ENGINEERING GRAPHICS & DESIGN

PRADEEP JAIN

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ISBN: 978-93-91505-06-6
Book Code: UG004EN

Engineering Graphics & Design
by Pradeep Jain
[English Edition]

First Edition: 2021

Published by:

Khanna Book Publishing Co. (P) Ltd.
Visit us at: **www.khannabooks.com**
Write us at: **contact@khannabooks.com**
CIN: U22110DL1998PTC095547

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Printed in India.

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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 grateful to AICTE for their meticulous planning and execution to publish the technical book for Engineering and Technology students.

I sincerely acknowledge the valuable contributions of the reviewer of the book Prof. B.S. Pabla, 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 I 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, I like to express my 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.

Pradeep Jain

Preface

The traditional engineering graphics course has undergone significant change due to emergence of computer-aided drafting and design (CAD) tools and the revision of engineering graphics curriculum to include computer based 2D and 3D modeling. The emphasis has shifted from drawing board based engineering graphics to CAD based modeling which has the advantages of speed, flexibility and convenience of drawing. In spite of these the focus on free hand sketching and development of ability to visualize the objects in 2D and 3D frame has been retained in the revised curriculum. The text book on "Engineering Graphics & Design" addresses the challenges of integrating computer aided drawing and design to develop the drafting manual skills and to integrate the computer based drafting without losing focus on the basic drawing skills. The book has been aligned to outcome based education to focus on the learning outcomes leading to attainment of program outcomes. Course outcomes and unit outcomes have been defined for each unit of the curriculum. COs have been mapped with the POs to enable the students to appreciate and work for attainment of program outcomes. The text matter has been reduced to make it easier to students. Each chapter is followed by a set of questions and the references for further reading. The first part of the book covers the conventional aspects of engineering drawing and the second part is devoted to the use of computer aided drafting.

It is sincerely hoped that the book will redefine the learning of engineering graphics and design with focus on development of visualization of engineering objects and use of computer software in developing 2D and 3D models.

Author will thankfully acknowledge the comments and suggestions for the future improvements of the book.

Pradeep Jain

Outcome Based Education

For the implementation of an outcome based education the first requirement is to develop an outcome based curriculum and incorporate an outcome based assessment in the education system. By going through outcome based assessments evaluators will be able to evaluate whether the students have achieved the outlined standard, specific and measurable outcomes. With the proper incorporation of outcome based education there will be a definite commitment to achieve a minimum standard for all learners without giving up at any level. At the end of the programme running with the aid of outcome based education, a student will be able to arrive at the following outcomes:

PO-1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.

PO-2. **Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

PO-3. **Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.

PO-4. **Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

PO-5. **Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

PO-6. **The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO-7. **Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

PO-8. **Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

PO-9. **Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO-10. Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

PO-11. Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

PO-12. Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

Course Outcomes

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

CO-1: Demonstrate the visual understanding of engineering drawing.

CO-2: Create working engineering drawings.

CO-3: Apply computer aided drafting for 2 D and 3 D modeling.

CO-4: Apply the modern engineering tools necessary for engineering practice.

CO-5: Read and communicate through engineering drawings.

Mapping of Course Outcomes with Programme Outcomes

Course Outcomes	Expected Mapping with Program Outcomes *(1- Weak Correlation; 2- Medium correlation; 3- Strong Correlation)*											
	PO-1	PO-2	PO-3	PO-4	PO-5	PO-6	PO-7	PO-8	PO-9	PO-10	PO-11	PO-12
CO-1	3	2	1	-	-	1	-	-	-	-	-	-
CO-2	3	3	2	2	-	-	-	-	-	-	-	-
CO-3	3	2	3	1	3	2	-	-	-	-	-	-
CO-4	3	2	1	1	3	2	-	-	-	-	-	-
CO-5	3	2	2	-	-	2	-	-	-	3	-	-

Abbreviations

2D	Two Dimensional
3D	Three Dimensional
Aux	Auxiliary
BIS	Bureau of Indian Standards
CAD	Computer-aided design
CO	Course Outcome
CUI	Customize User Interface
FV	Front View
GUI	Graphical User Interface
HP	Horizontal plane
HT	Horizontal trace
IS	Indian standard
ISO	International Organization for Standardization
LOS	Line of Sight
OSNAP	Object SNAP
PO	Programme Outcomes
PP	Profile Plane
RF	Representative Fraction
THK	Thick
TL	True Length
TV	Top View
UCS	User Coordinate System
UO	Unit Outcomes
VP	Vertical Plane
VT	Vertical Trace

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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 manipulate 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.

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 Introduction to Engineering Drawing

UNIT SPECIFIC

In this unit, "Introduction to engineering drawing" students will learn and understand various topics such as, principles of engineering graphics and their significance, usage of drawing instruments, lettering, conic sections including the rectangular hyperbola, cycloid, epicycloid, hypocycloid and involutes. Plain, diagonal and vernier scale.

RATIONALE

The purpose of this introductory chapter of course Engineering Graphics and Design is to enable engineers and craftsmen to communicate their ideas and facts with clarity and without ambiguity. This chapter mainly deals with the basic knowledge of creating working drawings and usage of various instruments.

PREREQUISITE

Basic knowledge of geometry and engineering drawing instruments.

UNIT OUTCOMES

Upon successful completion of this module, the students will be able to:

U1-O1: Apply common drafting tools to construct engineering drawing.
U1-O2: Create engineering drawings sheet layout.
U1-O3: Apply various types of lettering.
U1-O4: Create various types of scales.
U1-O5: Construct different types of conic sections.

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-1 Outcomes	Expected Mapping With Course Outcomes *(1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)*				
	CO-1	CO-2	CO-3	CO-4	CO-5
U1-O1	3	3			2
U1-O2	2	2			
U1-O3	2	2			
U1-O4	2	2			
U1-O5	2	2			

INTRODUCTION

"A picture is worth a thousand words". For centuries drawings have been used by humans as a major mode of communication by marking messages on sand, painting or drawing pictures of animals, trees and people on the walls of caves. The drawing itself is a way of communicating all necessary information about a concept or a graphic representation of some real entity, such as a machine part, building or tools. Engineering drawings are in use for more than 2000 years and it was first formally introduced by the French mathematician Gaspard Monge in the 18th century.

The first important step in the development of computer-aided drafting was made at the Massachusetts Institute of Technology (MIT) in 1963 where a system called Sketchpad was developed and demonstrated. The reason why manual drawings are introduced to students prior to computer-aided drawings is to make them learn the standards, conventions, tools and help them visualize how drawings are made.

1.1 DRAWING INSTRUMENTS

Engineering drawing is entirely a graphic language and instruments are essentially needed to record information on paper or sheet. The engineering drawing must be clear, neat and legible in order to serve its purpose, hence it is extremely important for engineers to display speed, accuracy, legibility and neatness in their drawing work. The quality of drawings depends to a very large extent on the quality, adjustment and care of the instruments used. The instruments and other aids used for drawing are listed below:

(i) Drawing Sheets
(ii) Drawing Board
(iii) T-Squares
(iv) French Curves
(v) Mini Drafter
(vi) Drawing Pencils
(vii) Instrument Box

Drawing Instruments

1.1.1 Drawing Sheets

Drawing sheet is the medium on which drawings are prepared by means of pencil or pen. Drawing sheet is a white paper on which an object is drawn; the sheet used for engineering drawing should be of good quality with uniform thickness and the surface of the sheet must be smooth.

Sizes of Drawing Sheets

Drawing sheets are available in standard sizes as shown in Table 1.1. A standard A0 size sheet is the one with an area of 1 m^2 and has these dimensions: 1189 x 841 mm. Every sheet number marked higher (A1, A2, and A3 in order) is half the size of the one immediately lower than it. The recommended standard sizes for drawing sheets are shown in figure 1.1.

Table 1.1: Standard sizes of drawing sheets

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

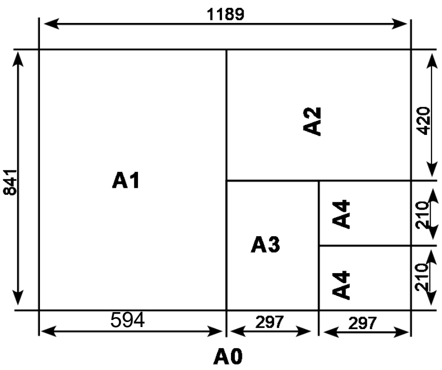


Figure 1.1: Standard Sizes of Drawing Sheets

Sheet Layout and Fixing Sheet

Drawing Sheet Layout

The layout of a drawing sheet consists of drawing space, title block and sufficient filing margins. Figure 1.2 shows a typical drawing sheet with following characteristics.

- **Borders:** A minimum of 10 mm space is to be left all around the trimmed edges of the sheet.
- **Filing margin:** A minimum of 20 mm space on the left hand side including the border is provided for making perforations.
- **Grid reference system:** The *grid reference system* is drawn on the sheet to permit easy location on the drawing such as details, alterations or additions. The length of the grids lies between 25 mm to 75 mm depending on the drawing sheet size. The grids along the vertical edges are labeled with capital letters and the grids along the horizontal edges are labeled with numerals. Numbering and lettering starts from the corner of the sheet opposite to the title box and is repeated on the other side also.

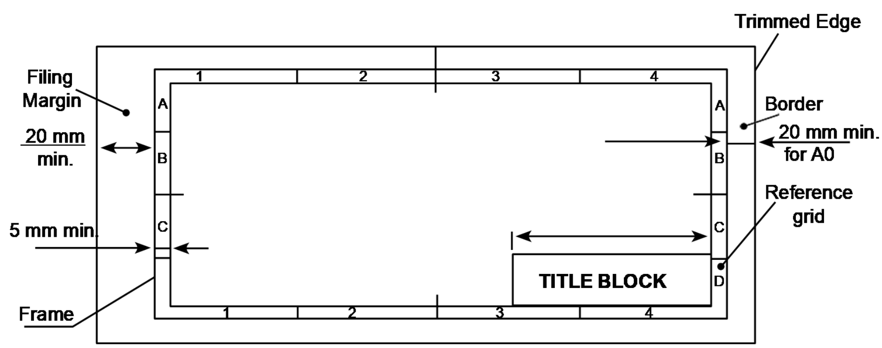


Figure 1.2: Drawing Sheet Layout

Drawing Sheet Title Box

The title box, which is an important feature and a necessity in every drawing sheet, is normally drawn at the bottom right hand corner of every drawing sheet as shown in figure 1.3. The title box provides technical and administrative details regarding the drawing. Though there are various dimensions for the title box, for engineering students it is advisable to use a title box of size 170 mm x 65 mm.

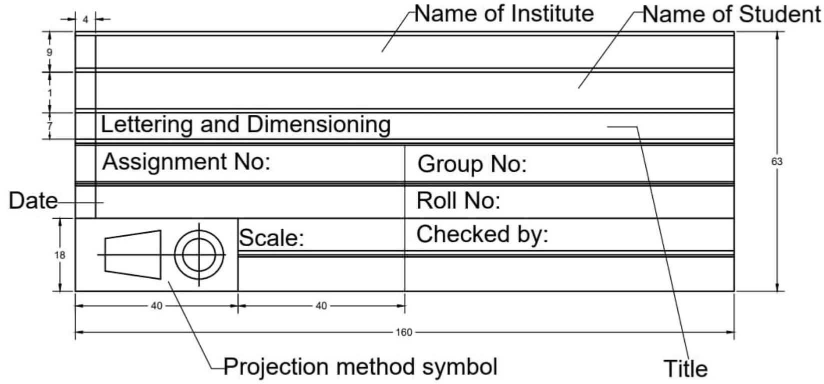


Figure 1.3: Drawing Sheet Title Block

1.1.2 Drawing Board

A drawing board is known to be a multipurpose desk which can be used for any kind of drawing, writing or sketching on a large sheet of paper or for drafting precise technical illustrations. An engineering drawing board is made of four to six strips of well seasoned soft wood such as pine, fir or oak having a thickness of about 25 mm with a working edge for T-square, as shown in figure 1.4.

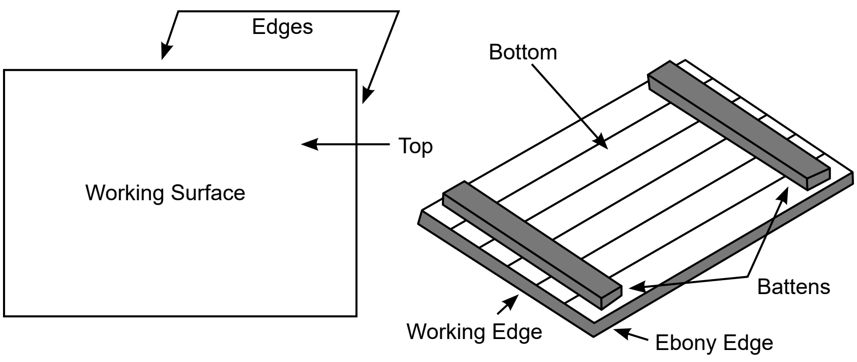


Figure 1.4: Drawing Board

Sizes of Drawing Boards

Generally, the standard sizes of drawing board depend on the size of drawing sheet required or used. The sizes of drawing boards that are recommended by the BIS (SP-46) are presented in table 1.2.

Table 1.2: Description of the size of standard drawing boards

Sr. No.	Designation	Dimensions (mm)		
		Length	Width	Thickness
1	D0	1500	1000	25
2	D1	1000	700	25
3	D2	700	500	15
4	D3	500	350	15

Important Observation
Board of D2 and D3 sizes are generally used in schools and colleges and large-sizes boards are used in drawing offices of engineering firms.

Drawing Clips

Drawing clips are used to hold drawing sheet on drawing boards at the required place, restricting its movement while working. The present trend is to use steel clips, if the size of the drawing paper is same as that of the drawing board. Drawing clips are attached at all the four corners of the drawing board to hold the paper in place.

1.1.3 Mini Drafter

A mini drafter is an engineering device utilized to draw parallel or inclined lines. It has the structure of an angle formed by two arms with scales marked and rigidly hinged to each other and combines the functions of T-square, set-squares, scales and protractor. The tool is mounted on the top left corner of the drawing board employing a clamping mechanism which is considered to be an integral part of the device. An L-shaped scale which is graduated in millimeters acts as the working edge of the mini drafter and also has a degree scale for the purpose of angle measurement. The working edge can be moved to any desired location on the drawing board. Figure 1.5 shows a typical mini drafter.

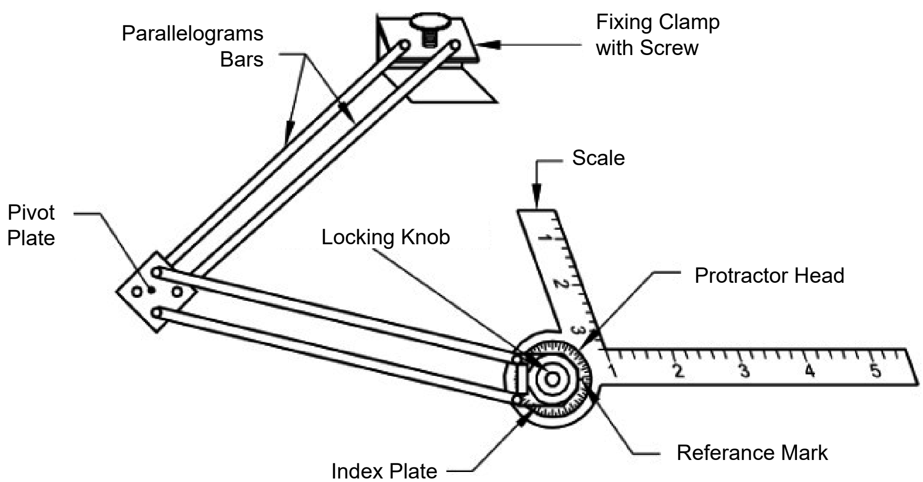


Figure 1.5: Mini Drafter

Clamping of Mini Drafter

The following steps are used for clamping the mini drafter on drawing board:

1. Set the protractor head with a reference mark indexing zero degree.
2. Then, fix the clamp of the mini drafter at the top left corner; either along the top horizontal edge or the left vertical edge of the board.
3. Following that, fix the drawing sheet to the drawing board with the scales of the mini drafter aligned either with the vertical or the horizontal border lines of the drawing sheet.

How to use Mini Drafter

Mini drafter contains two movable parts and one non-movable part. The movable part, known as the diagonal scale, which is the centre part of the device, helps to draw parallel lines in any shape with any angle. At the end of the device, there is a specific point on the round-shaped pad, a parallel line or angular line is drawn after matching the marked degree given in the scale with that point of the device.

1.1.4 T-Square

A T-Square is a straight edge with a short, thick cross piece that fits over the edge of a drawing board as shown in figure 1.6. It is used to draw horizontal lines and for guiding the triangles when drawing vertical and inclined lines. It is manipulated by sliding the working edge (inner face) of the head along the left edge of the board until the blade is in the required position.

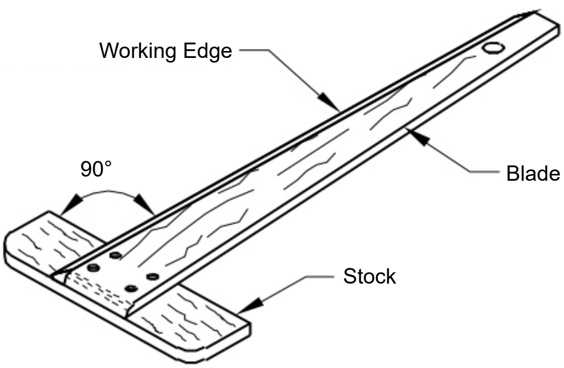


Figure 1.6: T–Square

The standard sizes of ‘T’ square recommended by the Bureau of Indian Standards are given in table 1.3.

Table 1.3: Standard sizes of ‘T’ square

S. No.	Designation	Blade length (mm)
1.	T0	1500
2.	T1	1000
3.	T2	700
4.	T3	500

1.1.5 French Curves

French curves are plastic (or wooden) templates having an edge composed of several different curves. French curves are used to draw smooth curves of almost any irregular shape that are not exactly circular as shown in figure 1.7.

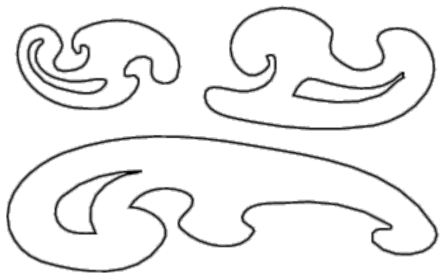


Figure 1.7: French Curves

1.1.6 Drawing Pencils

Pencil or lead stick is a primary tool in engineering drawings. Different grades of pencils are used for different purposes. Pencil lead is made up of graphite mixed with varying quantities of clay to produce various levels of hardness. There are 18 grades of pencils in order of descending hardness: 9H, 8H, 7H, 6H, 5H, 4H, 3H, 2H, H, F, HB, B, 2B, 3B, 4B, 5B, 6B, 7B, 8B, 9B where 9H is the hardest and 9B is the softest as shown in figure 1.8.

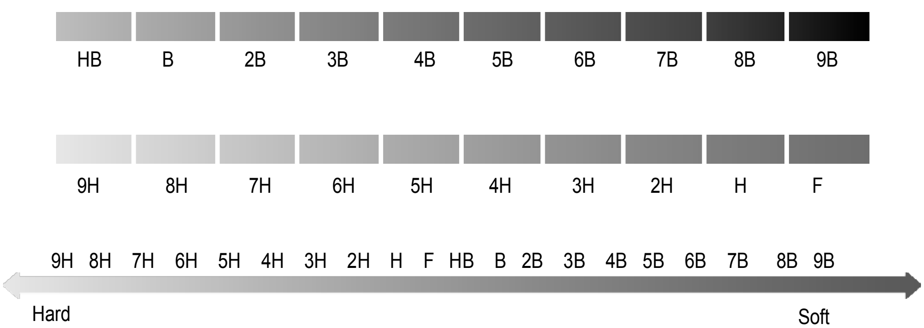


Figure 1.8: Grade of pencil leads

1.1.7 Instrument Box

Instrument box are used to prepare neat and accurate drawings and the accuracy of the drawings depends on the quality of instruments used to prepare them. Instrument box contains various instruments for specific work. Followings are the important instruments found in a typical instrument box.

1. Set square
2. Compass
3. Divider
4. Protector

1.2 LETTERING

Lettering is used for writing titles, sub-titles, dimensions and other details on a drawing. Main Features of lettering are legibility, uniformity and rapidity of execution. Typical lettering features used for engineering drawing is shown in figure 1.9.

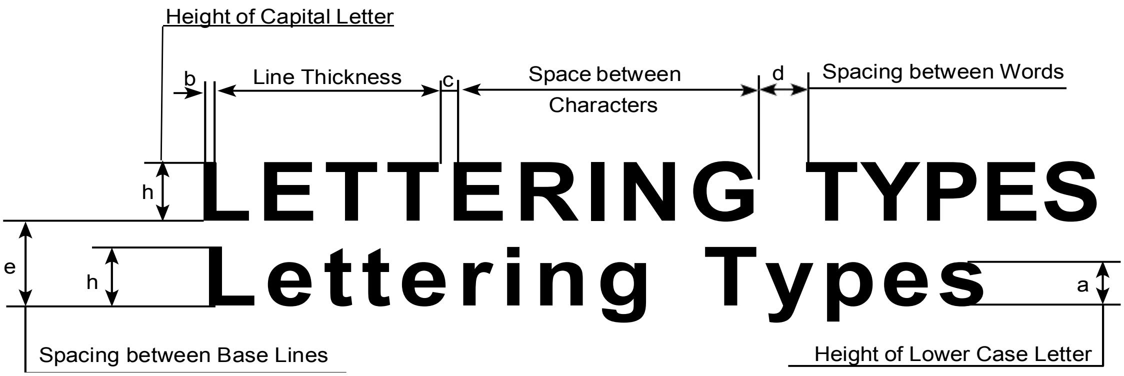


Figure 1.9: Lettering Features used in Engineering Drawing

1.2.1 Size of Letters

Size of letters is measured by the height (h) of the letters as well as numerals. Standard heights for letters and numerals recommended by BIS (SP-46) are 1.8, 2.5, 3.5, 5, 6, 10, 14 and 20 mm. The letter sizes generally recommended for various items are shown in Table 1.4.

Table 1.4: The letter sizes recommended for various items

S. No.	Item	Size (mm)
1.	Name of organization	10, 14, 20
2.	Drawing numbers, Letter denoting section planes	10, 14
3.	Title of drawing	7, 10
4.	Sub Titles and headings	5, 7
5.	Dimensioning, Notes, Schedules, Material lists	3.5, 7
6.	Alternate entries and Tolerances	3.5

1.2.2 Single Stroke Lettering

Single-stroke lettering is the simplest forms of letters and is usually employed in most of the engineering drawings. The word single-stroke means that the thickness of the line of the letter should be obtained in one stroke of the pencil. Single stroke lettering is classified as vertical lettering and inclined lettering.

Vertical Lettering

Vertical letter having thickness of each line of alphabet or numerals are same as the single stroke of a pencil.

Figure 1.10 shows the vertical lettering.

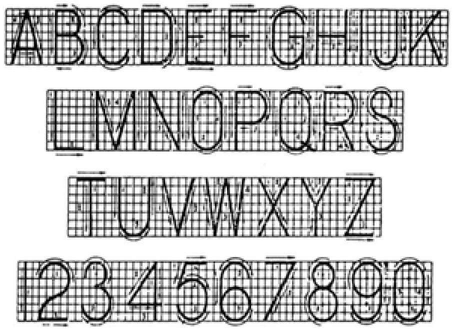


Figure 1.10: Vertical Lettering

Inclined or Italic Lettering

Inclined letter having thickness of each line of alphabet or numerals are same as the single stroke of a pencil and these letters are inclined at 75^0 to the horizontal as shown in figure 1.11.

