a line parallel to a given line, move the triangle and T-square as a unit until the hypotenuse of the triangle lines up with the given line. Hold the T-square firmly in position and slide the triangle away from the line and draw the required line along the hypotenuse.

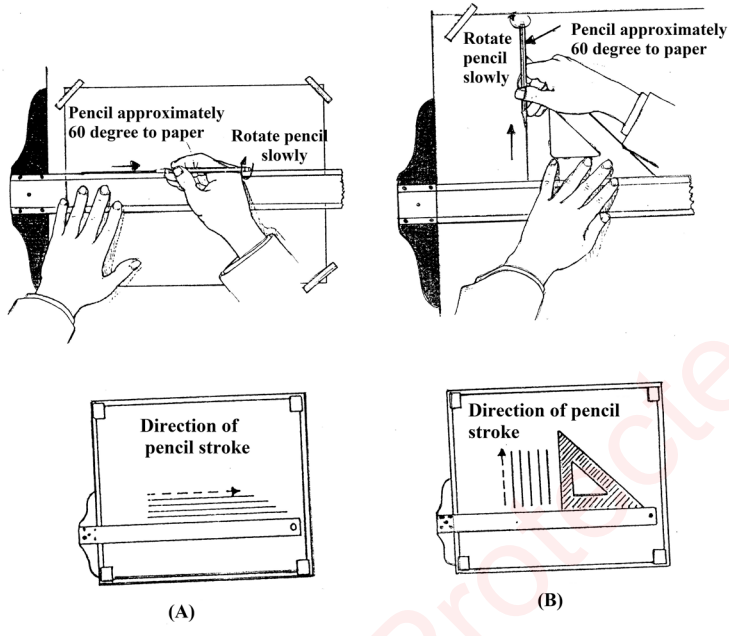


Fig 1.13: Drawing horizontal and vertical lines

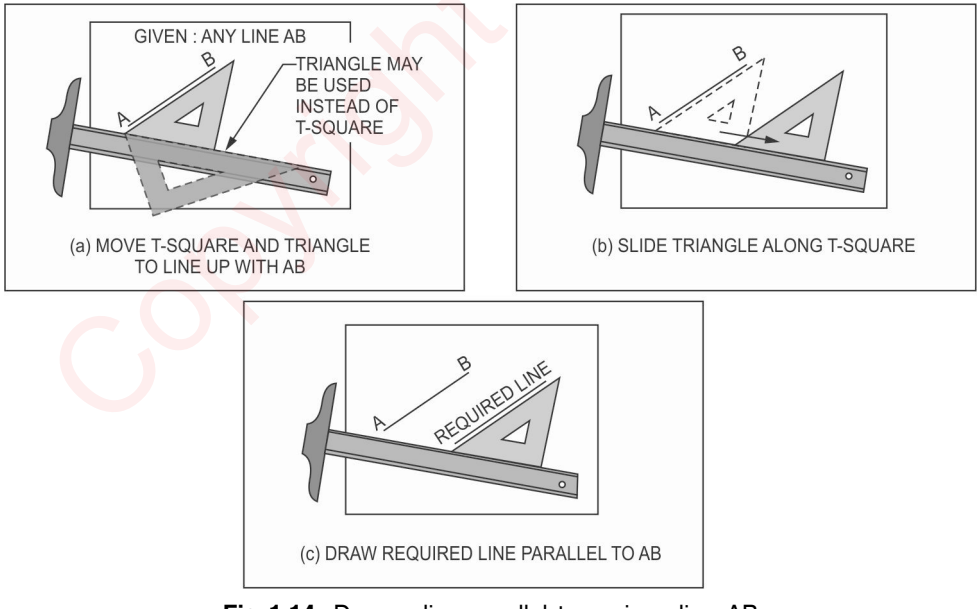


Fig 1.14: Draw a line parallel to a given line AB

4. Drawing perpendicular line

To draw a line perpendicular to a given line, move the T-square and triangle as shown in Fig.1.15.

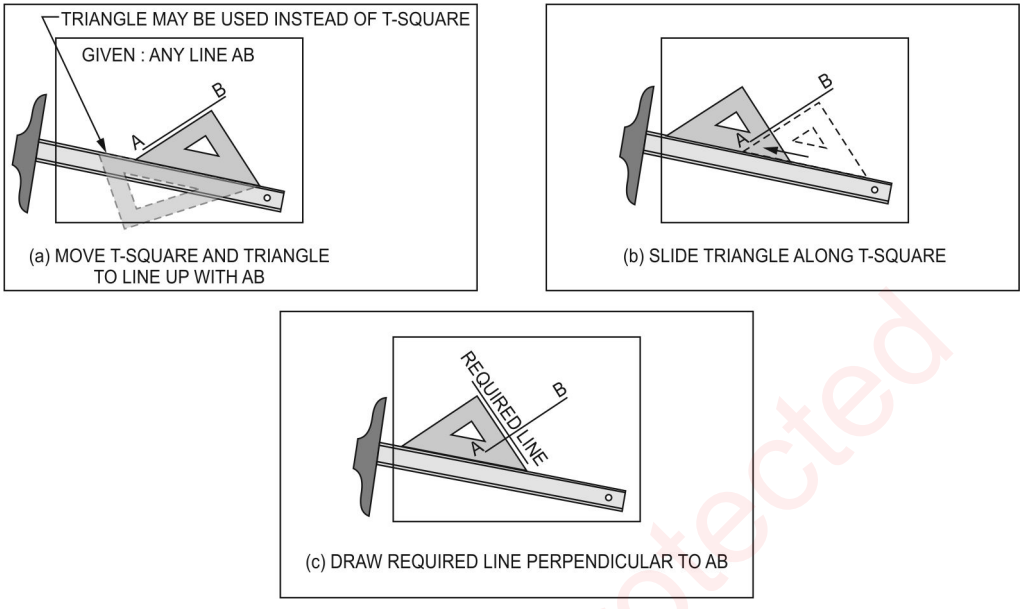


Fig:1.15: Draw a line perpendicular to a given line AB (method-1)

5. Drawing lines at 30°, 60° or 45°

To draw a line making 60° from the horizontal, arrange the triangle as shown in Fig.1.16. Angles of 30° and 45° may be drawn in a similar manner.

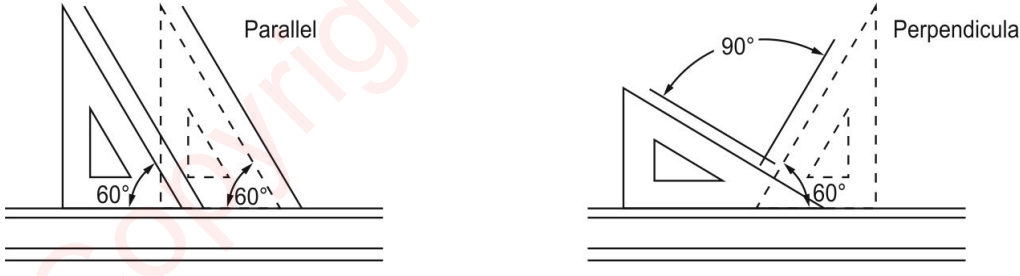
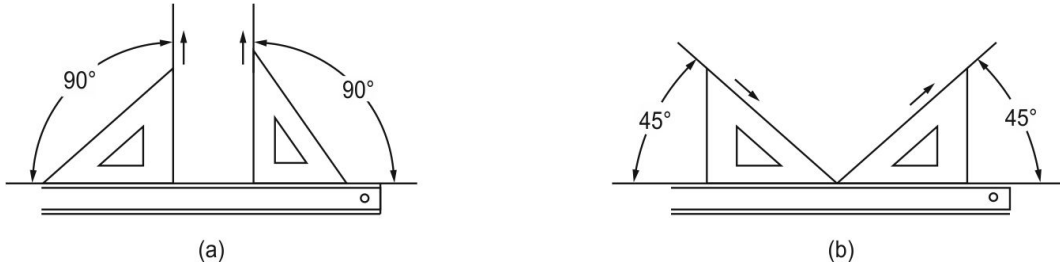


Fig:1.16: Draw a line perpendicular to a given line AB (method-2)

6. Drawing slanting lines

Fig.1.17 shows the use of 45°, and 30°, 60° set-squares in drawing slanting lines with T-square.



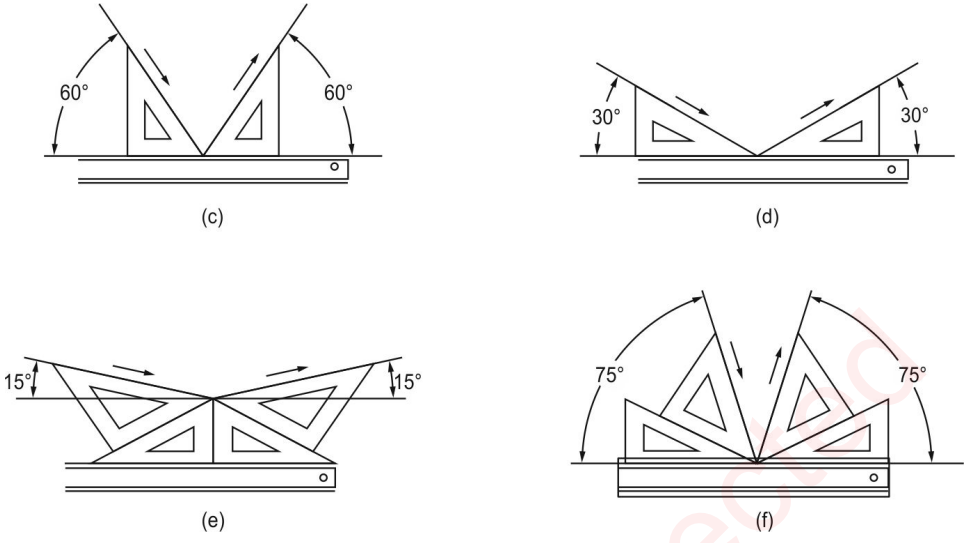


Fig.1.17: Drawing slanting lines with T-squares and set squares

Use of compass

Circles and arcs of circles are drawn with the help of the compass as shown in Fig.1.18. Compasses having the pencil and inking attachments are used for drawing circles of approximately 12 cm diameter. Their legs are kept straight. But if larger circles are to be drawn, bend the legs of the compass as shown in Fig.1.19, so that they will stand approximately perpendicular to the paper. Bending of the legs is essential in drawing ink circles as otherwise, both sides of the pen attachment will not touch the paper equally.

The following steps may be followed in using a compass for drawing a penciled circle:-

Step-1. Draw the centre lines of the required circle and set off the required radius on any of them.

Step-2. Put the needle point at the exact intersection of the centre lines.

Step-3. Adjust the opening of the legs of the compass to the required radius already marked on the centre line.

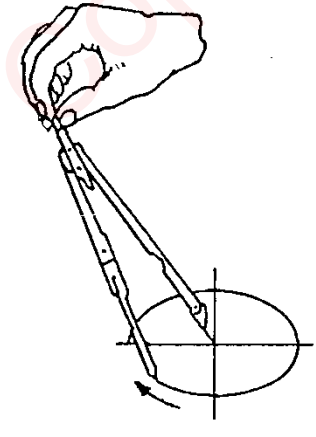


Fig.1.18: Using the compass

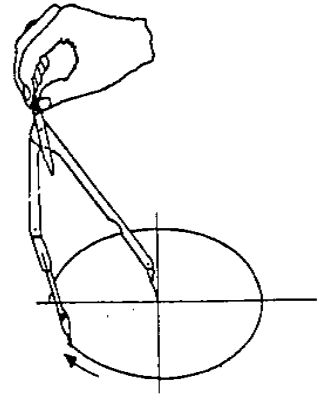


Fig.1.19: Bending the leg of the compass

Step-4. Give a slight inclination to the compass in the forward direction and draw the circle clock-wise while rotating the handle between the thumb and the first two fingers of the right hand (see Fig.1.18). In order to darken the penciled circle, repeat the same process in the clock-wise direction only.

Use of dividers

The dividers are similar to the compass in construction and are made in square, flat and round cross-section. Many dividers are made with a spring and thumb-screw in one leg, so that minute adjustments in the setting can be made by turning the small thumb-screw.

The dividers can be put to the following uses: -

1. Dividing distances in to a number of equal parts.
2. Transferring distances or far setting off series of equal distance.

The dividers are used for spaces of approximately 25mm or more. For less than 25mm spaces use the bow dividers. Never use the large dividers for small spaces when the bow dividers can be used.

Use of french curve

To draw a mechanical line over the freehand line with the aid of the irregular curve, it is only necessary to match the various segments of the irregular curve with successive portions of the freehand curve and to draw the line with pencil or ruling pen along the edge of the curve. It is very important that the irregular curve match the curve to be drawn for some distance at each and beyond the segment to be drawn for any one setting of the curve, as shown in Fig.1.20.

For symmetrical curves, such as an ellipse, use the same segment of irregular curve in two or more opposite places {Refer Fig.1.21(a)}. The irregular curve is matched to the curve and the line drawn from 1 to 2. Light pencil dashes are then drawn directly on the irregular curve at these points. Fig.1.21 (b) shows that the irregular curve is turned over and matched so that the line may be drawn from 2 to 1. In similar manner, the same segments is used again as shown in Fig.1.21(c) and 1.21(d). The ellipse is completed by filling in the gaps at the ends by using the irregular curve or French curve.

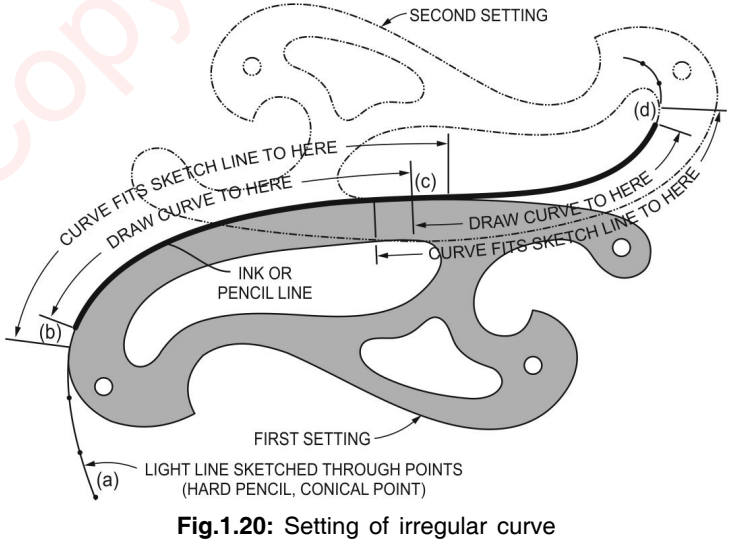


Fig.1.20: Setting of irregular curve

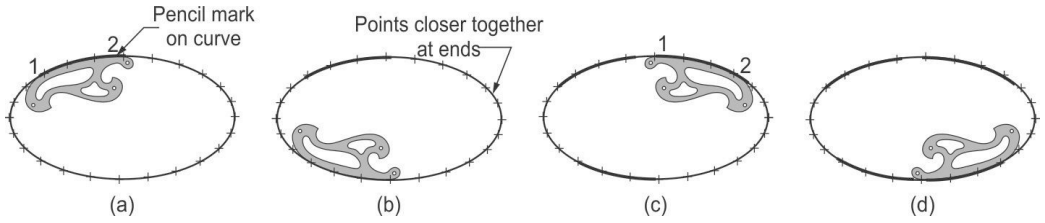


Fig.1.21: Drawing symmetrical figures by using irregular curve

1.1.4 To layout a drawing sheet

The Bureau of Indian Standards issued IS 15093:2002 and SP 46:2003 which specify the exact size and location for each item found on the drawing sheet. It is recommended that standard sizes and formats should be followed to improve readability, filing and handling of drawing sheets. Fig.1.22 shows a typical layout on drawing sheets. The drawing work should be done in rectangular working space. For this purpose, border lines are drawn. The recommended minimum boarder width of 20 mm for sizes A0 and A1 and a 10 mm minimum width for sizes A2, A3 and A4.

The lay-out must also provide a little block which should be located at the bottom right hand corner of the sheet. The title block contains the following information:

1. The title of the drawing,
2. Drawing number,
3. Name of the organization
4. Scale,
5. Date of drawing,
6. Initial of technical staff who prepared, verified and approved the drawing, etc.

Drawing elements

The size of the title block recommended is 65 mm x 170 mm and it is uniform for all sizes of drawing sheets. However, the drawing sheet lay-out with the contents of the little block may be modified to suit classroom training requirements.

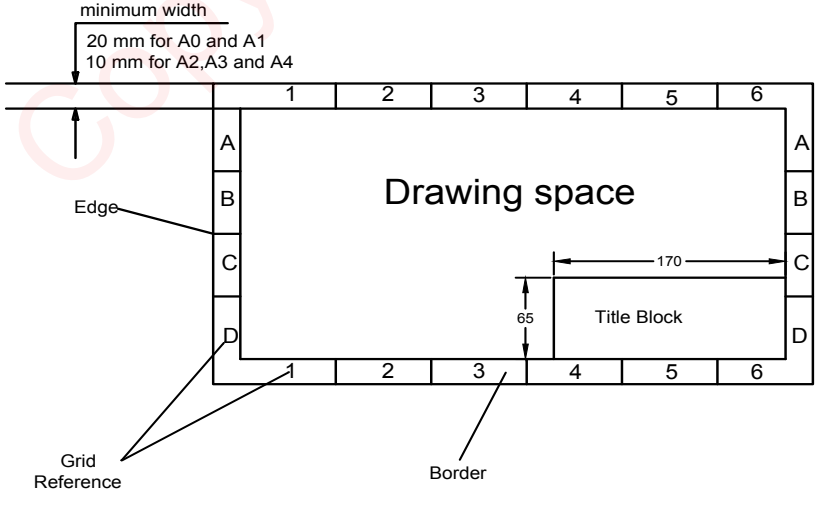


Fig.1.22: Layout of a drawing sheet

1.2 CONVENTION OF LINES AND THEIR APPLICATIONS

1.2.1 Need of Line and Line Work

Drawing is made up of lines. Therefore, use of correct type of line is important to describe an object correctly, so as to communicate the shape and size of the object clearly and completely to the reader. In engineering drawing, the figures are drawn by using different types of lines. Synonymously, the lines may be considered as alphabets of drawing and therefore each type of line, if used properly, speaks itself about its purpose for which it is being employed. In this section emphasis given to various basic types of lines, applications and their representation in engineering drawing as recommended by the latest BIS and its special publications SP 46:2003. The basic types of lines and their applications in engineering drawing given in SP 46:2003 are summarised in Table 1.3. For complete information, the concerned BIS code may be referred.

1.2.2 Types of Lines

There are 15 basic types of lines used in engineering practice which are identified by their designation numbers as per BIS specifications. These lines are discussed below:

Construction lines (Type 1.1: Continuous narrow Lines)

These lines are made for the construction of figures. They are shown in geometrical drawings. They should be drawn very thin and faint and hardly visible in the finished drawing.

Extension or projection lines (Type 1.1: Continuous narrow Lines)

These lines are continuous thin lines. They extend by about 3mm beyond the dimension lines.

Dimension lines (Type 1.1: Continuous narrow Lines)

The dimension line represents the dimensions of the feature in the drawing and is generally a continuous line drawn parallel to the edge. They are terminated at the outer ends by pointed arrowheads touching the outlines, extension lines or centre lines.

Hatching lines/Section lines (Type 1.1: Continuous narrow Lines)

These are used to indicate a surface, which is cut. A series of parallel lines is marked in the cut surface in an inclined fashion generally at an angle of 45^0 to the main outline of the section. They are uniformly spaced about 1mm to 2mm apart spacing among them.

Leader lines (Type 1.1: Continuous narrow Lines)

The leader lines are used to pin point a location or a feature in a drawing. They are drawn usually at an angle to the representation and touch the feature with an indication of an arrow.

Short centre lines (Type 1.1: Continuous narrow Lines)

These are used to mark the Centre of small circles represented in the drawings and are marked with a + (plus) sign.

Outlines or visible line (Type 1.2: Continuous wide Lines)

Lines drawn to represent visible edges and surface boundaries of objects are called outlines or principal lines. These are Continuous wide or thick lines.

Table 1.3: Types of lines and their applications in engineering drawing

Desig-nation No.	Broad category of line	Commonly used lines alongwith designation number and their representation	General Application
01	Continuous line	Continuous narrow line (1.1) Thickness = 0.13 mm	Imaginary lines of intersection, Construction, Projection, Short center, Extension, Dimension, Leader, Hatching, Outlines of revolved sections ,Root of screw threads, Interpretation lines of tapered features
		Continuous wide line (1.2) Maximum width or thickness = 0.25 mm	Visible outlines of parts in cut and section, when hatching is used, Visible outlines of parts in view, Reference lines in projection, ground lines, Arrow lines for making of views, cuts and sections, Border lines, title block etc.
		Continuous extra wide line (1.3) Maximum width or thickness = 0.5 mm 	Visible outlines, Visible outlines of parts in cut and section, when hatching is not used, Lines of special importance
		Continuous line, free hand narrow (1.4) 	Short break lines, Limits of partial or interrupted views
		Continuous lines, Zig zag hand narrow 	Long break lines, Limits of partial or interrupted views
02	Dshed lines	Dashed narrow line (2.1) 	Hidden lines or edges,Hidden outlines
		Dashed narrow line (2.2)	Hidden lines or edges

03	Dashed spaced line		
04	Long dashed dotted line	Long dashed dotted narrow line (Chain narrow line) (4.1)	Center lines, Lines of symmetry, Pitch circle of holes, Pitch circle of gears, Indication of cutting planes
		Long dashed dotted wide line (Chain wide line) (4.2)	Cutting plane representations at the ends and at locations of change of direction, Surface treatment indication locations
05	Long dashed double dotted line	Long dashed double dotted narrow line (5.1)	Centroidal lines , Locus lines Alternate positions of movable parts, Extreme positions of movable parts, Outlines of adjacent pats

Short break lines (Type 1.4: Continuous free hand narrow lines)

These lines are continuous, thin and wavy. They are drawn freehand and are used to show a short break or irregular boundaries.

Long break lines (Type 1.5: Continuous zig zag narrow lines)

These lines are used to indicate a break in lengthy objects such as bars and channels that are of uniform cross sections (regular or irregular) and are marked in zigzag wavy pattern with freehand.

Hidden lines or edge (Type 2.1: Dashed narrow Lines)

These are used to mark the invisible or hidden edge or features of an object and also called as dotted lines. They are made up short dashes of approximately equal lengths of 2 mm with a gap of 1 mm.

Centre lines (Type 4.1: Long Dashed dotted narrow Lines)

These are thin lines made up of long and short dashes, alternately spaced and consistent in length, beginning and ending with a long dash. These are drawn to represent the axis of the cylindrical, conical and spherical feature, including the holes in these bodies and are marked protruding slightly beyond the outlines of the object. These lines are also used to show the centres of the circles and arcs.

Cutting - plane lines (Type 4.2: Long Dashed dotted wide Lines)

The location of a cutting plane is shown by this line. It is a long, thin, chain line at ends only.

Chain thick lines (Type 4.2: long dashed dotted wide line)

These lines are used to indicate special treatment on the surface.

Long double-dots line (Type 5.1: long dashed double dotted narrow line)

These line representations are made with alternate dashes of 9-12 mm long, with two dots in between and spaced at 1mm apart among them from the ends of the dashes.

The application of various types of lines are shown in Fig.1.23(a) and (b).

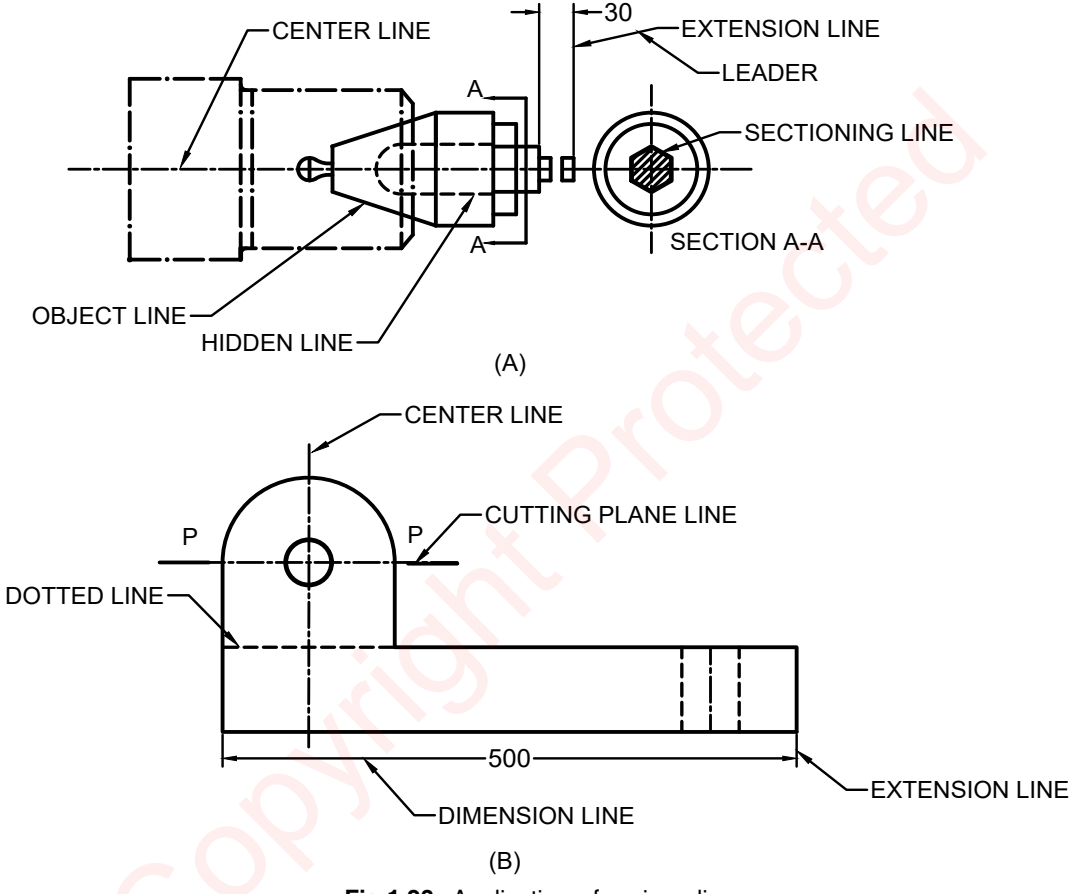


Fig.1.23: Application of various lines

SOLVED PROBLEMS (WITH INCREASING DIFFICULTY LEVEL)

Example 1.1 *List different types of line used in following drawing marked as A to F and check your response with the solution provided.*

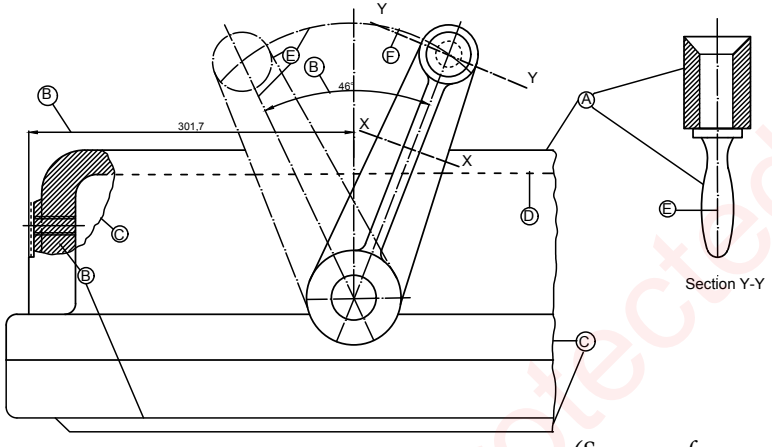
Solution

Continuous wide line-A: visible edges, visible outlines, main representations, maps, flow charts.

Continuous narrow line-B: dimension lines, extension lines, leader lines, reference lines, short centre lines, projection lines, hatching, construction lines, guide lines, outlines of revolved sections, imaginary lines of intersection.

Continuous narrow freehand line-C: Preferably manually represented termination of partial or interrupted views, cuts and sections, if the limit is not a line of symmetry or a center line.

Dashed narrow line-D: hidden edges, hidden outlines.
Long-dashed dotted narrow-E: center lines, axes, lines of symmetry, cutting planes.
Long-dashed dotted wide line-F: Cutting planes at the ends and changes of direction outlines of visible parts situated m front of cutting plane.



(Source reference [3])

1.3 ENGINEERING SCALE

1.3.1 Introduction

Whenever you start making drawing of an object, you intuitively, either reduce it, or enlarge it, or keep it of true size, so that it occupies allotted space on paper. If you make drawing of a building on paper, then you have to reduce dimensions proportionately, so that the drawing comes inside the paper. If you draw circuit of computer processor, naturally you have to draw a magnified drawing, so that minute circuits become visible & details can be labelled. And, if the size of object is equal to the size available on paper, then there is no need of reducing or enlarging the drawing, you can make true-size drawing.

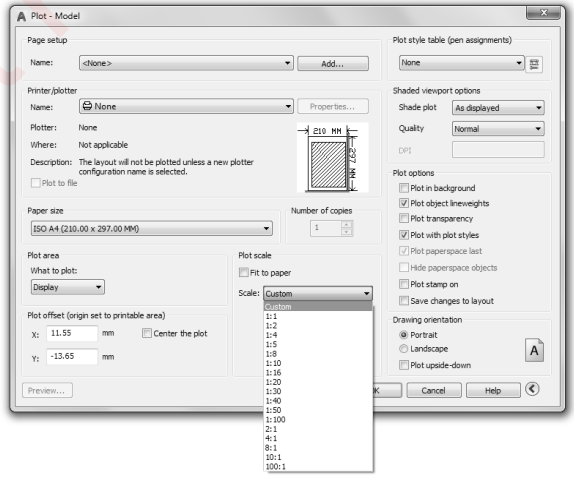


Fig.1.24: AutoCAD plot dialogue box showing options for plot scale of printed drawing

Now a day, almost all technical drawings are drafted with the help of some drafting software, for example Auto CAD. Since screen size of Auto CAD is infinite, you can always make true size drawing in Auto CAD. But when you try to take printout of a drawing made in Auto CAD, then before printing on paper, in the plot model dialogue box (Fig.1.24), the scale or the representative fraction of the drawing has to be specified or if you tick the fit to paper box (Fig.1.24) the scale factor is decided automatically by the software Auto CAD. Hence, it becomes essential to first develop understanding of the concept of representative fraction.

1.3.2 Representative Fraction or Scale Factor of a Drawing

Representative fraction of a drawing, is a numerical fraction value. Value of representative fraction indicates, how much times, the linear distance in the drawing is reduced or enlarged, compared to actual distance. If representative fraction of a drawing is (1/4) then, it means that linear distances on drawing are reduced 4 times, compared to actual distances. That is an actual line of 4 cm length, is drawn as 1 cm long line on the drawing. And if representative fraction of a drawing is (4/1) , then it means that linear distances on drawing are magnified by 4 times, compared to actual distances. That is a line of actual length 1 cm, is drawn as 4 cm long line in the drawing.

Mathematical formula for representative fraction of a drawing is

$$\textit{Representative fraction of a drawing} = \frac{\textit{Distance between two points A and B on drawing}}{\textit{Actual distance between the points A and B}}$$

From above formula, it is clear that representative fraction is a unitless and dimensionless value. For reduced drawings, its value is less than 1, for full size drawings, its value is equal to 1, & for enlarged drawings, it value is greater than 1.

Other variations of the formula for the representative fraction (R.F.) of a drawing are,

$$R.F. = \sqrt{\frac{\textit{Area of drawing}}{\textit{Actual area}}}$$

And in case of miniature 3D model of a big structure or a machine component, formula for R.F. becomes,

$$R.F. = \left(\frac{\textit{volume of model}}{\textit{volume of actual object}}\right)^{\left(\frac{1}{3}\right)}$$

Reduced drawing

Drawing, drawn with value of representative fraction less than 1, are called reduced drawings. When object size is larger than drawing sheet size, then you have to make reduced drawings.

Enlarged drawing

Drawing, drawn with value of representative fraction more than 1, are called enlarged drawings. When object size is very small compared to drawing sheet size, then you have to make magnified drawings, so that minute details become clearly visible.

True size drawing

Drawing, drawn with value of representative fraction equal to 1, are called true size drawings.

1.3.3 How to decide the value of representative fraction or scale factor of a drawing?

Whenever you start drafting a technical drawing on paper, first thing you have to decide is the representative fraction or scale factor of the drawing. The selection of correct value of representative fraction depends on two things,

1. Dimensions of the actual object
2. Dimensions of the space available on paper

Let me just explain to you, by a practical situation, how to reach the correct value of R.F. Suppose, you want to make floor plan of a room, with floor dimensions 3 m×4.5 m on A3 size drawing sheet (297 mm×420 mm).

Now, first step is, you bring the dimension of the object and paper in same units.

Actual floor dimension = 3 m x 4.5 m = 3000 mm x 4500 mm

Size of A3 size drawing sheet = 297 mm x 420 mm

Try to select values of R.F., only from values recommended in Table 1.4. If you select the value of R.F. to be 1:10, then for 3000 mm actual length, length on drawing will be 3000 ×1/10=300 mm and for 4500 mm actual length, length on drawing will be 4500×1/10=450 mm.

Table 1.4: Recommended values of R.F. as per IS: SP 46 (2003)

Category	Recommended Scales
Enlargement scales	50:1, 20:1, 10:1,5:1,2:1
Full size	1:1
Reduction 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

Now, 300 mm×450 mm is larger than A3 size drawing sheet (297 mm×420 mm). And therefore, you have to select next lower value of R.F., from recommended values, so that the drawing comes inside the drawing sheet.

The next lower of R.F. from recommended values is 1:20. If you select the value of R.F. to be 1:20, then for 3000 mm actual length, length on drawing will be 3000 ×1/20=150 mm and for 4500 mm actual length, length on drawing will be 4500×1/20=225 mm.

Now, 150 mm x 225 mm drawing, can easily come inside the A3 size drawing sheet, (297 mm×420 mm). Hence you finalise the R.F. value as 1/20.

If you have made true size drawing in Auto CAD, then before printing, in the plot model dialogue box, you have to specify value of scale as (1/20).

1.3.4 Engineer's Scale or Architect's Scale

With the availability of drafting software like Auto CAD, manual drafting of drawing has now become obsolete. Consequently, the engineer's scale or architect's scale, which are used for making manual drawings, are also out of fashion. But still then, the concept of engineer's scale is useful, in developing your understanding of scales and scale factor.

If you try to draw a reduced drawing, manually on paper, with a particular value of R.F., suppose (1/20), with a true scale, then every linear dimension has to be first divided by 20, and then the numerical value obtained is drawn by a true scale. This consumes time, and wastes your energy in repetitive calculations.

This repetitive process of calculation, can be easily automised, by the use of M3 engineer's scale, shown in Fig.1.25. M3 engineer's scale, is having two reducing scales, one with R.F. value (1/20) on one side, and other with R.F. value (1/10) on other side. If you draw a line of length 100 cm by the

reducing scale of R.F. value (1/20) then, it is actually a line of length (100/20) =5 cm (observe Fig.1.25). Hence you can define engineer's scale, as scale which automatically reduces or enlarges distances

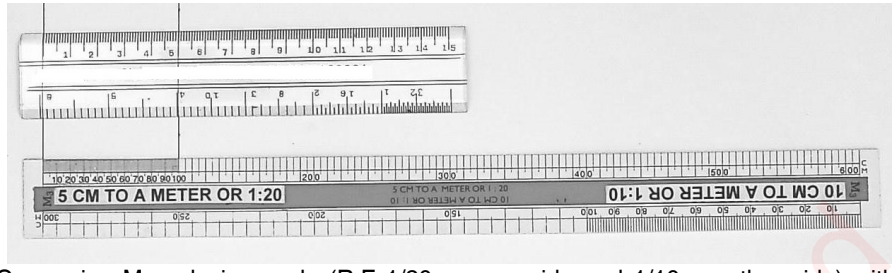


Fig.1.25: Comparing M_3 reducing scale (R.F 1/20 on one side and 1/10 on other side) with a full-size scale

in a ratio, whose value is equal to the R.F. value printed on the scale. Commonly available set of engineering scales and recommended R.F. values are shown in Table 1.5.

Table 1.5: Commonly available set of engineer's scales

Name	Representative fraction
M1	1:1,1:2
M2	1:5,1:2.5
M3	1:10,1:20
M4	1:50,1:100
M5	1:200,1:500
M6	1:300,1:600
M7	1:400,1:800
M8	1:1000,1:2000

1.3.5 Graphical Scale

Graphical scale, is a scale drawn on the drawing sheet itself. When you are using nonstandard value of R.F., for which engineer's scale is not available, then you first draw a graphical scale of required value of R.F., on the drawing sheet itself. This helps you, to make the drawing with given value of R.F., without calculation.

You can also see graphical scales, on the tourist maps and survey maps. The main advantage of graphical scale drawn there is, you can directly measure actual distance between two locations A and B on map, by the help of graphical scale printed on the map.

Two types of graphical scale will now be discussed.

Plain scale

A graphical scale, indicating two units only or a unit and its subunit only, or multiples of a unit and unit only, is called a plain scale. As an illustration, example 1.2 in the subsequent pages, explains the concept and procedure, for making graphical plain scale.

Diagonal scale

In Example 1.2, if you are asked to measure 17.4 km, you will find it difficult, to measure it by graphical plain scale drawn in Fig.1.26. The plain scale drawn, has least count of 1 km, and therefore it can measure either 17 km or 18 km, but it cannot measure 17.4 km. For measuring to the least count of 0.1 km, you require to draw a graphical diagonal scale.

Diagonal scale helps to measure distances in three units, i.e., unit, sub unit and sub-sub unit or 10 units, 1 unit and 0.1 unit or some other similar combination. As an illustration, example 1.3 in the subsequent pages, explains the principle and procedure for making graphical diagonal scale.

SOLVED PROBLEMS (WITH INCREASING DIFFICULTY LEVEL)

Example 1.2 *On a tourist map of Mumbai, 10 km of actual distance is represented by 5 cm length on the map. What is the representative fraction of the map drawing? Draw a plain scale on the map to read up to 30 km, and with least count of 1 km. Show a distance of 17 km marked on the scale.*

Solution

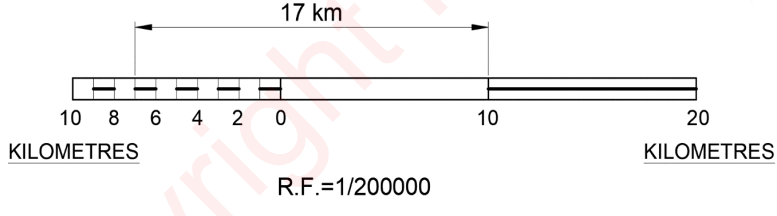


Fig.1.26

$$\text{Representative fraction of a drawing} = \frac{\text{Distance between two points A and B on drawing}}{\text{Actual distance between the points A and B}}$$

$$\text{Representative fraction of a drawing} = \frac{5cm}{10km} = \frac{5cm}{10 \times 1000 \times 100\ cm} = \frac{1}{200000}$$

Now, R.F. of drawing and R.F. of plain scale, for making or measuring elements of that drawing is always same.

$$\text{R. F. of plain scale} = \frac{\text{length of plain scale}}{\text{Actual length to be measured}}$$

$$\Rightarrow \text{Length of plain scale} = \text{R. F. of plain scale} \times \text{Actual length to be measured}$$

$$\Rightarrow \text{Length of plain scale} = \frac{1}{200000} \times 30km = \frac{1}{200000} \times 30 \times 1000 \times 100\ cm = 15\ cm$$

1. Draw a line segment of length 15 cm, and divide it into 3 equal parts. (Logic: - 15cm distance on drawing, represents an actual distance of 30 km, and therefore you divide it, into 3 equal parts, each part representing a distance of 10 km reduced 200000 times.)
2. At the end of first division, mark 0, & mark 10 & 20 at the end of subsequent divisions to its right. Divide first division, into 10 equal parts, each part representing a distance of 1km reduced 200000 times. On first division, mark kms from 1 to 10 starting from 0 and moving to left. (Logic: - 0 is not marked at the start of first division, but it is marked at the end of first division. The advantage is, there is no need of dividing all 3 divisions into 10 equal parts. Only dividing the first division into 10 equal parts works. Whether, you want to measure 17 km or 7 km or 27 km, it can be easily done with this technique.)
3. Show scale as a rectangle of height 3mm to 6mm, & for ease of measurement, draw thick and dark horizontal lines after alternate divisions. Mark distance of 17 km as shown in Fig.1.26.

Example 1.3 *On a tourist map of Mumbai, 10 km of actual distance, is represented by 5 cm length on the map. What is representative fraction of the map drawing? Draw a diagonal scale on the map to read up to 30 km, and with least count of 0.1 km. Show a distance of 17.4 km marked on the scale.*

Solution

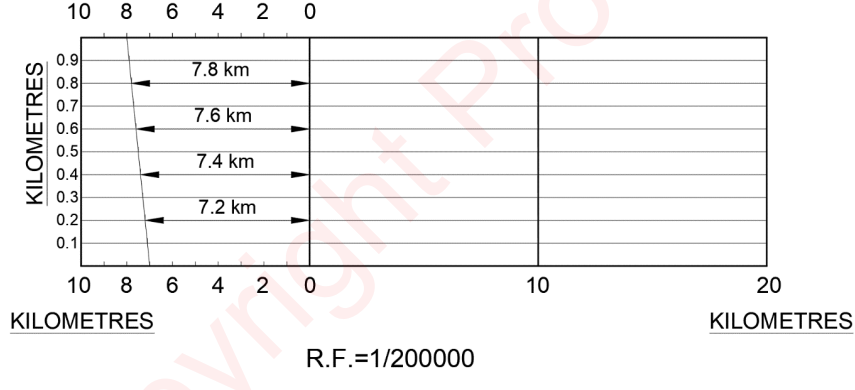


Fig: 1.27

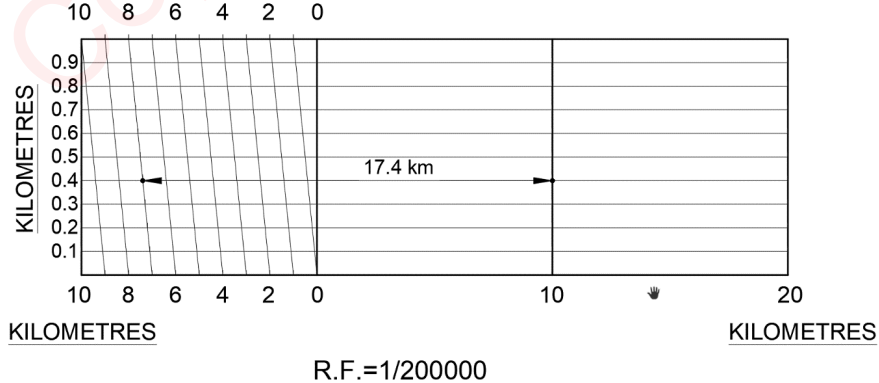


Fig: 1.28

Fig.1.27 explains the principle of diagonal scale i.e., how in a diagonal scale you are able to measure minute distances, which cannot be measured by plain scale. Just observe the line segment $\overline{00}$ and line segment $\overline{78}$. As you go from bottom to top the horizontal distance between the two-line segments increases from 7 km to 8 km (R.F. =1/200000). At horizontal level 0.1 the distance between the line segment $\overline{00}$ and line segment $\overline{78}$ is 7.1 km (R.F. =1/200000). At horizontal level 0.4 the distance between the line segment $\overline{00}$ and line segment $\overline{78}$ is 7.4 km (R.F. =1/200000) and so on.

Fig.1.28 is the solution. The explanation is in continuation of explanation for problem 1.1. From point 10 at left draw a vertical line, and divide it into 10 equal parts, because least count required is 0.1 km (If least count required would have been 0.2 km, then you would have divided vertical line from point 10 at left in 5 equal parts only). Mark the horizontal levels as 0.1, 0.2 …..0.9,1. Draw the rectangle and draw horizontal lines for all levels. The Diagonal scale is now complete.

To measure the distance of 17.4 km, simply go at the horizontal level of 0.4 and measure the distance as shown in Fig.1.28.

1.4 DIMENSIONING TECHNIQUES

Dimensioning is a technique of describing the sizes and locations of various parts in a drawing. It is a numerical value expressed in appropriate units of measurement. It is identified on a drawing along with lines, symbols and text to define the geometrical characteristics of an object. All dimensional information necessary to define a part or a component clearly and completely shall be shown directly on a drawing. Each feature shall be dimensioned once only on a drawing. Dimension shall be placed on the view or section that most clearly shows the corresponding features.

1.4.1 Elements of Dimensioning

The elements of dimensioning include extension line, dimension lines, leaders, arrowheads, origin indication and dimension itself. The various elements of dimensioning are shown in Fig.1.29.

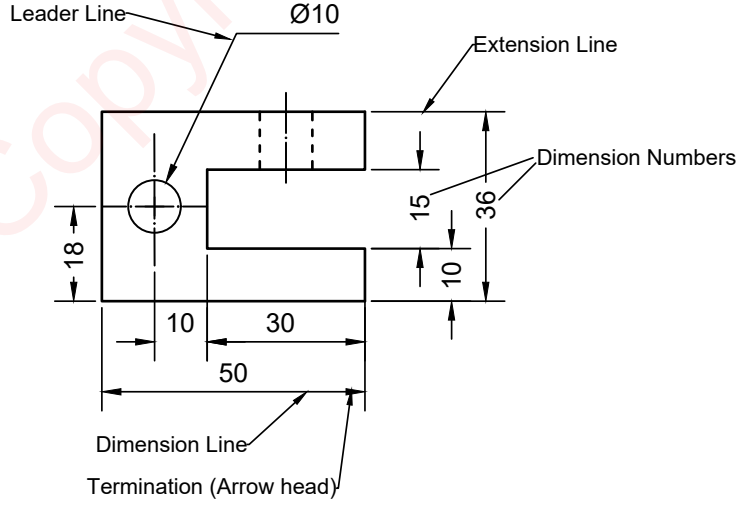


Fig.1.29: Elements of dimensioning

1.4.2 Methods of Dimensioning

There are two methods of dimensioning. The dimensions are indicated on a drawing as per one of the following methods as recommended in BIS (SP 46:2003).

Method 1- Aligned system

In this system, all dimensions are placed near the middle and above the dimension lines which are drawn without any break and written parallel to them. Dimensional value is indicated in such a way that they may be read from the bottom for horizontal dimensions and from the right side for vertical dimensions. Fig.1.30 shows this method of dimensioning.

Method 2- Unidirectional system

In this type of system, dimensions are readable from the bottom side of the given drawing-sheet. The dimension lines are broken near the middle for inserting the dimensions. Fig.1.31 shows this method of dimensioning. This system of dimensioning is generally used on large sized drawing, where it is not convenient to read the dimensions from the right-hand side.

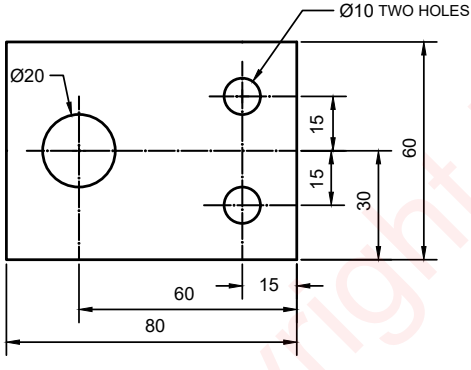


Fig.1.30: Aligned system of dimensioning

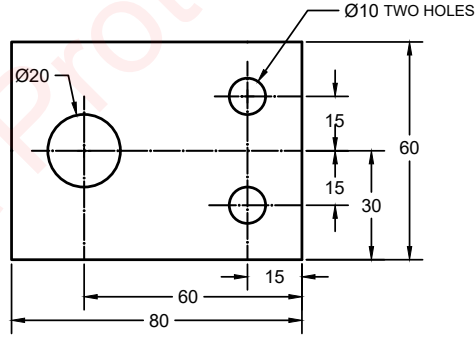


Fig.1.31: Unidirectional system of dimensioning

1.4.3 Arrangement of Dimensions

Chain dimensions

In this arrangement, dimension lines are drawn like a chain so that the successive arrowheads touch each other as shown in Fig.1.32. This is also known as continuous dimensioning.

Parallel dimensions

This type of arrangement is used when a number of dimension lines are spaced out parallel one to another and successively one below the other. This helps to refer a number of dimensioning from a common datum in a progressive manner as shown in Fig.1.33.

Combined dimensions

This arrangement is used, when both chain and parallel dimensions are indicated in the same drawing as shown in Fig.1.34.

Dimensioning by coordinates

In this type of dimensioning, a table can be used in place of other dimensioning styles.
This method is useful when a number of holes of different sizes have to be dimensioned. Fig.1.35 shows the dimensioning by coordinates.

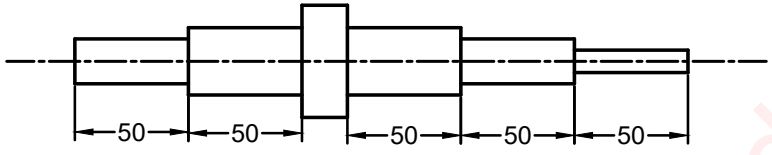


Fig.1.32: Chain–dimensioning

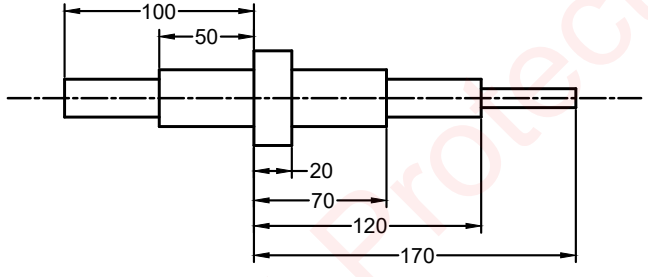


Fig.1.33: Parallel–dimensioning

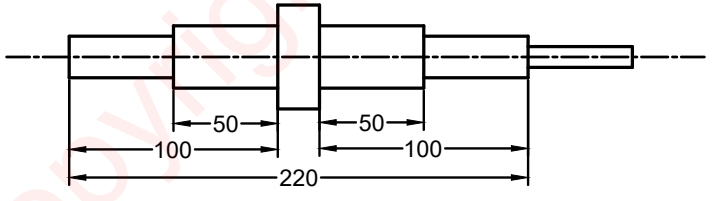
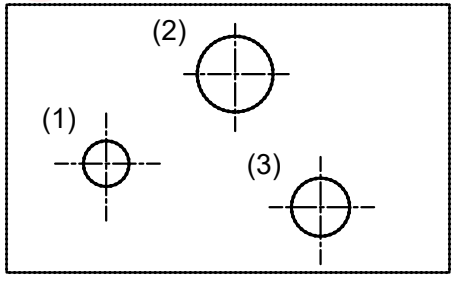


Fig.1.34: Combined–dimensioning



	X	Y	Ø
1	20	20	10
2	40	25	20
3	60	10	15

Fig.1.35: Dimensioning by coordinates

1.4.4 Example of Dimensioning Some Common Features

In engineering drawing some common features like circle, radii, chord and small arcs, angles etc., are used with different sizes and shapes. Some examples are shown to give an idea of indicating dimensions on these common features.

Dimensioning of circle

The dimension of a circle should be represented by the symbol Ø. If the circle is large in size it should be dimensioned with a dimension line drawn across the circle through its center at an angle with the diameter dimension shown in Fig.1.36 (a). If the space available inside the circle is insufficient to indicate the dimension, then it can be dimensioned using a leader or extension lines as shown in Fig.1.36 (b). If the space is insufficient to place the arrowheads inside, in the case of small diameter holes, then the diameter can be dimensioned as shown in Fig.1.36 (c). The circle should be dimensioned by one of the methods depending on size as shown in Fig.1.36(a), (b) and (c).

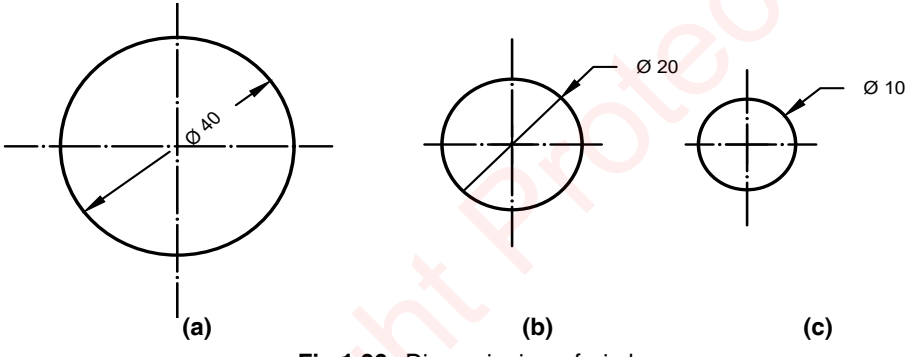


Fig.1.36: Dimensioning of circle

Dimensioning of radii

The dimension of a radii should be represented by the letter 'R'. As far as possible, the dimension line of a radius should pass through the centre of the arc. While dimensioning small radii, the arrow may be reversed. Leaders which indicate radii must always be radial lines. The radii should be dimensioned by one of the methods as shown in Fig.1.37.

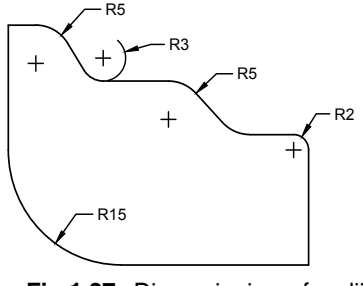


Fig.1.37: Dimensioning of radii

Dimensioning of chord and arc length

Fig.1.38(a) and 1.38(b) shows method of dimensioning chords and arcs.

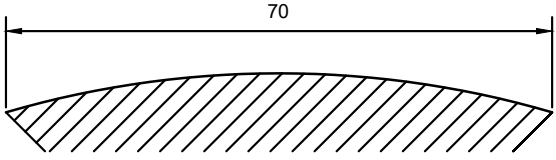


Fig.1.38(a): Dimensioning chords

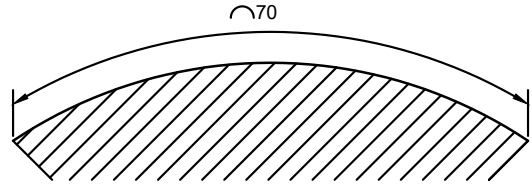


Fig.1.38(b): Dimensioning arcs

Dimension in narrow space

Sometimes the space between the extension lines is too small to insert the dimension value, so the dimension may be placed above the extended portion of the dimension line beyond the arrowheads, but preferably on right side as shown in Fig.1.39.

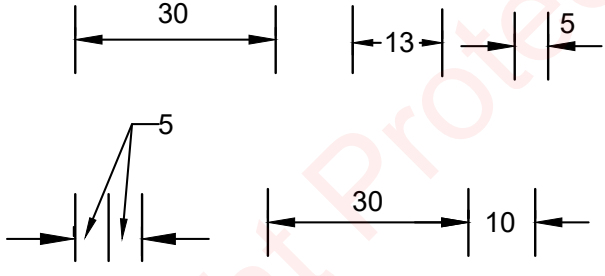


Fig.1.39: Dimension in narrow space

Dimensioning of angles

Angles are normally indicated in degrees. The different angles should be dimensioned by any one of the methods as shown in Fig.1.40.

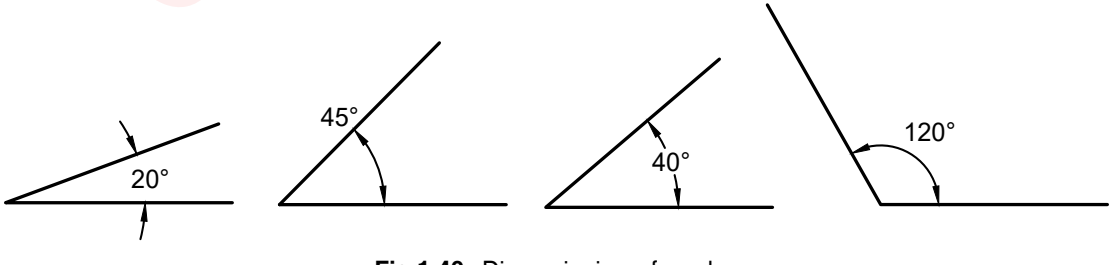
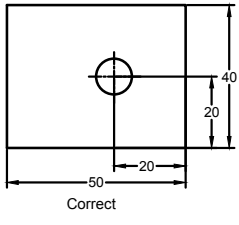


Fig.1.40: Dimensioning of angles

1.4.5 General Rules for Dimensioning

Some rules for dimensioning are given below:

1. Dimensions should be placed on the view which shows the relevant features clearly.
2. Dimensions marked in one view need not be repeated in another view.
3. Dimensions should be placed outside the view as show in Fig.1.41.
4. Dimensions should not be taken from hidden lines as shown in Fig.1.42.
5. Dimensions should be given from a base line, a centre line of the hole, or a finished surface. Dimensioning to a centre line should be avoided except when the centre line passes through the centre of the hole. Fig.1.43.
6. The crossing of dimension lines should be avoided as for as possible.
7. Aligned system of dimensioning is recommended.



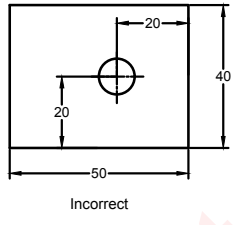
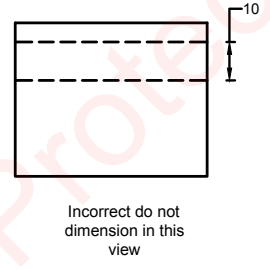


Fig.1.41: Dimensions should be placed outside the view



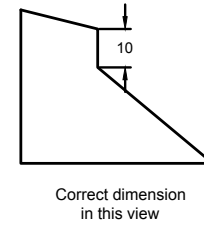
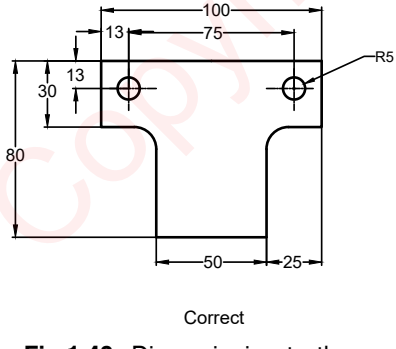


Fig.1.42: Dimensions should not taken from hidden lines



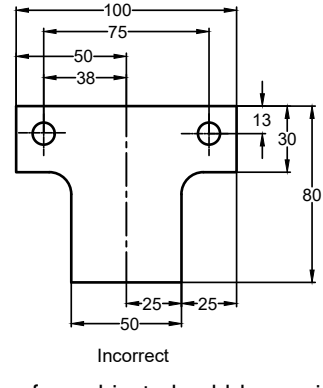


Fig.1.43: Dimensioning to the centre line of an object should be avoided

1.4.6 Drawing of Arrowheads

1. Arrow heads are drawn free hands.
2. Arrow head must be symmetrical about dimension line.
3. Arrow head may be open or solid depending upon the tip of a arrow being solid or open shown in Fig.1.44.

4. The size of arrow head should be proportionate to the thickness of drawing lines and size of drawing. Generally, the length of arrow head is about three times the depth. See Fig.1.45.

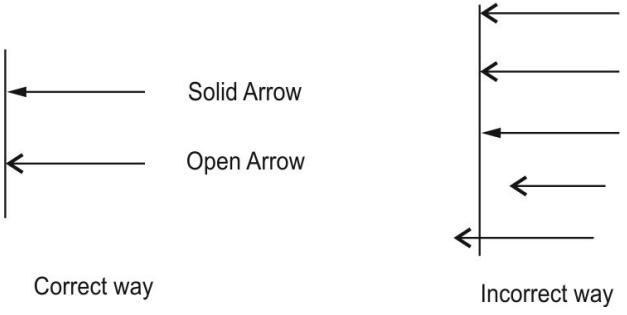


Fig.1.44: Solid and open arrow

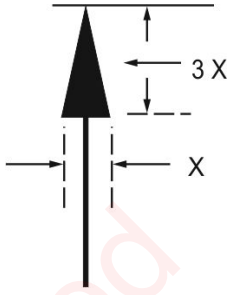
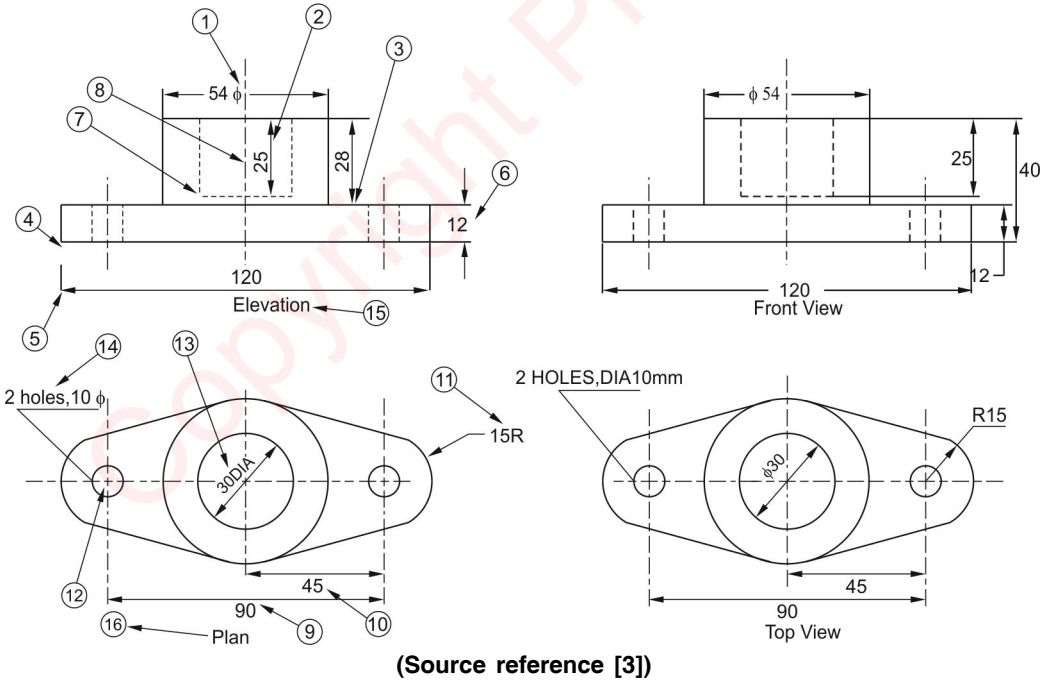


Fig.1.45: Size of arrowhead

SOLVED PROBLEMS (WITH INCREASING DIFFICULTY LEVEL)

Example 1.4 *Violation of some of the principles of drawing are indicated in following figure. The corrected version of the same as per BIS SP 46-2003 is given in the solution. The violations from 1 to 16 indicated in the figure are explained below.*

Solution



(Source reference [3])

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 aligned method.
7. Hidden lines should meet without a gap.
8. Centre line representation is wrong. Dots should be replaced by small dashes.
9. Horizontal dimension line should not be broken to insert the value of dimension in both aligned and uni-direction methods.
10. Dimension should be placed above the dimension line.
11. Radius symbol should precede the dimension.
12. Centre line should cross with long dashes not short dashes.
13. Dimension should be written by symbol followed by its values and not abbreviation.
14. Note with dimensions should be written in capitals.
15. Elevation is not correct usage.
16. Plan is obsolete in graphic language

1.5 GEOMETRICAL AND TANGENCY CONSTRUCTIONS

1.5.1 Introduction

The art of representation of geometrical object such as triangle, rectangle, square, circle, polygon etc, on a paper is known as geometrical drawings.

In the process of preparing a drawing, there will be many occasions when it will be necessary to utilize one or more of the geometric construction. These construction techniques will be helpful in solving problem involving the application of points, lines, angles and curved surfaces.

1.5.2 Definition

Definition of terms

Point: A point has position but no magnitude or the place of intersection of lines is called as point. In Fig.1.46 a point is represented by the intersection of two lines (a), by a short cross bar a line (b), by a small cross (c).

Line:

1. A straight line is the shortest distance between two pints.
2. A curved line is one which changes its direction from point to point (see Fig.1.47)

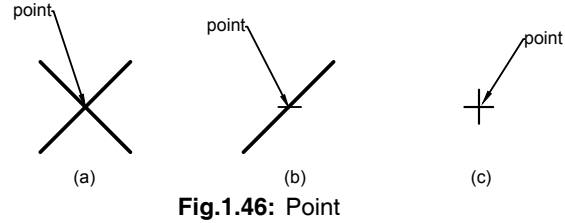


Fig.1.46: Point

Angle. An angle is formed by two intersecting lines.

The different kinds of angles are illustrated in Fig.1.48. Most of the angles used in technical drawing can be drawn easily with the mini drafter, T-square or straight edge and triangles. For drawing odd angles, use the protractor.

Triangles. A triangle is a plane figure bounded by three straight sides and the sum of the interior angles is always 180^0. The different kinds of triangles are shown in Fig.1.49.

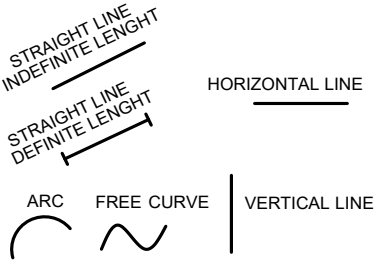


Fig.1.47: Types of line

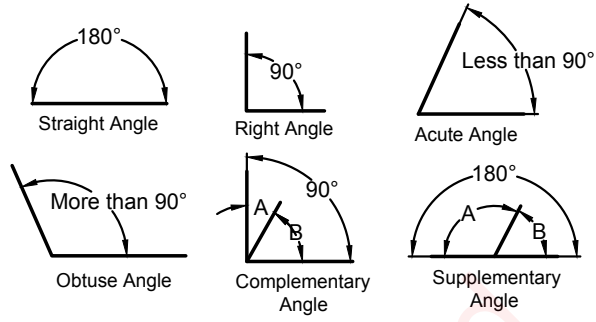


Fig.1.48: Types of angles

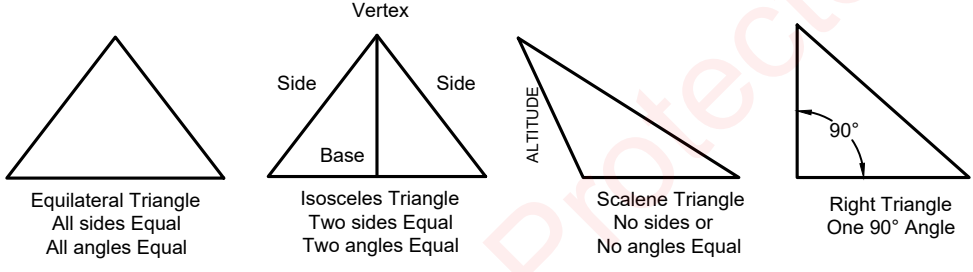


Fig.1.49: Different types of triangles

Quadrilaterals. A quadrilateral is a plane figure bounded by four straight sides. If the opposite sides are parallel, the quadrilateral is known as parallelogram as shown in Fig.1.50.

Circle. A circle is a closed curve all points of which are at the same distance from a point called the centre. Different parts of circles are shown in Fig.1.51.

Polygons. A polygon is a plane figure bounded by straight lines. If the polygon has equal angles and equal sides, it can be inscribed or circumscribed around a circle. Such polygon are known as regular polygon. The different types of polygons are illustrated in Fig.1.52.

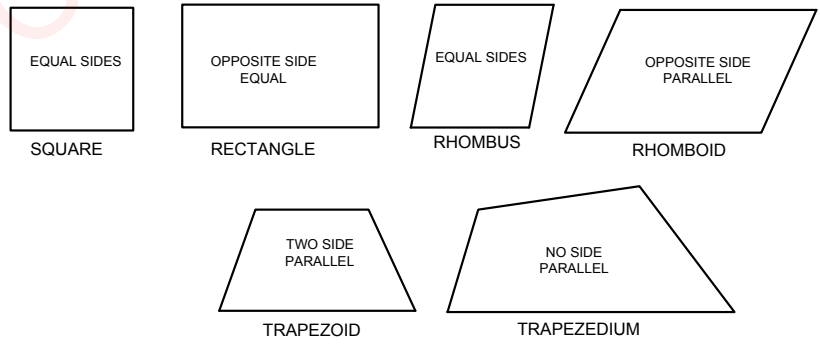


Fig.1.50: Types of quadrilaterals

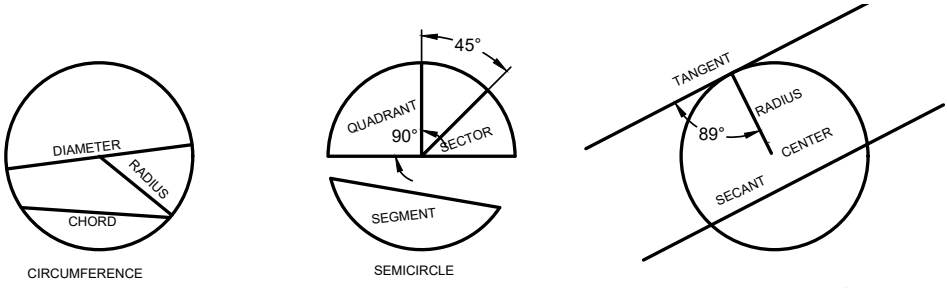


Fig.1.51: Different parts of circle

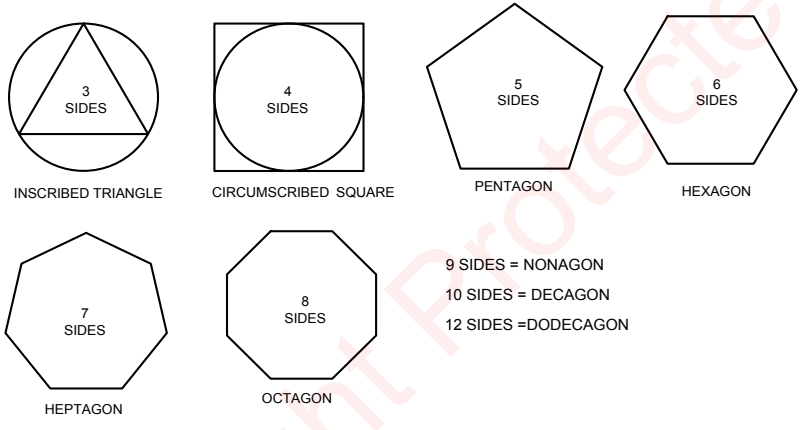


Fig.1.52: Different types of polygons

1.5.3 Lines and Angles

Bisecting a line or arc *(see Fig.1.53)*

1. Draw a line or arc AB of given length.
2. Set the compass for any radius greater than one-half of AB. Using A&B as centres, draw two arcs to intersect at C and two arcs to intersect at D.
3. The point at which line CD crosses AB is the centre.
4. Note: The dividers may be used to bisect a line or divide it into a number of equal parts.

Bisecting an angle *(see Fig.1.54)*

1. Draw angle BAC.
2. With A as a centre and the compass set at any convenient radius, draw an arc cutting line AB at D and line AC at E.
3. Set the compass at a radius greater than one-half of DE.
4. With D and E as centres, draw two arc to intersect at Q.
5. Draw a line from Q to A. The line QA bisects the angle.

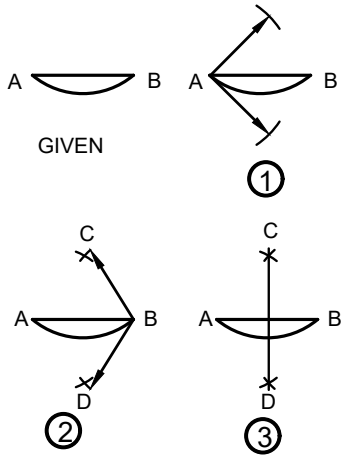


Fig.1.53: Bisecting a line or arc

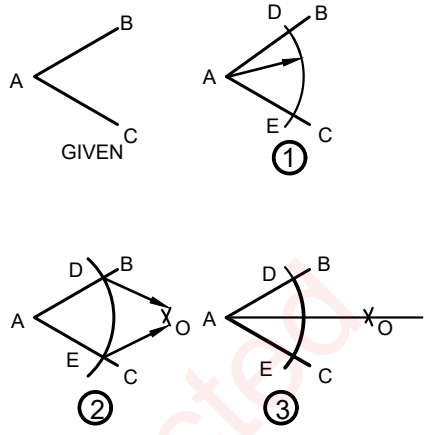


Fig.1.54: Bisecting an angle

Drawing a line parallel to another line at a given distance *(see Fig.1.55)*

1. Let AB be the straight line and CD the given distance at using a line parallel to AB is to be drawn. Using any point E and F on line AB as centres and CD as the radius, draw two arcs. The line GH drawn tangent to the two arcs is the regular line.
2. Curved lines: Let AB be the curved line and CD the given distance. Draw a series of arcs from AB with a radius equal to CD. With the help of french curve, draw line tangent to these arcs (see fig1.56).

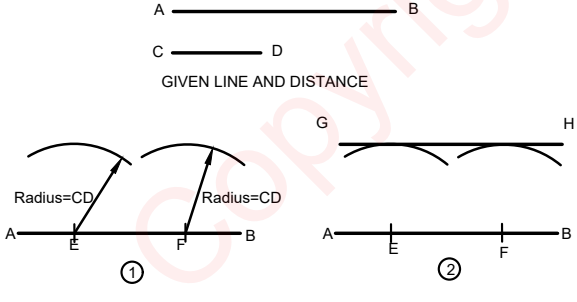


Fig.1.55: Drawing a line parallel to another line at a given distance

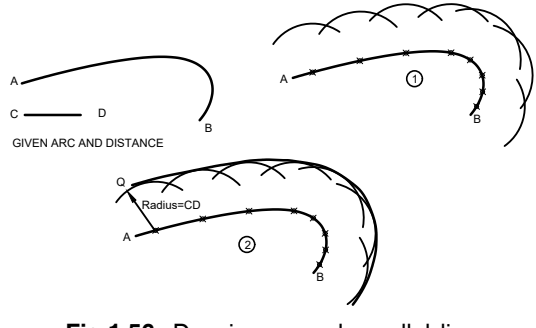


Fig.1.56: Drawing curved parallel lines

Dividing a line into equal parts

Fig.(1.57.) shows a line AB which is to be divided into six equal parts;

1. From one end of this line draw line AC at any convenient angle.
2. Starting at A on line AC lay off six equal spaces either with dividers or a scale.
3. From the termination point of the last space D, draw a line connecting D with B.
4. With the edge of a triangle set parallel with line DB, draw lines from the points on line AC to line AB. The division points will be found where the parallel lines intersect line AB.

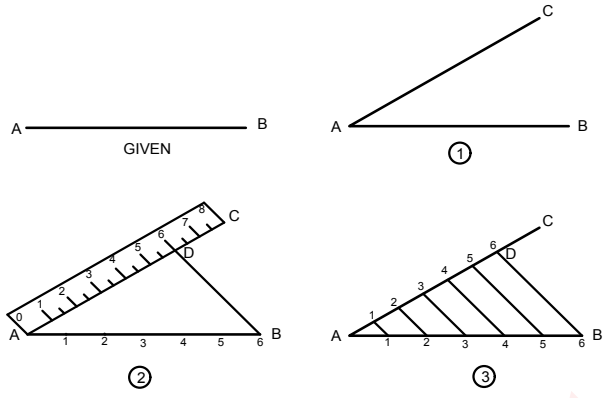


Fig.1.57: Dividing a line into six equal parts

1.5.4 Construction of Angles by using Compass

Constructing an angle of 60°

AB is the base line, with A as centre and any suitable radius describe an arc to obtain point C. Using the same radius and point C as centre, mark of point D. Join AD. The angle DAC thus obtained is of 60°.

Constructing an angle of 30°

Construct an angle of 60° as in the preceding construction. With C and D as centres and the same radius AC strike an arc to obtain point E. Join AE. The angle EAC thus obtained is of 30°.

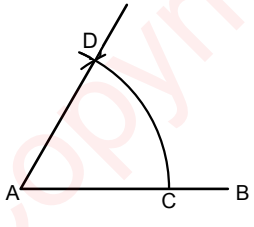


Fig.1.58: Construct an angle of 60°

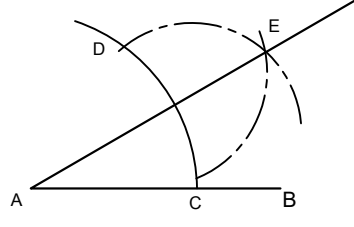


Fig.1.59: Construct an angle of 30°

Constructing an angle of 90°

Let AB is the base line as shown in Fig.1.60. With a convenient radius AC and A as centre, strike the arc. Using the same radius and centre C, mark off D. Similarly, mark of E. With D and E as centres and using the same radius obtain point F. Join AF. Then, the angle FAC thus obtained is of 90°.

Constructing an angle of 75°

Construct an angle of 90° as in the preceding construction. Using a constant radius, bisect the arc DO to obtain point G. Join AG. The angle GAC thus obtained is of 75°. See Fig.1.61.

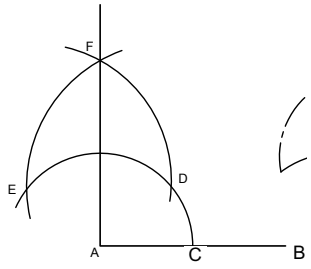


Fig.1.60: Construct an angle of 90°

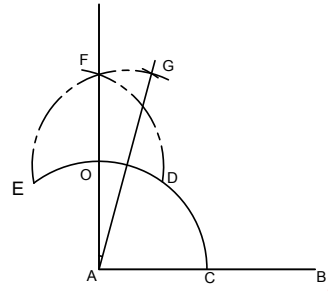


Fig.1.61: Construct an angle of 75°

1.5.5 Construction of Triangles and Quadrilaterals

Drawing a triangle with sides given of length, A, B and C *as shown in Fig.1.62(a)*

1. Draw one side, as C, in desired position and strike an arc with radius equal to side A.
2. Strike another arc with radius equal to side B.
3. Draw sides from intersect of arcs as shown in Fig.1.62(3).

Drawing a right angle triangle length of hypotenuse and one side is given

1. Draw line AB equal to length 'S' given in Fig.1.63(a).
2. With AB as a diameter equal to S, draw a semi circle.
3. With A as centre and R as radius, draw an arc intersecting the semi-circle at C. Draw AC and CB to complete the right angle triangle shown in Fig.1.63.

Drawing an equilateral triangle

1. Draw line of AB of given length.
2. With A and B as centres and AB as radius, strike arcs to intersect at C.
3. Draw lines AC and BC to complete the triangle as shown in Fig.1.64.

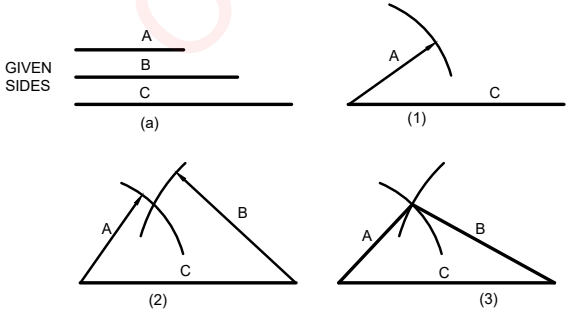


Fig.1.62: Draw a triangle with sides given

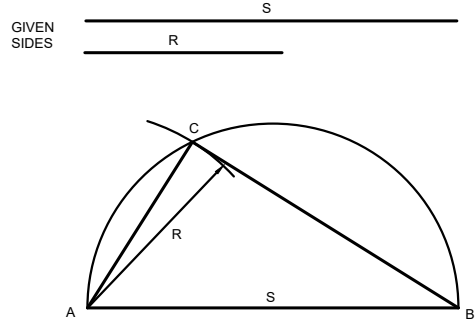


Fig.1.63: Draw a right angle triangle length of hypotenuse and one side is given

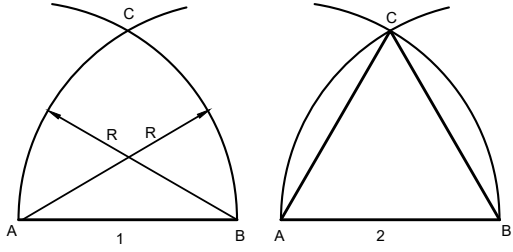


Fig.1.64: Construct an equilateral triangle

Drawing a square

1. Select a point A which will act as one corner of the needed square.
2. Construct horizontal and vertical lines through point A.
3. Measure the desired length to point B and construct a vertical line through it.
4. From point A (or B), construct a 45° line establishing point C.
5. Construct a horizontal line through point C establishing line CD and complete the required square ABCD as shown in Fig.1.65.

Constructing a rectangle, given the diagonal BD and one side of it

1. Bisect BD at O.
2. With O as centre and OB as radius, draw a circle.
3. With B and D as centres and radius equal to the length of the given side, draw arcs to cut the circle at C and A as shown in Fig. 1.66.
4. Join AD, BC and then AB and CD. Thus, ABCD is the required rectangle.

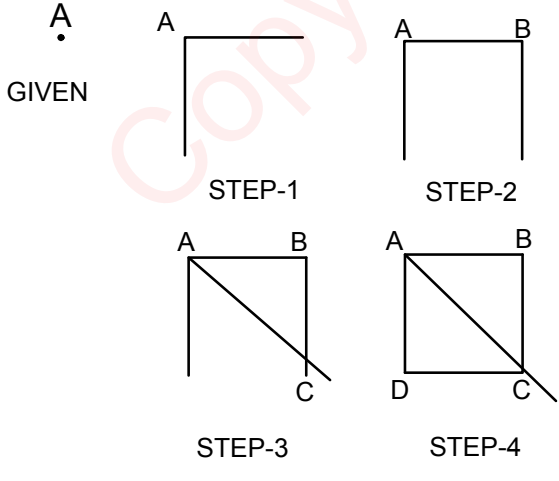


Fig.1.65: Draw a square by measuring only one side

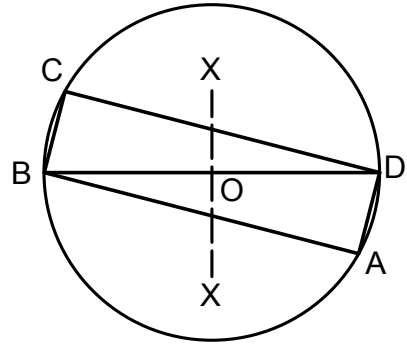


Fig.1.66: Constructing a rectangle, given the diagonal and one side

1.5.6 Construction of Circles

Constructing a circle or arc through three points

Given three points A, B and C.

1. Construct lines AB and BC.
2. Bisect lines AB and BC using procedure as shown in Fig.1.67.
3. The intersection of the two bisectors becomes the radius point and an arc or circle can be formed through points A, B and C.

Locating a circle's center

(Refer Fig.1.68) Given a circle of any diameter

1. Draw any chord AB, within the circle.
2. Construct lines AC and BD perpendicular to line AB.
3. Construct line AD and BC. Intersection ' X ' will be the circle's center.

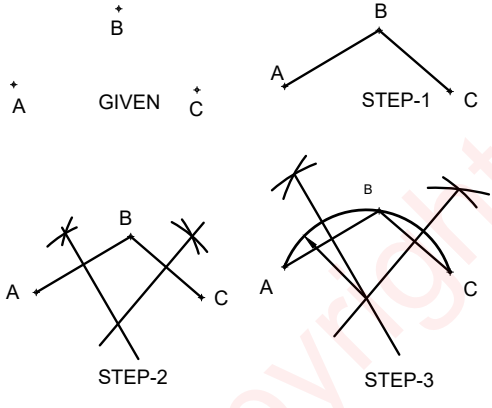


Fig.1.67: Construction of a circle or an arc through three points

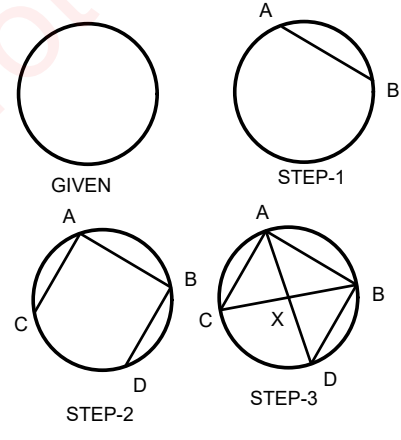


Fig.1.68: Locating a circle's center

1.5.7 Construction of Regular Polygons

Construction of hexagon with drafting tools

Fig.1.69 shows how to construct a hexagon when the dimension given in the distance across the flats as opposed to the distance across the corners.

1. Bisect the line to locate its centre, using half the line's length as a radius construct a circle.
2. Using the 30° - 60° triangle draw the 30° and 60° inclined lines tangent to the circle and two horizontal lines also tangent to the circle.
3. Draw hexagon's heavy object lines and erase all construction lines.

Construction of hexagon inside a circle

(Refer Fig.1.70) Given a circle with center point.

1. Draw a horizontal line through the center of the circle forming points A and B.
2. Using the 30° - 60° triangle construct the hexagon's four sides from points A and B.
3. Draw hexagon's horizontal top and bottom lines.

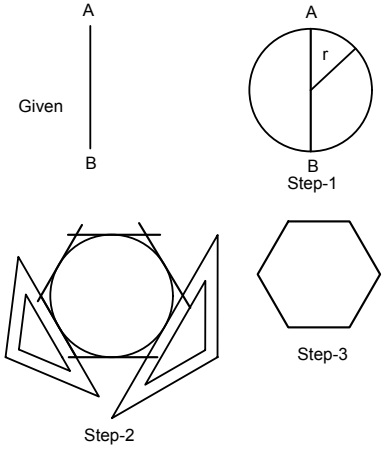


Fig.1.69: Construction of hexagon

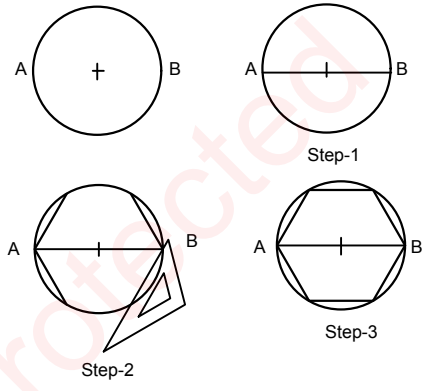


Fig.1.70: Draw a hexagon inside a circle

Construction of octagon

Fig.1.71 shows how to construct a octagon when the distance across the flats is given.

1. Locate the centre of line AB. Using AZ as the radius, draw a full circle.
2. Construct two horizontal and two vertical lines tangent to the circle's arc. Using the 45° triangle construct four inclined lines tangent to the circle's arc.
3. Draw octagon's heavy object lines and erase all construction lines.

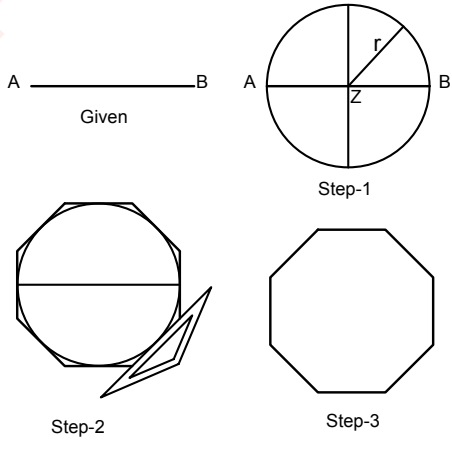


Fig.1.71: Draw the octagon

1.5.8 Tangency Construction

The outline of many parts of machines, apparatus, instruments and devices consists of straight lines and arcs gradually merging into one another. Such gradual continuation of one line whether straight or curved into another straight or curved is termed as tangency.

Drawing a tangent to a circle

(Refer Fig.1.72) Given a circle of any diameter with a centre point O and a point P beyond it, draw a tangent from P to the circle.

1. Join points P and O.
2. Bisecting a line PO and describe a semicircle upon it.
3. The point of contact between the circle and the semicircle is X. Join points P and X produce to Y.
4. Line PY is the tangent.

Drawing a common exterior tangent to two unequal circles

(Refer Fig.1.73) Given two circles of unequal diameter with centres O1 and O2.

1. Join O1 O2 and bisect it.
2. On O1 and O2 draw a semi-circle.
3. Mark off CE equal to DO2
4. With O1 as centre and radius O1E strike an arc EF. Join O1F and produce it to obtain the point A.
5. Draw BO2 parallel to O1A.
6. Join AB which is the required tangent as shown in Fig.1.73.

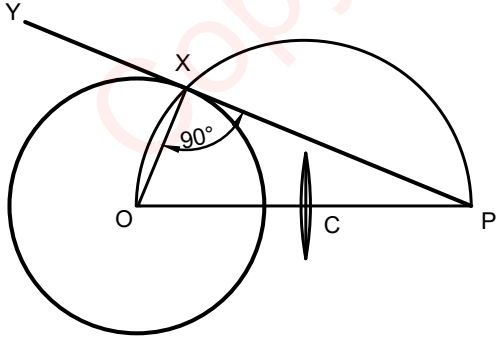


Fig.1.72: Draw a tangent to a circle

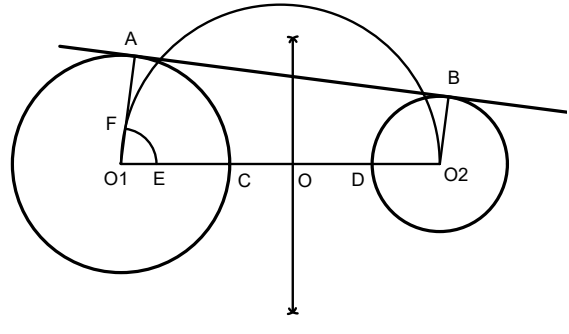


Fig.1.73: Draw a common exterior tangent to two unequal circles

Drawing a tangent to a circle through a point

Given point P on the circle.

1. Move the mini drafter horizontal scale and triangle as a unit until one side of the triangle passes through the point P and the centre of the circle.
2. Slide the triangle until the other side passes through point P, and draw the required tangent. See Fig.1.74

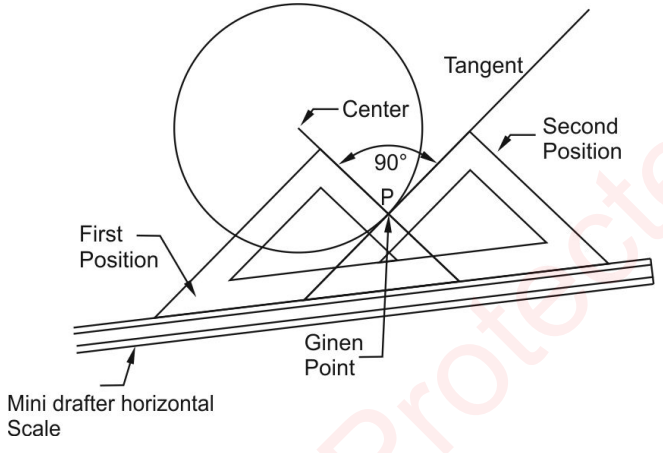


Fig.1.74: Draw a tangent to a circle through a point

SOLVED PROBLEMS (WITH INCREASING DIFFICULTY LEVEL)

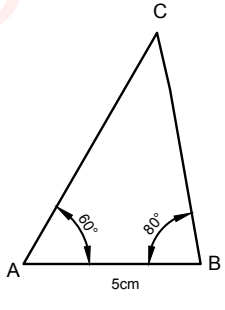
Example 1.5 *Construct the circumcentre of the ΔABC with AB = 5 cm, angle A = 60° and angle B = 80° draw the circumcircle and find the circumradius of the ΔABC.*

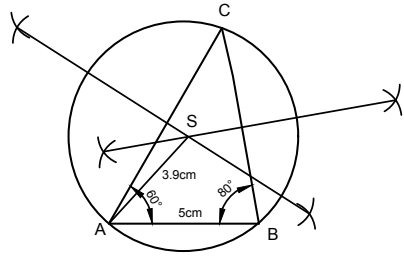
Solution

Step-1. Draw the ΔABC with the given measurements

Step-2. Construct the perpendicular bisector of any two sides (AC and BC) and let them meet at S which is the circumcentre.

Step-3. S as centre and SA = SB = SC as radius, draw the Circumcircle to passes through A,B and C. Circumradius = 3.9 cm.



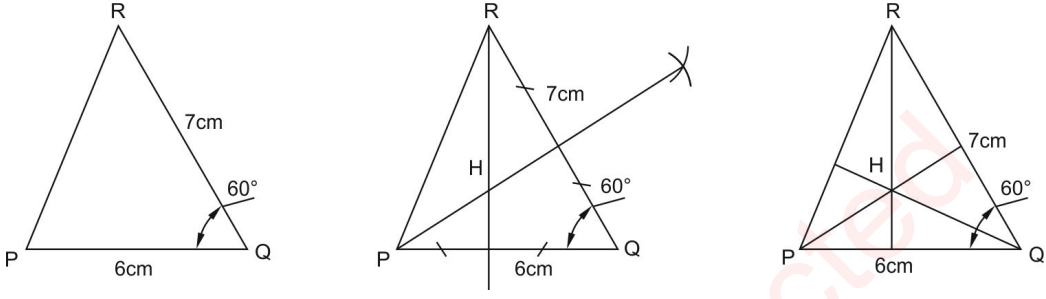


Example 1.6 *Construct ΔPQR whose sides are PQ = 6 cm Q = 60⁰ and QR = 7 cm and locate its Orthocentre.*

Solution

Step-1. Draw the ΔPQR with the given measurements.

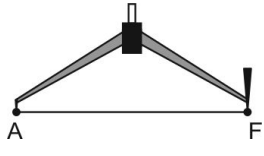
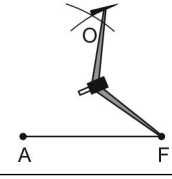
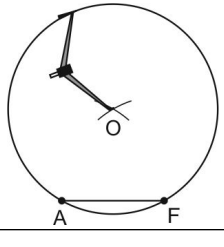
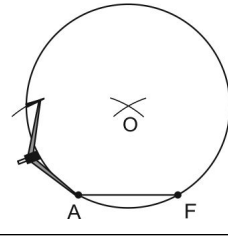
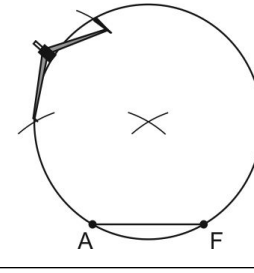
Step-2. Construct altitudes from any two vertices R and P, to their opposite sides PQ and QR respectively. The point of intersection of the altitude H is the Orthocentre of the given ΔPQR.



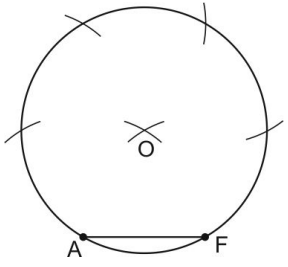
Example 1.7 *Construct a hexagon with 50 mm side (Horizontal)*

Solution

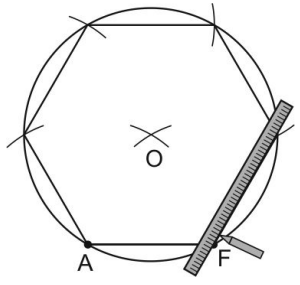
This construction is very similar to constructing a hexagon inscribed in a circle, except we are not given the circle, but one of the sides instead. Steps 1-3 are there to draw this circle, and from then on the constructions are the same. The center of the circle is found using the fact that the radius of a regular hexagon (distance from the center to a vertex) is equal to the length of each side.

1. Start with a line segment AF of 50 mm length. This will become one side of the hexagon. Because we are constructing a regular hexagon, the other five sides will have this length also. 	2. Set the compasses point on A, and set its width to F. The compasses must remain at this width for the remainder of the construction. 	3. From points A and F, draw two arcs so that they intersect. Mark this as point O. This is the center of the hexagon's circumcircle. 
4. Move the compasses to O and draw a circle. This is the hexagon's circumcircle - the circle that passes through all six vertices 	5. Move the compasses on to A and draw an arc across the circle. This is the next vertex of the hexagon. 	6. Move the compasses to this arc and draw an arc across the circle to create the next vertex. 

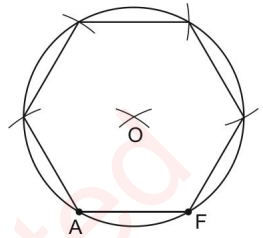
7. Continue in this way until you have all six vertices. (Four new ones plus the points A and F you started with.)



8. Draw a line between each successive pairs of vertices.

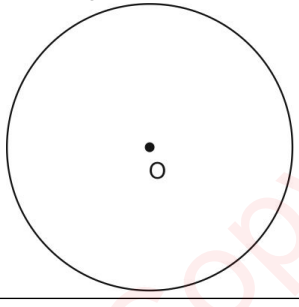


9. Done. These lines form a regular hexagon where each side is equal in length to AF=50 mm.

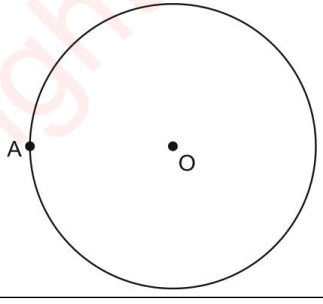


Example 1.8 *Construct an inscribed square with side 25 mm.*
Solution

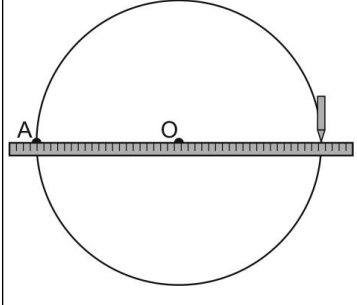
1. Start with the given circle, center O and radius of 25 mm. If the circle center point is not given, you can construct the center using the method shown in Finding the center of a circle.



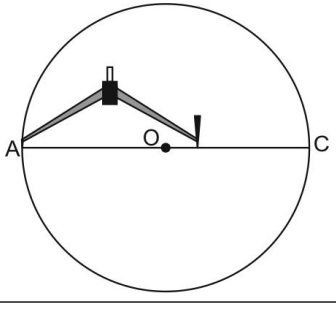
2. Mark a point A on the circle. This will become one of the vertices of the square.



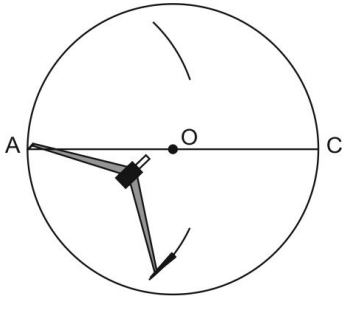
3. Draw a diameter line from the point A, through the center and on to cross the circle again, creating point C.



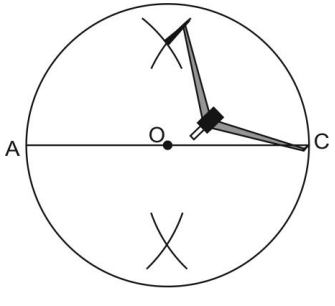
4. Set the compass on A and set the width to a little more than the distance to O.

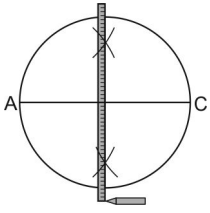
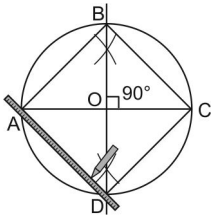
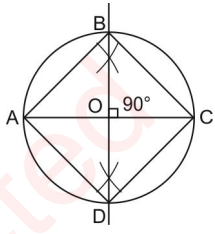


5. Draw an arc above and below O.



6. Move the compass to C and repeat.



7. Draw a line through where the arc pairs cross, making it long enough to touch the circle at top and bottom, creating the new points B and D. This is a diameter at right angles to the first one AC. 	8. Draw a line between each successive pairs of points A, B, C, D 	9. Done. ABCD is a square inscribed in the given circle. 

UNIT SUMMARY

1. Indian Standards Institutions(ISI) is renamed as Bureau of Indian Standards (BIS). BIS has developed different standard code of practice to maintained the uniform standard of engineering drawing throughout the country.
2. The engineering or technical drawing can be prepared with manuals instruments on drawing sheets. Accuracy of drawings largely depend on the quality of instruments. Different types of instruments and materials staring from drawing board, T- square , Mini-drafter, drawing sheet etc., have been shown along with their dimensions as recommended by BIS. The usage of various drawing instruments and layout of a drawing sheet as per Indian standards are also described in this unit.
3. In engineering drawing, the figures are drawn by using different types of lines. The different types of lines, applications and their representation in engineering drawing as recommended by the BIS and its special publications SP 46:2003 are explained comprehensively.
4. Drawing of an object may be the same size as that of the object (full size) or it may be larger or smaller than the object. The ratio of reduction or enlargement depends on the relative sizes of the object and of the sheet of paper on which the drawing is to be made. In this unit calculation of representative fraction for construction of a scale is described. Method of construction of plain and diagonal scales are explained with some numerical examples.
5. Dimensioning is a technique of describing the sizes and locations of various parts in a drawing. It is a numerical value expressed in appropriate units of measurement. It is identified on a drawing along with lines, symbols and text to define the geometrical characteristics of an object. Elements of dimensioning, Method of dimensioning as per BIS and drawing of Arrowheads are described with examples in this unit.
6. In engineering drawing technical students frequently use a number of geometrical constructions which are the application of plane geometry. So in this unit we also given more stresses on construction of geometrical drawings based on plane geometry as bisecting a line or arc, angles, dividing a line into equal number of parts, construction of angles , triangles, , square circles , regular polygon and tangency construction.

EXERCISES

A. Objective Questions

A1. Fill in the blanks with appropriate words.

1.1 When drawing, the head of the T-square is made to slide against the _____edge of the board.

1.2 Set-squares are used for drawing all _____ lines except the horizontal lines.

1.3 Chisel point pencil is used for _____.

1.4 The dimensions of A0 size drawing sheet as per BIS is _____ x _____ mm.

1.5 The centre line is used to indicate _____ and _____.

1.6 Section lines are usually drawn at a _____ angle.

1.7 Drawing of buildings are drawn using _____ scale

1.8 When two angles together make 90^0, they are said to be _____angle.

1.9 All dimensions are shown from a common datum in _____ dimensioning.

1.10 The extension line should extent about 3mm beyond the _____.

Answers of Fill in the Blanks Questions

1.1 working edge; 1.2 straight; 1.3 long thin lines; 1.4 841x1189 mm; 1.5 axis of a cylindrical and spherical feature; 1.6 45^0; 1.7 reducing scale; 1.8 complementary angle; 1.9 parallel; 1.10 dimension line

A2. Multiple Choice Questions

1.1 The compass is used for drawing

(a) perspective (b) square (c) circle (d) triangle

1.2 A Mini drafter is used for drawing

(a) smooth and irregular curves (b) parallel and perpendicular lines

(c) free hand lines only (d) vertical lines only

1.3 A French curve is used to draw

(a) polygons (b) circles (c) triangles (d) smooth freeform curves

1.4 Centre lines are drawn as

(a) continuous narrow lines (b) dashed narrow lines

(c) long-dashed dotted narrow lines (d) long dashed double dotted narrow lines

1.5 Which of the following latest standard is used for line conventions according to Bureau of Indian Standards?

(a) SP46 (b) ISD609 (c) IS696 (d) ISO9000

1.6 A long dashed double dotted narrow line is used to represent
(a) centre line (b) border line (c) centroidal line (d) hidden line

1.7 According to BIS, dimension lines, hatching and extension lines are drawn as
(a) continuous wide line (b) continuous thin line
(c) continuous extra wide line (d) continuous free hand lines

1.8 Representative fraction in a scale is ratio of
(a) (Length of a line in actual)/(length of line in the drawing)
(b) (Length of a line in drawing)/(Actual length of line on the object)
(c) (Length of a line in drawing)/(length of line in isometric drawing)
(d) Length of a line on drawing X actual length of line

1.9 The unit of R.F. is
(a) cubic centimeter (b) square centimeter (c) centimeter (d) unit less

1.10 A 2 cm long line on a drawing represents a distance of 70 cm, R.F. of the drawing is
(a) 1/70 (b) 1/35 (c) 35 (d) (1/35)1/2

1.11 If the area of drawing of roof of a house is 1/100 of actual area of roof then the representative fraction of drawing is
(a) 1/100 (b) 100/1 (c) 1/10000 (d) 1/10

1.12 A solid of volume 125 m3 is represented by a similar solid model of volume 1 cm^3. Scale factor for the scale is
(a) 1:12500 (b) 1:500 (c) 1:125 (d) 1:2500

1.13 When measurements are required in three consecutive units, the appropriate scale is
(a) plain scale (b) diagonal scale (c) isometric scale (d) scale of chords

1.14 The dimension for diameter of a circle should be
(a) preceded by the symbol 'ϕ' (b) suffixed by the symbol 'ϕ'
(c) preceded by the symbol 'D' (d) suffixed by the symbol 'D'

1.15 Which of the following is not a specified method for dimensioning?
(a) chain dimensioning (b) perpendicular dimensioning
(c) parallel dimensioning (d) progressive dimensioning

Answers of Multiple Choice Questions

1.1 (c); 1.2 (b); 1.3 (d); 1.4 (c); 1.5 (a); 1.6 (c); 1.7 (b); 1.8 (b); 1.9 (d); 1.10 (b); 1.11 (d); 1.12. (b); 1.13 (b); 1.14 (a); 1.15 (b)

B. Subjective Questions

1.1 Why is drawing called the language of engineers?

1.2 List the different types of drawing instruments.

1.3 List the standard sizes of the drawing boards according to BIS. Which size of drawing board is suitable for the practice of engineering students?

1.4 Name the angles that can be draw with the help of a pair of set squares.

1.5 How are pencil graded in an engineering drawing work? Describe shapes of the pencil lead along with their field of applications.

1.6 List the standard sizes of the drawing sheets according to BIS. Which size is suitable for technical drawing?

1.7 Explain the procedure to draw parallel lines drawn with the help of a set square? Explain with a neat sketch.

1.8 Explain the use of compass and dividers.

1.9 Explain layout of Drawing sheet with a neat sketch.

1.10 Specify the information that can be included in the title block of a drawing sheet?

1.11 State the various types of lines and their general application?

1.12 Differentiate between the meaning of continuous thick and dashed thin lines.

1.13 Distinguish among a full size, a reduced size, and an enlarged size drawing.

1.14 Compare engineer's scale with graphical scale.

1.15 Explain with a neat sketch of dimensioning of different sizes of circles.

1.16 Differentiate between aligned and unidirectional dimensioning system.

1.17 Differentiate between chain and parallel dimensioning.

PRACTICALS

As per curriculum the practicals related to Unit-1 are:

1. Draw horizontal, vertical, 30^0, 45^0, 60^0 and 75^0 lines, different types of lines, dimensioning styles using Tee and Set squares/ drafter. (do this exercise in sketch book)
2. Write alphabets and numerical (Vertical only) (do this exercise in sketch book)
3. Draw regular geometric constructions and redraw the given figure (do this exercise in sketch book) Part I.
4. Draw regular geometric construction and redraw the given figure (do this exercise in sketch book) Part II.

Practical-1 Draw Lines and Dimensioning

Practical statement

Draw horizontal, Vertical, 30 degree, 45 degree, 60 and 75 degrees lines, different types of lines, dimensioning styles using Tee and Set squares/ drafter.

Practical significance

Usually complex drawings are created by combining and modifying several different basic primitive shapes, such as lines, circles, arcs, polygons etc.

Relevant theory

Refer Section 1.1.3 (Use of T-Square, Set-Squares and Drafter)
Refer Section 1.2 (Types of Lines)
Refer Section 1.4 (Dimensioning styles)

Practical outcomes (PrO)

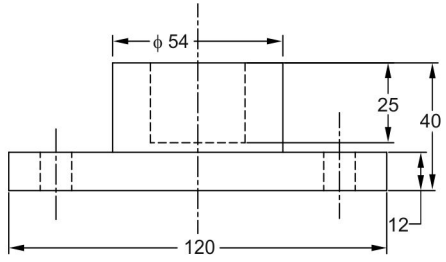
The practical outcomes are derived from the curriculum of this course:

PrO1 Use T-Square and Set-Squares to draw lines at different orientations.

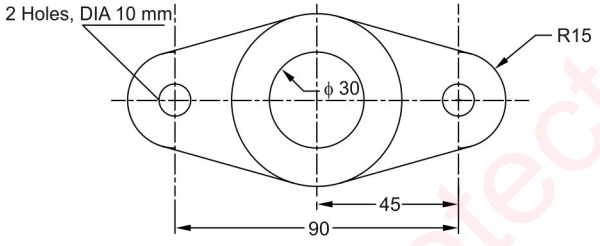
PrO2 Use appropriate line and dimension style.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

1. Draw horizontal, vertical, 30 degree, 45 degree, 60 and 75 degrees lines using Tee and Set squares/ drafter.
2. Draw different types of lines using T and Set squares/ drafter.
3. Draw different dimensioning styles.
4. Draw the following figure in sketch book:



Front View



Top View

Resources required

Sr. No	Suggested Resources Required Machines/Tools/ Instruments with Vital specifications	Qty	Actual Resources used Machines/Tools/ Instruments with broad specifications (to be filled by the student)		Remarks (if any)
1	Drawing Sheet / Sketch Book	1			
2	Drawing Table with Drawing Board	1			
3	Mini Drafter T-square Set squares (45^0 and 30^0-60^0) Protractor Drawing instrument box (containing set of compasses and dividers)	1 Set			
4	Pencils and Pencil leads(H, HB, B)	3			
5	Insulation Electric Tap	1N.			

Precautions

1. Fix the drawing sheet and drafter properly on the Drawing board.
2. Use appropriate type of pencil based on the type of line use.
3. Minimize the use of eraser.
4. While drawing keep a hanky/cloth under your hands to avoid patching on the sheet.
5. Don't use blades or paper cutters to sharpen the Pencils.

Suggested procedure

1. As outlined in section 1.1.3 of the book. (Use of T-Square, Set-Squares and Drafter)
2. As outlined in section 1.2 of the book. (Types of Lines)
3. As outlined in section 1.4 of the book. (Dimensioning Styles)
4. As outlined in the solution of problem 1.4

Observations and calculations

As shown in 1.13 to 1.17, Solution of Problem 1.4 of the book

Practical related questions

1. Explain the use of set squares.
2. Which method and combination of setsquares and T-squares should be used to create a perpendicular line to the given line?
3. Explain how Centre lines are drawn.

Disposal of waste

Classify the waste materials to be thrown in this experiment in the following bins:

Type of Waste	Colour of Bins	Name of item
Biodegradable waste	Green bin	Five-year old drawing sheets completed form both the sides. Pencil shavings during sharpening
e-Waste	Black bin	
Plastic and metal waste	Blue bin	
Any Other		

Environment friendly approach: Reuse, Reduce and Recycle

Reuse: The drawing sheets should be used from both the sides.

Reduce: Depending on the number of exercises appropriate size of drawing sheet should be used to reduce wastage in the form of empty space.

Recycle: Five-year old drawing sheets can be recycled.

Suggested learning resources

Refer Unit-1

Suggested assessment scheme

(to be filled by teacher)

The given performance indicators should serve as a guideline for assessment regarding process and product related marks.

Performance indicators		Weightage/ Marks	Marks Awarded
Process related: 5 Marks* (33%)			
1	Fixing the drawing sheet properly	1	
2	Use of appropriate pencil and instrument while drawing different entities	2	
3	Not taking support of others to draw given problem(s)	2	
Product related: 10 Marks* (67%)			
4	Neatness, Cleanliness on drawing sheet/sketch book	1	
5	Dimensional accuracy of the output	1	
6	Uniformity in drawing and line work	1	
7	Creating given drawing with all problems solved	5	
8	Dimensioning the given drawing and writing text	1	
9	Submission of drawing in time	1	
	Total	**100 %**	

* Marks and percentage weightages for product and process assessment will be decided by the teacher.

Name of the Student: …….. **Marks Awarded**			**Signature of Teacher with date**
Process Related	Product Related	Total	

Practical-2 Write Alphabets and Numerical (Vertical Only)

Practical statement

Print 10 mm single stroke capital letters and numerals in vertical style using either scale or setsquare and by free hand.

Practical significance

Apart from graphical elements (lines, arcs, circles etc) technical drawings will also contain written information. This written information is referred as "lettering". While writing annotations like Text, Labels, Dimensions, Part list, Symbols, leaders, balloons etc., Alphabets and Numbers are to be hand written during manual drawing as per BIS SP:46-2003.

Relevant theory

Size of letters

- Size of Letters is measured by the height h of the CAPITAL letters as well as numerals. The standard heights recommended by BIS SP: 46-2003 are in the progressive ratio of "square root 2". They are namely 2.5 - 3.5 - 5 - 7 - 10 – 14 and 20 mm. The height of lower case letter (without tail or stem) are 2.5, 3.5, 5, 7, 10 and 14 mm. Size of the letters may be selected based upon the size of drawing.

Procedure for lettering

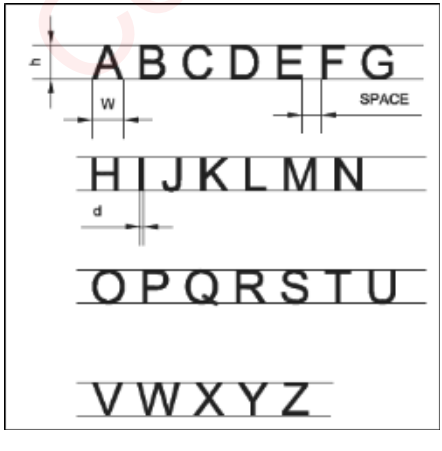
- Thin horizontal guide lines are drawn first at a distance ' h' apart.
- Horizontal lines of the letters are drawn from left to right. Vertical, Inclined and curved lines are drawn from top to bottom.
- After completing the lettering the guidelines should not be erased.

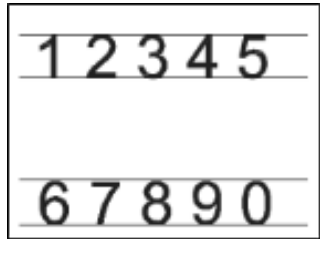
Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

PrO1 Write text and numbers on the given 2D entity.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)





Resources required

Same as mentioned in Practical-1

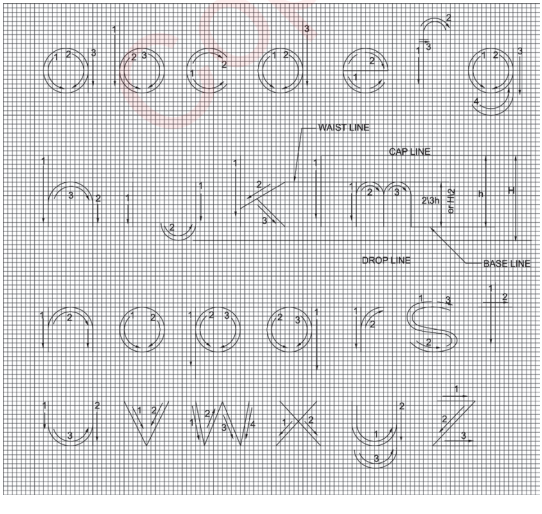
Precautions

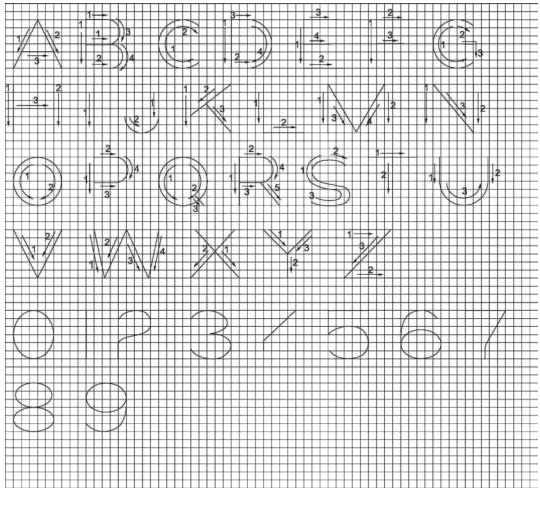
Same as mentioned in Practical-1

Suggested procedure

1. Draw horizontal thin parallel lines (guide lines) of 10 mm distance. h = 10 mm distances denotes the height of the letter.
2. Mark the width of the letters recommended by BIS (IS:9609-1983). The width of different letters in terms of `d' is as follows. d indicates stroke thickness and is equal to h/10. For example, as per following table to write J the width should be (10/10 x4) =4mm
3. For curved letters use smooth free hand curve.
4. Print straight line letters using either scale or setsquares.
5. To maintain the uniform thickness of line, use conical point soft grade pencil and avoid too much of sharpness (HB).
6. Guidelines of both top and bottom should always be drawn with sharp pencil (2H).

Width (w)	Capital letters
1	I
4	J
5	C, E, F, L
6	B,D,G,H,K,N,O,P,R,S,T,U & Z
7	A , M , Q , V , X , Y
9	W





Observations

Practical related questions

1. State the values of Height and Width of letters J and I if the guide lines are 10 mm apart.
2. State the values of Height and Width of numbers 6 and 1 if the guide lines are 10 mm apart.

Disposal of waste

Same as mentioned in Practical-1

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1

Suggested learning resources

Refer Unit-1

Suggested assessment scheme

Same as mentioned in Practical-1

Practical-3 Draw Regular Geometric Constructions-I

Practical statement

Draw regular geometric constructions. In a situation two pulleys are connected through a belt (Open belt arrangement), draw the setup with circles representing the pulleys and their external tangents representing the sides of the belt.

Practical significance

Complex 2D drawings are created using primitive shapes like Circles, Rectangles, Triangles, Polygons, and Ellipses etc. Hence student must know how to draw these geometrical shapes and their tangents.

Relevant theory

Refer Section 1.5 of the book.

Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

PrO1 Draw regular geometric constructions and their tangents.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

Diameter of larger Pulley = 40mm
Diameter of smaller Pulley = 30mm

Center distance between the pulleys = 60 mm

Resources required

Same as mentioned in Practical-1

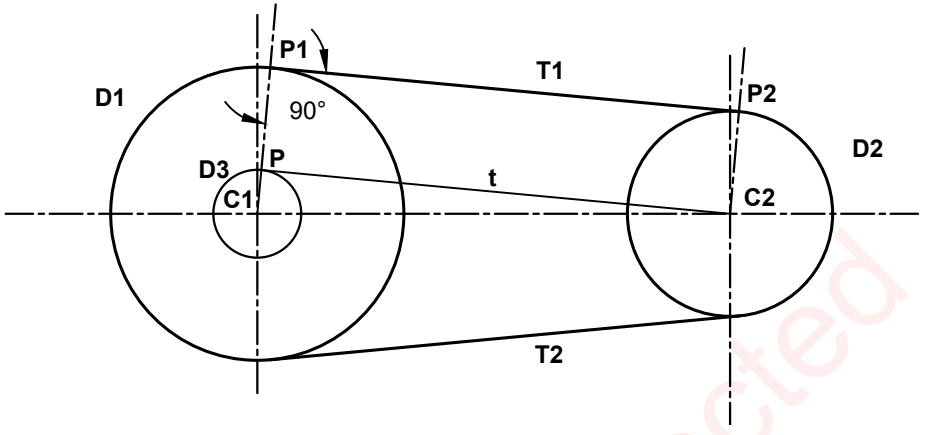
Precautions

Same as mentioned in Practical-1

Suggested procedure

1. Draw a line of 60 mm and and mark two end points as C1 and C2.
2. With centre C1 and C2 draw two circles of diameters 40mm and 30mm respectively.
3. Draw a concentric circle (D3) of dia 10 mm (40mm – 30mm) on 40mm circle (D1).
4. From centre C2 draw a line 't' touching circle D3 at P.
5. Join C1 and P (angle P is right angle) and extend it up to the circle D1 meeting at P1.
6. Draw C2-P2 parallel to C1-P1.
7. Join P1 and P2 forming the (common) tangent T1 to circles D1 and D2.
8. Similarly draw tangent T2. Tangents T1 and T2 are called external tangents.

Observations



Practical related questions

1. Identify the angle of wrap on the larger pulley.
2. If it is a cross belt situation how would be your approach to draw the tangents.

Disposal of waste

Same as mentioned in Practical-1

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1

Suggested learning resources

Refer Unit-1

Suggested assessment scheme

Same as mentioned in Practical-1

Practical-4 Draw Regular Geometric Constructions-II

Practical statement

Draw regular geometric constructions like a pentagon with a given length of side.

Practical significance

Complex 2D drawings are created using primitive shapes like Circles, Rectangles, Triangles, Polygons, and Ellipses etc. Hence student must know how to draw these geometrical shapes and their tangents.

Relevant theory

Refer Section 1.5 of the book.

Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

PrO1 Draw regular geometric constructions and their tangents.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

Regular Pentagon with 50mm side is given. One side of the Pentagon should be horizontal. Also draw an inscribed circle in the Pentagon drawn.

Resources required

Same as mentioned in Practical-1

Precautions

Same as mentioned in Practical-1

Suggested procedure

Method-1

1. Draw a line AB equal to the given length of side (50mm).
2. Bisect AB at P.
3. Draw a line BQ equal to AB in length and perpendicular to AB.
4. With centre P and radius PQ, draw an arc intersecting AB produced at R. AR is equal to the diagonal length of the pentagon.
5. With centers A and B and radii AR and AB respectively draw arcs intersecting at C.
6. With centers A and B and radius AR draw arcs intersecting at D.
7. With centers A and B and radii AB and AR respectively draw arcs intersecting at E.
8. ABCDE is the required pentagon.

Method-2

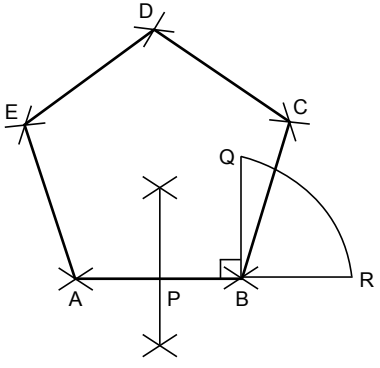
1. Draw a line AB equal to the length of the given side (60mm).
2. Draw a line B1 such that <AB 1 = 108° (included angle)
3. Mark C on Bl such that BC = AB

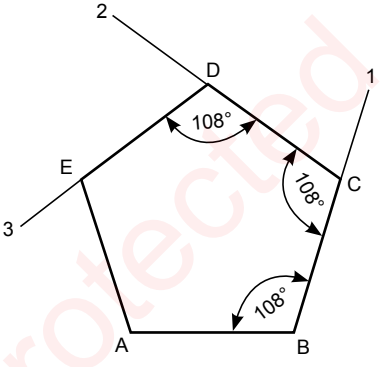
4. Repeat steps 2 and 3 and complete the pentagon ABCDE

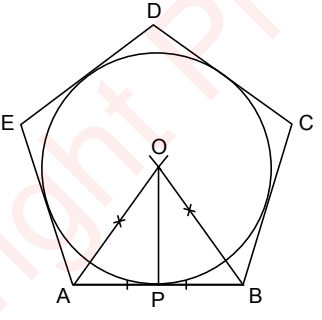
Inscribed Circle

1. Bisect any two angles by lines intersecting each other at O.
2. From O, draw a perpendicular to any one side of the pentagon cutting it at P.
3. With centre O and radius OP, draw the required circle.

Observations







Practical related questions

1. How included angle was estimated in Method-2.
2. If a circumscribed circle is required, then what should be the approach.

Disposal of waste

Same as mentioned in Practical-1

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1

Suggested learning resources

Refer Unit-1

Suggested assessment scheme

Same as mentioned in Practical-1

DESIGN INNOVATIVE PRACTICALS/PROJECTS/ ACTIVITIES

Student shall form a group of 5-6 students and undertake one or two micro project(s)/activity(ies) under the guidance of faculty and present it as group with individual participation as well. A sample list is given below:

1 Draw parallel lines at 30° with horizontal by using T-square and set-squares.
2 Draw vertical lines, when one of the lines makes 15°, by using T-square and set-square.
3 Draw a line AB 70 mm long. Divide the line in to four equal parts by using the divider.
4 Construct the square by using a 45° set-square.
5 You have an A2 size drawing sheet and you want to make plan of a house with plot area 30 feet by 50 feet on it. What value of R.F you will use?
6 Construct a graphical scale to measure a distance up to 50 feet. Take R.F value as 1/100
   a. Least count should be 1 foot
   b. Least count should be 1 inch
7 Draw a line 119 mm long at any angle and show the proper procedure for bisecting it.
8 Draw a line 150 mm long. Using the drafting tools, divide the line into 5 equal parts.
9 Construct a 45^0 angle with 75 mm sides. Bisect this angle.
10 Construct an angle of 15^0 by using compass and show the proper procedure.
11 Construct an equilateral triangle of 60 mm height.
12 Construct a triangle having the base 50 mm, altitude 20 mm and one side 40 mm.
13 Construct a square by measuring only 60 mm side.
14 Through three points A, B and C, draw a circle where AB = 30 mm and BC = 20 mm.
15 From a point outside a circle of 100 mm diameter, draw two tangents containing an angle of 60^0.
16 Draw a circle about a triangle of sides 60 mm, 35 mm and 55 mm.
17 Draw two circles 50 mm diameter and 40 mm diameter respectively with centre 60 mm apart. Draw open-belt tangents to the circles.
18 Draw a hexagon within a circle of radius 35 mm.
19 Draw an octagon when the distance across the flats is 50 mm.

KNOW MORE

- Teacher and students shall collect few production/construction/electrical/electronics actual industrial component drawings.
- Teacher should download the free software available to demonstrate generation of drawing with virtual drawing instruments during class room sessions.
- During the input sessions the teacher must give examples related to the relevant branch viz.

mechanical and allied disciplines/electrical and allied disciplines/electronics etc.

- Show video/animation films to explain mentioned concepts.
- Use charts and industrial drawing/drawing sheets developed by experienced faculty to teach standard symbols and current industrial/teaching practices.
- Teachers should ask students to use url/qr codes available in the book for further understanding/practicing of the concepts.

Applications (Real life / Industrial)

1. *Drawing instruments* are used to prepare neat and accurate drawings viz. civil drawings, architectural drawings, structural drawings, mechanical systems drawings, electrical drawings, plumbing drawings etc.
2. *Different types of lines* used in construction of figures, geometrical shapes and objects, civil construction drawings, mechanical engineering drawings etc. In mechanical engineering drawings, lines types are used to represent the outlines of adjacent parts, extreme positions of movable parts in the assembly drawings. In building construction works, continuous narrow lines are used to represent doors, windows, stairs, ramps and sloping areas.
3. *Geometric constructions* are used while drawing components/features in mechanical drawing, civil drawing, production drawing, construction drawing, architectural drawing etc.; Head of the bolts; Nuts; Gear tooth; Sprocket tooth; Cams; Keyways; Specially designed doors and windows; Water bodies; Interior; Textile drawings.
4. In engineering drawing as soon as you start drawing any component whether it is from mechanical, electrical, civil, electronics etc., the very first thing is to choose a *scale*. Hence no object, element and component domestic or industrial can be drawn without scale.
5. *Dimensioning* is used to give information of any 2D and 3D entity on drawing sheet. Dimensioning is used in Engineering drawing, Machine drawing, Constructional drawing, Electrical wiring layout drawing, Electronics component and placement drawing, Production drawing, Orthographic views, isometric views etc. Dimensioning is also used to provide information of size related to components and features like-through hole, blind hole, stepped hole, counter dilled hole, countersunk hole, spot faced, thickness, height, length, width, radius, diameter, arc, chord length, curves, angle, repeated features, pitch length, taper, chamfer, fillet, restricted areas, undercut, slots, groove, screw, nut, stud, rivet, bolt, washer, seal, key, shaft, axel, stud, rod, lever, couplings, gears, chains, bearings, belts, doors, windows, truss, circuit layout, plumbing line, etc.

Inquisitiveness and Curiosity

Other than the classroom and drawing practice sessions , following are the suggested student-related co-curricular activities which can be undertaken to accelerate the attainment of the various outcomes in this unit:

1. Students should collect Production drawings, Building Drawings, Layouts from nearby workshops/industries/builders/contractors and try to
   a. redraw types of lines used
   b. redraw lettering styles used

c. list BIS code referred and the symbols/annotations/dimensioning used

d. list the type of scales used. Compare the size of component on drawing sheet with actual component.

e. redraw the same with the help of basic drawing instruments and tools such as Minidafter, T-square, Compass, Set-squares , French curve etc.

2. Students should collect some tourist map and try to find the distance between any two locations with the help of graphical scale drawn on it.
3. Surf the web and list the scale used in wrist watch. Observe the scale used for a drawing a hostel building, hospital building, 2BHK flat. Observe the scale used for a drawing a typical connecting rod of a bike. Observe a scale used for the drawing of integrated circuit.
4. Each student should explain at least one problem of geometrical construction to other students of his class in presence of course teacher.
5. Student should maintain a separate A3 size sketch book which will be the part of term work and submit it along with drawing sheets. Draw minimum 5 problems related to geometrical construction in the sketch book (The problems on sheet and in the sketch book should be different).
6. Student should identify and list at least five components related to their engineering branch where they have seen the used of geometrical construction

REFERENCES & SUGGESTED READINGS

1. Engineering Drawing Practices for School and Colleges SP 46:2003, published by Bureau of Indian Standards, Government of India, Third Reprint, October 1998; ISBN:. 81-7061-091-2, Manak Bhavan , 9 Bahadur Shah Zafar Marg , New Delhi
2. A work book of Engineering Drawing by T.T.T.I., Bhopal, 1999
3. A text Book of Engineering Drawing, K Venkata Reddy, BS Publication, 2008
4. Bureau of Indian Standards: https://en.wikipedia.org/wiki/Bureau_of_Indian_Standards
5. Use of Virtual drawing instruments: https://www.triumphcloud.com/
6. Virtual drawing environment: https://www.mathspad.co.uk/

2 Orthographic Projections

UNIT SPECIFICS

- Concept and Applications of orthographic, perspective, isometric and oblique projections.
- Orthographic projection, First angle and Third angle method, their symbols.
- Axonometric projection
- Oblique projection
- Perspective projection
- Conversion of pictorial view into orthographic views.

The understanding of these topics will be developed after reading the contents, completing the solved problems, activities, performing exercises and viewing the ICT and web resources mentioned in this unit.

At the end of the unit, summary is provided to recapitulate the topics covered and applications are mentioned so that the learner can correlate the presented knowledge with real life and industrial situations. Few activities are mentioned to create inquisitiveness and curiosity in the learner. Subjective and objective questions are provided for the reinforcement of the knowledge and a list of references and suggested readings is also provided for further learning. Video resources along with QR codes are mentioned for getting more information on various topics of interest which can be surfed or scanned through mobile phone for viewing. Practical with details are provided to horn the drawing and drafting skills.

RATIONALE

When you desire to make drawing of an object, then the first decision that you have to take is, what type of drawing you want to make? If you are communicating shape and size of an object to a layman, you make a pictorial drawing show three dimensional image of object. But if you are communicating with technical person, then in addition to pictorial drawing, you may even go for an orthographic drawing and sectional orthographic drawing. The advantage of orthographic drawing and sectional orthographic drawing is you can show details more clearly and precisely.

PRE-REQUISITES

Before reading this unit the student is advised to revisit following:

- Maths: coordinate and plane geometry
- Unit-1 of this book

UNIT OUTCOMES

After reading the contents, completing the solved problems, activities, exercises and viewing the ICT and web resources mention in this unit the students are expected to:

U2-O1: Explain the concept and applications of orthographic, perspective, isometric and oblique Projections

U2-O2: Differentiate between First angle and Third angle projection methods alongwith their symbol specified by BIS

U2-O3: Draw the orthographic views in First Angle Projection from given pictorial views of object containing plain surfaces, slanting surfaces, slots, ribs, cylindrical surfaces.

2.1 CONCEPT AND APPLICATIONS OF ORTHOGRAPHIC, PERSPECTIVE,

Unit-2 Outcome	EXPECTED MAPPING WITH COURSE OUTCOMES *(1- Weak Correlation; 2- Medium Correlation; 3- Strong Correlation)*					
	CO-1	CO-2	CO-3	CO-4	CO-5	CO-6
U2-O1	3	2	3	3	2	-
U2-O2	3	3	3	2	3	2
U2-O3	3	1	2	2	2	1

ISOMETRIC AND OBLIQUE PROJECTIONS

2.1.1 Introduction to Projections

Projection means that you project image of an object on a plane. Key concepts important for understanding the meaning of word projection are:

Object

In Fig.2.1 (a) and (b) rectangle ABCD is the object. The object can be a point, line, plane, solid or a sectioned solid whose image you are trying to project.

Plane of projection

It is the plane on which you project the image of object. It is also called reference plane or picture plane.

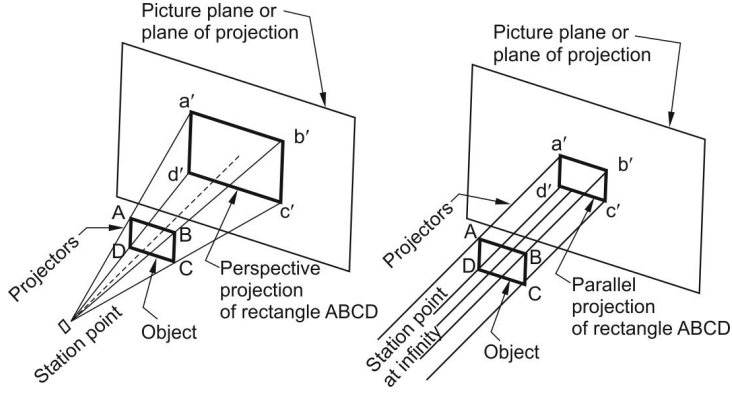


Fig.2.1: Perspective projection versus parallel projection

The station point

It is the point from which the projectors start. In perspective projection the station point is at a finite distance whereas in parallel projection the station point is at infinity.

Projectors

Projectors are the straight lines drawn from contour points or corner points of the object on the plane of projection.

Projection

When piercing point of projectors and plane of projection are joined in proper sequence you get the projection of the object.

2.1.2 Classification of Projection Methods

Projection methods are classified into two broad categories

1. Perspective projection {Fig.2.1(a)}
2. Parallel projection {Fig.2.1(b)}

Perspective projection

Fig.2.1 gives a comparative pictorial description of parallel projection and perspective projection. Fig.2.1(a) shows the concept of perspective projection. Observe that in perspective projection the projectors converge to a point called station point. Fig.2.1(b) shows the concept of parallel projection. Observe that in parallel projection, projectors are parallel to each other i.e., the station point is at infinity.

Parallel projection

When parallel projectors are drawn from contour of object on the picture plane or reference plane, then figure obtained on joining piercing point of projectors and reference plane in proper sequence is called the parallel projection of the object on reference plane. Parallel projection is classified into two subcategories

1.Oblique projection {Fig.2.2 (a)}

2.Orthographic projection {Fig.2.2 (b)}

The key differentiating point is, in oblique projection projectors are inclined to the plane of projection, whereas in orthographic projection, the projectors are always perpendicular to the plane of projection.

Fig.2.3 gives detailed classification of different methods of making a drawing. Except multiview projection all other drawings are pictorial drawing.

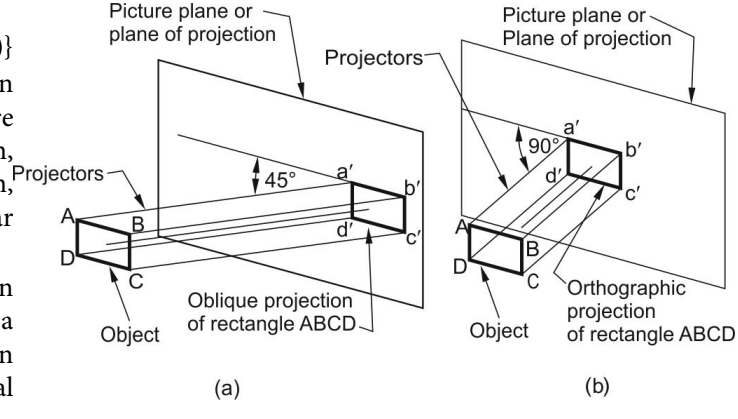


Fig.2.2: Oblique projection versus orthographic projection

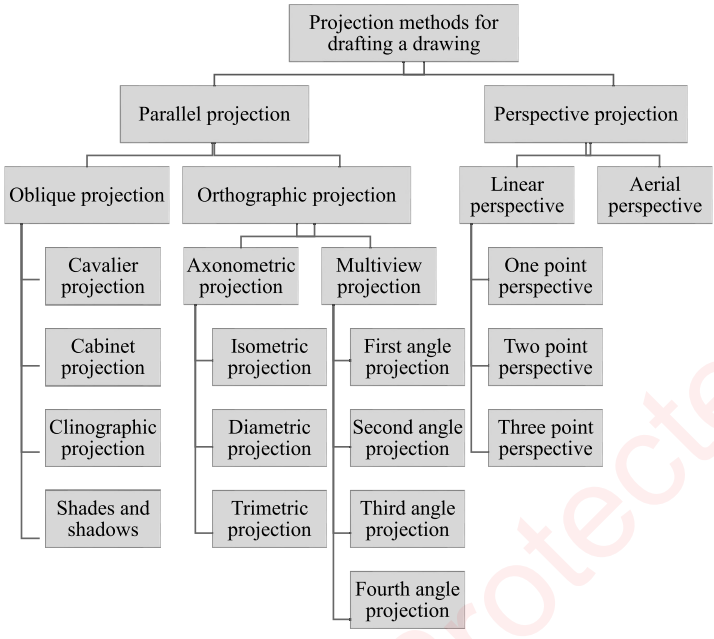


Fig.2.3: Classification of projection methods

2.2 ORTHOGRAPHIC PROJECTION

The only two necessary and sufficient condition for orthographic projection are

1. Projectors should be parallel to each other and
2 Projectors should be perpendicular to plane of projection

Fig.2.4 (a) shows pictorial view of 3D arrangement for orthographic projection of a solid. The face of solid containing maximum information is kept parallel to the plane of projection. Fig. 2.4(b) shows separately the orthographic projection of solid placed as shown in Fig. 2.4(a). Notice that orthographic

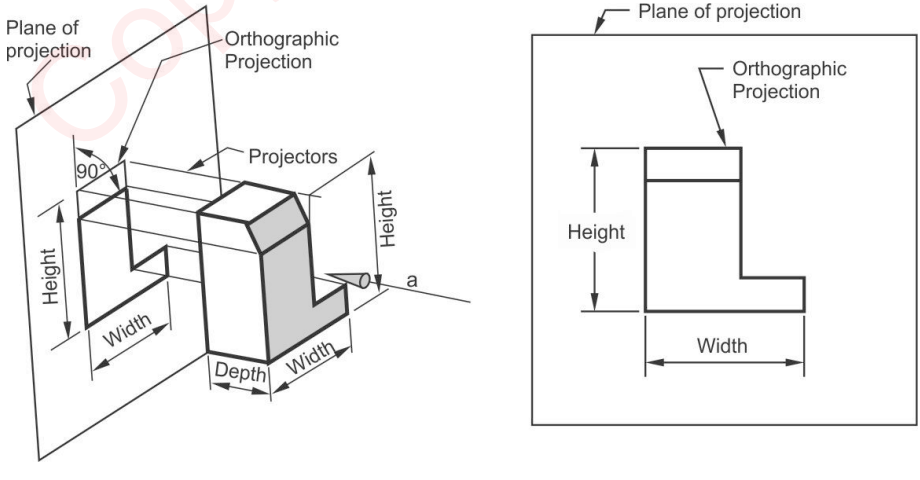


Fig.2.4: Pictorial view of 3D arrangement for orthographic projection of a solid

projection obtained is giving information only about width and height of solid but it is giving no information about the depth of solid. Hence more orthographic views required.

Fig.2.5 (a) shows orthographic projection of a solid when all three mutually perpendicular coordinate axis of solid are equally inclined to the plane of projection. The angle of X, Y and Z axis of solid with plane of projection is 36.46°. Such type of orthographic projection is called isometric projection and it will be discussed in full details in chapter 3. Dashed line made in Fig.2.5 (b) represent invisible lines from direction of looking. General practice is to avoid dashed lines in isometric drawing.