Figure 1.11: Inclined or Italic Lettering

1.2.3 Roman Lettering

The lettering in which all the letters are formed by thick and thin elements is called roman lettering. It may be vertical or inclined. It can be written with a chisel pointed pencil or D-3 type speed ball pen as shown in figure 1.12.

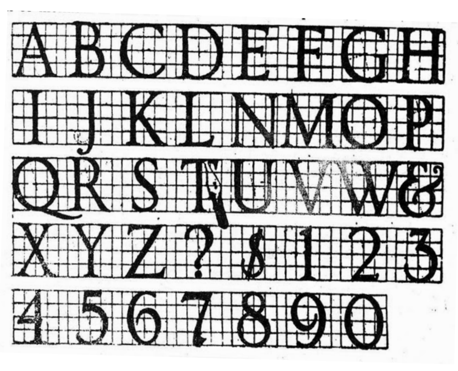


Figure 1.12: Roman lettering

1.3 SCALES

A scale is defined as the ratio of linear dimension of an object as represented in original drawing to the actual linear dimension of the same object.

Representative Fraction

Dimensions of large objects must be reduced to accommodate on standard size drawing sheet. The ratio of the dimension of the object shown on the drawing to its actual size is called the Representative Fraction (RF).

$$\text{Representative Fraction} = \frac{\text{Length of line in drawing}}{\text{Actual length of line object}}$$

Types of Scales

Scales may be divided into the following categories:

(i) Plain Scale
(ii) Diagonal Scale
(iii) Vernier Scale
(iv) Comparative Scale
(v) Scale of Chords

1.3.1 Plain Scale

A plane scale consists of a line divided into suitable number of equal parts or units, in which first part is subdivided into small parts as shown in figure 1.13.

Features of the plain scale:

1. The zero is placed at the end of the first main division i.e., between units and its sub units.
2. From the zero mark, the units are numbered to the right and its sub – division to the left.
3. Units of the divisions and sub-divisions are stated below at the respective ends.
4. The length of the scale is obtained by the following formula:

Length of the scale = R.F x Maximum length to be measured by the scale

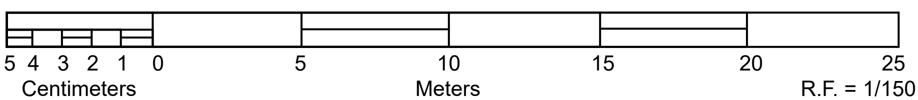


Figure 1.13: Plain scale

Procedure for construction of plane scale:

1. Find the Representative Fraction using the formula

$$\text{R. F.} = \frac{\text{Length of line in drawing}}{\text{Actual length of line object}}$$

2. Draw a straight line equal to length of scale.
3. Divide it into equal number of parts and mark zero at the end of first main division.
4. Make sub-divisions on first main division as per the requirement.

5. Number the units to the right and sub-divisions to the left from the zero mark.
6. Write the names of units and sub-divisions at respective ends.

1.3.2 Diagonal Scale

A diagonal scale consists of a set of parallel lines crossing obliquely with other set of lines such that their intersections create subdivisions. It is mainly used when measurements are required in three units such as cm, m and km. The scale has two parts: the lower part which is a plane scale and the upper part which has oblique lines as shown in figure 1.14. On this scale it is possible to read up to $1/100^{th}$ of a meter which is not possible on plane scale.

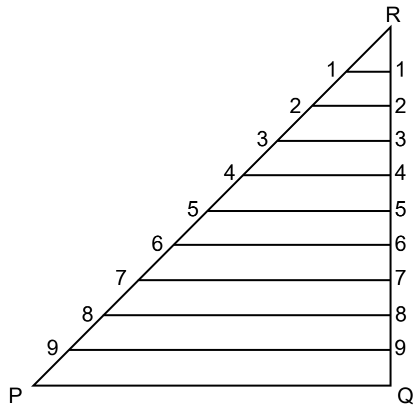


Figure 1.14: Diagonal Scale

Procedure for construction of diagonal scale:

1. First draw a line of length PQ.
2. At the end Q, draw a line perpendicular to PQ and mark off ten divisions (equal distances of any length) along it starting from Q to R.
3. Join PR and lines parallel to PQ and mark them 1-1, 2-2, 3-3 and so on.
4. Triangles 1-1R, 2-2R, 3-3R up to PQR are similar.

1.3.3 Vernier Scale

Vernier scale is modified form of diagonal scale, used to measure fractional parts of the smallest division of the main scale. The vernier scale is divided into two parts: primary or main scale which is a fixed scale and a vernier scale which is a moving graduated scale. The graduations on vernier scale are derived from those on the primary scale and it slides along the edge of long fixed scale (main scale).These are very accurate for reading very small units. Figure 1.15 shows the vernier scale.

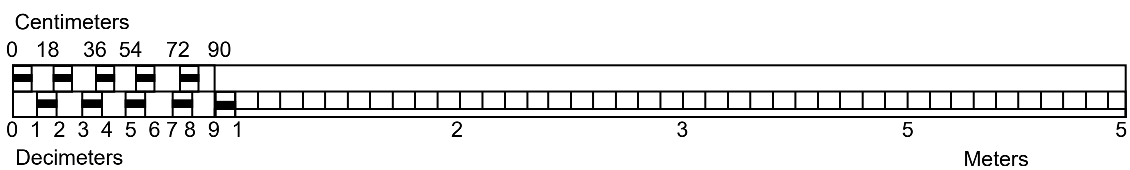


Figure 1.15: Vernier scale

1.3.4 Comparative Scale

Comparative scale or corresponding scale consists of two scales of same representative fraction but graduated to read different units. They may be plain scales or diagonal scales or vernier scales or separately constructed scales one above the other. They are used to compare distances expressed in different systems such as kilometers and miles. Figure 1.16 shows the comparative scale.

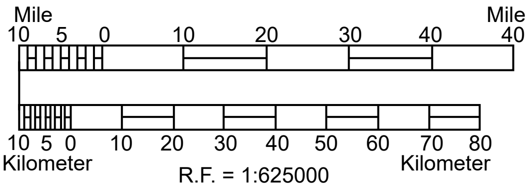


Figure 1.16: Comparative Scale

1.3.5 Scale of Chords

It is a method of constructing and measuring angles using a linear scale when protractor is not available. Angles are measured by comparing the angles subtended by chords of an arc at the center of the arc.

Procedure for construction scale of chord:

1. Draw a line RS of any length.
2. At S draw a perpendicular ST.
3. With S as center and ST as radius draw an arc RT.
4. Divide the arc in 9 equal parts each division of 10° and mark them as 10,20,30,40 and so on. This may be done :
   (a) By dividing the arc *RT* into three equal parts by drawing arcs with centers *R* and T and radius equal to *RS.*
   (b) By dividing each of these parts into three equal parts by trial and error method.
5. Each of the nine equal parts subtends an angle of 10° at the centre S.
6. Transfer each division-point from the arc to the straight line RS-produced, by taking R as centre and radii equal to chords R -10, R -20 and so on.
7. Construct linear scale by drawing rectangle and mark the divisions with zero below R and number the divisions subsequently as 10°, 20°, 30° and so on as shown in figure 1.17.

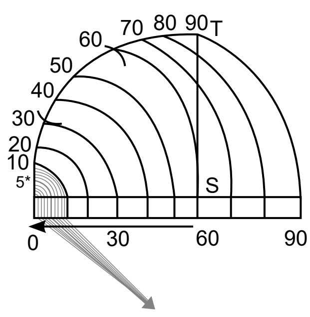


Figure 1.17: Scale of chords

Procedure to construct an angle (33°) using the scale of chords:

1. Draw a line *PQ* equal to *RS i.e. a.*
2. Draw the arc QM of radius a and centre at P.
3. On the arc QM mark the point K at a distance equal to b from Q, the distance corresponding to 33^o as shown in Fig. 1.18.
4. Join *P* and *K,* and the angle *KPQ* is equal to 33^o.

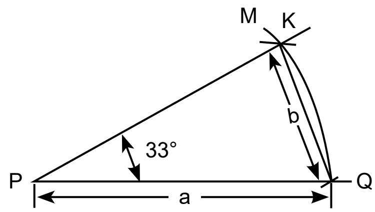


Figure 1.18: Construction of an Angle by Scale of Chords

SOLVED EXAMPLES

Example 1.1: Construct a plain scale of representative fraction (R.F.) = 1/5 to show centimeters and decimeters.

Solution:

1. Calculate the length of scale.
   Length of scale = R F × maximum length of scale
   L = 1/5 × 10 × 10 (1m=10dm, 1dm=10 cm) = 20 cm (Length of line to be drawn on sheet)
2. Draw a line of 20 centimeter and divide it into 10 equal parts each representing 1 decimeter.
3. Put zero at the end of first main division and mark 1, 2, 3, 4, 5, 6, 7, 8 and 9 at each subsequent division to the right.
4. Divide first division into 10 equal parts each representing 1 centimeter.
5. Write centimeters on left and decimeters on right.
6. To mark distance of 7.6 decimeters take 7 decimeters from main scale and 6 subdivisions equal to .6 decimeters or 6 centimeters from the scale made on left side of zero mark as shown in figure 1.19.

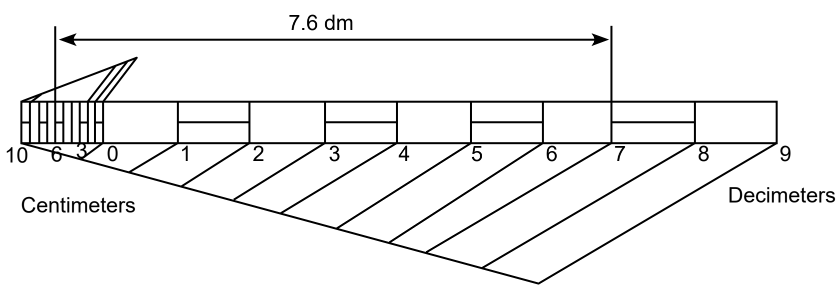


Figure 1.19: Plain Scale R.F. 1/5

Example 1.2: Construct a diagonal scale to show meters, decimeters and centimeters. Take RF = 3:100 and measure of scale to be 5 meters. Also show the length of 3.69 m on the scale.

Solution:

1. Calculate length of scale:
   Length of scale = RF x maximum length = 3/100 × 5 × 100 (1 m=100 cm) = 15 cm
2. Draw a line of length 15 cm and divide it into 5 equal parts each representing 1 meter.
3. Divide the first division into 10 equal subdivisions. Each sub division showing 1 decimeter.
4. At the starting point, erect a perpendicular of suitable length and complete the rectangle. Divide the line drawn into 10 equal parts and draw horizontal lines along each sub division throughout the rectangle.
5. Draw perpendiculars at meter divisions 0, 1, 2, 3 and 4
6. Join the top most left point with point 9 (first sub division on horizontal scale on the left side of zero mark.
7. Draw lines parallel to line drawn in earlier step through the remaining points 8, 7, 6, 5, 4, 3, 2, 1, and 0.
8. Mark a distance of 3 meters 6 decimeters and 9 centimeters on the scale as shown in figure 1.20.

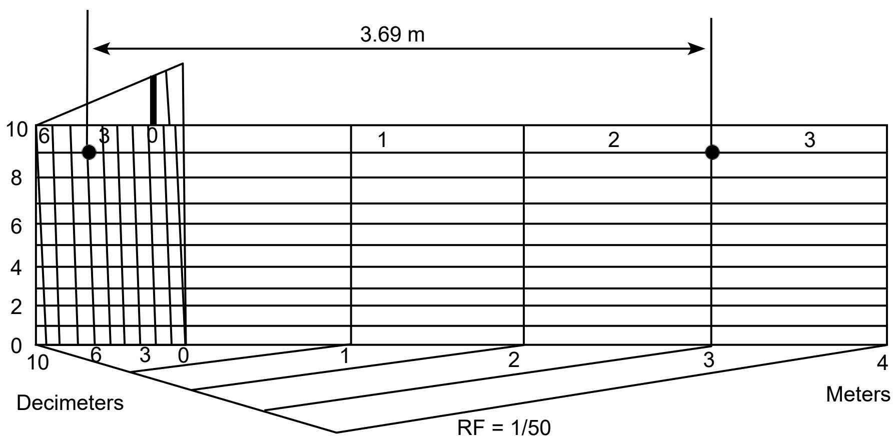


Figure 1.20: Diagonal scale R.F. 1/300

Example 1.3: Draw a vernier scale with R.F. = 1/25 which can read up to 4 meters and show decimeters, centimeters and millimeters. Mark lengths 2.33 meters on the scale.

Solution:

1. Calculate length of scale:
   Length of scale = 1/25 × 4 × 100 = 16cm (1 m=100 cm)
2. Draw a line of 16 cm and divide it into 4 equal parts where each part is equal to 1 meter.
3. Divide each of these 4 parts into 10 equal subparts to show decimeter.
4. Take 11 parts of decimeter length each equal to 1 cm on left side of zero and divide them into 10 equal parts. Each part shows length of 11 cm or 1.1 dm.

5. Complete the scale and mark 2.33 meter as shown in figure 1.21.

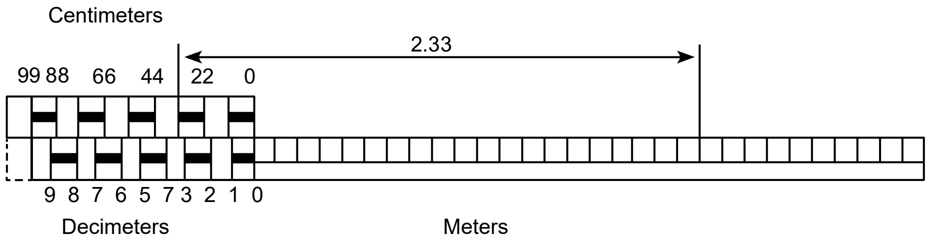


Figure 1.21: Vernier Scale with R.F. =1/25

1.4 CURVES AND CONIC SECTIONS

Curves are natural formulations of small line segments and have radius of curvature. Curves can be written in parametric form as x = x (t), y = y (t).

Conic sections are defined as the locus of a point moving in a plane such that the ratio of its distance from a fixed point and a fixed straight line is always constant. Conics are obtained by cutting a right circular cone with imaginary cutting planes in different positions. Various types of curves and conics which are commonly used in engineering practice as given below:

i. Ellipse
ii. Parabola
iii. Hyperbola
iv. Cycloids
v. Epicycloids
vi. Hypocycloids
vii. Involutes

Curves and Conic Sections

1.4.1 Ellipse

An ellipse is obtained when a section plane which is inclined to the axis, cuts all the generators of the cone on one side of the apex as shown in figure 1.22. It is also defined as a curve traced by a point, moving in a plane such that the sum of its distances from two fixed points is always the same. Use of elliptical curves is made in arches, bridges, dams, monuments, manholes, glands and stuffing-boxes.

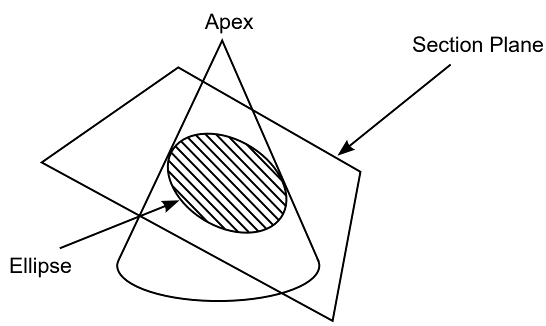


Figure 1.22: Ellipse

Methods of Generating Ellipse

Methods commonly used for generating ellipse:

1. **Focus-Directrix or Eccentricity Method:** General method of constructing ellipse when the distance of the focus from the directrix and its eccentricity are given.
2. **Concentric Method**: This method is applicable when the major axis and minor axis of an ellipse are given.
3. **Oblong Method:** This method is applicable when the major axis and minor axis or the conjugate axes with the angle between them is given.

Generation of Ellipse using Focus –Directrics or Eccentricity Method

Procedure for generating ellipse using Focus-Directrix method:

1. Draw directrix DD' and draw a perpendicular at any point A.
2. Mark focus point F at given distance CF.
3. Divide CF in 7 equal parts and mark vertex V at 4th part from C considering eccentricity as 3/4.
4. From V draw a perpendicular, VF = VA.
5. Draw a line CA such that CVA is a triangle.
6. Mark a point 1 and draw perpendicular to meet CA at 1'.
7. With F as center and 1-1' as radius, draw arcs to cut the perpendicular through 1 and mark the points as P1, P1'.
8. Repeat the process for 2, 3 and so on.
9. Join P1, P2, P3 etc to obtain smooth curve as shown in figure 1.23.

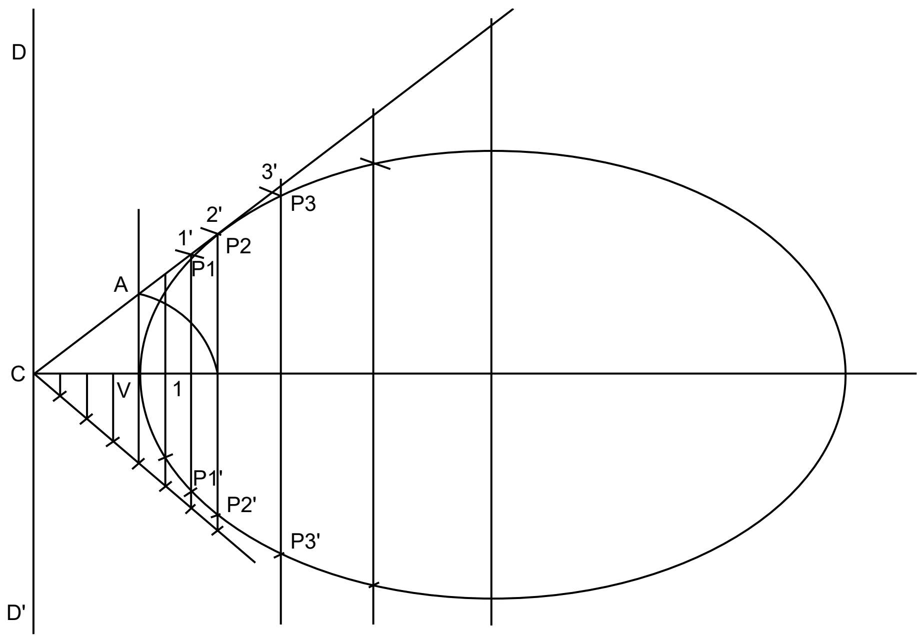


Figure 1.23: Generation of Ellipse

1.4.2 Parabola

A parabola is obtained when a section plane which is inclined to the axis but parallel to one of the generators of a cone cuts the cone as shown in figure 1.24. A parabola is also defined as locus of a point moving in a plane such that ratio of its distances from a fixed point and a fixed straight line is constant and equal to one. Use of parabolic curves is made in arches, bridges, sound reflectors and light reflectors.

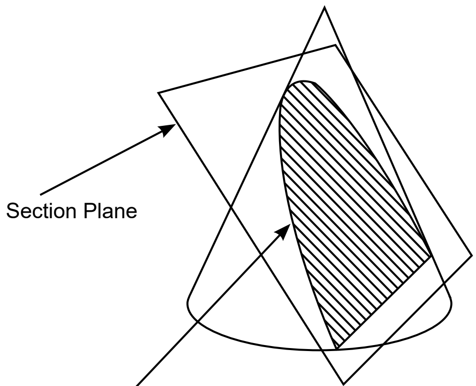


Figure 1.24: Parabola

Methods of Generating Parabola

Methods commonly used for generating parabola:

1. **Focus-Directrix or Eccentricity Method**: General method of constructing parabola when the distance of the focus from the directrix is given.
2. **Rectangle Method and Parallelogram Method**: This method is applicable when the axis (or abscissa) and the base (or double ordinate) of a parabola are given or the conjugate axes with the angle between them is given.
3. **Tangent Method:** This method is applicable when the base and the inclination of tangents at open ends of the parabola with the base are given.

Generation of Parabola using Focus–Directrics Method

Procedure for generating parabola using Focus-Directrix method:

1. Draw the directrix AB and the axis CD.
2. Mark focus F on CD at a given distance from C.
3. Bisect CF at V the vertex considering eccentricity is 1 (VF/VC = 1).
4. Mark a number of points 1, 2, 3 etc. on the axis and through them, draw perpendiculars to it.
5. With centre F and radius equal to C-1, draw arcs cutting the perpendicular through 1 at P_1 and P'_1.
6. Similarly, locate points P_2 and P'_2, P_3 and P'_3 etc. on both the sides of the axis.

7. Draw a smooth curve through these points. This curve is the required parabola. It is an open curve as shown in figure 1.25.

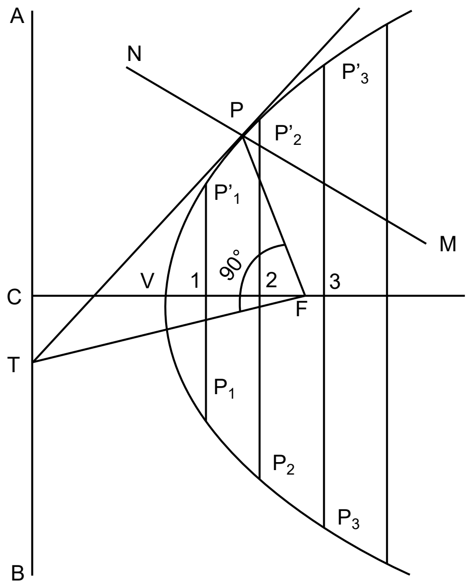


Figure 1.25: Generation of Parabola

1.4.3 Hyperbola

A hyperbola is obtained when a section plane which makes a smaller angle with the axis than the one made by generator of a cone as shown in figure 1.26. It is a curve traced out by a point moving in such a way that the product of its distances from two fixed lines at right angles to each other is a constant. The fixed lines are called asymptotes. A **rectangular hyperbola** for which the asymptotes are perpendicular, also called an equilateral hyperbola or right hyperbola. This occurs when the semi major and semi minor axes are equal.

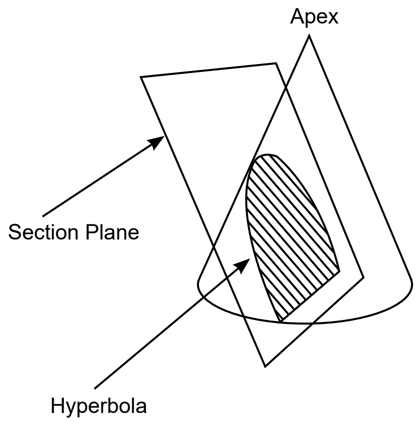


Figure 1.26: Hyperbola

Methods of Generating Hyperbola

Methods commonly used for generating hyperbola:

1. **Focus-Directrix Or Eccentricity Method**: General method of constructing hyperbola when the distance of the focus from the directrix is given
2. **Rectangle or Ordinate-Abscissa Method**: This method is applicable when the axis (or abscissa) and the base (or double ordinate) of a hyperbola are given.