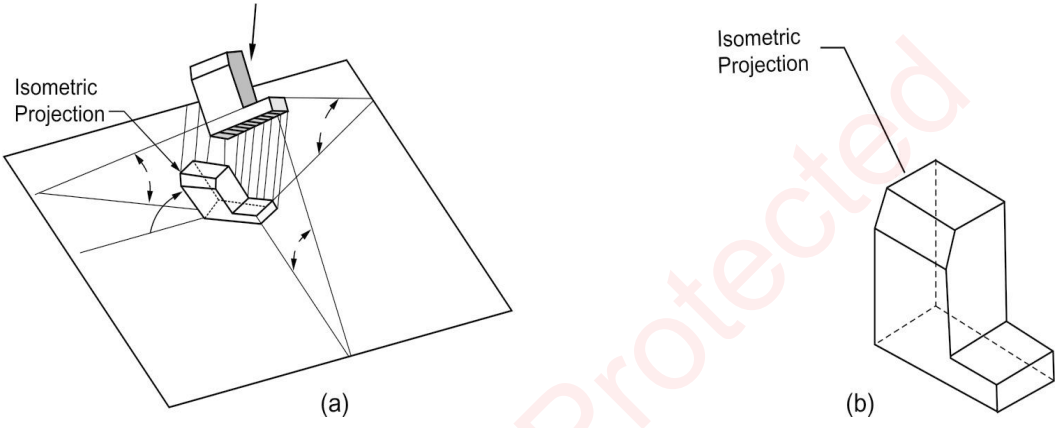


Fig.2.5: (a) Orthographic projection of a solid when all three mutually perpendicular coordinate axis of solid are equally inclined to the plane of projection and (b) Isometric projection

2.2.1 Multiview Orthographic Projection of Solid

In Fig.2.4(a) orthographic projection of the solid is giving information only about the width and height of solid. For information about depth another orthographic view showing depth is required. Hence the need of multiview orthographic projection. In multiview orthographic projection you project multiple orthographic views of the solid from different directions.

In multiview orthographic projection (Fig.2.6), observer views the object from multiple directions i.e., the projectors are drawn along different directions 'a', 'b', 'c', 'd', 'e' & 'f' and view projected on respective projection planes. As shown in Fig.2.8 the most informative view, of the object to be represented is normally chosen as the principal view (front view). This view is named 'A' as the direction of viewing is 'a'. Plane of solid containing maximum information is kept perpendicular to 'a' direction of projectors.

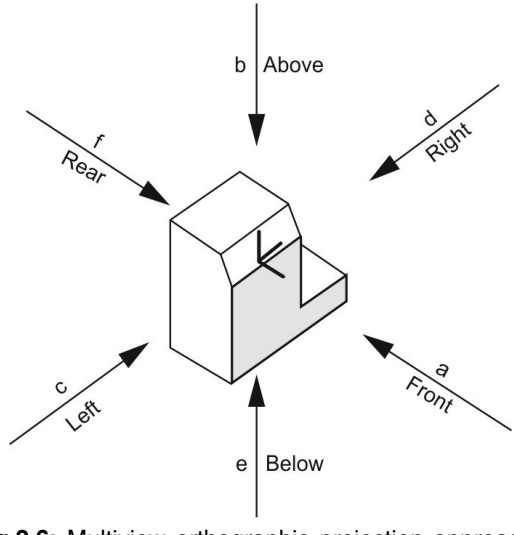


Fig.2.6: Multiview orthographic projection approach

As per IS sp-46 the designation of views is as given in table 2.1.

Table 2.1: Designation of views

Direction of observation	Designation of view
View from direction 'a' is called view from front	A
View from direction 'b' is called view from above	B
View from direction 'c' is called view from left	C
View from direction 'd' is called view from right	D
View from direction 'e' is called view from bottom	E
View from direction 'f' is called view from rear	F

2.2.2 Concept of Four Quadrant

To get complete information about shape and size of an object, orthographic projection of an object are taken on at least two mutually perpendicular planes. Fig.2.7 illustrates the four quadrants or dihedral angle.

1. A quadrant consists of two principal planes of projection inclined to each other at an angle of 90°.
2. The vertical reference plane V.P. and the horizontal reference plane H.P. divides the space in four regions. Once front direction of looking 'a' and top direction of looking 'b' are mentioned then we can define first quadrant, second quadrant, third quadrant and fourth quadrant as shown in Fig.2.7.
3. First quadrant is region of space above H.P. and in front of V.P.
4. Second quadrant is region of space above H.P and behind V.P.

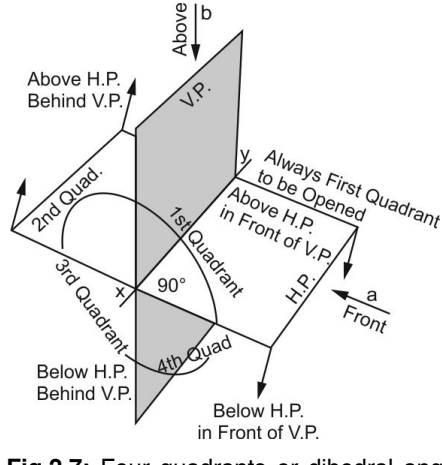


Fig.2.7: Four quadrants or dihedral angle

5. Third quadrant is region of space below H.P and behind V.P.
6. Fourth quadrant is region of space below H.P and in front of V.P.

View from front or elevation or front view

Fig.2.8 shows the pictorial arrangement for obtaining view from front or elevation or front view. In multiview orthographic projection, to obtain view from front of the solid, you keep most informative face of solid parallel to the vertical reference plane V.P. Direction of projector is along direction 'a'. Piercing point of projectors and V.P. are joined in proper sequence to obtain view from front of

the solid. Note that in Fig.2.8 view from front is giving information about width and height of solid model but it is not giving information about depth of solid model.

View from above or plan or top view

Fig.2.9 shows the pictorial view of 3D arrangement for obtaining view from above or plan or top view of the solid model. Direction of projector is along direction 'b' and the projection plane H.P. is perpendicular to the direction 'b'. Note that depth and width of the solid model are visible in view from above but height is not visible.

In Fig.2.10 the flat chamfer surface is replaced by a vertical and horizontal surface but still then both view from above and view from front are same. That means only view from above and view front are not able to explain the shape and size of solid model shown, view from left or right is also required.

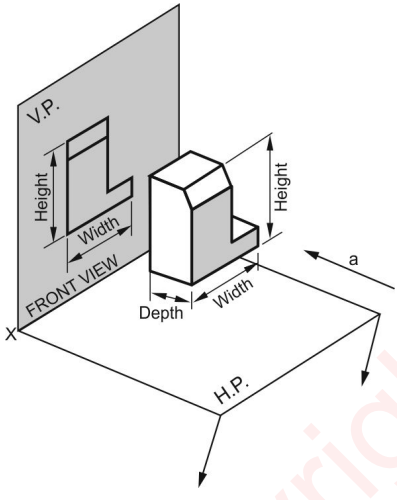


Fig.2.8: Pictorial view of 3D object with chamfer for obtaining view from front

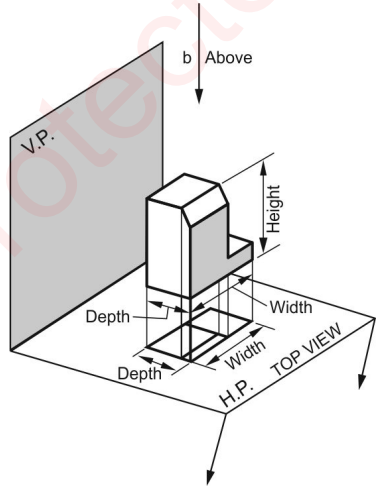


Fig.2.9: Pictorial view of 3D object with chamfer for obtaining view from above

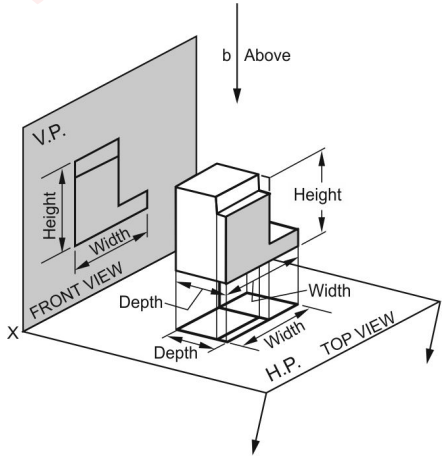


Fig.2.10: Pictorial view of 3D object with a step for obtaining top view

View from left

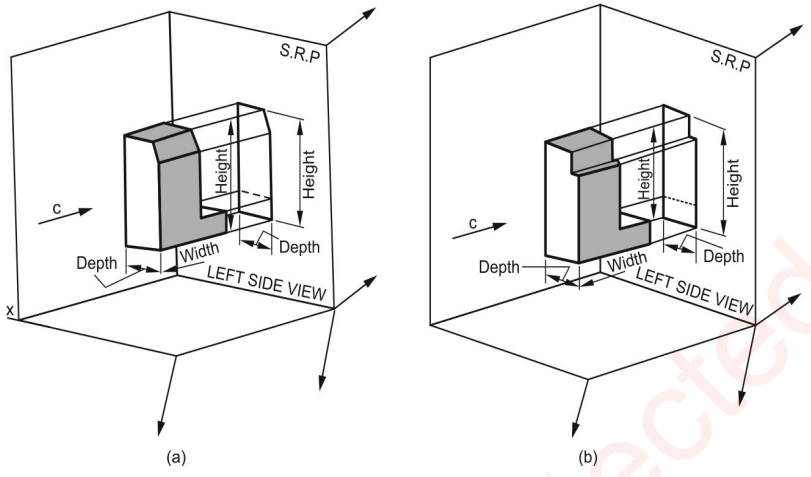


Fig.2.11: (a) Pictorial arrangement for obtaining the left side view of the solid with chamfer
(b) Pictorial arrangement for obtaining the left side view of the solid with step

Fig.2.11(a) gives the pictorial arrangement for obtaining the left side view of the solid. Notice that the side view gives information about the depth and height of solid model but it does not give information about width of the solid model. Depth information is also available in top view and height information is also available in front view then what is the necessity of side view? On comparing Fig.2.11(a) and Fig.2.11(b) you reach the conclusion that, to show that it is a chamfer surface and not a step side view is the required view.

2.2.3 Methods for Multiview Orthographic Projection

IS-sp 46 defines two methods for drafting multiview orthographic projections.

1. First angle projection method, and
2. Third angle projection method

First angle projection method

The key point about first angle projection method for multiview orthographic projection of a solid is that, object lies between the observer and plane of projection. Another way of thinking is, in first angle projection method object is positioned in first quadrant i.e., in front of V.P. and above H.P.

Fig.2.12 shows the pictorial drawing for object kept in first quadrant i.e., above H.P. and in front of V.P. When projectors are projected along front direction 'a' perpendicular to vertical reference plane orthographic projection is obtained on the vertical reference plane, and is called view from front or elevation. View from front or elevation gives information about width and height of the solid model. When projectors are projected along top direction 'b' perpendicular to horizontal reference plane, then orthographic projection is obtained on the horizontal reference plane and is called view from above or top. View from above gives information about width and depth of the solid model. When projectors are projected along left direction 'c' perpendicular to the side reference plane or profile plane, then orthographic projection is obtained on the profile plane and is called view from left. View from left gives information about depth and height of the solid model.

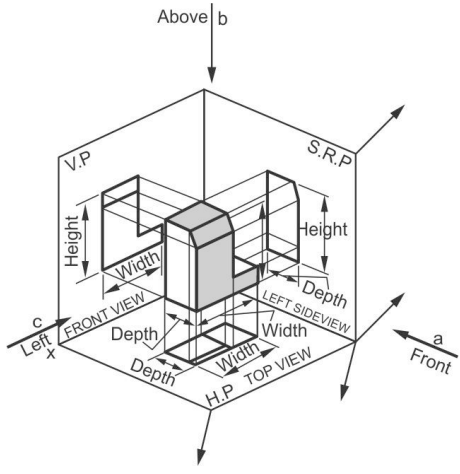


Fig.2.12: The pictorial drawing for an object kept in first quadrant

Now when you want to draw the multiple orthographic views of the solid model on a 2-dimensional plane paper you require to make all the projection planes coplanar. In first angle projection method, it is customary to revolve the H.P. about its line of intersection with vertical reference plane by an angle of 90°, so that it becomes coplanar with V.P. and comes below the vertical reference plane as shown in Fig.2.13(a). Curved arrowheads in Fig.2.12 shows the direction in which H.P. is rotated and Fig.2.13 (a) shows the pictorial drawing after H.P. is rotated. Similarly, in first angle projection method, it is customary to revolve the profile plane about its line of intersection with vertical reference plane by an angle of 90° so that it becomes coplanar with V.P. and comes at the same level as vertical reference plane. Arrowheads in Fig.2.12 shows the direction in which profile plane is rotated and Fig.2.13 (a) shows the pictorial drawing after profile plane is rotated. Fig.2.13 (b) shows final multiview orthographic projection of the solid by first angle projection method. Notice that line which are not visible are made dashed.

For this simple solid only three views are enough and there all six orthographic views are not shown.

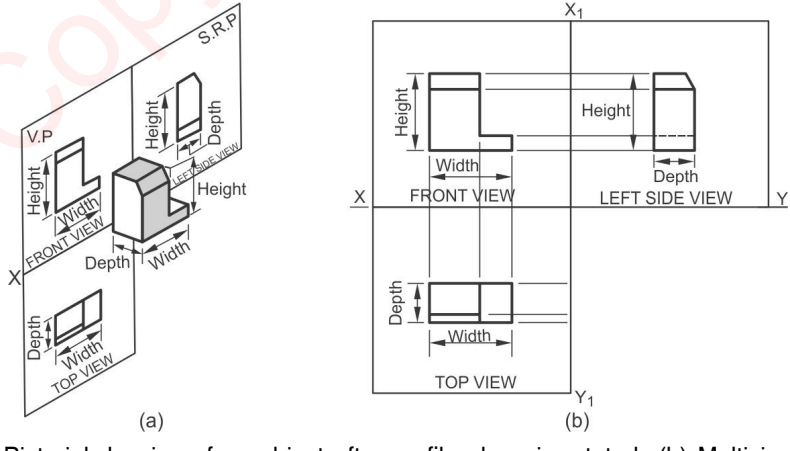


Fig.2.13: (a) Pictorial drawing of an object after profile plane is rotated (b) Multiview orthographic projection of the object by first angle projection method

Fig.2.14(a) shows the symbol for orthographic drawings made in first angle projection system. Pictorial view in Fig.2.14(b) shows that when a frustum of a cone is positioned in first quadrant such that its axis is parallel to H.P. and V.P. then its front view and left side view are used as symbol for first angle projection system.

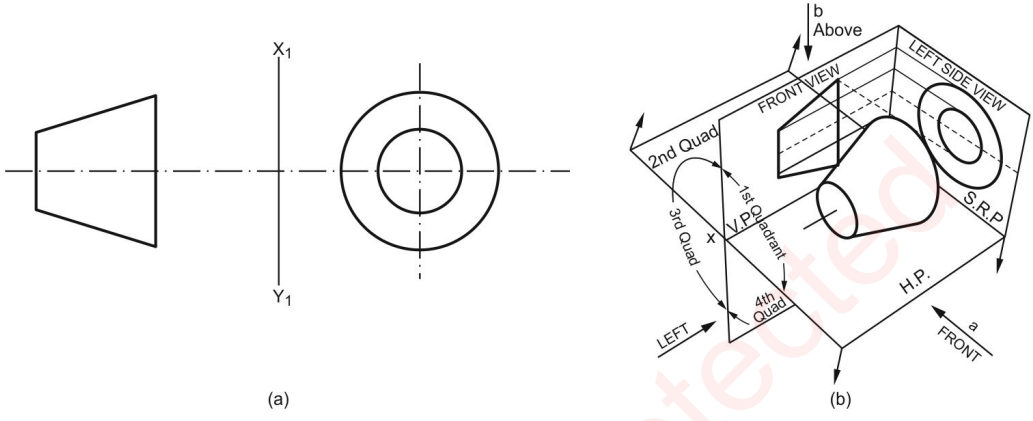


Fig.2.14: (a) Symbol for orthographic drawings made in first angle projection system (b) A frustum of a cone is placed in first quadrant with its axis parallel to H.P. and V.P.

Third angle projection method

The key point, about third angle projection method for drafting multiview orthographic projections of a solid is that, transparent plane of projection lies between the observer and object. Another way of thinking is, in third angle projection method object is positioned in third quadrant i.e., behind V.P. and below H.P.

Fig.2.15 shows the pictorial drawing for object kept in third quadrant i.e., below H.P. and behind V.P. To obtain view from front or elevation, projectors are projected along front direction **'a'** toward the contour points of the solid model and perpendicular to transparent vertical reference plane. Notice that the vertical reference plane is lying between observer and object. Front view or elevation is obtained on the vertical reference plane by joining in proper sequence piercing points of projector and vertical reference plane. View from front or elevation gives information about width and height of the solid model.

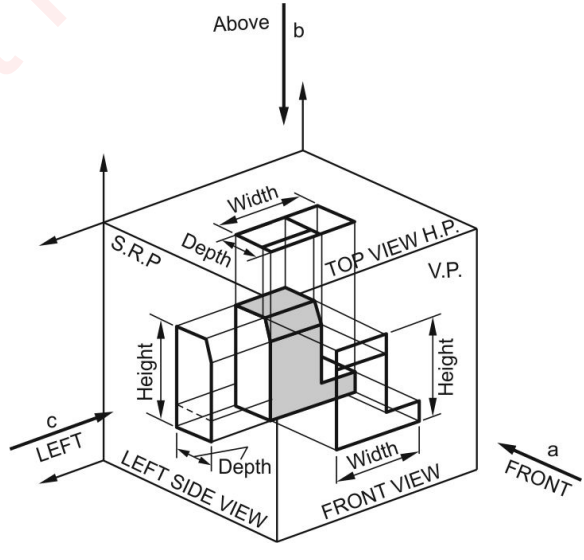


Fig.2.15: Pictorial drawing for an object kept in third quadrant

To obtain view from above or top, projectors are projected along above direction 'b' toward the contour points of the solid model and perpendicular to transparent horizontal reference plane. Notice that the horizontal reference plane is lying between observer and object. View from above

or top is obtained on the horizontal reference plane by joining in proper sequence piercing points of projector and horizontal reference plane. View from above or top gives information about width and depth of the solid model.

To obtain left side view, projectors are projected along above direction 'c' toward the contour points of the solid model and perpendicular to transparent side reference plane or profile plane. Notice that the side reference plane is lying between observer and object. View from left or plan is obtained on the side reference plane by joining in proper sequence piercing points of projector and

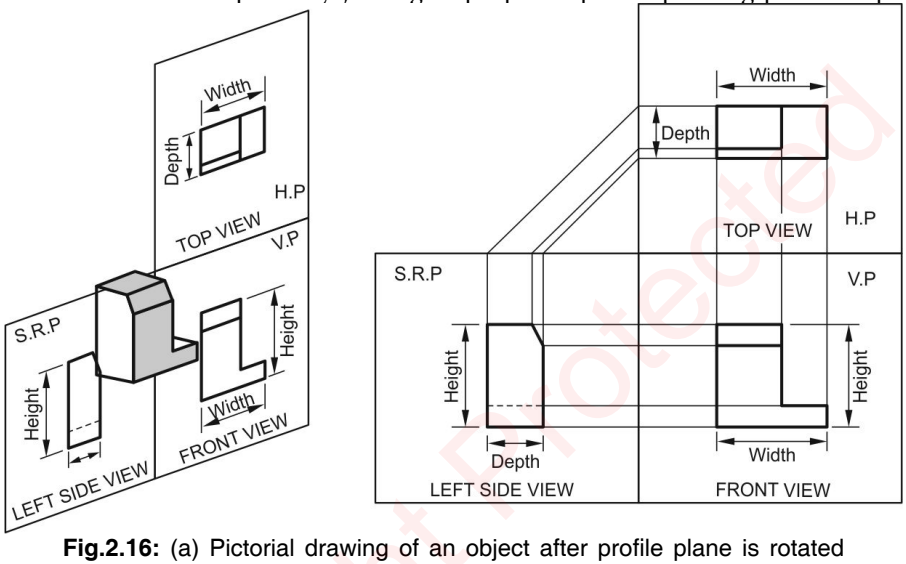


Fig.2.16: (a) Pictorial drawing of an object after profile plane is rotated
(b) Multiview orthographic projection of the object by third angle projection method

side reference plane. View from left gives information about height and depth of the solid model.

Now when you want to draw multiple orthographic views of the solid model on a 2-dimensional plane paper, you require to make all the projection planes coplanar. In third angle projection system, it is customary to revolve the H.P. about its line of intersection with vertical reference plane by an angle of 90° so that it becomes coplanar with V.P. and comes above the vertical reference plane as shown in Fig.2.16(a). Curved arrowheads in Fig.2.15 shows the direction in which H.P. is rotated and Fig.2.15(a) shows the pictorial drawing after H.P. is rotated. Similarly, in third angle project system, it is customary to revolve the profile plane about its line of intersection with vertical reference plane by an angle of 90° so that it becomes coplanar with V.P. and comes at the same level as vertical reference plane. Curved arrowheads in Fig.2.15 shows the direction in which profile plane is rotated and Fig.2.16(a) shows the pictorial drawing after profile plane is rotated. Fig.2.16 (b) shows final orthographic projection of solid by third angle projection method. Notice that line which is not visible is made dashed. For this simple solid only three views are enough and therefore all six orthographic views are not shown.

Fig.2.17(a) shows the symbol for orthographic drawings made in third angle projection system. Pictorial view in Fig.2.17(b) shows that when a frustum of a cone is positioned in third quadrant such that its axis is parallel to H.P. and V.P. then its front view and left side view are used as symbol for thired angle projection system.

Based on the above illustrations, it is quite evident that the views obtained by the two methods of projection is completely identical in shape, size and all other details. The difference lies in their relative positions only. Table 2.2 shows the comparison of first angle and third angle projection methods.

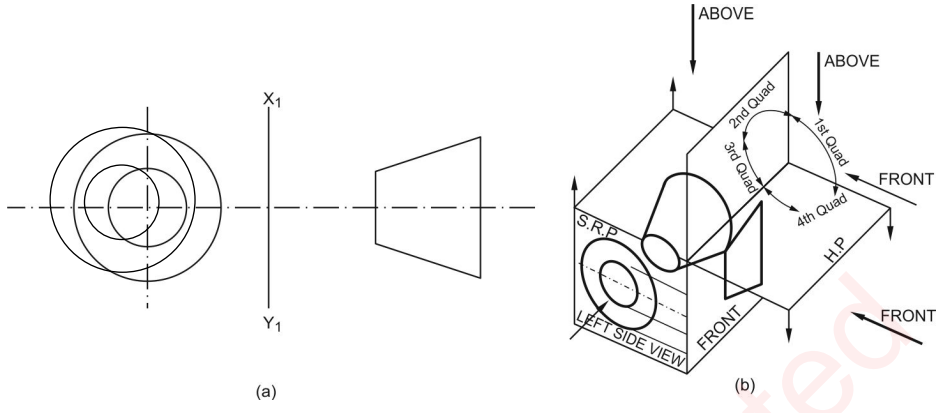


Fig.2.17: (a) Symbol for orthographic drawings made in third angle projection system
(b) A frustum of a cone is placed in third quadrant with its axis parallel to H.P. and V.P.

Table 2.2: Comparison of first angle and third angle projection method

Sl. No.	First Angle	Third Angle
1.	Object is kept above H.P. and in front of V.P. (i.e. in first quadrant)	Object is kept below H.P. and behind the V.P. (i.e. in third quadrant)
2.	Top view is below the front view	Top view is above the front view
3.	Right hand side view is on the left hand side of the front view	Right hand side view is right hand side of the front view.
4.	Left hand side view is on the right hand side of the front view.	Left hand side view is on the left hand side of the front view.
5.	Picture plane is placed behind the object with respect to observer.	Picture plane is placed in front of object with respect to observer.
6.	Each view represents the side of the object remote from it in the adjacent view.	Each view represents the side of the object near to it in the adjacent view.

2.3 AXONOMETRIC PROJECTION

Single view parallel orthographic projection obtained when X, Y and Z axis of solid are inclined to plane of projection is called axonometric projection. Main advantage of Axonometric projection is it gives 3- dimensional image of the object.

Axonometric projections are further classified into three sub categories, namely, Isometric, Dimetric and Trimetric projections (Fig.2.18).

2.3.1 Isometric Projection

When all three-coordinate axis of solid are equally inclined to plane of projection then the single view orthographic projection obtained is called isometric projection. Fig. 2.5 shows the isometric projection of a solid. Notice that it gives a 3- dimensional image of the object. Isometric projection will be discussed in detail in chapter-3.

2.3.2 Dimetric Projection

When two-coordinate axis of solid are equally inclined to plane of projection, and the third axis is inclined at a different angle to plane of projection then the single view orthographic projection obtained is called dimetric projection.

2.3.3 Trimetric Projection

When all three-coordinate axis of solid are inclined at different angle to plane of projection, then the single view orthographic projection obtained is called trimetric projection.

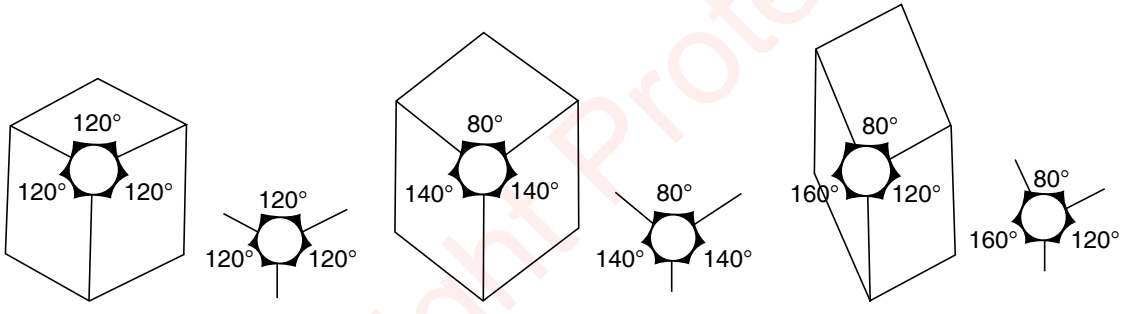


Fig. 2.18: Axonometric projections: Isometric, Dimetric and Trimetric projections

2.4 OBLIQUE PROJECTION

Single view parallel projection when projectors are inclined to the projection plane at angle other than 90° is called oblique projection

Fig.2.19 shows the pictorial view of oblique and orthographic projection of a cube and Fig.2.20 shows the actual oblique and orthographic projection of the same cube. Generally, you imagine the face of solid containing maximum information parallel to the projection plane. In the oblique projection, you see true shape and size of the plane of solid parallel to projection plane. Projection of lines perpendicular to projection plane is called receding lines. Though theoretically receding lines can be drawn at any angle general practice is to draw receding lines at angle of 30°, 45°, 60°. You get a pictorial 3-dimensional view of the solid in oblique projection.

Oblique projection is further classified in three sub categories:

2.4.1 Cavalier Oblique Projection

Projection of lines perpendicular to projection plane, i.e., receding lines, appear to be of true length in cavalier oblique projection. This is possible only when in 3-dimension angle between projectors and projection plane is 45°.

2.4.2 Cabinet Oblique Projection

Fig.2.21 shows in pictorial view, the Cabinet oblique projection and orthographic projection of a cube and Fig.2.22 shows the actual Cabinet oblique projection and orthographic projection of the same cube. Receding lines i.e., projection of lines perpendicular to projection plane appear to be of half true length. This is possible only when in 3-dimension angle between projectors and projection plane is $\tan^{-1}(2)=63.43°$. On comparing Fig.2.20 and Fig.2.22 you conclude that cabinet oblique projection gives a more proportionate 3-dimensional look of the cube compared to its cavalier oblique projection.

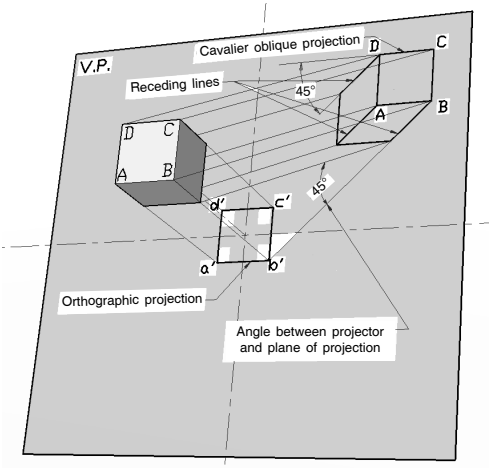


Fig.2.19: Pictorial drawing of concept of cavalier oblique projection of a cube

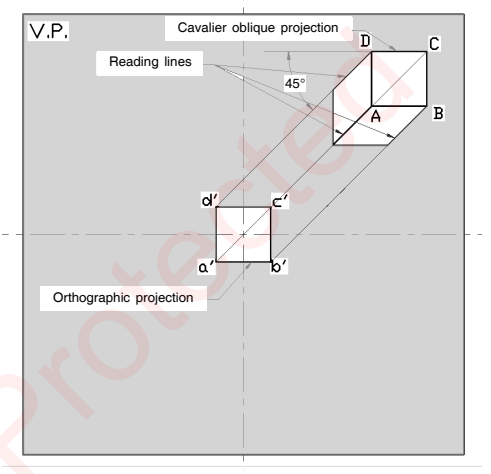


Fig.2.20: Cavalier oblique projection of a cube

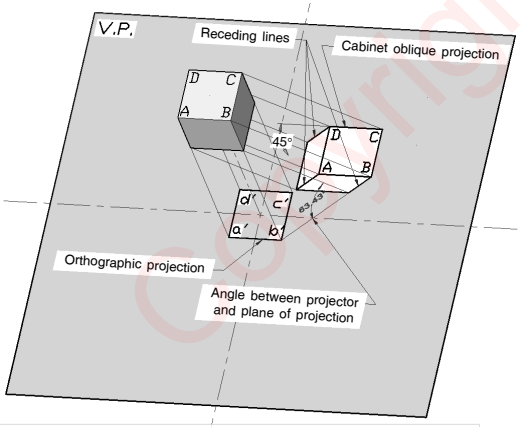


Fig.2.21: Pictorial drawing of concept of cabinet oblique projection of a cube

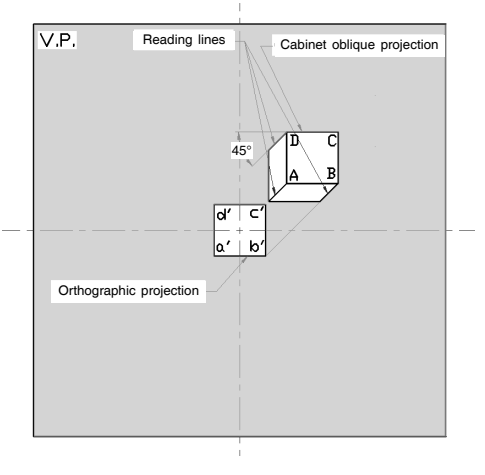


Fig.2.22: Cabinet oblique projection of a cube

2.4.3 General Oblique Projection

Receding lines i.e., line perpendicular to projection plane appear to be reduced length in any arbitrary proportion. That is in 3-dimension angle between projectors and projection plane is having any value except 45°and 63.43°and 90°and 0°.

Though all three methods of oblique projection given unnatural view of the object, still then cabinet projection is more near to actual view. Main advantage of oblique projection is

1. oblique projection can be drawn faster
2. In oblique projection you see true shape and size of plane parallel to projection plane
3. oblique projection gives a reasonably informative pictorial view of the solid.

2.5 PERSPECTIVE PROJECTION

There are various methods for drawing pictorial 3-dimension drawing of a solid. Some of them are :

1. Cavalier oblique projection
2. Cabinet oblique projection
3. Isometric projection
4. Diametric projection
5. Trimetric projection
6. Perspective projection

Best approximation of the view seen by human eye is perspective projection. It is 3-dimensional representation of an object on a plane surface, as it would appear to the eye when viewed from a fixed position.

Fig.2.23 shows a pictorial view describing how perspective projection is obtained in 3-dimensions. A picture plane is placed between observer eye and object. Projectors are drawn from corner points of the steps model to the observer eye. Points of intersection of projectors and picture plane are joined in proper sequence to obtain the perspective of the object on the picture plane.

Perspective projection is further classified into three categories depending on the orientation of X, Y & Z axis of the object relative to picture plane.

2.5.1 One-Point Perspective Projection

When two axes of a solid are parallel to picture plane then it is called one-point perspective projection. Fig.2.23 shows a pictorial view showing relative position of object and picture plane to obtain one-point perspective projection. Fig.2.24 shows the one-point perspective projection of the model of steps.

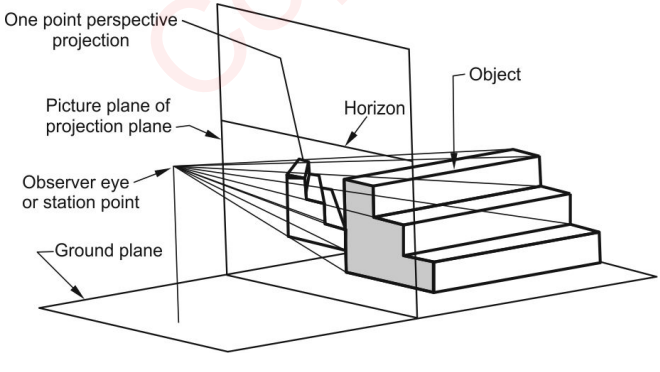


Fig.2.23: Pictorial view describing how perspective projection is obtained in 3-dimensions

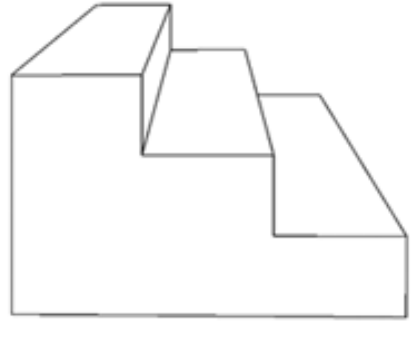


Fig.2.24: One-point perspective projection of the model of steps.

2.5.2 Two-Point Perspective Projection

When only one axes of a solid is parallel to picture plane and the remaining two axes are inclined to picture plane, then it is called two-point perspective projection.

Fig.2.25 shows a pictorial view showing relative position of object and picture plane to obtain two-point perspective projection. Only vertical lines are parallel to the plane of projection. Fig.2.26 shows the two-point perspective projection of the model of steps.

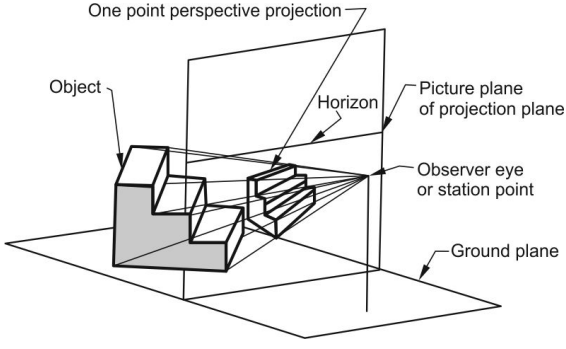


Fig.2.25: Relative position of object and picture plane to obtain two-point perspective projection

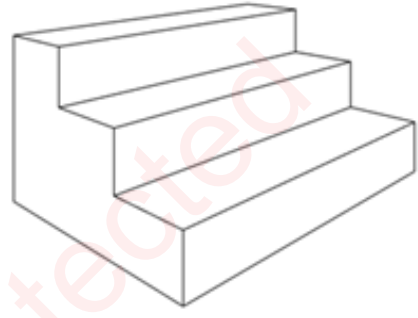


Fig.2.26: Two-point perspective projection of the model of steps.

2.5.3 Three-Point Perspective Projection

When all three axes of a solid are inclined to picture plane then the perspective projection obtained is called three-point perspective projection.

Further discussion on perspective projection like how to draw the perspective projection is out of scope of this book.

2.6 CONVERSION OF PICTORIAL VIEWS INTO ORTHOGRAPHIC VIEWS

Multiview orthographic projection has the distinct advantage of clarity and details compared to pictorial drawings. Hence, operator on the shop floor, is generally provided with the multiview orthographic drawings. To show internal details more clearly, he is also provided multiview sectional orthographic drawings. Hence ability to convert pictorial views, or 3D solid into orthographic views is of key importance for an engineer working in design and drawing department.

Whenever you are required to make multiview orthographic projection of a machine component the input data is either the pictorial drawing of the component, or the real machine component in 3 dimensions, or pictorial imagination of machine component in your brain. Now a days drafting software have the capability of directly converting a solid modelling into multiview orthographic projection.

SOLVED PROBLEMS (WITH INCREASING DIFFICULTY LEVEL)

Example 2.1 *Draw orthographic views of the model of steps shown to communicate all details about its shape and size.*

Solution As per IS specification SP46 letter **'a'** indicates the front direction of looking. Since the object is simple, only view from above and view from front are sufficient to explain the shape and size of the object.

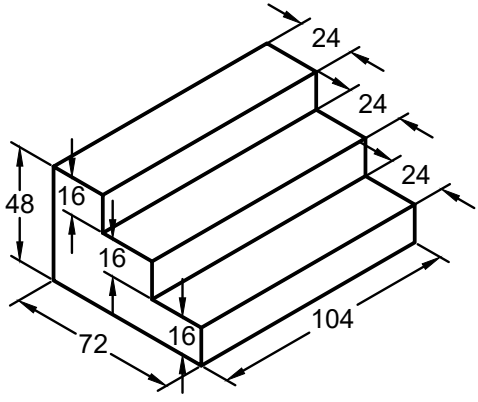


Fig.2.27: Isometric drawing of model of steps

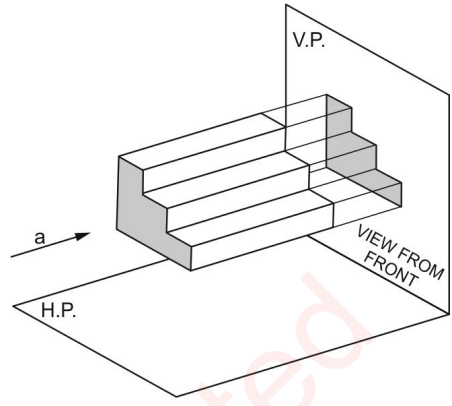


Fig.2.28: Pictorial modelling showing view from front of model of steps on vertical projection plane

First step in making view from front is to imagine the solid model positioned in first quadrant and project in 3 dimensions its view from front. Fig.2.28 shows the imagination of the view from front in three dimensions. Fig.2.29 shows the visualisation of view from top for model of steps.

Generally, in orthographic drawing starting is made from the view in which true shape of faces of the solid is visible. Since this model of steps is a having some planes parallel to H.P. and some planes parallel to V.P. starting can be made from view from above as well as view from front. Fig.2.30 shows the orthographic views for the model of steps. Only view from front and top are sufficient to explain complete information about its shape, there is no need of any other orthographic view.

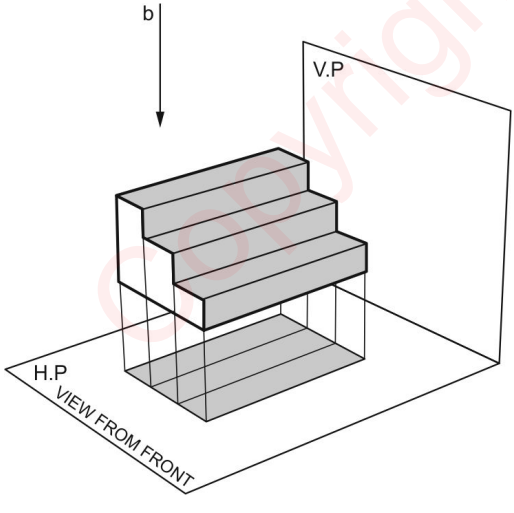


Fig.2.29: Pictorial modelling showing view from above of model of steps on horizontal projection plane

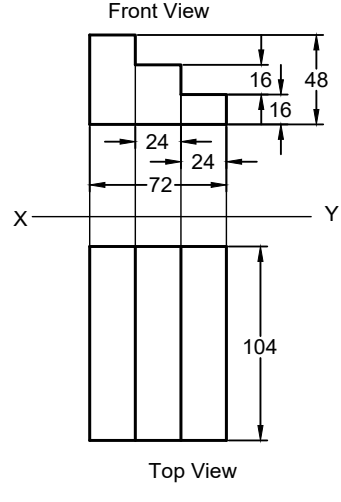


Fig.2.30: Orthographic views of the model of steps

Example 2.2 *Draw orthographic views required to communicate all shape and size details for the fixture with triangular rib shown in Fig.2.31.*

Solution.

As shown in Fig.2.32 imagine view from front of the clamping fixture in three dimensions.

As shown in Fig.2.33 imagine view from above of the clamping fixture in three dimensions.

Only view from front and top are not sufficient to explain the triangular shape of the rib. For explaining triangular shape of rib view from left or right is required. As shown in Fig.2.34 imagine view from right of the clamping fixture in three dimensions. Fig.2.35 shows the required orthographic views for the model of clamping fixture with triangular rib.

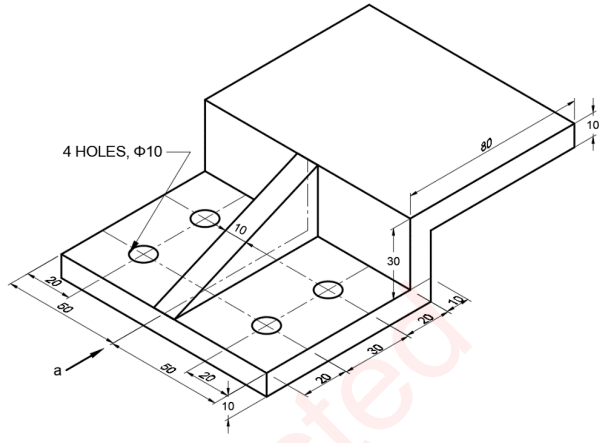


Fig.2.31: Clamping fixture with triangular rib

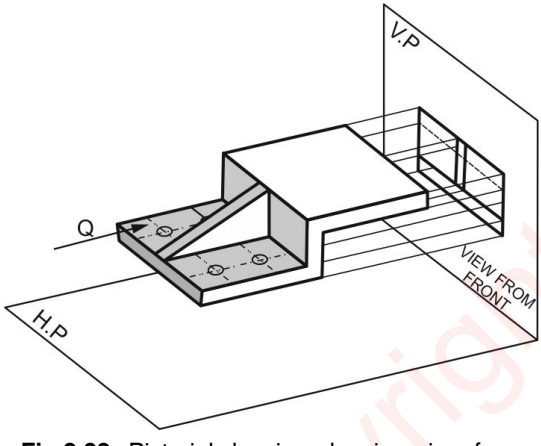


Fig.2.32: Pictorial drawing showing view from front of the clamping fixture

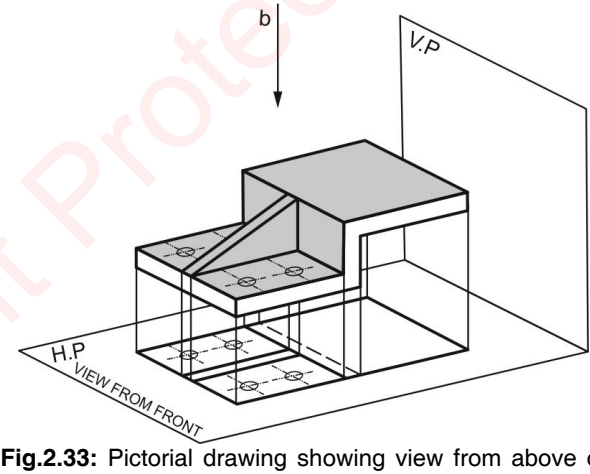


Fig.2.33: Pictorial drawing showing view from above of the clamping fixture

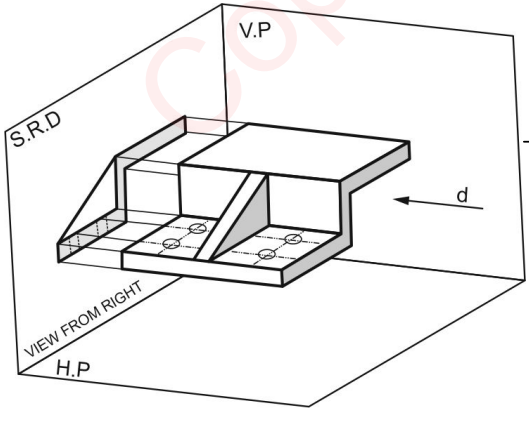


Fig.2.34: Pictorial drawing showing view from right of the clamping fixture

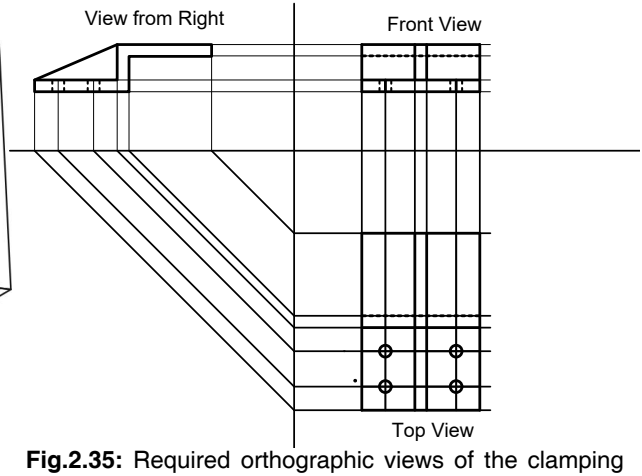


Fig.2.35: Required orthographic views of the clamping fixture

Example 2.3 *Draw orthographic views of the slotted machine component shown to communicate all details about its shape and size.*

Solution

As shown in fig.2.37 imagine view from front of the slotted machine component in three dimensions. As shown in fig.2.38 imagine view from above of the slotted machine component in three dimensions. Fig.2.39 shows the required orthographic views for the model of the slotted machine component.

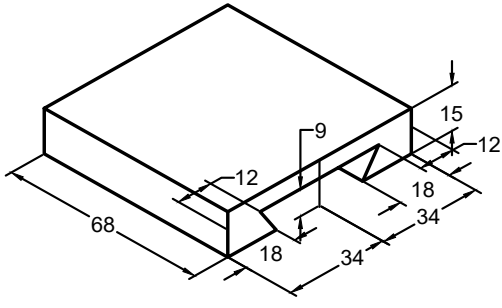


Fig.2.36: Slotted machine component

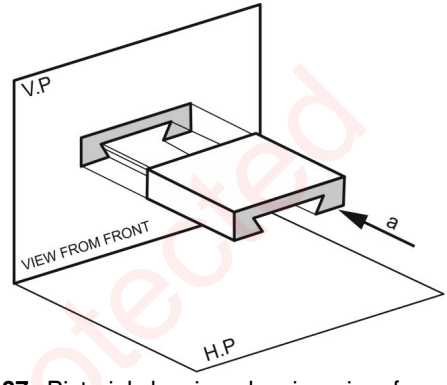


Fig.2.37: Pictorial drawing showing view from front

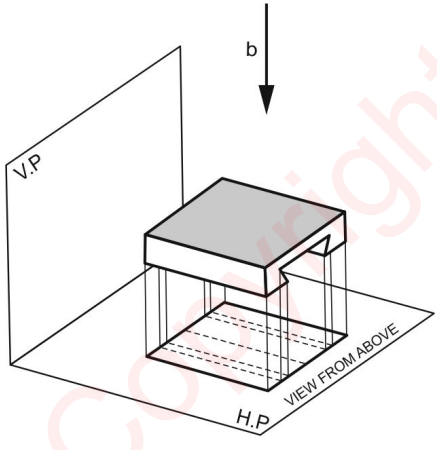


Fig.2.38: Pictorial drawing showing view from above

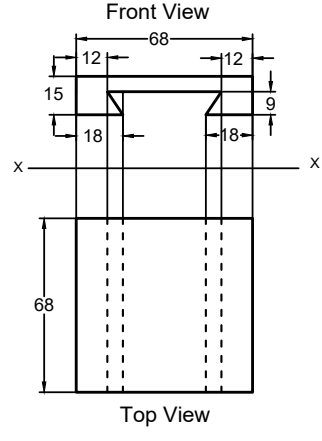


Fig.2.39: Required orthographic views of the slotted machine component

Example 2.4 *Draw required orthographic views of the upper brass of a Plummer block.*

Solution

As shown in fig. 2.41 imagine view from front of the upper brass of Plummer block in three dimensions. As shown in fig. 2.42 imagine view from above of the upper brass of Plummer block in three dimensions. As shown in fig. 2.43 imagine view from left of the upper brass of Plummer block in three dimensions. Fig. 2.44 shows the required orthographic views for the model of the upper brass of Plummer block.

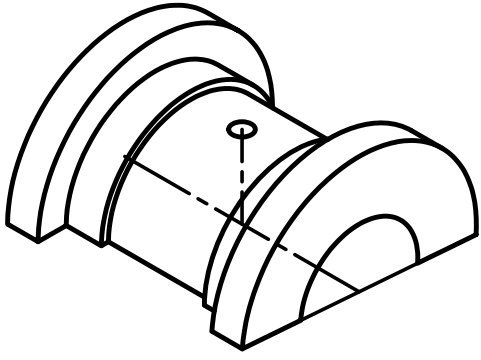


Fig.2.40: Isometric drawing of the upper brass

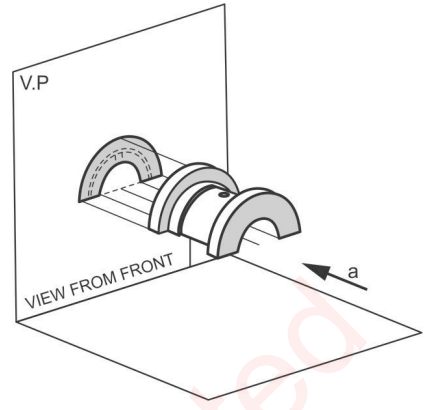


Fig.2.41: Pictorial drawing showing view from front

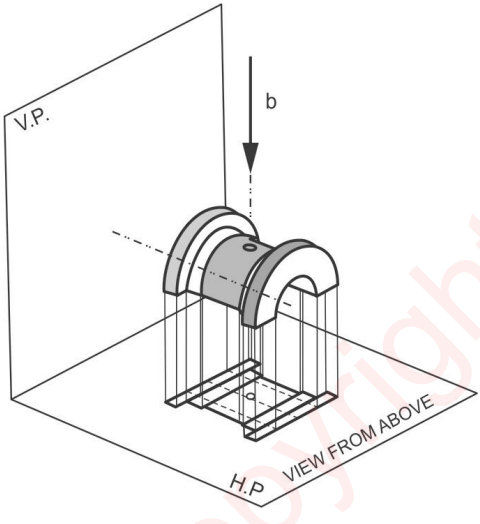


Fig.2.42: Pictorial drawing showing view from above

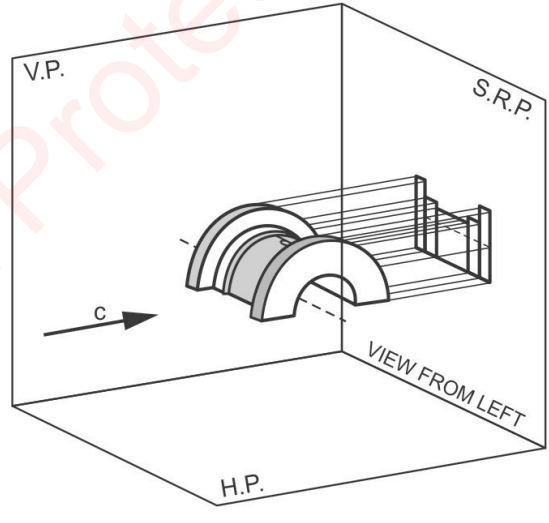


Fig.2.43: Pictorial drawing showing view from left

Simple Orthographic Projections

Orthographic Projections of objects with slant & curved surfaces

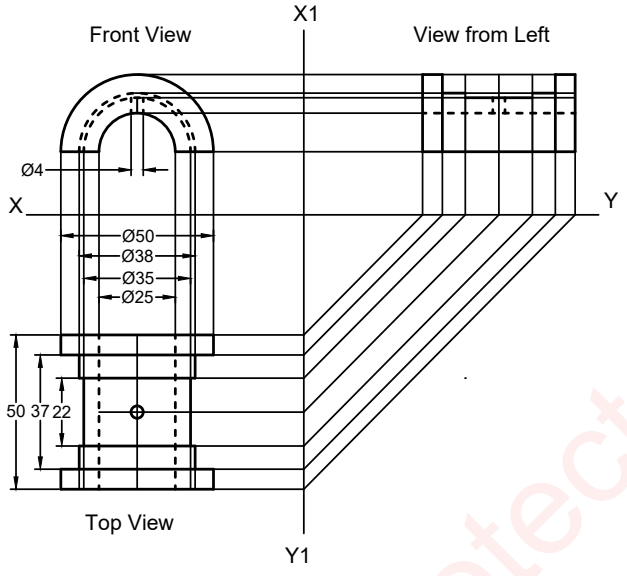


Fig.2.44: Required orthographic views of the upper brass of Plummer block

UNIT SUMMARY

1. Parallel projection is classified into two subcategories;
   a. Oblique projection and b. Orthographic projection.
2. In orthographic projection projectors are always parallel to each other and perpendicular to the plane of projection.
3. The three basic elements for a projection are; object, projectors and reference planes. In orthographic projection, the projection of the object on the vertical plane (V.P.) is known as front view and on the horizontal plane (H.P.) is known as top view.
4. In first angle projection method, the object lies between the observer and plane of projection. Here, the object is positioned in first quadrant i.e., in front of V.P. and above H.P.
5. In third angle projection, transparent plane of projection lies between the observer and object. Here, the object is positioned in third quadrant i.e., behind V.P. and below H.P.
6. Single view parallel orthographic projection obtained when X, Y and Z axis of solid are inclined to plane of projection is called axonometric projection.
7. When all three-coordinate axis of solid are equally inclined to plane of projection then the single view orthographic projection obtained is called isometric projection.
8. When projectors are inclined to the projection plane at angle other than 90° then the single view parallel projection is called oblique projection.
9. Perspective projection is 3-dimensional representation of an object on a plane surface, as it would appear to the eye when viewed from a fixed position.
10. At present in India, first angle projection is recommended by BIS.

EXERCISES

A. Objective Questions

A1. Fill in the blanks with appropriate words

2.1 Multi-view drawing is another name for_____.

2.2 Projection of an object shown by three view is called as _____.

2.3 The top view and _____ view is always in line vertically.

2.4 The horizontal and vertical planes intersect forming four quadrants. These quadrants are called_____.

2.5 For orthographic projection, BIS recommends the_____ projection method.

2.6 In orthographic projections, the xy line is called as _____.

2.7 In orthographic projection projectors are always _____ to each other.

2.8 The top view of an object is projected on _____ plane.

2.9 In oblique projection projectors are _____ to the plane of projection.

2.10 In third angle projection, the picture plane is placed _____ the object with respect to observer.

Answers of Fill in the Blanks Questions

2.1 Orthographic projection; 2.2 Orthographic; 2.3 Front; 2.4 Angles; 2.5 First angle projection; 2.6 Reference Line; 2.7 Parallel; 2.8 Horizontal; 2.9 Inclined; 2.10 in front of;

A2. Multiple choice questions

2.1 In orthographic projection, the xy line is also known as ……..

(a) Horizontal line (b) Vertical line

(c) Reference line (d) Horizontal trace

2.2 In orthographic projections, the projectors are assumed to be

(a) diverge from a point (b) Oblique to the plan of projection

(c) Perpendicular to each other (d) Parallel to each other

2.3 The orthographic projection of an object on vertical reference plane is known as ……..

(a) Front view (b) Top view

(c) Rear view (d) Side view

2.4 When projector are drawn from corner points of object perpendicular to horizontal reference plane then figure obtained by joining points of intersection of projector and horizontal reference plane in proper sequence is called ……..

(a) Plan of object (b) Elevation of object

(c) Front view of object (d) Side view of object

2.5 Orthographic projection means ……..
(a) Projecting an object such that the projectors are perpendicular to each other
(b) Projecting ……. projectors are oblique to each other
(c) Projecting an object such that the projectors are parallel to each other and oblique to the plane of projection
(d) Projecting …….. the projectors are parallel to each other and perpendicular to the plane of projection

2.6 In first angle projection method, the relative positions of the object, plane of projection and observe are ……..
(a) Plane of projection is placed in between (b) Observer is placed in between
(c) Object is placed in between (d) Object is placed before the plane of projection and observer

2.7 According to BIS SP-2003, which of the following projection is used for orthographic projection ?
(a) First angle projection (b) Second angle projection
(c) Third angle projection (d) Fourth angle projection

2.8 Principal planes refer to ……..
(a) Vertical plane and auxiliary plane (b) Vertical and Horizontal plane
(c) Central plane and ground plane (d) Auxiliary plane and picture plane

2.9 The symbol recommended by BIS for indicating the angle of projection showing two views of the frustum of a ……
(a) Cone (b) Prism (c) Pyramid (d) Sphere

2.10 In orthographic projection, the height dimension of an object is seen in ……..
(a) Front and top view (b) Front and side view
(c) Top and side view (d) Front, top and side view

Answers of Multiple Choice Questions

2.1 (c); 2.2 (d); 2.3 (a); 2.4 (a); 2.5 (d); 2.6 (c); 2.7 (a); 2.8 (b); 2.9 (a); 2.10 (b)

B. Subjective Questions

2.1 Explain the term projection. List the elements required for a projection.
2.2 Give classification of a projection method for making a drawing.
2.3 Define an orthographic projection? How does it differ from an oblique projection?
2.4 Explain reference planes and their utility.
2.5 Differentiate between a perspective projection and parallel projection.
2.6 Explain the term quadrant and its utility to obtain orthographic projection.
2.7 Explain the concept of first and third angle projection method with the help of suitable sketches.

2.8 Differentiate between the first angle and third angle projection method on the basis of: (i) Position of different views and (ii) Position of object, observer and plane.

2.9 Define multi view projection. Describe briefly the method of obtaining multi view projection of an object.

2.10 Give the symbolic representation of first-angle projection.

2.11 Fig.2.45 shows the pictorial views of four objects. The direction of view is shown by arrow. Draw its front view and top view using the first angle projection.

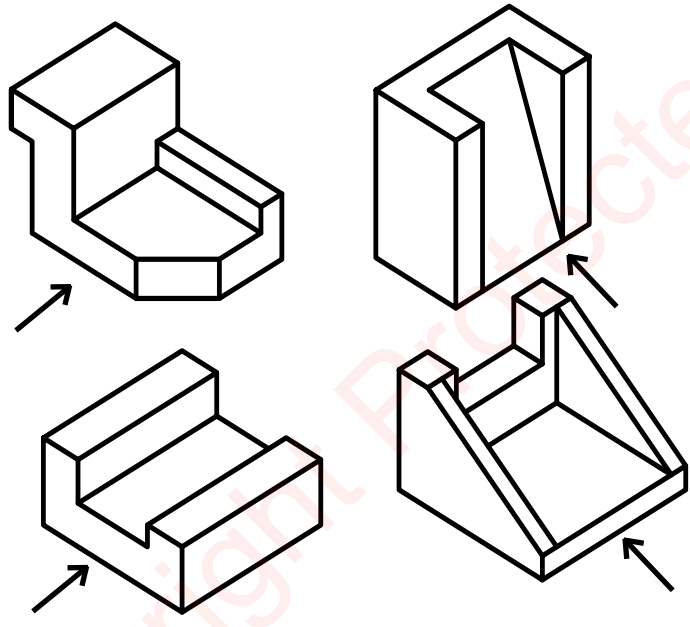


Fig.2.45: Pictorial views for subjective problem 2.11

2.12 Draw three views of the objects shown in Fig.2.46 using first angle projection.

2.13 Draw front and top view of the six objects given in Fig.2.47 using first angle projection. The direction of view is shown by arrow.

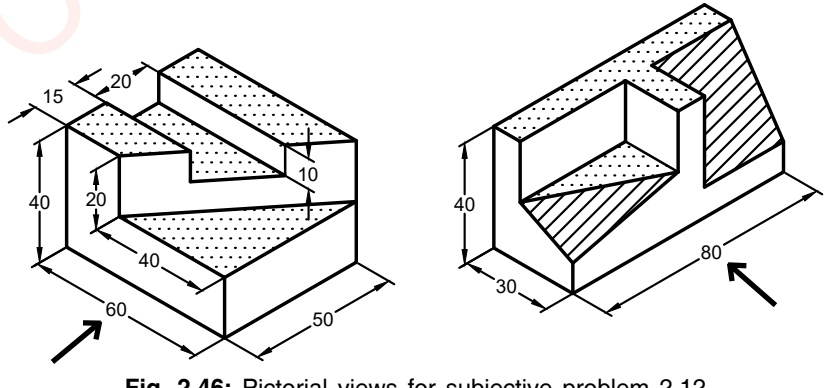


Fig. 2.46: Pictorial views for subjective problem 2.12

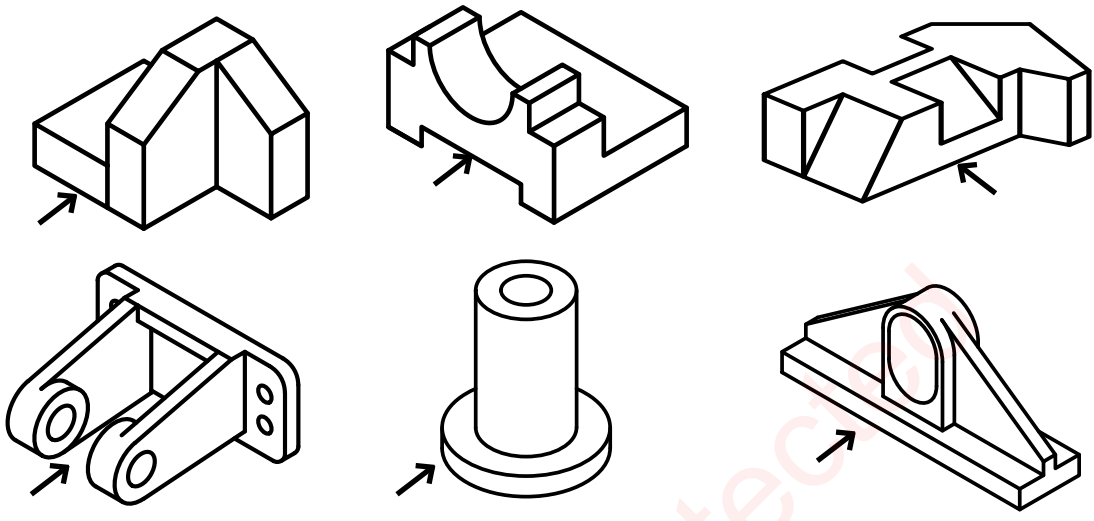


Fig. 2.47: Pictorial views for subjective problem 2.13

PRACTICALS

As per curriculum the practicals related to Unit-2 are:

1. Draw a problem on orthographic projections using first angle method of projection having plain surfaces and slanting.
2. Draw another problem on orthographic projections using first angle method of projection having slanting surfaces with slots.
3. Draw two problems on orthographic projections using first angle method of projection having cylindrical surfaces, ribs.

Practical-1 Draw Orthographic Projections of Object with Plain and Slant Surfaces

Practical statement

Draw orthographic projections of an object having plain and slanting surfaces using first angle projection method.

Practical significance

After designing components the next step is to manufacture/construct them hence production/ construction drawings are required by the technicians. To communicate with these technicians orthographic and sectional orthographic drawings are needed which shows all possible details more clearly and precisely so that the component/entity can be manufactured/constructed. The technicians are trained to read the orthographic views. These views are generally drawn with first angle projection method.

Relevant theory

Refer Section 2.0 of the book.

Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

PrO1: Draw orthographic views in first angle projection from given isometric views of objects having plain and slanting surfaces.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

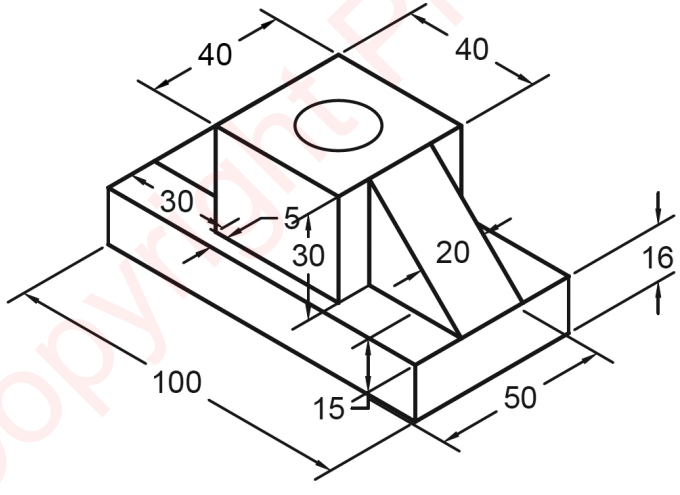
Resources required

Same as mentioned in Practical-1 of Unit-1

Precautions

Same as mentioned in Practical-1 of Unit-1

Suggested procedure



1. Looking to the given Isometric view of the object it can be concluded that the object is basically made up of a rectangular block base (100 x 16) with another rectangular block (40 x 30) centrally located on top of it along with two symmetrically placed Ribs (50 mm thick) on the two sides.
2. Start with front view and draw a rectangle of 100mm by 60mm to represent the front view of base block and another rectangle of size 40 mm length and 30 mm height centrally above it to indicate top block.
3. Join the top edges of the square block with that at the base block to mark the sloping ribs, on either side as shown in the front view.
4. Now draw the projectors to develop Top view. Sketch a rectangle of length 100 mm and

width 50 mm to represent base block and mark a square of 40 mm sides in the centre, to represent the top the centre block in top view. Mark the hole and the side edges to indicate the ribs as shown in the top view.

5. Project the side view of the base unit and the centre block along with the ribs as shown in the right-side view. Represent the hole with dotted lines.

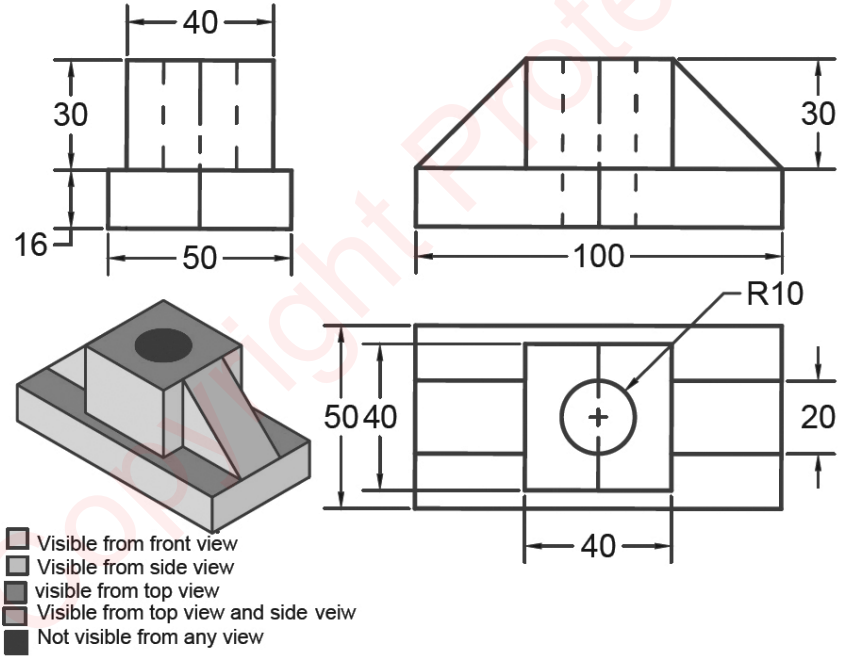
Observations

Practical related questions

1. In right side view the dotted lines are not of full length while the hole is a through hole. Justify.
2. Explain the purpose of ribs.

Disposal of waste

Same as mentioned in Practical-1 of Unit-1



Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1 of Unit-1

Suggested learning resources

Refer Unit-2

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-1

Practical-2 Draw Orthographic Projections of Object with Slant Surfaces and Slots

Practical statement

Draw orthographic projections of an object having slanting surfaces and slots using first angle projection method.

Practical significance

After designing components the next step is to manufacture/construct them hence production/ construction drawings are required by the technicians. To communicate with these technicians orthographic and sectional orthographic drawings are needed which shows all possible details more clearly and precisely so that the component/entity can be manufactured/constructed. The technicians are trained to read the orthographic views. These views are generally drawn with first angle projection method.

Relevant theory

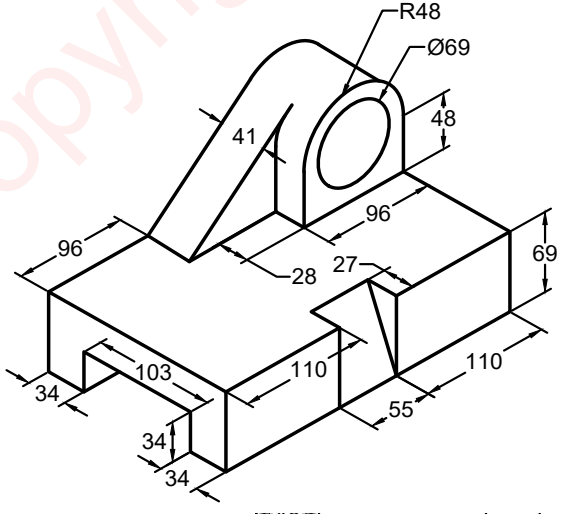
Refer Section 2.0 of the book.

Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

PrO1: Draw orthographic views in first angle projection from given isometric views of objects having slanting surfaces and slots.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)



Resources required

Same as mentioned in Practical-1 of Unit-1

Precautions

Same as mentioned in Practical-1 of Unit-1

Suggested procedure

1. Looking to the given Isometric view of the object it can be concluded that the base of the object is a rectangular block which is supporting a semicircular block above it. The semicircular block is connected tangentially with a sloping rib with the base block. A longitudinal slot in the base and a front inclined central slot are provided in the rectangular block.
2. Start from the front view and draw a rectangle of length 275 mm and height 69 mm to represent base block. Draw a full circle of radius 34.5 mm and center 48 mm above and 48 mm left from the right hand top corner of the rectangle.
3. Add a semicircle with same center as that of circle but radius equal to 48 mm on top of the circle. Draw two vertical tangents to connect this semicircle with rectangle. Mark a point at 96 mm from the top left corner of the rectangle and connect this point to the semicircle through an external inclined tangent. Draw two vertical parallel lines 55 mm apart in the center of the rectangle to represent central inclined slot. This completes the front view.
4. Project the Rectangle, Circle, Semicircle, Longitudinal slot and Central inclined slot from Front view to Top view. The corresponding width values of these entities will be taken from the isometric view. Looking at the figure given in the Observation section list the steps followed:

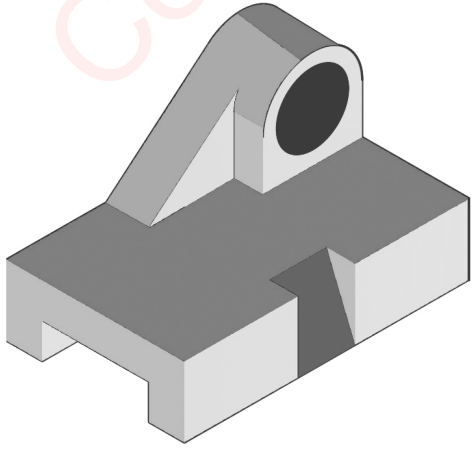
   a)…… ……

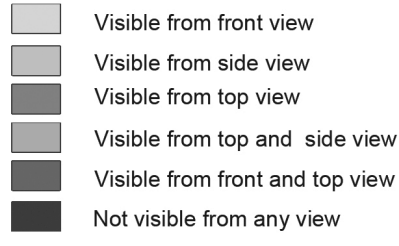
   b)…… ……
5. In Right side view, an L-shaped arrangement indicating the base block and the bearing block is shown. Project the end points of the rib. The slope of the central cut-out is marked at a distance of 27 mm. Looking at the figure given in the Observation section list the steps followed:

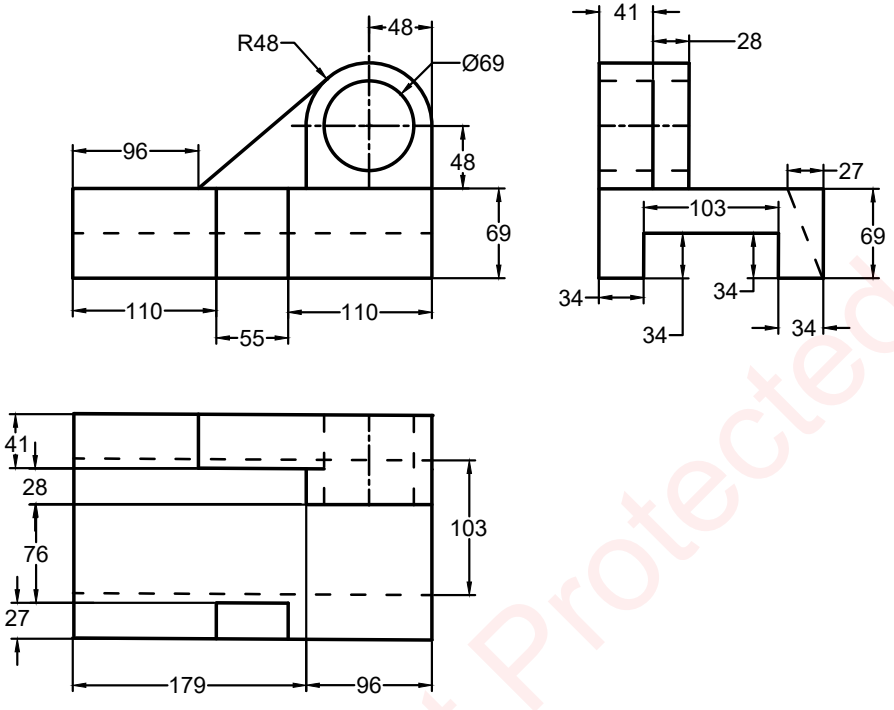
   a)…… ……

   b)…… ……

Observations







Practical Related Questions

1. In right side view the dotted lines are not of full length while the hole is a through hole. Justify.
2. Explain the purpose of ribs.

Disposal of Waste

Same as mentioned in Practical-1 of Unit-1

Environment Friendly Approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1 of Unit-1

Suggested Learning Resources

Refer Unit-2

Suggested Assessment Scheme

Same as mentioned in Practical-1 of Unit-1

Illustrative Example-1

Practical-3 Draw Orthographic Projections of Object with Cylindrical Surfaces and Ribs

Practical statement

Draw orthographic projections of an object having cylindrical surfaces and ribs using first angle projection method.

Practical significance

After designing components the next step is to manufacture/construct them hence production/ construction drawings are required by the technicians. To communicate with these technicians orthographic and sectional orthographic drawings are needed which shows all possible details more clearly and precisely so that the component/entity can be manufactured/constructed. The technicians are trained to read the orthographic views which are generally drawn with first angle projection method

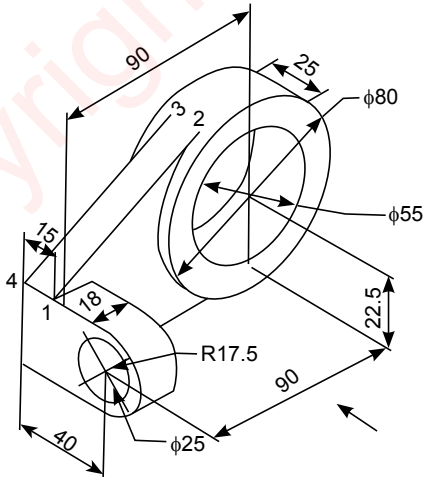
Relevant theory

Refer Section 2.0 of the book.

Practical outcomes (PrO)

PrO1: Draw orthographic views in first angle projection from given isometric views of objects having cylindrical surfaces and ribs.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)



Resources required

Same as mentioned in Practical-1 of Unit-1

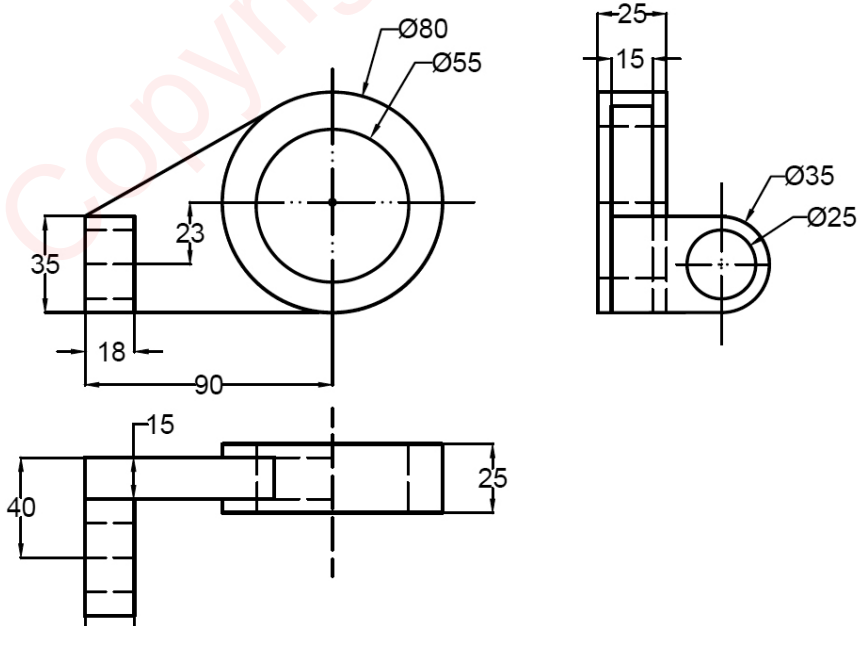
Precautions

Same as mentioned in Practical-1 of Unit-1

Suggested Procedure

1. In the given object there are two different sized hollow cylindrical blocks placed perpendicular to each other. Both are connected with a central L-shaped rib, which is inclined from the top and horizontal from the bottom. The bigger one is placed in the front surface and the smaller one is placed in the left hand-side surface of the L-shaped structure.
2. In front view start with to center lines perpendicular toeach other followed by two concentric circles of 55 mm and 80 mm diameter to represent the front view of the bigger cylindrical block. Locate the axis of the side block which is 22.5 mm from the centre of these concentric circles. Draw the outline of smaller cylindrical block using a rectangle of height of 35mm and width 18mm centrally placed about the axisas shown in the front view.
3. Now draw the inclined and horizontal external tangents of the bigger circle connecting to the rectangle. Draw parallel dotted lines in the rectangle to represent hole in the smaller cylindrical block.
4. The outlines of the central rib are drawn as tangential lines to the circle as shown in the FV.
5. The top views of the three units are projected as rectangles and are joined as shown in Fig. The top surface of the rib is shown in thick lines, while the bottom surface is identified in dotted lines.
6. The left-side view of the assembly is drawn in an L-shaped fashion as shown in figure. The cylindrical holes in the respective blocks are represented as circles in the appropriate views and with dotted lines in the other views. The rib is shown through two rectangle in Right side and Top view of thickness 15mm while the thickness of larger cylindrical block is 25mm and is visible in Right side view and Top view. The thickness of Smaller cylindrical block is 18mm and is visible in Front view and Top view.

Observations



Practical related questions

1. Identify the distances of the axis of smaller cylindrical block from back surface of the rib and back surface of bigger cylindrical block. Are these distances are same or different.
2. Identify the distance between the axes of bigger and smaller cylindrical blocks.

Disposal of waste

Same as mentioned in Practical-1 of Unit-1

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1 of Unit-1

Suggested learning resources

Refer Unit-2

Illustrative Example-2

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-1

KNOW MORE

- Teacher shall collect few production/construction/electrical/electronic actual industrial component drawings to show them to the students with input.
- During the input sessions the teacher must give examples related to the relevant branch viz. Mechanical and allied disciplines/ Electrical and allied disciplines/Electronics etc.
- Teacher must know orthographic view of objects having plain, slanting and cylindrical surface
- Show video/animation films to explain mentioned concepts.
- Teachers should ask students to use url/qr codes available in the book for further understanding/ practicing of the concepts.

Applications (Real life / Industrial)

It is a standard practice of engineering companies to provide a manufacturer or fabricator an accurate, measured drawing that can be followed when making something.

Orthographic drawings are extensively used by designers – particularly architects, product/ industrial designers and engineers, as they are ideal for visualization and projection between views for design of buildings, mechanical components, machine elements , locking arrangements, jigs and fixtures, furniture etc.

Inquisitiveness and Curiosity

Other than the classroom and drawing practice sessions, following are the suggested student-related co-curricular activities which can be undertaken to accelerate the attainment of the various outcomes in this unit:

1. Student should maintain a separate A3 size sketch book which will be the part of term work and submit it along with drawing sheets. Following assignment on orthographic views should be drawn with a suitable dimension in the sketch book-
   a. Laptop computer
   b. Steel Almira
   c. Refrigerator
   d. Staircase
   e. Single point cutting tool
2. Each student of the batch will try to collect few pictorial views related to production drawings/ construction drawings/ plumbing drawings from local workshops/builders /electrical and mechanical contractors and try to generate the orthographic views using first angle and third angle projection.
3. Each student of the batch will surf the web and list the product / components in which orthographic views are used to manufacture or fabricate in different field of engineering.

REFERENCES & SUGGESTED READINGS

1. Engineering Drawing Practices for School and Colleges SP 46:2003, published by Bureau of Indian Standards, Government of India, Third Reprint, October 1998; ISBN:. 81-7061-091-2, Manak Bhavan, 9 Bahadur Shah Zafar Marg , New Delhi
2. A Text Book of Engineering Drawing, K Venkata Reddy, BS Publication, 2008
3. Engineering Drawing, N.D. Bhatt, Charotar Publishing House, Anand,Gujarat, 2014
4. NPTEL Course Material on Engineering Drawing, P.S. Robi, Department of Mechanical Engineering, Indian Institute of Technology Guwahati, India: https://nptel.ac.in/courses/112/105/112105294/

3 Isometric Projections

UNIT SPECIFICS

This unit presents information related to following topics:

- Introduction to isometric projections
- Isometric scale
- Isometric view and isometric projection
- Draw isometric view of plane geometrical figures containing lines, circles and arcs shape.
- Conversion of orthographic views into isometric view/projection

The understanding of these topics will be developed after reading the contents, completing the solved problems, activities, performing exercises and viewing the ICT & web resources mentioned in this unit.

At the end of the unit, summary is provided to recapitulate the topics covered and applications are mentioned so that the learner can correlate the presented knowledge with real life and industrial situations. Few activities are mentioned to create inquisitiveness and curiosity in the learner. Subjective and objective questions are provided for the reinforcement of the knowledge and a list of references and suggested readings is also provided for further learning. Video resources along with QR codes are mentioned for getting more information on various topics of interest which can be surfed or scanned through mobile phone for viewing. Practical with details are provided to horn the drawing and drafting skills.

RATIONALE

Nearly all the engineering drawing for manufacture in industry is produced in orthographic projection. In an orthographic projection, we draw separately front view, top view, right-side view, etc., while in an isometric drawing, the front side and the top all are combined in one figure only. This thing may happen in a pictorial projection also, but the necessary condition in an isometric drawing is that the principle lines are parallel to the isometric axes and the actual dimensions of a solid object can be measured directly from it. Engineers or designers often use pictorial drawings and sketches to explain the complication of the multi-view drawing or communicate ideas both among technical and with non-technical staff. Most commonly used pictorial drawing is isometric drawing.

PRE-REQUISITES

Before reading this unit the student is advised to revisit following:

- Maths: Plane geometry and Unit-2 of this book

UNIT OUTCOMES

After reading the contents, completing the solved problems, activities, exercises and viewing the ICT and web resources mention in this unit the students are expected to:

U3-O1: Construct isometric scale.

U3-O2: Differentiate between isometric view and isometric projection

U3-O3: Draw isometric views of given simple plane geometrical figures containing lines, circles and arcs shapes only

U3-O4: Draw isometric views from given orthographic views.

Unit-3 Outcome	EXPECTED MAPPING WITH COURSE OUTCOMES *(1- Weak Correlation; 2- Medium Correlation; 3- Strong Correlation)*					
	CO-1	CO-2	CO-3	CO-4	CO-5	CO-6
U3-O1	-	-	2	3	3	1
U3-O2	-	-	2	3	3	1
U3-O3	1	2	3	3	3	2
U3-O4	2	2	3	3	3	3

3.1 INTRODUCTION TO ISOMETRIC PROJECTIONS

The term isometric comes from the Greek for "equal measure". In isometric projection, all the three edges of the object are equally inclined to the planes forming an angle of 120° with each other and called the isometric axes as shown in Fig.3.1. The three lines OM, OQ and OP meeting at point O and making 120^0 angles with each other are called isometric axes or isoaxes.

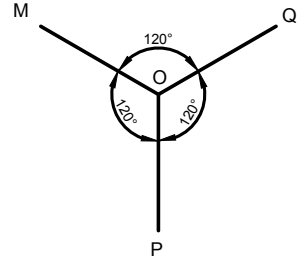


Fig.3.1: Isometric axes

The principles of isometric projection may be understood from the isometric projection of a cube. The isometric projection of a cube is obtained, when the line of sight is parallel to its solid diagonal. If a cube is resting on one of its corners on H.P. with its solid diagonal perpendicular to V.P. Fig.3.2 shows the front view of the cube in isometric projection when hidden lines are removed. It gives a realistic view of the cube. From the Fig.3.2 you may be noted that:

1. the three lines OM, OQ and OP meeting at point O, represent three edges of the cube which are equally inclined to the V.P. and therefore are equally fore-shortened.
2. the line OP is vertical and other two lines OM and OQ make 30degree angle with the horizontal.

3. all other lines representing the edges of the cube are parallel to one or the other of the above three lines i.e. OM, OQ and OP and are also fore-shortened equally.

In other words, in isometric projection, the three planes will be equally foreshortened and the axes are equally spaced at 120° apart. The axes OM, OQ and OP are called isometric axes, refer Fig.3.2. Lines parallel to these axes are called isometric lines and planes parallel to the faces of the cube in the isometric projection are called isometric planes. As the three axes are fore-shortened equally, single isometric scale can be used.

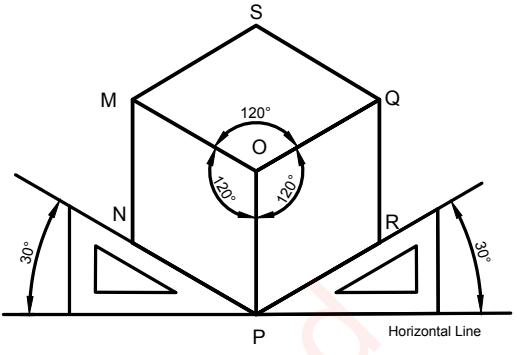


Fig.3.2: Isometric projection

This projection is more appropriate in the case of small objects, but larger objects may appear to be unnatural. Perspective projection is used to represent larger objects.

3.2 ISOMETRIC SCALE

This scale is used only for making the isometric views. The projection by means of which the true distance is reduced to isometric distance is known as isometric scale.

In isometric projection, the measurement of dimensions is reduced, therefore a special scale is required to draw the projections. Refer to Fig.3.3(a), construct a square with MQ as diagonal representing the true shape of the top surface. It may be noted from the same figure that the lines MS and MT are inclined at 30° and 45° to MY respectively. It is also evident from Fig.3.3(b), that MT is the true length and MS is its isometric length of projection.

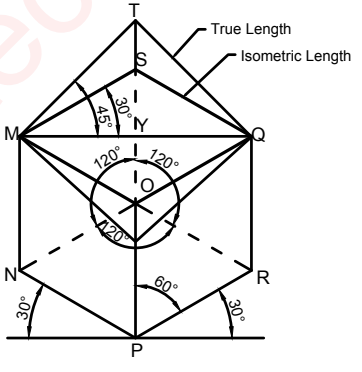


Fig.3.3(a): Isometric projection

From right angle triangle, Δ MYT and Δ MYS

MY = MT cos 45°

MY = MS cos 30°

Now, MT cos 45° = MS cos 30°

$$\text{So, } \frac{MS}{MT} = \frac{\cos 45^\circ}{\cos 30^\circ} = \frac{1/\sqrt{2}}{\sqrt{3}/2} = 0.81649568 = 0.816 = \frac{816}{1000}$$

Hence, isometric length = 0.816 x true length.

Thus, all the dimensions in isometric projection are 0.816 times the true length or approximately 82% of the true length can be obtained either by multiplying it by factor or by construction of isometric scale. The isometric scale shown in Fig.3.3 (b) has been constructed as below steps:

1. Draw a horizontal line MY.
2. Draw lines MS and MT making 30° and 45° with line MY.

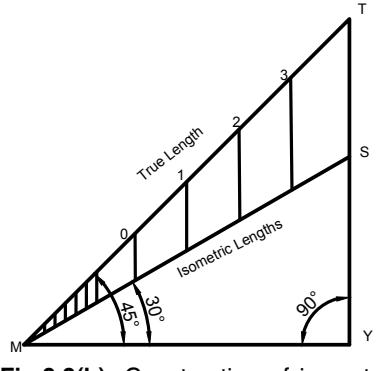


Fig.3.3(b): Construction of isometric scale

3. Construct a regular scale to true length on the 45° line (MT) such as 0,1,2,3,4 cm and project the same point on the 30° line MS.
4. Drawn lines should be perpendicular to the base line MY, meeting upto MS only.
5. Mark the corresponding division points on the line MS to obtain the isometric scale.

It should be noted very carefully that all the measurements should be made along isometric axis or isometric lines only".

The isometric scale can be drawn in a simple form as shown in Fig.3.4. The true length is marked as PQ. A line PR is drawn at 15˚ to PQ to intersect another line inclined at 45˚ to horizontal and drawn from Q. Then 45˚ inclined lines are drawn from the divisions on the true length scale PQ, so that the corresponding isometric lengths are obtained on the line PR. Even though it is easy to calculate the isometric lengths (82%) with the help of a calculator, it is a custom to show isometric scale nearby the isometric projection (on top or right side) for identification.

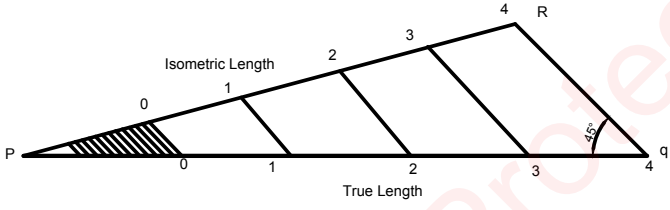
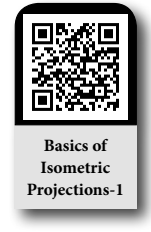


Fig.3.4: Simplified isometric scale



3.3 ISOMETRIC PROJECTION AND ISOMETRIC VIEW

In isometric projection, isometric lengths (0.816 times of the true length) are used to prepare the drawing. In isometric view, the true lengths are used to prepare the drawing, neglecting the foreshortened lengths of all the edges of the object. Thus, isometric view of an object is larger than

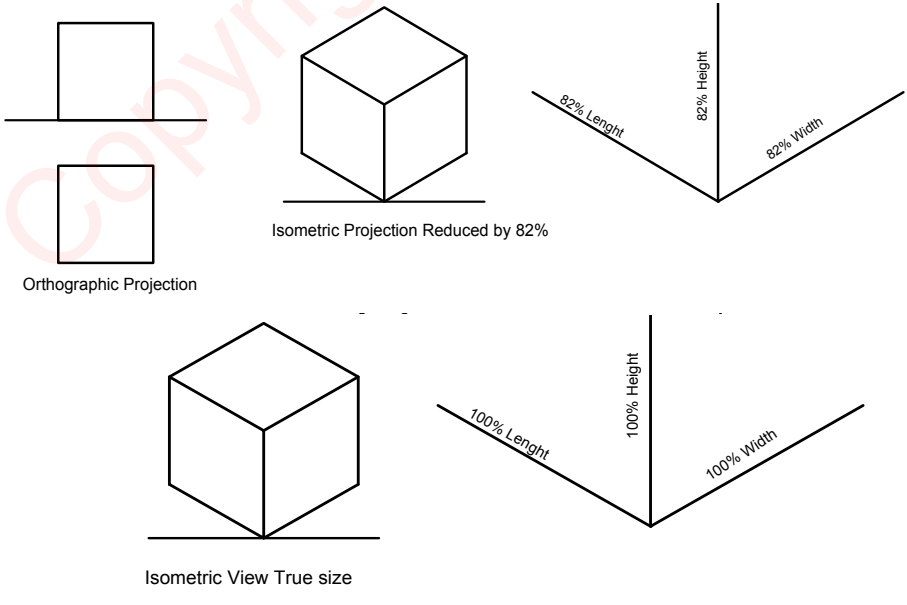


Fig.3.5: Difference between isometric projection and isometric view

the isometric projection of the object. Due to ease of construction and advantage of measuring the dimensions directly from the drawing, it has become a general practice to use the true scale instead of isometric scale. Thus, isometric projection seems to be smaller in size than the isometric view. An isometric projection and an isometric view of a cube are shown in Fig.3.5.

3.3.1 Isometric dimensioning

The general rules for the dimensioning of orthographic projection is applicable for isometric projection also, except the following:-

1. All the extension lines and dimension lines should be parallel to the isometric axes and they should lie on any of the isometric planes.
2. The dimensional values should be placed at the middle of the dimension line. The dimensional values in x direction should be readable from the right side, while those in y direction should be readable from the left side. The dimensional values in z direction should be readable horizontally from the right side.
3. It is usual practice to avoid the hidden lines, unless they are essential to make the drawing clear.

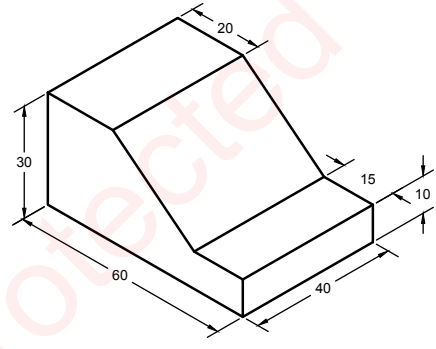


Fig.3.6: Isometric dimensioning

Fig.3.6 shows the recommended dimensioning layout for isometric drawing by BIS.

3.4 DRAW ISOMETRIC VIEW OF PLANE GEOMETRICAL FIGURES

Isometric drawing of plane geometrical figure can be obtained simply by keeping in mind that :

1. All lines that are parallel on the object are parallel on the isometric projection.
2. Vertical lines will be vertical in projection and horizontal lines will be at 30^0 inclined in projection.
3. The lines parallel to an isometric axis is called an isometric line and it is foreshortened to 82%.
4. The lines which are not parallel to any isometric axis, called as non-isometric lines, are not equally foreshortened. For example, diagonal MQ and SO [see Fig.3.3(a)] are of equal lengths in front view but are of different lengths in the isometric projection. The non-isometric lines are drawn by locating positions of their ends on isometric planes and then joining them.

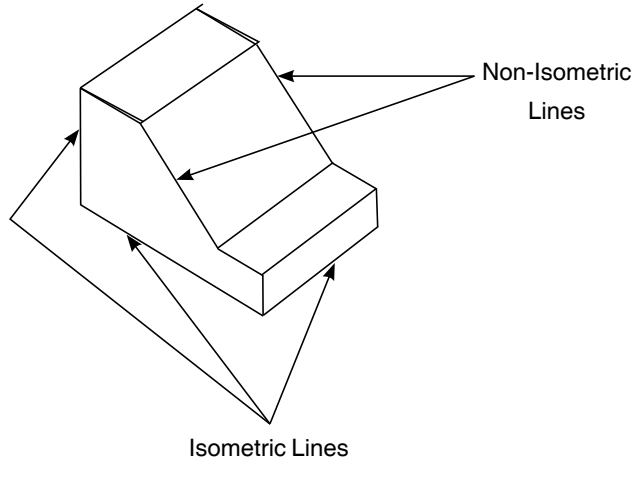
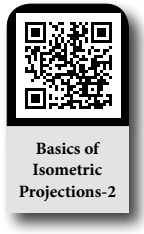


Fig.3.7: Lines in isometric projection

3.4.1 Methods of Constructing Isometric Drawing



The inclined lines of an object are represented by non-isometric lines in isometric projections. These are drawn by one of the following methods.

1. Box Method
2. Co-ordinate or Offset Methods.

Box method

In this method, the object is assumed to be enclosed in a rectangular box and both isometric and non-isometric lines are drawn by locating their respective points of contact with the surfaces and edges of the box. It is always helpful to draw or imagine the orthographic views first and then proceed for isometric drawing. It is used to draw prisms, cylinders and frustum of solids. An example of isometric drawing by box method is illustrated below:

Example 3.1 *Draw an isometric projection of a block with isometric lines only.*

Solution (by Box-Method)

To make the procedure clear a very simple object, containing only isometric lines is taken and Fig.3.8 shows the construction of the isometric projection by box method.

- At (a), the front and top view of the object are given.
- At (a') the three isometric axes are drawn, such that one of the axes is vertical (upward) and the other two making 30° angles with the horizontal.
- Along these axes, dimensions L, W and H are laid off, and as shown at (b) the isometric box is completed.
- Then, taking the measurements along the isometric lines, the details of the object are transferred from the orthographic view to the isometric box.
- Finally all the details which were drawn by light lines are darken to make the figure more distinct as at (c).

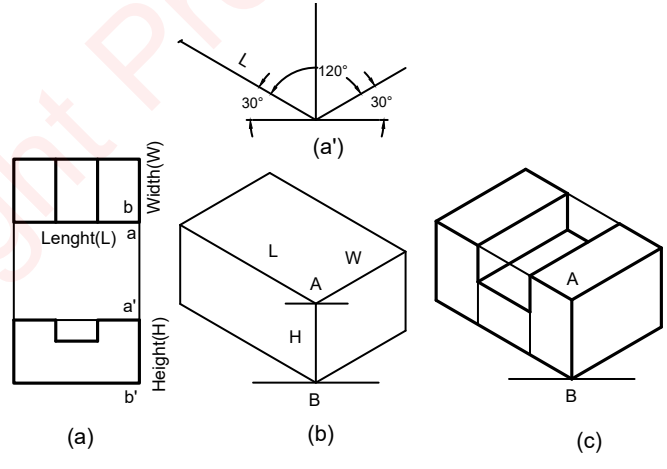


Fig.3.8: Draw an isometric projection of a block by box-method

Co-ordinate or Offset Methods

This method of making an isometric drawing is better suited to objects made-up of a number of planes at different angles or contains irregular curved surfaces. In case of planes, the ends of the edges are located by dropping perpendiculars from each point to an isometric reference plane. The perpendiculars may be located on the drawing, as these are isometric lines and the respective measurements are transferred from the orthographic views. In case of curved surfaces, the curved feature may be obtained by plotting the points on the curve, located by the measurements along

isometric lines. This method is used to draw isometric projection of pyramids, cones and truncated solids.

The box method is generally convenient for solving most of the problems.

Example 3.2 *Draw an isometric view of a square ABCD as shown in Fig.3.9 (a).*

Solution

The square can be projected isometrically in three different ways, as shown in Fig.3.9(b, c and d).

Case-1

Step-1. Draw a horizontal line and mark a point (b), on it. Draw a line ba, making a 30^0 angle, towards the left. The length is equal to the side of the square AB.

Step-2. From point a, draw a vertical line ad equal to AD.

Step-3. From point b, draw a vertical line bc, equal in length to BC.

Step-4. Join d and c by a straight line which is also inclined at 30^0 to the horizontal.

b c d a is the isometric drawing of the square ABCD and it gives a feeling as if we are viewing perpendicular to the line ba and from the left to the right side (Fig.3.9b).

Case-2

Another way in which the square could be projected is given in Fig.3.9(c). The procedure is exactly similar but the 30^0 angle is to be drawn on the right-hand side. This gives a feeling as if we are looking the square in a direction perpendicular to the side ab and from the right to the left.

Case-3

The third way is projecting the square horizontally. The procedure is as follows:

Step-1. Draw a horizontal line and mark a point a on the line.

Step-2. From point a, draw two lines, making 30^0 angles with the horizontal line but on the opposite sides.

Step-3. Cut lengths equal to the side of the square AB and AD. Draw parallel lines from these ends to the opposite sides, completing the rhombus abcd.

Step-4. The Fig.3.9(d) represents the isometric drawing of the square ABCD when the square is considered horizontal.

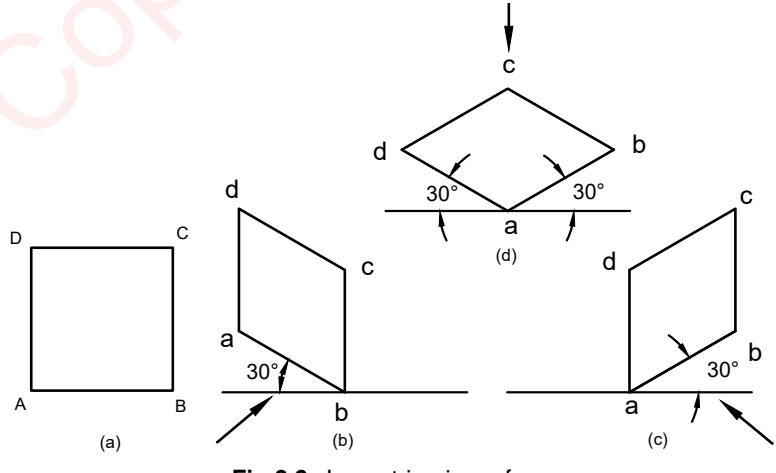


Fig.3.9: Isometric view of square

Example-3.3 *Draw angles in isometric as shown in Fig.3.10(a).*

Solution

Isometric angles cannot be measured directly with the protractor. Angles are constructed by locating the end points of inclined lines by measurements parallel to the isometric axes. In Fig.3.10(a), angle is given in degrees, it is necessary to convert it into linear measurement.

Step-1. Draw the "construction box" with isometric axes.

Step-2. Draw the triangle and convert 30° in linear measurement "A"

Step-3. This dimension "A" should be transferred with dividers to the isometric drawing and draw parallel inclined lines.

Step-4. Darken all final lines.

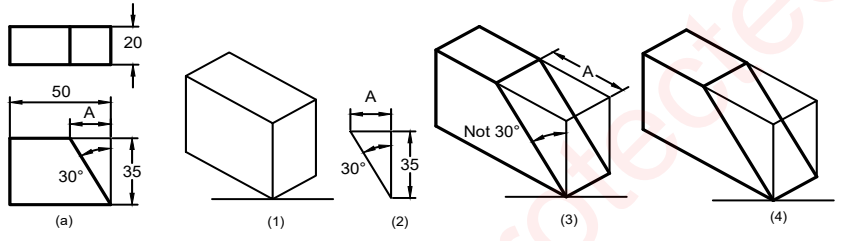


Fig.3.10: Angles in isometric

Example 3.4 *Draw an isometric view of a triangle ABC as shown in Fig.3.11(a).*

Solution

The triangle ABC can be iso-metrically projected in different ways, as shown in Fig.3.11(b) and (c).

Step-1. Side 'ab' is the isometric lines while 'bc' and 'ca' are non-isometric lines. Therefore, enclose the triangle in a rectangle 'abde'.

Step-2. Draw an isometric view of the rectangle 'abde'. For this,

i. As the surface of the triangle is vertical, draw a vertical line AE equal to 'ae'.

ii. Draw line AB equal to 'ab', inclined at 30^0 to the horizontal.

iii. Make the rectangle ABDE.

Step-3. Make a point C on the edge DE such that EC = 'ec'. Join AC, CB and obtain the required isometric view ABC of triangle 'abc'.

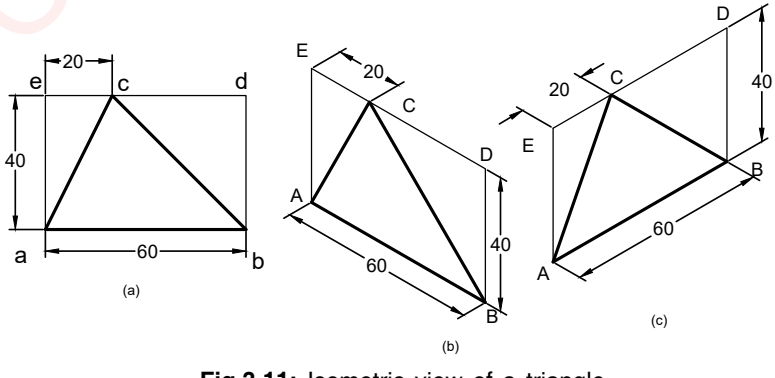


Fig.3.11: Isometric view of a triangle

Example 3.5 *Draw an isometric view of a pentagon with 30 mm length of side.*

Solution

Fig.3.12(i) shows a rectangular pentagon a,b,c,d,e of 30mm length of side, to draw the isometric view of a given pentagon.

Step-1. Enclose the given pentagon in a rectangle pqrs as shown in Fig.3.12(ii)

Step-2 . Draw the isometric view of rectangle pqrs and locate the corners abcde of the pentagon.

Mark points A and B on side QP, such that PA = pa, PB = pb

Mark a point E on side PS such that PE = pe

Mark point D as the midpoint of side SR.

Mark point c on side RQ such that RC = cr as shown in Fig.3.12(iii).

Step-3 Join all the points BA, AE, ED, DC and CB. This represents the required isometric view of given pentagon a,b,c,d,e shown in Fig.3.12(iv).

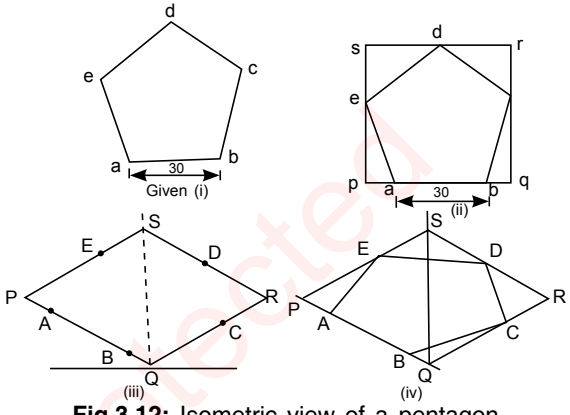


Fig.3.12: Isometric view of a pentagon

Example 3.6 *Draw an isometric drawing of a circle of 40mm diameter using four centre method.*

Solution

Four centre method is an approximate method of drawing isometric circles. The isometric drawings of a circle in a vertical plane and in a horizontal plane are shown in Fig.3.13.

Step-1. Construct the square ABCD enclosing the circle and touching it at 1,2,3 and 4. See Fig.3.13(a).

Step-2. When the circle is in a vertical position, draw AB at 30° to the horizontal and from A&B,

draw vertical line AD and BC. Join DC. Mark the mid points 1,2,3 and 4. Note the points

where A3 and C1 cut the diagonal BD at e and f.

Step-3. With these two points e and f as centres, draw arcs with radius e4(= f2) equal to "r" and with A and C as centres and radius A3 (= C1) equal to "R" draw arcs. 1,2,3,4 is the required isometric view of the circle, when it is in a vertical plane as shown in Fig.3.13(b).

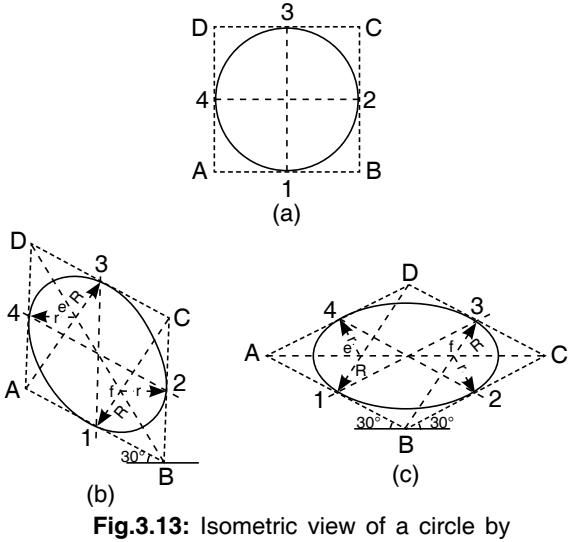


Fig.3.13: Isometric view of a circle by four centre method

In similar manner, the isometric view of the circle in a horizontal plane can be drawn. Here, AB and BC are at 30° to the horizontal line as shown in Fig.3.13(c).

Example 3.7 *Draw the isometric projection of a square plate of side 60 mm with fillet radius of 12 mm in left lower and upper corners as shown in Fig.3.14(a) as is visible from front.*

Solution

- Construct the square abcd using isometric scale.
- Draw the two isometric axes, one at 30° to horizontal and another axis vertical. Mark corner 'C' at the junction of the isometric axes and construct the rhombus ABCD as described earlier.
- Mark the points M and N at a distance equal to fillet radius 12 mm from corner A on AD and N on AB.
- Draw lines from M and N perpendicular to AD and AB to get the center C1. With the center C1, draw the arc MN, with the given fillet radius C1 M and the same is repeated to get the arc at lower left corner with the center C2. Hence, complete the projection.

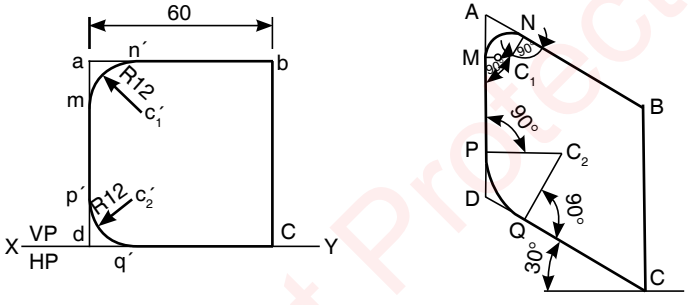


Fig.3.14: Isometric projection of a square plate

SOLVED PROBLEMS (WITH INCREASING DIFFICULTY LEVEL)

Problems related to Conversion of orthographic views into isometric view

Example 3.8 *Fig.3.15(a) shows the orthographic projections of an object. Draw its isometric view.*

Solution The required isometric view is shown in Fig.3.15(b).

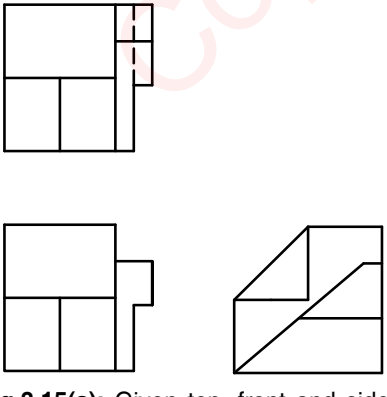


Fig.3.15(a): Given top, front and side views

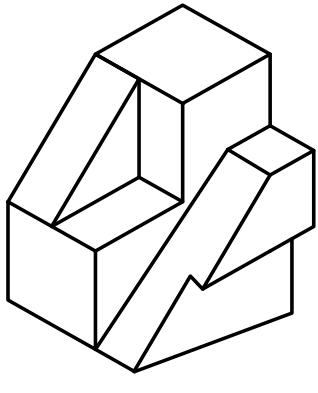


Fig.3.15(b): Isometric view

Example 3.9 *Fig.3.16(a) shows the top view and front view of cut block. construct the isometric view along with its dimension.*
Solution Fig.3.16(b) shows the required isometric view.

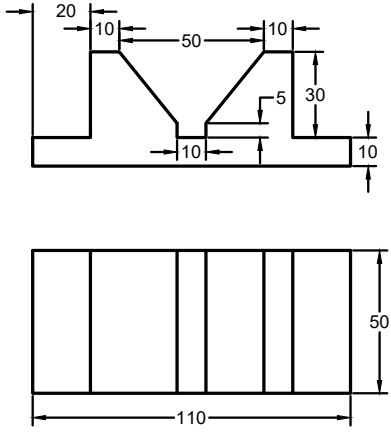


Fig.3.16(a): Given top and front views

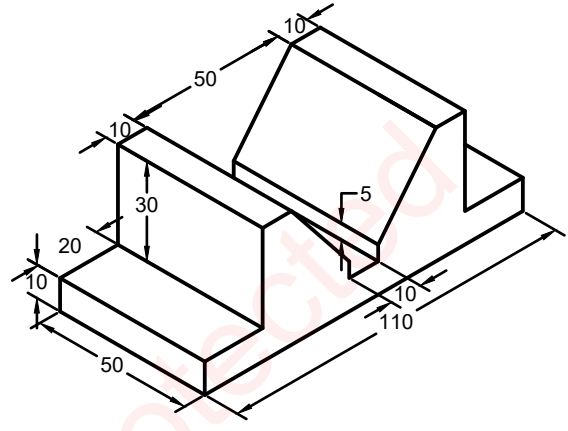


Fig.3.16(b): Isometric view

Example 3.10 *Fig.3.17(a) shows the top view and front view of an object. Construct the isometric view.*
Solution Fig.3.17(b) shows the required isometric view.

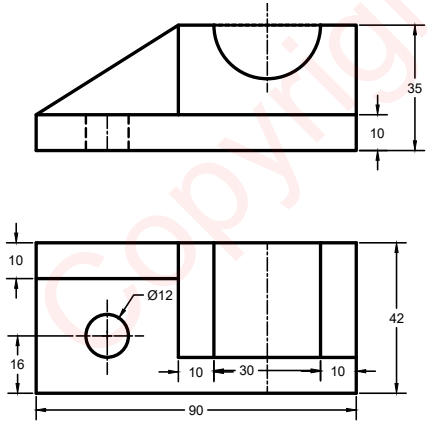


Fig.3.17(a): Given top and front views

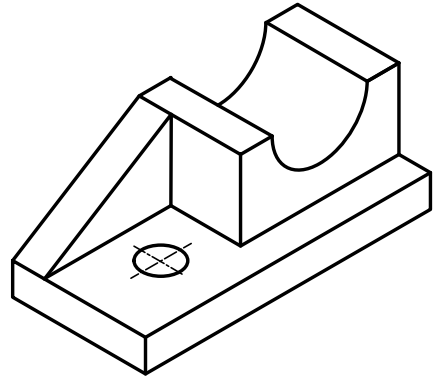


Fig.3.17(b): Isometric view

Example 3.11 *The top view and front view of the E block is shown in Fig.3.18(a). Draw its isometric view.*
Solution Fig.3.18(b) shows the required isometric view.

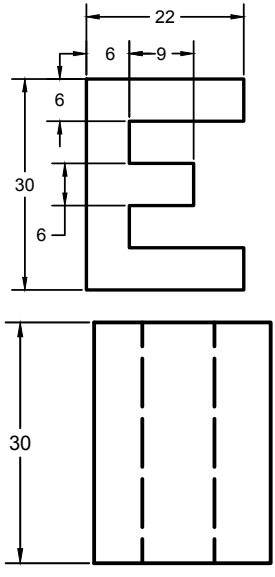


Fig.3.18(a): Given top and front views

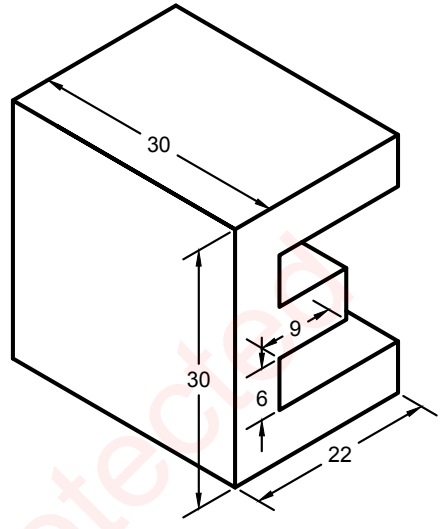


Fig.3.18(b): Isometric view

Example 3.12 *Fig.3.19(a) shows a square headed bolt of 25 mm diameter and 85 mm long with a square neck of 25mm thickness and bolt head no chamfer. Draw the isometric view of a square bolt.*
Solution Fig.3.19(b) shows the required isometric view.

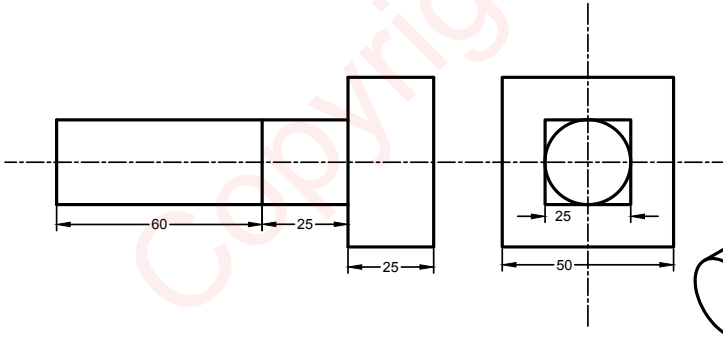


Fig.3.19(a): Given top and front views

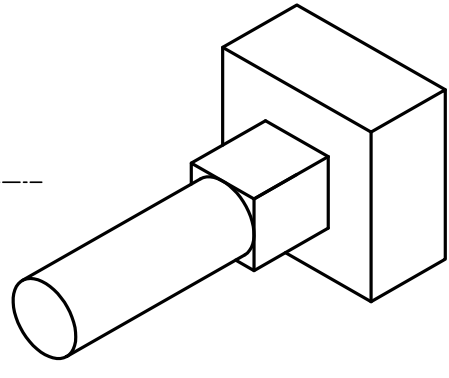


Fig.3.19(b): Isometric view

Example 3.13 *The top view and front view of an object is shown in Fig.3.20(a). Construct an isometric scale and draw its isometric projection.*
Solution Fig.3.20(b) shows the used isometric scale and Fig.3.20(c) shows the required isometric projection.

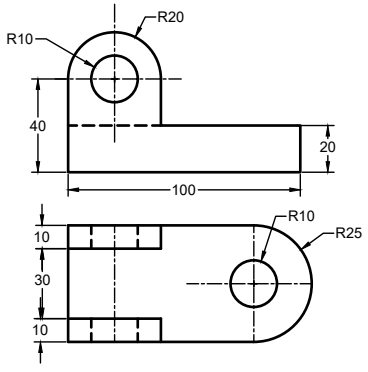


Fig.3.20(a): Given top and front views

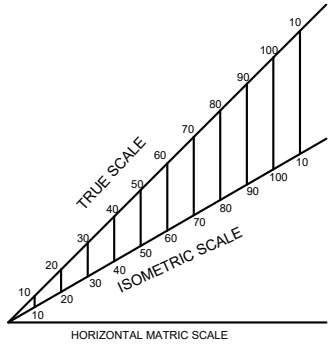


Fig.3.20(b): Isometric scale

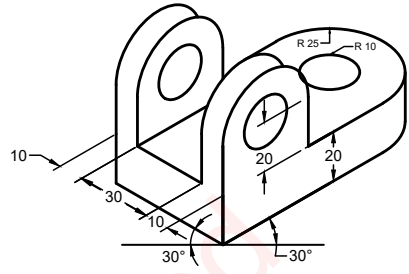


Fig.3.20(c): Isometric projection

Example 3.14 *The top view and front view of an object is shown in Fig.3.21(a). Draw the isometric projection.*

Solution Fig.3.21(b) shows the required isometric projection.

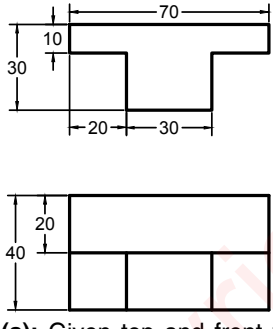


Fig.3.21(a): Given top and front views

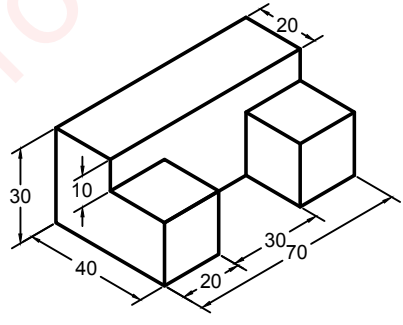


Fig.3.21(b): Isometric projection

Example 3.15. *The top view and front view of an object is shown in Fig.3.22(a). Draw the isometric projection. Enter dimensions on the isometric drawing.*

Solution. Fig.3.22(b) shows the required isometric projection.

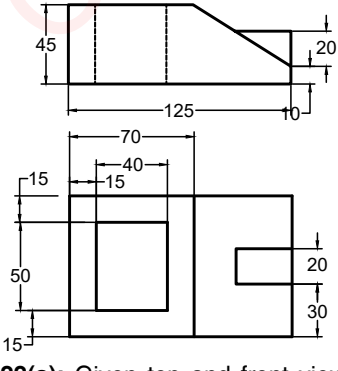


Fig.3.22(a): Given top and front views

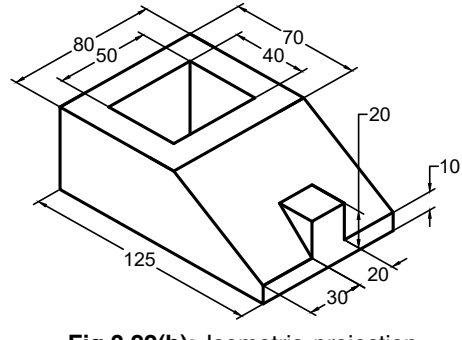


Fig.3.22(b): Isometric projection

Example 3.16 *Fig.3.23(a) shows the orthographic projections of an object. Using isometric scale, draw its isometric projection.*

Solution Fig.3.23(b) shows the required isometric projection.

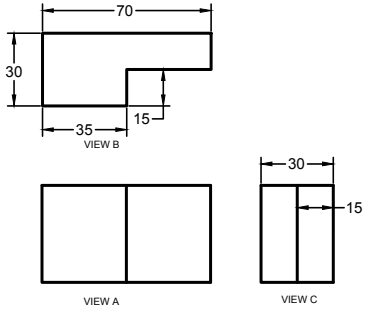


Fig.3.23(a): Given the orthographic projections

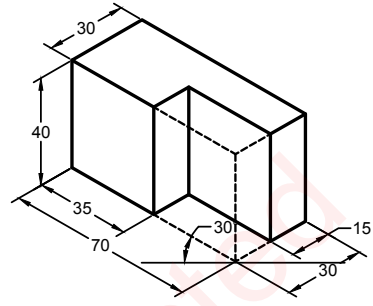


Fig.3.23(b): Isometric projection

Example 3.17 *Two views of a hut model are shown in Fig.3.24(a). Draw its isometric view. Need not dimension the isometric view.*

Solution Fig.3.24(b) shows the required isometric view

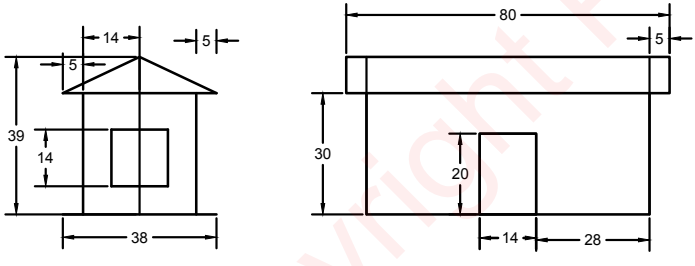


Fig.3.24(a): Given the two orthographic views

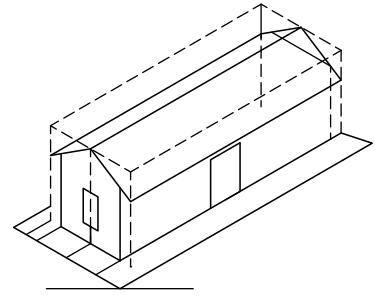


Fig.3.24(b): Isometric view

UNIT SUMMARY

1. Isometric projection/view is a 3-dimensional (pictorial) view/projection whose principal faces and axes are equally inclined to the planes forming an angle of 120° with each other.
2. Isometric projection is to be drawn with isometric scale.
3. Isometric views are to be drawn with True scale.
4. In isometric projection, all edges are equally foreshortened in the ratio $\sqrt{2} : \sqrt{3}$ to that of actual dimensions.
5. Isometric view of an object is larger than the isometric projection of the object.
6. Lines parallel to the isometric axes are termed as isometric lines. Lines which are not parallel to isometric axes are known as non-isometric lines.

7. Planes representing the faces of the cube as well as other planes parallel to these planes are called isometric planes. Planes not parallel to isometric planes are non-isometric planes.
8. Isometric projection of prisms, cylinders and frustum of solids are drawn using 'Box Method'.
9. Isometric projection of pyramids, cones and truncated solids are drawn using 'Co-ordinate' or 'off-set method'.
10. In 'Isometric dimensioning', it is usual practice to avoid the hidden lines, unless they are essential to make the drawing clear.
11. Isometric projection of a square is a rhombus and Isometric projection of a circle is an ellipse.
12. To draw isometric projection of a circle, 'Four centre method is an approximate method but comparatively simple method.

A. Objective Questions

EXERCISES

A1. Fill in the blanks with appropriate words

3.1 When a drawing is prepared with an Isometric scale is known as.....

3.2 The axes of an Isometric view are ..... degree apart to each other.

3.3 The lines which are parallel to the Isometric axes are known as .....

3.4 .....scale is used to convert the true length into Isometric Lengths.

3.5 In Isometric Projection, the vertical line on the object remains .....

3.6 Isometric view of an object is ..... than the Isometric projection of the object.

3.7 Isometric projection of a square is a ..... shape.

3.8 To draw isometric projection of a circle, ... method is comparatively simple method.

3.9 In Isometric projection, all edges are equally foreshortened in the ratio ..... to that of actual dimension.

3.10 ..... Projection is more suitable for small objects.

3.11 True lengths are used for drawing Isometric .....

Answers of Fill in the Blanks Questions

3.1 Isometric Projection; 3.2 120; 3.3 Isometric Lines; 3.4 Isometric; 3.5 vertical; 3.6 larger; 3.7 rhombus; 3.8 Four Centre method; 3.9 $\sqrt{(2)}:\sqrt{3}$; 3.10 Isometric; 3.11 View

A2. Multiple Choice Questions

3.1 Foreshortening term is not used for
(a) Isometric view (b) Orthographic View (c) Oblique View (d) Cabinet View

3.2 An Isometric scale is smaller than actual scale by
(a) 0.67 times (b) 0.71 times (c) 0.81 times (d) 0.89 times

3.3 Four Centre method is used to draw isometric view of a
(a) triangle (b) circle (c) square (d) rhombus

3.4 The angle that Isometric lines make with each other is
(a) 60° (b) 75° (c) 90° (d) 120°

3.5 On Isometric plane, a circle appears as
(a) square (b) quadrant of a circle (c) an ellipse (d) an involute

3.6 The utility of an Isometric Scale is
(a) to measure foreshortened lines (b) to project the object on projection plane
(c) to measure diagonal lines (d) to measure centre line

3.7 Pictorial views drawn on Isometric scale are called
(a) isometric view (b) isometric projection (c) isometric drawing (d) multi view drawing

3.8 Shape of a square in an Isometric plane remains appear as
(a) square box (b) parallelogram (c) square prism (d) rhombus

3.9 The appearance of an Isometric Projection as compare to an Isometric View is
(a) smaller (b) larger (c) more realistic (d) equal

3.10 The ratio between the isometric and true length is
(a) 2 / √3 (b) √2 / 3 (c) √2 / √3 (d) 1- (√2/√3)

3.11 Select the correct Isometric view corresponding to the orthographic view shown in Fig.3.25.

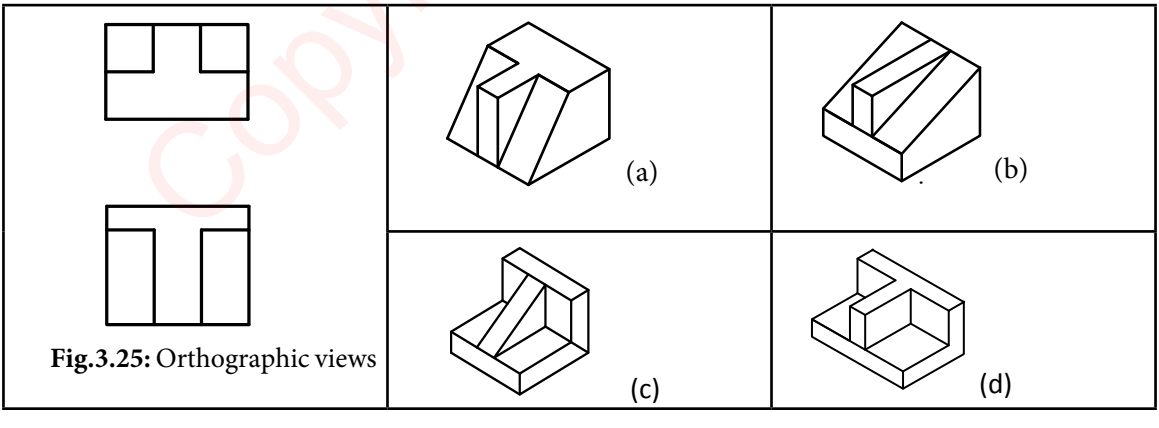


Fig.3.25: Orthographic views

Answers of Multiple Choice Questions

3.1 (b); 3.2 (c); 3.3 (b); 3.4 (d); 3.5 (c); 3.6 (a); 3.7 (b); 3.8 (d); 3.9 (a); 3.10 (c); 3.11 (d)

B. Subjective Questions

3.1 Define isometric line and non-isometric line with the help of a sketch.

3.2 Define isometric axes and isometric planes.

3.3 Differentiate between isometric projection and isometric drawing.

3.4 Determine the relationship between true length and isometric length.

3.5 Describe the meaning of foreshortening in isometric view.

3.6 Explain the utility of an isometric scale.

3.7 Describe the constructional steps to be followed to draw an isometric Ssale.

3.8 List the different methods to draw isometric projection. Describe offset method.

3.9 Explain the principle followed to draw an isometric projection of an object by box method with a neat sketch.

3.10 Write the principles of dimensioning in isometric projection.

3.11 State the merit of drawing isometric view.

3.12 Describe the four centre method of drawing isometric projection of a circle with a neat sketch.

3.13 Draw the isometric drawing of the components shown in Fig.3.26

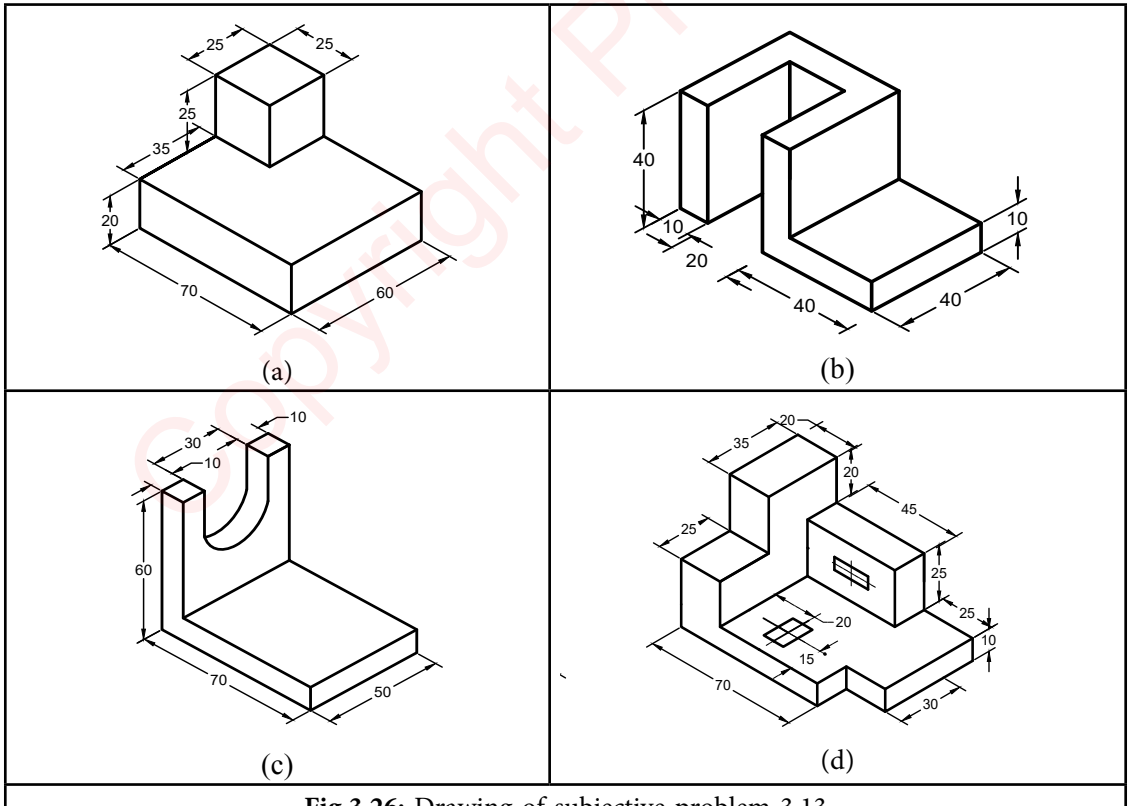


Fig.3.26: Drawing of subjective problem 3.13

3.14 Draw the isometric drawing of a rectangular prism of base 25 mm x 40 mm and the height 55 mm.

3.15 Draw the isometric view of square prism with a side of base 30mm and axis 50mm long when the axis is (a) vertical and (b)horizontal.

3.16 A cube, 30 mm edge, is placed centrally on the top of another square block, of 50 mm edge and 15 mm thick. Draw the isometric drawing of the two solids.

3.17 The top view and front view of an object is shown in Fig.3.27. Construct an isometric scale used and draw its isometric projection.

3.18 Fig.3.28 shows the front and top view of a tray. Draw its isometric view with dimension.

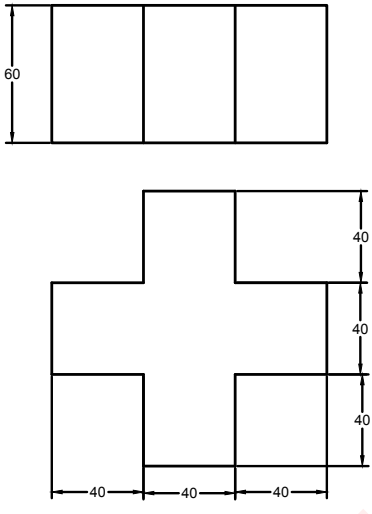


Fig.3.27: Drawing of subjective problem 3.17

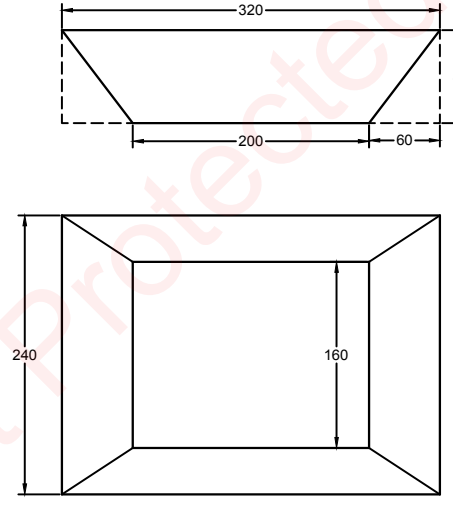


Fig.3.28: Drawing of subjective problem 3.18

3.19 Draw isometric view of the stepped cut block, from the given two view in Fig.3.29.

3.20 Two views of a pin support are shown in Fig.3.30. Draw its isometric view. Need not dimension the isometric view.

3.21 The front and top view of an ink bottle are given in Fig.3.31. Draw its isometric view without labeling the dimension.

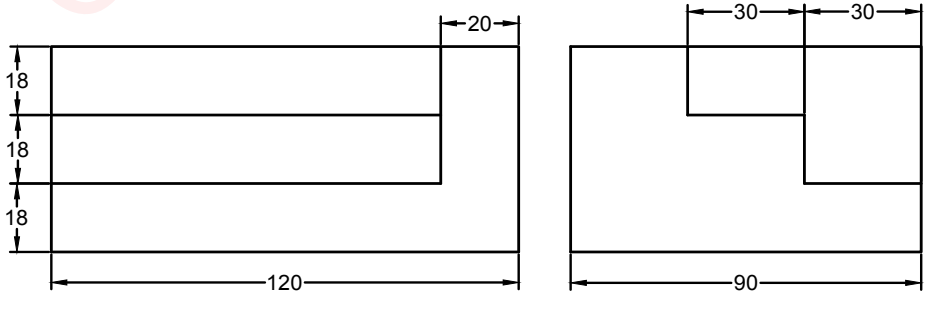


Fig.3.29: Drawing of subjective problem 3.19

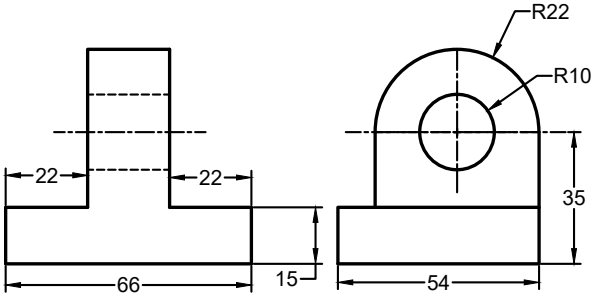


Fig.3.30: Drawing of subjective problem 3.20

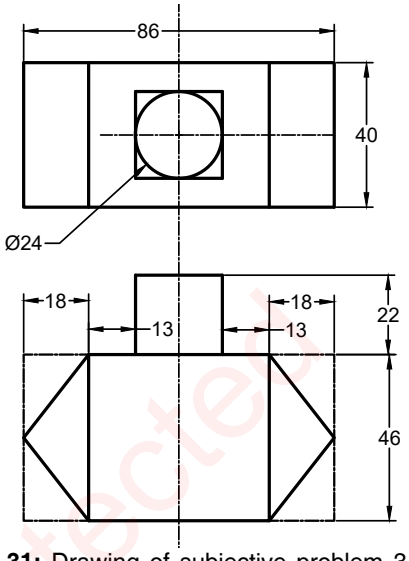


Fig.3.31: Drawing of subjective problem 3.21

PRACTICALS

As per curriculum the practicals related to Unit-3 are:

1. Draw two problems on Isometric view of simple objects having plain and slanting surface by using natural scale.
2. Draw some problems on Isometric projection of simple objects having cylindrical surface by using isometric scale.

Practical-1 Draw Isometric View of Object with Slant Surfaces

Practical statement

Draw two problems on Isometric view of simple objects having plain and slanting surface using natural scale.

Practical significance

Generally the orthographic views of an object are difficult to imagine and are separately drawn for top, front and side views. To visualize the object properly views like isometric, oblique and perspective are generally used. All the threes views are visible in a single isometric view and hence easy to understand. On the other hand for a designer, while developing new ideas, isometric sketching plays an important role in arriving at a final acceptable solution which can be shared with others and feedback can be taken even from a non technical person.

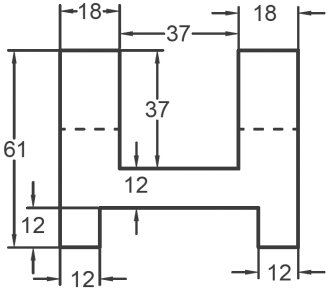
Relevant theory

Refer Section 3.0 of the book.

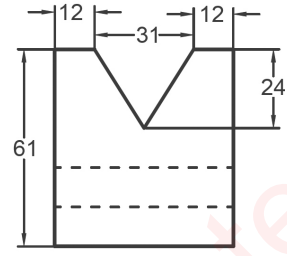
Practical outcomes (PrO)

PrO1: Draw isometric view from given orthographic views of the object with slant surfaces.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)



Front view



Right side view

Resources required

Same as mentioned in Practical-1 of Unit-1

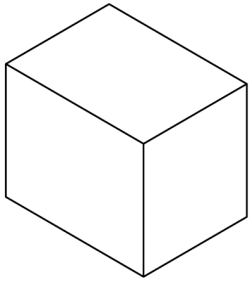
Precautions

Same as mentioned in Practical-1 of Unit-1

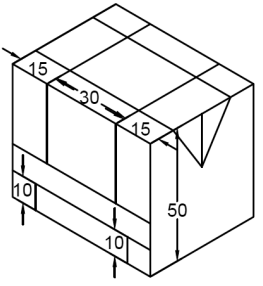
Isometric Projection-Object with slant surfaces

Suggested procedure

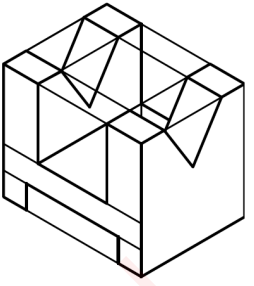
1. Study the given front and side view of the object.
2. Sketch the isometric axes.
3. Enclose the given object in a box of height as maximum height available in side or front view (50mm), of width as maximum width available in side view (45mm) and finally the length as maximum length available in front view (60mm). Sketch the box with light pencil (2H).
4. Remember that any vertical line in front and side view will remain vertical in isometric front and side view. Any horizontal line in front and side view will become 30° inclined in isometric front and side view while all line of top view will become 30° inclined from horizontal.
5. Draw the isometric view of the imaginary box keeping above rule in mind as shown in figure of Step-1.
6. The direction of arrow represents the front view of the isometric imaginary box. Now try to locate the points from orthographic front view to this front face of the box (as indicated by black bold dots).
7. Lay-off the distances in each face to locate all the features and points as shown in figure of Step-2.
8. Sketch the lines through these points and parallel both isometric axes as shown in figure of Step-3.
9. Complete the view by darkening the required lines as shown in observation section.
10. Dimension the final isometric view and add the necessary notes.