Generation of Hyperbola using Focus–Directrics Method

Procedure for generating hyperbola using Focus-Directrix method:

1. Draw the directrix AB and the axis CD.
2. Mark the focus F on CD at a given distance from C.
3. Divide CF into 5 equal divisions and mark V the vertex, on the second division from C. Thus, eccentricity = VF / VC = 3 /2

   To construct the scale for the ratio 3/2 draw a line VE perpendicular to CD such that VE = VF. Join C with E. Thus, in triangle CVE, VE/VC = VF / VC =3/2
4. Mark any point 1 on the axis and through it draw a perpendicular to meet CE-produced at 1.
5. With centre F and radius equal to 1-1', draw arcs intersecting the perpendicular through 1 at P1 and P'1.
6. Similarly, mark the number of points 2, 3 etc., and obtain points P2 and P'2, P3 and P'3 etc.
7. Draw the hyperbola through these points as shown in figure 1.27.

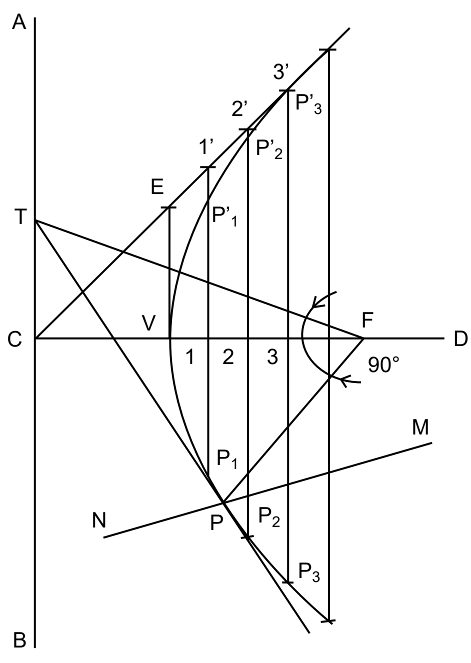


Figure 1.27: Generation of Hyperbola

1.4.4 Cycloid

A cycloid is a curve generated by a fixed point on the circumference of the circle as the circle rolls along a straight line without slipping. The moving circle is called the "Generating circle" and the straight line is called the "Directing line" or the "Base line". The point on the Generating circle which generates the curve is called the "Generating point".

Following are the steps used to generate a cycloid for the given diameter (D) of generating circle:

1. Draw a generating circle of diameter D and a base line equal to circumference of the generating circle.
2. Divide the circle and the base line in to equal number of parts.
3. Through divisions on circle draw perpendiculars to intersect the line O-C12 at C1, C2 and so on.
4. Set the compass the radius of the circle and centers as C1, C2, C3-C12 to cut the arcs on the lines from circle through 1, 2, 3 etc.
5. Join the points to form the cycloid as shown in figure 1.28.

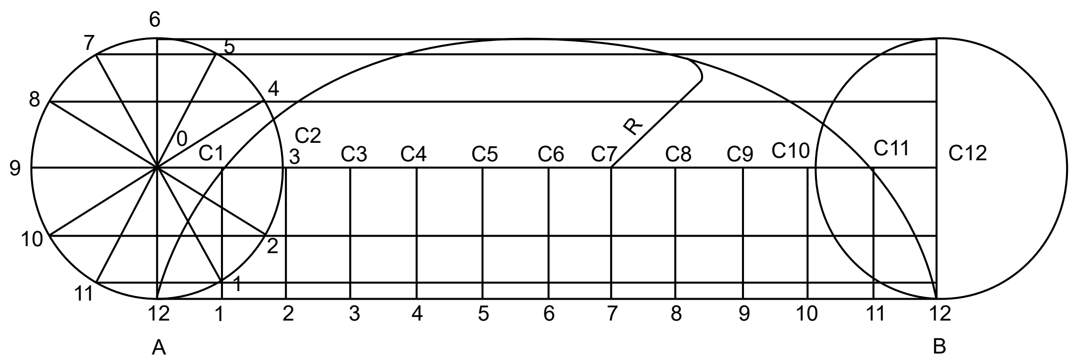


Figure 1.28: Generation of Cycloid

> **Important Observation**
> Cycloid curves are generally used for tooth profile of gears.

1.4.5 Epicycloids

The curve is called epicycloids when the generating circle rolls along the circumference of another circle outside it.

Procedure for generate epicycloids for the given radius (r) of the generating circle and radius (R) of directing circle:

1. With O as centre and radius OP (base circle radius), draw an arc PQ, the included angle $\theta = (r/R) \times 360°$.
2. With O as centre and OC as radius, draw an arc to represent locus of centre. Divide arc PQ in to 12 equal parts and name them as 1', 2', 12'.
3. Join O1', O2' … and produce them to cut the locus of centers at C1, C2 ….C12.
4. Taking C1 as centre and radius equal to r, draw an arc cutting the arc through 1 at P1.
5. Taking C2 as centre and with the same radius, draw an arc cutting the arc through 2 at P2, similarly obtain points P3, P4, …., P12.

6. Draw a smooth curve passing through P1, P2….. , P12, which is the required epicycloids as shown in figure 1.29.

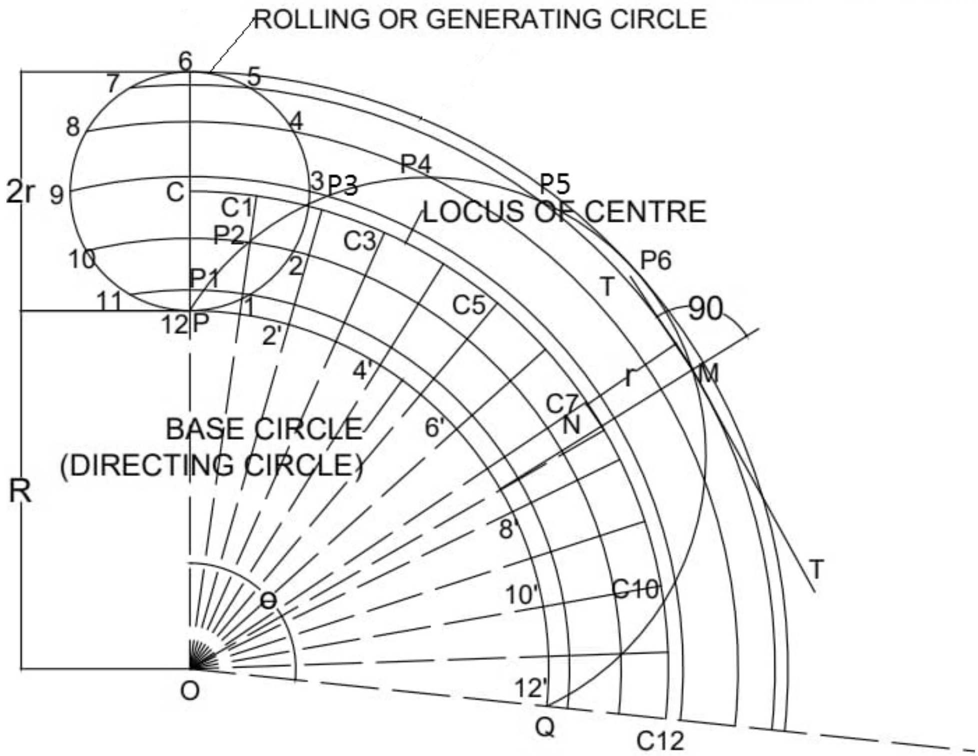


Figure 1.29: Generation of Epicycloids

1.4.6 Hypocycloid

Hypocycloid is obtained when the generating circle rolls along the circumference of another circle but inside it.

Procedure for generation of a hypocycloid of the radius of generating circle is r and directing circle radius is R.

1. Draw a circle of radius r and divide the circle in 12 equal divisions.
2. Calculate $\theta = r/R * 360^o$
3. Take a point O' as center and radius R mm; draw an arc PQ subtending angle θ at O'.
4. On O'P mark PC1 radius of rolling circle. With O' as center and radius r draw the rolling circle.
5. Divide the rolling and generating circles into 12 equal parts.
6. With C1, C2 etc. as centers and radius equal to r, draw arcs to intersect the arcs made with O' as center through 7-12.
7. Draw a smooth curve by joining points as shown in figure 1.30.

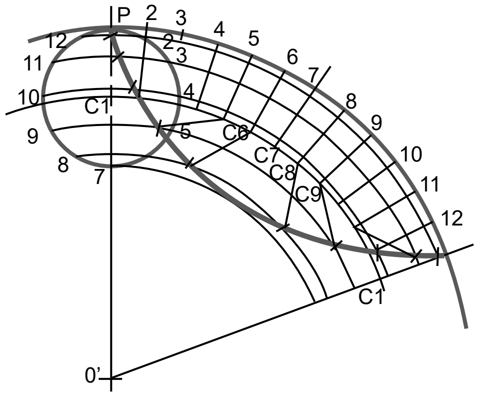


Figure 1.30 Construction of Hypocycloid

1.4.7 Involutes

An involute is defined as the path of a point on a straight line which rolls without slipping along the circumference of a cylinder. It can also be defined as a curve traced by the free end of a thread unwound from a circle or a polygon in such a way that the thread is always tight and tangential to the circle or side of the polygon.

Following are the steps for generation of Involutes of a circle of diameter D:

1. Draw a circle of diameter D.
2. Divide the circle into 12 equal parts and mark as 1, 2, 3, etc., in clockwise direction starting from a point next to the bottom most one.
3. Mark the centre point of the circle as O.
4. Draw a tangent AC from point 12 for a length of L = πd, (d – diameter of circle).
5. Divide AC into 12 equal points and the name points 1', 2', 3'...etc.,
6. Draw tangents from 1, 2, 3, etc., as shown in figure.
7. With 11-11' as radius 11 as centre cut an arc on the tangent drawn from 11 and name the point as P11, similarly obtain other points P9, P10, ..etc.
8. Join all the points by a smooth curve to obtain involutes as shown in figure 1.31.

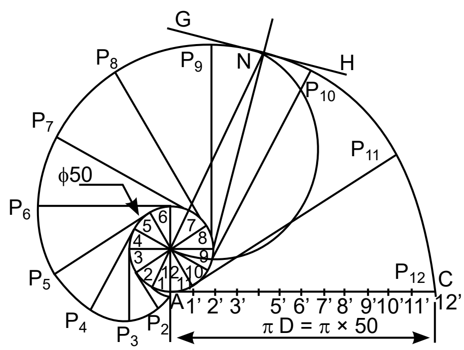


Figure 1.31: Generation of Involutes

UNIT SUMMARY

- The engineering drawings depict lot of information such as the shape and size of an object, its dimensions, scale, material details, date and name of person who made the drawing.
- Drawings have been used as major mode of communication by marking message on sand or scratching drawings of animals, trees, and people on rock walls of caves.
- The first important step in the development of computer-aided drafting was made at the Massachusetts Institute of Technology (MIT) in 1963 where a system called Sketchpad was developed and demonstrated.
- Drawing sheet is the medium on which drawings are prepared by means of pencils or pen.
- Engineering students generally use A2 or A3 size drawing sheets.
- A engineering drawing board is made of four to six strips of well seasoned soft wood such as pine, fir or oak of thickness about 25 mm.
- Mini drafter is a device used to draw parallel or inclined lines very effectively with ease.
- T-Square is a straight edge with a short, thick crosspiece that fits over the edge of a drawing board, used primarily to draw horizontal lines.
- Lettering is used for writing of titles, sub-titles, dimensions, scales and other details on a drawing.
- In Single-stroke vertical lettering the alphabets made are of same thickness.
- Inclined or italic letters are inclined at 75’ to the horizontal.
- The lettering in which all the letters are formed by thick and thin elements is called Roman Lettering.
- The ratio of the dimension of the object shown on the drawing to its actual size is called the Representative Fraction (RF).
- A plain scale represents either two units or only one unit and its fraction.
- A diagonal scale consists of a set of parallel lines crossing obliquely with other set of lines such that their intersections create subdivisions. It is mainly used when measurements are required in three units such as cm, m and km.
- Vernier scale is used to measure fractional parts of the smallest division of the main scale.
- Comparative scale or corresponding scales consists of two scale of same RF but graduated to read different units.
- Conic is defined as the locus of a point moving in a plane such that the ratio of its distance from a fixed point and a fixed straight line is always constant.

EXERCISES

1.1 Multiple Choice Questions

1. The following is used to draw curves which are not circular.
   (a) Compass (b) Protractor
   (c) French curves (d) Pro circle

2. The areas of the two subsequent sizes of drawing sheet are in the ratio
   (a) 1:5 (b) 1:4
   (c) 1:2 (d) 1:10
3. The following is not included in title block of drawing sheet.
   (a) Sheet No (b) Scale
   (c) Method of projection (d) Size of sheet
4. Representative Fraction is the
   (a) ratio of the length in drawing to the actual length
   (b) ratio of the actual length to the length in drawing
   (c) reciprocal of actual length
   (d) square of the length in drawing
5. The length of the object in drawing is 50 mm; the scale is given as 1:5. Find the actual length of the object.
   (a) 50 cm (b) 10 cm
   (c) 25 cm (d) 20 cm
6. Eccentricity of,
   (a) Parabola is equal to one
   (b) Hyperbola is greater than one
   (c) Ellipse is less than one
   (d) All of the above
7. What is the type of scale in which the representative fraction is 1:1?
   (a) Enlarged scale (b) Reduced scale
   (c) Full size scale (d) Graphical scale
8. The graduations of which scale is derived from the primary scale?
   (a) Comparative scale (b) Vernier scale
   (c) Plain scale (d) Diagonal scale
9. What is the length of the scale, the representative fraction is 1:50000 and the scale must measure up to 25 km?
   (a) 0.5 cm (b) 50 cm
   (c) 5 cm (d) 0.5 cm
10. Which of the following is used to set or measure angles when a protractor is not available?
    (a) Plain scale (b) Diagonal scale
    (c) Scale of chords (d) Comparative scale
11. When measurement is required in three consecutive units, the appropriate scale is :
    (a) Plane scale (b) Diagonal scale
    (c) Isometric scale (d) Scale of chords
12. What type of curve is created by the intersection of a plane parallel to side of cone?
    (a) Parabola (b) Hyperbola
    (c) Ellipse (d) Cycloid

13. The curve generated by a point on the circumferences of a circle, which rolls without slipping along outside of another circle is known as:
    (a) Hypocycloid (b) Epicycloid
    (c) Cycloid (d) Torchoid
14. Which set of lead grades has a grade out of sequence?
    (a) H, HB, B, 3B (b) 7B, H, F, 3H
    (c) 6B, B, H, 4H (d) 9H, HB, B, 2B
15. Which of the following constructions use parabolic curves?
    (a) Cooling towers (b) Water channels
    (c) Light reflectors (d) Man-holes

Answers Keys (Exercise 1.1)

1. (c), 2. (c), 3. (d), 4. (a), 5. (c), 6. (d), 7. (c), 8. (b),
9. (b), 10. (c), 11. (b), 12. (a), 13. (b), 14. (b), 15. (c)

1.2 Short and Long Answer Type Questions

Category I

1. Define engineering drawing and why it's called universal language of engineers?
2. Discuss the different types of drawing.
3. Name different drawing instruments.
4. What are the standard sizes of drawing sheets according to BIS?
5. Define layout of a drawing sheet.
6. List the content of a title block.
7. What is lettering? What are the main requirements of lettering?
8. Explain different styles of letters used in engineering drawing.
9. Write the following in single stroke vertical capital letters 20 mm height.
   (a) Drawing is the Language of Engineers
   (b) All India Council for Technical Education
10. What is a scale? Why are they used in engineering drawing?
11. What are the different types of scale?
12. Explain representative fraction?
13. Differentiate plain scale and diagonal scale?
14. Define involutes and cycloids.
15. Differentiate epicycloids and hypocycloids.

Category II

16. Explain the principal of construction of diagonal scale with neat sketch.
17. Explain the principal of construction of vernier scale with neat sketch.
18. Generate and draw an epicycloids for the given radius (r) of the generating circle and radius (R) of directing circle:

19. Generate and draw a cycloid for the given diameter (D) of generating circle.
20. Generate and draw an involute of a circle of diameter D:

1.3 Practical Problems

1. The actual length of 500 m is represented by a line of 15 cm on a drawing. Construct a vernier scale to read up to 600 m. Mark on the scale a length of 549 m.
2. A car is running at a speed of 50 km/hour. Construct a diagonal scale to show 1 km by 3 cm and to measure up to 6 km. Mark also on the scale the distance covered by the car in 5 minutes 28 seconds.
3. Construct the angles of 45° and 120° using the scale of chords.
4. Construct an ellipse when distance of focus from directrix is 60 mm and eccentricity is 3/4.
5. Construct a parabola, when the distance of the Focus from the directrix is 50 mm.
6. Construct a hyperbola, when the distance of the Focus from the directrix is 55 mm.
7. Draw a hypocycloid if the radius of generating circle is 20 mm and directing circle is 70 mm.
8. Draw a cycloid if the diameter of rolling circle is 30 mm.
9. Draw involutes of a circle of 40 mm diameter.
10. A thin circular disc of 60 mm diameter is allowed to roll without slipping from upper edge of sloping plank which is inclined at 18° with the horizontal plane. Draw the curve traced by the point on the circumference of the disc.

KNOW MORE

Types of Lines Used in Engineering Drawing

For general engineering drawings, the types of lines recommended by the Bureau of Indian Standards must be used. Table 1.5 shows the different types of lines and their application in engineering drawing:

Table 1.5: Types of lines

Type	Line	Description	Application
A		Continuous THICK	Visible outlines, Visible edges.
B		Continuous THIN	Demension line, Projection lines, Leader line, Imaginary lines of intersections, Outlines of revolved sections.
C		Continuous THIN Freehand	Boundaries of Limits of Partial or Interrupted views.
D		Continuous THIN Zig-Zag	Long break line
E	------------------	Dashed THICK	Hidden outlies, Hidden edges.
F	....	Dashed THIN	Centre line, Lines of Symmetry Trajectories
G	- - - - - - - - - - - - - -	Chain THIN	

Type	Line	Description	Application
H		Chain THIN and THICK at ends & Changes of Direction	Cutting Planes
I		Chain THICK	Indication of lines or surfaces to which special treatment required
J		Chain THIN Double Dash	Outlines of adjacent parts, Alternate and Extreme positions of movable parts, Centroridal lines, Initial outlines, Prior to formaing.

DESIGN PROJECT/ACTIVITIES

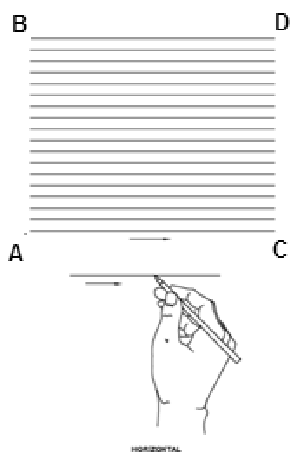
- **Activity:** Sketch by free hand horizontal thick and thin lines:
- **Concept:** Free Hand Sketching is such a drawing which is drawn without measuring instruments. This drawing is drawn with the help of pencil and eraser only.

Procedure:

1. Sketch two vertical thin guide lines AB & CD.
2. Mark points on the vertical lines AB & CD, 5 mm intervals approximately.
3. Draw the lines by free hand between the two points sketch thick and thin alternatively

Technique:

1. Lengthy lines can be drawn with the forearm motion and short lines are drawn with the wrist motion.
2. Keep uniform pressure while sketching.
3. Horizontal lines are drawn from left to right as shown in below figure.
4. While sketching straight lines between two points keep your eyes on the point to which the line is to go rather than the point of pencil.
5. Avoid of drawing whole length of line in one single stroke.
6. Prevent using eraser often.



INTERESTING FACTS

- Complex technical drawings were made in renaissance period (1300-1500), such as the drawings of Leonardo da Vinci.
- Modern engineering drawing, with its precise conventions of orthographic projection developed in france during Industrial Revolution (1770-1850).
- In 1960, Ivan Sutherland MIT's Lincoln Laboratory created SKETCHPAD, which demonstrated the basic principles and feasibility of computer technical drawing.

APPLICATIONS (REAL LIFE / INDUSTRIAL)

Engineering drawing is the preferred method of drafting in all engineering fields including, but not limited to, civil engineering, electrical engineering, mechanical engineering and architecture.

Technical drawings are used in many different applications. They are needed in any setup, which involves design, and in any subsequent forms of conversion process. The most common applications of technical drawings can be found in the fields of manufacturing, engineering and construction.

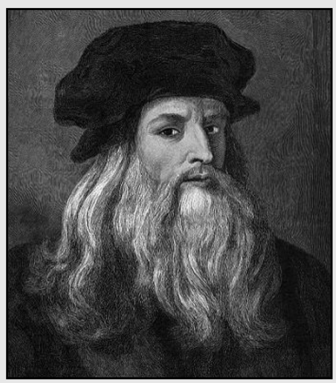
INQUISITIVENESS AND CURIOSITY TOPICS

- **Artistic Drawing:** An artistic drawing range from the simplest line drawing to the most famous paintings. Regardless of their complexity, artistic drawings are used to express the feelings, beliefs, philosophies, and ideas of the artist. Artists often take an abstract approach in communicating through their drawings, which in turn gives rise to various interpretations.
- **Technical Drawing:** A technical drawing is a means of clearly and concisely communicating all of the information necessary to transform an idea or a concept in to reality. Therefore, a technical drawing often contains more than just a graphic representation of its subject; it also contains dimensions, notes and other specifications.

CASE STUDY

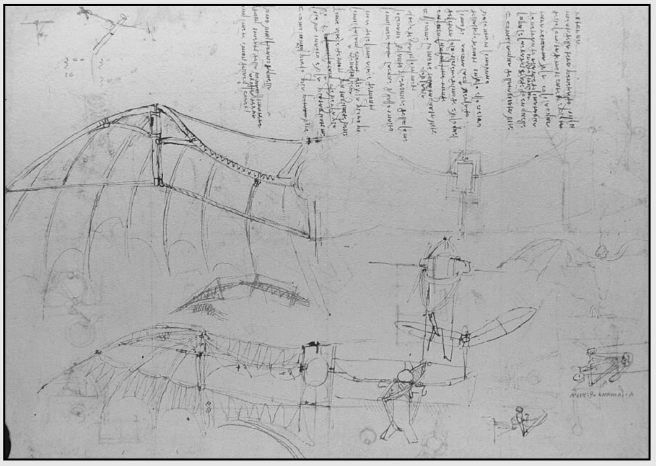
Environmental / Sustainability / Social / Ethical Issues

Flying Machine Design by Leonardo da Vinci



Leonardo da Vinci was a true genius who graced this world with his presence from April 15, 1452 to May 2, 1519. He is among the most influential artists in history, having left a significant legacy not only in the realm of art but in science as well, each discipline informing his mastery of the other. Da Vinci lived in a golden age of creativity among such contemporaries as Raphael and Michaelangelo and contributed his unique genius to virtually everything he touched. Today, no name better seems to symbolize Renaissance age than Leonardo da Vinci.

Da Vinci not only developed his skill in drawing, painting and sculpting during his apprenticeship, but through others working in and around the studio, he picked up knowledge in such diverse fields as mechanics, carpentry, metallurgy, architectural drafting and chemistry. His research tools being almost exclusively his eyes. In Leonardo's notebooks, there are many war machines, such as a vehicle propelled by two men through crankshafts. Also, there were flying machine designs and bridges projects. Thus, we can admit that the "dark ages" has lighted up the technical drawing study. Nowadays, it is a little difficult to imagine how they did these work by hand- made production, using drawing instruments, which were early versions of the ruler, set square, compass, and protractor.



Courtesy of LeonardoDaVinci.net

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

1. Engineering Drawing Practice for Schools and Colleges, Bureau of Indian standards Manak Bhavan, 9 Bahadur Shah Zafar Marg New Delhi 110002.
2. Engineering Drawing, National Instructional Media Institute, Chennai, Directorate General of Training Ministry of Skill Development & Entrepreneurship government of india.
3. https://nptel.ac.in/courses/112/105/112105294/
4. https://www.youtube.com/watch?v=_M4U0d56dr8
5. https://youtu.be/qkPZgVbtiHE
6. https://youtu.be/bOtL_H3ma7I
7. https://youtu.be/_PHul1YFxrI

2 Orthographic Projection

UNIT SPECIFIC

This unit "Orthographic Projection" covering, principles of orthographic projections-conventions - projections of points and lines inclined to both the planes; projections of perpendiculars and oblique planes.

RATIONALE

Learning of orthographic projection enables students and engineers to create and read working drawings. Theory of orthographic projection is used to design various manufacturing components and buildings.

PREREQUISITE

Basic knowledge of preparing engineering drawings using prevailing drawing standards and instruments.

UNIT OUTCOMES

Upon successful completion of this module, the students will be able to:

U2-O1: Draw orthographic projections of basic objects.
U2-O2: Understand the methods of projection.
U2-O3: Construct projection of lines.
U2-O4: Construct projection of planes.

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-2 Outcomes	Expected Mapping With Course Outcomes *(1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)*				
	CO-1	**CO-2**	**CO-3**	**CO-4**	**CO-5**
U2-O1	2	3			2
U2-O2	2	3			2
U2-O3	2	3			2
U2-O4	2	3			2

2.1 THEORY OF PROJECTION

Projecting the image of an object to the plane of projection is known as projection. To represent an object, straight lines from various points on the contour of object are drawn on the paper, and the image formed by joining the points in the right order is called *projection* of the object.

A line of sight (LOS) is an imaginary ray of light between an observer and an object also known as projectors. A plane of projection is an imaginary plane upon which the image created, by the line of sight, is projected. Figure 2.1 shows the theory of projection.

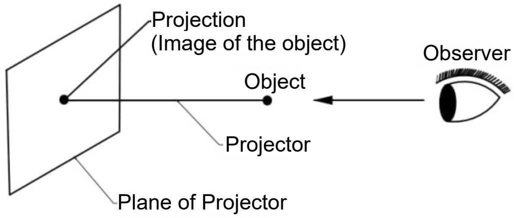


Figure 2.1: Theory of Projection

2.2 TYPES OF PROJECTION

The two broad types of projections, both with several sub classifications, are parallel projection and perspective projection as shown in figure 2.2.

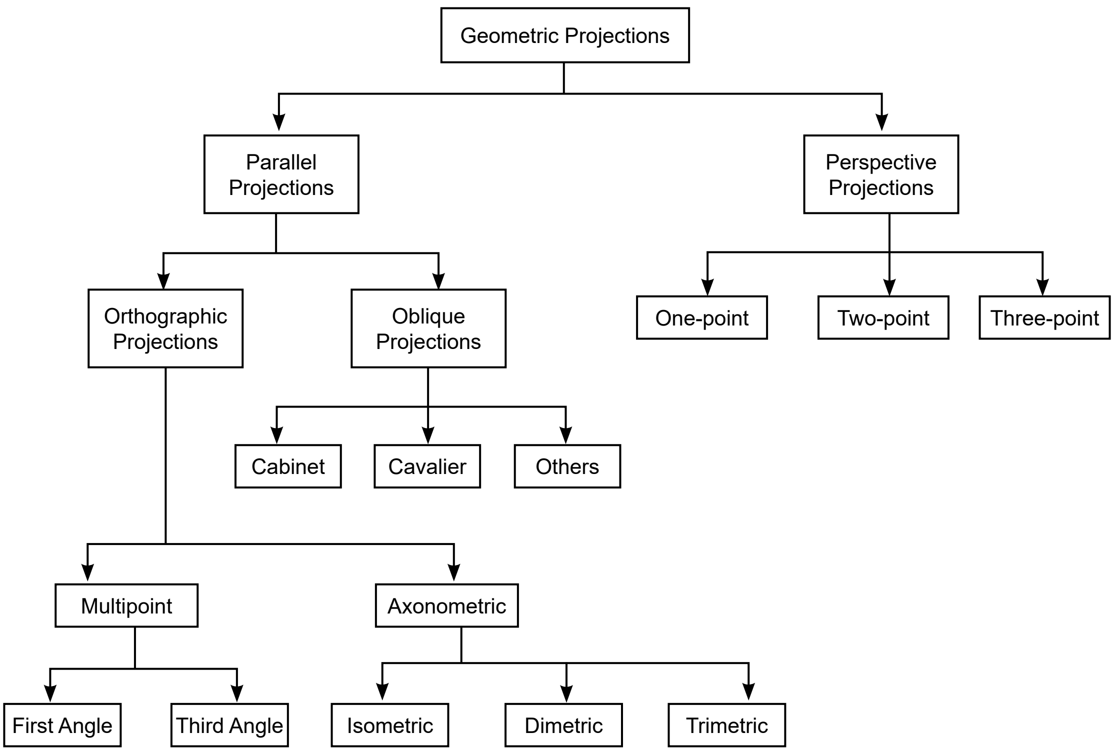


Figure 2.2: Types of Projection

2.3 PARALLEL PROJECTION

Parallel projection is a type of projection where the line of sight (LOS) or projectors are parallel and perpendicular to the plane of projection. Parallel projections are divided into the following two categories:

1. Orthographic projection
2. Oblique projection

Orthographic Projections

Orthographic projection is a method of representing the exact or true shape of an object on a sheet of paper, in two or more views. Orthographic projections are further classified as follows:

1. **Multiview projection:** This is defined as an orthographic projection method in which the object is behind the projection plane and is oriented in such a manner that only two of its dimensions are visible.
2. **Axonometric projection:** This is defined as a parallel projection method used to create a pictorial drawing of an object, by rotating the object along one or more of its axes relative to the plane of projection.

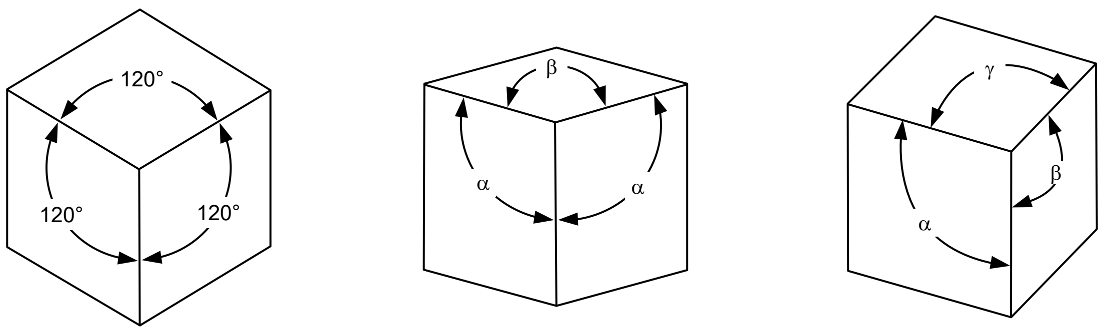
   Axonometric projections are of three types: isometric, dimetric and trimetric projection defined as follows:

   a. **Isometric projection:** This is the most commonly used form of axonometric projection in engineering drawing. Here all three angles are equal as shown in figure 2.3(a). The isometric projection is the least pleasing to the eye, but is the easiest to draw.

   b. **Dimetric projection:** As shown in figure 2.3(b), when two of the three angles are equal, the projection is classified as a dimetric projection. Dimetric projection is less pleasing to the eye, but is easier to draw than trimetric projection.

   c. **Trimetric projection:** In this type of projection the direction of viewing is such that all three axes of space appear unequally foreshortened as shown in figure 2.3(c). The scale along each of the three axes and the angles among them are determined separately as dictated by the angle of viewing. Trimetric projections are difficult to draw and are rarely used.

Orthographic Projections



(a) (b) (c)

Figure 2.3: Isometric, Dimetric and Trimetric Projections

Oblique Projections

Oblique projections are obtained when one face of the object is parallel and an adjacent face is inclined at 45° to the plane of projection. Oblique projections are classified as cavalier and cabinet projection as follows:

(a) **Cavalier projection:** In this projection method the projection lines or projectors make an angle of 45° with the projection plane and all lines (including receding lines) are made to their true length (TL) as shown in figure 2.4(a).

(b) **Cabinet projection:** In this projection method the receding lines are shortened by one-half of true length (TL) to compensate for distortion and to approximate closer to what the human eye would see, as shown in figure 2.4(b).

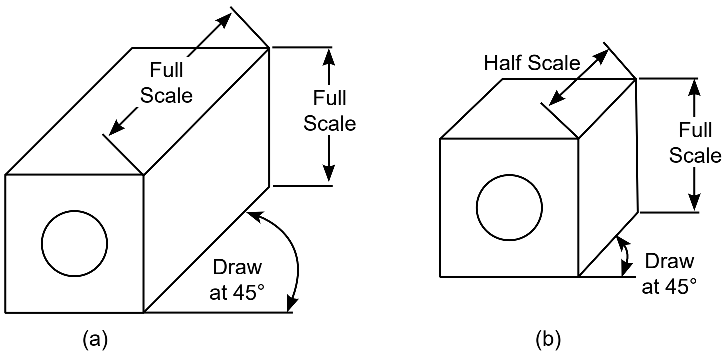


Figure 2.4: Oblique Projections

2.4 PERSPECTIVE PROJECTION

Perspective projections are drawings which attempt to replicate what the human eye actually sees when it views an object. In perspective projection, the observer is assumed to be situated at a definite position from the object. Figure 2.5 shows the theory of perspective projection of an object on a picture plane.

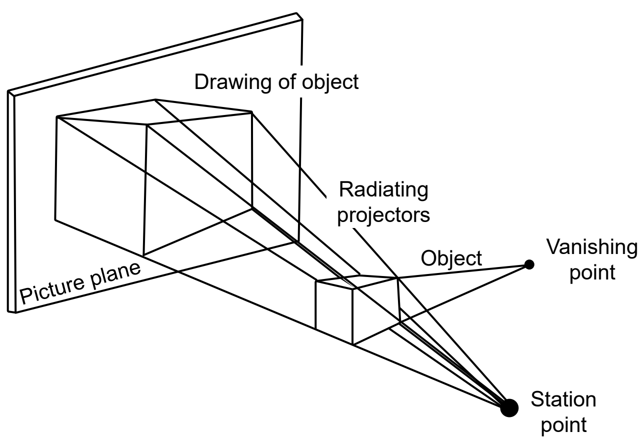


Figure 2.5: Theory of Perspective Projection

2.5 DIFFERENCE BETWEEN PARALLEL AND PERSPECTIVE PROJECTION

The difference between parallel and perspective projection are given in table 2.1:

Table 2.1: Difference between parallel and perspective projection

S. No.	Parallel Projection	Perspective projection
1.	The distance of the object from the center of projection is infinite.	The distance of the object from the center of projection is finite.
2.	Parallel projection does not form a realistic view of object.	Perspective projection forms a realistic picture of the object.
3.	Parallel projection can give an accurate view of object.	Perspective projection is not accurate as parallel projection.
4.	Projector is parallel.	Projector is not parallel.
5.	Parallel projection can preserve the relative proportion of an object.	Perspective projection cannot preserve the relative proportion of an object.

2.6 ORTHOGRAPHIC MULTI VIEW PROJECTION

Orthographic multi-view projection is defined as a method of projection in which different views of an object are projected on different reference planes, while ensuring to remain perpendicular to the respective plane. Figure 2.6 shows the three views of an object, top view, front view, and side view.

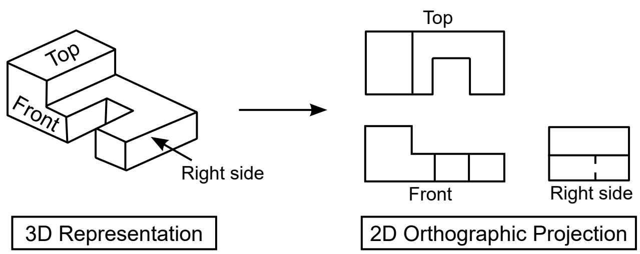


Figure 2.6: Orthographic Multi-view Projection

2.6.1 Principal Planes

The projection of one face will not provide an overall description of the object; to establish an object's true height, width and depth requires a front, top and side views, and the planes of projection which are called the principal planes of projection. The angles formed between the horizontal and the vertical planes are called the first, second, third and fourth angles as shown in the figure 2.7. The three principal (or primary) planes of projection are known as the vertical, horizontal and profile planes are defined as follows:

1. **Vertical plane:** The vertical plane of projection is the plane onto which the front view (FV) of the multi-view drawing is projected. The front view of an object shows its height and width.
2. **Horizontal plane:** The horizontal plane of projection is the plane on which the top view **(**TV**)** of the multi-view drawing is projected. The top view of an object shows the width and depth.

3. **Profile plane:** *The* profile plane of projection is the plane on which the side view of the multi-view drawing is projected. The side view of an object shows the depth and the height.

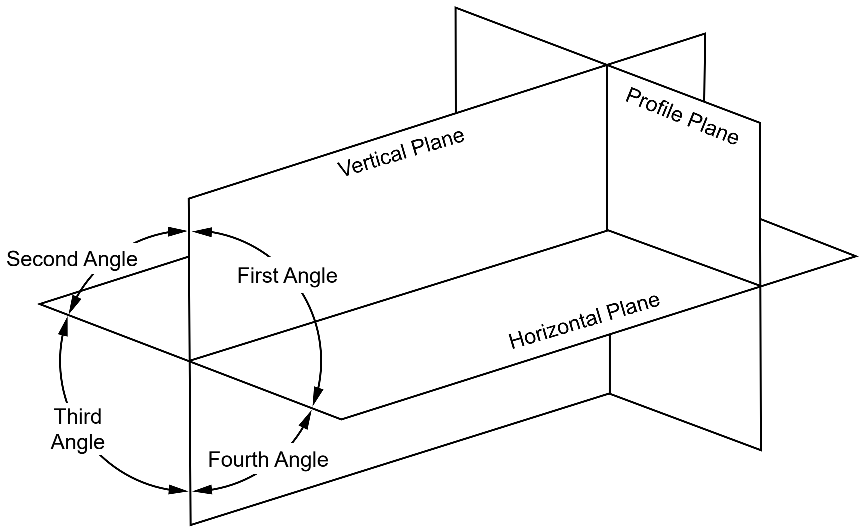


Figure 2.7: Principal Planes

2.6.2 Orthographic Projection Views

There are six types of orthographic projection views used in engineering drawing as shown in figure 2.8.

1. Front view
2. Back view
3. Top view
4. Bottom view
5. Right view
6. Left view

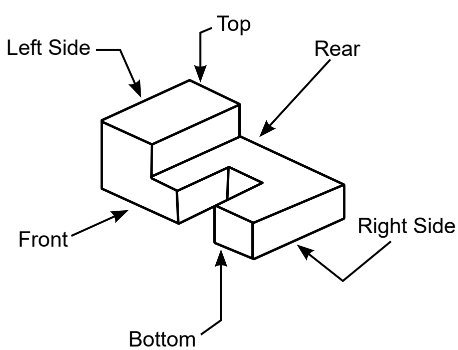


Figure 2.8: Orthographic Projection Views

2.6.3 Orthographic Projection Methods

Orthographic projection is the method of representing three-dimensional objects in two dimensions, such that the object is viewed in parallel lines that are perpendicular to the drawing plane. Instead of a single three-dimensional view, it uses multiple two-dimensional views of the same object.

There are two ways of drawing in orthographic projection -***first angle projection*** and ***third angle projection*** defined as follows:

1. **First angle projection:** In first angle projection, the object is considered to be positioned in the first quadrant. The front view of the object is obtained, by looking at the object from the right side of the quadrant and tracing it in the correct sequence and by the points of intersection between the projection plane and the rays of sight being extended. In terms of conventional representation, first angle projection is usually denoted by the symbol depicted in figure 2.9.

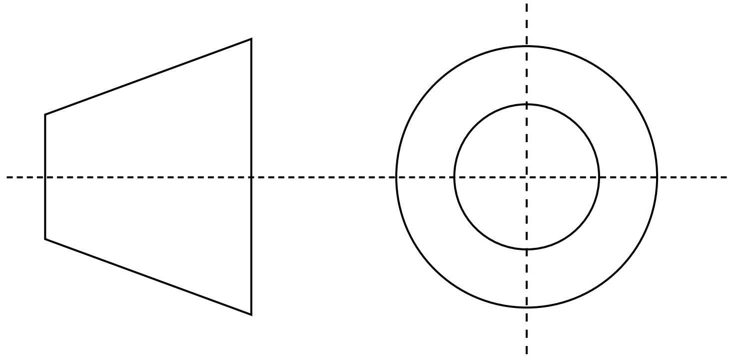


Figure 2.9: Symbol of First Angle Projection

2. **Third angle projection:** In third angle projection, the object is imagined to be positioned in the third quadrant. The plane comes between the observer and object. Since the planes are between the observer and the object, they are imagined to be transparent and the object is visible to them. A front view and top view is projected on to the vertical and horizontal planes respectively. In terms of conventional representation, third angle projection is denoted by the symbol depicted in figure 2.10.

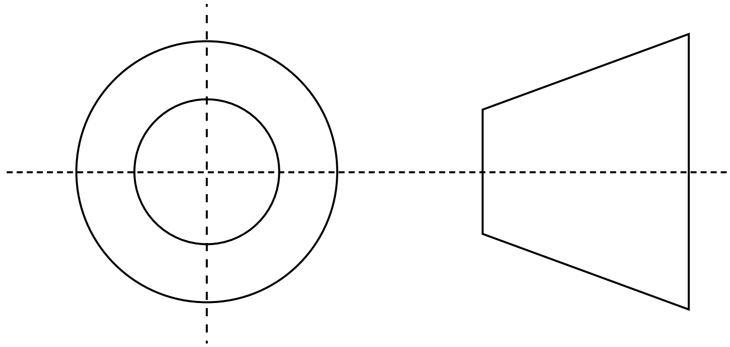


Figure 2.10: Symbol of Third Angle Projection

2.6.4 Difference between First Angle and Third Angle Projection

The difference between first angle and third angle projection are given in table 2.2:

Table 2.2: Difference between first angle and third angle projection

S. No.	First Angle Projection	Third Angle Projection
1.	The object is assumed to be situated in first quadrant.	The object is assumed to be situated in third quadrant.
2.	Object is places in between the observer & the plane.	Planes are transparent and between the observer and object.
3.	The bottom view is placed above the front view.	The bottom view is placed below the front view.
4.	The top view is placed below the front view.	The top view is placed above the front view.
5.	The left side view is placed at the right of the front view	The left side view is placed at the left of the front view
6.	The right side view is placed at the left of the front view	The right side view is placed at the right of the front view

Important Observations

First angle projection is used in USA, whereas Third angle projection is used in European countries. Bureau of Indian Standards has recommended first angle projection method in Indian institutes and organizations. In this entire course, first angle projection method is used.

2.7 PROJECTION OF POINTS

A point can be represented by small dot situated in space at any position but it has no magnitude. The position of a point in engineering drawing is defined with respect to its distance from the three principal planes. Projections of a point are obtained by extending projectors; perpendicular to principal planes and the horizontal plane (HP) is rotated clockwise until the VP and HP come into the same line. Projection on the HP, VP and profile plane is called the top view, front view and side view of a point respectively. The following conventions are used to draw projections of a point:

- Capital letters A, B, C etc., are used to represent objects in space.
- The top views are represented by lower case letters a, b, c etc.
- The front views are represented by small letters with apostrophe as, a', b', c' etc.
- Projections in top and front views are drawn using thick lines, construction lines and projection lines are drawn using thin lines.
- In engineering drawing practice, the top view is also known as plan and the front view is known as elevation.

2.7.1 Projection of a Point in First Quadrant

The following steps are used to obtain projection of a point (A) situated in the first quadrant as shown in figure 2.11:

1. Consider a point (A) placed in the first quadrant at a distance of h mm above the HP and at a distance of d mm in front of the VP.
2. Front view (a') is projected onto the VP and the top view (a) is projected onto the HP.

3. HP is rotated in a clockwise direction for 90^0 with reference to line XY.
4. Mark a point (a') at a height h mm above XY and a point (a) at a distance d mm below XY.
5. The projector joining (a') and (a) is always perpendicular to XY.

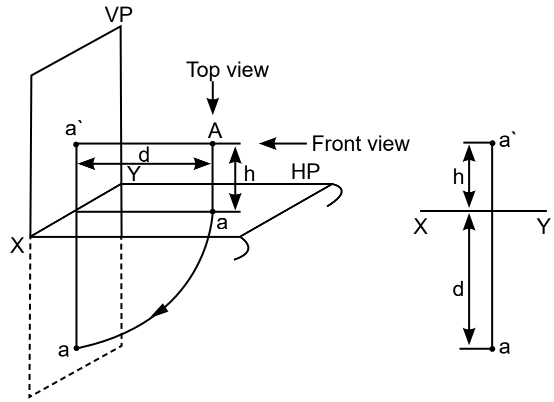


Figure 2.11: Projection of a Point in First Quadrant

2.7.2 Projection of a Point in Third Quadrant

The following steps are used to obtain the projection of a point (C) situated in the third quadrant as shown in figure 2.12:

1. Consider a point (C) placed in the third quadrant at a distance h mm below the HP and a distance d mm behind the VP.
2. The front view (c') is projected onto the VP and the top view (c) is projected onto the HP.
3. The HP is rotated in a clockwise direction for 90^0 and is obtained in a vertical position.
4. It is drawn with reference to line XY. Mark a point (c') at a height h mm below XY and (c) at a distance d mm above XY.

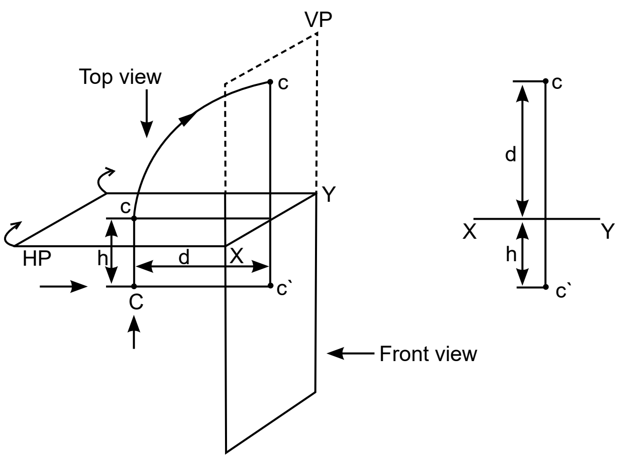


Figure 2.12: Projection of a Point in Third Quadrant

SOLVED EXAMPLES

Example 2.1: Draw the projections of a point 'A' 25 mm above the HP and 35 mm in front of the VP.

Solution: Following steps are used to draw projection of a given point 'A' as shown in figure 2.13:

1. Draw a point A at a distance of 25 mm from HP and 35 mm from VP.
2. Look for the projections of point from top and the front.
3. The top view (a) is seen on the HP and the front view (a') on the VP.
4. To bring both the projections on same surface HP is rotated by 90° clockwise direction.
5. Along the reference axis XY a' is plotted above XY at a distance of 25 mm and (a) is plotted below XY at a distance of 35 mm in the same line.

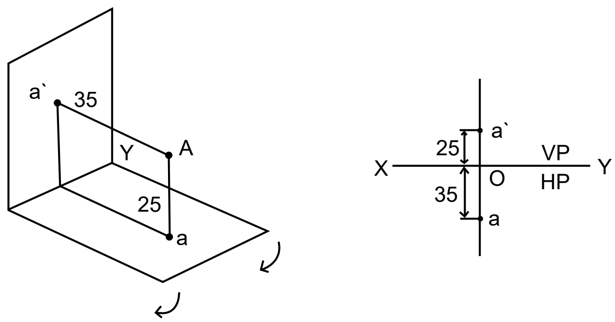


Figure 2.13: Projection of Point A

Example 2.2: Draw the projections of a point 'P' 10 mm above the H.P. and 30 mm behind the V.P.