STEP 1

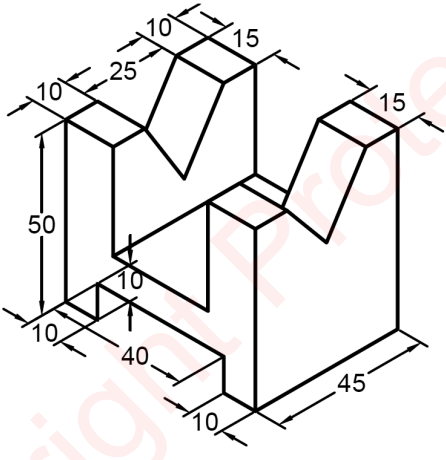


STEP 2



STEP 3

Observations



Practical related questions

1. Explain the difference between Isometric view and Isometric projection.
2. Explain the natural/True scale and isometric scale.
3. Explain the method to locate the root or notch point of the V in isometric view.

Disposal of waste

Same as mentioned in Practical-1 of Unit-1

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1 of Unit-1

Suggested learning resources

Refer Unit-3

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-1

Practical-2 Draw Isometric Projection of Object with Cylindrical Surfaces

Practical statement

Draw Isometric projection of simple objects having cylindrical surface using isometric scale. A frustum of a cone of base diameter 50mm, top diameter 30mm and height 45mm is resting upon its base on HP. Draw its isometric projection.

Practical significance

Generally the orthographic views of an object are difficult to imagine and are separately drawn for top, front and side views. To visualize the object properly views like isometric, oblique and perspective are generally used. All the threes views are visible in a single isometric view and hence easy to understand. On the other hand for a designer, while developing new ideas, isometric sketching plays an important role in arriving at a final acceptable solution which can be shared with others and feedback can be taken even from a non technical person.

Relevant theory

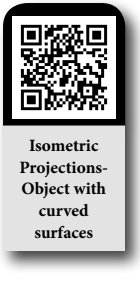
1. Refer section 3.0 of the book.
2. Difference between Isometric View and Isometric Projection (refer section 3.3 of the book).
3. Refer section 3.4 (Example 6.0) of the book.
4. All isometric lines are shortened to 82% of their original lengths. To get the length of Isometric lines multiply their original lengths by 0.82. Following figure shows the isometric lines of a cube of 1 unit sides.

Practical outcomes (PrO)

PrO1: Draw isometric projection from given orthographic views of the object with cylindrical surfaces.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

A frustum of a cone of base diameter 50mm, top diameter 30mm and height 45mm is resting upon its base on HP. Draw its isometric projection.



Isometric Projections-Object with curved surfaces

Resources required

Same as mentioned in Practical-1 of Unit-1

Precautions

Same as mentioned in Practical-1 of Unit-1

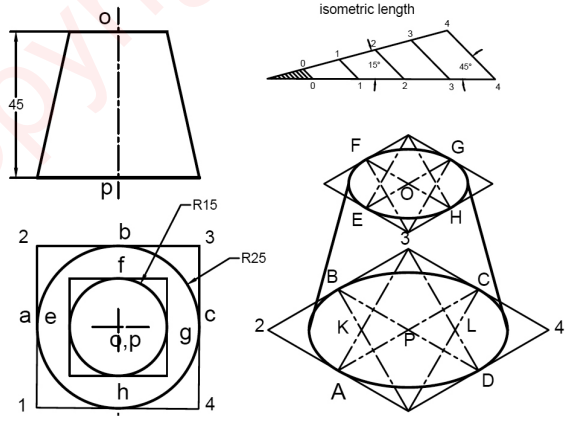
Suggested Procedure

1. Draw the Top and front orthographic views of the frustum of cone and enclose the circles of top view inside the squares.
2. Construct the isometric scale as outlined in section 3.2 of the book.
3. Sketch the isometric axes.
4. Draw the isometric square 1234 (Rhombus) for the base of the cone as shown in the figure.

All sides of the isometric square are isometric hence each should be of 0.82 x 50 = 41mm length.

5. Now construct the isometric (Ellipse) ABCD inside the isometric square 1234 using four center method.
   - Mark A,B,C and D as midpoints of the sides 1-2,2-3,3-4 and 4-1 respectively. Join (the ends of the minor diagonals) 1 to meet points B & C and 3 to meet points A & D.
   - Let B1 and A3 intersect at point K and D3 and C1 intersect at a point L. Then K, 3, L and 1 are the four centers for drawing the ellipse.
   - With center 1 and radius 1B draw Arc B-C. With center 3 and Radius 3A draw Arc A-D. With center K and radius KA draw Arc A-B. With centre L and radius LD draw Arc C-D.
   - These Arcs join in the form of an Ellipse ABCD which represents the required isometric as shown in the figure.
6. Now draw the vertical axis PO of length 0.82 x 45 = 36.9mm from the center of the base ellipse in upward direction.
7. To draw top isometric circle about center O, draw 30° inclined lines EG and FH through point O of length equal to 0.82 x 30 = 24.6mm each. Draw second Isometric Square 5678 (Rhombus) parallel to these lines and construct second isometric circle EFGH (Ellipse) inside this rhombus using same procedure as outlined in Step-5.
8. Draw the outermost generators of the frustum of cone as tangents to the two ellipses.
9. Finish the view using proper line type and dimensioning. All construction lines should be draw thin (with 2H/H pencil) and the final entities should be thick (with HB/B pencil)

Observations



Practical related questions

1. Explain the difference between Isometric view and Isometric projection.
2. Explain the natural/True scale and isometric scale.
3. Explain the method to locate the point O in isometric projection.

Disposal of waste

Same as mentioned in Practical-1 of Unit-1

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1 of Unit-1

Suggested learning resources

Refer Unit-3

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-1

KNOW MORE

- Teacher shall collect few production/construction/electrical/electronic actual industrial component drawings to show them to the students with input.
- During the input sessions the teacher must give examples related to the relevant branch viz. Mechanical and allied disciplines/electrical and allied disciplines/electronics etc.
- Teacher must know isometric view of objects having plain, slanting and cylindrical surface
- Show video/animation films to explain mentioned concepts.
- Use charts and industrial drawing/drawing sheets developed by experienced faculty to teach standard symbols and current industrial/teaching practices.
- Teachers should ask students to use url/qr codes available in the book for further understanding/practicing of the concepts.

Applications (Real life / Industrial)

Isometric drawings are extensively used by designers–particularly architects, product/industrial designers & engineers, as they are ideal for visualising concept of complicated structures, assemblies, components, machine tools etc.

Isometric drawings are extensively used by interior designers for furniture design, furnishing residential buildings etc. Objects drawn using isometric projection gives a clear understanding of the object. This can be anything from single point cutting tools to CNC machines, sneakers to office revolving chairs. To draw any component in isometric drawings the very first thing is to draw a isometric axes and/or isometric scale.

- Surf the web and list the isometric view used in industrial product
- Surf the web and list the isometric view used by architects and Builders
- Surf the web and list the isometric view used by interior designers and game designers

It is a standard practice of engineering companies to create isometric drawings for marketing and selling of their products/components. Some of the examples are as follows:

1. Mobile phone	8. Television
2. Bearings	9. Laptop
3. Car	10. Pumps
4. Cutting tools	11. Compressors
5. CNC machines	12. Water coolers
6. Washing machine	13. Furniture design
7. Refrigerators and Air conditioners	14. Flanges, Valves, Elbows and other fittings

Inquisitiveness and Curiosity

Other than the classroom and drawing practice sessions, following are the suggested student-related co-curricular activities which can be undertaken to accelerate the attainment of the various outcomes in this unit:

Student should maintain a separate A3 size sketch book which will be the part of term work and submit it along with drawing sheets. Following assignment should be drawn in the sketch book:

1. A wooden sitting chair; Laptop computer; Wall clock; Steel almira; Refrigerator; Dovetail stop; Staircase
2. Isometric views: Each student of the batch will try to collect at least one production drawings/ construction drawings/ plumbing drawings from local workshops/builders /electrical and mechanical contractors and try to generate isometric views from the orthographic views given in the drawings.
3. Isometric and orthographic views: Each batch will collect a single point cutting tool from workshop and draw its isometric and orthographic views with a suitable enlarged scale. In carpentry shop each batch will try to make wooden model from these views.
4. Isometric views: The teacher will assign one set of orthographic projections and ask the student to develop 3D thermocol models of the same.

REFERENCES & SUGGESTED READINGS

1. Engineering Drawing Practices for School and Colleges SP 46:2003, published by Bureau of Indian Standards, Government of India, Third Reprint, October 1998; ISBN:. 81-7061-091-2, Manak Bhavan , 9 Bahadur Shah Zafar Marg , New Delhi
2. Engineering Graphics for Degree, K.C.John , PHI publication,2010
3. A text Book of Engineering Drawing, K Venkata Reddy, BS Publication, 2008
4. NPTEL Course Material on Engineering Drawing, P.S. Robi, Department of Mechanical Engineering, Indian Institute of Technology Guwahati, India: https://nptel.ac.in/courses/112/105/112105294/

4 Free Hand Sketches of Engineering Elements

UNIT SPECIFICS

This unit presents information related to following topics:

- General Guidelines for freehand sketching
- Free hand sketches of machine elements
- Free hand sketches of orthographic view
- Free hand sketches of isometric view

The understanding of these topics will be developed after reading the contents, completing the solved problems, activities, performing exercises and viewing the ICT and web resources mentioned in this unit.

At the end of the unit, summary is provided to recapitulate the topics covered and applications are mentioned so that the learner can correlate the presented knowledge with real life and industrial situations. Few activities are mentioned to create inquisitiveness and curiosity in the learner. Subjective and objective questions are provided for the reinforcement of the knowledge and a list of references and suggested readings is also provided for further learning. Video resources along with QR codes are mentioned for getting more information on various topics of interest which can be surfed or scanned through mobile phone for viewing. Practical with details are provided to horn the drawing and drafting skills.

RATIONALE

After studying unit 2 and 3, you are well familiar with orthographic projection and isometric projection and also done enough practice work for the conversion of orthographic views into pictorial drawing and vice-versa. In this unit, we will discuss the application of principles of freehand sketching. Free-hand sketching plays an important role in the preparation of drawing in engineering field. Therefore, it is essential to the designer to records his clear and concise ideas in the form of sketches which are later converted into working production drawings.

It is usually assumed that skill in sketching may be acquired more easily than proficiency in instrumental drawings. This is not so because, both in sketching and instrumental drawings, the same basic principles of drawing must be employed. A lot of effort and practice are required to sketch say, two parallel lines, a circle, etc. then to draw the same. A student is therefore advised to acquire the sketching skills through constant effort and practice because later in his career, he must be in a

better position to express his ideas through, on the spot sketches, not only to his superiors but also to his sub-ordinates.

PRE-REQUISITES

Before reading this unit the student is advised to revisit following:

- Unit-2 and Unit-3 of this book.

UNIT OUTCOMES

After reading the contents, completing the solved problems, activities, exercises and viewing the web resources mentioned in this unit the students are expected to:

U4-O1: Sketch proportionate freehand sketches of machine elements such as nuts, bolts, studs, locking devices etc.

U4-O2: Draw sketches of orthographic views of the given objects on squared graph paper.

U4-O3: Sketch an isometric view from given orthographic views on isometric grid sheet.

Unit-4 Outcome	EXPECTED MAPPING WITH COURSE OUTCOMES *(1- Weak Correlation; 2- Medium Correlation; 3- Strong Correlation)*					
	CO-1	CO-2	CO-3	CO-4	CO-5	CO-6
U4-O1	-	1	2	3	3	-
U4-O2	-	1	3	3	3	-
U4-O3	-	1	3	3	3	-

4.1 GENERAL GUIDELINES FOR FREEHAND SKETCHING

Freehand sketching is one of the most effective ways to communicate a pictorial or verbal idea to a workman. For freehand sketching of an object, consider the following general guidelines

1. When sketching, use a suitable paper and pencil.
2. Carefully examine the object to be sketched and describe the shape of the object completely.
3. Showing the relative parts in fair proportion but not to any particular scale.
4. Add all essential dimensions wherever necessary.
5. Have notes to specify, for example, the material used, manufacturing methods etc.

4.1.1 Materials for Sketching

Freehand sketches show generally what a viewer visualises rather than technical details. The main sources of freehand sketching are imagination and modifying existing sketches. These freehand sketches are drawn with minimum resources, which include the following things:

- **Pencil** - The pencil should be soft enough to give a good texture on a medium rough paper. Though any of the soft pencil in the range F to 2H can be selected but the HB grade pencil is mostly preferred.
- **Paper** - Depending upon the condition of work and the purpose of drawing, a variety of papers are used for sketching. An engineer working in a design office should use a drawing sheet or sketch paper of a good quality. For sketching the different views of a component and to produce dimensioned sketches, squared graph paper (Fig.4.1) is used. Similarly, isometric grid sheet (Fig.4.2) is used to prepare isometric 3 dimensional sketches. In the beginning, one can practice freehand sketching using plain paper, so that develop a good sense of proportion, accuracy of observation and proper use of pencil work.
- **Eraser** - A good quality soft eraser is required to make the sketches neat and clean.

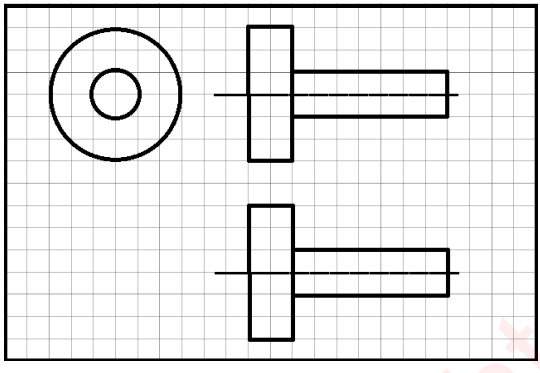


Fig.4.1: Squared graph sheet

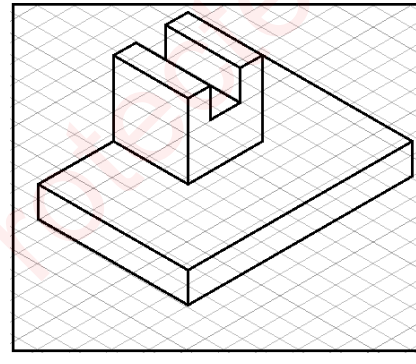


Fig.4.2: Isometric grid sheet

4.1.2 General Sketching Elements to Draw Shapes of Objects

The shapes of objects are made-up of flat and curved surfaces, which are represented by straight and curves lines and hence the student must first develop skill in sketching these elements rapidly and accurately.

4.1.3 Sketching of Straight Lines

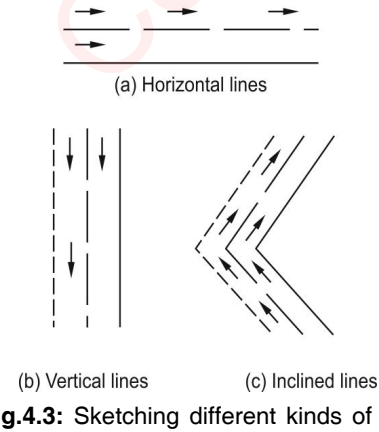


Fig.4.3: Sketching different kinds of line

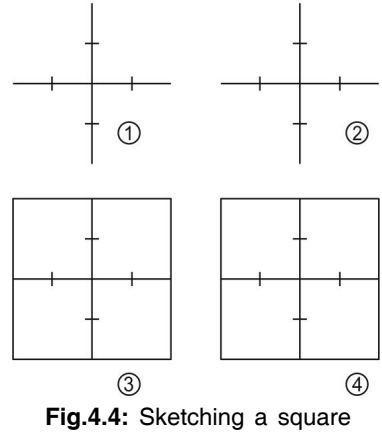


Fig.4.4: Sketching a square

The first step in making any straight line is to determine its length by lightly marking its end points. Sketchy dashes are then tried between these end points, correcting the angle of direction in each stroke. The gaps are then closed in stages and the line is thin darkened to full intensity. Fig.4.3 indicates the directional motion to be adopted for drawing different kinds of lines such as horizontal, vertical or inclined.

4.1.4 Sketching a Square

Fig.4.4 indicates the steps to be followed in sketching a square.

4.1.5 Sketching of Circles and Arcs

Small circle and arcs can be drawn easily in one or two strokes without using any guiding block. There is a systematic method for sketching large circle and arcs.

One method of sketching a large circle, is estimate the radii and marked it on horizontal and vertical centre lines to locate a square. Sketched inside the square as shown in Fig.4.5.

In another method, the radii are estimated and marked off on the lines indicated to locate the points through which the circle will be sketched. See Fig.4.6.

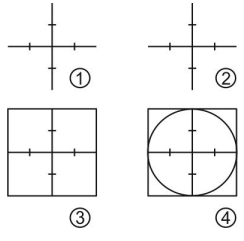


Fig.4.5: Sketching a circle

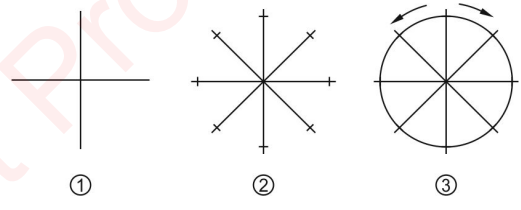


Fig.4.6: Another method of sketching a circle

4.2 FREEHAND SKETCHING OF MACHINE ELEMENTS

All machines are made of several component parts. The devices used for holding together two parts of a machine or structure are called fasteners. The fasteners are classified into two main categories:

1. Temporary fasteners and
2. Permanent fasteners

A screw thread is the functional element used on a bolt, stud, set-screw or any other threaded piece or components. All screwed pieces fall under the category of temporary fasteners. Temporary fasteners are widely used in joining the machine parts and other engineering products where frequent dismantling is required. Nuts and bolts, screws, studs and pins are examples of temporary fasteners, whereas riveting, welding, soldering and brazing etc. are permanent fastening.

4.2.1 Parts of a Screw Thread

In a study of threads and their representation on a drawing, it is necessary to define a few important terms used in specifying any thread. Different parts of a screw thread are shown in Fig.4.7 and defined below.

Outside or major diameter: It is the diameter at the crest of the thread measured at right angles to the axis of the screw.

Core or minor diameter: It is the diameter at the core or root of the thread. It is the smallest diameter of the screw and is equal to the outside diameter minus twice the depth of the thread.

Effective diameter: It is equal to the length of the line, perpendicular to and passing through the axis, and measured between the points where it cuts the flanks of the thread.

Nominal diameter: It is the diameter of the cylindrical piece on which the thread is cut. The screw is specified by this diameter.

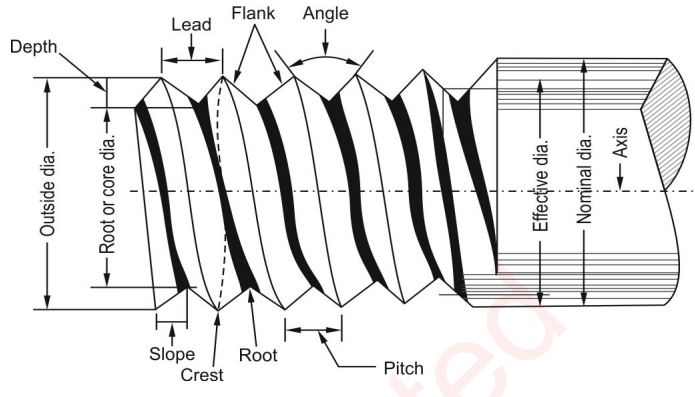


Fig.4.7: Different parts of a screw thread

Pitch: It is the distance measured parallel to the axis, between a point on one thread form and a corresponding point on the adjacent thread form, i.e. from crest to crest or root to root. It may also be described as the reciprocal of the number of thread forms per unit length, i.e. P = 1/N.

Root: The root is the inner-most or bottom surface created by two adjacent flanks of the thread.

Crest: It is the outer-most or top surface part of the thread.

Flank: The surface between the crest and the root is called the flank of the thread.

Angle: It is the angle between the flanks measured on an axial plane.

Lead: It is the distance measured parallel to the axis from a point on a thread to a corresponding point on the same thread after one complete revolution. It can also be described as the distance moved by a nut in the axial direction in one complete revolution. The lead is equal to the pitch in case of single-start threads.

Depth of thread: The depth is the distance between the crest and the root, measured at right angles to the axis. It is equal to half the difference between the outside diameter and the core diameter.

Slope of thread: It is equal to one-half the pitch of the thread.

4.2.2 Types of Thread Profile

It is the shape of a thread on a section plane containing the axis of thread. It is often called the profile of thread. Basically two types of profiles are there, one is square and another is triangular. All the available and used types of profiles are minor modifications of either square or triangular thread. Each type has different application. Pitch of the profile depends upon the nominal diameter. Fig.4.8 shows various profiles, their shape and proportions in terms of pitch. Use of each profile is discussed below.

V thread (metric thread)

The Bureau of Indian Standards has adopted V-thread profile recommended by the International Organisation for Standards (ISO) a metric screw thread for use in our country.

This types of screw threads are also known as Unified Thread. It has a 60° thread angle between the

adjacent inclined surfaces. Since such a profile has sharp edges that can cause injury while using and hence avoided.

A practical modification of this groove is a metric thread in which the sharp crest is truncated. External threads are flat at crest of length P/8 and root is rounded with radius of 0.1P (Fig.4.8b). This profile is used by most of the thread fasteners.

British Standard Whitworth (B.S.W) thread

A similar shape as that of V threads, but the included angle as 55^0 is used for British Standard Whitworth thread (Fig.4.8c). The depth of thread is 0.64P and radii at root crest are 0.14P.

Square thread

Since these types of screw threads are in the form of a square, it is called square thread. The flanks or the sides of this thread are perpendicular to the axis of the thread. The depth and thickness of the thread are equal to half of the pitch (Fig.4.8d). A modified square thread has a slight taper of 5^0 on the both flanks to form included angle of 10^0 (Fig.4.8e).

It is the simplest and strong type of thread profile and hence used for transmission of motion and power as in vices, clamps etc. and for converting a rapid rotary motion into slow linear motion, for example, the lead screw of a lathe, screw presses, jacks etc.

Acme thread

An Acme thread is a slight modified form of a square thread and the sides are inclined at 14.5^0 (Fig.4.8f). A half nut is used to connect and disconnect the power from these screws. It is strong thread because of wider base with the added advantage of easy operation of engagement and disengagement of the half nut due to the inclined surfaces.

It is suitably used for power screws like the breaks screw, jack screws and on the value operating screw for axial power transmission.

Buttress thread

Buttress thread is designed to transmit power in one direction only. It is commonly used in large guns, in jacks and in other mechanisms that have high-strength requirements. It has efficiency of square thread and strength of V-thread, earlier it was produced with one phase as vertical, but never profile have slope of 7^0 with vertical (Fig.4.8g), while other face at 45^0.

Knuckle thread

A semi-circular profile of radius P/4 is used for these threads (Fig.4.8h). This profile is suitable if threads are to be formed by moulding or rolling like sheet metal covers for jars.

Thread presentation

True representation of thread is difficult and time consuming and hence not used. In true representation, crest and root lines are helix, which is laborious. In drawing threads are generally represented by simplified or conventional method. Fig.4.9 and 4.10 shows the simplified and conventional representation of thread.

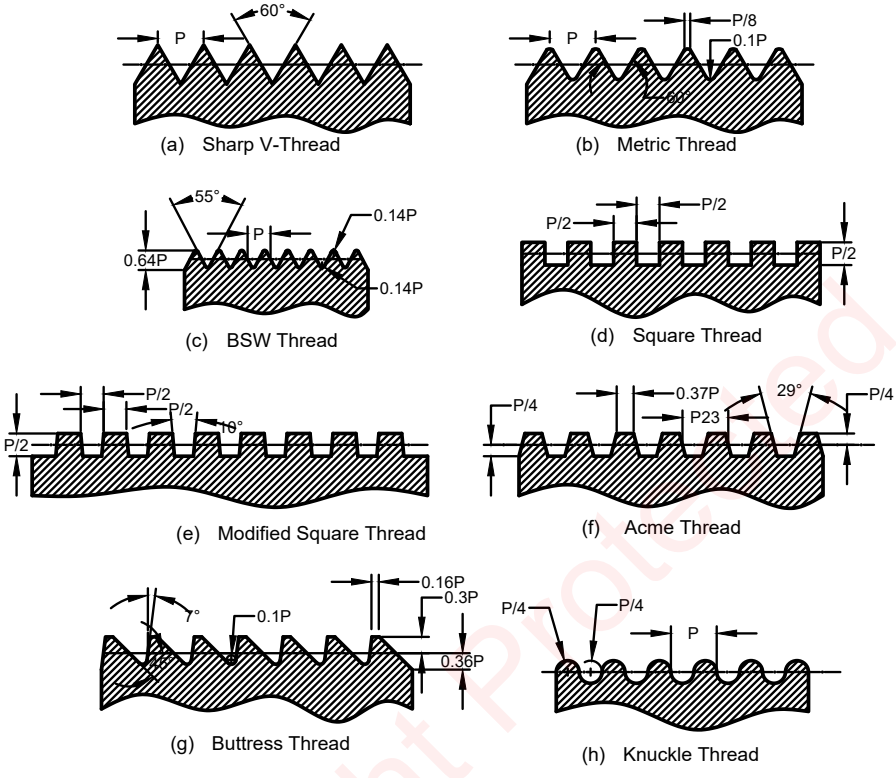


Fig.4.8: Different types of thread profile

Simplified representation of thread

In simplified method, the V profile of V threads is not drawn. The crest lines are represented by thin lines at distance P, inclined at helix angle and root lines by thick line in between the thin lines (Fig.4.9). In the side view, the outer diameter is drawn by full circle, while the root diameter is drawn a circle slightly more than 3/4 circle. Gap of the circle for root diameter can be shown in any position.

Conventional representation of threads

This is the simplest method. A thin line parallel to the major diameter at distance equal to the depth of the thread is drawn (Fig.4.10). It gives no idea about the hand of helix and start of threads, etc., hence these are to be mentioned separately by a note. If not mentioned, it is presumed that it is a single start, right handed thread.

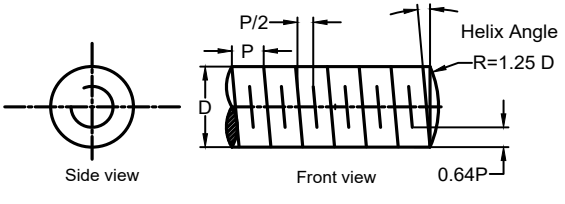


Fig.4.9: Simplified representation of thread

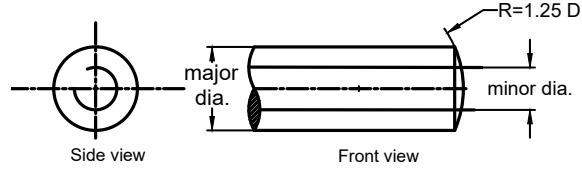


Fig.4.10: Conventional representation of thread

4.2.3 Drawing a Bolt, Nut and Washer

The bolt and nut joint or screwed fastening is a temporary joint. A bolt is a cylindrical part having a hexagonal or square head on one side and threads, partially or completely on the cylindrical part. The counter part of the bolt is a nut that is also either hexagonal or square and has internal threads with same pitch as that of the bolt. The parts to be joined together are drilled with a hole of the same size as the body diameter of the bolt or slightly more for easy insertion. The bolt is then passed through the parts to be joined and then the nut is tightened on the threads of the bolt. When the nut is rotated, it advances on the threaded body of the bolt and clamps the parts.

A washer is placed below the nut so that the surface of the part is not spoiled due to tightening of the nut. It also increases the contact area, thereby reducing the local intensity of stress. Fig. 4.11 is a display of pictorial view of hexagonal bolt with hexagonal nut and washer. Fig. 4.12 is a detailed drawing because it gives the required details of individual parts of an assembly of hexagonal bolt, nut and Fig.4.13 shows orthographic views of the same assembly.

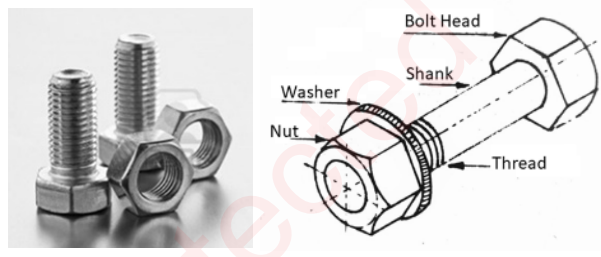


Fig.4.11: Pictorial view of hexagonal bolt with hexagonal nut and washer

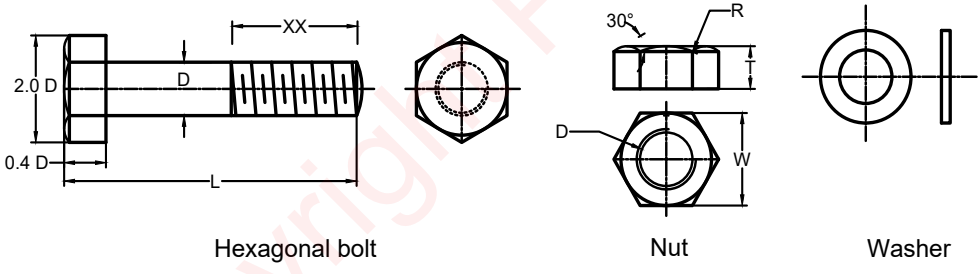


Fig. 4.12: Detailed drawing of hexagonal bolt, nut and washer

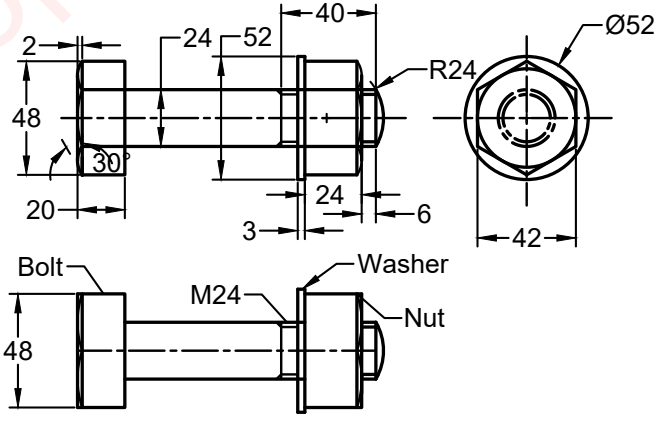


Fig.4.13: Orthographic views of hexagonal bolt, nut and washer

4.2.4 Studs

Studs are used for components that are removed frequently like cylinder heads, cover etc. A stud does not have a head and is cylindrical in shape having threads of the same hand (generally right hand) on both sides as shown in Fig.4.14. The length of the stud can be divided in three areas:

1. **Metal End:** this is fully tightened in one of the components. Its length varies from D to 1.5D.
2. **Plain Portion:** It is the unthreaded portion of the stud. Its length is about 0.5 D, but can have more length also. This portion accommodates other parts.
3. **Nut End:** The nut is tightened at this end. Its length is kept as 2D+ 6 to 25mm.

The metal end is tightened into the tapped hole (threaded hole) of the body. The remaining portion guides the removable part to be positioned. Then the nut is tightened. Engine cylinder heads are tightened using studs and not bolts.

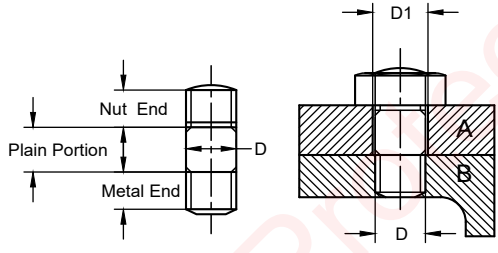


Fig. 4.14: A stud

4.2.5 Screws

A screw is smaller in size both for diameter and length than a bolt and has threads for the entire length of the shank. The head is not hexagonal but circular. The various shapes of the heads used for the screws are shown in Fig.4.15. There are many types of screws and are described below.

Cap Screw: When a screw is tightened in tapped hole, it is known as cap screw.

Machine Screw: If a screw uses a nut on the other end, it is called as machine screw (Fig.4.16). They have a good finish and are used for light applications. They are made of steel or brass.

Set Screw: A set screw is tightened in a tapped hole in an outer part to press the inner part to keep it in position, e.g., a screw in the knob of a microwave, washing machine etc. The ends are therefore given a pencil shape like cone, dog or cup as shown in Fig.4.17.

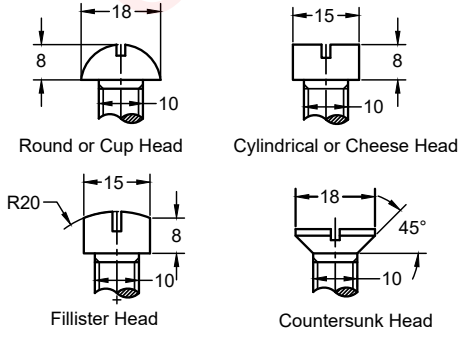


Fig. 4.15: Shapes of screw heads

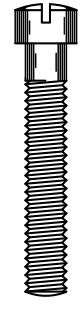


Fig. 4.16: Machine screw

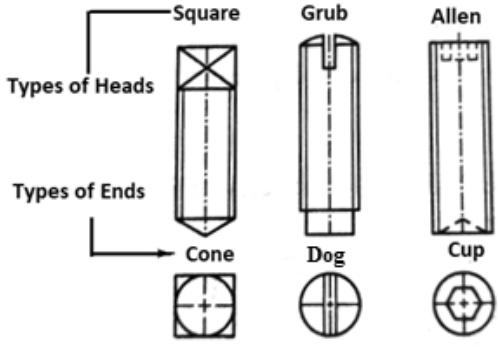


Fig. 4.17: Types of screw ends

4.2.6 Locking arrangements for Nuts

When bolts and nuts are used in machine elements having vibrations, there are chances of getting them loosened. The loosening can go to the extent that the nut may come out completely and cause hazards. To keep the nut at the tightened position, locking devices are used. A large number of locking devices are available, some of which are as follows:

1. Jam Nut or lock nut: A lock nut is an additional nut placed theoretically below the main nut. This is about one half or two third thickness of standard nut.
2. Castle nut: It is a hexagonal nut with cylindrical upper part. This part is slotted in line with the centre of each face. A split pin is inserted through two slots in the nut and a hole in the bolt. This used in automobile industry.
3. Sawn nut: It has a slot sawn half way through. After the nut is tightened, the small screw is screwed which produces more friction between the nut and the bolt preventing the loosening of the nut.
4. Penn, ring or grooved nut: It has a upper hexagonal part and a lower cylindrical part. The bottom cylindrical portion is recessed to receive the tip of locking set screw.
5. Locking with pin: The nuts are locked by means of taper pin or cotter pin.
6. Locking with plate: A plate or locking plate is used to lock the bolt.
7. Spring lock washer: As the nut is tightened, one edge of the washer will be digging itself in the that piece thus increasing the resistance so that the nut will not be loosened.

Some of the locking devices are shown in Fig.4.18.

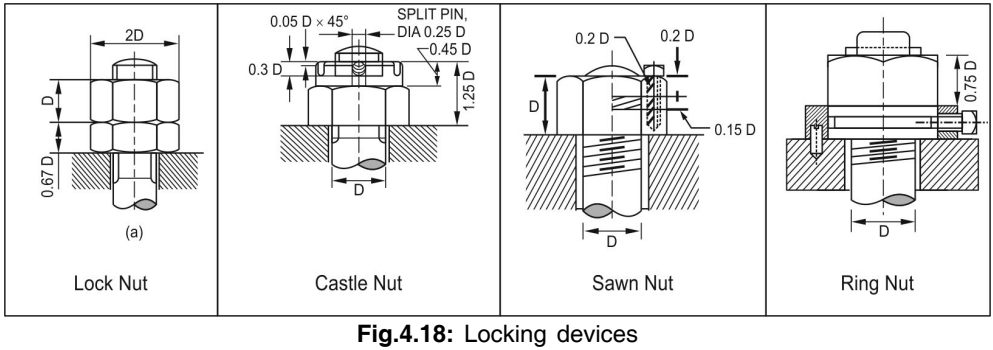


Fig.4.18: Locking devices

4.2.7 Sketching Machine Parts

When making a technical sketch of a machine part begin with constructing its overall outlines (the so-called "dimension cage") in light lines. In our example [Fig.4.19(a)] such a dimension cage is a rectangular box. Since any machine part may be thought of as consisting of separate elements which resemble some geometrical solids, sketch all elements step by step until the whole machine part is obtained. Finish the sketch by adding surface shading, as in Fig.4.19(b).

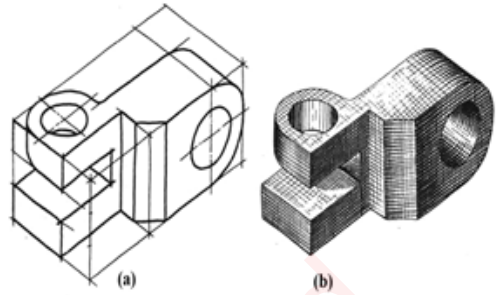


Fig.4.19: Typical machine element

When sketching a machine part from a model, for example, a cam, as in Fig.4.20(a), the student, while carefully scrutinizing its shape, must also compare the size of its separate elements, thus developing his sense of proportionality. Presented in Fig.4.20 are two technical sketches made from one and the same model. It is quite obvious that the sketch in Fig.4.20 (b) is incorrect, since proportionality is not observed. The distance b is not uniform and therefore the line of intersection (a portion of an ellipse) of the flat lug with the cylinder is incorrect. As compared with the model, dimension k is considerably enlarged, the thickness of the cylinder wall is reduced, as is the depth of the slot e. The dimension e must be approximately 2.5 times larger than a, while in Fig.4.20(b) they are equal. Fig.4.20(c) gives a correct version of the sketch.

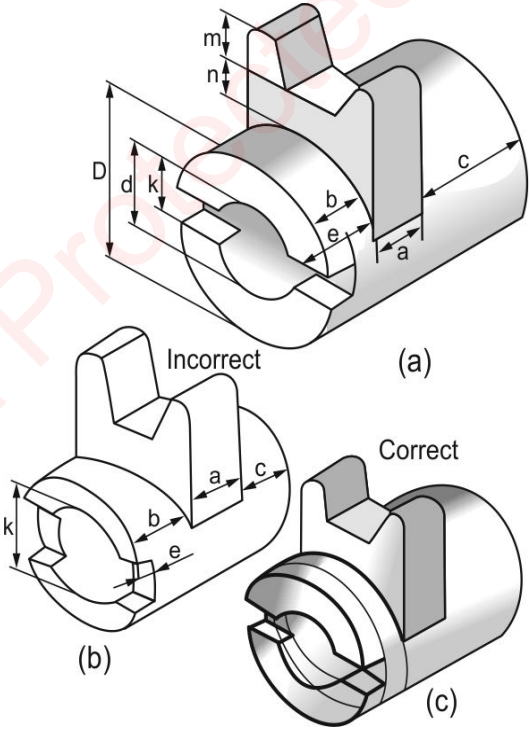


Fig. 4.20: A cam

4.2.8 Hints on drawing a Hexagonal nut

1. Work out the required dimensions that is nominal diameter of bolt, height of nut, size across flats, radius of front chamfer and radius of crest of the thread etc.
2. Draw the centrelines, horizontal and vertical, to locate the centre of circled into view.
3. Commence drawing from top view, as it contains circles. Draw the two circles for root and crest of threads and also draw the third circle for chamfer on the nut.
4. Outside the chamfering circle, complete the hexagon.
5. Project the corners of the hexagon, thus drawn, to front view and draw two parallel horizontal line apart by height of nut, already calculated, to cut these projections.
6. Draw the front chamfer arc in front view and project its point of intersection with the two inner vertical projections, horizontally.
7. Locate the centre of the remaining arc and complete the views.

4.3 FREE HAND SKETCHES OF ORTHOGRAPHIC VIEW

Theory of orthographic projection has already been discussed in detail in unit 2. All the rules and conventions discussed there are applicable in orthographic sketching. Orthographic sketches are made either on plain paper or squared graph sheet.

Orthographic sketching may require sketching any combination of the six principal views of the object. Usually the front, top and right side views are selected for describing an object in an orthographic sketch. In general, a combination of views that show the most detail and the least number of hidden lines should be chosen.

The procedure followed in making a sketch is almost the same as that in drawing with instruments. Though the sketch is not made to scale, yet it should be fairly proportionate and should be according to all the conventional practices of projection. A well-prepared sketch should give the same good appearance as a good instrument drawing. The pictorial view of an object is given in Fig.4.21 (a). The steps to be followed in making a freehand sketch of orthographic views of a given object are illustrated below in Fig.4.21.

Step 1. Study the given object until its shapes and functions are understood. Decide the views which best describe the shape of the object.

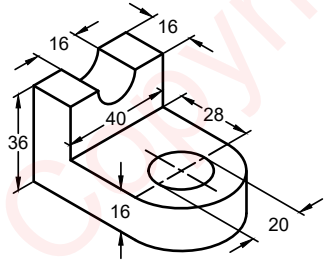
Block in the views in light lines and space the views to give a balanced appearance to the sketch.

Step 2. Add the details in each view in light construction lines, so that they need not be erased.

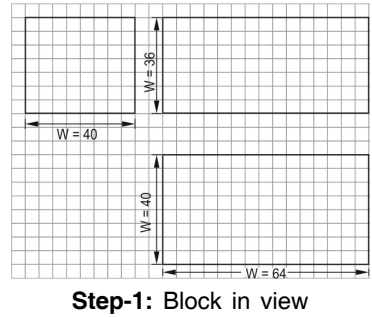
Step 3. Sketch the required arc and circle in each view by any one of the method as shown in Fig.4.5 and 4.6.

Step 4. Sketch the additional required lines and brightening the circle and arc in each view.

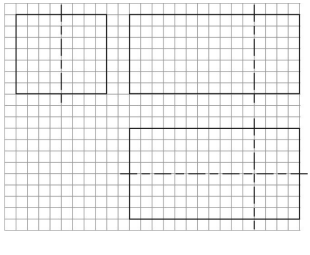
Step 5. Sketch the required hidden lines, brightening the visible lines, add the necessary dimensions and name the views of the drawing.



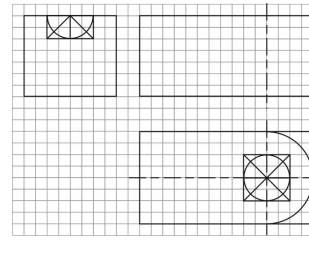
Given (a)



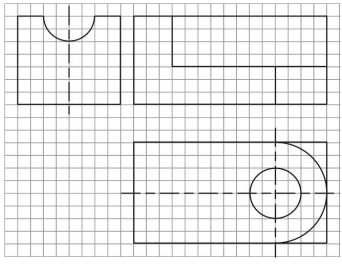
Step-1: Block in view



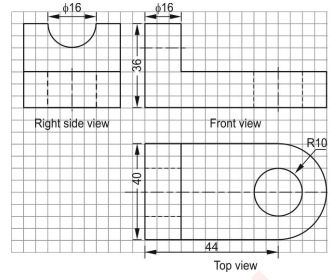
Step-2: Add details in light lines



Step-3: Construct arc and circle



Step-4: Add necessary lines



Step-5: Add necessary hidden lines, dimensions and name the views

Fig.4.21: Steps in sketching of orthographic view

4.4 FREE HAND SKETCHES OF ISOMETRIC VIEW

Theory of isometric drawing has already been discussed in detail in unit 3. All the rules and conventions discussed there are applicable in isometric sketching. Isometric sketches are made either on plain paper or isometric grid paper. Though any of the methods used in making isometric sketching can be adopted for making isometric sketches, yet the Box-construction method is mostly preferred.

The procedure followed is the same as that used in isometric drawing. Two orthographic views of the object are given in Fig.4.22(a). The following steps are usually used for freehand sketching of isometric view:

Step 1. Study the orthographic views of the object and decide the position in which it should be placed to reveal the maximum details. Lay out the isometric axes, which should be nearly 120˚ apart from each other, i.e. one axis vertical and the other two at 30˚ (approximately) with horizontal.

Step 2. Taking the proportions length, width and height along the isometric axes, complete the enclosing box. As the accuracy of the sketch depends upon the accuracy of the isometric box, the following points should be observed carefully: -

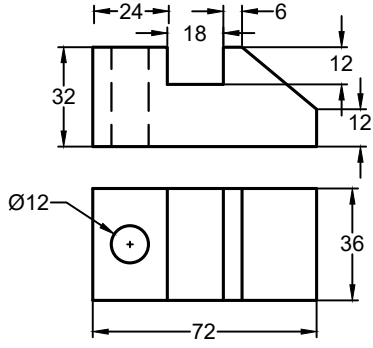
- Vertical edges must be vertical.
- Transverse lines must be parallel, may be little converging but never diverging.
- The angle of the inclines axes should never be more than 30˚.

Step-3. Transfer the details from orthographic views to the isometric box. Take the proportions along or parallel the isometric axes only. For non-isometric lines, first locate the end points to establish the line.

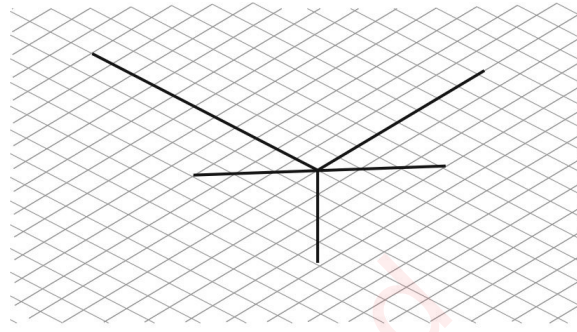
Step-4 Sketch additional required non- isometric lines and circle & arcs and brightening the desired surfaces.

Step 5. Brighten the required lines and complete the view with all the necessary dimensions.

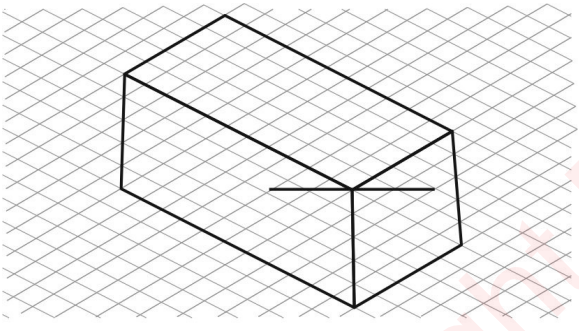
Fig.4.22 illustrates the step-by-step construction of an isometric sketch.



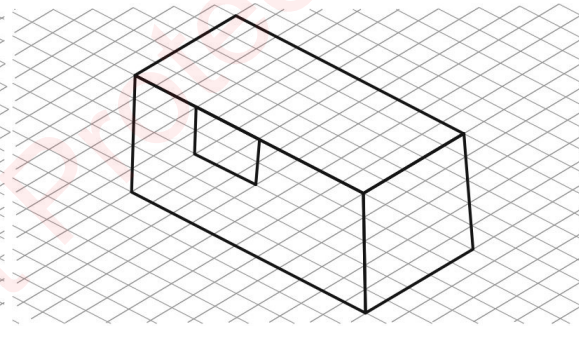
Given (a)



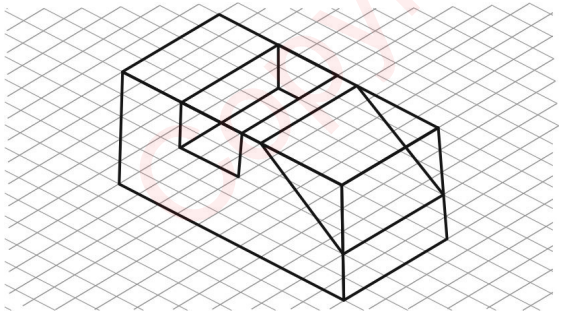
Step-1



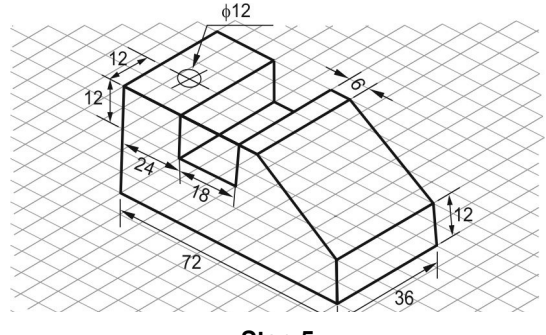
Step-2



Step-3



Step-4



Step-5

Fig.4.22: Steps in sketching of an isometric view

UNIT SUMMARY

1. Freehand sketching is one of the most effective ways to communicate a pictorial or verbal idea to a workman.
2. Three things absolutely essential for sketching work are pencil, eraser and paper.
3. A pencil of the grade HB is preferred for sketching works.
4. The sketches drawn using only pencil and eraser are called 'Free Hand Sketches'.
5. Designers, engineers, artists and teachers use this for immediate graphical communication.
6. In all fields of engineering these are essential to draw the diagrams, prepare the layouts, think of a design etc.
7. To start with grid sheets help to practise well.
8. Vertical lines are to be drawn from top to bottom.
9. Inclined lines are to be drawn from lower left to upper right.
10. Arcs of small radii and circles are conveniently drawn by constructing squares.
11. A sketch is considered to be good when its features are shown in correct proportions.

EXERCISES

A. Objective Questions

A1. Fill in the blanks with appropriate words

4.1 In industry, freehand sketches are employed to present the ideas of the ---------- to others or the management.

4.2 A pencil of the grade ----------is preferred for sketching works.

4.3 Sketching a ----------is also useful in sketching circles.

4.4 Circular feature in isometric projection, appear as ---------- feature.

4.5 Shop drawings are supplied in the form of ---------- views.

4.6 A washer is used to protect ---------- surface and distribute stresses.

4.7 Locking devices are used so that the --------- does not open.

Answers of Fill in the Blanks Questions

4.1 Designer/Engineer; 4.2 HB; 4.3 Square; 4.4 Elliptical; 4.5 Orthographic Views; 4.6 Bottom surface; 4.7. Nut

A2. Multiple Choice Questions

4.1 Flank in a screw member is the
(a) unthreaded portion (b) inclined surface of the thread
(c) extreme surface of the screw member (d) inner most part of the thread

4.2 Thread angle of BSW thread is
(a) 55° (b) 60° (c) 29° (d) 45°

4.3 Buttress threads are used for applications where there is
(a) heavy load (b) high speed
(c) power load is in one direction only (d) high load and speed

4.4 Length of bolt is specified as measured from
(a) top of head to end of bolt (b) bottom of head to end of bolt
(c) where the threads start to end (d) bottom of head to start of threads

4.5 A stud is used where
(a) bolt cannot be used (b) heavy load is on bolts
(c) parts are removed frequently (d) no space is there for bolt head

4.6 A screw has its top head that is generally
(a) circular (b) square (c) hexagonal (d) octagonal

4.7 A lock nut is different from ordinary nut with regard to
(a) height (b) diameter (c) type of threads (d) shape

Answers of Multiple Choice Questions

4.1 (b); 4.2 (a); 4.3 (c); 4.4 (b); 4.5 (c); 4.6 (a); 4.7 (a)

B. Subjective Questions

4.1 List the applications of free hand sketching.

4.2 List the materials for free hand sketching.

4.3 Explain the procedure to draw freehand lines on a technical sketch at an angle of 30° and 45° with the horizontal.

4.4 Write the steps to be followed in freehand sketching of:
(a) square (b) circle and (c) ellipse

4.5 Define the term fastening and its types. Under what circumstances each type is used?

4.6 Give different applications of thread.

4.7 Define at least five parts of a screw thread.

4.8 Explain the conventional method for drawing threads.

4.9 Describe the steps to be followed in sketching orthographic views of an object.

4.10 Write the steps to be followed in sketching isometric projection of an object, from the given orthographic views.

4.11 Make freehand Sketches of the following machine elements:
- (a) Hexagonal headed bolt and nut taking bolt diameter as 40 mm.
- (b) Square headed bolt and nut taking bolt diameter as 24 mm.
- (c) Nut and lock nut.
- (d) Ring nut.

4.12 Draw two views of bolt and nut of diameter 20mm and the length 75mm with square head.

4.13 Draw front, side and top view of a hexagonal bolt of diameter 24mm and length 85mm along with its nut and washer.

4.14 Draw freehand the isometric projection of the following :
- (a) A cube of 50 mm side
- (b) Hexagonal prism with side of 50 mm and 80 mm long

4.15 Make sketches of two simple models of machine parts as shown in Fig.4.23 (a) and (b).

4.16 Two orthographic view of an object are given in Fig.4.24. Add third view in both cases. You may draw isometric views in given isometric block with grid.

4.17 Sketch by freehand the front, top and left side view of each object as given in the Fig.4.25 on squared graph paper.

4.18 Sketch the isometric view of each on isometric grid as shown in Fig.4.25.

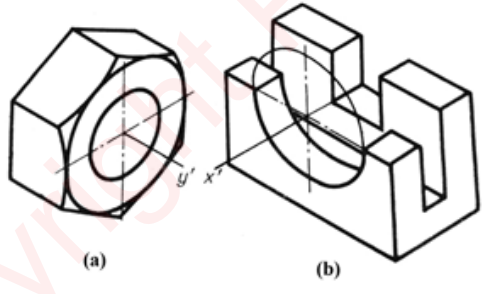


Fig.4.23: Pictorial views for subjective problem 4.15

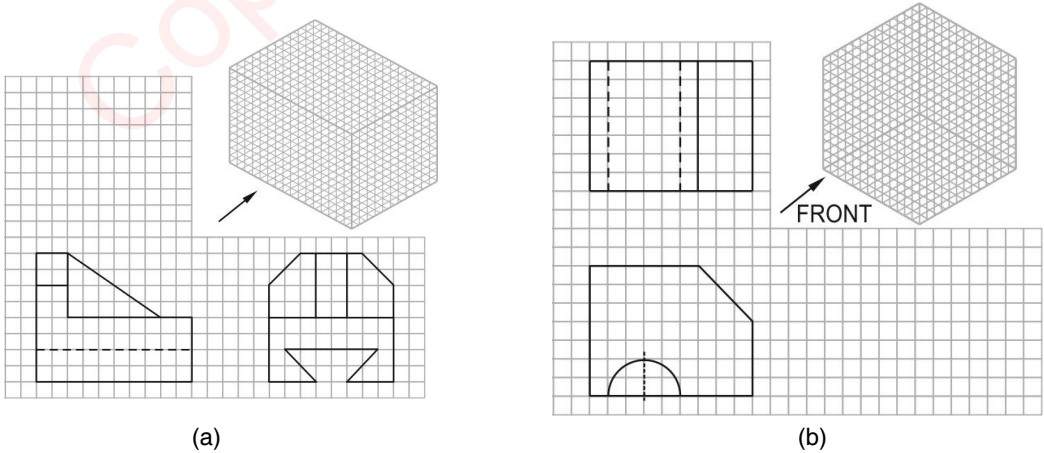


Fig.4.24: Pictorial views for subjective problem 4.16

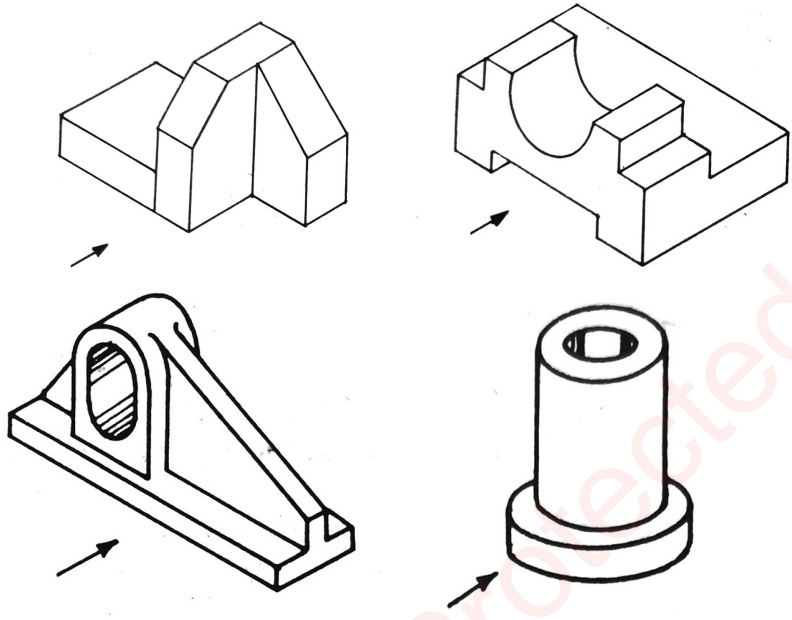


Fig.4.25: Pictorial views for subjective problem 4.17 and 4.18

PRACTICALS

As per curriculum the practicals related to Unit-4 are:

1. Draw free hand sketches/ conventional representation of machine elements in sketch book such as thread profiles, nuts, bolts, studs, set screws, washers, Locking arrangements.
2. Problem based Learning: Given the orthographic views of at least three objects with few missing lines, the student will try to imagine the corresponding objects, complete the views and draw these views in sketch book.

Practical-1 Draw Free Hand Sketches/Conventional Representation of Machine Elements

Practical statement

Draw free hand sketches/ conventional representation of machine elements in sketch book such as thread profiles, nuts, bolts, studs, set screws, washers, Locking arrangements.

Practical significance

Generally in the process of engineering design, the first step is conceptual design, i.e. formulation of idea. This concept has to be transferred on paper, generally by freehand sketching without the aid of drawing instruments because it require several modification based on the discussion and feedback in arriving at a final acceptable solution.

Relevant theory

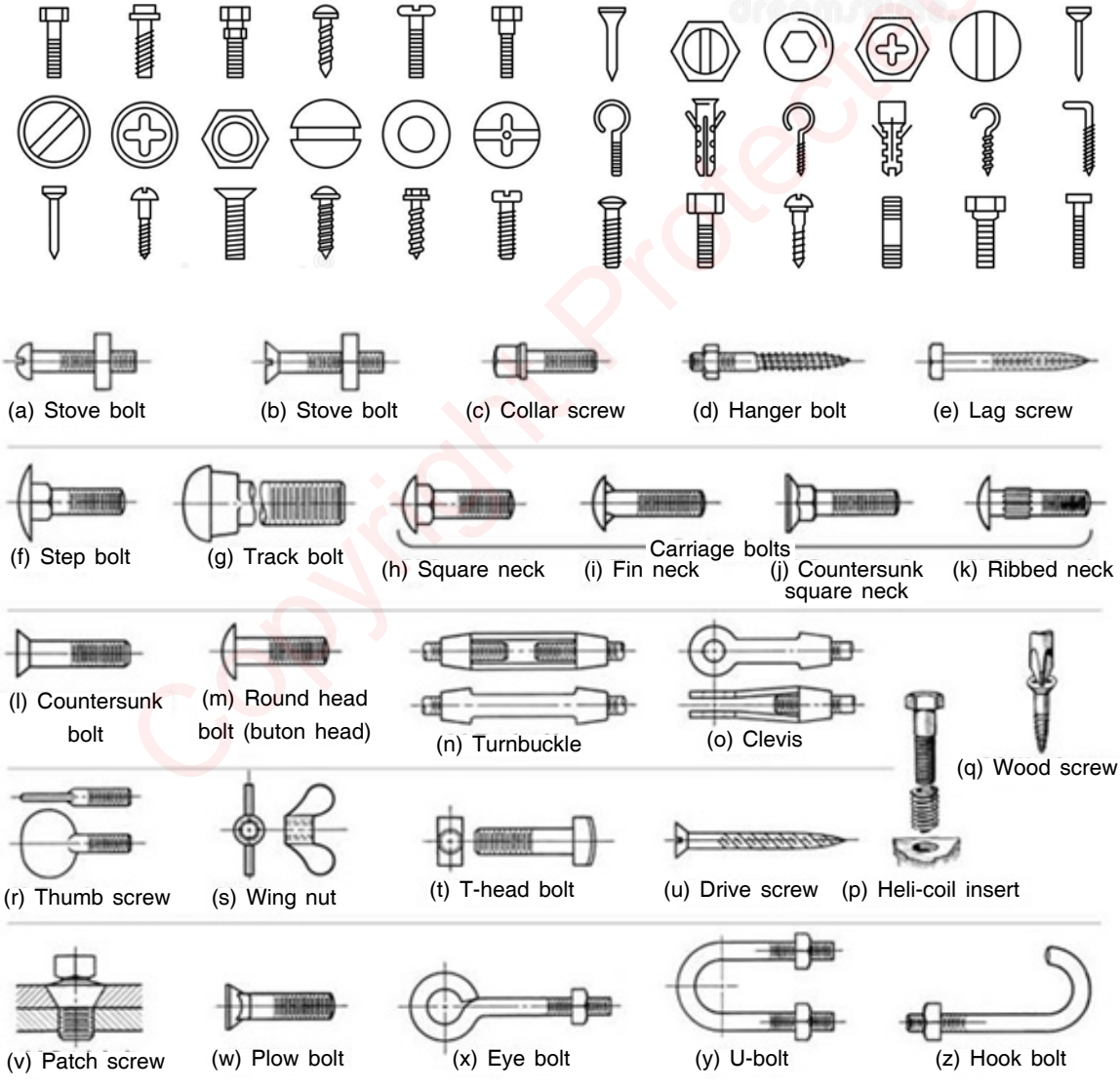
Refer section 4.0 of the book.

Practical outcomes (PrO)

PrO1: Draw free hand sketches/ conventional representation of machine elements..

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

Create 10 free hand sketches of following fasteners:



Resources required

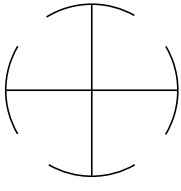
Same as mentioned in Practical-1 of Unit-1 except drawing instruments, T-Square, Set Square, Mini-Drafter. Further, Machine element Conventional Representation BIS codes are also required. Rectangular and Isometric grid or graph papers are also required.

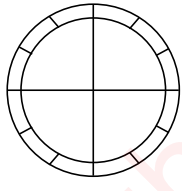
Precautions

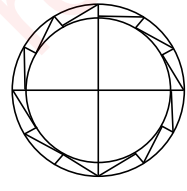
1. Use appropriate type of Pencil based on the type of line use.
2. Don't use blades or paper cutters to sharpen the Pencils.

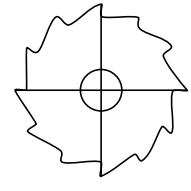
Suggested procedure

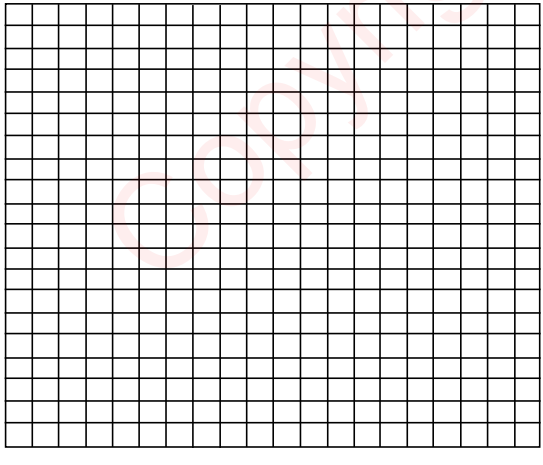
1. To make a good sketch, it is necessary that one should be able to draw straight lines, curves, etc. without any external aid. In the beginning, one should practice to draw horizontal and vertical lines. It may not be exactly to scale, but it should be proportional for better understanding.
2. For example if free hand sketching of a ratchet is required then following suggestive steps can be taken up:

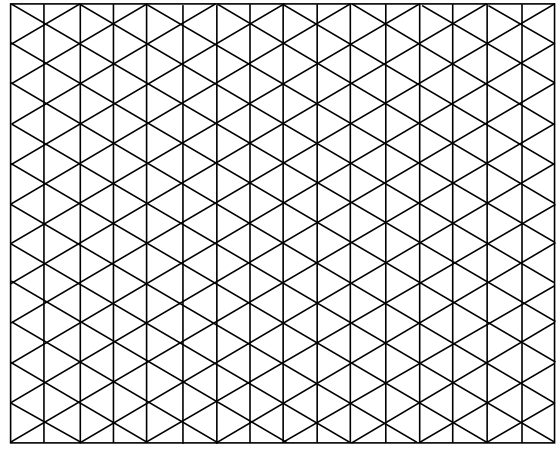

Step 1


Step 2


Step 3

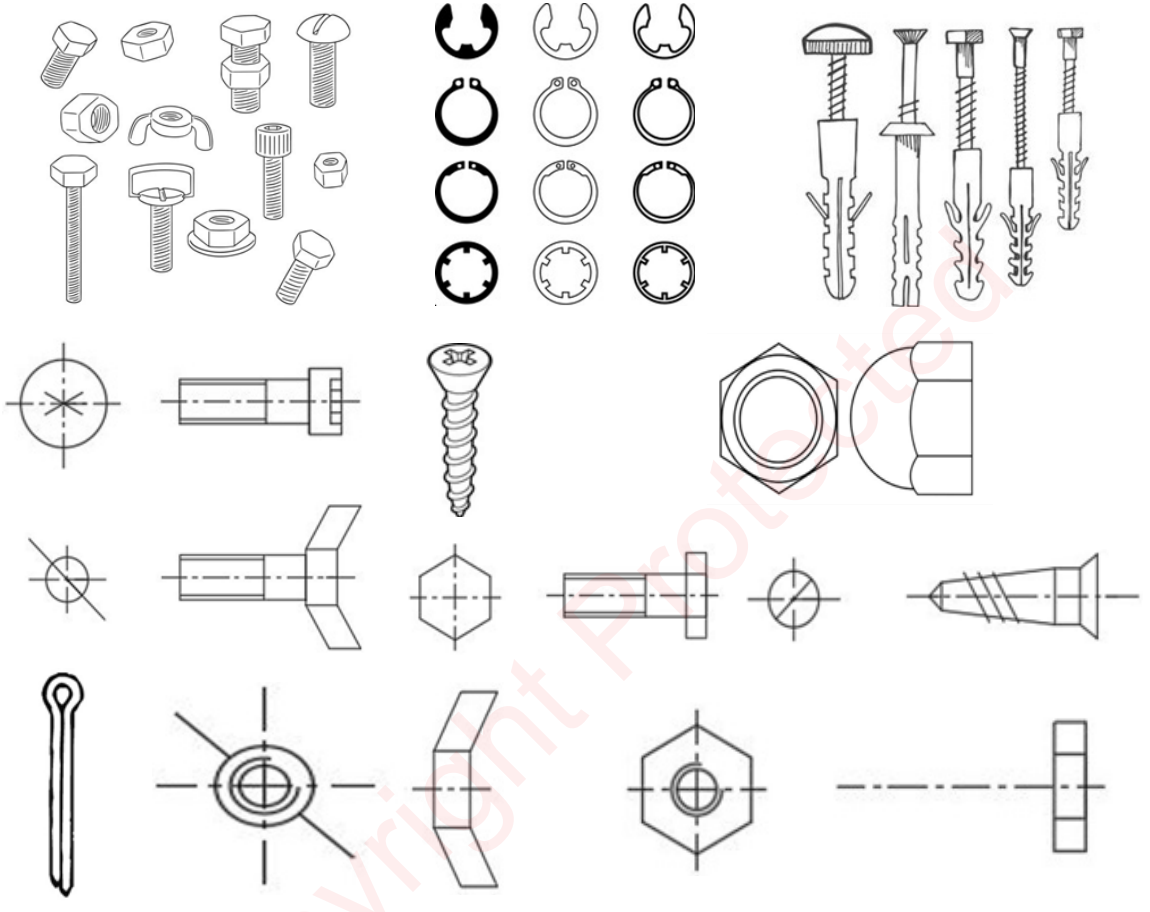

Step 4





3. Initially a Rectangular grid or an Isometric grid can be used to draw the components in orthographic and Isometric views.

Observations



Practical related questions

1. Explain the difference between a rectangular and Isometric grid.
2. Write five applications of each Screw and Nut-Bolt

Disposal of waste

Same as mentioned in Practical-1 of Unit-1

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1 of Unit-1

Suggested learning resources

Refer Unit-4

Suggested assessment scheme

Performance indicators		Weightage/ Marks	Marks Awarded
Process related: 5 Marks* (33%)			
1	Use of appropriate pencil while drawing different entities	3	
2	Not taking support of others to draw given problem(s)	2	
Product related: 10 Marks* (67%)			
3	Neatness, Cleanliness on drawing sheet/sketch book	1	
4	Closeness of free hand sketches to actual counterparts	3	
5	Uniformity in drawing and line work	1	
6	Creating given drawing with all problems solved	4	
7	Submission of drawing in time	1	
	Total	**100 %**	

* Marks and percentage weightages for product and process assessment will be decided by the teacher.

Name of the Student: …….. **Marks Awarded**			**Signature of Teacher with date**
Process Related	Product Related	Total	

Practical-2 Draw Missing Views and Information

Practical statement

Given the orthographic views of at least three objects with few missing lines, the student will try to imagine the corresponding objects, complete the views and draw these views in sketch book (Problem based Learning).

Practical significance

Practice to identify missing views and information is very helpful in further strengthening of engineering drawing concepts, Orthographic views, Isometric views and visualization ability in a student.

Relevant theory

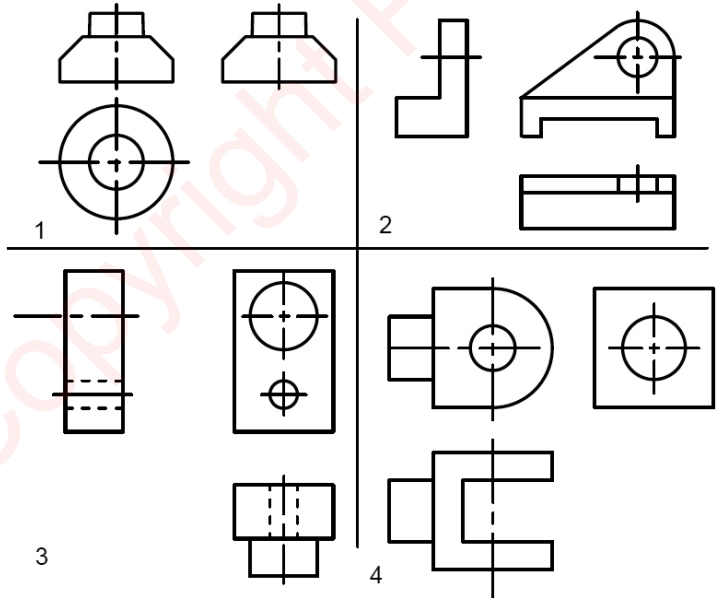
Refer section 4.0 of the book.