Solution: Following steps are used to draw projection of a given point 'P' as shown in figure 2.14:

1. Draw a point P at a distance of 10 mm above the H.P. and 30 mm behind the V.P.
2. Look for the projections of point from the top and front.
3. The top view (p) is seen on the H.P. and the front view (p') on the V.P.
4. To bring both the projections on same surface H.P. is rotated 90° clockwise.
5. Along the reference axis XY both p' and p are plotted above XY at a distance of 10 mm and 30 mm respectively.

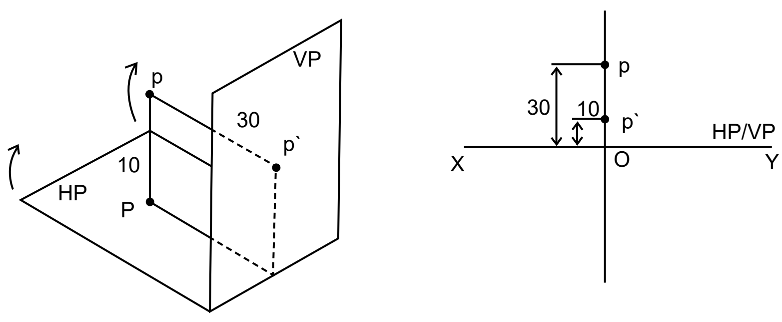


Figure 2.14: Projection of Point P

Example 2.3: Draw the projections of a point 'B' 20 mm below the H.P. and 40 mm behind the V.P.

Solution: Following steps are used to draw projection of a given point 'P' as shown in figure 2.15:

1. Draw a point B at a distance of 20 mm below H.P. and 40 mm behind V.P.
2. Look for the projections of point from the top and front.
3. The top view is seen on the H.P. and the front view on V.P.
4. To bring both the projections on same surface H.P. is rotated 90° clockwise.
5. Along the reference axis XY b' is plotted below XY at a distance of 20 mm and b is plotted above XY at a distance of 40 mm in the same line.

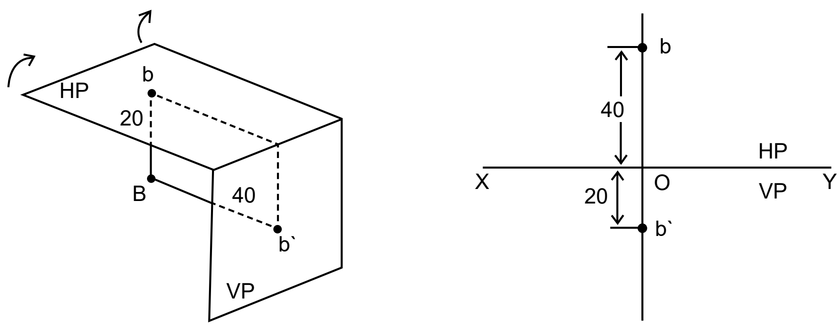


Figure 2.15: Projection of point B

Example 2.4: Draw the projections of a point 'C' 45 mm below the H.P. and 35 mm in front of the V.P.

Solution: Following steps are used to draw projection of a given point 'C' as shown in figure 2.16:

1. Draw a point C at a distance of 45 mm below the H.P. and 35 mm in front the V.P.
2. Look for the projections of point from the top and front.
3. The top view is seen on the HP and front view on the VP.
4. To bring both the projections on same surface HP is rotated 90° clockwise.
5. Along the reference axis XY c' and c are plotted below XY at a distance of 45 mm and 35 mm respectively.

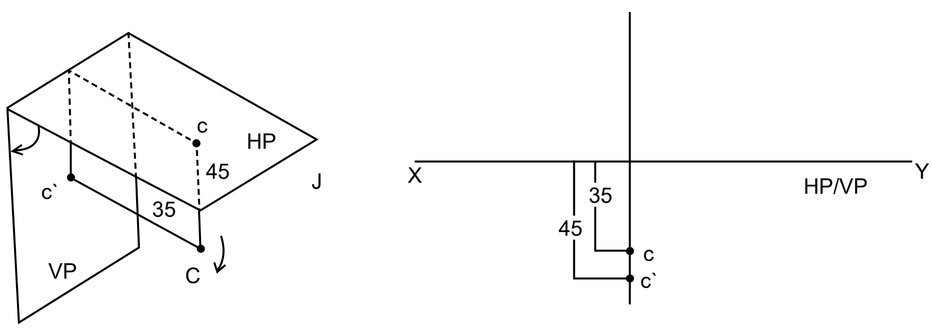


Figure 2.16: Projection of Point B

2.8 PROJECTION OF LINES

A straight line can be defined as the shortest distance between two points. The orthographic projection of a line can be obtained by projecting the end points on the principal planes of projection and then connecting these projected points. The position of a line can be fixed in space by various combinations of inputs such as distances of extreme points from the principal planes, the actual length of a line, inclinations with reference planes and distance between end projectors.

2.8.1 Projection of a Line Parallel to both the Planes

Consider a line AB kept parallel to both HP and VP as shown in figure 2.17(a). Its projections on both the planes are parallel along reference axis (XY) and are of true lengths. The HP is rotated in the clockwise direction through an angle of 90° and is obtained in the vertical position.

Followings are the steps used to draw projection with references to the line XY as given in figure 2.17(b).

1. Draw the line XY.
2. Draw the front view a′b′, a line parallel to XY and having true length (TL)
3. Project the top view ab which is also a line parallel to XY having true length (TL).

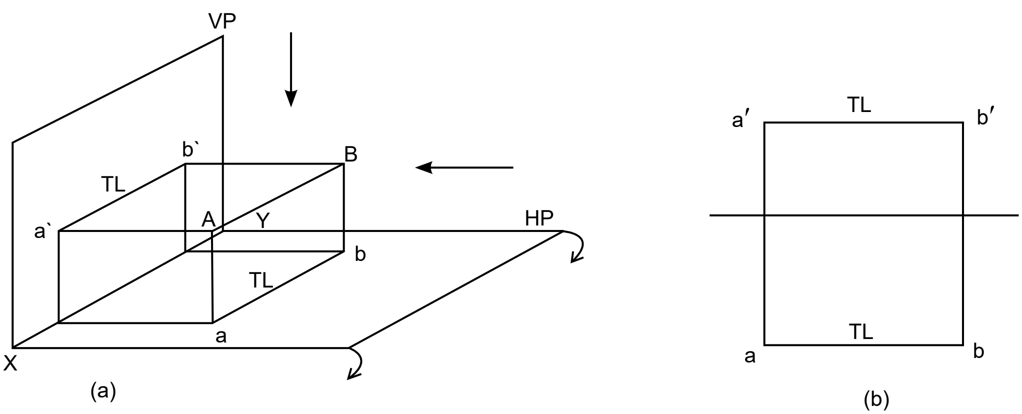


Figure 2.17 : Projection of Line Parallel to both the Planes

2.8.2 Projection of a Line Perpendicular to HP and Parallel to VP

Consider a straight line AB kept perpendicular to HP and parallel to VP as shown in figure 2.18(a). Its front view is projected onto VP which is a line having true length. The top view is projected onto HP which is a point one end b of which is visible while the other end a is invisible and is enclosed within ( ).

The HP is rotated in the clockwise direction through an angle of 90^0 and is obtained in the vertical position.

Followings are the steps used to draw projection with references to the XY line as given in figure 2.18(b).

1. Draw the line XY.
2. Draw the front view a’b’, which is a line perpendicular to XY and having true length (TL).

3. Projected the top view ab. The end b is visible and the invisible end a is marked inside ( ).

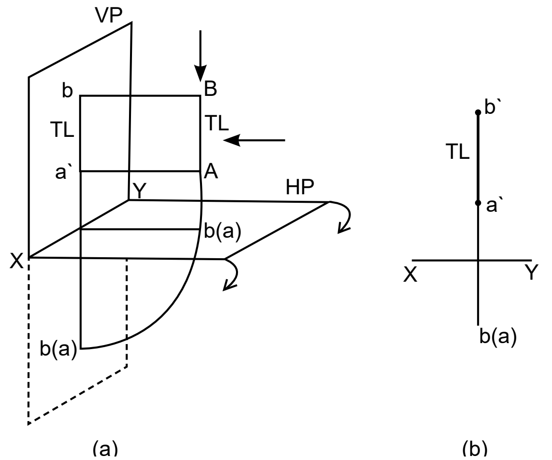


Figure 2.18: Projection of line Perpendicular to HP and Parallel to VP

2.8.3 Projection of a Line Perpendicular to VP and Parallel to HP

Consider a straight line AB perpendicular to VP and parallel to HP as shown in figure 2.19(a). Its top view is projected onto HP which is a line having true length (TL). The front view is projected on to VP which is point, the end b' of which is visible and the other end a' is invisible which is shown enclosed in ( ).The HP is rotated in a clockwise direction through an angle of 90° and is obtained in the vertical position. The followings steps are used to draw projection with references to the line XY as given in figure 2.19(b).

1. Draw the line XY.
2. Draw the top view (a b), a line perpendicular to XY and having true length (TL).
3. Projected the front view (a' b'). The end (b') is visible and the invisible end (a') is marked inside ( ).

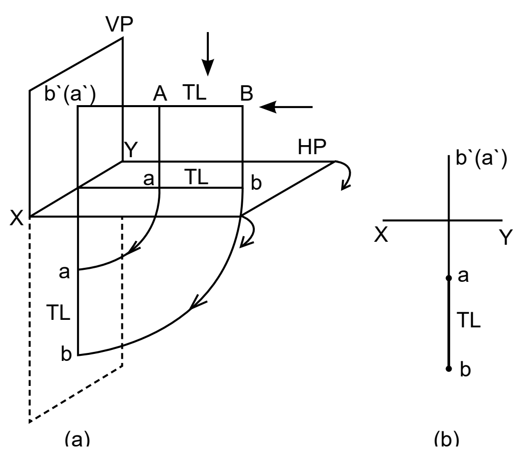


Figure 2.19 : Projection of line perpendicular to VP and parallel to HP

2.8.4 Projection of a Line Inclined to HP and Parallel to VP

Consider a straight line AB kept inclined to HP and parallel to VP as shown in figure 2.20(a). Its front view is projected onto VP which is an inclined line at an angle θ to XY and having true length (TL). The top view is projected onto HP which is also a line but smaller than the true length and parallel to XY. The inclination of the line with HP is always represented by the symbol θ. The HP is rotated in the clockwise direction through 90° and is obtained in the vertical position.

The followings steps are used to draw projection with references to the line XY as given in figure 2.20(b):

1. Draw the line XY.
2. Draw the front view (a' b'), a line inclined at an angle θ to XY having true length (TL).
3. Project the top view (a b) which is also a line parallel to XY and smaller than true length.

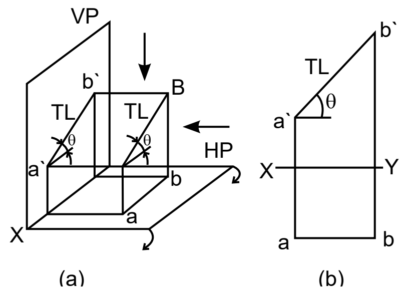


Figure 2.20: Projection of Line Inclined to HP and Parallel to VP

2.8.5 Projection of a Line Inclined to VP and Parallel to HP

Consider a straight line AB inclined to VP and parallel to HP as shown in figure 2.21(a). Its top view is projected onto HP which is a line inclined at an angle ϕ to XY and having true length (TL). The front view is projected onto VP which is also a line but smaller than true length and is parallel to XY. The inclination of the line with VP is always represented by the symbol ϕ. The HP is rotated in the clockwise direction through an angle of 90° and is obtained in the vertical position.

The followings steps are used to draw projection with references to the line XY as given in figure 2.21(b):

1. Draw the line XY
2. Draw the top view (a b), a line inclined at an angle ϕ to XY and having true length (TL).
3. Project the front view (a' b'), which is also a line parallel to XY but smaller than true length.

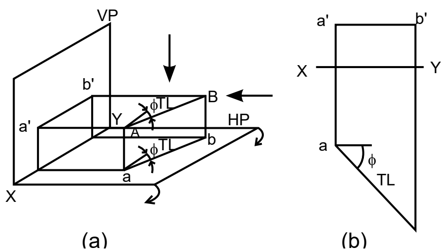


Figure 2.21: Projection of Line Inclined to VP and Parallel to HP

2.8.6 Projection of Line Contained by HP and Parallel to VP

When line CD is parallel to V.P. and is in H.P. its top view cd is a line parallel to V.P. and front view c'd' is on line XY as shown in figure 2.22. Both represent the true length of the line.

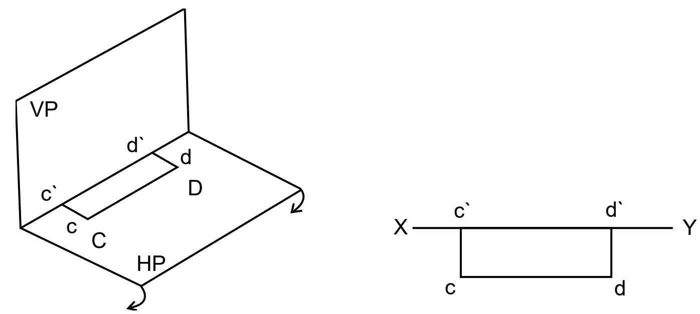


Figure 2.22: Projection of Line Contained by HP and Parallel to VP

2.8.7 Projection of Line Contained by VP and Parallel to HP

When a line AB is parallel to H.P. and is in V.P. its top view ab is a line on XY and front view (a' b') is parallel to H.P as shown in figure 2.23. Both represent the true length of the line.

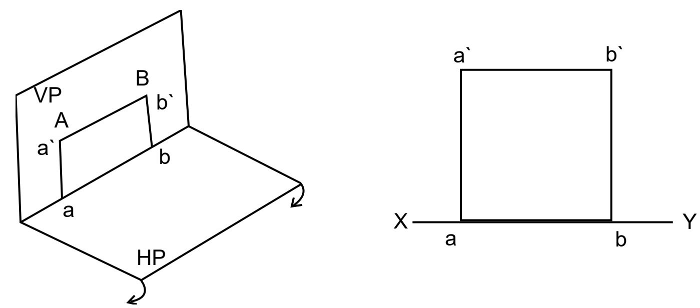


Figure 2.23: Projection of Line Contained by VP and Parallel to HP

2.8.8 Projectin of Line Contained by VP and HP

The front view (a' b') and top view (ab) both coincide with each other and lie on XY line and represent the true length of line AB as shown in figure 2.24.

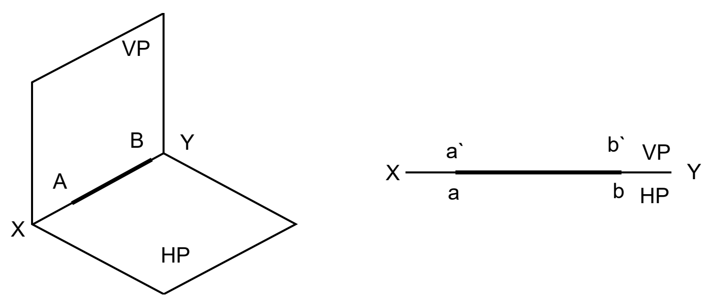


Figure 2.24 : Projection of Line Contained by VP and HP

2.8.9 Projection of a Line Inclined to both HP and VP

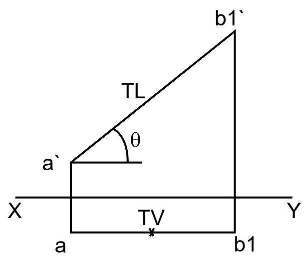
When a line is placed inclined to both HP and VP, its projections obtained in top and front views are smaller than the true length of the line and inclined to the XY line. So it is impossible to project and draw the top or front view of the line directly. Any one of the following methods may be used to draw the projections.

1. Rotating line method
2. Rotating trapezoidal plane method
3. Auxiliary plane method

Rotating Line Method: Consider a line AB is placed inclined at θ to HP and ϕ to VP. Draw its projections assuming that the line is placed in the first quadrant.

The following steps are listed to find the top view (plan) and front view (elevation) lengths, and then they are rotated to the required position to represent the projections of the line in the given position.Mark the projections of the end A by considering it as a point. Its front view (a') will be obtained above XY and top view (a) will be obtained below the line XY.

Step 1: Assume that the line is kept inclined to HP and are parallel to VP. Draw the front view (a' b1'), it is a line inclined at θ to XY and having true length (TL). Project and get the top view (ab1) length which is parallel to XY line as shown in figure 2.25(a). Then this will be rotated to the required position.



Projection of lines inclined to both the planes

Figure 2.25(a): Projection of a Line inclined to both HP and VP

Step 2: Assume the line is kept inclined to VP and parallel to HP. Draw the top view ab_2 , it is a line inclined at ϕ to XY and having true length (TL). Project and get the front view (a' b_2') length which is parallel to XY line as shown in figure 2.25(b) . Then this will be rotated to the required position.

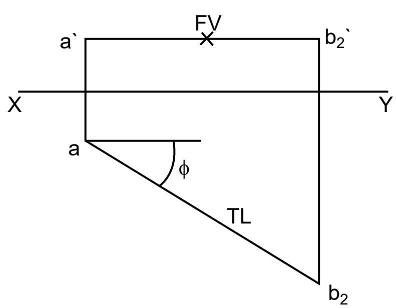


Figure 2.25 (b): Projection of a Line inclined to both HP and VP

Step 3: Draw the locus of the other end B of the line in top and front views. Draw the locus of the front view (b') as a line passing through (b'1) and parallel to XY line. Draw the locus of the top view (b) as a line passing through (b_2) and parallel to XY line as shown in figure 2.25(c).

Note: Step 1 and step 2 are shown together.

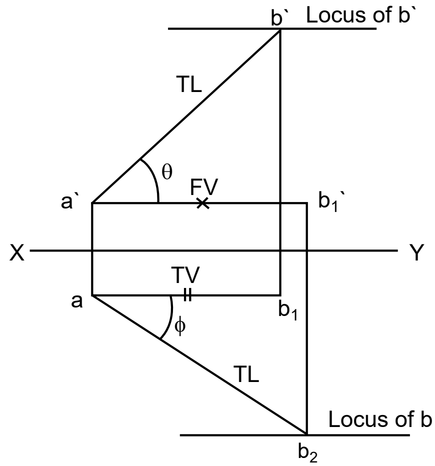


Figure 2.25 (c): Projection of a Line Inclined to both HP and VP

Step 4: Rotate the top view ab_1 and front view (a'b2') to the required position. Take a as centre, top view length ab as radius, draw an arc to intersect with the locus of b at b. Join a and b to get the top view (ab) of the line in required position. Taking a' as centre, front view (a'b2') as radius, draw an arc to intersect the locus of b' at b'. Join a' and b' to get the front view a'b' of the line in required position as shown in figure 2.25(d).

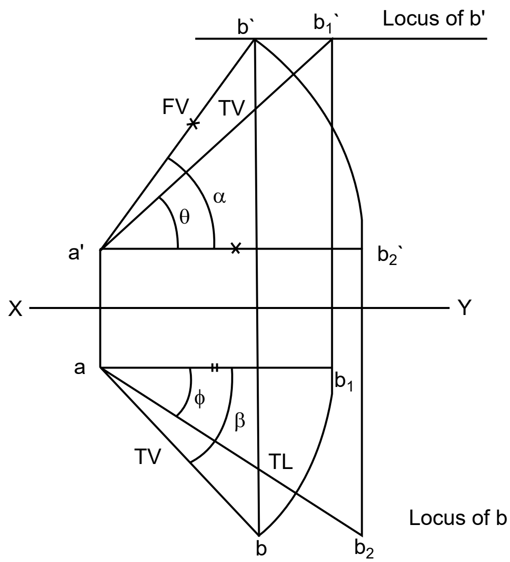


Figure 2.25 (d): Projection of a Line Inclined to both HP and VP

2.8.10 Projection of a Line Contained by one or both the Planes

The following steps are used for projection of a line contained in one or both plane as shown in figure 2.26.

1. Line AB is in the H.P. Its top view ab is equal to AB; its front view a' b' is in xy.
2. Line CD is in the V.P. Its front view c'd' is equal to CD; its top view cd is in xy.
3. Line EF is in both the planes. Its front view e' f' and the top view ef coincide with each other in xy.
4. Hence, when a line is contained by a plane, its projection on that plane is equal to its true length; while its projection on the other plane is in the reference line.

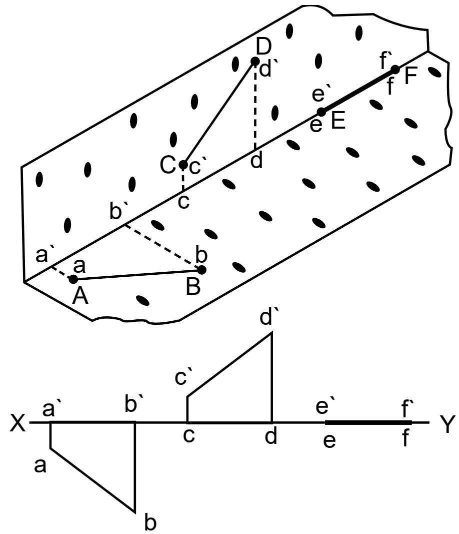


Figure 2.26: Projection of a Line Contained by one or both Planes

SOLVED EXAMPLES

Example 2.5: The front view of a 75 mm long line AB measures 55 mm. The line is parallel to the H.P.and one of its ends is in the V.P. and 25 mm above the H.P. Draw the projections of the line and determine its inclination with the V.P.

Solution: As the line is parallel to the H.P., its front view will be parallel to *xy.* Following steps are used to draw the projection of a given line PQ as shown in figure 2.27

1. Mark *a,* the top view of one end in *xy,* and a', its front view, 25 mm above *xy.*
2. Draw the front view *a'b',* 55 mm long and parallel to *xy.*
3. With *a* as centre and radius equal to 75 mm, draw an arc cutting the projector through *b'* at *b.*
4. Join *a* with *b. ab* is the top view of the line. Its inclination with *xy,* viz. ϕ is the inclination of the line with the V.P.

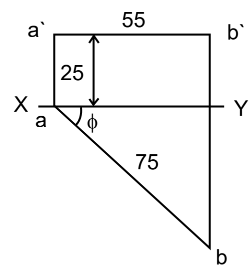


Figure 2.27: Projection of Line AB

Example 2.6: A line AB 75 mm long has its end A in both HP and VP. The line is kept inclined at 45° to HP and 30° to VP.

Solution: Following steps are used to draw the projection of a line AB as shown in figure 2.28 :

Mark the projections of end A by considering it as a point. Its front view (a') and top view (a) are marked as a common point on the XY line.

1. Assume that the line is kept inclined to HP and parallel to VP. Draw the front view (a'b1') which is inclined at 45° to XY and has a length of 75 mm. The top view length (ab1) is projected and obtained on XY line.
2. Assume that the line is kept inclined to VP and parallel to HP. Draw the top view ab_2 which is inclined at 30° to XY and has a length of 75 mm. The front view length a'b2' is projected and obtained on line XY.
3. Draw the locus of b', passing through b_1' and parallel to XY line. Also draw the locus of b passing through b_2 and parallel to XY line.
4. Rotate the top view ab_1 by taking "a" as centre, ab_1 as radius to get the intersection point b with the locus of b.
5. Join a and b to complete the top view ab of the line.
6. Rotate the front view (a'b2') by taking a' as centre, a'b2' as radius to get the intersection point b' with the locus of b'.
7. Join a' and b' to get the front view a'b' of the line.

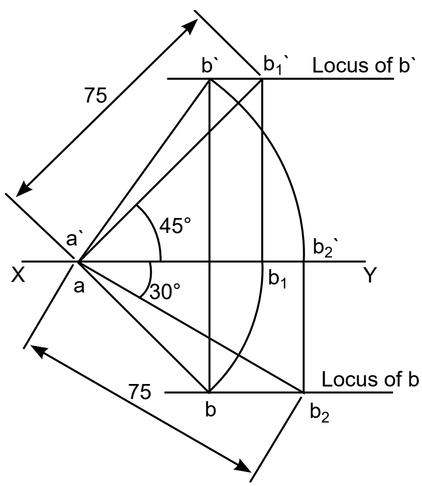


Figure 2.28: Projection of Line AB Inclined to both the Planes

Example 2.7 : FV of line AB is 50° inclined to xy and measures 55 mm long while it's TV is 60° inclined to xy line. If end A is 10 mm above HP and 15 mm in front of VP, draw it's projections,find TL, inclinations of line with Hp & Vp.

Solution : Following steps are used to draw the projection of a given lins as shown in figure 2.29 :

1. Draw xy line and one projector.
2. Locate a' 10 mm above xy and a 15 mm below xy line.
3. Draw locus from these points.

4. Draw FV 50° to xy from a’ and mark b’ cutting 55 mm on it.
5. Similarly draw TV 60° to xy from a and draw projector from b’ locate point b and join a b.
6. Then rotating views as shown, locate true lengths ab1 & a’b1’ and their angles with HP and VP.

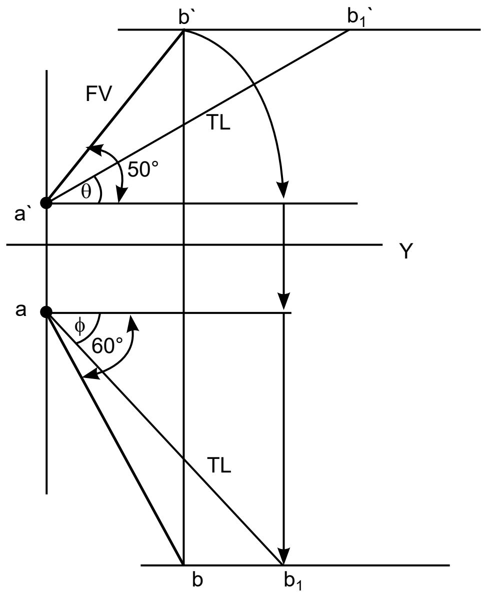


Figure 2.29: Projection of Line for Given FV and TV

2.9 PROJECTION OF PLANES

A plane is a two dimensional figure with length and breadth. A plane figure may be assumed to be contained by a plane,and its projections can be drawn, if the position of that plane with respect to the principal planes of projection is known. Figure 2.30 shows different shapes of plane such as square, rectangle, circle, pentagon, hexagon, etc.

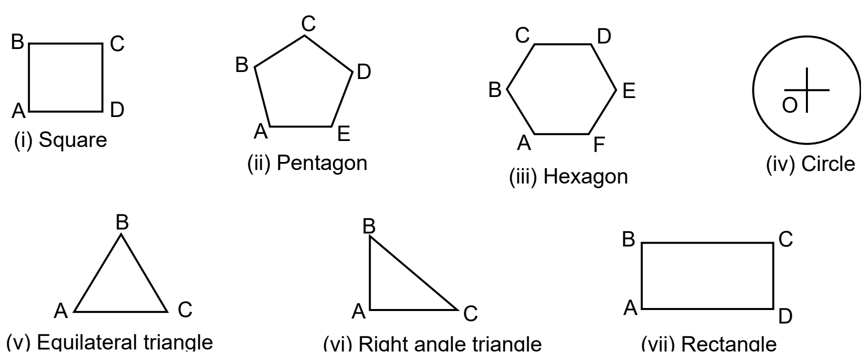


Figure 2.30: Different Shapes of Plane

2.9.1 Types of Projection Planes

There are two types of projection planes used in engineering drawings:

1. **Perpendicular planes:** The planes perpendicular to one or both the reference planes i.e., H.P. and V.P. are termed as perpendicular planes.These planes can be devided into the following types:
   (i) Planes perpendicular to H.P. and parallel to V.P.
   (ii) Planes perpendicular to V.P. and parallel to H.P.
   (iii) Planes perpendicular to V.P. and H.P. both.
   (iv) Planes perpendicular to V.P. and inclined to H.P.
   (v) Planes perpendicular to H.P. and inclined to V.P.
2. **Oblique Planes :**Planes which are inclined to both the reference planes are called *oblique planes.*

2.9.2 Projection of Planes Perpendicular to H.P. and Parallel to V.P.

Figure 2.31 shows a rectangle ABCD perpendicular to H.P. and parallel to V.P., the characteristics of the projection of the rectangle are as follows :

1. Its front view will be a rectangle (a'b'c'd') showing the true shape and size of the plane.
2. Its top view will be a line (ad-bd) which is parallel to XY.
3. It has only a horizontal trace and no vertical trace.

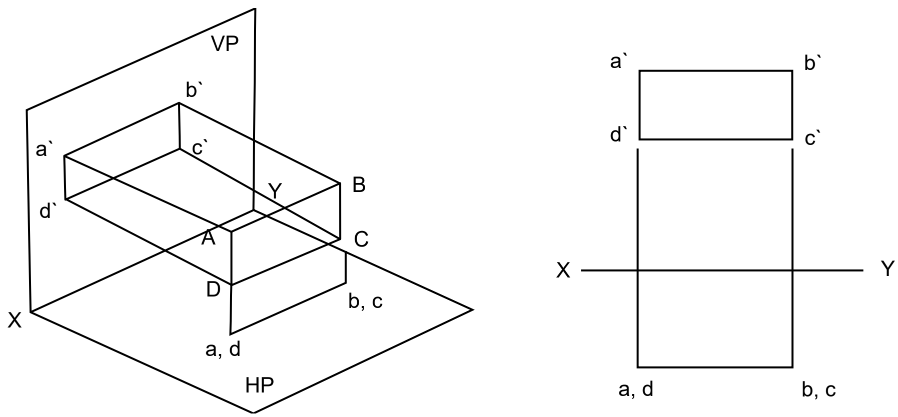


Figure 2.31: Projection of Planes Perpendicular to H.P. and Parallel to V.P.

2.9.3 Projection of Planes Perpendicular to V.P. and Parallel to H.P.

Figure 2.32 shows a rectangle ABCD in space perpendicular to V.P. and parallel to H.P., the characteristics of the projection of the rectangle are as follows :

1. The top view (abcd) gives the actual shape and size of the object.
2. The front view is a straight line parallel to XY.

3. It has only vertical trace and no horizontal trace.

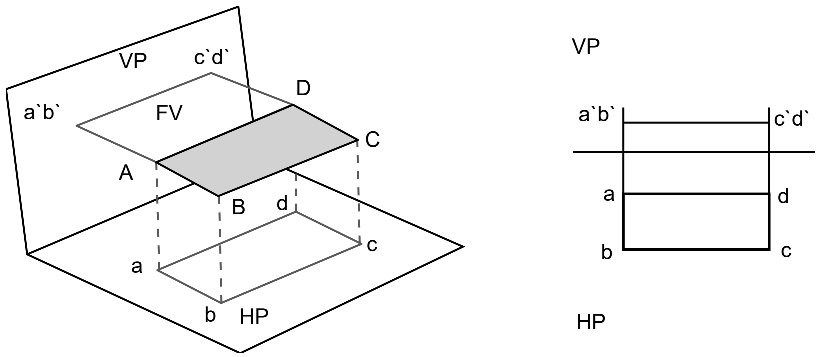


Figure 2.32: Projection of Planes Perpendicular to V.P. and Parallel to H.P.

2.9.4 Projection of Planes Perpendicular to H.P. and V.P.

Figure 2.33 shows a rectangle ABCD in space perpendicular to H.P. and V.P., the characteristics of the projection of the rectangle are as follows :

1. It has both vertical and horizontal traces.
2. The front view(straight line) a'b'-c'd' coincide with V.T.
3. The top view (straight line) ad-cb coincide with H.T.

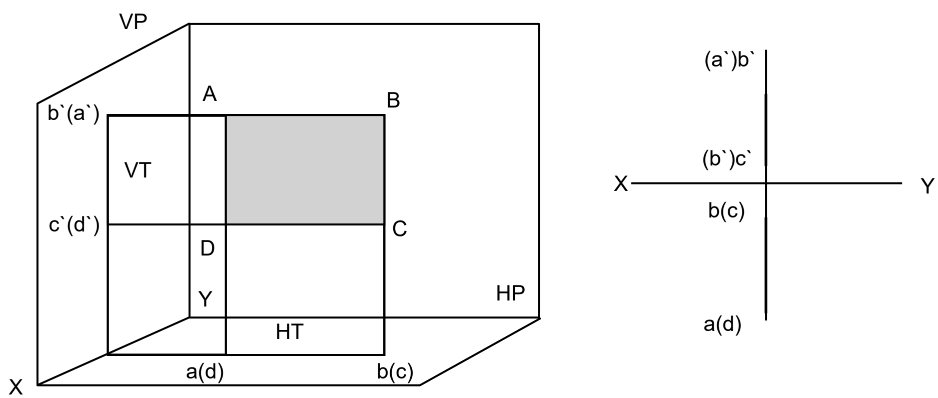


Figure 2.33: Projection of Planes Perpendicular to H.P. and V.P.

2.9.5 Projection of Planes Perpendicular to H.P. and Inclined to V.P.

Figure 2.34 shows a rectangle ABCD in space perpendicular to H.P. and inclined V.P. at angle ϕ, the characteristics of the projection of the rectangle are as follows :

1. It has horizontal trace inclined at angle ϕ to XY and vertical trace perpendicular to XY line.
2. The front view is shortened and less than the actual dimensions.

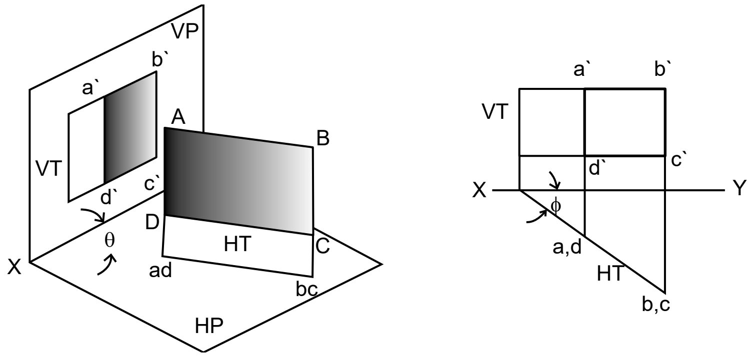


Figure 2.34: Projection of Planes Perpendicular to H.P. and Inclined to V.P.

2.9.6 Projection of Planes Perpendicular to V.P. and Inclined to H.P.

Figure 2.35 shows a rectangle ABCD in space perpendicular to V.P. and inclined to H.P. at an angle θ , the characteristics of the projection of the rectangle are as follows :

1. It has vertical trace inclined at angle θ to XY and horizontal trace perpendicular to XY line.
2. The top view is shortened and less than the actual size.

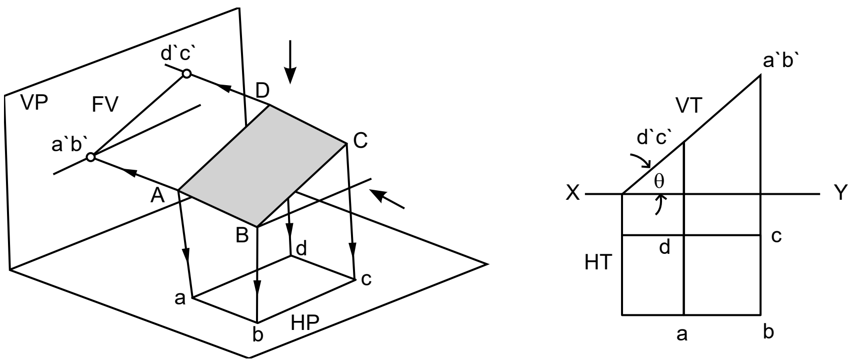


Figure 2.35: Projection of Planes Perpendicular to V.P. and Inclined to H.P.

Important Observations

- When a plane is perpendicular to H.P. and parallel to V.P., it has only one trace i.e., horizontal trace and when a plane is perpendicular to V.P. and parallel to H.P., it has only one trace which is vertical trace.
- In cases when plane is perpendicular to both the planes H.P. and V.P., it has both vertical and horizontal traces.
- When a plane is perpendicular to H.P. and inclined to V.P. it has both vrtical and horizontal traces. Front view gives reduced shape of plane and top view makes true angle with V.P.
- When a plane is perpendicular to V.P. and inclined to H.P., it has both vrtical and horizontal traces. Top view gives reduced shape of plane and front view makes true angle with H.P.
- When a plane is perpendicular to both H.P. and V.P. both front view and top view give reduced shape of the plane.

SOLVED EXAMPLES

Example 2.8: A hexagonal plate of side 30mm is places with a side on V.P. and surface inclined at 50° to V.P. and perpendicular to H.P. Draw the projections.

Solution: Following steps are used to draw the projection of a given hexagonal plate as shown in figure 2.36:

1. Draw a regular hexagon a'b'c'd'e'f' of side 30mm with a side resting on V.P assuming plane to be parallel to V.P. and perpendicular to H.P.
2. Tilt the top view at angle of 50° and project the front view.

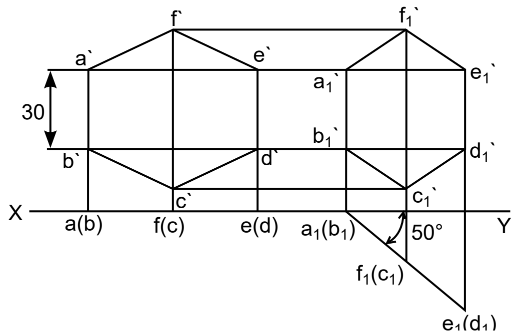


Figure 2.36: Projection of Hexagonal Plate

Example 2.9: Draw the projections of a circle of 5 cm diameter having its plane vertical and inclined at 30° to theV.P. Its center is 3 cm above H.P. and 2 cm in front of V.P.

Solution: Following steps are used to draw the projection of a given circle of 5 cm diameter as shown in figure 2.37:

1. Draw a circle of diameter 5 cm with center 3 cm above the line XY.
2. Divide it into 12 equal parts and project top view on line XY
3. Reproduce a new top view separately, making an angle 30° to the V.P.
4. Project the new front view upwards with the help of new top view and initial top view. The new front view is an ellipse.

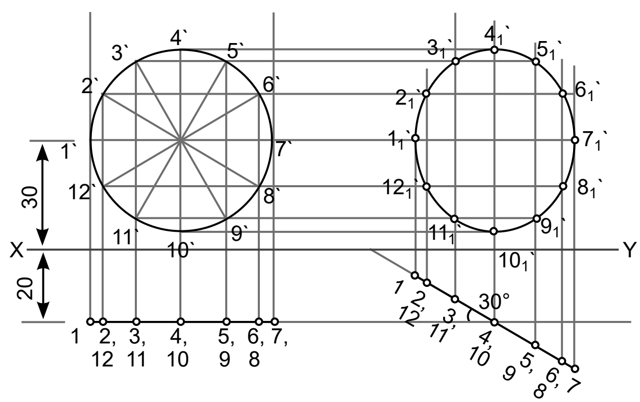


Figure 2.37: Projection of a Circle of 5 cm Diameter

Example 2.10: Draw the projections of a rhombus with diagonals 100 mm and 40 mm long.The smaller diagonal is parallel to both the refernce planes and larger diagonal is inclined at 35° to the H.P.

Solution: Following steps are used to draw the projection of rhombus as shown in figure 2.38:

1. Draw top view of rhombus in true shape abcd in a way that ac is paralle to XY line.Project the corresponding top view which is a straight line.
2. Tilt and incline a'c' at angle 35° to the H.P. Project the second top view $a_1b_1c_1d_1$.
3. Reproduce another top view such that smaller diagonal b_1d_1 is parallel to both the planes and project the final front view accordingly.

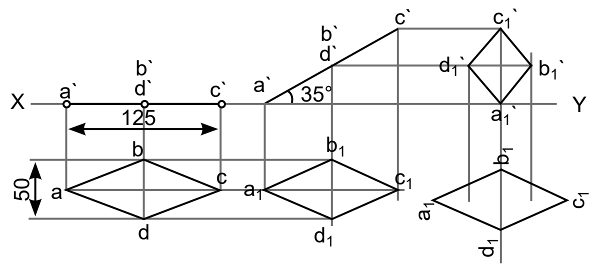


Figure 2.38: Projection of a Rhombus

Example 2.11: A pentagonal plane of 25 mm side is resting in H.P. on one of its sides which makes an angle of 60° with V.P. and the surface of plane makes an angle of 30° with H.P. Draw the projections.

Solution: Following steps are used to draw the projection of a given pentagonal of 25 mm side as shown in figure 2.39:

1. Assume that the pentagon is lying on H.P. with an edge perpendicular to V.P. Draw top view(pentagon) abcde and front view a'b'c'd'e'(line) on XY line.
2. Draw second top view a'e'b'd'c' which is a line inclined at 30° to the horizontal plane and project it to form second top view.
3. Turn the second top view such that the edge ae makes an angle 60° with the V.P. and project the third and final front view upwards from third top view and horizontally from second front view.

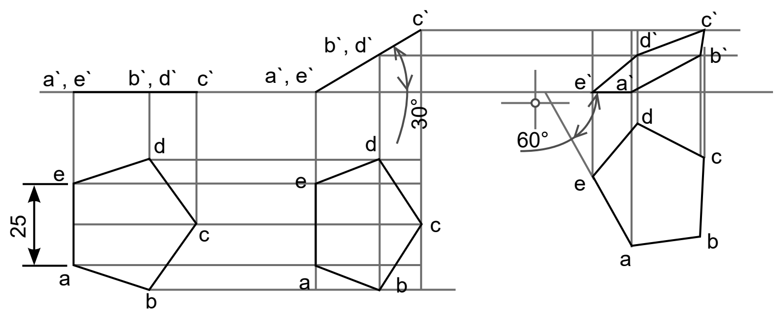


Figure 2.39: Projection of Pentagonal of 25 mm Side

Example 2.12: A rectangular plate 30 mm * 50 mm is resting on HP on one small side which is at angle 30° to V.P., while the surface of plane makes an angle of 45° with H.P. Draw the projections.

Solution: Following steps are used to draw the projection of given rectangular plate as shown in figure 2.40:

1. Draw the top view abcd(rectangular) and front view a'd'-c'd'(line) such that shorter side is perpendiculat to V.P.
2. Draw the second top view $a_1b_1c_1d_1$ as a square of side 30 mm (smaller side) and project the 2*nd* front view at an angle 45° by drawing vertical projections.
3. Tilt $a_1b_1c_1d_1$ such that edge a_1b_1 makes an angle 30° with V.P. Project the front view $a_1'b_1'c_1'd_1'$.

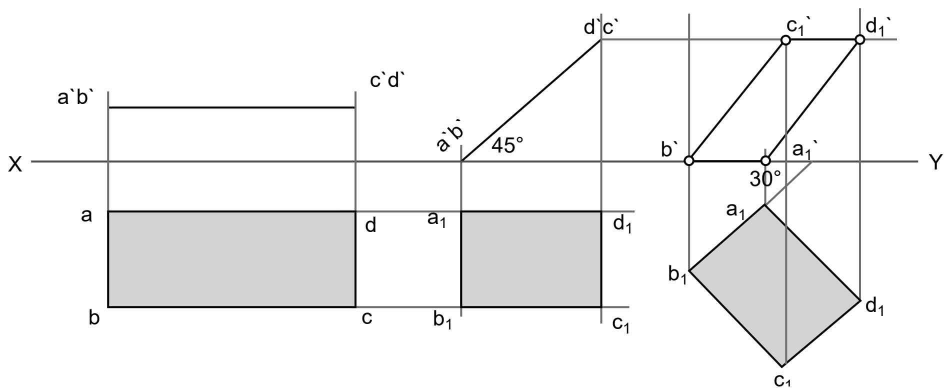


Figure 2.40: Projection of Rectangular Plate

UNIT SUMMARY

- Orthographic projection is a method of representing exact or true shape of a 3D object on 2D sheet in two or more views.
- Reference plane is formed when two planes intersecting at right angles to each other are used for projection.
- Vertical Plane is the plane which is vertical and is used for projecting front view of an object.
- Horizontal Plane is the plane which is horizontal and is used for projecting top view of an object.
- Front view or elevation of an object and is projected on vertical plane(VP).
- Top view or plan is an object is projected on horizontal plane(HP).
- Vertical plane is always fixed while Horizontal plane is rotated clockwise (90°) about XY, until VP and HP come in same line.
- Inclination of a straight line to a reference plane is the angle which line makes with the plane.
- Actual length of the line is called the *true length*.
- A plane is a two dimensional figure with length and breadth. Its thickness is always neglected.
- The planes perpendicular to one or both the reference planes i.e., H.P. and V.P. are termed as perpendicular planes.
- A line when parallel to both the planes HP and VP, then the line has true length in both the front and top views.
- If the line is inclined only to HP the Front view is a line having the true length (TL) and true inclination θ.
- If the line is inclined only to VP the Top view is a line having the true length (TL) and true inclination Φ.
- First angle projections method the objects are placed in 1st quadrant (FV above x-y line and TV below x-y line) which is above HP and in front of VP.
- Third angle projections method the objects are placed in 3rd quadrant (FV below x-y line and TV above x-y line) which is below HP and behind VP.