Practical outcomes (PrO)

PrO1: Draw missing views and information in the given orthographic views and isometric views.

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

Identify the missing views and lines in the following situations (First Angle Projection).



Resources required

Same as mentioned in Practical-1 of Unit-4.

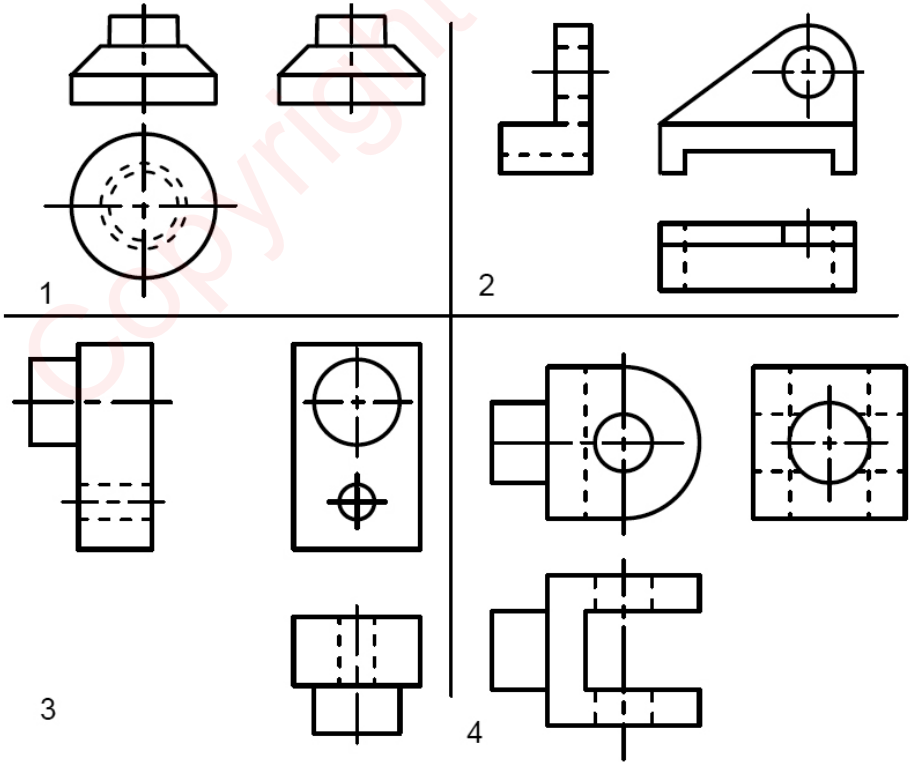
Precautions

Same as mentioned in Practical-1 of Unit-4.

Suggested procedure

1. In situation-1 if you examine critically there is no missing information in front and side views but when we look from the top of the object then three circles should be visible hence one circle is missing in the top view.
2. In situation-2 the front view shows a circular hole while it is hidden in top and side views hence in top and side views it should be represented by a pair of two parallel dotted lines. In top view the dotted lines are present but in side view the dotted lines for circular hole are missing. Similarly in front view there is a rectangular through slot in the base hence its corresponding dotted lines in top and side views should be available. One dotted line in side view and two dotted lines in top view are to be drawn to make them complete. As one of the edges of the hole is in line with one of the edges of slot hence in top view some part of the right dotted lines for hole and slot are overlapping.
3. In situation-3 the projection of bigger circular block is missing in the side view. Hence a rectangle of height equal to diameter of circular block and width equal to thickness of circular block should be drawn in side view to make it complete.
4. In situation-4, the front view has got a circular through hole which is hidden in other two views hence dotted lines corresponding to this circular hole in top and side views should be drawn. Similarly, in left side the inner edge of rectangular slot should be represented by two vertical dotted lines in the given side view.

Observations



Practical related Questions

1. In situation-3 two vertical dotted lines are shown. Explain the presence of these lines and why they are dotted.
2. In situation-4 four vertical dotted lines are shown. Explain the presence of these lines and why they are dotted.

Disposal of waste

Same as mentioned in Practical-1 of Unit-4.

Environment friendly approach: Reuse, Reduce and Recycle

Same as mentioned in Practical-1 of Unit-4.

Suggested learning resources

Refer Unit-4

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-4.

KNOW MORE

- Teacher shall collect few production/construction/electrical/electronic actual industrial component drawings to show them to the students with input.
- During the input sessions the teacher must sketches some objects by freehand on white / black board related to the relevant branch viz. mechanical and allied disciplines/electrical and allied disciplines/electronics etc.

Applications (Real life / Industrial)

Freehand sketching has applications in the following areas :-

- In R&D section, while designing new products or any changes in the existing design the designer makes sketches or schematic diagrams to conveyed his ideas to a draftsman or others.
- In production and maintenance departments, freehand sketches are prepared to give details about the part to be repaired or modification needed in an existing structure of machine etc.
- Freehand sketching helps the teacher to explain concepts in the classroom.

Inquisitiveness and Curiosity

Other than the classroom and Drawing practice sessions, following are the suggested student-related co-curricular activities which can be undertaken to accelerate the attainment of the various outcomes in this unit:

a. Student should maintain a separate A3 size sketch book which will be the part of term work and submit it along with drawing sheets. Following assignment on orthographic views should be drawn with a suitable dimension in the sketch book-

i. Electric motor
ii. Car model
iii. Computer monitor
iv. Electronic circuit board
v. Staircase
vi. Multi views of a house

REFERENCES & SUGGESTED READINGS

1. Engineering Drawing Practices for School and Colleges SP 46:2003, published by Bureau of Indian Standards, Government of India, Third Reprint, October 1998; ISBN:. 81-7061-091-2, Manak Bhavan, 9 Bahadur Shah Zafar Marg , New Delhi
2. Online course material on Free hand sketching, T. Jeyapoovan Thangasamy, Department of Mechanical Engineering, Hindustan Institute of Technology and Science, Chennai, India, https://fdocuments.in/document/lesson-14-freehand-sketching-orthographic-projections-part-i.html
3. Online course material on Free hand sketching, https://deseng.ryerson.ca/dokuwiki/_media/mec222:asc3.pdf
4. Video on Sketching Orthographic Views, http://www.cadmodelinghub.com/, https://www.youtube.com/watch?v=VkMG9Se1AGM
5. Video on Freehand Sketching for Engineers - Video 3 - Orthographic and Circles, Marklin https://www.youtube.com/watch?v=nEaqezJIEMs

5 Computer Aided Drafting Interface

UNIT SPECIFICS

This unit presents information related to following topics:

- Computer Aided Drafting (CADr): Concept and benefits
- Computer hardware and computer aided design and drafting software
- System requirements and understanding the interface for AutoCAD software
- AutoCAD software installation process
- AutoCAD software opening screen and different pallet/tool bars
- Invoking commands in AutoCAD software
- File features: Opening new and existing drawing file, saving the file, creating templates
- Setting up parameters for new drawing like units, limits
- Use of drawing aides like grid, snap, ortho, object snaps
- Function keys, short-cut key characters, undoing and redoing action
- Exiting AutoCAD software

The understanding of these topics will be developed after reading the contents, completing the solved problems, activities, performing exercises on AutoCAD software and viewing the ICT and web resources mentioned in this unit.

At the end of the unit, summary is provided to recapitulate the topics covered and applications are mentioned so that the learner can correlate the presented knowledge with real life and industrial situations. Few activities are mentioned to create inquisitiveness and curiosity in the learner. Subjective and objective questions are provided for the reinforcement of the knowledge and a list of references and suggested readings is also provided for further learning. Video resources along with QR codes are mentioned for getting more information on various topics of interest which can be surfed or scanned through mobile phone for viewing.

RATIONALE

From many years, drawing has been an indispensible part of engineering as it is the link between engineering design and manufacturing activities. Information related to dimensions, tolerances, shape, size and other details for the manufacturing are communicated to the manufacturing personnel in form

of drawings made as per the prescribed drafting standards. The manufacturing personnel and designers can understand and interpret the drawings in the same sense based on these drafting standards. The engineering drawings can be made both as per the conventional methods as seen from Unit 1 to 4 of this book and also using computers (Unit 5 and 6 of this book). The traditional methods uses manual drafting instruments like drafting board, pen, pencil, ink, tracing paper, etc. The drawings made using computers are termed as Computer Aided Drafting, Computer Aided Design & Drafting, Computer Assisted Drafting, Computer Augmented Drafting, Computer Automated Drafting, etc. This Unit outlines the basics of Computer Aided Drafting and interface of Computer Drafting hardware with software. It also describes initial steps to be followed to start any drafting software like AutoCAD, AutoCAD main window, setting up of new drawing and how to invoke commands.

PRE-REQUISITES

Before reading this unit the student is advised to revisit following:

- Engineering Drawing Unit-1 to 2 of this book
- Maths: Coordinate and Plane geometry
- Basic Computer and MS window operations

UNIT OUTCOMES

After reading the contents, completing the solved problems, activities, performing exercises on AutoCAD software and viewing the ICT and web resources mentioned in this unit the students are expected to:

U5-O1: Explain Computer aided drafting (CADr), associated terminology and advantages

U5-O2: Interface computer hardware and software interface for CADr

U5-O3: Perform file related operations in AutoCAD software

U5-O4: Set up new drawing in AutoCAD software

U5-O5: Use status toolbar options for accurate drawing

Unit-5 Outcome	EXPECTED MAPPING WITH COURSE OUTCOMES *(1- Weak Correlation; 2- Medium Correlation; 3- Strong Correlation)*					
	CO-1	CO-2	CO-3	CO-4	CO-5	CO-6
U5-O1	-	-	-	-	-	1
U5-O2	-	-	-	-	-	2
U5-O3	-	-	-	-	-	3
U5-O4	-	-	-	-	-	3
U5-O5	-	-	-	-	-	3

5.1 INTRODUCTION

Any machine part to be produced is primarily designed and then its drawing is made. Based on the drawing the industry will manufacture it. This drawing happens to be the communication between the designers and the shop floor of the manufacturing. Thus, engineering drawing is a vital communication tool for engineers.

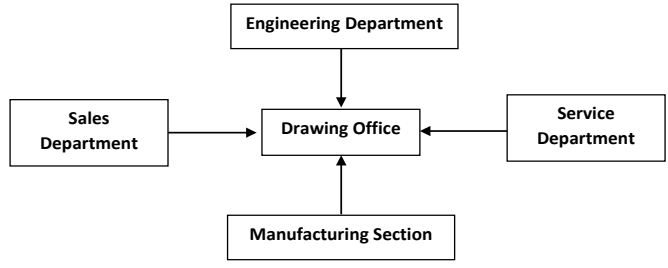


Fig.5.1: Typical industrial drawing office work flow

In the past these drawings were made manually but now computer aided drafting software are used.

Fig.5.1 provides the flow of work in a typical industrial drawing office.

Many software are available for drawing but the most popular is AutoCAD. This book provides information related to AutoCAD 2020.

5.2 COMPUTER AIDED DRAFTING

CADr means Computer Aided Drafting. It is a process of preparing a drawing of an object on computer screen using computer drafting software.

Many types of drawings are there in different fields of engineering and sciences. In the area of aeronautical engineering and mechanical various drawings related to machine components and layouts are prepared. In the area of civil engineering various layouts and plans are prepared. In the field of electrical engineering, the layouts of power distribution system and circuit diagrams are prepared.

Computers have brought drastic changes in many fields including engineering drawing. Not only the time involved in computer aided drawing or drafting is a fraction of that involved in manual drafting but also the revision can be done quickly without disturbing the entire drawing. It is very easy to draw different views of the object using CADr. To use CADr software effectively the user must learn and practice all the commands. This unit and next intend to develop the ability in the learner to draw, modify, edit, manage and print any simple to complex 2D entities using AutoCAD software. The use of CADr approach offers high graphics capabilities to support designer in

- Visualizing and conceptualizing his ideas
- Drawing, modifying, saving and printing very quickly
- Making and using animations
- Performing design calculations
- Applying colors, ergonomics, aesthetics and other formatting features easily

In CADr the drawing is made on the computer screen using computer aided design and drafting software like AutoCAD, Creo, Catia, Unigraphics, Solidworks, etc. Modification and editing is very easy in these digital drawings and once complete the drawing can be saved in the external or internal memory like computer Hard Disk/CD/Floppy/External HD/Pen Drive. Once the drawing is complete then it can be printed using Printers or Plotters. Upto A3 size drawings printers can be used while for large and colored drawings plotters are used. The printed drawing quality is independent of skills of the person contrary to manual engineering drawing.

5.3 BENEFITS OF CADR

The implementation of the CAD system provides variety of benefits to the industries in design and production as given below:

1. Enhanced productivity in drafting
2. Less time for preparation of drawing
3. Less man power requirement
4. Easy alterations as per clients requirements
5. Reduced transcription errors in drawing
6. Enhanced accuracy of drawing
7. Assistance in document preparation
8. Better evolved designs
9. Improvement in aesthetics and interpretation of drawing by using colours
10. Development of orthographic views and projections with geometric and dimensioning tolerances
11. Sectioned views with different patterns of hatching
12. Usage of blocks to facilitate development of assembly and sub assembly drawings
13. Part list and bill of material preparation
14. Machining, tolerance and surface finish symbols at the desired surfaces
15. Hydraulic and pneumatic circuit diagrams with symbols
16. Ease in printing/plotting to any scale

5.4 HARDWARE AND VARIOUS CAD SOFTWARE AVAILABLE

The CAD system comprises of computer hardware system and software.

5.4.1 CAD Hardware

The CAD hardware is the computer system and it may be

1. a Workstation computer or
2. a Personal computer (PC) or
3. a Laptop computer

The workstation consists of a high resolution 19/24 inches monitor and graphics accelerator card for quick display, facility to work with multiple windows, higher Random Access Memory (RAM) and higher Hard Disk storage. The workstation looks similar to a personal computer but has a higher performance and is almost five times more expensive. It has high speed and better memory and display capabilities.

Laptop computers are portable computers that can be taken in a bag to any place. A typical Laptop computer includes motherboard with processor, sound cards, graphics card, network card, internal hard disk, different ports to connect external devices, screen, keyboard, speakers, camera and mouse pad. They come along with a battery to provide power to the system and the battery can be charged using a power adapter.

A Personal Computer/Workstation (CAD Hardware)(Fig.5.2) typically consists of

1. High-resolution Monitor (21/19/14 ")
2. Central Processing Unit with numeric co-processor, 8 to 32 MB RAM, a hard disk drive, a cache memory, a colour graphics card.
3. Speakers
4. Printer/Plotter
5. Keyboard
6. Three-button scroll Mouse/Digitiser

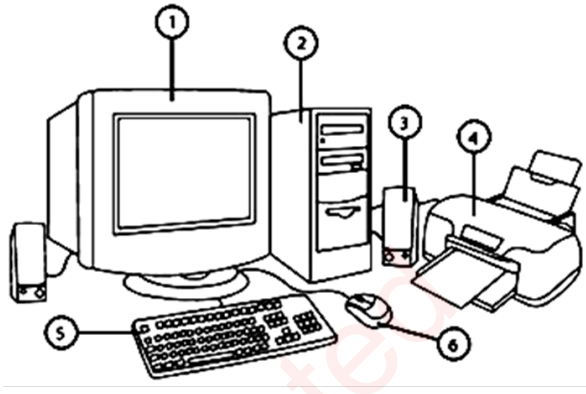


Fig.5.2: Typical computer system

Central Processing Unit (CPU)

A central processing unit (CPU), also called as central processor, main processor or just processor is the brain of a computer, containing all the circuitry needed to process input, store data, and output results. The CPU performs basic arithmetic, logic, controlling, and input/output (I/O) operations specified by the instructions in the program. The internal components in a typical CPU are:

1. Motherboard
2. Networking cards
3. Graphics card
4. Processor
5. Hard drive
6. USB ports
7. Monitor ports
8. Power supply

Input devices

The input devices are used for making selections from a menu, typing information/commands, placing cursor at appropriate position and selecting entities from the screen etc. Sending the commands/ information into the computer produces the digital drawing on the computer screen. Following devices (Fig.5.3) are used as input devices but Keyboard and Mouse are most commonly used input devices.

a. a keyboard, for giving inputs through alphabet, number and arrow keys;
b. a mouse is a small box type device that can control movements in the same direction when pushed across a surface;
c. thumb wheels, horizontal movement is controlled by one wheel and vertical movement regulated by second wheel;
d. a light pen, for pointing the pen to directly select the position on screen;
e. a joystick, being a vertical lever mounted in a box controls movements in all directions;
f. a tracker ball, the rolling of the ball controls movements in all directions;
g. a puck or (h) stylus in conjunction with the digitising tablet, can enter the complete

drawing from a sketch or half-completed drawing by attaching it to the digitiser and used to indicate the important points on lines, curves, etc., and entering into the relevant commands.

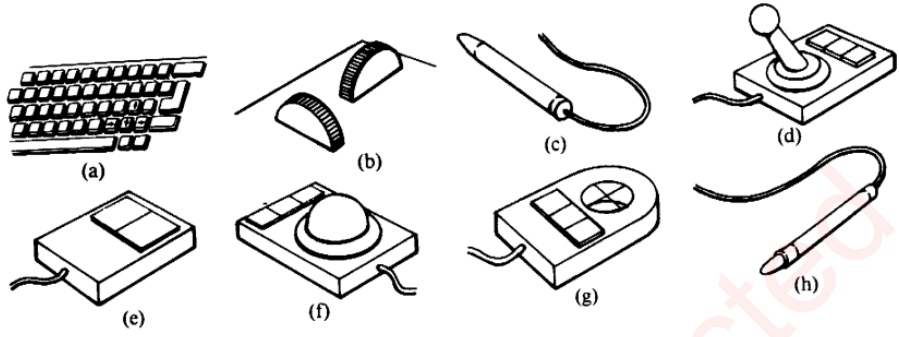


Fig.5.3: Input devices

Output devices

The output devices receive data from the computer processor and give a digital output on the screen or a hard copy through a printer. The main output devices are computer monitors and printers/ plotters. The printers are of two types being impact type or non-impact type.

1. impact printers. These are now less popular. They are dot matrix type and form shapes by selecting relevant small dots from print heads.
2. non-impact printers. These include electrostatic, ink-jet and laser printers, viz.:
   a. electrostatic printers. These create shapes by burning away a thin metallic coating on the special printing paper;
   b. ink-jet printers. These use a print head that directs a jet of ink at the paper to create the required shapes;
   c. the laser printers use a fine beam of laser light to create the required shapes.

The two basic types of plotter are flatbed and drum type:

a. flatbed plotters. These have a flat area to place the paper of any type and thickness and the pens of different thickness and different colours of ink can be used in any direction with plot head.;
b. drum plotters. They have a rotating drum that can move paper in two directions. The pens have a limitation being movement only across the drum. The drawings are created by the combination of pen movement and drum rotation. In comparison to flatbed plotters the drawings are of lower precision and longer length.

5.4.2 CAD Software

The CAD software is an interpreter or translator which allows the user to perform specific type of application or job related to CAD. The following software are available for drafting:

1. AutoCAD
2. Solid Works
3. CATIA

4. NX Unigraphics
5. Creo
6. HYPERMESH
7. ANSYS
8. MSc. NASTRAN
9. FLUENT – GAMBIT

The above software have their own graphic user interface and areas of applications.

AutoCAD software (student version)

AutoCAD package is best suited for accurate and perfect drawings of engineering designs. The drawing of machine parts, isometric views and assembly drawings are possible in AutoCAD. It is quite exhaustive and is internationally known for drafting purposes. It is developed and marketed by Autodesk, Inc. AutoCAD was first released in December 1982. AutoCAD is a software application for 2D and 3D computer-aided drafting as a desktop application and since 2010 as a mobile-web and cloud-based app, currently marketed as AutoCAD 360. The version AutoCAD 2020 is being described in this book.

Autodesk Inc. provides licensed education and commercial versions of AutoCAD, discounted version for education purpose, one and three year education versions for students of AICTE approved technical institutions and one month free trial version for any user. All educational versions are same except the DWG files created or modified using student version have an internal bit-flag set (the "educational flag") and if such files are printed using any version of AutoCAD the printout has 'education version' written on all four sides of the printed drawing.. The Autodesk Education community provides registered students and faculty with free access to different Autodesk applications.

5.5 SYSTEM REQUIREMENTS AND UNDERSTANDING THE INTERFACE

The minimum computer hardware requirement for installing AutoCAD is

- Operating System: Microsoft Windows 10 (64-bit only), 8.1 (64-bit only), or 7 SP1 (64-bit only)
- Processor: 2.5 GHz (3+ GHz recommended)
- Memory: 8 GB (16GB recommended)
- Disk space: 6.0 GB
- Display: 1920 x 1080 resolution with true color (high resolution & 4K displays: resolutions up to 3840 x 2160 supported on Windows 10, 64-bit systems (with capable display card)
- Display Card Basic: 1 GB GPU with 29 GB/s Bandwidth and DirectX 11 compliant (Recommended: 4 GB GPU with 106 GB/s Bandwidth and DirectX 11 compliant)
- Keyboard
- Three button mouse
- Printer or a Plotter

5.6 AUTOCAD INSTALLATION PROCESS

Refer website: https://www.autodesk.com/education/free-software/all

1. Click AutoCAD
2. Create your login account using the MRCET mail id. xyz@mrcet.ac.in (you can access the software for 3 years)
3. After you create your account, sign in and choose
   a. Version: AutoCAD 2020
   b. Operating System: 32 or 64 bit (To find the information, Right click on My Computer or My PC and select properties.)
   c. Language: English (so you can have more effective technical support)
4. Serial number and Product key will be displayed. This information is required at the time of activation after installing the software
5. Download can be carried in two ways:
   a. Download Now (Recommended)
   b. Browser Download
6. After downloading the file, double click on the installation file and then click Yes to complete the installation
7. Now click on Install
8. Check the box I accept the click next
9. For the standalone License type default option, enter the serial key & product key details found on the software database for this software version
10. Click Install and Click Finish to complete the installation

5.7 STARTING THE AUTOCAD PROGRAM

Presumption: The authors presume in the book that AutoCAD software is installed on the computer system and the user has the basic knowledge of computers. To start, click the AutoCAD icon on the desktop or click the start button at the left corner of the screen. The sequence of selections is provided below; please follow it to launch the software.


Launching AutoCAD and Opening drawing


AutoCAD Main Screen

5.8 AUTOCAD OPENING SCREEN (THE USER INTERFACE)

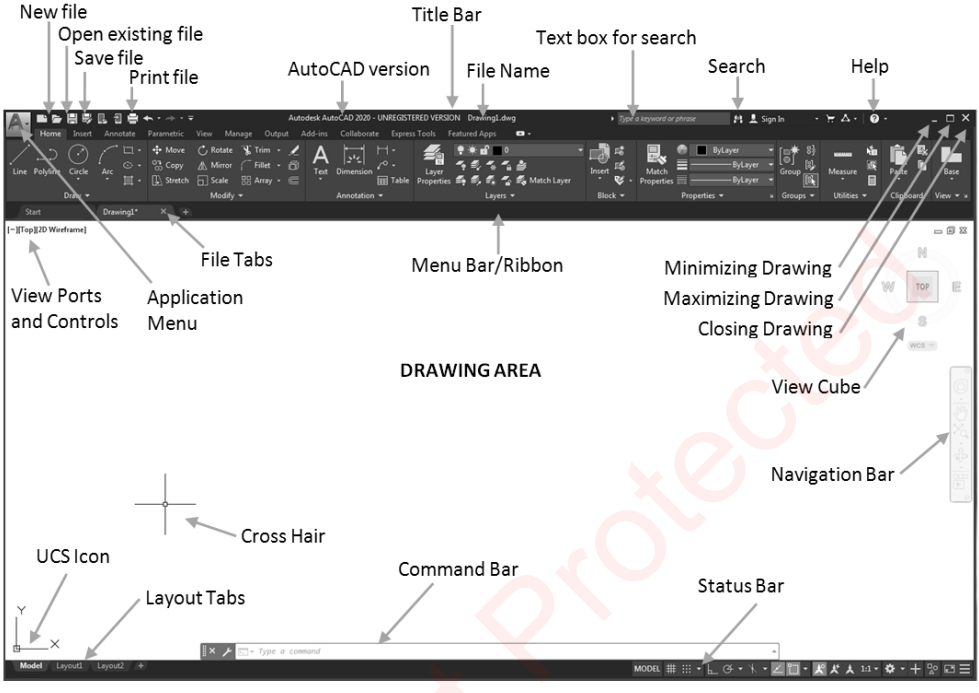
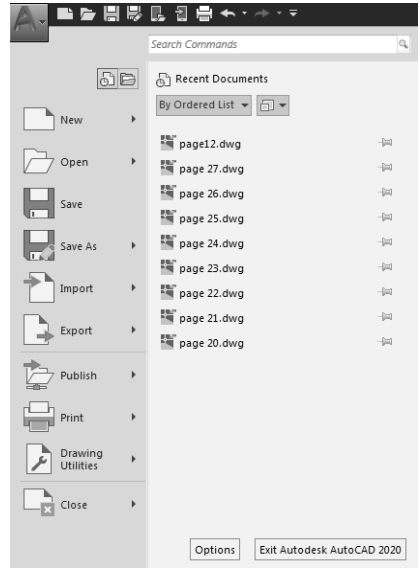


Fig.5.4: AutoCAD 2020 opening screen

5.8.1 Application Menu

Menus are available through the application button in the upper left corner of the drawing window. This menu contains the commands used to create, save, print and manage your drawing.



5.8.2 Command Prompt

The rectangular horizontal window at lower side of the screen is called the command area. The instructions given to the computer through keyboard is shown in this area. It is important to read the command prompt when working with an unfamiliar command. To enter a command using the keyboard, type the command name on the command line and press ENTER or the Spacebar.

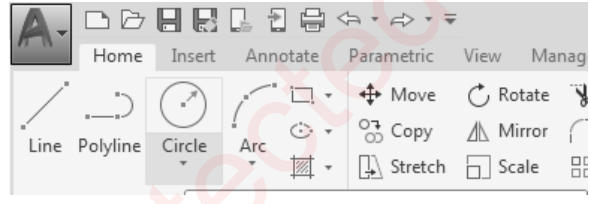
5.8.3 Navigation Bar

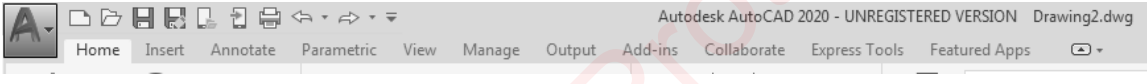
The navigation bar is a user interface element where you can access both unified and product-specific navigation tools. Unified navigation tools are those that can be found across many Autodesk products.

5.8.4 Quick Access Toolbar

The Quick Access toolbar, displayed in the Drafting & Annotation workspace is located at the very top of the drawing window next to the Application button. The Quick Access toolbar may be customized by adding or removing commands. This is done by right clicking on the toolbar and selecting Customize Quick Access toolbar or selecting the arrow at the end of the toolbar.

Icons to open a new file, open an existing file, save a file, undo, redo and plot followed by a file name are present at the top. A default File name [Drawing1.dwg] in this sequence is also present as shown below. The details related to these actions are provided in sections 5.10 to 5.14.





5.8.5 Drawing Area & Cross Hair

The drawing area happens to be the major area in the centre of the screen. The drawing has to be created in this area. A cross hair is displayed here that can be moved using a mouse. It identifies pick and drawing points within the drawing area. Use the crosshairs, which are controlled by your pointing device, to locate points and to select-draw-modify objects. The cross hair position is indicated by coordinate values shown at the left end of the status bar.

5.8.6 Status Bar

The status bar (Fig.5.5) displays the cursor location, drawing tools and the tools that affect the drawing environment. It also provides quick access to some of the most commonly used drawing tools, coordinates of the cross hair (Cursor) and buttons for Grid, Snap, Object snap, Object snap tracking, Polar tracking, Ortho and Scale etc.

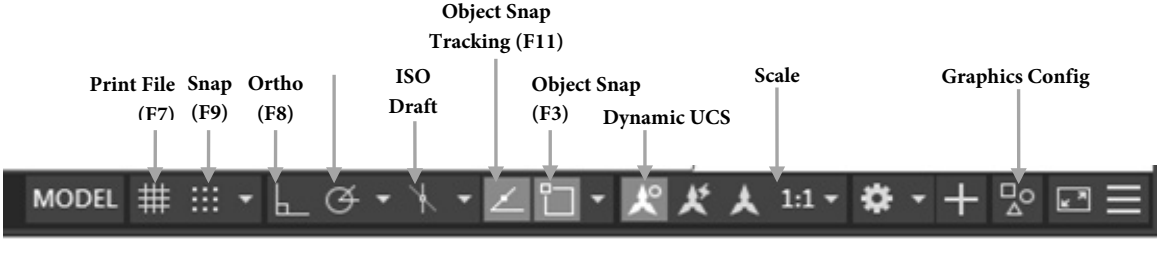
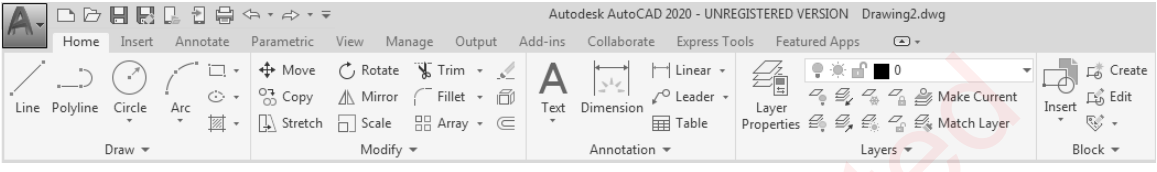


Fig.5.5: AutoCAD 2020 status bar

5.8.7 Menu Bar

The Menu bar displays options like Home, Insert Annotate etc., as shown below. If this is not visible, you can type the system variable Menubar at the command line and set its value as 1 to display. A zero value hides this bar. Ribbon related to each tab/option can be activated by clicking it. For example Home tab has toolbars like Draw, Modify, Annotation, Layers, Block, and Properties etc. Any toolbar can be dragged and placed at any convenient place on the screen using mouse.



It is mandatory to remember the shape of each icon. In case you cannot remember then if the mouse is brought over the icon for a while then a tool tip display shows the job assigned to the icons.

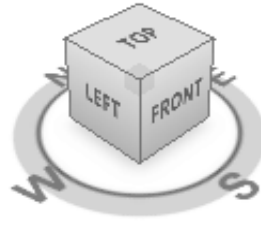
5.8.8 ViewCube

The ViewCube is a navigation tool that allows you to switch between viewing directions. This is very useful in 3D space. It is located in the upper right corner of the drawing area.

5.8.9 UCS and WCS

The UCS icon by default is located at the lower left corner of the drawing area coinciding with WCS. AutoCAD uses the 3-dimensional coordinate system. This coordinate system is termed as the World Coordinate System (WCS). In the drawing area its icon is shown at the lower left corner, using arrows to show the X and Y axis and a square is shown at the intersection (Fig. 5.1). The Z direction is taken at right angles to the screen and outwards.

There is another movable user defined coordinate system called the User Coordinate System (UCS). Its origin can be set anywhere in the WCS or coincide with the WCS (in the Fig.5.1 it coincides with the WCS). To differentiate, when working with UCS the square at the center of the icon is not displayed. Its display can be put On or Off using UCSICON command. Its position and orientation can be changed using UCS command.

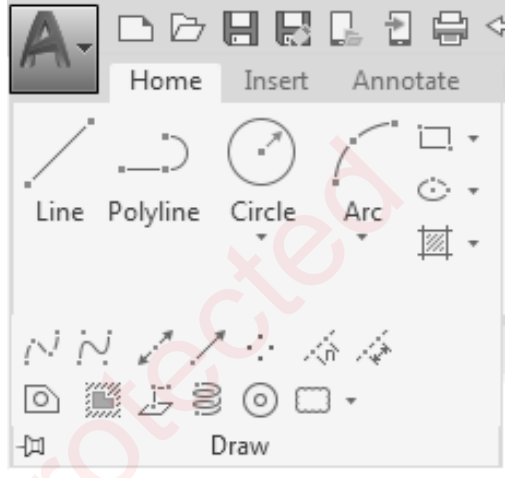


The current UCS model name is displayed in the UCS menu located below the view cube. The menu helps in the following manner, restoration of named UCS's saved with the model, switch to the WCS, or define a new UCS. The WCS item on the menu is used to switch the coordinate from UCS to WCS. In the new UCS, the current UCS can be rotated based on 1, 2 or 3 points to define a new UCS. If the new UCS is clicked a new UCS appears with a default name or unnamed. The name option can be used to save the new defined UCS. The view cube can orient the new UCS or WCS providing direction of modeling. Orientation of view cube with WCS helps navigate the model in the north and up directions. The setting for controlling are called as ViewCube settings.

5.8.10 Draw Tool Bar

The Draw tool bar or pallet is a user interface element which contains following commands which are used to draw any 2D and 3D entities. Detail treatment of this tool bar is given in next chapter.

- LINE, XLINE, MLINE, PLINE
- POLYGON
- RECTANG
- ARC
- CIRCLE
- DONUT
- SPLINE
- ELLIPSE
- BLOCK
- POINT
- HATCH
- BHATCH
- REGION

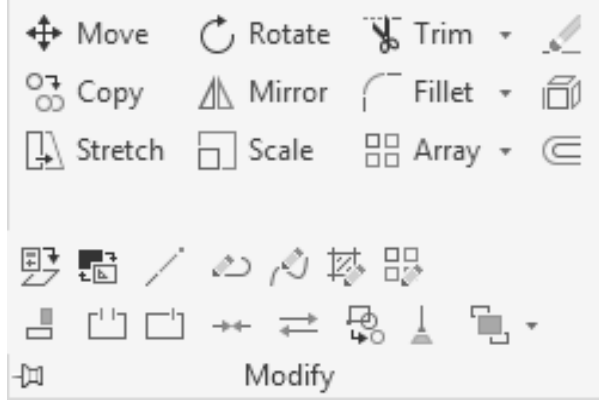


5.8.11 Modify Tool Bar

The Modify tool bar or pallet is a user interface element which contains following commands which are used to modify/edit already drawn 2D and 3D entities. Detail treatment of this tool bar is given in next chapter.

- ERASE
- COPY
- MIRROR
- OFFSET
- ARRAY
- MOVE
- ROTATE
- SCALE
- TRIM
- EXTEND
- CHAMFER
- FILLET
- BREAK
- EXPLODE

Draw and Modify Toolbars



5.8.12 Layout Line

The layout line on the screen is at the bottom. This helps to select the layout. The default is Model Layout. Switch your drawing between model (drawing) space and paper (layout) space. You generally

create your designs in model space and then create layouts to plot and print your drawing in paper space.

5.8.13 Command Line

The Command line is just above the Layout line in the middle of the screen. See Fig. 5.4 where "Type a Command" is written. The area is used to type commands. The prompts for data (↵).

5.9 INVOKING COMMANDS IN AUTOCAD

You can use any of the following methods to start commands.

1. Menu bar: Click an option on Menu bar. From the popup menu, select a choice by clicking the mouse on it.
2. Toolbars/Ribbon: Click on an icon of the ribbon with the left mouse button.
3. Command line: At the Command line type a command using the keyboard and then press ENTER key (↵).
   a. To repeat the last command : press ENTER or SPACEBAR or right click in the drawing area and choose repeat.
   b. To repeat one of the last six commands: right click in the command window or text window and from the shortcut menu choose recent commands.
   c. To repeat the same command multiple times: at the command prompt enter multiple and at the next prompt enter the command you want to repeat.
   d. To cancel a command: press ESC.

5.10 OPENING A DRAWING

5.10.1 Open an Existing Drawing

Click on the red letter A on the top left of your screen or press second icon on the quick access bar. This opens options like Drawing, Drawing from AutoCAD Web, Sheet Set, DGN, Sample files as shown in Fig. 5.6. Click the first option- Drawing which displays the browser from where you can 'Look in' and choose any existing file from the list. As soon as the desired file is selected the software opens it on the screen.

5.10.2 Open a New Drawing

Opens a new drawing based on the measurement system you choose- English (inches) or metric(mm). Click on the red letter A on the top left of your screen or press first icon on the quick access bar. Click on the 'New' button and your screen should then look like Fig.5.7. Select Drawing option and the software opens another form which contains default AutoCAD templates. If you want to use any readymade template then choose any one from the list, otherwise if you want to open new drawing without template then press a small down arrow button adjacent to Open and select 'open with no template' Imperial for units in inches and Metric for units in mm as depicted in Fig.5.7.

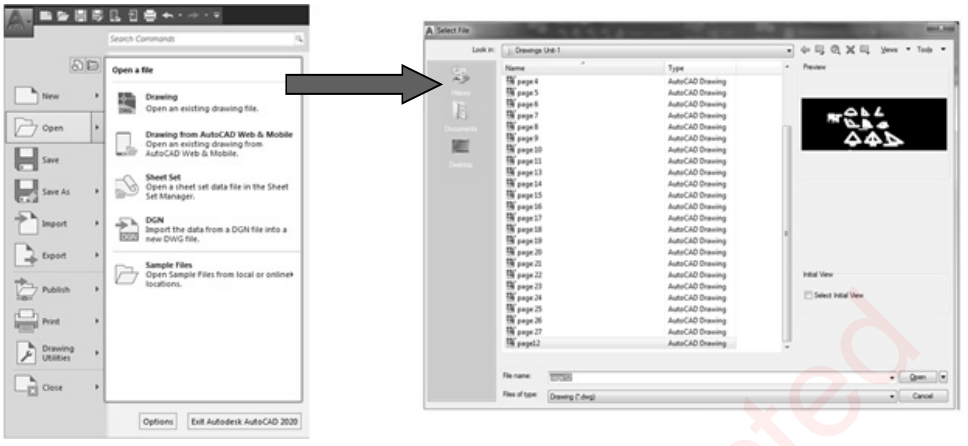


Fig.5.6: Opening an existing file

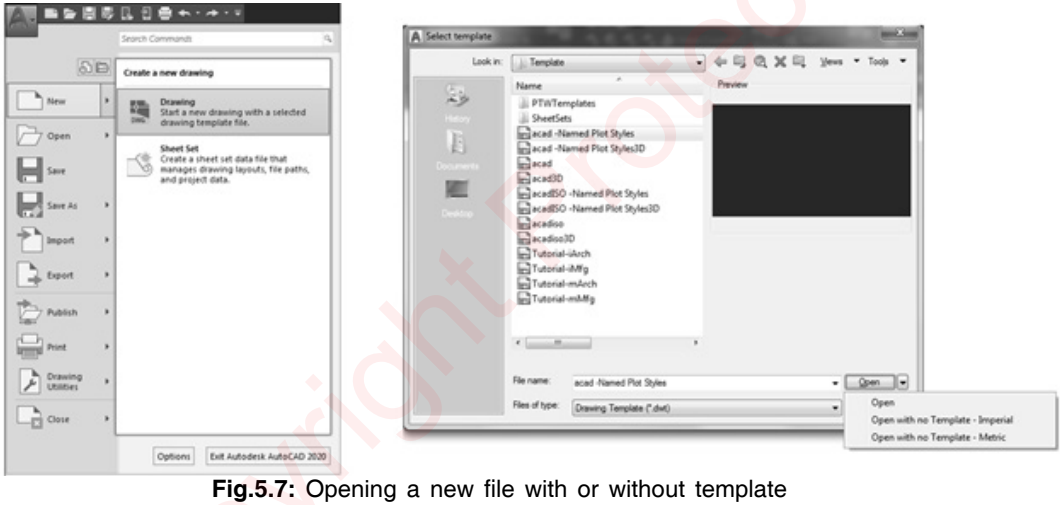


Fig.5.7: Opening a new file with or without template

5.10.3 Use a Template

If in above case you choose option 'Opens' then a new drawing will be opened based on a readymade template you select from the list. The list displays template files (.dwt extension) that exist in the drawing template file location as specified in the Options dialog box. Template file store all the settings for a drawing and can also include predefined layers, dimension styles and views.

5.11 SAVING DRAWING FILES

All the three commands SAVE, SAVE AS and QSAVE allow you to save your drawing by writing it to a permanent storage device such as Hard Drive/Pendrive/External Hard Disc Drive. SAVE command saves the current named drawing. The SAVE AS command sets the name of the current drawing to the new file name you specify. QSAVE command saves the current named drawing without asking you to enter a filename, thus allowing you to do a quick save. You can invoke save command:

- By selecting it from Application icon list.
- From the Quick Access toolbar and
- By typing in Command line : SAVE, SAVE AS , QSAVE and press Enter key.

5.12 CREATING USER DEFINED TEMPLATES

A drawing template file is a drawing file that has been saved with a .dwt file extension and it specifies the styles, settings and layouts in a drawing, including title blocks. You can specify a different template file from the templates' list to start a new drawing as per your requirement.

To create a custom (user defined) template:

- Click New Drawing.
- In the Select template dialog box, select acad.dwt from the list of templates and then modify the template.
- You can change preferences or controls in the drawing, add or replace the border or title block, import or configure standards and erase or add drawing elements.
- Click 'Save As' AutoCAD Drawing Template.
- By default, the software saves the new template in the folder ProgramData\Autodesk\.....\ Template.
- For File name, enter a name for the new template and click Save.
- In the Template Options dialog box, enter a brief description of the template and specify the measurements.

5.13 DRAWING AIDS

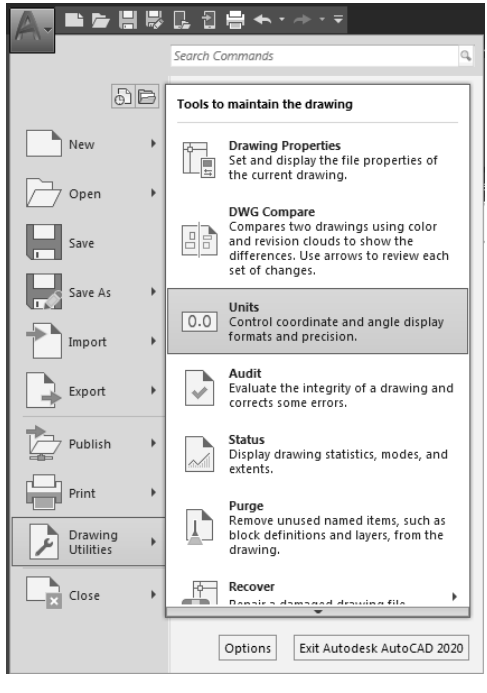
5.13.1 Setting Units

The type of units can be set for the drawing using UNIT command.

- Typing in Command line : UN
- By clicking application button and choosing Click Units under Drawing utilities.

In both the cases the Drawing Units dialog box is displayed.

- Length tile is displayed in the upper left corner. In this tile, click the arrow in the combo box and select a Type and Fractions (Fig.5.8).
- For Length, Decimal is most common and for Angle, Decimal Degree is most common.
- Precision is the number of digits after decimal point to appear with the dimension, e.g., 12.1 or 12.67.
- Select appropriate units in combo box. The effect of this setting is displayed in the Sample Output tile at the bottom of the dialog box.



- By clicking the Direction button at the bottom of the dialogue box the direction to measure angles is set. The Direction control dialog box is displayed. East direction is taken as horizontal direction. Click on any one of the radio buttons as desired and then click the OK.
- Click the OK button to save and exit.

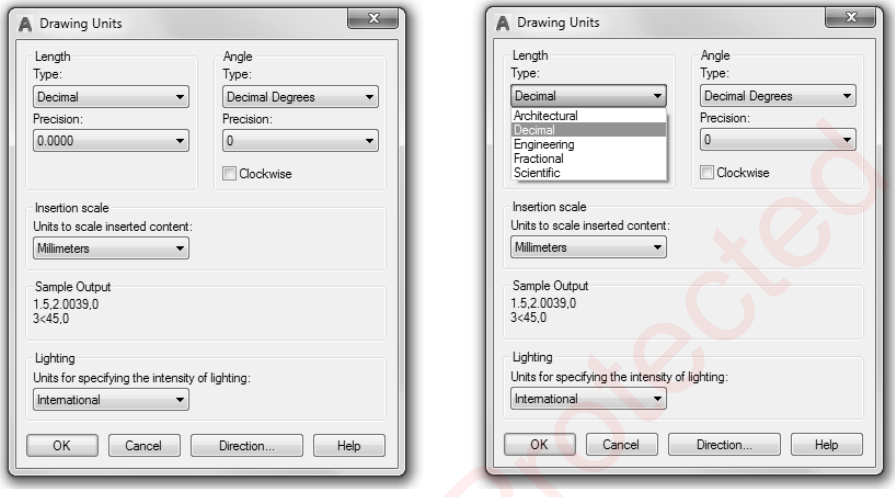


Fig.5.8: Setting drawing units

5.13.2 Drawing Limits

The Limits command in AutoCAD is used to set an invisible rectangular boundary in the drawing area or viewport. You can set the limits of the drawing area in Metric or English before you start a drawing. Setting the limits serves as a reference tool that marks the area in your drawing in which you are currently working. This is the area which will be displayed when you 'Zoom' 'All'(Z ↵ A ↵). It is also the area in which the visible grid and snap grid are extended to. If limits are on, you cannot draw any entity outside the limits. You can see the drawing area by making the Grid on. The drawing boundaries are usually set to match the size of a sheet of drawing paper. This means that when the drawing is plotted and a hard copy is made, it will fit on the drawing paper. To set limits:

- Click Format menu Drawing Limits.
- Enter the coordinates for a point at the lower-left corner of the grid limits.
- Enter the coordinates for a point at the upper-right corner of the grid limits and press ENTER.

5.13.3 Snap, Grid and Ortho

Grid is a display of small dots on the screen at equally specified distances in the horizontal and vertical directions. This grid can also be rotated. While plotting the drawing the grid points do not appear. For activation of GRID, the grid button on the status bar can be clicked. The status bar is at the bottom of the screen (Fig.5.5) or by pressing Function key F7 or by typing the command as mentioned:

Command: GRID

Specify grid spacing (X) or [ON/OFF/Snap/Aspect] <10.0000>:

Specify a value of grid spacing

Snap is the smallest invisible distance of increment that can be set for the mouse. If Snap is set on, the mouse moves in steps of the set increment. To select any intermediate point between the dots of the grid, put Snap off by pressing F9 or on Status bar (Fig. 5.5) click the Snap button or type the command as under:

Command: SNAP

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <10.00000>:

Specify snap increment or put it off

Ortho, Orthogonal can be written as Ortho in short. It means 90° to each other. On setting the mode ON the lines are drawn along the X and Y direction. The Ortho is set as on either by clicking on ORTHO button on the status bar (Fig.5.5) or by pressing F8 Function Key or by typing the command at command line as mentioned below:

- Command: ORTHO
- Enter mode [ON/OFF] <ON>: Type an option: On or Off as desired.
- On the Status Bar, right-click on snap mode, find and select Snap Settings.
- In the Drafting Settings dialog box, Snap and Grid tab, click Snap on.
- Under Snap Type, make sure Grid Snap and Rectangular Snap are selected.
- In the Snap X Spacing box, enter the horizontal snap spacing value in units.
- The check box, Equal X and Y Spacing, applies to both snap and grid spacing.
- Modify other settings as needed.
- Snap mode can be turned on and off with the F9 key. You can suppress snap temporarily by holding down F9 while you create or modify an object.

5.13.4 Object Snaps 

To make drawings quickly and accurately AutoCAD provides tracking and Object snap tools. The Object snap features select locations accurately based on existing objects in your drawings. Without object snaps the resolution of your screen makes it impossible to select points with the accuracy.

For making accurate selection the object snap is used like the objects center, endpoint, midpoint and so on.

You can select this command

- From the status toolbar: clicking the down arrow adjacent to Object Snap button on the Status bar or Right-click it (Fig.5.9).
- Typing in Command line: DSETTINGS,
- OSNAP

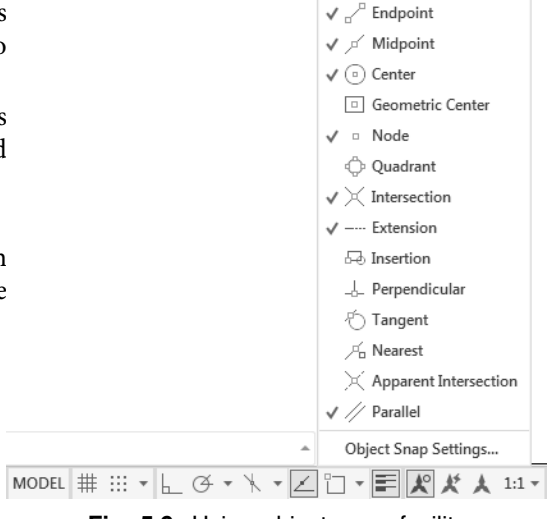


Fig. 5.9: Using object snap facility

5.14 FUNCTION KEY ASSIGNMENTS

To change the status of coordinate display, ortho etc. you can use Function and Control keys. The following is the list of function and control keys.

F1 Displays Help
F2 Toggles between the text and graphics screen
F3 Running Object Snap on/off
F4 Tablet (an input device) on/off
F5 Switches between the top, front and side views for isometric drawings
F6 Switches dynamic UCS to on or off
F7 Grid on/off
F8 Ortho on/off; if on, lines are either horizontal or vertical
F9 Snap mode on/off
F10 Polar option on/off
F11 Object Snap Tracking on/off

5.15 SHORT-CUT KEY CHARACTERS

Short-cut characters are given in table 5.1

Table 5.1 Short-cut key command name function

A	ARC	Draws an arc
C	CIRCLE	Draws a circle
E	ERASE	Deletes the selected object(s)
F	FILLET	Creates a radius at the intersecting lines
G	HATCH	Displays Boundary Hatch and Fill dialog box
L	LINE	Draws a line
M	MOVE	Moves a drawing from one place to another but drawing coordinates change
O	OFFSET	Draw a parallel line or arc at specified distance
P	PAN	Moves drawing on screen without changing its coordinates
R	REDRAW	Redraws the whole drawing
S	STRETCH	Stretches an entity by specified distance
T	TEXT	Writes text

V	VIEW	Displays View dialog box
W	WBLOCK	Displays Write Block dialog box
X	EXPLODE	Breaks a group of entities into separate entities
Z	ZOOM	Enlarges or reduces the view

UNIT SUMMARY

In this unit you have learned:

1. The meaning of Computer Aided Drafting and related terminology.
2. About computer aided hardware and software required for CADr.
3. How to install AutoCAD Student version.
4. About AutoCAD main window user interface.
5. Opening a new/exiting, closing and saving a drawing.
6. How to create a user defined template.
7. Setting of units and limits.
8. Various Drawing aids like Snap, Grid, Object Snap to draw accurate drawing.
9. About Short Cut keys and Function keys.

EXERCISES

A. Multiple Choice Questions

5.1 RAM for a computer means

(a) Regular and Accurate Memory (b) Random Access Memory

(c) Rapid Access Management (d) Right Actual Mode

5.2 Size of a display unit is measured as

(a) Width of screen (b) Height of screen

(c) Distance across diagonal corners of screen (d) Average of width & height of screen

5.3 A Mouse is a

(a) pointing device on screen (b) disturbance in a computer

(c) device to locate fault (d) device to draw complex drawings

5.4 Files can be accessed over internet if a computer has a

(a) VGA card (b) Network card

(c) Modem card (d) Sound card

5.5 The device to make a hard copy of A0 size of drawing from the computer is
(a) Digitizer (b) Printer
(c) Scanner (d) Plotter

5.6 AutoCAD is a software for
(a) drafting (b) designing
(c) designing and drafting (d) drafting and manufacturing

5.7 Menu bar on the AutoCAD screen contains
(a) File, Edit, View, etc. (b) Draw, Modify, etc.
(c) Cut, Copy, Paste, etc. (d) None of the above

5.8 Commands given in AutoCAD appear
(a) in the drawing area (b) on the toolbar of commands
(c) at the command line (d) at the status line

5.9 Snap is turned on or off using function key
(a) F4 (b) F6
(c) F7 (d) F9

5.10 Function key F8 is used to toggle
(a) Coordinate display (b) Grid display
(c) Status bar (d) Ortho

Answers of Multiple Choice Questions

5.1 (b); 5.2 (c); 5.3 (a); 5.4 (c); 5.5 (d); 5.6 (a); 5.7 (a); 5.8 (c); 5.9 (d); 5.10 (d)

B. Subjective Questions

5.1 List the advantages of Computer Aided Drafting.
5.2 Describe the various displays on an AutoCAD screen.
5.3 Explain the method to open a new file in AutoCAD
5.4 Describe the functions assigned to Function keys (F1 to F11) in AutoCAD.
5.5 Discuss the importance of UCS icon. How can you put UCS icon on and off?

PRACTICALS

1. As per curriculum the practical related to Unit-5 are: Nil

KNOW MORE

- Teacher should download the command reference guide of AutoCAD 2020.
- During the input sessions the teacher must give examples related to the relevant branch viz. Mechanical and allied disciplines/ Electrical and allied disciplines/Electronics etc.
- Teacher should demonstrate all above commands through AutoCAD software and guide students in Hands-on sessions.
- Show video/animation films to explain mentioned concepts.
- Use videos/spoken tutorials to teach AutoCAD software.
- Teacher must know 3D modeling also using AutoCAD 2020.

Applications (Real life / Industrial)

Computers have brought drastic changes in many fields including engineering drawing. Not only the time involved in computer aided drawing or drafting is a fraction of that involved in manual drafting but also the revision can be done quickly without disturbing the entire drawing. It is very easy to draw different views of the object using CADr. Currently, industries, professionals and practicing engineers are using computer aided drafting software for creating digital production drawings in mechanical and allied engineering domains, digital building and construction drawings in civil engineering and architecture domain, digital pipe lines and layouts in chemical and food processing fields, digital layouts of power distribution system and electric circuits in electrical engineering domain. To use CADr software effectively the user must learn and practice all the commands, shortcut and functional keys.

Inquisitiveness and Curiosity

1. Download student version of AutoCAD and tryout above commands.
2. Open a new drawing with and without template and observe the difference.
3. Open an existing drawing file and save it with other name.
4. Open an existing file and check its Units and Limits.
5. Open a new file and set Units to mm. Make the grid ON/OFF and Snap ON/OFF. Change the setting of Snap and observe the movement of cursor.

REFERENCES & SUGGESTED READINGS

1. Machine Design Includes AutoCAD Second Edition, Ajeet Singh, Tata McGraw Hill Education Private Limited, New Delhi, 2018.
2. AutoCAD 2020: A Problem-Solving Approach, Basic and Intermediate, 26th Edition, Sham Tickoo, CADCIM Technologies, 2019.
3. AutoCAD 2013, Command Reference Guide, Autodesk Inc.
4. AutoCAD Shortcuts Guide, Autodesk Inc.
5. Autodesk knowledge network, tutorials, documentation, downloads, troubleshooting articles, https://knowledge.autodesk.com/support

6 Computer Aided Drafting

UNIT SPECIFICS

This unit presents information related to following topics:

- Draw basic entities like line, circle, arc, polygon, ellipse, rectangle, multiline, polyline.
- Method of Specifying points: Absolute and relative cartesian and polar coordinates.
- Modify and edit commands like trim, extend, delete, copy, offset, array, block, layers.
- Dimensioning the digital drawing: linear, horizontal, vertical, aligned, rotated, baseline, continuous, diameter, radius, angular. Dim scale variable. Editing dimensions.
- Writing text on the digital drawing: Single and multiline line text.
- Selecting various plotting parameters such as paper size, drawing orientation, plot scale, plot offset, plot area, print preview

The understanding of these topics will be developed after reading the contents, completing the solved problems, activities, performing exercises on AutoCAD software and viewing the ICT & web resources mentioned in this unit.

At the end of the unit, summary is provided to recapitulate the topics covered and applications are mentioned so that the learner can correlate the presented knowledge with real life and industrial situations. Few activities are mentioned to create inquisitiveness and curiosity in the learner. Subjective and objective questions are provided for the reinforcement of the knowledge and a list of references and suggested readings is also provided for further learning. Video resources along with QR codes are mentioned for getting more information on various topics of interest which can be surfed or scanned through mobile phone for viewing. Practical with details are provided to horn the drawing and drafting skills.

RATIONALE

Today every industry by default needs computerized drawings. The computerized drawings have their advantages of storage, retrieval, ease of modification, transmission etc. Further, the most critical factor is the accuracy of creation of overall design and specially curves which no hand drawing can match. To improve the employability quotient of the students many engineering institutions worldwide have adopted this approach. Usually digital drawings are created by combining and modifying several different basic primitive shapes, such as lines, circles, and arcs to create more complex shapes. AutoCAD is widely used design and drafting software to create these digital drawings. The version AutoCAD

2020 is being described in this book. The outcomes of this unit are performance based, hence, once you complete the unit, you will be able to perform the outcomes listed in the unit. In this unit the student will learn concepts of creating and modifying drawing objects, including lines, arcs and circles etc. This unit provides step-by-step instructions to use each command along with practice exercise for the command usage.

PRE-REQUISITES

Before reading this unit the student is advised to revisit following:

- Basic AutoCAD commands mentioned in Unit-5
- Maths: Coordinate and Plane geometry

UNIT OUTCOMES

After reading the contents, completing the solved problems, activities, performing exercises on AutoCAD software and viewing the ICT & web resources mentioned in this unit the students are expected to:

U6-O1: Use Draw tool bar of AutoCAD software to draw simple 2D entities

U6-O2: Use Modify tool bar of AutoCAD software to modify and edit 2D entities

U6-O3: Create drawings using Blocks, Array and Layer commands in AutoCAD

U6-O4: Apply dimensions to simple 2D entities using AutoCAD software

U6-O5: Write text in drawings using AutoCAD software

U6-O6: Plot drawing in AutoCAD software

Unit-6 Outcome	EXPECTED MAPPING WITH COURSE OUTCOMES *(1- Weak Correlation; 2- Medium Correlation; 3- Strong Correlation)*					
	CO-1	CO-2	CO-3	CO-4	CO-5	CO-6
U6-O1	3	-	2	-	-	3
U6-O2	3	-	2	-	-	3
U6-O3	3	-	2	-	-	3
U6-O4	-	3	-	-	-	2
U6-O5	-	3	-	-	-	2
U6-O6	-	-	-	-	-	2

6.1 INTRODUCTION

In this unit the student will learn concepts of creating and modifying drawing objects, including lines, arcs and circles etc. and also use Object Snaps, Object Snap Tracking, Polar Tracking, Coordinate methods and Dimensioning to create accurate drawings. Unit also covers knowledge elements related

to creation and modification of Layers, Blocks, Text writing and Hatching. The students will also learn how to navigate through drawings with Pan and Zoom commands, Do and Undo and finally Exiting from AutoCAD after saving the drawing. The section ahead provides instructions to use each command and also supplements with practice exercises to use the commands.

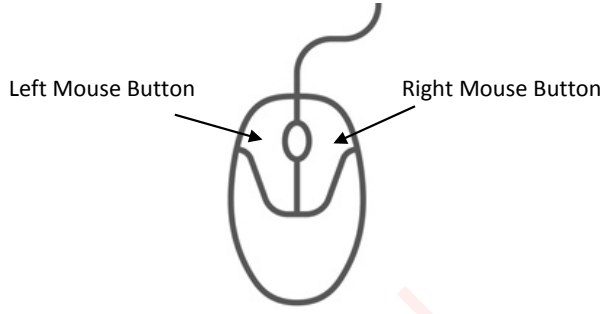


Fig.6.1: Typical Mouse: left and right clicks

6.2 USE OF MOUSE

It is assumed that you are using a mouse with a middle roller-button as depicted in Fig. 6.1.

Pick or Click	Quickly press and then release the left mouse button
Right-click	Quickly press and release the right mouse button
Double-click	Rapidly click the left mouse button twice
Drag	Press and hold down the left mouse button while you move the mouse
Point	Move the mouse until the mouse pointer (cursor) on the screen is positioned over the item you want
Select	Position the mouse pointer/drawing cursor over an item and click the left mouse button

6.3 HOW TO SELECT A COMMAND

AutoCAD provides you with two different methods of selecting commands:

1. One is selecting a tool from the Ribbon/Pallet,
2. The alternative is to type the command in the command line.

Both methods give the same output. Choose the method as per your preference. For example:

Method 1: Selecting the tool from the Ribbon (Fig. 6.2)

1. First select a **Tab** such as **Home**.
2. Locate the correct **Panel/Tool Bar/Pallet** such as **Draw** (Fig. 6.3).
3. Select a **tool** such as **Circle**.
   a. If the tool includes multiple types, it will show a down-arrow. On selecting the down menu arrow a submenu appears. Select the desired type such as 2-Point.
   b. The latest selection will then become the current displayed tool because AutoCAD assumes that you may need that tool again.

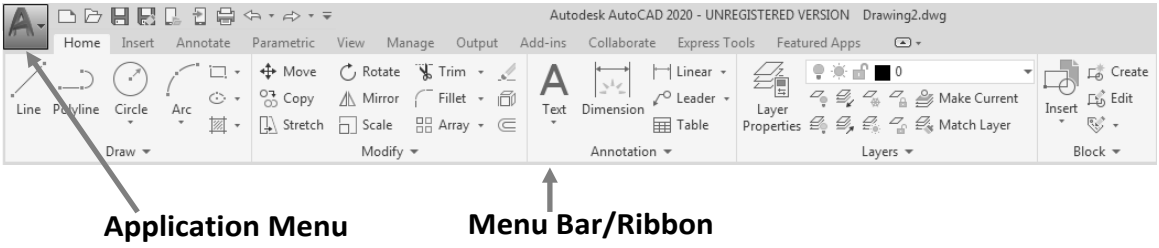


Fig. 6.2: Ribbon in AutoCAD 2020 main screen (user interface)

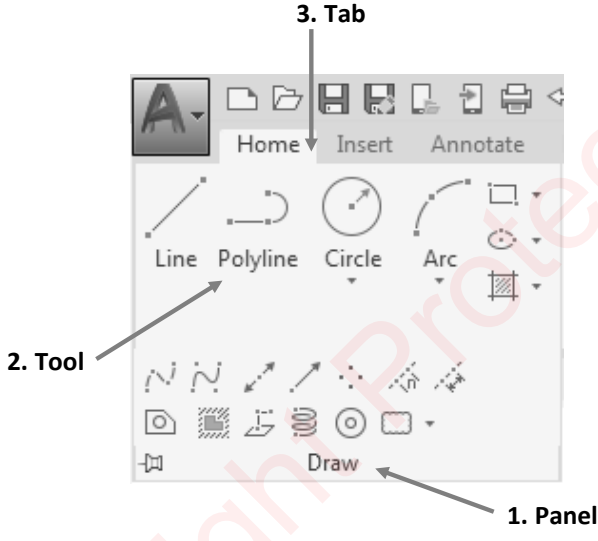


Fig. 6.3: Selecting a tool from the Ribbon

Method 2: Keyboard entry

The command may be input by typing the command on the command line or in the dynamic input tool tip (if dynamic input is on).

6.4 COMMAND LINE

Used to enter a command in the Command Line.

- Type the first letter of a command, such as 'c' for circle.
- A list of commands that begin with the letter 'c' will appear. Select the desired command from the list because 'c' is also related to Close, Change properties, Combine etc.
- On entering the command like circle the prompt and option gets displayed on the command line. The prompt for Circle command asks you to: 'Specify center point for circle' or [3P/2P/Ttr/(tan tan radius)]
- The information within [ ] brackets are options that you may select. Clickable options are displayed in blue while options displayed in black must be typed in the command line OR selected from the option menu.
- History gets recorded of the used commands. The history can be displayed using F2 or

the up arrow on the command line at the right hand end. Recent Commands tool displays recently selected commands.

- Press <Ctrl+9 > to toggle the command line on and off.
- By pressing space bar on keyboard you can activate the previous command used.

6.5 DYNAMIC INPUT TOOLTIP

This is another method of inputting commands, values and select options. Turn on the dynamic input button in the Status Bar to activate it. In case of Dynamic Input, the command will be entered in the tooltip box after the cursor.

- Place the cursor in the Drawing Area (important).
- Type the first letter of a command, such as 'c' for circle.
- A list of commands that begin with the letter 'c' will appear.
- Select the command from the list.
- If you press the ↓ down arrow, the options will appear below the prompt (Fig. 6.4).

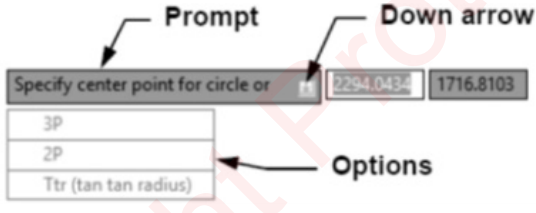


Fig. 6.4: Dynamic input method to enter a command

6.6 TOOL/ICON TIP HELP

- When you hover your cursor over a tool/icon an initial ToolTip will appear telling you the name of the tool with a brief description (Fig 6.5 a).
- If the cursor is placed little longer on the icon, a graphic display directly from the help system will appear (Fig 6.5b).

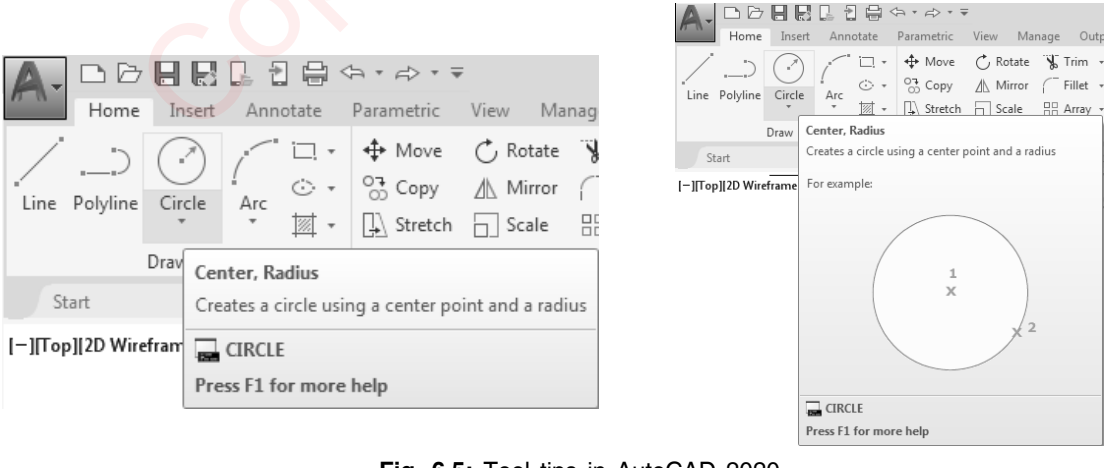


Fig. 6.5: Tool tips in AutoCAD 2020

6.7 COORDINATE SYSTEM

Absolute coordinate system. In absolute coordinate system the points are located in respect to the origin (0,0). The absolute coordinates are specified by entering X and Y coordinates. For example point 2,3 represents point with X=2 (i.e horizontal) and Y=3 (i.e vertical) from origin.

Relative Cartesian coordinates. Relative coordinates specify the X and Y distance from a previous point. They are called relative coordinates because they only have meaning relative to a point previously specified. You tell AutoCAD that the coordinates are relative by using the @ symbol. For example a Relative coordinate of @5,2 will go to the right 5 units and up 2 units from the last point entered.

Polar coordinates. Another common situation is to know the distance and angle of a point from either 0,0 or a previous point. In this case, you can use polar coordinates, which can be either absolute or relative. Most commonly, you use relative polar coordinates. Polar coordinates take the form distance<angle. (To type the angle symbol, use the less than symbol on your keyboard.) Relative polar coordinates must have the @ sign before the coordinate.

Direct distance entry. One shortcut for entering coordinates is direct distance entry. After you specify the start point of a line, at the Specify next point or [Undo]: prompt, simply move the mouse cursor in the direction you want the line to go and type in the line's length. It works best in orthogonal mode or with polar tracking, discussed next, which makes it easy to specify exact angles. You can use direct distance entry for any command that requires you to specify a distance and a direction, including both drawing and editing commands.

6.8 CREATING ELEMENTARY OBJECTS (DRAW TOOL BAR)

In AutoCAD, the screen acts as a drawing sheet and the drawing is not created by drawing instruments rather pressing buttons OR typing command related to basic 2D entities like Line, Circle, Arc, Rectangle, Polygon etc. To create a drawing in AutoCAD, Draw tool bar is used (Fig. 6.6 and 6.7). To create any entity (Point, Line, Arc, Circle etc.) either click the related icon available in the Draw Tool Bar OR type the respective command at Command line and then specify/supply the values as prompted. Just keep an eye on the messages prompted by the software and supply the required data. If you feel that something went wrong then press Esc key and the command will get terminated. If you prefer to separate the Draw Panel/Tool bar/Pallet from the Ribbon, you may drag the Panel off the Ribbon to a new convenient location on the screen.

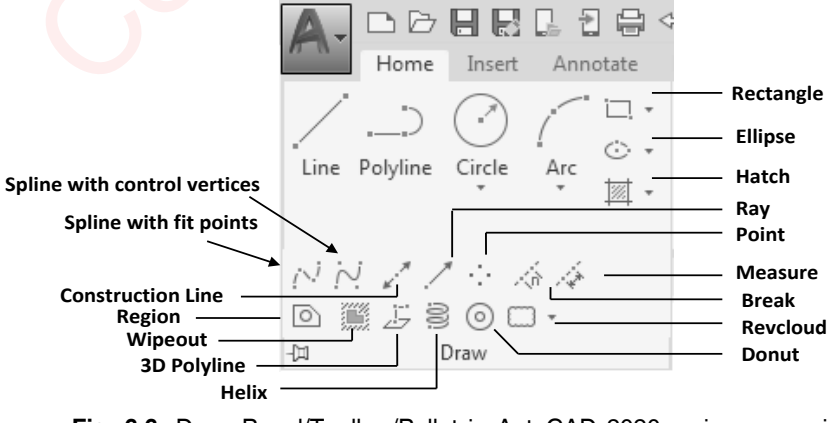


Fig. 6.6: Draw Panel/Toolbar/Pallet in AutoCAD 2020 main screen ribbon

6.8.1 Point

- The Point command will insert a point marker in your drawing at a position which you pick or at any coordinate location which you enter in the Command window.
- Other ways of defining a point can be accessed through the fly-out menu.
- By default the point style is a simple dot, but it can be changed any other using the point style dialogue box. (Try commands PDMODE & PDSIZE with different values)
- Tool Bar: Menu → Draw → Point
- Command: PO

6.8.2 Line

- Creates straight line segments
- It is used to draw lines continuously. It is possible to separately edit each line segment.
- To draw Horizontal and Vertical lines make Ortho ON.
- **Tool Bar**: Menu → Draw → Line
- **Command**: L
- Specify first point: press ENTER to continue from the last drawn line or arc or specify a point.
- The different options are:
  a. Continue: This helps to continue the line from the most recently drawn endpoint of the line.
  b. Close: End the line segment at the beginning of the first line segment, which forms a closed loop of line segment.
  c. Undo: Erases the most recent segment of a line sequence.