EXERCISES

2.1 Multiple Choice Questions

1. Which of the following describes the theory of orthographic projection?
   (a) Projectors parallel to each other and perpendicular to the plane of projection
   (b) Projectors parallel to each other and parallel to the plane of projection
   (c) Projectors parallel to each other and oblique to the plane of projection
   (d) Projectors perpendicular to each other and parallel to the plane of projection

2. Projection of an object shown by three views is known as
   (a) Perspective (b) Isometric
   (c) Oblique (d) Orthographic

3. In first angle projection method, the relative positions of the object, plane and observers are:
   (a) Object is placed in between
   (b) Plane is placed in between
   (c) Observer is placed in between
   (d) May be placed in any order
4. The object we see in our surrounding usually without drawing came under which projection:
   (a) Perspective projection (b) Oblique projection
   (c) Isometric projection (d) Orthographic projection
5. For orthographic projections, B.I.S. recommends the following method ?
   (a) First angle projection (b) Third angle projection
   (c) Second angle projection (d) Fourth angle projection
6. A point P is above Horizontal Plane (HP) and in front of Vertical Plane (VP). The point is in?
   (a) First quadrant (b) Second quadrant
   (c) Third quadrant (d) Fourth quadrant
7. Projection of a point in first quadrant will be?
   (a) Front view in VP (b) Front view in HP
   (c) Front view in PP (d) None of the above
8. The side view of an object is drawn in:
   (a) Vertical plane (b) Horizontal plane
   (c) Profile plane (d) Any of the above
9. A point lying in the HP has its top view above XY line. Its front view will be:
   (a) On XY line (b) Above XY line
   (c) Below XY line (d) Any of these
10. A point whose elevation and plan are above XY, is situated in:
    (a) First angle (b) Second angle
    (c) Third angle (d) Fourth angle
11. Which one of the following position is not possible in projection of a line?
    (a) Line perpendicular to HP, parallel to VP
    (b) Line perpendicular to VP, parallel to HP
    (c) Line parallel to both VP and HP
    (d) Line perpendicular to both VP and HP
12. When the line is parallel to both VP and HP, we can get its true length in?
    (a) Top view (b) Front view
    (c) Both (a) and (b) (d) Side view
13. When the line is inclined to HP and parallel to VP, we gets its true length in?
    (a) Front view (b) Top view
    (c) Side view (d) None of the above

14. Following method(s) is (are) used to find true length of a straight line?
    (a) Rotation method (b) Trapezoidal method
    (c) Both (a) and (b) (d) None of the above
15. When a surface of an object is inclined to a plane of projection, it will appear in the view
    (a) Foreshortened (b) Tue size and shape
    (c) Line (d) Point

Answers Keys (Exercise 2.1)

1. (a), 2. (d), 3. (a), 4. (a), 5. (a), 6. (a), 7. (a), 8. (c), 9. (b), 10. (b),
11. (d) 12. (c), 13. (a), 14. (c), 15. (a)

2.2 Short and Long Answer Type Questions

Category I

1. What is the principle of projection?
2. What is the difference between first angle and third angle projection?
3. Which angle projection is recommended by B.I.S in indie?
4. Why the projection of a object is not drawn in second and fourth quadrant?
5. State the quadrants in which the following points are situated :
   (a) A point P, its top view is 20 mm below XY and front view in XY.
   (b) A point Q, its top view is 25 mm above XY and front view is 40 mm below XY.
6. Name the method to determine the true length and true inclination of a straight line.
7. Which are the types of projections used in engineering drawing?
8. Explain the purpose and theory of multi view projections.
9. Define straight lines and trace of a line.
10. Name and define the principal planes of projection.

Category II

11. Write the steps and draw the projection of a line inclined to both the reference planes.
12. Explain projection of different planes position with respective to principal planes.
13. Classified and explain different types of projections.
14. Compare with examples , orthographic , isometric and perspective projections.
15. Why orthographics projection is important in manufacturing industry?

EXERCISE 2.3: Practical Problems

1. Projectors of point C and D are 40 mm apart. Draw projections of C and D if C is 15 mm above HP and 30 mm in front of VP. Point D is 25 mm below HP and 30 mm behind VP
2. Draw the projections of a point R which lies 20 mm behind VP and 15 mm below HP. In which quadrant R is located?
3. There is a point F which is 20 mm above HP and its shortest distance from XY is 45 mm. The point lies in first quadrant. Show its plan and elevation.

4. A line AB 60 mm long is parallel to V.P. & H.P. both. The end A is 30 mm above H.P. and 40 mm in front of V.P. Draw the projection of line AB.
5. A line AB is 75 mm long and it is 30° & 40° Inclined to H.P. and V.P. respectively. End point A is 12 mm above H.P. and 10 mm in front of V.P. Draw projections given that line is in 1st quadrant.
6. Draw the projections of a pentagon with side 35 mm having one side in H.P. The plane is inclined at 30° to H.P. and the side with which it rests makes an angle of 50° with H.P.
7. A Rectangle 40 mm and 70 mm sides is resting on HP on one small side which is 40° inclined to VP, while the surface of the plane makes 45° inclination with HP. Draw it's projections.
8. A regular pentagon of 30 mm sides is resting on HP on one of the side while its opposite vertex (corner) is 30 mm above HP. Draw projections when side in HP is 30° inclined to VP.
9. A circular plate having diameter 60 mm is resting on H.P. on point A of its rim(circumference).The plane is inclined at 45° to the H.P. and (i) the top view of diameter makes an angle of 30° with V.P. (ii) Diameter makes an angle 30° with the V.P.
10. A line AB, 60 mm long and inclined at 30° to the ground, has its end A on theground at 15 mm behind V.P. Its front view measures 45 mm. Draw the top view of the line AB and determine its inclination with the V.P. The H.P. is 45 mm above the ground.
11. The front view of line inclined at 30° to V.P is 65mm long. Draw the projections of a line, when it is parallel to and 40 mm above H.P. and one end being 20 mm in front of V.P.
12. Draw the projections of following points:
    (a) Point A is in HP and 40 mm behind VP
    (b) Point B is in HP and 40 mm in front of VP
    (c) Point C is in VP and 40 mm above HP
    (d) Point D is in both HP and VP
    (e) Point E is 20 mm above HP and 40 mm behind VP
    (f) Point F in VP and 50 mm below HP

INTERESTING FACTS

- Marcus Vitruvius Pollio, (flourished 1st century BC), Roman architect, devised the term orthographic (from the greek word *orthos* (= "straight") and graphē(= "drawing") for the projection.
- In an Orthographic projection, two views are necessary to understand the nature of the object. Sometimes, it is necessary to create the third view from the two available views so that all the details of the object are obtained. Thus creating the missing view is of importance in orthographic projection.

KNOW MORE

Detail Drawings

- Detail drawings provide a detailed description of the geometric form of a part of an object such as a building, bridge, tunnel, machine, plant, and so on. They tend to be large-scale drawings that show in detail parts that may be included in less detail on general arrangement drawings.

- Detail drawings may be used to demonstrate compliance with regulations and other requirements, to provide information about assembly and the junctions between components, to show construction details, detailed form, and so on, that would not be possible to include on more general drawings.
- Detail drawings consists of 2-dimensional orthographic projection representing plans, sections and elevations and may be drawn to scale by hand, or prepared using Computer Aided Design (CAD) software. However, increasingly, building information modeling (BIM) is being used to create detailed three-dimensional representations of buildings and their components.

APPLICATIONS

The primary role or function of orthographic projection is to convert design data into construction information and to clearly communicate that information to building industry, code officials, product manufacturers, suppliers and fabricators. Some of the applications of the orthographic projection are as follows:

- Engineers and Architects use various views to depict detail of an object.
- Real estate agents use floor plans of building to show prospective buyers of houses.
- Designers produce working drawings for prototyping and manufacturing.

INQUISITIVENESS AND CURIOSITY TOPICS

Auxiliary views and plane

Many objects have such shapes that their principal planes and lines of sight are not parallel to the regular principal projection planes. The projections on principal planes are either insufficient to describe the complete geometry or difficult to draw. There are infinite possible auxiliary views of an object.

Auxiliary views can help in finding:

- True lengths and inclinations of a line.
- Point view of a line.
- Edge view of a plane.
- True shape of a plane.
- Projections of solids
- True shapes and inclinations of solids etc.

Auxiliary plane is assumed in position which best facilitates auxiliary views leaving principal views clear.

ACTIVITY

Drawing orthographic projections

Create two-dimensional representations of three-dimensional objects shapes using blocks and then draw orthographic projections of those shapes.

Assessment Plan

- Place the same object on each table and have the students draw it in orthographic projection.
- Draw an object on the board in isometric, give the proper dimensions, and have students draw the object in orthographic projection.

CASE STUDY

The Designer and Orthographic Drawing

An orthographic drawing, sometimes called a working drawing, is usually the last drawing produced by a designer. It normally has three accurate views of a product, a front view, side view and plan view. Dimensions (measurements) are also drawn on each view, ensuring the manufacturer can make the product to the precise size and the designers requirements. A parts list is also included. This has the precise measurements for every part of the product and includes details such as materials and finish.

Designers often use remote manufacturing, in an attempt to keep costs low. Designers also produce working drawings so that prototypes can be manufactured, and then tested. This leads to improvements being made to the product. Working drawings are usually produced using CAD, although skilled designers still draw them be hand, at least in the early stage of the designing process. Designers find sketching in orthographic projection very useful.

Information found on a working drawing / orthographic drawing:

- All necessary views required for manufacturing.
- All the necessary measurements (called dimensions).
- A standard format for working drawings.
- A parts list which includes all the information needed to make each part of the product.

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

- https://youtu.be/cQHDAfrptUc
- https://youtu.be/ytwEDvX-l44
- https://youtu.be/TsTCaHYl9oo
- https://technologystudent.com/index.htm

3 Projection of Solids

UNIT SPECIFIC

In this unit, students will learn about solids and their classifications, projection of solids in different positions and orientation, such as when the axis is perpendicular, parallel and inclined to the reference planes.

RATIONALE

The topic "projection of solids", aims at developing the ability to draw and read projection of solid objects by presenting views of different sides of the object. This topic is very useful to create working drawings for visualization and shop floor production; it is also building foundation for further topics related to projection of solids.

PREREQUISITE

Knowledge and understanding of orthographic projection theory and practice.

UNIT OUTCOMES

Upon successful completion of this module, the students will be able to:

U3-O1: Understand various types of solids.

U3-O2: Construct the projections of simple solids in various positions.

U3-O3: Create floor plans for buildings.

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-3 Outcomes	Expected Mapping With Course Outcomes (1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)				
	CO-1	CO-2	CO-3	CO-4	CO-5
U3-O1	2	1			2
U3-O2		3			2
U3-O3		3			2

INTRODUCTION

A solid is three dimensional object, which has length, breadth and thickness, bound by surfaces, and may be plane or curved. Projection of solids can be represented by orthographic views, the number of which depends on the type of solid and its orientation with respect to the planes of projection.

3.1 CLASSIFICATION OF SOLIDS

Solids may be classified into two major groups:

(i) Polyhedral , and
(ii) Solids of revolution

3.1.1 Polyhedral

A polyhedron is defined as a solid bound by plane surfaces. When all the surfaces are similar, equal and regular, the polyhedron is called a regular polyhedron. Regular polyhedra can be further classified as:

(i) Tetrahedron: The tetrahedron has four equal faces, six straight edges, and four vertex corners. Each face has an equilateral triangle as shown in figure 3.1.

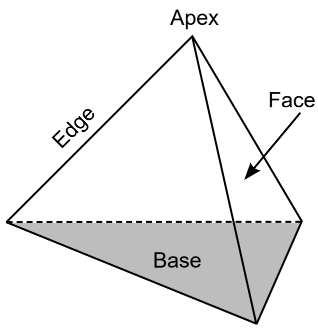


Figure 3.1: Tetrahedron

(ii) Cube or hexahedron: The cube or hexahedron is a polyhedral solid has six equal faces, eight corners, and 12 edges as shown in figure 3.2

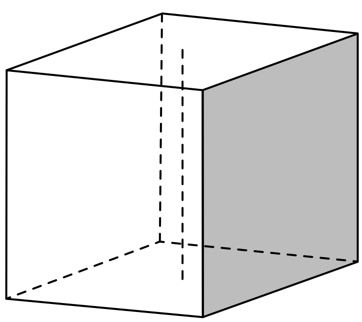


Figure 3.2: Cube or Hexahedron

(iii) Octahedron: The octahedron is a polyhedral solid has six vertices, twelve edges, and eight equal equilateral triangular faces as shown in figure 3.3.

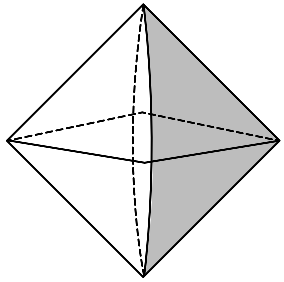


Figure 3.3: Octahedron

(iv) Dodecahedron: The dodecahedron is a polyhedral solid has twenty vertices , thirty edges and twelve equal and regular pentagonal faces as shown in figure 3.4.

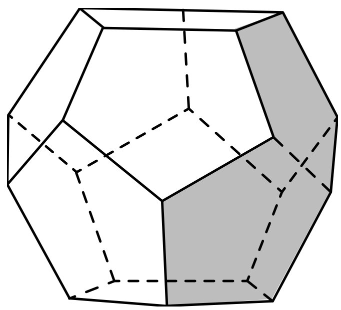


Figure 3.4: Dodecahedron

(v) Icosahedron: The icosahedron is a polyhedral solid has twelve vertices, thirty edges and twenty equal and regular triangular faces as shown in figure 3.5.

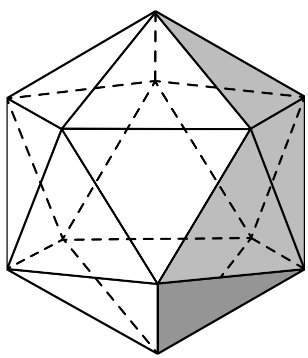


Figure 3.5: Icosahedron

(vi) Prism: A prism is a polyhedral figure that having two equal and similar faces called its bases, parallel to each other and joined by other faces which are parallelograms. The imaginary line joining the centres of the bases is called the axis. A prism is said to be right and regular prism when its axis is perpendicular to bases and all the faces are equal rectangles as shown in figure 3.6 .

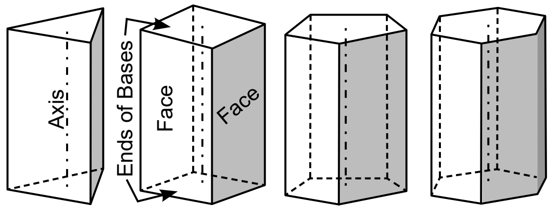


Figure 3.6: Regular Prisms

(vii) Pyramid: A pyramid is a polyhedral formed by a plane surface as its base and a number of isosceles triangles as its side faces, all meeting at a point called vertex(apex). the imaginary line connecting the apex and the center of the base is called the axis. Inclined or slant faces are called inclined triangular side faces.

A right pyramid is one in which the axis of the pyramid is perpendicular to its base.The pyramid is named according to the base, e.g. a pyramid with a square base is called square pyramid or a pyramid with a pentagonal base is called pentagonal pyramid as shown in figure 3.7.

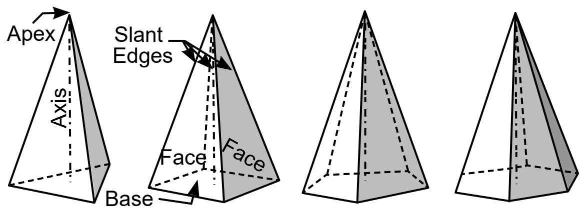


Figure 3.7: Right pyramids

3.1.2 Solids of Revolution

Solids of revolution are generated when some of the plane figures are revolved about one of their sides. Solids of revolution are classified as:

(i) Cylinder: The solid which is generated when a rectangle is revolved about one of its sides is called a cylinder. It has two equal and circular bases, the fixed line joining the centre of the circular bases is called the axis of cylinder as shown in figure 3.8.

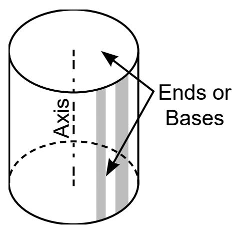


Figure 3.8: Cylinder

(ii) Sphere: A sphere is a solid which is generated when a semi-circle is revolved about its diameter which remains fixed as shown in figure 3.9. The mid-point of diameter is the center of sphere.

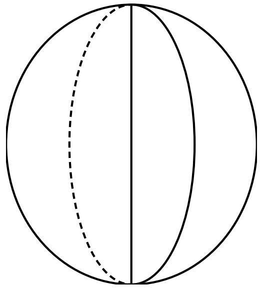


Figure 3.9: Sphere

(iii) Cone: A cone is a solid which is generated when a right triangle is revolved about one of its perpendicular sides. The fixed line about which triangle revolves is called the axis and the circle described by the other side is called the base as shown in figure 3.10.

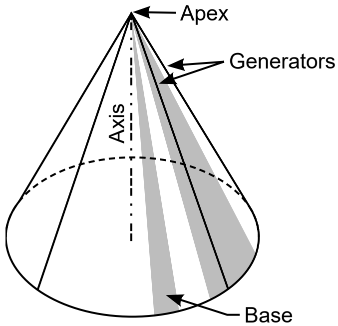


Figure 3.10: Cone

(iv) Frustum: When a solid cone or pyramid is cut by a plane, parallel to its base and top portion is removed, the remaining portion of the solid is called a frustum as shown in figure 3.11.

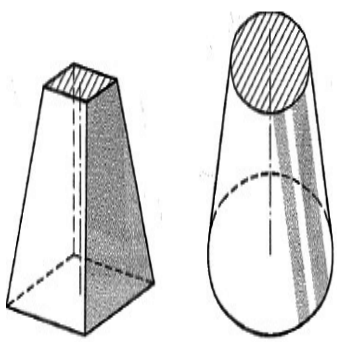


Figure 3.11: Frustum

v. **Truncated:** When a solid prism or cylinder`is cut by a plane which inclined to the base and the top portion is removed, the remaining portion of the solid is called truncated as shown in figure 3.12.

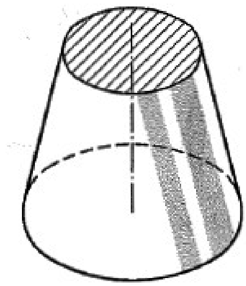


Figure 3.12: Truncated

3.2 TERMINOLOGY

1. **Edge or generator:** For Pyramids & Prisms, edges are the lines separating the triangular faces or rectangular faces from each other. For Cylinders, generators are the straight lines joining different points on the circumference of the bases with each other.
2. **Apex :** For a cone and a pyramid, apex is the point where all the generators or the edges meet.
3. **Axis of solid:** For a cone and a pyramid, *axis* is an imaginary line joining the center of the base to the apex. For a cylinder and a prism, *axis* is an imaginary line joining the centers of the ends or bases.
4. **Base:** The *base* of a solid is defined as the lower surface on which an object stands. The bottom line of a shape such as a triangle or rectangle.
5. **Right solid:** A solid is said to be a *right solid* when its axis is perpendicular to its base.
6. **Oblique solid:** A solid is said to be an *oblique* solid if its axis is inclined at an angle other than 90° to its base.
7. **Regular solid:** A solid is said to be a *regular* solid if all the edges of the base or the end faces of a solid are equal in length and form regular plane figures.

3.3 PROJECTION OF SOLIDS IN DIFFERENT POSITIONS

The position of solids in space may be specified by the location of either the axis, base, edge, diagonal or face with the principal planes of projection.

The following general positions of solids are considered for the projection:

1. Axis is perpendicular to HP.
2. Axis is perpendicular to VP.
3. Axis is parallel to both the HP and VP.
4. Axis is inclined to VP and parallel to HP.
5. Axis is inclined to HP and parallel to VP.
6. Axis is inclined to both the planes (VP and HP).

The position of a solid with reference to the principal planes may also be grouped as follows:

1. A solid resting on its base.
2. A solid resting on anyone of its faces, edges of faces, edges of bases, generators and slant edges.
3. A solid suspended freely from one of its corners.

3.3.1 Projection of a Solid with its Axis Perpendicular to HP

A solid when placed on HP with its axis perpendicular to it, will have its base on HP. The top view which shows the true shape is drawn first and the front view is projected from it.

Example: Projection of a cylinder of radius r and height h, resting on HP. Its axis is perpendicular to HP and parallel to VP.

These are the steps used to draw the projection of the given cylinder, resting on HP and with its axis perpendicular to HP. See figure 3.13.

1. Assuming a cylinder is placed with its base on HP.
2. First draw its top view, which shows its true shape, as the axis is perpendicular to the HP.
3. Mark the circle center 'o' and divide it into 8 equal parts.
4. Project the front view on XY, which will be a rectangle of width 2r and height h.

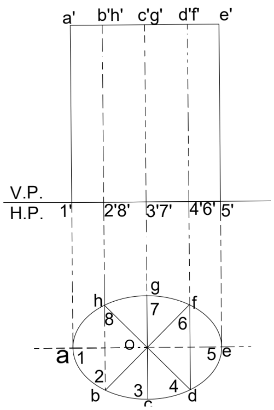


Figure 3.13: Projection of a Cylinder with Axis Perpendicular to HP

3.3.2 Projection of a Solid with its Axis Perpendicular to VP

When a solid is placed with its axis perpendicular to VP, the base of the solid will always be perpendicular to HP and parallel to VP. In the front view, the base will be projected in true shape. Therefore, the front view is drawn first and then the top view is projected from it.

Example: Projection of a cylinder of radius r and height h, its axis perpendicular to VP and parallel to HP

These are the steps used to draw a projection of a given cylinder with its axis perpendicular to VP and parallel to HP. See figure 3.14.

1. First draw its front view, which shows its true shape, as the axis of a cylinder is perpendicular to the VP.
2. Mark the circle center 'o' and divide it into 8 equal parts.

3. Project the top view below the XY line, which will be a rectangle of width 2r and height h.

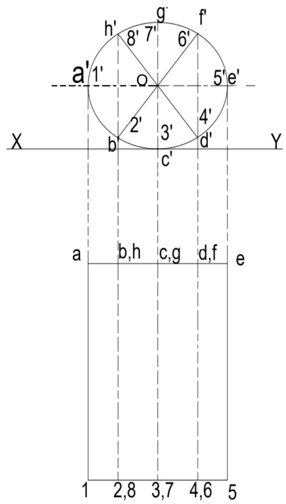


Figure 3.14: Projection of a Cylinder with its Axis Perpendicular to VP

3.3.3 Projection of a Solid with its Axis Parallel to both HP and VP

As the axis of a solid object is parallel to both the plane HP and VP, and perpendicular to the profile plane, the side view is drawn first.

Example: Projections of a cone of base radius r and h, is resting on HP on one of its faces. The axis is parallel to both the reference planes.

These are the steps used to draw a projection of a cone, when its axis is parallel to both HP and VP. See figure 3.15.

1. First draw a circle representing the side view keeping a point of circle on line XY.
2. Mark the center of the circle 'o' and divide the circle into 8 equal parts.
3. Project the front view horizontally from the side view of the cone.
4. Project down the top view from the front view and the side view.

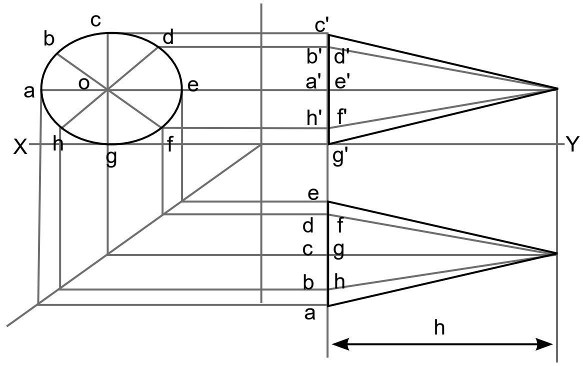


Figure 3.15: Projection of a Cone with Axis Parallel to both HP and VP

3.3.4 Projection of a Solid with its Axis Inclined to the VP and Parallel to the HP

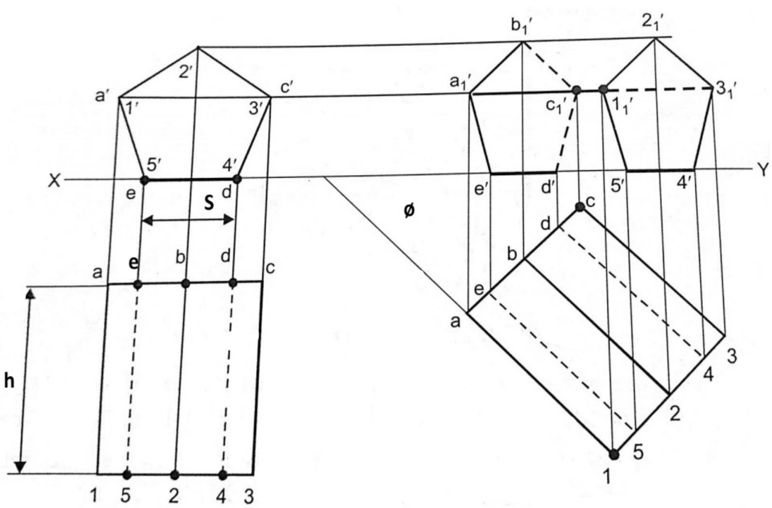
When a solid has its axis inclined to VP and is parallel to the HP, its projections are drawn through the following steps.