Example 6.1 *Draw following shape with any dimensions*
Solution

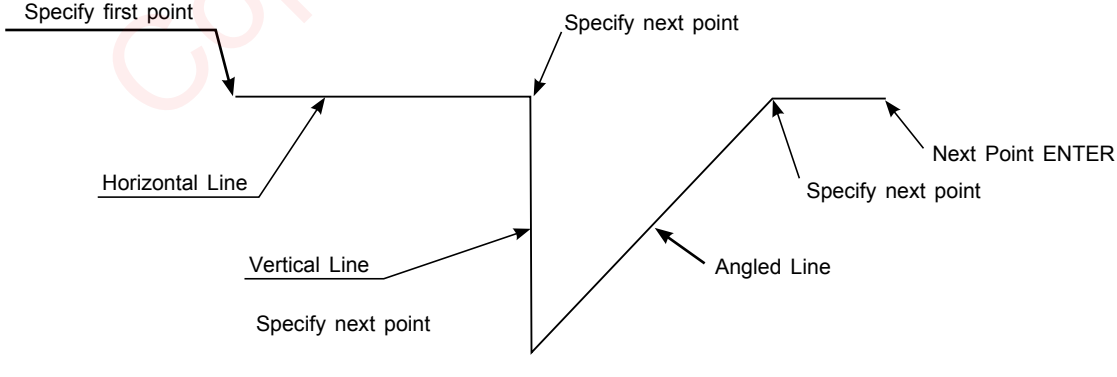


Fig. 6.7: Solution of example 6.1

1. Select the Line command.
2. Place the First endpoint anywhere in the drawing area.
3. Turn Ortho on by selecting the Ortho button or <F8>. (The "Ortho" button will change to a neon blue when on).
4. Move the cursor to the right and press the left mouse button to place the next endpoint. (The line should appear perfectly horizontal).
5. Move the cursor down and press the left mouse button to place the next endpoint. (The line should appear perfectly vertical).
6. Now turn Ortho off by selecting the Ortho button. (The "Ortho" button will change to gray when off).
7. Move the cursor up and to the right on an angle (the line should move freely now) and press the left mouse button to place the next endpoint.
8. Save the drawing as: Ex-1-Line (or any name)

6.8.3 Construction lines

- Creates an infinite line which passes through two picked points.
- Are very useful for creating construction frameworks or grids.
- Construction lines are not normally used as objects in finished drawings. Therefore, it is usual to draw all your construction lines on a separate layer (section 6.13) which will be turned off or frozen prior to printing.
- **Tool Bar:** Menu → Draw → Construction Line 
- **Command:** XLINE
- Specify a point or [Hor/Ver/Ang/Bisect/Offset]: Specify a point or enter an option.
- The different options are:
  a. Hor: Creates a horizontal construction line.
  b. Ver: Creates a vertical construction line.
  c. Ang: Creates a construction line at a specified angle.
  d. Bisect: Create a construction line that bisects an angle defined by 3 points.
  e. Offset: Creates a construction line that is offset from an existing line by a specified distance.

6.8.4 Multiple Lines

- Creates multiple parallel lines.
- Commonly used to create wall of any civil construction and Road in Top view.
- Tool Bar: Menu → Draw → Multiple Lines 
- Command: MLINE
- Specify start point or [Justification/Scale/Style] : Specify a point or enter an option.

6.8.5 Polyline

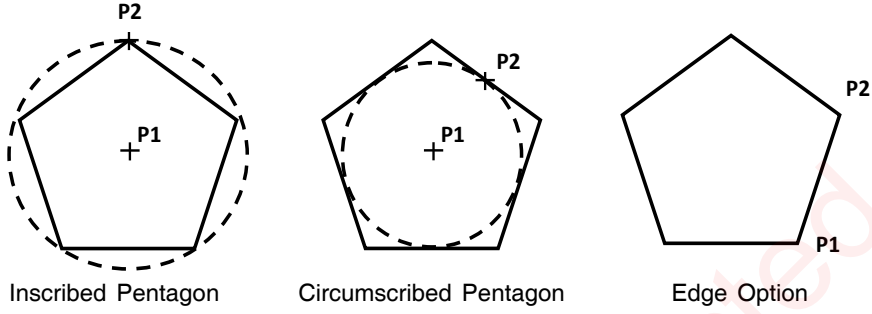
- Create two-dimensional polylines.
- A polyline is very similar to a line. It is created in the same way a line is drawn. It requires first and second endpoints. But a polyline has additional features, as follows:
  a. Polyline is one object, even though it may have many segments.
  b. You may specify a specific width to each segment.
  c. You may specify a different width to the start and end of a polyline segment.
  d. In polyline you can switch from arc to line and vice versa.
- Tool Bar: Menu → Draw → Poly Lines 
- Command: PLINE
- Specify start point :Specify point (1)
- Current line-width is <current>
- Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: Specify a point (2) or enter an option.
- The different options are:
  a. Width: Specify the starting and ending width. You can create a tapered polyline by specifying different starting and ending widths.
  b. Arc: This option allows you to create a circular polyline less than 360 degrees.
  c. Close: The close option is the same as in the Line command. Close attaches the last segment to the first segment.
  d. Length: This option allows you to draw a polyline at the same angle as the last polyline drawn. This option is very similar to the Offset command. You specify the first endpoint and the length. The new polyline will automatically be drawn at the same angle as the previous polyline.

6.8.6 Polygon

- Creates a polygon of any number of equal sides.
- Tool Bar: Menu → Draw → Polygon 
- Command: POLYGON
- Enter number of sides <current> : Enter a value between 3 and 1024 and press ENTER.
- Specify center of polygon or [Edge]: Specify point of center
- Enter ‘c’ (Circumscribed about Circle).
- Specify the radius length. Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.
- The different options are:
  a. Centre of Polygon: Defines the center of the polygon.
  b. Inscribed in Circle: Indicates the radius of a circle on which all vertices of the polygon line fall.

c. Circumscribe about circle: Indicates the distance from the centre of the polygon to the midpoints of the edges of the polygon.

d. Edge: Defines a polygon by specifying the endpoints of the first edge.



6.8.7 Rectangle

- Creates a Rectangle
- Tool Bar: Menu → Draw → Rectangle
- Command: RECTANGLE
- Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: Enter an option or specify a point (P1)
- Specify other corner point : Specify a point (P2)
- The different options are (Fig.6.8):
  a. Chamfer: Creates a rectangle with chamfer by specifying chamfer distances
  b. Fillet: Creates a filleted rectangle by specifying fillet radius
  c. Width: Controls the line width of rectangle
  d. Thickness: Controls thickness of rectangle
  e. Elevation: For drawing a rectangle at the specified height from the XY plane

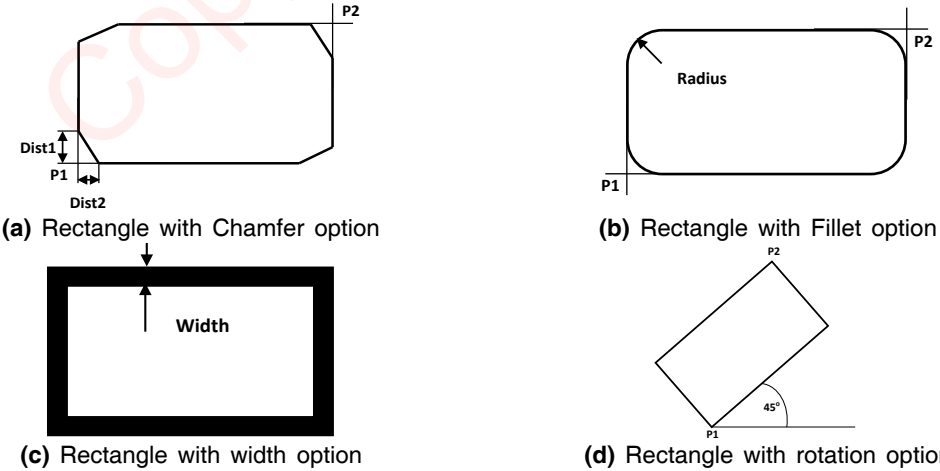


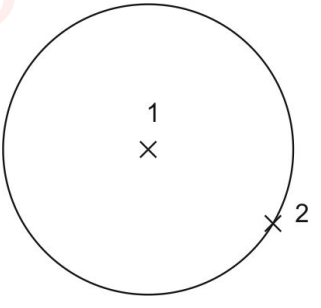
Fig.6.8: Draw rectangle with different options

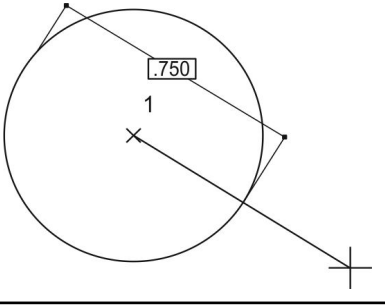
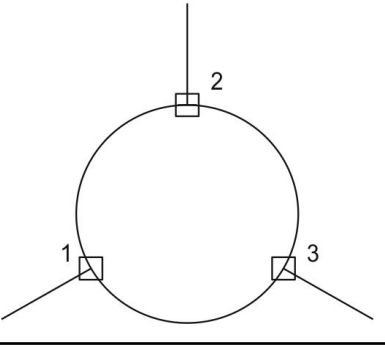
6.8.8 Circle

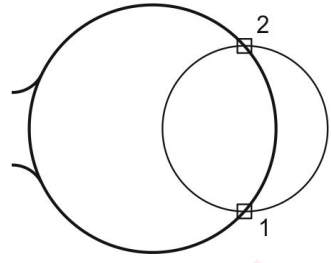
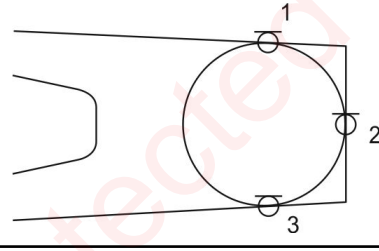
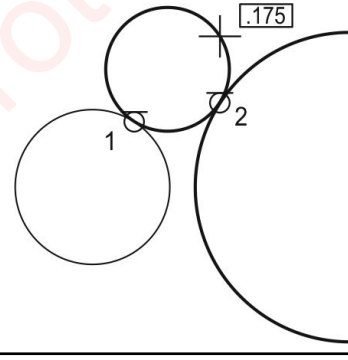
- Creates a circle
- Tool Bar: Menu → Draw → Circle
- Command: CIRCLE
- Specify center point for circle or [3P/2P/Ttr(tan tan radius)] : provide a point on the screen or give coordinates of the point or enter an option
- The different options are:
  a. Center, Radius: By defining the center point and radius of the circle
  b. Two Point: By specifying the endpoints of circle diameter
  c. Three Point: By specifying 3 points on the circle
  d. Tangent, Tangent Radius: It draws a circle which is tangent to two objects (line, circle or arc) and with a specified radius.

Center, Radius

- Specify the center point for the circle by moving the cursor to the desired location in the drawing area or provide the coordinates of center point (1)and click the left mouse button.
- Now as you move the cursor away from the center point, a circle will start forming.
- When the circle of desired size is visible (2), press the left mouse button, or you can type the radius, give value and then press ENTER.



Centre point and Diameter Defines the diameter of the circle.	
3P (Three Points) To draw a circle when three points on circumference are selected.	

2P (Two Points) To draw a circle when two endpoints of the diameter are selected.	
Tan, Tan, Tan To draw a circle tangent to three given objects.	
Tan, Tan, Radius To draw a circle with radius value and tangent to two given objects.	

6.8.9 Arc

- Creates an arc. In AutoCAD an arc can be drawn in 11 distinct ways.
- The default method of drawing an arc is 3point method. Other options are:
  a. Start, Center, End: Specify start point, center point and end point of arc
  b. Start, Center, Angle: Specify start point, center point and included angle of arc
  c. Start, Center, Length: Specify start point, center point and cord length
  d. Start, End, Angle: Specify start point, end point and included angle
  e. Start, End, Direction: Specify start point, end point and starting direction of arc
  f. Start, End, Angle: Specify start point, end point and radius of arc
  g. Center, Start, End: Specify center point, start point and end point of arc
  h. Center, Start, Angle: Specify center point, start point and included angle of arc
  i. Center, Start, Length: Specify center point, start point and cord length
  j. Continue: Continue drawing an arc from a previously drawn arc

- **Tool Bar:** Menu → Draw → Arc
- **Command:** ARC

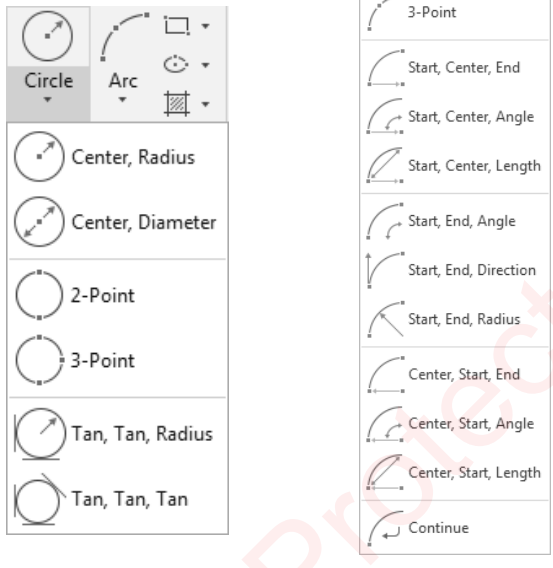


Fig. 6.9: Different options to draw circle and arc in AutoCAD

- The arc can be created using the Arc command and using any of the three specified values given below. Specify the value asked by the prompt then type the characters displayed in uppercase within brackets and then press ENTER key
  a. Start point
  b. Center point
  c. End point
  d. Radius
  e. Included angle
  f. Chord length
  g. Direction at start point.

6.8.10 Donut

- Draws filled rings, annulus and circles.
- Tool Bar: Menu → Draw → Donut
- Command: DONUT
- Specify inside diameter of donut <current>: Specify a distance or press ENTER
- Specify outside diameter of donut <current>: Specify a distance or press ENTER
- Specify center diameter of donut <exit>: Specify a point by clicking on screen or providing coordinates or press ENTER/Esc to end the command.

6.8.11 Spline

- Creates a quadratic of cubic spline (NURBS) curve
- Tool Bar: Menu → Draw → Spline 
- Command: SPLINE
- Specify first point or [Object]: Specify a point or enter 'O'
- Depending on the method the first point can be the first fit point or the first control vertex.
- Specify next point: Specify/Click next point this will create additional spline segments until ENTER key is pressed.

6.8.12 Ellipse

- Creates an ellipse or an elliptical arc
- Tool Bar: Menu → Draw → Ellipse 
- Command: ELLIPSE
- There are 3 methods to draw an Ellipse (Fig. 6.9). Either specify 3 points of the axes or define the center point and the axis points or define an elliptical Arc.

Axis, End method

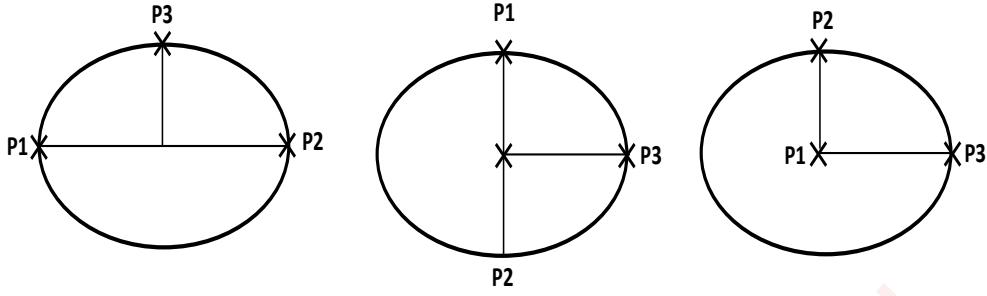
- Command: ELLIPSE
- Specify axis endpoint of ellipse [Acr/Center/Isocircle]: Specify/Click the primary point on the major or minor axis (P1). [Refer Fig.6.10(a)]
- Specify/Click other endpoint of the axis (P2)
- Specify distance to other axis or [Rotation]: Specify/Click third point (P3) perpendicular to the first axis.

Center method

- Command: ELLIPSE
- Specify center of ellipse: Specify/Click center of ellipse (P1) [Refer Fig.6.10(b)]
- Specify endpoint of axis: Specify/Click first axis endpoint (either axis) (P2)
- Specify distance to other axis or [Rotation]: Specify/Click third point (P3) perpendicular to the first axis.

Isocircle option

- In isometric view the appearance of a circle is like an ellipse. In AutoCAD it is called as isocircle and is useful for isometric view.
- Set the snap style option of SNAP to isocircle and isometric snap on. Function key F5 can be used to obtain different shapes of an isocircle depending on the type of view (front, side and top).



Specifying Major axis first (P1/P2), then Minor axis (P3)

Specifying Minor axis first (P1/P2), then Major axis (P3)

Specify Center (P1) then P3 and P2

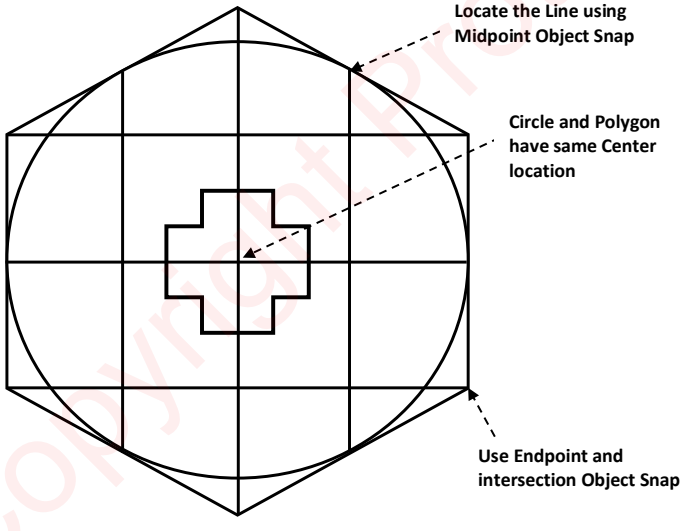
(a) Axis, End method

(b) Center method

Fig. 6.10: Different options to draw ellipse in AutoCAD

Example 6.2 *Draw following shape with any dimensions.*

Solution

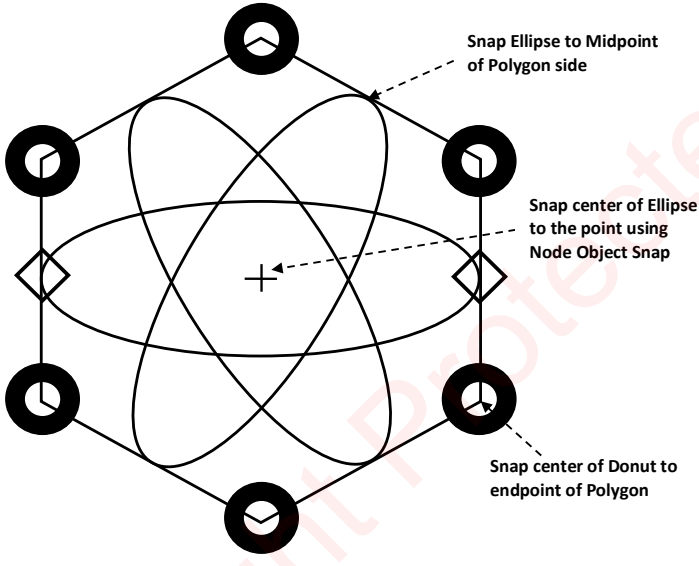


1. Start a New file.
2. Draw the Circle using the Center / Radius option.
3. Using Object Snap draw the Circumscribed Polygon next.
4. Locate the center of the Polygon at the center of the Circle using Center object snap and locate the radius of the Polygon on the Circle using Quadrant object snap.
5. Ortho <F8> on.
6. Use Object Snaps Midpoint and Endpoint to draw lines.
7. Complete the middle shape using line command and object tracking option on.
8. Save the drawing as: Ex-2

Example 6.3 *Draw following shape with any dimensions.*

Solution

1. Start a New file
2. To draw the given figure use Polygon, Ellipse and Donut commands as per instructions shown in the figure
3. Save the drawing as: Ex-3



6.8.13 Block

- Creates a block definition from objects you select.
- A Block is a cluster of objects that are converted into one object. The applications of the block command are drawing of a nut, bolt, rivet, gear tooth, transistor, bathroom fixture, window, screw, tree pipe fittings.
- First a Block must be created then it is inserted into the drawing. In terms of space inserted block uses less space than a set of objects.

Creating a Block

- To create a block, first you have to draw all the objects which will be converted into a Block
- Tool Bar: Menu → Draw → Block 
- Command: BLOCK
- AutoCAD displays the Block Definition dialog box (Fig. 6.11).
  1. Enter the New Block name in the Name box
  2. Select the Pick point button (Or you may type the X, Y and Z coordinates). As you click on this button the Block Definition box will disappear and you will return temporarily to the drawing.

3. Choose the location for insertion point of the block. Usually this point is the Center, Midpoint or Endpoint of an object. On insertion of the block it is the insertion point to which the block gets attached.

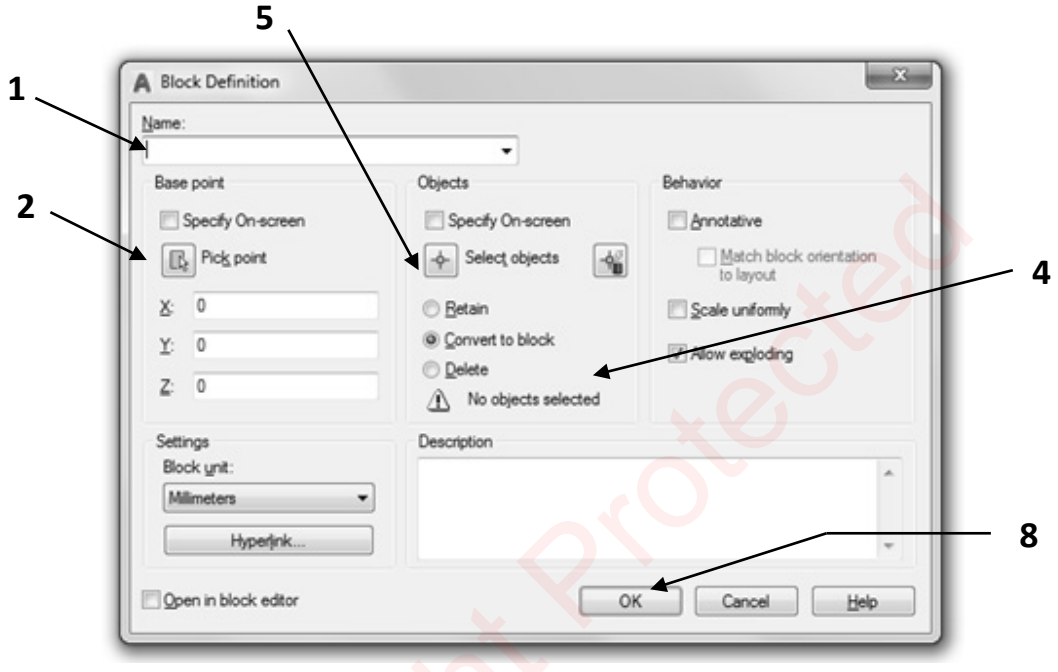
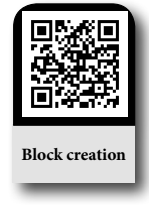


Fig.6.11: Block definition dialog box in AutoCAD

4. Select one of the options:
   a. Retain: After the block creation the original objects will remain on the screen.
   b. Convert to block: After the block creation the original objects will disappear from the screen and instantly reappear as a block.
   c. Delete: After the block creation the original objects will disappear from the screen.
5. After this the select objects button should be clicked. Now select the objects individually or by window options and then press ENTER.
6. The Block Definition box disappears and the control return temporarily to the drawing.
7. Select the OK button.
8. Now new block is created and stored in the drawing's block definition table.

Inserting a Block

- A Block can be inserted at any location in the same or another drawing. During insertion one can scale, rotate or explode the block.
- Select the Insert command through Ribbon or keyboard:
  a. Ribbon → Insert Tab → Block Panel → Insert Block
  b. Keyboard → INSERT then press ENTER



- Insert Window will appear. Choose the desired block.
- Choose the location on screen to insert the block and press left mouse button.
- To insert block in another file save the block with WBLOCK command.

6.8.14 Hatch

- Fills a specified boundary using a suitable pattern. The hatching will be carried out inside a closed defined area.
- The type of pattern and pattern variables can be chosen from a library of patterns available.
- Tool Bar: Menu → Draw → Hatch
- Command: BHATCH
- Enter a pattern name or [?/Solid/User Defined] <current>: Enter a predefined or custom pattern name, enter an option or press ENTER key.
- This will display the Boundary Hatch and Fill dialog box (Fig. 6.12).
- The hatch dialogue box has two tabs Hatch and Gradient, click on the Hatch tab.
- Let the Type be pre-defined as default type.
- In the pattern combo box click arrow to select the type of pattern. More patterns will appear on clicking the three dots button on the right side. Hatch Pattern Palette dialog box is displayed. Choose a pattern from this dialog box and by clicking OK button you can come back to Hatch and Gradient dialogue box.
- The pattern selected is shown in a window below that combo box.
- In the Angle combo box let the Angle be zero and Scale as 1.0000.
- To select the boundary for hatching, you can use any one of the two buttons on right side;
- Add: Pick Points or Add: Select Objects.
- If you choose Pick Point option, the dialog box disappears and then you click anywhere in a closed boundary where hatching is required. Dashed lines will display the selected boundary.
- If you choose Select Object button you have to select all the objects forming a boundary by clicking on them one by one.
- Press ENTER key after the selections are made.
- Click Preview, the hatching is displayed.
- At this stage if you are not satisfied with the spacing and angle of the hatching lines then you can increase/decrease the scale and angle value. Again click Preview and if satisfied Press OK button.
- Instead of hatch you can fill colour in the enclosed area by clicking on Other predefined tab and then Solid. Click Gradient tab to shade an area in which the colour fades gradually in a chosen pattern.

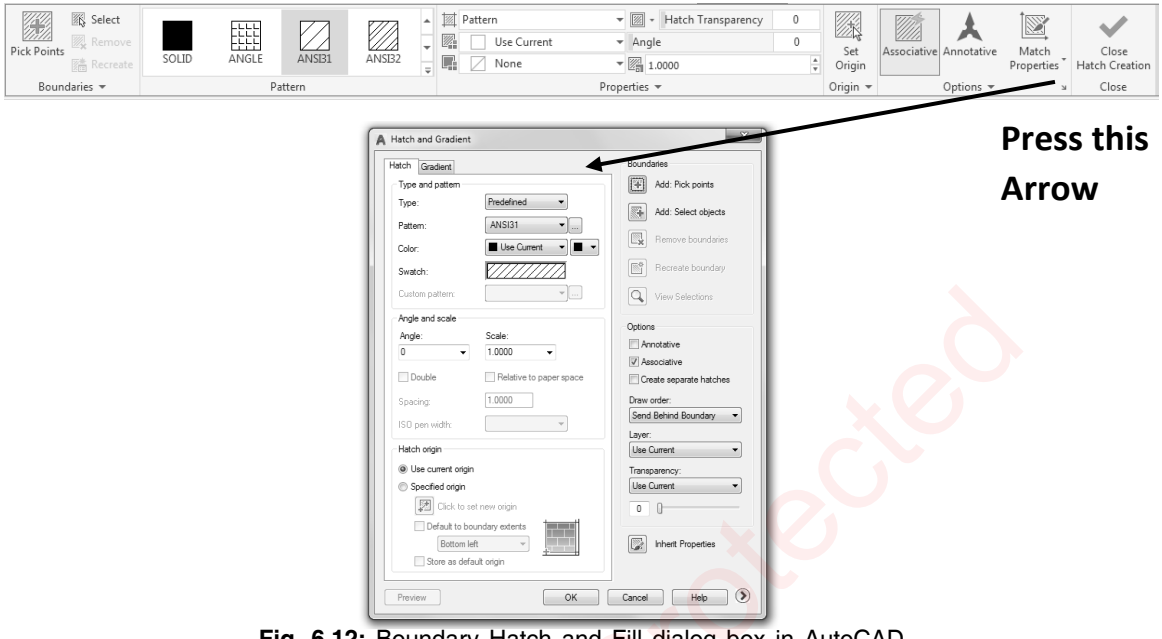


Fig. 6.12: Boundary Hatch and Fill dialog box in AutoCAD

6.8.15 Region

Creates a region of objects from a selection set of existing objects

Define Regions

- Tool Bar: Menu → Draw → Region
- Command: REGION
- Select objects for creating the Region, these objects should form an enclosed area like a circle or a closed polygon.
- Press ENTER key. The Command prompt provides a message that indicates the number of loops detected and the number of regions created.

Define Regions with Boundaries

- Tool Bar: Menu → Draw → Boundary
- Command: BOUNDARY
- Boundary Creation dialog box will be displayed, select Region in the Object Type list.
- Click Pick Points.
- Click a point inside each closed area that you want to make as a region and press ENTER.

6.9 MODIFYING COMMANDS

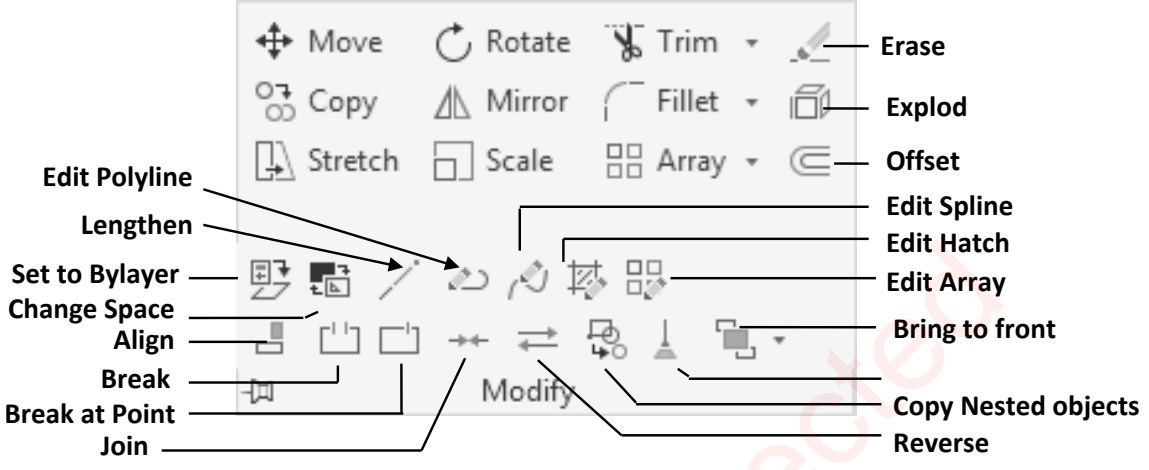


Fig.6.13: Icons in Modify Panel/Toolbar/Pallet in AutoCAD 2020 main screen Ribbon

6.9.1 Erase

- Removes selected objects from a drawing
- Objects can be raised by three methods from the drawing
- Tool Bar: Menu → Modify → Erase
- Command: ERASE

Method 1

- Click the Erase command and then select the objects.
- Select objects: Select the object to be erased individually or using window options.
- Select objects: Press ENTER. The selected objects will be deleted.

Method 2

- Select the object to be erased individually or using window options.
- Click the <Delete> key on the keyboard.

Method 3

- Select the object to be erased individually or using window options.
- Press the right mouse button.
- Select Erase from the Shortcut Menu using the left mouse button.

6.9.2 Copy

- Creates duplicates of already created objects.
- The steps required are:

a. Select the objects to be copied.

b. Select a base point.

c. Select a new location for the new copy.

- Tool Bar: Menu → Modify → Copy 
- Command: COPY 
- Select objects to be copied (for multiple selection press shift and select) and press ENTER Key
- Specify base point or displacement, or [Multiple]: Choose a point for a single copy or enter 'm' for multiple copies.
- Specify/Click a second point. The selected objects are copied to the new location.
- If the second point is very near to the base point and object snap is on then sometimes copy may not appear. Press F3 to put the snap off and repeat COPY command.

Copy an Object using a Displacement

- Tool Bar: Menu → Modify → Copy
- Select the objects to copy and press ENTER.
- The displacement can be entered using Cartesian, polar, cylindrical or spherical coordinates.
- The @ sign should not be used as the software assumes it as relative coordinates.
- At the prompt for the second point, press ENTER.
- The selected objects will be copied to a new location based on the relative coordinate values entered.

6.9.3 Mirror

- Creates a mirror image copy of objects
- Tool Bar: Menu → Modify → Mirror
- Command: MIRROR
- Select objects to mirror: Use an object selection method and press ENTER to finish
- Specify first point of mirror line: Specify a point (1)
- Specify second point of mirror line : Specify a point (2) (Fig. 6.14 and 6.15)
- To control mirroring of text when using the Mirror command do the following before the use of Mirror command:

  a. Command Line: type mirrtext then press ENTER

  b. If you want the text to mirror (reverse reading), type: 1 and ENTER

  c. If you do not want the text to mirror, type: 0 and ENTER

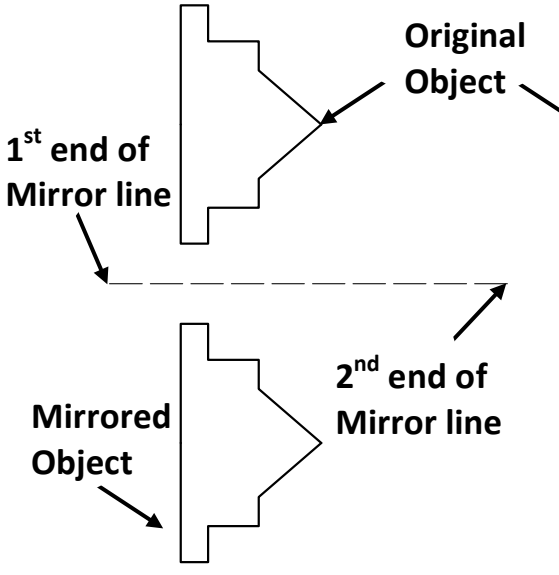


Fig. 6.14: Use of Mirror Command: Mirror Line Horizontal

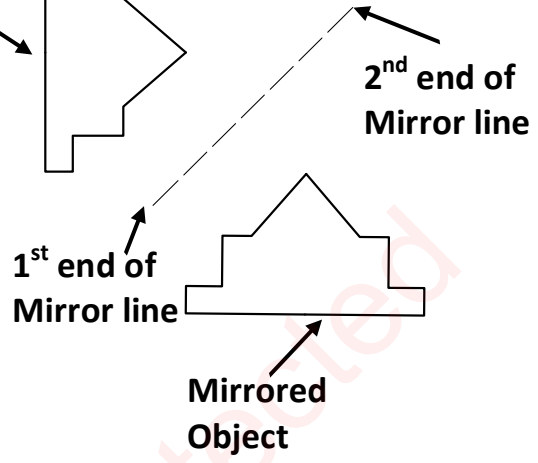


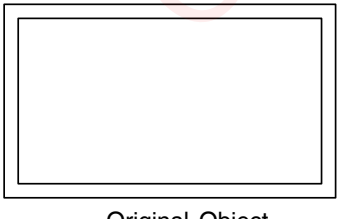
Fig. 6.15: Use of Mirror Command: Mirror Line at an Angle

6.9.4 Offset

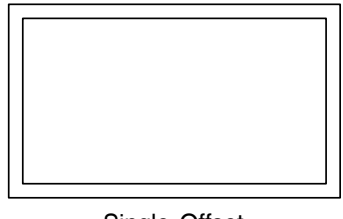
- Duplicates an object parallel to the original object at a specified distance. You can offset lines, arcs, circles, ellipses, 2D polylines and splines.
- Tool Bar: Menu → Modify → Offset 
- Command: OFFSET
- Specify offset distance of [Through] <current> Specify offset distance or enter 't' (Through).

Offset distance option

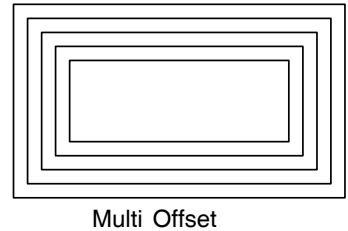
- You can either enter a value or use the pointing device to determine a distance with two points. Press ENTER Key
- The point needs to be clicked/specified inside or outside of the original object to indicate the offset direction



Original Object



Single Offset



Multi Offset

Through option

- Specify/Click a point through which the offset object will pass.
- Select the object to offset and press ENTER Key.

6.9.5 Array

This command facilitates you to create multiple copies in a Rectangular, Polar (Circular) fashion and even on a Path (Fig. 6.16).

- Tool Bar: Menu → Modify → Array 

- Command: _ARRAY

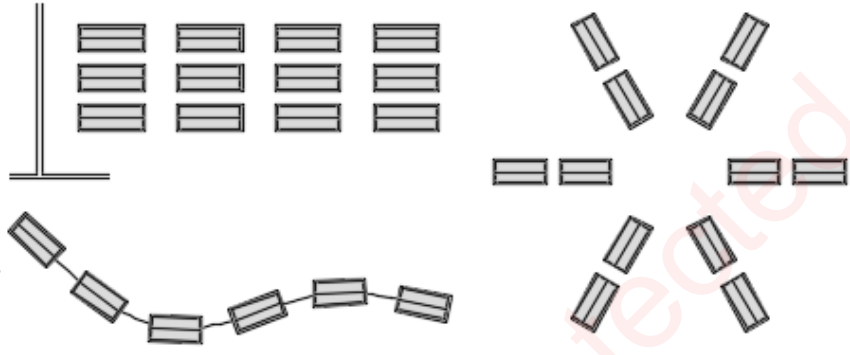


Fig. 6.16: Example of types of Array

Rectangular Array

In this option multiple copies (instances) of selected object(s) are arranged in a rectangular or matrix form with rows and column pattern. The user has to provide number of rows and columns followed by distance between the rows and distance between the columns. The copies are places at equal intervals and once the copies are made the user can edit any instance independently.

- Tool Bar: Menu → Modify → Array → Rectangular Array 
- Command: ARRAY
- Select Objects: User has to select the object(s) to be arrayed individually one after another or using window selection methods. Press ENTER to stop.
- Enter the type of array [Rectangular/Polar]<current> Rectangular Array and press ENTER key.
- The Array Creation Tab appears (Fig.6.17) and a 3 x 4 default grid array of the object will be displayed on the screen.
- On this preview the spacing and number of rows and columns can be adjusted in following manner:

  Using first ► or ▲ symbols, the user can change the spacing between the columns or rows.

  Using last ► or ▲ symbols the user can change the total spacing between the base point, can add extra columns/rows and can change the axis angle.

  Using ■ symbol the total row and column spacing and numbers can be changed simultaneously.

  To move the complete array use the Base Point grip ■.
- Above changes in the array can also be made directly through Array Creation Tab.
- Number of rows (---): Provide a non-zero integer to represent the number of rows. For one row more than one column needs to be specified and vice versa.

- Number of columns (|||): Specifies the number of columns.
- Distance between rows or specify unit cell: Specifies the distance between rows, including the length of the object to be arrayed. To add rows downward, specify a negative value for the distance between rows.
- Distance between columns (|||): Specifies the distance between columns. A negative distance value will add columns to the left.
- To display changes in the array press ENTER key.
- Click Close Array tab to finalize the array display.

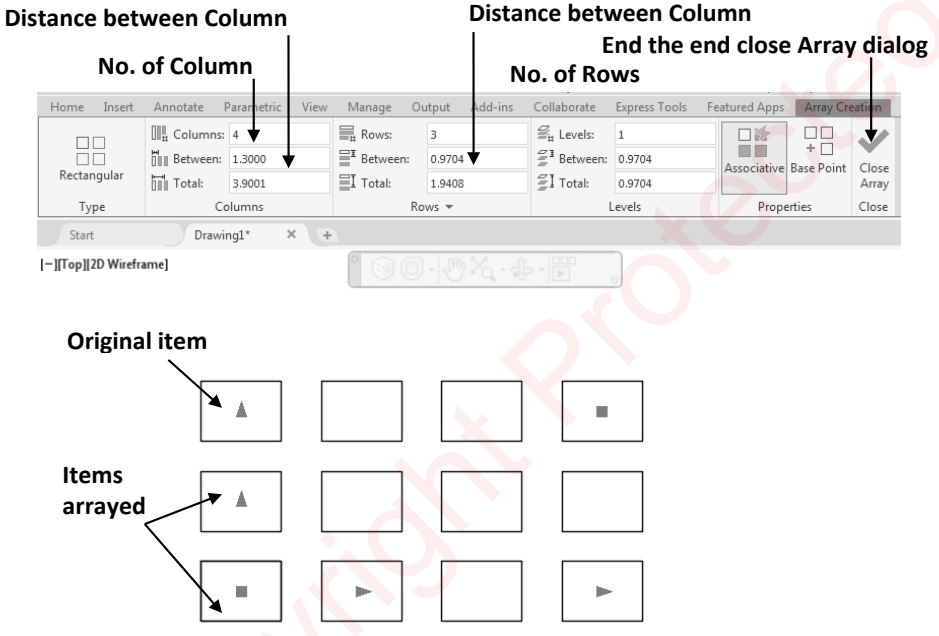
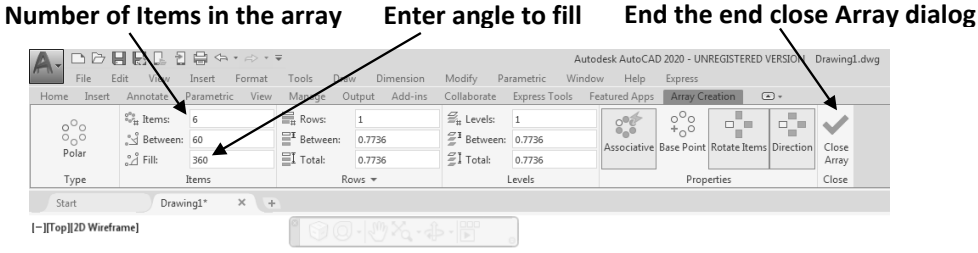
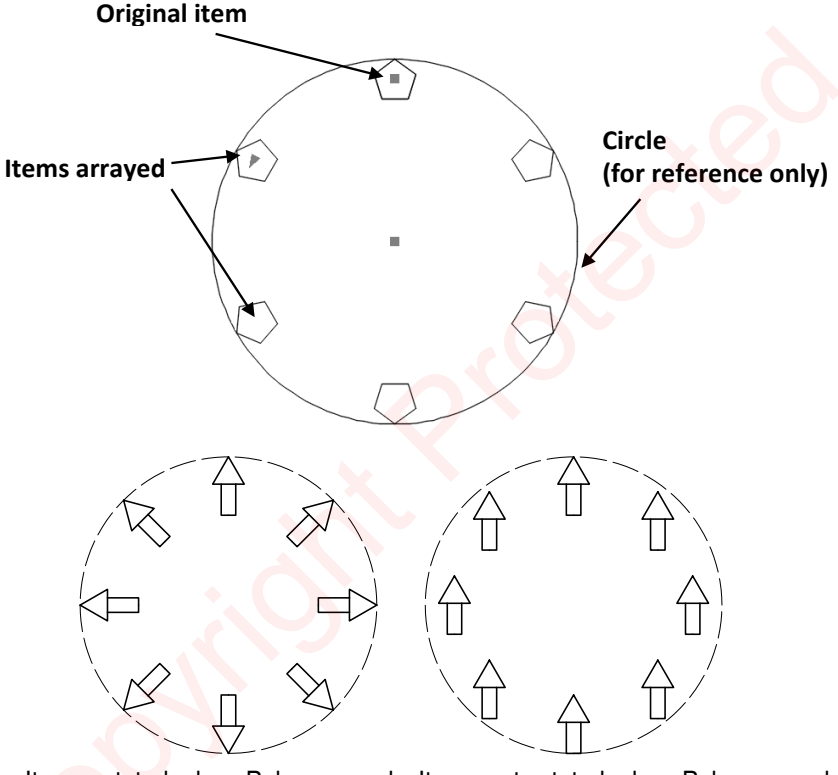


Fig. 6.17: Rectangular Array Creation Tab

Polar Array

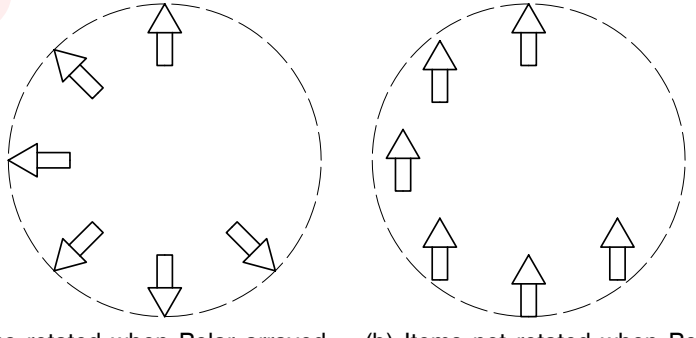
- In this option multiple copies (instances) of selected object(s) are arranged in a circular fashion.
- In this option the user has to select the object(s), provide center of the invisible (reference) circle on which the instances will be placed after circular arraying, number of instances, angle of fill or angle between each copy and angle to fill. User can also have option to rotate/ unrotate the instances after arraying.
- Tool Bar: Menu → Modify → Array → Polar Array 
- Command: ARRAY
- Draw the object(s) to be arrayed and it is advisable to draw a circle also with dotted line for reference.
- Select the object(s) to be arrayed





Items rotated when Polar arrayed Items not rotated when Polar arrayed

Fig. 6.18: Polar Array creation tab with 360° fill angle



(a) Items rotated when Polar arrayed (b) Items not rotated when Polar arrayed

Fig. 6.19: Polar Array Creation Tab with 225° fill angle

- Select Objects: Select more objects or ENTER to stop.
- Enter the type of array [Rectangular/Polar]<current> Enter Polar Array and press ENTER key.
- Specify center point of array or [Base point / Axis of Rotation]: Specify/Click the center point of the reference (invisible) circle. It is always better to draw a circle for reference and choose the center of this circle.
- The Array Creation Tab appears with default polar array of 6 items (Fig. 6.18). Enter number of instances and enter angle between the instances and enter fill angle. If the number of instances are 12, angle between items are 30^0 and angle of fill is 360^0 then the 12 items are evenly distributed within 360^0 as shown in Fig.6.18. In another situation if the number of items are say 6, angle between items are 45^0 and angle of fill is 225^0 then the 6 items are placed as shown in Fig.6.19.
- To display changes in the array press ENTER key.
- Click Close Array tab to finalize the array display.

6.9.6 Move

- Displaces objects at a specified distance in a specified direction.
- Tool Bar: Menu → Modify → Move
- Command: MOVE
- Select objects: Select object(s) individually or using window selection methods and once over press ENTER key.
- Specify base point or displacement. Specify/Click a point (P1) (base point) (usually on the object).
- Specify second point of displacement or <use first point as displacement> Specify a point (P2) and press the left mouse button or press ENTER key (Fig.6.20)

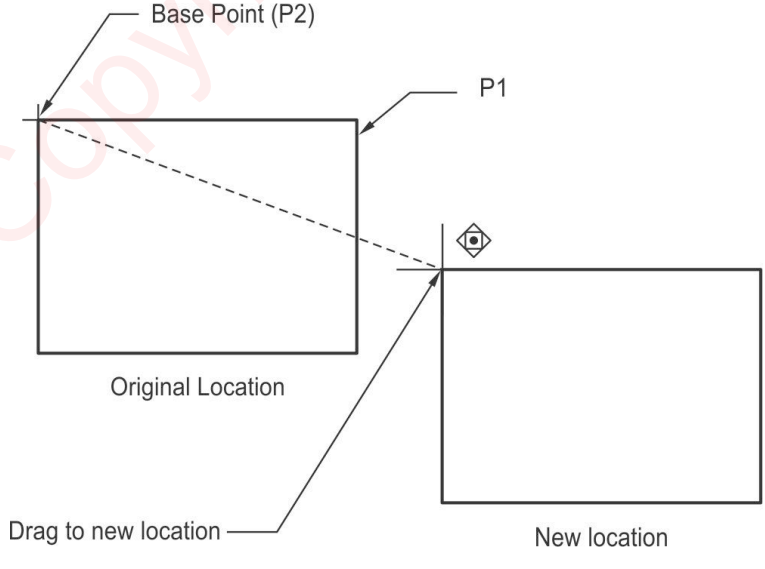


Fig. 6.20: Move command

6.9.7 Rotate

- Rotates objects about a base point (pivot point)
- After selecting the objects and the base point, you will enter the rotation angle from its current rotation angle or select a reference angle followed by the new angle.
- A positive rotation angle revolves the objects Counter-Clockwise and negative rotation angle revolves the objects Clockwise.
- Tool Bar: Menu → Modify → Rotate 
- Command: ROTATE
- Select object(s) individually or using window selection methods and once over press ENTER key.
- Specify/Click a point about which the rotation will be done (base point)
- Specify rotation angle or [Reference]: Specify an angle, specify a point, or enter 'r'
- Any of the following can be done:
  a. Enter the angle of rotation.
  b. Drag the object around its base point and specify a point location to which you want to rotate the object.
  c. Enter 'c' to create a copy of the selected objects.
  d. Enter 'r' to rotate the selected objects from a specified reference angle to an absolute angle.
- Reference: Specifies the absolute current rotation angle and desired new rotation angle. The Reference option can be used to align or straighten the object with other features in a drawing.

6.9.8 Scale

- Enlarges or reduces selected objects equally in the X, Y and Z directions.
- Tool Bar: Menu → Modify → Scale
- Command: SCALE 
- Select object(s) individually or using window selection methods and once over press ENTER key.
- Specify/Click a point about which the scaling will be done (base point)
- Specify scale factor or [Reference]: Specify a scale or press ENTER key
- Scale Factor: Multiples the dimensions of the selected objects by the specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects.

6.9.9 Trim

- Trims objects at a cutting edge defined by other objects.
- Tool Bar: Menu → Modify → Trim 
- Command: TRIM
- Select a cutting edge or edges and press ENTER key.
- Selects the object to be trimmed. The cutting edge can be any line, arc, circle, etc. The selected

entities beyond the selected cutting edge are trimmed. The position of the selection decides the side to be trimmed. If two cutting edges are used then the entity between the two cutting edges is trimmed.

6.9.10 Extend

- Extend an object to meet another object
- Tool Bar: Menu → Modify → Extend
- Command: EXTEND
- Select boundary edges.
- Select objects: Use an object selection method and press ENTER when you finish
- Select object to extend or {Project/Edge/Undo]: Select an object or enter an option

6.9.11 Chamfer

- Bevels or chamfers the edges of two 2D objects or the adjacent faces of a 3D solid
- Tool Bar: Menu → Modify → Chamfer
- Command: CHAMFER
- Select first line or [Polyline/Distance/Angle/Trim/Method]
- Select the first of two objects or the first line segment of a 2D polyline to define the chamfer (Fig. 6.21).
- Select the second object or line segment of a 2D polyline to define the chamfer.
- Polyline: Inserts a chamfer line at each vertex of a 2D polyline where two straight line segments meet.
- Distance: Sets the chamfer distances from the intersecting points of the first and second objects.
- Angle: Sets the chamfer distance from the intersecting point of the selected objects and the XY angle from the first object or line segment.
- Trim: Controls whether the selected objects are trimmed to meet the endpoints of the chamfer line.
- Method: Controls how the chamfer line is calculated from the intersecting point of the selected objects or line segments.

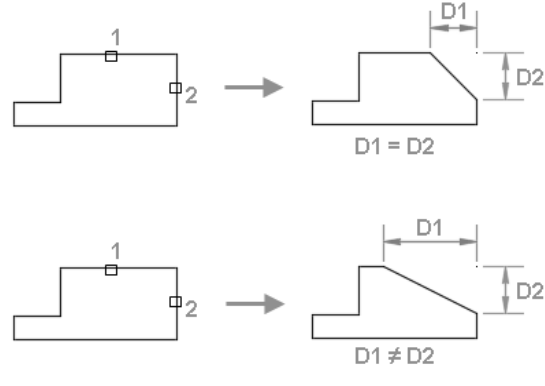


Fig. 6.21: Chamfer command

6.9.12 Fillet

- Rounds and fillets the edges of objects.
- The objects may or may not touch each other. If two parallel lines are selected, it will construct a full radius (Fig.6.22).
- Tool Bar: Menu → Modify → Fillet
- Command: FILLET
- Enter 'r' (Radius) at the command prompt.
- Enter a new fillet radius value.
- Once the fillet radius has been set, select the objects or line segments that define the points of tangency for the resulting arc or press ENTER to terminate the command.

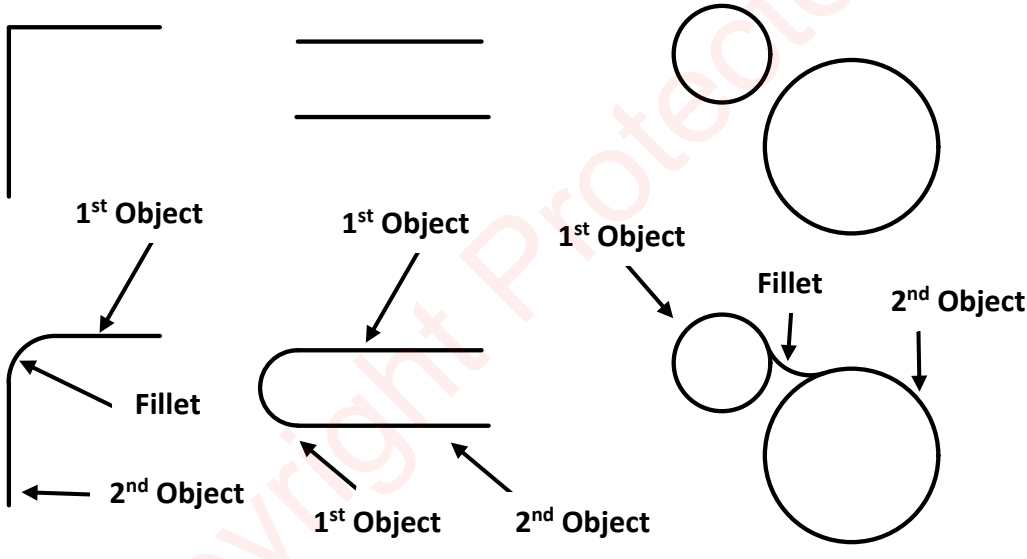


Fig. 6.22: Fillet command

6.9.13 Break

- Splits an object in two
- Tool Bar: Menu → Modify → Break
- Command: BREAK
- Select the object that user want to break.
- By default, the point at which you select the object is the first break point. To select a different pair of break points, enter 'f' (First) and specify the first break point.
- Click/specify the second break point.
- To break an object without creating a gap, enter @0,0 to specify the previous point.

6.9.14 Explode

- Breaks a compound object into its component objects
- Tool Bar: Menu → Modify → Explode 
- Command: EXPLODE
- Select the objects to be exploded.
- Exploding has no visible effect for most of the objects.

Illustrative Example-2: Array

6.10 CONTROLLING DRAWING DISPLAY

6.10.1 Redraw

- Refreshes the display in the current viewport.
- Removes temporary graphics left by VSLIDE and some operations from the current viewport. To remove stray pixels, use the REGEN command.
- Tool Bar: Menu → View → Redraw
- Command: REDRAW

6.10.2 Regen

- REGEN command directs AutoCAD to re-generate the drawing to update it. The command refines the drawing by smoothing out arcs and circles.
- Tool Bar: Menu → View → Regen
- Command: REGEN

6.10.3 Zoom

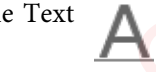
- The Zoom command is used to move closer to or farther away from an object. This is called Zooming In and Out. Zoom command is useful to work on minute details of a drawing.
- Tool Bar: Menu → View → Zoom 
- Command: ZOOM

6.10.4 Pan

- Shifts the view any direction without changing the viewing direction or magnification.
- Position the cursor at the start location and press the left mouse button down. Drag the cursor to the new location. You can also press the mouse scroll wheel or middle button down and drag the cursor to pan.
- Tool Bar: Menu → View → Pan 
- Command: PAN

6.11 TEXT AND DIMENSIONS

6.11.1 Text

- Words, messages and numbers can be inserted as required on an engineering drawing. The alphanumeric keyboard is used extensively for non-graphical input such as text. The text style, height, text angle, aspect ratio, colour, etc. are some of the attributes associated with text. These attributes can be changed as per requirements.
- Text is used to write tile blocks, write specifications, label parts on a drawing.
- Single line text helps to create one or more lines of text independently. Each line behaves as an independent object and can be moved, formatted or modified to select options on the shortcut menu, user should click right mouse button in the text box.
- Tool Bar: Menu → Annotation panel → Single Line Text 
- Command: TEXT
- Specify the insertion point.
- On pressing ENTER the program creates new text just below the last text object.
- If you press ENTER, the program inserts the new text immediately below the last text object you created, if any.
- Enter a height or click to specify the height of the text. If a specific text height is set in the current text style, this prompt is skipped.
- Enter an angle value or click to specify the rotation angle.
- Enter the text. While typing, the text may be displayed horizontally and at a legible size.

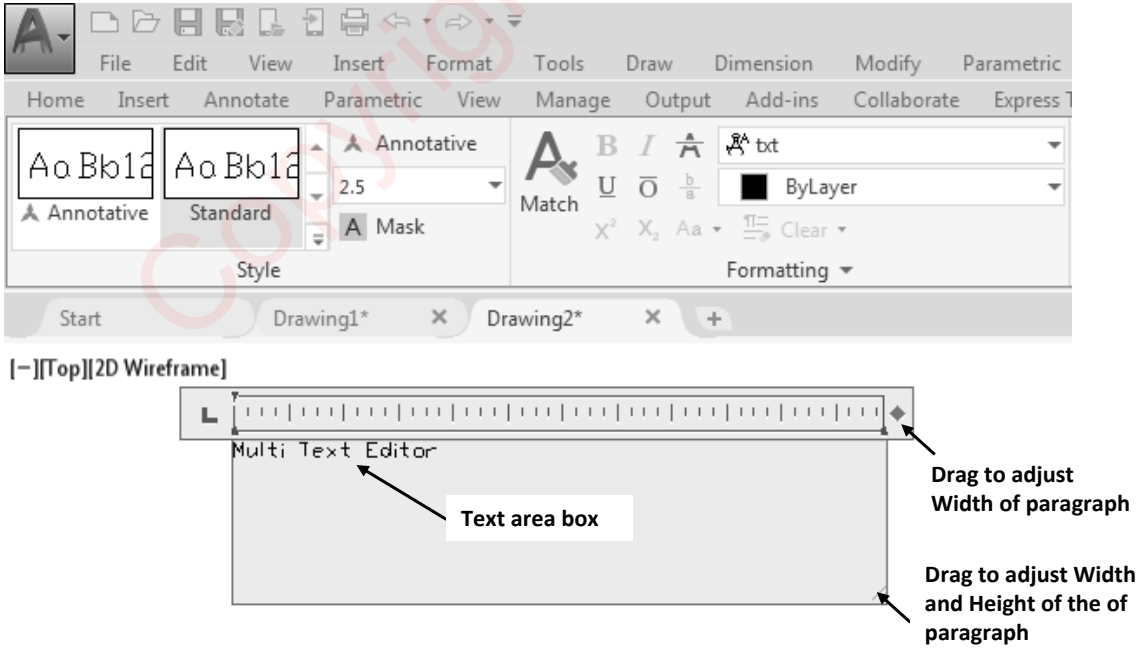


Fig. 6.23: Text Editor contextual tab

- To create another single-line text, do one of the following:
  a. Press ENTER to start another line of text immediately below.
  b. Click a location for the next text object.
- To terminate the command press ENTER on the blank line.

6.11.2 Multiline Text

- Multiline text consists of any number of text lines or paragraphs that fit within a width you specify
- Tool Bar: Menu → Annotation panel→ Multiline Text 
- Command: TEXT
- The width of the bounding box for multiline text objects can be defined by clicking or specifying opposite corners of the box.
- If the ribbon is active, the Text Editor contextual tab displays (Fig.6.23)

6.11.3 Dimensioning

Create linear dimensions with horizontal, vertical and aligned dimension lines. Create baseline or chained dimensions. Place dimension lines at an angle to the extension lines, or make extension lines oblique.

- Tool Bar: Annotate tab → Dimensions panel → Dimension
- Under dimensioning you have options like Linear, Aligned, Angular, Arc Length, Radius, Diameter, Ordinate, Jogged to dimension different entities.
- Command: DIM
- Select a line or specify the first and second extension line origin points.
- Move your pointing device to the desired position and orientation of the dimension.
- Before specifying the dimension line location, you can edit or rotate the text.
- Click to place the dimension line.
- Repeat steps to continue dimensioning or press ENTER to end dimensioning.
- Continued, baseline and ordinate dimensions can be created using the options in this command.
- To dimension isometric drawing use aligned dimensioning then through command line enter
- DIMED (DIMEDIT) command. Select obligue option, select object (dimension you want to convert to isometric style) press ENTER and enter obliquing angel 30^0 and 150^0 as the case may be.

Create a Linear Dimension

- Tool Bar: Annotate tab → Dimensions panel → Linear
- Command: DIMLINEAR
- Specify the first and second extension line origin.
- At the prompt, enter 'r' (Rotated).
- Enter an angle for the dimension line and Click to place the dimension line (Fig.6.24).

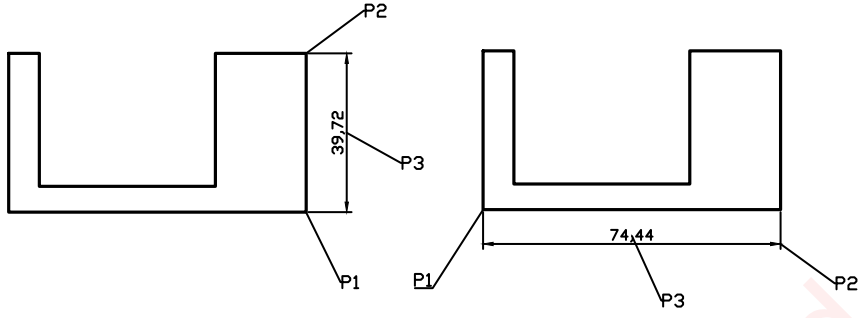
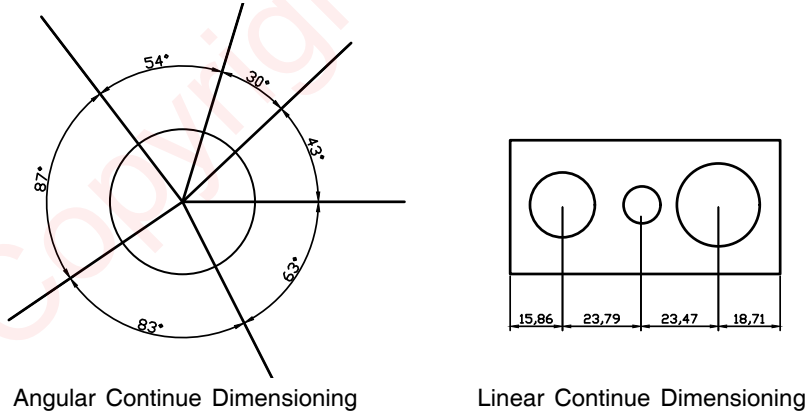


Fig. 6.24: Linear Dimensioning

Create a Continued or Chain Dimension

- Using this option, series of dimensions can be created in-line with an existing dimension. There is no need to specify the continue extension origin when this option is used just after Linear dimensioning command (Fig.6.25).
- Tool Bar: Annotate tab → Dimensions panel → Continue
- Command: DIMCONTINUE
- If prompted, select the dimension to continue. This prompt is skipped if the first extension line origin can be assumed from the origin of the second extension line of the last created linear or angular dimension.
- To specify the next extension line origin object snap should be used.
- To terminate the command press, ENTER twice.



Angular Continue Dimensioning Linear Continue Dimensioning

Fig. 6.25: Continue Dimensioning

Create a Baseline Dimension

- Through Baseline Dimensioning option user can setup a baseline for successive dimensions (Fig.6.26).
- Tool Bar: Annotate tab → Dimensions panel → Baseline
- Command: DIMBASELINE

- If prompted, select the base dimension. This prompt is skipped if the first extension line origin can be assumed from the last created linear or angular dimension.
- Use an object snap to select the second extension line origin, or press ENTER to select any dimension as the base dimension.
- The second dimension line is placed by default at the distance mentioned in the Dimension Style Manager, Lines tab under the Baseline Spacing option.

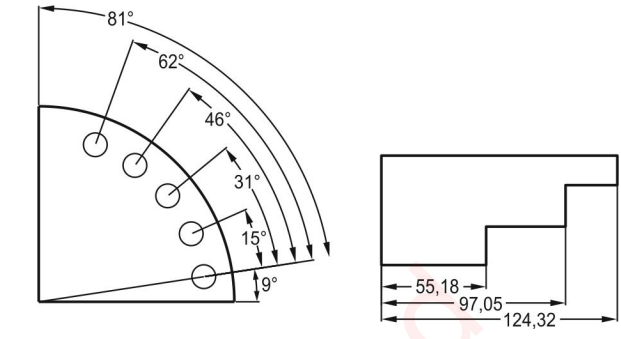


Fig. 6.26: Baseline Dimensioning

- To specify the next extension line origin object snap should be used.
- As required go on selecting extension line origins.
- To terminate the command press ENTER twice.

6.12 LAYERS

A Layer in AutoCAD is like a transparency on which a drawing is created. In a single drawing many layers can be used. This facility is very useful in all engineering disciplines e.g. in Mechanical Engineering drawings, component outline may be on one layer, hatching or sectioned views of the component may be on another layer, dimensioning may be on another layer, text and part list many be on next layer and so on.

Through this command you can create a new layer, delete a layer and rename layers, change their properties, set property overrides in layout viewports and add layer descriptions (Fig.6.27).

How to use Layers

- First choose a layer and then start drawing on it. It is advisable to draw similar nature objects on same layer. For example, in civil/architectural construction drawing, create a layer with name 'Wall', now make it current and then draw the floor plan on it. Similarly create another layer with name 'Electrical', make it current and draw the electrical objects/ wire diagram on it. Finally make third layer, name it Plumbing', make it current and draw hot/cold water pipe line layout, pipe fittings, bathroom fittings etc. on it.
- In a drawing once layers are made then they can be controlled individually like visibility of objects, properties assignment such as color and linetype, text and dimensioning style, making it visible/invisible, editable/non editable, printable/non printable etc.
- When an object is drawn on a layer then by default it takes the property of that layer but if required then this can be override and properties of an individual object can be changed. Suppose, the colour of a layer is red and when any object is drawn on this layer it will automatically of red colour as the object's color property is set to BYLAYER. But if the user wants to change the object colour to yellow then that can be done.
- Frozen option makes a layer invisible and Thaw makes it visible.

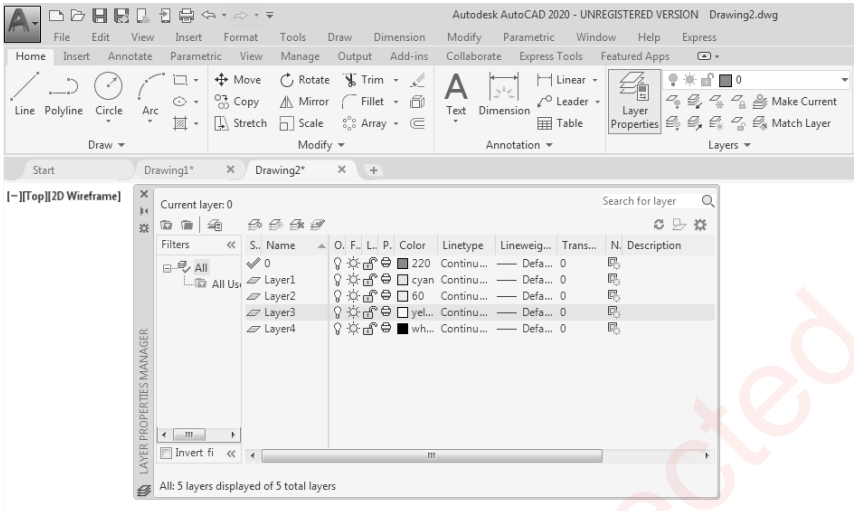


Fig. 6.27: Layer Property Manager

Create a Layer

- Tool Bar: Home → Quick Access Toolbar → Layer 
- Command: LAYER
- In the layer property manager layers panel will be displayed.
- Click New Layer icon 
- This will add a layer name space in the layer list.
- Type a new layer name in the highlighted space.
- For complex drawings with many layers, enter descriptive text in the Description column.
- Specify the settings and default properties of the new layer by clicking in each column.

Select a Layer

- In the layer property manager layers panel will be displayed.
- Select the drop-down arrow. Highlight the desired layer and press the left mouse button.
- The selected layer becomes the 'current' layer. All objects will be created on this layer until you select a different layer.

Rename a Layer

- In the Layer Properties Manager, click to select a layer.
- Click the layer name or press F2.
- Enter a new name.

Remove a Layer

- In the Layer Properties Manager, click to select a layer.
- Click Delete Layer 
- The following layers cannot be deleted:

- Layers 0 and Defpoints
- Layers containing objects, including objects in block definitions
- The current layer
- Layers used in an external reference
- To remove all unused layers, use PURGE 

Status of a Layer

The following controls can be accessed using the Layer drop-down arrow.

- On or Off: A layer is visible if it is on otherwise invisible. User can edit or plot only those layers which are on.
- Freeze or Thaw: A frozen layer is invisible while a thawed layer is visible hence they are similar to on and off with extra features like at the time of zooming in and out the frozen layers are not regenerated which saves computational time; thawed layers can be edited or plotted; objects of frozen layer cannot be erased.
- Lock or Unlock: Locked layers are visible hence will be plotted but cannot be edited.

Modifying a Layer

Using layer properties manager you can modify a layer in following ways:

- Changing name of the Layer
- Changing Layer Colour
- Changing Line weight
- Changing Line type
- Making it editable/non-editable, printable/non-printable and visible/invisible

6.13 METHODS OF SELECTING OBJECTS

Many AutoCAD commands prompt you to "select objects". This means select the objects that you want the command to effect. There are 3 methods.

Method 1

Pick is very easy and should be used if user wants to select 1 or few objects.

To select object(s) using this method just place the mouse cursor on the object without pressing it. This will highlight the object and indicates that the object is ready for selection. If it is the same object you intended to select click the left mouse button to finally select it otherwise, shift the mouse cursor towards left/right or top/bottom to highlight the desired object. This appearance change is called "Rollover Highlighting". To select multiple objects just hover on the next object and click left mouse button and so on.

Method 2

Crossing. Select the object(s) by placing the mouse cursor on the top and towards right of the object(s), clicking the left mouse button and releasing it (do not hold the button in pressed condition) and then dragging it to diagonally opposite corner viz. below and towards left of the object(s) and again clicking left mouse button. A window with green shade and dotted boundary will be formed and anything inside this and crossed by this window will get selected.

Window. In this case just do opposite to previous case, that is, place the mouse cursor at the top and towards left of the object(s), click the left mouse button and release it (do not hold the button in pressed condition) and then drag it to diagonally opposite corner viz. down and towards right of the object(s) and again left click the mouse. A window with blue shade and solid boundary will be formed and anything inside this window get selected.

Method 3

Lasso Selection, is a little more difficult than Window Crossing Selection.

Place your cursor in the area up and to the right of the objects that you want to select (P1) then press and hold the left mouse button (Do not release the mouse button). Then move the cursor in an anticlockwise direction until you have crossed the objects you want to select (P2) then release the left mouse button. Only the objects that the Lasso Window crosses will be selected.

6.14 UNDO AND REDO

The Undo and Redo tools allow you to cancel out or to do again the previous activities in the drawing. Suppose, if an object is deleted by mistake, then the previous "erase" command can be undo and the object will reappear. Till closing of the drawing many time undo/redo can be attempted.

6.15 MODEL AND PAPER LAYOUT

- There are two distinct working environments, called "model space" and "paper space," in which you can work with objects in a drawing.
  a. By default, you start working in a limitless 3D drawing area called model space (In the book we are covering only 2D entities). You begin by deciding whether one unit represents one millimeter, one centimeter, one inch, one foot, or whatever unit is most convenient. You then draw at 1:1 scale.
  b. To prepare your drawing for printing, switch to paper space. Here you can set up different layouts with title blocks and notes; and on each layout, you create layout viewports that display different views of model space. In the layout viewports, you scale the model space views relative to paper space. One unit in paper space represents the actual distance on a sheet of paper, either in millimeters or inches, depending on how you configure your page setup.
- Model space is accessible from the Model tab and paper space is accessible from the layout tabs.
- To activate Model space user has to click Model tab places in the lower-left corner of the main window (Fig.6.28).
- Depending on whether you are in Model Space or Paper Space, the tools in status bar change.
- If the Model and Layout tabs are hidden, set the LAYOUTTAB system variable to 1.
- By default, a Model tab and several named Layout tabs are displayed at the bottom-left corner of the drawing area. Click either MODEL or PAPER on the status bar at the bottom of the application window.

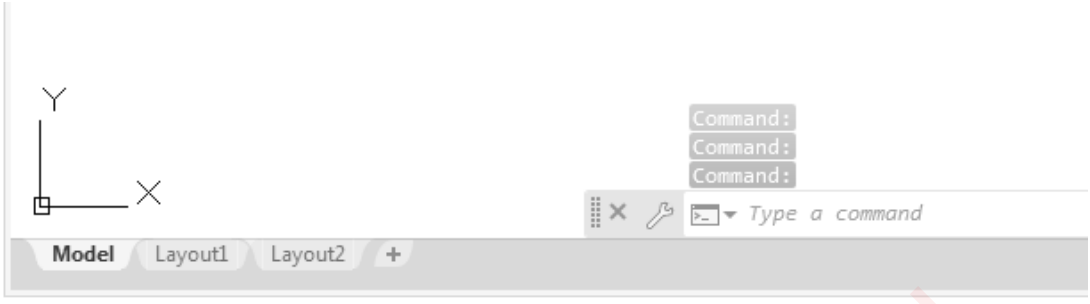


Fig. 6.28: Model and Paper Space

6.16 PLOTTING OF DRAWINGS

- Plot command gives instructions to a plotter/printer/file for printing of a drawing.
- To print a drawing: Perform Zoom / All to center the drawing within the plot area.
- Tool Bar: Quick Access Toolbar → Plot
- Command: PLOT
- This opens a dialogue box having two pages, namely 'Plot device and Plot Settings'.
- The plotter configuration or its equivalent has to be selected in the Plot Device page and the following options to be set in the Plot setting page:
  - Page Size: A4 (210x297 mm) or A3 or etc. (As per the specifications of Printer)
  - Units: mm
  - Drawing Orientation: Portrait / Landscape
  - Plot Area: In the Plot area tile, below 'What to plot: combo box', click the arrow. Four options are displayed: Display, Extents, Limits and Window. The meaning of each is as under:

a. Display option plots whatever is displayed on screen.

b. Extents option chooses the entire drawing even if it is outside the specified limits.

c. Limits option plots the drawing falling within specified limits set by LIMIT command.

d. Window option allows choosing a specified area within a window on the screen. The Window option is usually chosen. The dialog box disappears and you are guided to select the object(s) which you want to plot. Select the drawings by making a window across the object(s) by clicking on two diagonal corners. The dialog box appears again.

  - Plot Scale: In the Plot scale tile, choose a scale. Alternately, you can also click the check box Fit to paper if you do not wish the drawing to be plotted on a particular scale. AutoCAD recalculates the scale and displays its new value. If you select Fit to paper option, Plot scale option will remain gray and you cannot choose.
  - Centering the Plot: In the 'Plot offset' (origin set to printable area) tile, click in 'Center the plot' check box. The offsets in X and Y directions automatically are displayed. If you do not want the plot in the center, you can define your own values in the X: and Y: offset text boxes.
  - Drawing Orientation: Click the arrow in the circle on the right bottom corner of 'Plot-Model dialog box'. The dialog box opens more options. In Drawing orientation tile,

click the radio button Portrait or Landscape. Portrait option keeps longer side of the drawing vertical while Landscape keeps longer side horizontal. Click Plot upside-Down check box if you want to invert the drawing by 180°

- Previewing and Plotting: After all above settings and before plotting, preview the drawing. At the bottom of the dialog box, click the Preview button. If satisfied, press ENTER key or OK button in the dialog box to get a print of the selected drawing, If not satisfied, change the settings.

6.17 EXITING AUTOCAD

To close/exit AutoCAD software or to close file(s):

- Save all open drawings.
- Exit using one of the following:
  a. Click Application Menu → Click 

  b. Close file using right top corner cross symbol 
- If you want to close only current drawing then click application menu, go to close option and select 'current drawing'. If you select 'all drawing' option then all the currently open drawings will be closed. In both the cases 'do you want to save the changes ?' warning message will be displayed to remind to save the changes. The option Yes, No or Cancel has to be selected.

SOLVED PROBLEMS (WITH INCREASING DIFFICULTY LEVEL)

Example 6.4 *Draw following figure in AutoCAD.*

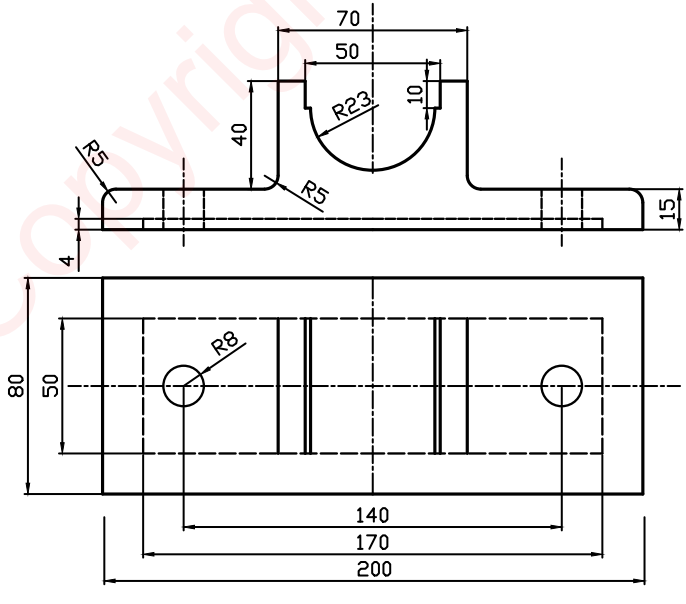


Fig. 6.29: Drawing for Example 6.4

Solution

1. Open a new file and create three layers. Let the names be Layer 0, 1, 2 and 3. Specify the style for each layer.
2. On the Layers toolbar, click the arrow in the Settings combo box. Click on Layer 0.
3. Draw the outline using RECTANG, LINE, CIRCLE and ARC commands.
4. In the right panel of Layer Property Manager dialog box click on Layer 2.
5. Using OFFSET command draw the inner rectangle of top view. Draw dashed lines of Front view also. Notice that the linetype automatically is dashed and is in blue color.
6. In the right panel of Layer Property Manager dialog box click on Layer 3. Draw the center lines in both the views. Notice that the linetype automatically is Center and is in cyan color.
7. In the right panel of Layer Property Manager dialog box click on Layer 1.
8. Click on the DIMLINEAR icon. AutoCAD prompts for first extension line origin. Click on left corner of the bottom line of top view. Then specify second extension point as the other right extreme corner of the same line. Now move the cursor where you want the dimension to appear and click there. The dimension 200 appears in red color.
9. Repeat Step 8 for all horizontal and vertical lines.
10. Click on DIMDIAMETER icon. AutoCAD prompts to select an arc or circle. Click on the circle in the top view. Now move the cursor where you want the dimension and click.
11. Click on DIMRADIUS icon. AutoCAD prompts to select an arc or circle. Click on the arc in the front view. Move the cursor where you want the dimension and click.
12. Repeat Step 11 to dimension the fillets.
13. Save the file with name Ex6.4

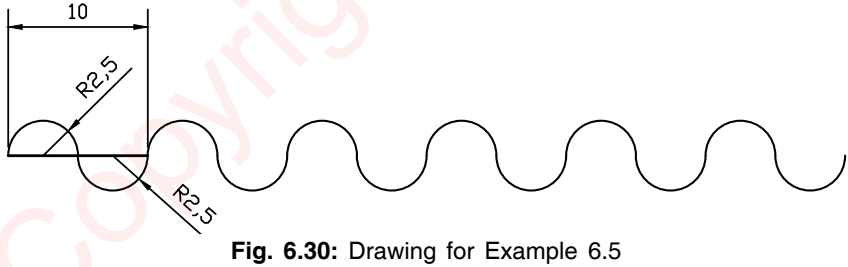


Fig. 6.30: Drawing for Example 6.5

Example 6.5 *Draw profile of knuckle thread with a pitch of 20 mm as shown in Fig. Show at least 6 threads.*

Solution

1. Using ARC command draw only first 2 arcs of 2.5 mm radius anywhere on the screen.
2. Use ARRAY command. In the Array dialog box, click Rectangular option.
3. Click the Select objects button. The dialog box disappears and the drawing is displayed. Click on both the arcs.
4. Specify number for Rows as 1 and columns as 6 in the text boxes.
5. Specify Colum offset as 10. Value of Row offset has no effect here as the number of rows is only 1. Let the Angle of array be 0.

6. Click Preview button on lower right side of the dialog box and then press ENTER Key.
7. Five more thread profiles are copied. Total will be six including the original object.
8. Save the file with name Ex 6.5

Example 6.6 *Draw six squares at equal angles between them of 12 mm sides on a pitch circle diameter as 120 mm as shown in following figure.*

Solution

1. Use LINE command to draw horizontal line O-P of 60 mm length.
2. Use RECTANG command to draw square. For one corner use From object snap. Click at point P for base point. Specify Offset distance as (@)-6,-6 (Point A from point P). Press F3 to put object snap Off. For the other corner C specify relative coordinates as @12,12. Press F3 again to put snap ON.
3. Use ARRAY command. In the Array dialog box, click Polar option.
4. Click the Select objects button. Click on the square.
5. Click a button against Center point Click on the left end of line at the point O as center of array.
6. Choose the option "Total number of items and angle to fill".
7. Specify total Number of items as 6.
8. Specify Angle to fill as 360°.

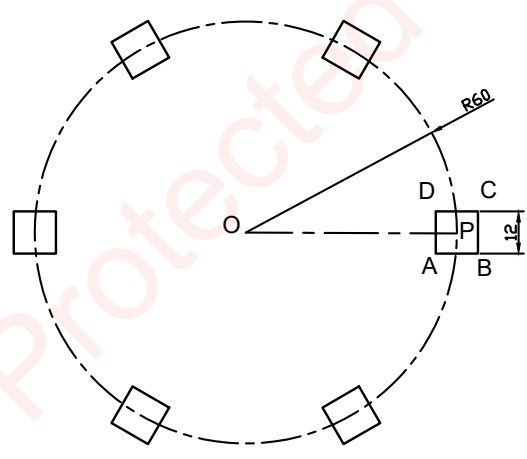


Fig. 6.31: Drawing for Example 6.6

9. Click the check box in Rotate items as they are copied. A tick sign appears. If it is already there, ignore this step.
10. Click Preview button on lower right side of the dialog box and then press ENTER Key.
11. Five more squares are copied around the center point. Total will be six including the original object.
12. Repeat steps 3 to 9 but at step 8 let the Rotate items as they are copied check box remain unchecked. See the effect that all the squares remain horizontal.
13. Save the file with name Ex 6.6

UNIT SUMMARY

In this unit you have learned about:

1. Commands under Draw and Modify Tool bars.
2. Simple, direct distance entry, absolute & relative coordinate system.
3. Circle (centre-radius-diameter, 2point, 3point, tan-tan-tan-radius).
4. Ellipse (center-axis-end) & Ellipse arc.

5. Rectangle, Dimension style manager.
6. Polygon (Inscribed in circle& circumscribed about circle& edge)
7. Polyline & Pedit.
8. Arc, Spline, Solid, Donut.
9. Fillet, Chamfer, Rotate Scale, Stretch, and Join.
10. Move, Copy, Zoom, Pan, Offset, Extend, Trim.
11. Mirror, Mirror text, Break, Trim, Erase, Undo, Redo.
12. Function keys and setting.
13. Text (T -Multiple & Dt- single line text & change).
14. Array (rectangular & polar).
15. Block (making & inserting, Attributed definition, dynamic block editor).
16. Point, mpoint, ddptype, Divide, Measure, hatching, gradient, editing, linetype, Ltscale.
17. Template, setting of units, Explode.
18. Layer (making layer & setting), layer tools properly, grip (options), status.

EXERCISES

A. Multiple Choice Questions

6.1 1. Commands given in AutoCAD appear
(a) in the drawing area (b) on the toolbar of commands
(c) at the command line (d) at the status line

6.2 Ortho, Snap, Grid, etc. are displayed on the
(a) object snap toolbar (b) view toolbar
(c) right hand side menu (d) status bar

6.3 Snap is turned on or off using function key
(a) F4 (b) F6
(c) F7 (d) F9

6.4 Function key F8 is used to toggle
(a) Coordinate display (b) Grid display (c) Status bar (d) Ortho

6.5 Object snap is used to increase
(a) speed, accuracy and ease of drafting (b) speed of drawing
(c) accuracy while drafting (d) memory utilization

6.6 In 2 point option of CIRCLE command, the two points to be defined are
(a) center and a point on periphery (b) two diametrically opposite points on periphery
(c) any two points on periphery (d) two diametrically opposite points on a horizontal line

6.7 The Direction option in ARC command is the direction of a
(a) tangential line at the end point

(b) tangential line at the start point
(c) radial line between the center and start point
(d) radial line between the center point and end point

6.8 Rotation option in ELLIPSE command draws an ellipse whose
(a) major axis is oriented by the specified rotation angle
(b) minor axis is oriented by the specified rotation angle
(c) major axis remains in the direction specified by the end points
(d) major/minor axis ratio becomes equal to sine of the rotation angle

6.9 COPY command allows to make
(a) only one copy of a single entity at a time
(b) many copies of many objects
(c) one or many copies of one or many objects
(d) many copies of only one entity

6.10 MIRROR command is used to make
(a) a copy of an object
(b) a mirror copy of an object with or without original object
(c) a mirror copy of an object only about X axis
(d) a mirror copy of an object without original object

6.11 OFFSET command at a time allows offsetting an object
(a) only on one side of object at a fixed distance
(b) on both sides of object at a fixed distance
(c) only on one side of object at a variable distance
(d) on both sides of object at a variable distance

6.12 Rotation angle in TEXT command
(a) inclines the lines of the text by a specified angle
(b) inclines each character by that angle but the line remains horizontal
(c) rotates the text in the form of circular arc
(d) displays the text upside down

6.13 The command to edit text is
(a) TEXT
(b) DDEDIT
(c) ETEXT
(d) MTEXT

6.14 Which of the following options will print the complete drawing
(a) Display
(b) Extents
(c) Limits
(d) Window

6.15 The size of the printout
(a) remains the same as in the drawing
(b) can be increased only while plotting
(c) can be decreased only while plotting
(d) be increased or decreased while plotting

6.16 Insertion base point of a block
(a) should be on the block
(b) can be anywhere on the block or away from the block
(c) has to be at its base
(d) has to be on a flat surface

6.17 The gap between two adjacent objects in a rectangular array
(a) has to be more than the size of object
(b) can be less or more than the size of object
(c) should be at least 1.5 times the object
(d) depends upon many parameters

6.18 ARRAY with polar option is applicable for objects
(a) which are drawn using polar coordinates
(b) where at least one entity of object is drawn using polar coordinates
(c) where at least two entities of object are drawn using polar coordinates
(d) drawn by any method of coordinates system

6.19 If a block is to be used in another drawing file, the command to save the block is
(a) INSERT (b) BLOCK (c) WBLOCK (d) MINSERT

6.20 WBLOCK command allows to create a file from
(a) a block (b) selected objects (c) complete drawing file (d) any one of above

Answers of Multiple Choice Questions

6.1 (c); 6.2 (d); 6.3 (d); 6.4 (d); 6.5 (a); 6.6 (b); 6.7 (b); 6.8 (c); 6.9 (c); 6.10 (b); 6.11 (a); 6.12 (a); 6.13 (b); 6.14 (b); 6.15 (d); 6.16 (b); 6.17 (b); 6.18 (d); 6.19 (c); 6.20 (d)

B. Subjective Questions

6.1 Describe the use of LIMITS command.
6.2 Describe the use of Snap and Grid.
6.3 Explain the use of Object Snap. Discuss the different modes of operation.
6.4 List the icons on the Draw Toolbar and discuss any five icons for drawing entities.
6.5 Give the command sequence to make an equilateral triangle using three lines.
6.6 Write any 3 options to draw a circle.
6.7 Describe the method to draw an arc.
6.8 Write steps to draw a polygon with given center and an edge length.
6.9 Explain the difference between Window and Crossing window while selecting an object.
6.10 Explain the difference between COPY and MOVE command? Explain by examples.
6.11 Explain the use of EXPLODE command.
6.12 Explain the utility of SCALE command. Can you have different scales in X and Y directions while using this command?
6.13 Write steps to use a CHAMFER command to specify different chamfer distances for each side command with zero radius.
6.14 Describe the use of FILLET command. How do you change the fillet radius. What is the effect of using this command with zero radius?

PRACTICALS

As per curriculum the practicals related to Unit-6 are:

1. Draw basic 2D entities like: Rectangle, Rhombus, Polygon using AutoCAD (Print out should be a part of progressive assessment).
2. Draw basic 2D entities like: Circles and Arcs using AutoCAD (Printout should be a part of progressive assessment).
3. Draw basic 2D entities like: Circular and rectangular array using AutoCAD (Printout should be a part of progressive assessment).
4. Draw blocks of 2D entities comprises of Rectangle, Rhombus, Polygon, Circles, Arcs, circular and rectangular array, blocks using AutoCAD (Print out should be a part of progressive assessment).
5. Draw basic branch specific components in 2D using AutoCAD (Print out should be a part of term work).
6. Draw complex branch specific components in 2D using AutoCAD (Print should be a part of progressive assessment).

Practical-1 Draw Rectangle, Rhombus, Polygon using AutoCAD

Practical statement

Draw basic 2D entities like Rectangle, Rhombus, Polygon using AutoCAD.

Practical significance

Usually complex digital drawings are created by combining and modifying several different basic primitive shapes, such as lines, circles, arcs, polygons etc.

Relevant theory

Refer Section 6.9.2 (Draw Line)
Refer Section 6.9.6 (Draw Polygon)
Refer Section 6.9.7 (Draw Rectangle)
Refer Section 6.12 (Text and Dimension)

Practical outcomes (PrO)

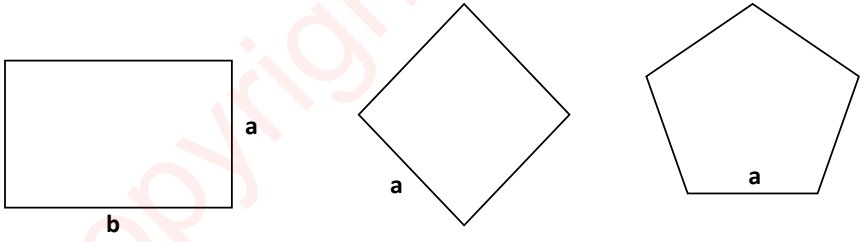
The practical outcomes are derived from the curriculum of this course:

PrO1: Use Draw tool bar commands of AutoCAD to create basic 2D entities.

PrO2: Dimension the given 2D entity using AutoCAD

PrO3: Write text on the given 2D entity using AutoCAD

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)



a=100 mm and b=300 mm

Resources Required

Sr. No	Suggested Resources required Machines/Tools/ Instruments with vital specifications	Qty	Actual Resources used Machines/Tools/Instruments with broad specifications (to be filled by the student)		Remarks (if any)
1	AutoCAD 2020 software Student Edition	1 User	AutoCAD 2020 software student edition	1 User	Or any Licensed version of AutoCAD

2	Computer System PC/ Laptop loaded with Window and Anti-Virus software	1			As per section 5.4.1

Precautions

1. Save the drawing at regular interval to avoid data loss.
2. When not in use shut down the computer system.
3. Use Antivirus software in the computer system

Suggested Procedure

Preliminary Steps:

- It is presumed in this book that AutoCAD software is installed on the computer and the user has a basic knowledge of computers.
- Make the computer system ON.
- Click the AutoCAD icon on the desktop or
- Click the START button at the lower left corner of the screen, followed by PROGRAM and then clicking AutoCAD 2020 in the list. Wait for few seconds to get the software completely launched and till the main AutoCAD software window is displayed on the screen.
- Open New drawing with institute specific template.
- Set limits to 230 mm x 310 mm (A4 size).
- Set units to metric.
- Make grid ON and Snap ON with snap setting at 5mm.

To draw Rectangle in AutoCAD:

1. Tool Bar: Menu → Draw → Rectangle 
2. Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: Enter an option or specify a point (P1) anywhere on the screen.
3. Specify other corner point : Specify a point (P2) as per given sides 'a' and 'b'.
4. Dimensioning of the rectangle:
5. Tool Bar: Annotate tab → Dimensions panel → Linear 
6. Specify the first and second extension line origin.
7. Click to place the dimension line on the width of the rectangle.
8. Repeat step 10 to 12 to place dimension on length side of the rectangle.
9. Save the drawing as: Pr-12.

To draw Rhombus in AutoCAD:

1. Tool Bar: Menu → Draw → Rectangle 
2. Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: Enter an option or specify a point (P1) anywhere on the screen with sufficient distance from the previously drawn entity.

3. Specify other corner point : Specify a point (P2) just vertically below then P1 such that distance between P1 and P2 is a√2.
4. The output is a Rhombus.
5. Dimensioning the Rhombus:
6. Tool Bar: Annotate tab → Dimensions panel → Aligned
7. Specify the first and second extension line origin.
8. Click to place the dimension line on any one side of the Rhombus.
9. Save the drawing..

To draw Polygon in AutoCAD: Say a Pentagon:

1. Tool Bar: Menu → Draw → Polygon
2. Click first point on the screen with sufficient distance from the previously drawn entities.
3. Enter number of sides <current> : 100 mm and press. ENTER.
4. Specify center of polygon or [Edge]: edge and press ENTER.
5. Specify edge length as 50 and press ENTER.
6. Pentagon completed
7. Dimensioning of the Pentagon.
8. Tool Bar: Annotate tab → Dimensions panel → Aligned
9. Specify the first and second extension line origin.
10. Click to place the dimension line on any one side of the Pentagon.
11. Write the text on all the three entities using the procedure outlined in 6.12.1
12. Save the drawing.

Observations

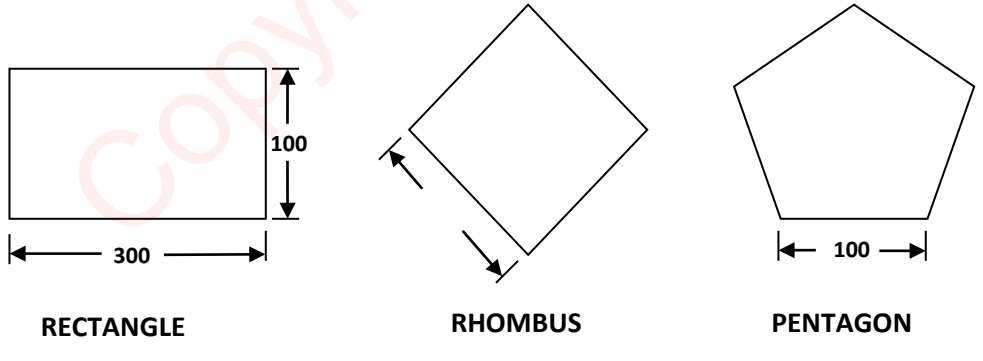


Figure: AutoCAD Drawings of Rectangle, Rhombus, Polygon

Practical related questions

1. Can we draw these figures using line command ? How?
2. If circumscribed circle radius 'r' is given how a pentagon of 'a' side will be drawn?

Suggested Learning Resources

Refer Unit-6

Suggested Assessment Scheme

(to be filled by teacher)

The given performance indicators should serve as a guideline for assessment regarding process and product related marks.

Performance indicators		Weightage/ Marks	Marks Awarded
Process related: 10 Marks* (66%)			
1	Setting parameters for new drawing and using institute specific template	2	
2	Type of command (s) used to create the desired drawing	2	
3	Ease of using keyboard and mouse to create the drawing	2	
4	Time taken	2	
5	Response to practical viva questions	2	
Product related: 5 Marks* (33%)			
6	Dimensional accuracy of the output	3	
7	Dimensioning and text writing	2	
	Total	**100%**	

* Marks and percentage weightages for product and process assessment will be decided by the teacher.

Name of the Student: …….. **Marks Awarded**			**Signature of Teacher with date**
Process Related	Product Related	Total	

Practical-2 Draw Circles and Arcs using AutoCAD

Practical statement

Draw basic 2D entities like Circles and Arcs using AutoCAD.

Practical significance

Usually complex digital drawings are created by combining and modifying several different basic primitive shapes, such as lines, circles, arcs, polygons etc.

Relevant theory

Refer Section 6.9.8 (Draw Circle)

Refer Section 6.9.9 (Draw Arc)

Refer Section 6.12 (Text and Dimension)

Practical outcomes (PrO)

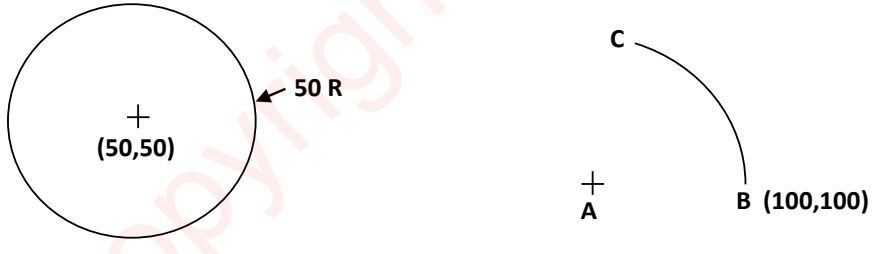
The practical outcomes are derived from the curriculum of this course:

PrO1: Use Draw tool bar commands of AutoCAD to create basic 2D entities.

PrO2: Dimension the given 2D entity using AutoCAD

PrO3: Write text on the given 2D entity using AutoCAD

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)



All dimensions are in mm and in case of Arc the included angle ABC is 75°

Resources required

Same as mentioned in Practical-1 of Unit-6.

Precautions

Same as mentioned in Practical-1 of Unit-6.

Suggested procedure

Preliminary Steps: *Same as mentioned in Practical-1 of Unit-6.*

To draw Circle in AutoCAD: with Center-Radius option

1. Follow the procedure for drawing a circle with center and radius option as detailed out in

section 6.9.8 with coordinates of center point as (50, 50) and radius 50 mm given in this practical.

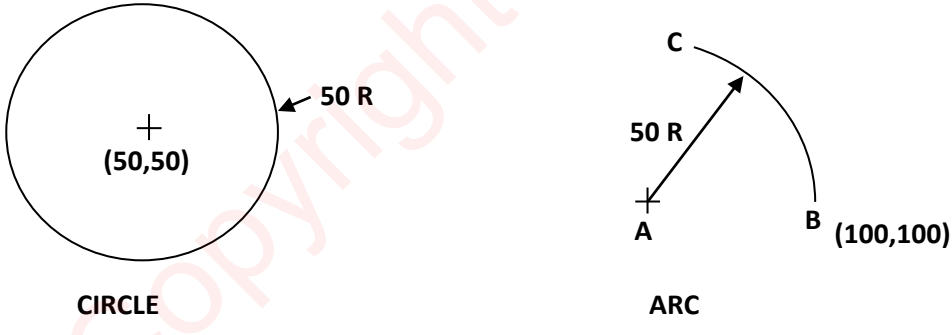
2. Dimensioning the Circle:
3. Tool Bar: Annotate tab → Dimensions panel → linear → Radius
4. Select the circle and positions the dimension.
5. Save the drawing as: Pr-13.

To draw Arc in AutoCAD: with Center-Start-Angle option

1. Tool Bar: Menu → Draw → Arc 

2. Specify center point of the arc: specify point at (0,100)
3. Specify start point of the Arc: specify point at (100,100)
4. Specify included angle: 75.
5. Arc completed
6. Dimensioning of the Arc:
7. Tool Bar: Annotate tab → Dimensions panel → Radius
8. Select the Arc and positions the dimension.
9. Write the text on the two entities using the procedure outlined in 6.12.1
10. Save the drawing.

Observations



AutoCAD Drawings of Circle and Arc

Practical related questions

1. Can we draw given Arc using Center-Start-Radius option? How?
2. What are the other ways of drawing a circle in AutoCAD?

Suggested learning resources

Refer Unit-6

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-6.

Practical-3 Draw Circular and Rectangular Array using AutoCAD

Practical statement

Draw basic 2D entities like: Circular and rectangular array using AutoCAD.

Practical significance

Polar and Rectangular arrays are required to develop complex digital drawings of components with repeated pattern like placement of rivets in riveted joint; circular/square/rectangular perforations in computer cabinets/motor cases; different treads on screw/nut/not/screw jack/lead screw etc.; window patterns; holes on the periphery of circular plates/flanges/ buttons on key board, regular shaped pattern on various house hold/industrial components.

Relevant theory

Refer Section 6.10.5 (Polar and Rectangular Arrays) Refer Section 6.12 (Text and Dimension)

Practical outcomes (PrO)

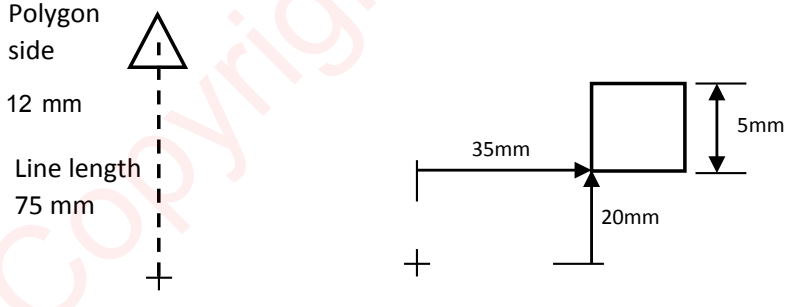
The practical outcomes are derived from the curriculum of this course:

PrO1: Use Draw tool bar commands of AutoCAD to create Polar and Rectangular arrays.

PrO2: Dimension the given 2D entity using AutoCAD

PrO3: Write text on the given 2D entity using AutoCAD

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)



Polar array: Fill angle 360^0; Number of items in array: 12; Included angle: 75^0
Rectangular array: Column: 5; Column distance: 20mm; Row:4; Row distance: 15mm

Resources required

Same as mentioned in Practical-1 of Unit-6.

Precautions

Same as mentioned in Practical-1 of Unit-6.

Suggested procedure

Preliminary Steps: *Same as mentioned in Practical-1 of Unit-6.*

To draw Polar array:

Draw a line of 75 mm length using line command:

1. Click line command.
2. Specify first point: 100,150.
3. Click next point: @0,75 and pres Esc from keyboard.
4. Line completed.

Draw a polygon with radius 12mm and 3 sides:

1. Enter number of sides:3
2. Specify center point or [edge]:top end point of previously drawn line.
3. Specify radius of circumcircle:12
4. Polygon completed.

Polar array:

1. Select the Polar Array command: Home Tab → Modify Panel → Array
2. Select Polar Array.
3. Select Objects: select the object to be arrayed (Polygon).
4. Select Objects: select more objects or <Enter> to stop.
5. Specify center point of array or [Base point / Axis of Rotation]: select lower end point of the line.
6. The Array Creation Tab appears and the default polar array with 6 items is displayed.
7. Enter items: 12
8. Enter Fill: 360
9. Press <Enter> to display the selections.
10. Select Close Array if display is correct.
11. Polar Array completed.
12. Dimensioning the Array: side of the polygon, length of the line and included angle between two polygons.
13. Save the drawing as: Pr-14a.

Repeat the same procedure with different values:

1. Make a copy of line and polygon and place it at reasonable distance from previous polar array on the screen.
2. Repeat the steps 9 to 14
3. Enter items: 6
4. Enter Between: 45
5. Enter Fill: 360

6. Select Close Array if display is correct.
7. New Polar array completed.
8. Save the drawing.

To draw Rectangular array:

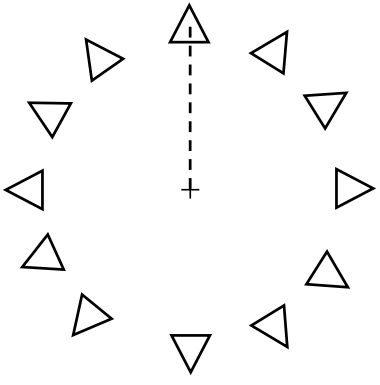
Draw square of size 5mm side

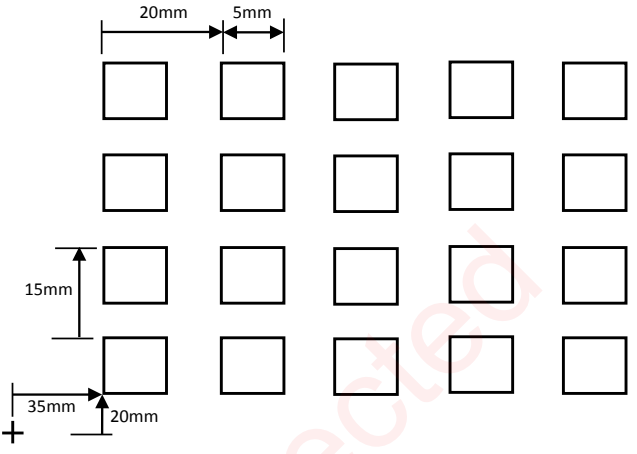
1. Start a new drawing with same parameters.
2. Click Rectangle command
3. Specify first point: specify point at (35,20)
4. Specify second point[Area/Dimensions/Rotations]: enter d press ENTER
5. Specify length for rectangle: 5
6. Specify width for rectangle: 5
7. Rectangle completed

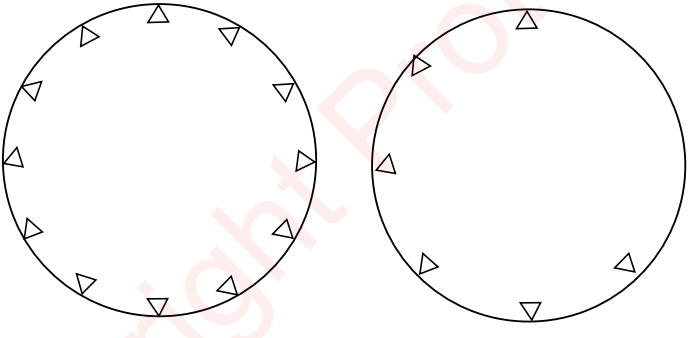
Rectangular array:

1. Select the Rectangular Array command: Home Tab → Modify Panel → Array
2. Select Rectangular Array.
3. Select Objects: select the object to be arrayed (Square) and press ENTER.
4. The Array Creation Tab appears with a 3 row and 4 column default rectangular array.
5. Enter Columns: 5
6. Between distance: 20
7. Enter Rows: 4
8. Between distance: 15
9. Press <Enter> to display the selections.
10. Rectangular Array completed.
11. Dimensioning the Array: side of the original square, Column distance, Row distance, horizontal and vertical distance of one corner of the original square from UCS.
12. Save the drawing as P-14b.

Observations







Practical related questions

1. Can we select more than 1 entity for array? How?
2. Can we edit array after its creation? How?

Suggested learning resources

Refer Unit-6

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-6.

Practical-4 Draw Blocks of 2D Entities using AutoCAD

Practical statement

Draw blocks of 2D entities comprises of Rectangle, Rhombus, Polygon, Circles, Arcs, circular and rectangular array, blocks using AutoCAD

Practical significance

Blocks are used to use same entity or combination of 2D entities several times. Generally, in cases where same subassembly or components are used again and again with same or different orientation/ scale then those entity or group entities are converted into block and saved with a name/identification so that whenever required can be inserted to save time and efforts. For example, various types of nut-bolts, bearings, gears, bets, sprockets, screws, chairs, doors, windows, resistors, capacitors, battery etc.

Relevant theory

Refer Section 6.9 and 6.10 (Draw and Modify Tool bars)

Refer Section 6.9.13 (Blocks)

Refer Section 6.10.5 (Polar and Rectangular Arrays)

Refer Section 6.12 (Text and Dimension)

Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

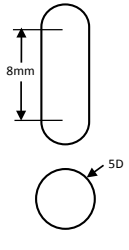
PrO1: Use Draw tool bar commands of AutoCAD to create Polar and Rectangular arrays.

PrO2: Dimension the given 2D entity using AutoCAD

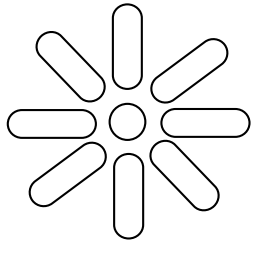
PrO3: Write text on the given 2D entity using AutoCAD

Practical setup (Drawing/Sketch/Circuit Diagram/Work Situation)

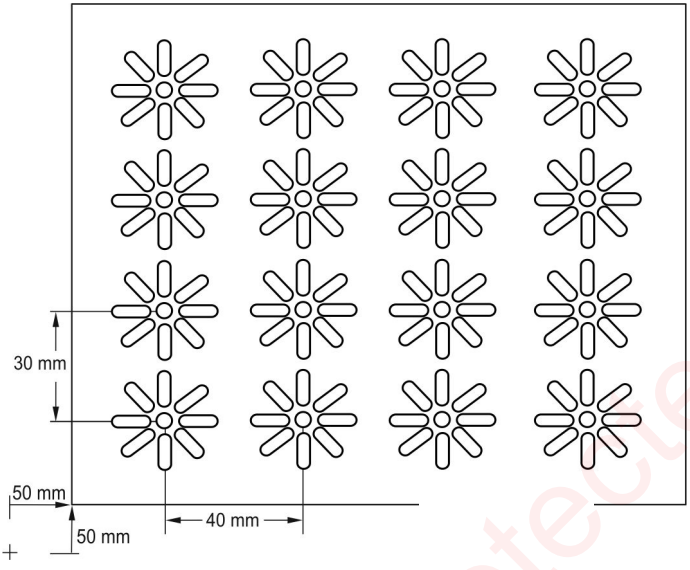
- Draw objects as shown in Fig.(A)
- Draw a Polar Array to create a Block as depicted in Fig.(B)
- Polar array parameters: Fill angle 3600 and number of items in array is 8.
- Draw a Rectangular array as depicted in Fig.(B)
- Rectangular array parameters: Column: 4; Column distance: 40mm; Row:4; Row distance: 30mm



(A) OBJECTS



(B) POLAR ARRAY CONVERTED TO BLOCK



(C) RECTANGULAR ARRAY

Resources required

Same as mentioned in Practical-1 of Unit-6.

Precautions

Same as mentioned in Practical-1 of Unit-6.

Suggested procedure

Preliminary Steps: *Same as mentioned in Practical-1 of Unit-6.*

Draw the Circle

1. Click circle command.
2. Specify center: 80, 80.
3. Enter radius: 2.5
4. Circle completed.

Draw the Rectangle

1. Click Rectangle command
2. Specify first point: specify point at (50,50)
3. Specify second point[Area/Dimensions/Rotations]: enter d press ENTER
4. Specify length for rectangle: 180
5. Specify width for rectangle: 150
6. Rectangle completed

Draw the elliptical slot:

1. Click line command
2. Specify first point: specify point at (78,85)

3. Specify second point: @0,8
4. Click offset command
5. Select object and press enter
6. Specify offset distance: 4
7. Specify side: Right side
8. Click fillet command
9. Click fillet command
10. Enter fillet radius: 2
11. Select first object: select first line on the upper side.
12. Select second object: select second line on the upper side.
13. Full fillet will be created.
14. Repeat fillet command for lower side of the two line also.
15. Elliptical slot is completed.

Draw the Polar array:

1. Select the Polar Array command: Home Tab → Modify Panel → Array
2. Select Polar Array.
3. Select Objects: select the object to be arrayed (elliptical slot).
4. Select Objects: select more objects or <Enter> to stop.
5. Specify center point of array or [Base point / Axis of Rotation]: select center of the circle.
6. The Array Creation Tab appears and the default polar array with 6 items is displayed.
7. Enter items: 8
8. Enter Fill: 360
9. Press <Enter> to display the selections.
10. Select Close Array if display is correct.
11. Polar Array completed.
12. Save the drawing.

Create Block of the Polar array.

1. Click Menu → Draw → Block
2. AutoCAD displays the Block Definition dialog box (Fig. 6.11).
3. Enter the New Block name in the Name box: xyz
4. Select the Pick point button. The Block Definition box will disappear and you will return temporarily to the drawing.
5. Select the insertion point for the Block: select the center of the circle.
6. Select convert to block. Click the Select objects button.
7. The Block Definition box will disappear and you will return temporarily to the drawing.
8. Select the objects: middle circle and eight elliptical slots, then press ENTER.
9. Select the OK button.

10. The new block is now stored in the drawing's block definition table.
11. Block created.
12. Save the drawing.

Draw Rectangular array:

1. Select the Rectangular Array command: Home Tab → Modify Panel → Array
2. Select Rectangular Array.
3. Select Objects: select the complete block.
4. The Array Creation Tab appears with a 3 row and 4 column default rectangular array.
5. Enter Columns: 4
6. Between distance: 40
7. Enter Rows: 4
8. Between distance: 30
9. Press <Enter> to display the selections.
10. Rectangular Array completed.
11. Save the drawing.

Observations

Output drawing as shown in Fig(C)

Practical related questions

1. Can we select more than 1 entity for array? How?
2. Can we edit array after its creation? How?

Suggested learning resources

Refer Unit-6

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-6.

Practical-5 Draw Basic Branch Specific Components in 2D using AutoCAD

Practical statement

Draw basic branch specific 2D components like Nut in Mechanical and other engineering disciplines using AutoCAD.

Practical significance

Usually digital drawings of simple Mechanical components, Electrical components, Computer and Electronics components, chemical and process components are created by combining and modifying several different basic primitive shapes, such as lines, circles, arcs, polygons etc.

Relevant theory

Refer Section 6.9 (Draw Tool bar)

Refer Section 6.10 (Modify Tool bar)

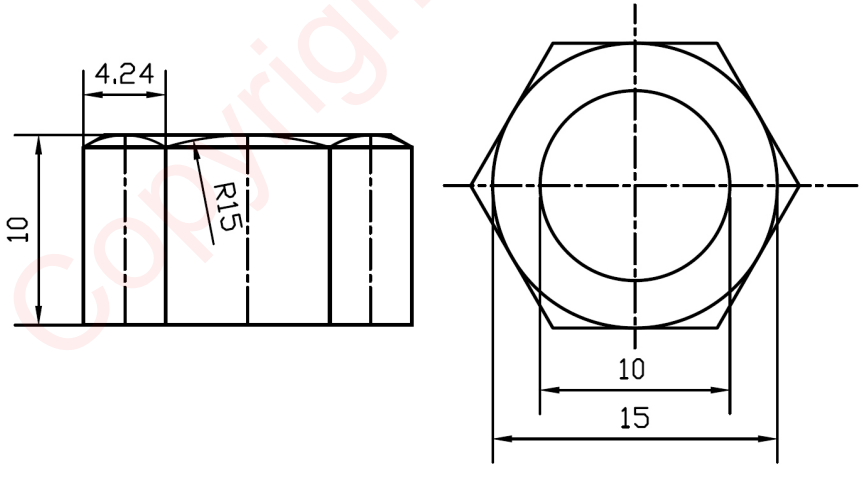
Refer Section 6.12 (Text and Dimension)

Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

PrO1: Use Draw and Modify tool bar commands of AutoCAD to create simple components related to various engineering disciplines.

Practical Setup (Drawing/Sketch/Circuit Diagram/Work Situation)



Resources required

Same as mentioned in Practical-1 of Unit-6.

Precautions

Same as mentioned in Practical-1 of Unit-6.

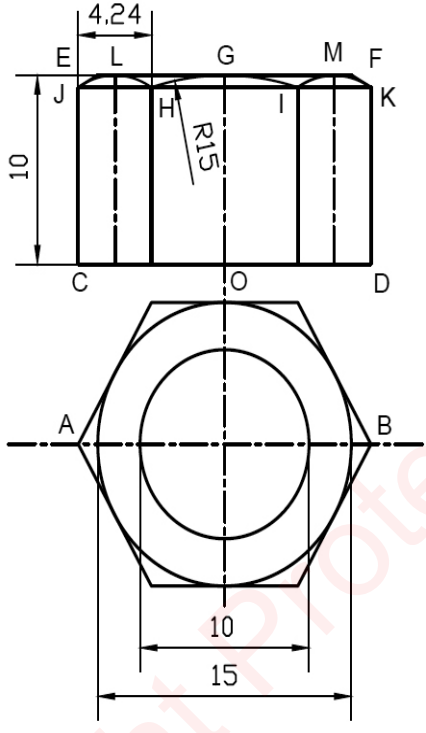
Suggested procedure

Preliminary Steps: Same as mentioned in Practical-1 of Unit-6.

To draw Rectangle in AutoCAD:

1. Start with Top view of the given Nut.
2. Draw horizontal line A-B of 15-mm length from the origin (50,50) using LINE command.
3. At its midpoint draw two circles of 5 mm and 7.5 mm radius using CIRCLE command.
4. With same center draw a hexagon with circumscribed option and radius 7.5 mm using POLYGON command.
5. Top view of Nut completed.
6. To draw Front view set ORTHO ON by pressing F8 and copy line A-B, vertically above at a suitable height as line C-D (the base line of front view).
7. Draw line E-F using OFFSET command: specify offset distance 10, select object to offset as line C-D and for side click above line C-D.
8. Draw vertical center line from point G with the help of MID object snap.
9. Draw vertical lines from the corners of the hexagon up to line E-F.
10. Use TRIM command to delete the vertical lines below the line C-D.
11. Draw a circle with 2P option. Specify first point as point G and other @0, –30 (Radius 15 mm, hence diameter 30).
12. Use TRIM command to delete the circle outside the points H and I.
13. Draw horizontal lines H-K passing through the points H and I and then line H-J.
14. The distance between points H and J is 4.28. It can be found either by dimensioning or by DIST command.
15. Use OFFSET command to get vertical lines from point L and M. Specify offset distance as 2.14 (half of 4.28).
16. Using ARC command with 3 points option, draw arcs J-L-H and I-M-K.
17. Use TRIM command to remove vertical lines between horizontal lines E-F and J-K.
18. Draw inclined line of any length, say 5 mm, and angle 30° at point J and angle 150° at point K.
19. Use FILLET command with 0 radius to fillet the inclined lines and the top horizontal line L-M.
20. Use DDLTYPE command to load center line.
21. Click Modify on Menu bar and choose Modify to change the center lines from continuous lines to center lines. Save the drawing as P-16.

Observations



Practical related questions

1. What is the diameter of Nut?
2. Write the relations between diameter and height/ width of a standard Nut.

Suggested learning resources

Refer Unit-6

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-6.

Practical-6 Draw Complex Branch Specific Components in 2D using AutoCAD

Practical statement

Draw complex branch specific 2D components like Square Thread Lead Screw of Lath in Mechanical and other engineering disciplines using AutoCAD.

Practical significance

Usually digital drawings of complex Mechanical components, Electrical components, Computer and Electronics components, chemical and process components are created by combining and modifying several different basic primitive shapes, such as lines, circles, arcs, polygons etc.

Relevant theory

Refer Section 6.9 (Draw Tool bar)

Refer Section6.10 (Modify Tool bar)

Refer Section 6.12 (Text and Dimension)

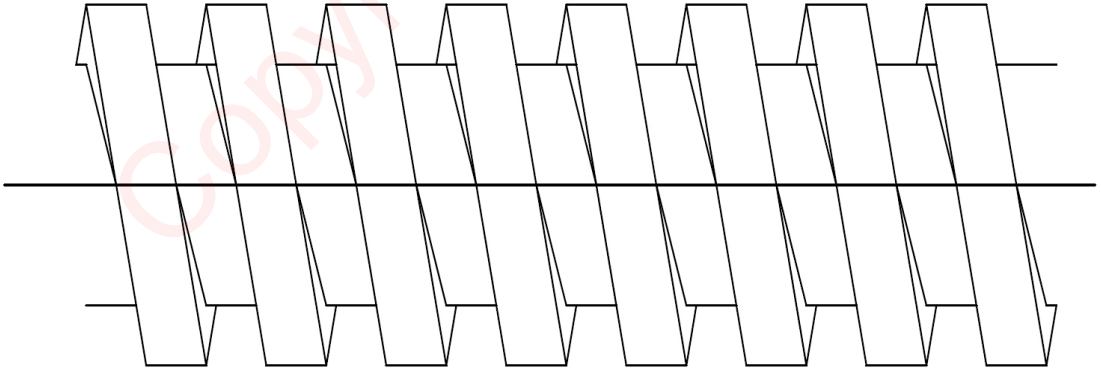
Practical outcomes (PrO)

The practical outcomes are derived from the curriculum of this course:

PrO1: Use Draw and Modify tool bar commands of AutoCAD to create simple components related to various engineering disciplines.

Practical Setup (Drawing/Sketch/Circuit Diagram/Work Situation)

Draw square threads with core diameter 40 mm, nominal dia 60 mm and the pitch 20 mm and eight threads as shown in Fig.



Resources required

Same as mentioned in Practical-1 of Unit-6.

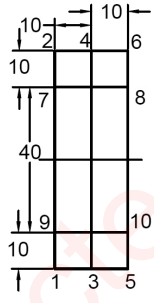
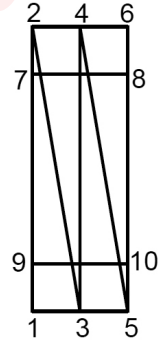
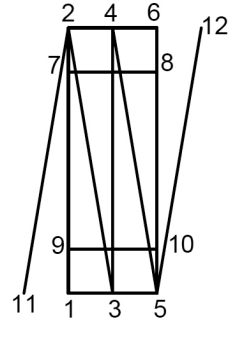
Precautions

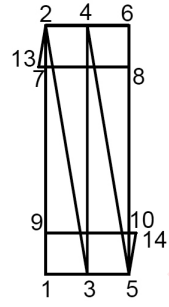
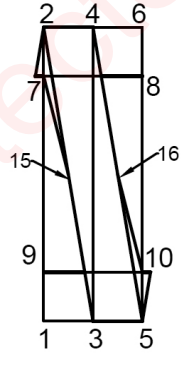
Same as mentioned in Practical-1 of Unit-6.

Suggested procedure

Preliminary Steps: Same as mentioned in Practical-1 of Unit-6.

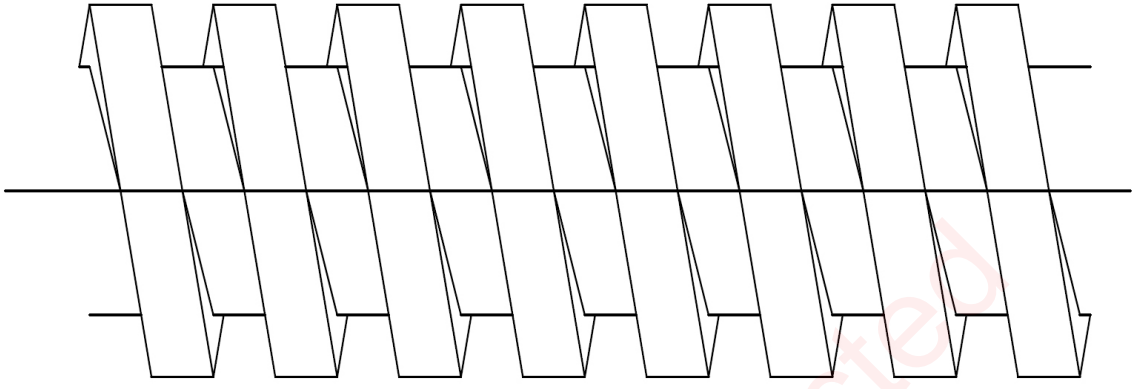
To draw Rectangle in AutoCAD:

1. Draw vertical line 1-2 of 100 mm length using LINE command. Draw lines 3-4 and 5-6 with offset distance 10 mm using LINE command as shown in the adjoining figure. 2. Join points 2-6 and 1-5 using LINE command. Using OFFSET command with offset 10 draw lines 7-8 and 9-10. 3. Draw the center line in the middle of the lines.	 STEP 1
4. Join point 2 to 3 and 4 to 5 to get inclined lines 2-3 and 4-5.	 STEP 2
5. Use MIRROR command: select object to mirror line 2-3 and mirror axis as line 1-2 to get line 2-11. Similarly use MIRROR command to mirror line 4-5 with mirror axis as 5-6 to get line 5-12.	 STEP 3

6. Use EXTEND and TRIM commands to trim lines 2-11 and 5-12 to get lines 2-13 and 5-14 respectively.	 STEP 4
7. Copy line 2-13 at point 1 to get point 17. Copy small horizontal line from point 9 up to inclined line and paste it from upper inclined line to get point 18. 8. Draw lines from points 7 to 15 and 10 to 16.	 STEP 5
9. Delete all the unwanted lines and leave only the lines as shown in adjoining figure. Set the line weight of these lines as 0.5 by Modify-Properties.	 STEP 6

10. Use ARRAY command. Select all objects for array as shown in step 9 by making a window across them. Specify number for Rows as 1 and columns as 8 in the text boxes. Specify Column offset as 20. Let the Angle of array be 0.
11. Click Preview button on lower right side of the dialog box and then press Enter Key.
12. Save the drawing as P-17.

Observation



Practical related questions

1. Is it possible to create only upper half portion of the object shown in step 9 and then use MIRROR/COPY/SYMMETRY commands(s) to get the full object shown in step 9? How?

Suggested learning resources

Refer Unit-6

Suggested assessment scheme

Same as mentioned in Practical-1 of Unit-6.

KNOW MORE

- Teacher should download the command reference guide of AutoCAD 2020.
- During the input sessions the teacher must give examples related to the relevant branch viz. Mechanical and allied disciplines/ Electrical and allied disciplines/Electronics etc.
- Teacher should demonstrate all above commands through AutoCAD software and guide students in Hands-on sessions.
- Show video/animation films to explain mentioned concepts.
- Guide student(s) in preparing Institute Template drawing work.
- Use videos/spoken tutorials to teach AutoCAD software.
- Teachers should ask students to use url/qr codes available in the book for further understanding/practicing of the concepts.
- Teacher should take more sessions on commands like Block, Layers, Text writing, Dimensioning, Array and Hatching.
- Teacher must know 3D modeling also using AutoCAD 2020.

Applications (Real life / Industrial)

1. Usually digital drawings are created by combining and modifying several different basic primitive shapes, such as lines, circles, and arcs detailed out in this unit to create more complex engineering disciplined based shapes like gears, springs, joints, couplings, armature, nut-bolts, bearings, sprockets, cams, rotors, turbine/compressor blades, pumps, motors, IC engine parts, automobile components, medical instruments, mechanical components, plastic components, electronic components, house hold items, artifacts, etc. AutoCAD is widely used design and drafting software to create these digital drawings in almost all industries.
2. Digital Production drawings are used in mechanical and allied industries from a very small component to a big assembly.
3. Digital drawings are also used as construction drawings by civil engineers and architects for houses, buildings, hospitals, hotels, malls, airports, overhead tanks, bridges, roads etc.
4. Digital electrical and electronics drawings are used for electrical machines, transformers, drives, actuators, insulators, circuits, circuit bords, electrical and electronic components, computer cabinets, computer monitor cases, keybord-mouse drawings, pannels, electronic hardware etc.
5. Digital drawings are also used in piping, textile, oil and refinery and food processing industries by chemical engineers, pharma engineers and process engineers etc.

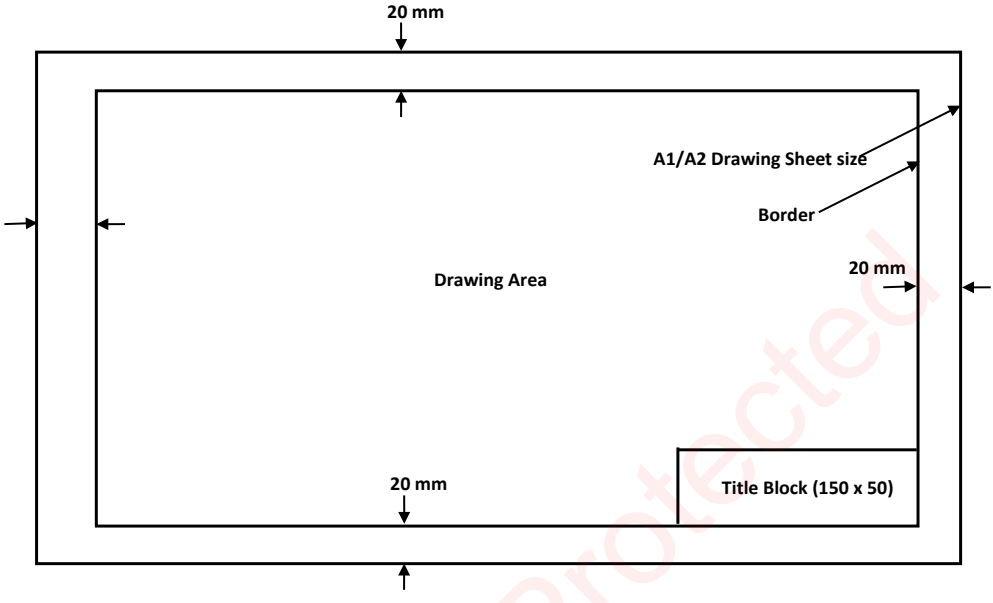
Inquisitiveness and Curiosity

1. As engineering graphics is a skill oriented course and requires continuous practice to understand and develop competence on use of various commands and techniques mentioned in the unit, therefore, students are advised to draw and practice little complex 2D geometry of industrial components to horn the skills further. Pick any component of your engineering discipline and try it out in AutoCAD using mentioned steps.
2. Students should collect Production drawings, Building Drawings, Layouts from nearby workshops/industries/builders/contractors and should attempt to redraw the same using AutoCAD software using commands mentioned in this unit.

Design Innovative Practical / Projects / Activities

Student shall form a group of 5 - 6 students and undertake one or two micro project(s) /activity(ies) under the guidance of faculty and present it as group with individual participation as well. A sample list is given below:

1. Develop Drawing Template of your institute with following dimensions and details.



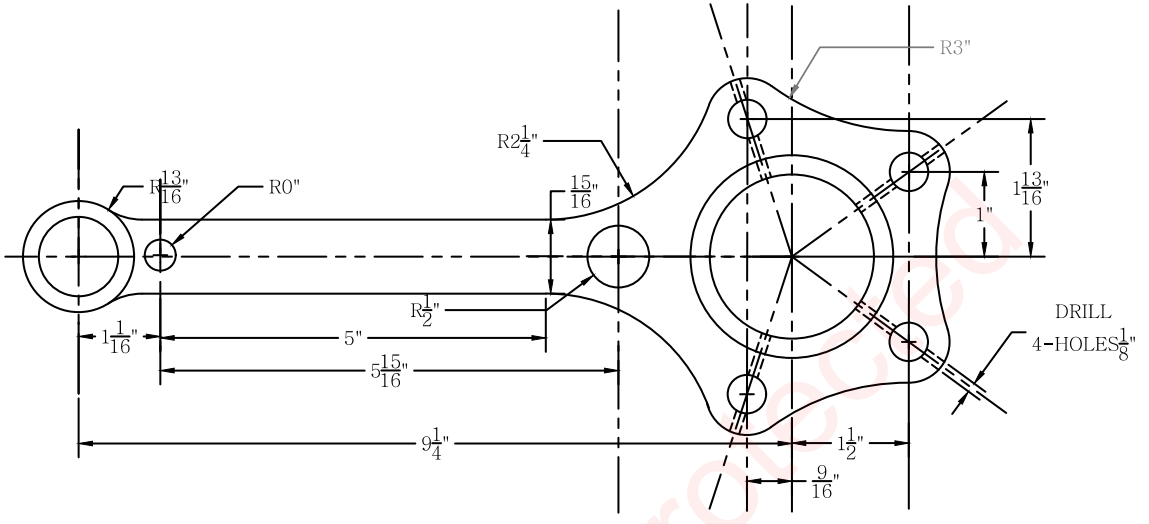
Institute Drawing Template

Name:	**Title:**	
Class:	**Description :**	
Roll No.:	**Sheet No:**	**Date:**
Institute Name: Institute Logo	**Projection:**	

Title Block with institute Logo

2. Draw following view of a typical Connecting rod.

   For dimensions in mm please refer same figure on cover page of the book.



REFERENCES & SUGGESTED READINGS

1. Machine Design Includes AutoCAD Second Edition, Ajeet Singh, Tata McGraw Hill Education Private Limited, New Delhi, 2018.
2. AutoCAD 2020: A Problem-Solving Approach, Basic and Intermediate, 26th Edition, Sham Tickoo, CADCIM Technologies, 2019.
3. AutoCAD 2013, Command Reference Guide, Autodesk Inc.
4. AutoCAD Shortcuts Guide, Autodesk Inc.
5. Autodesk knowledge network, tutorials, documentation, downloads, troubleshooting articles, https://knowledge.autodesk.com/support

APPENDICES

APPENDIX-A: Records for Practical

Sl No	Page No	Name of the Experiment	Date			Marks	Signature
			Actual	Repeat	Record		
1	Draw horizontal, Vertical, 30 degree, 45 degree, 60 and 75 degrees lines, different types of lines, dimensioning styles using Tee and Set squares/ drafter.						
2	Write alphabets and numerical (Vertical only)						
3	Draw regular geometric constructions and redraw the given figure Part I						
4	Draw regular geometric construction and redraw the given figure Part II						
5	Draw a problem on orthographic projections using first angle method of projection having plain surfaces and slanting.						
6	Draw another problem on orthographic projections using first angle method of projection having slanting surfaces with slots.						
7	Draw two problems on orthographic projections using first angle method of projection having cylindrical surfaces, ribs.						
8	Draw two problems on isometric view of simple objects having plain and slanting surface by using natural scale.						

9	Draw some problems on isometric projection of simple objects having cylindrical surface by using isometric scale.						
10	Draw free hand sketches/ conventional representation of machine elements in sketch book such as thread profiles, nuts, bolts, studs, set screws, washers, locking arrangements.						
11	Problem based Learning: Given the orthographic views of at least three objects with few missing lines, the student will try to imagine the corresponding objects, complete the views and draw these views in sketch book.						
12	Draw basic 2D entities like: Rectangle, Rhombus, Polygon using AutoCAD.						
13	Draw basic 2D entities like: Circles, Arcs, circular using AutoCAD.						
14	Draw basic 2D entities like: Circular and rectangular array using AutoCAD.						
15	Draw blocks of 2D entities comprises of Rectangle, Rhombus, Polygon, Circles, Arcs, circular and rectangular array, blocks using AutoCAD.						
16	Draw basic branch specific components in 2D using AutoCAD.						
17	Draw complex branch specific components in 2D using AutoCAD.						

ANNEXURES

Annexure-I General and Specific Instructions when working in the Drawing hall/ Computer aided drafting software lab

General Instructions

1. It is a fact that the objective of the practical sessions is learning and also a verification of the knowledge that you have gathered. The practical are designed in such a way so as to support and reinforce the concepts learned in theory.
2. By doing the practical with your own interest only it is possible to be familiar with all the fine points and to expose you to drawing instruments and computer aided drafting software.
3. Always perform the practical with an attitude of learning and with your interest to verify the theoretical knowledge that you have gathered.
4. Be very particular to arrive in time in the drawing hall/laboratory and always with proper preparation with a clear knowledge about the practical.

Specific Instructions

1. When working in the drawing hall to prepare drawings as per list of practical and assignment it is mandatory to come with drawing sheet, sketch book, pencils, eraser, sharpener, napkin, and all required drawing instruments as listed in the corresponding practical.
2. The completed work on the drawing sheet or in sketch book has to be shown to your teacher/ instructor before leaving the drawing hall.
3. All the students have to maintain individual copy of the completed drawing sheets/sketch book.
4. Dimensioning and other annotations activities should always be done with proper units for each drawing practice problem.
5. Try to use drawing sheet in optimum manner by proper planning of space required for each practice problem and use both sides of the drawing sheet to complete the assigned practice problems.
6. During the working hours in the practical sessions you are supposed to fully utilize the duration and do not leave the drawing hall before the completion of the working hours. If you finish early, you may spend the remaining time to complete similar extra practice problems in the drawing sheet/sketch book.
7. In case of practical related to computer aided drafting software lab, save the work done at regular interval and shut down the computer properly after the completion of practical. Save the completed work on pen drive or email or submit it to the digital submission box/drive.
8. Try to practice more drawing problems to horn the skills.

REFERENCES FOR FURTHER LEARNING

Lists of some of the books are given below which may be used for further learning of the subject (both theory and practical) by the interested students:

1. Bureau of Indian Standards, Engineering Drawing Practice for Schools and Colleges IS: SP-46, BIS, Government of India, Third Reprint, October 1998; ISBN: 81-7061-091-2.
2. N.D. Bhatt, Engineering Drawing, Charotar Publishing House, Anand, Gujrat 2010; ISBN: 978-93- 80358-17-8.
3. Jain & Gautam, Engineering Graphics & Design, Khanna Publishing House, New Delhi; ISBN: 978-93-86173-478.
4. D.A. Jolhe, Engineering Drawing. Tata McGraw Hill Edu. New Delhi, 2010; ISBN: 978-0-07-064837-1.
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7. N.S. Parthasarathy, Vela Murali, Engineering Drawing, Oxford University Press, 2015.
8. K. Venkata Reddy, A Text Book of Engineering Drawing, BS Publication, 2008.
9. D. M. Kulkarni, A.P Rastogi, A.K. Sarkar, Engineering Graphics with AutoCAD. PHI Learning Private Limited-New Delhi (2010); ISBN: 978-8120337831.
10. T. Jeyapoovan, Essentials of Engineering Drawing and Graphics using AutoCAD, Vikas Publishing House Pvt. Ltd, Noida, 2011; ISBN: 978-8125953005.
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12. Sham, Tickoo. AutoCAD 2016 for Engineers and Designers. Dreamtech Press; Galgotia Publication, New Delhi, 2015; ISBN 978-9351199113.
13. Yasser Shoukry, Jaiprakash Pandey, Practical Autodesk AutoCAD 2021 and AutoCAD LT 2021: A no-nonsense, beginner's guide to drafting and 3D modeling with Autodesk AutoCAD, Packt Publishing, 1st edition, 2020.
14. B.V.R. Gupta, M. Raja Roy, Engineering Drawing With AutoCAD, Third Edition, Dreamtech Press, 2020.
15. Roop Lal, Ramakant Rana, A Textbook of Engineering Drawing : Along with an introduction to AutoCAD (R) 2015, I K International Publishing House Pvt. Ltd, 2015.
16. D.A. Hindoliya, A Textbook of Engineering Graphics, B.S. Publication, 2014, ISBN:978-93-83635-21-4.

CO AND PO ATTAINMENT TABLE

Course outcomes (COs) for this course can be mapped with the programme outcomes (POs) after the completion of the course and a correlation can be made for the attainment of POs to analyze the gap. After proper analysis of the gap in the attainment of POs necessary measures can be taken to overcome the gaps.

Table for CO and PO attainment

Course Outcome	Attainment of Programme Outcomes *(1- Weak Correlation; 2- Medium correlation; 3- Strong Correlation)*						
	PO-1	PO-2	PO-3	PO-4	PO-5	PO-6	PO-7
CO-1							
CO-2							
CO-3							
CO-4							
CO-5							
CO-6							

The data filled in the above table can be used for gap analysis.

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