1. Draw the front and top view considering the solid in its simple position.
2. Tilt the top view by the given angle ϕ (axis inclined to VP).
3. Then project all the points to get the final front view.

Example: Projections of a pentagonal prism whose base is s and axis is h. It is resting on one of its rectangular faces on HP and axis inclined at ϕ to VP

These are the steps used to draw the projection of a pentagonal prism with its axis parallel to HP and inclined to VP. See figure 3.16.

1. The axis is inclined to VP so in the first view the axis is kept perpendicular to VP and true shape is drawn in a front view.
2. Draw the front view, a pentagon such that its edge lies on line XY.
3. Project the top view, below line XY.
4. Alter the position of the top view, i.e. reproduce it so that the axis is inclined at ϕ to *xy*.
5. Draw the final front view by projecting all the points upwards from this top view and horizontally from the first front view.



Projection of solids with axis inclined to one and parallel to another reference plane

Figure 3.16: Projection of a Pentagonal Prism Axis parallel to HP and Inclined to VP

3.3.5 Projection of a Solid with its Axis Inclined to the HP and Parallel to the VP

When a solid has its axis inclined to HP and is parallel to the VP, its projections are drawn through following steps.

1. Draw the front and top view considering the solid in its simple position.
2. Tilt the front view by the given angle θ (axis inclined to HP).
3. Then project all the points to get the final top view.

Example: Projections of a hexagonal pyramid of side s and h is resting with one of its edges of base on HP and makes an angle of θ with HP and is parallel to VP

Assume a hexagonal pyramid is placed with its base on HP and its axis to be perpendicular to the HP.

The following steps are used to draw the projection of a hexagonal pyramid with its axis parallel to VP and inclined to HP. See figure 3.17.

1. Draw the top view of a regular hexagon abcdef, keeping the right hand side cd perpendicular to XY.
2. Project the front view on line XY.
3. Reproduce the front view such that base edge c'd' touches line XY making an angle θ with HP.
4. Project all the points downwards to form the second and final top view.

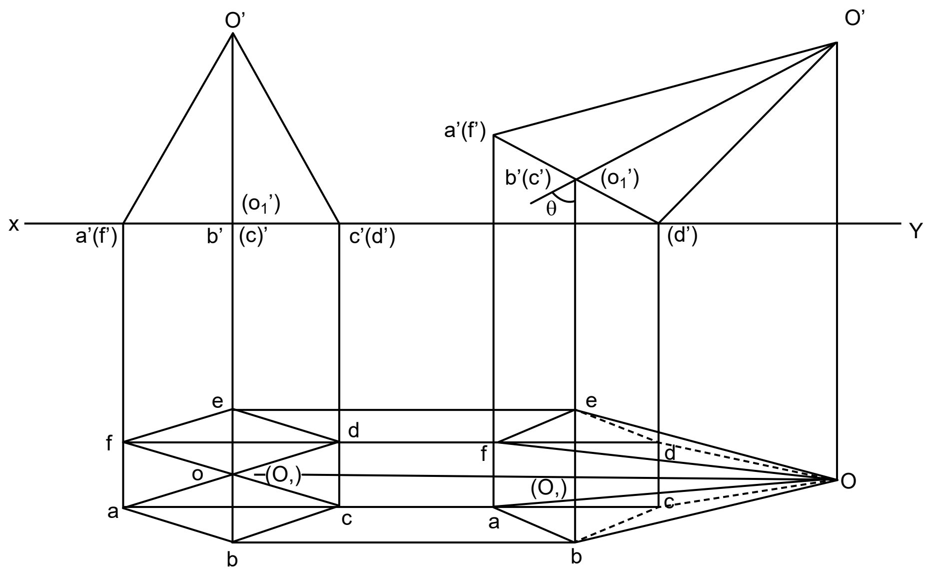


Figure 3.17: Projection of a Hexagonal Pyramid Axis Inclined to HP and Parallel to VP

3.3.6 Projection of a Solid with its Axis Inclined to the HP and VP

A solid is said to be inclined to both the planes when:

i. The axis is inclined to both the planes
ii. The axis is inclined to one plane and an edge of the base is inclined to the other.

There are two methods for drawing the projections of solids:

1. **Projection of solids by change of position method**
   In this method the solids are placed first in the simple position and then tilted successively in one or more stages to obtain the final position.
2. **Projection of solids by auxiliary plane method**
   In this method, the solids are placed initially in the simple position and then one or more auxiliary planes are setup to obtain the projection in the final position.

Example: Projections of a right circular cylinder of radius ‘r’ and axis ‘h’ resting on one point of base circle on VP and its axis makes an angle of ϕ with VP. The front view of the axis is inclined at θ to HP. The following steps are used to draw the projection of a right circular cylinder of radius ‘r’ and axis ‘h’. See figure 3.18.

1. Draw the front view and top view in their initial positions.
2. The top view is a rectangle of height h and front view is a circle of radius r.
3. Tilt the top view at an angle of ϕ with line XY.
4. Project the second front view about line XY.
5. Reproduce the second front view, make an angle of θ to HP.
6. Project the top view accordingly.

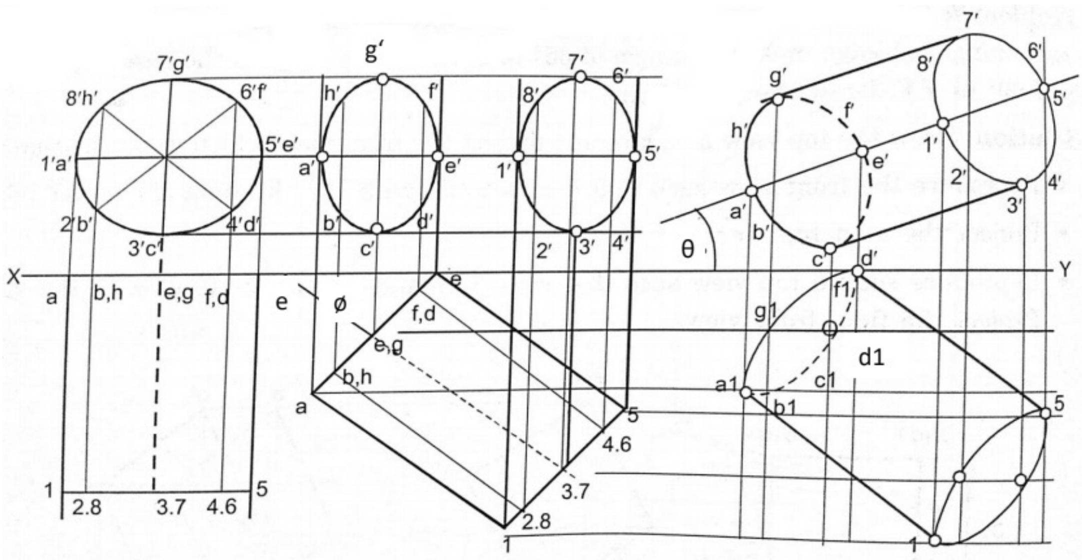


Figure 3.18: Projection of a right circular cylinder

SOLVED EXAMPLES

Example 3.1: A pentagonal prism of 30 mm side and 60 mm axis is resting with one of its edges of base on HP and makes an angle of 30° with HP and is parallel to VP. Draw its projections.

Solution: The following steps are used to draw the projection of the given pentagonal. See figure 3.19.

1. Draw the top view of a regular pentagon abcde, keeping the right hand side cd perpendicular to XY. Assume a pentagonal prism placed with its base on HP.
2. Project the front view above line XY.
3. Reproduce the front view such that base edge c’d’ touches line XYmaking an angle of 30° with HP.

4. Project all the points downwards to form a second and final top view.

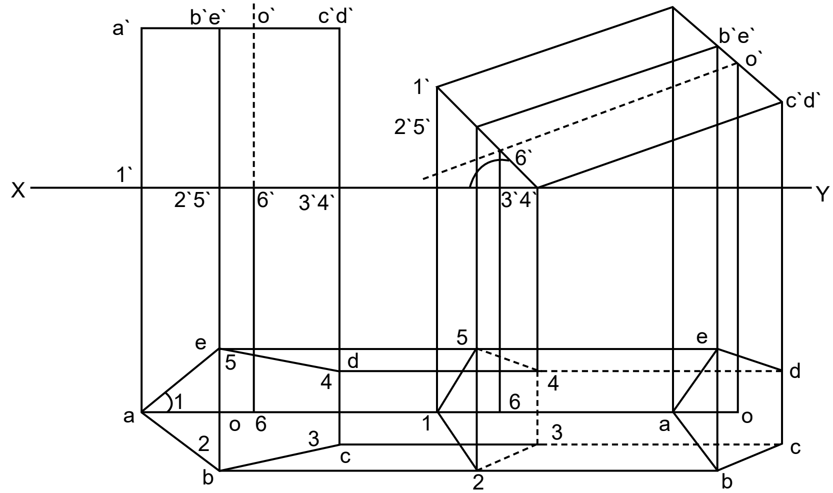


Figure 3.19: Projection of a Pentagonal Prism

Example 3.2: Draw the projections of a cylinder 35 mm diameter and axis 50 mm long resting on a point of base circle on HP and its axis making an angle of 45° to HP and parallel to VP.

Solution: The following steps are used to draw the projection of a given cylinder. See figure 3.20.

1. Draw the top view of a cylinder which is a circle of radius 35 mm. Assume that the cylinder is placed with its base on HP.
2. Project the top view to form the front view, ,which is a rectangle of height 50 mm.
3. Tilt the front view at an angle of 45° with line XY.
4. Project the final top view.

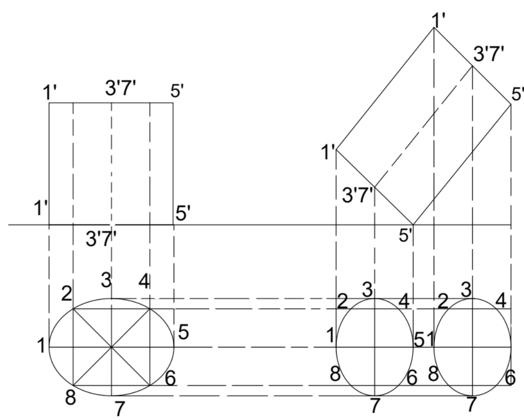


Figure 3.20: Projection of a Cylinder

Example 3.3: A cube of 35 mm side has one of the edges on HP in a way that one square face containing that edge makes an angle of 35° to HPand the edge of the base makes an angle of 60° to VP. Draw the projections.

Solution: The following steps are used to draw the projection of the given cube. See figure 3.21.

1. Draw the top view of cube, a square and project the front view which is also a square. Assume that a cube is placed with its base on HP.
2. Reproduce the front view such that it makes an angle of 35° with HP keeping 2’3’on line XY.
3. Project the next top view.
4. Reproduce the second top view such that edge 23 makes an angle of 60° with the VP.
5. Project the final front view.

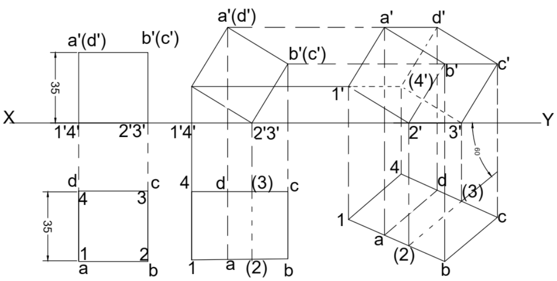


Figure 3.21: Projection of a Cube

Example 3.4: A pentagonal pyramid, 35 mm base and 60 mm long axis is suspended freely from one corner of its base so that the plane on which the axis lies is parallel to VP. Draw the views.

Solution: The following steps are used to draw the projection of given pentagonal pyramid. See figure 3.22.

1. Draw the top view, a regular pentagon keeping ab perpendicular to line XY. Assume a pyramid is placed with its base on HP.
2. Project the front view and locate the CG position which is 1/4*th* of the height from the base.
3. Keeping g’d’ vertical, now project the second front view.
4. Project all the points downwards and sideward to draw the top view and the side view.

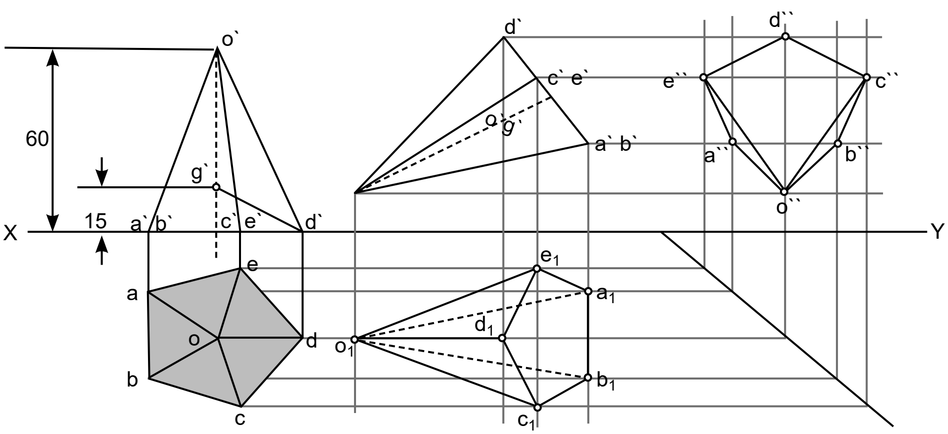


Figure 3.22: Projection of a Pentagonal Pyramid

Example 3.5: A square pyramid of 40 mm base side and axis of 60 mm length rests on it apex on HP. Draw the projections if one slant edge is vertical and triangular face through it is perpendicular to VP.

Solution: The following steps are used to draw the projection of a given square pyramid. See figure 3.23.

1. Draw the top view, the square which shows the TL.
2. Draw the projections to draw the front view. The pyramid is resting on its apex, so the view will be an inverted triangle.
3. Reproduce the second front view keeping o'p' vertical. Draw the projections to form the top view.
4. Reproduce the second top view keeping p_1s_1 perpendicular to VP.
5. Project the final front view.

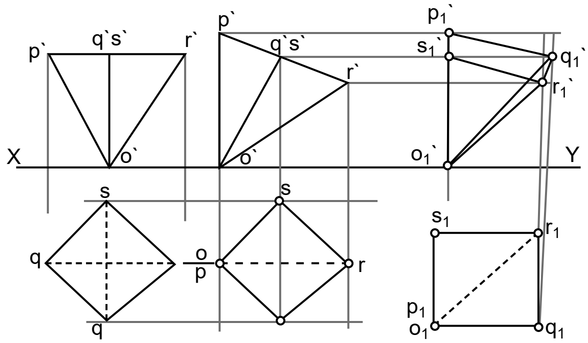


Figure 3.23: Projection of a Square Pyramid

3.4 FLOOR PLAN

A floor plan an aerial view is a two-dimensional scale drawing of the planning, size and direction of rooms, doors, walls and windows including the location of the heating and cooling facilities, electric lines and plumbing.

It is necessary to define the space and its limitations before starting the planning and construction. A floor plan should also meet the needs of the family which may include their preferences, storage requirements; specific features for particular rooms etc. such as keeping bedrooms and libraries away from noise. It is also important to measure living area the windows, closets, doors and electrical outlets.

Features of a floor plan:

(i) Floor plan should be such that it fits the priorities and living patterns. Example keeping kitchen near the dining area.

(ii) Flexibility should be there in floor plan so that any change could be made easily.

(iii) Versatility and proper sizing of regions such as rooms, hallway etc is necessary to accommodate more people and furniture.

(iv) An ideal layout keeping a balance between practical considerations and architectural designs to ensure safety and comfort.

Following are the steps used for drawing floor plan:

1. First select an area to draw the plan including dimensions.
2. Take measurements of walls, doors and furniture to design accurate floor plan. Make sure that total area fits in the design.
3. Add walls for each room in the building and scale them properly.
4. Add features to the space like doors, windows along with other important appliances that must be placed in a specific location.
5. Add furniture, if required as per the plan

Example drawing of floor plan

Following figure 3.24, drawing of a ground floor plan showing the dimensions and location of each room, position of doors, windows, bed, sink,closets,location of furniture etc.

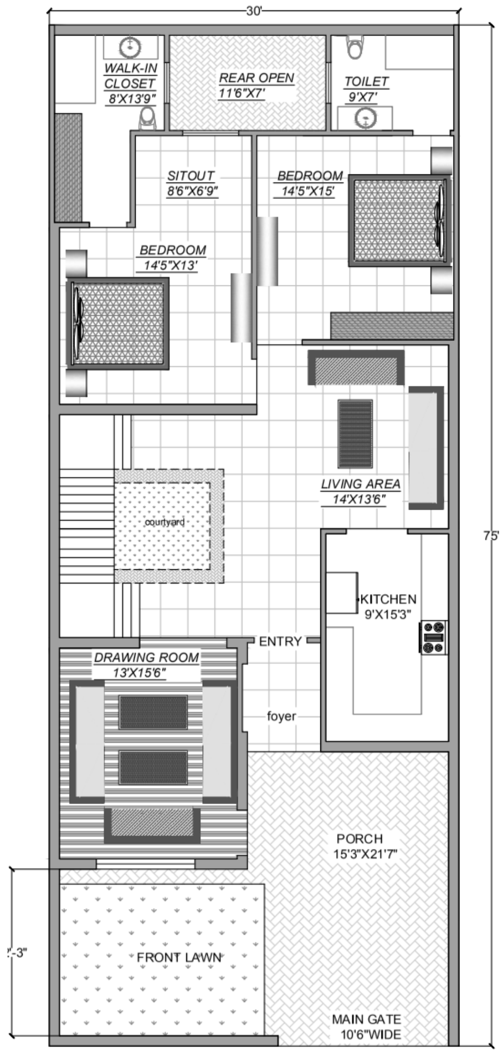


Figure 3.24: Drawing of Floor Plan

UNIT SUMMARY

- A solid has three dimensions, viz. length, breadth and thickness. To represent a solid on a flat surface having only length and breadth, at least two orthographic views are necessary.
- A polyhedron is defined as a solid bound by planes called faces. When all the faces are equal and regular, the polyhedron is said to be regular.
- Solids of revolution are generated by rotating a plane surface about one of its sides.
- A solid is said to be a right solid when its axis is perpendicular to its base.
- A solid is said to be an oblique when its axis is inclined at an angle other than 90° to its base.
- When the solid is placed with the base on HP position, in the top view, the base will be projected in its true shape.
- When the axis of the solid is perpendicular to VP, the front view is drawn first and then the top and side views are drawn from it.
- If the axis of a solid object is parallel to both the HP and VP and perpendicular to the profile plane, the side view is drawn first.

EXERCISES

3.1 Multiple Choice Type Questions

1. Axis of which of the following solids is perpendicular to their bases?
   (a) Oblique solid (b) Right solid
   (c) Both (a) and (b) (d) None of the above
2. Following is (are) solids of revolution.
   (a) Sphere (b) Cone
   (c) Cylinder (d) All of the above
3. When the solid is cut by a plane inclined to its base then it is known as
   (a) Full solid (b) Truncated solid
   (c) Frustum of solid (d) Half solid
4. _______ is a kind of polyhedron having two parallel identical faces or bases.
   (a) Pyramid (b) Prism
   (c) Solids of revolution (d) None of the above
5. A ___ is type of polyhedron having a base and an apex.
   (a) Prism (b) Solids of revolution
   (c) Pyramid (d) None of the above
6. Which of the following position is not possible in solids
   (a) Axis of a solid parallel to HP, perpendicular to VP
   (b) Axis of a solid parallel to VP, perpendicular to HP
   (c) Axis of a solid parallel to both HP and VP
   (d) Axis of a solid perpendicular to both HP and VP

7. The minimum number of orthographic view required to represent a solid on a flat surface is _________.
   (a) 1 (b) 2
   (c) 3 (d) 4
8. If a solid's axis is perpendicular to one of the reference planes then the projection of solid on to the same plane gives the true shape and size of its ___________.
   (a) lateral geometry (b) base
   (c) cross-section (d) surface
9. When the axis of solid is perpendicular to H.P, the ______view should be drawn first and ____ view then projected from it.
   (a) front , top (b) top, side
   (c) side, front (d) top, front
10. When the axis of solid is parallel to H.P and V.P, then ______view should be drawn first and ____ and _______view then projected from it.
   (a) front, top, side (b) top, side, front
   (c) side, front, top (d) top, front, side
11. The front view, side view and top view of a cylinder standing on a horizontal plane _______.
   (a) circle, rectangle and rectangle
   (b) rectangle, rectangle and circle
   (c) rectangle, circle and rectangle
   (d) circle, triangle and triangle
12. When the axis of the solid is parallel to both HP and VP the view which reveals the true shape of the base is
   (a) Front view (b) Top view
   (c) Side view (d) Both top and side view
13. The number of stages that are necessary to get the orthographic views of a solid having inclination to both the reference plane.
   (a) 1 (b) 2
   (c) 3 (d) 4
14. Front view of a cube resting on HP on one of its faces, and another face parallel of VP, is
   (a) Rectangle (b) Square
   (c) Parallelogram (d) Rhombus
15. If frustum of a cone is placed on HP on its base, its top view will consist of
   (a) a point (b) a circle
   (c) two circles (d) an ellipse and a circle

Answer Keys (Exercise: 3.1)

1. (b), 2. (d), 3. (b), 4. (b), 5. (c), 6. (d), 7. (b),
8. (b), 9. (d), 10. (c), 11. (b), 12. (c), 13. (c), 14. (b), 15. (c)

3.2 Short and Long Answer Type Questions

Category I

1. Define solids and its classifications.
2. Define polyhedral and its classifications.
3. Define solid of revolutions and its types.
4. What is the difference between a prism and a pyramid?
5. What is the difference between frustum and truncated?
6. Write the steps used to draw projection of a solid object when its axis is perpendicular to H.P.
7. Write the steps used to draw projection of a solid object when its axis is perpendicular to V.P.
8. Write the steps used to draw projection of a pentagonal prism when its axis is parallel to H.P and V.P.
9. Write the steps used to draw projection of a cylinder when its axis inclined to both H.P and V.P.
10. Fill in the blanks:
    (i) An object having three dimensions is called a ______
    (ii) The solid which is generated when a rectangle is revolved about one of its sides is a ______
    (iii) A solid having four equal equilateral triangle faces is called ________

Category II

11. Classify different types of solids commonly used in engineering drawings.
12. What is floor plan ? How to create floor plan drawings ?
13. Explain the features of floor plan drawings.
14. Explain in details applications of projection of solids in industries.
15. Explain the terminology of projection of solids.

3.3 Practical Exercises

1. Draw the projections of a pentagonal prism whose base is 30 mm and axis is 60 mm long. It is resting on one of its rectangular faces on H.P. and axis inclined at 45° to V.P.
2. A hexagonal pyramid of 30 mm side and 60 mm axis is resting with one of its edge of base on H.P. and makes an angle of 35° with H.P. and is parallel to V.P. Draw its projections.
3. Draw the projections of a right circular cylinder of 25 mm radius and 60 mm axis resting on one point of base circle on V.P. and its axis makes an angle of 50° with V.P. The front view of the axis is inclined at 30° to H.P.
4. A cone of base radius 25 mm and 60 mm axis is resting on H.P. on one of its face.The axis is parallel to both the reference planes. Draw the projections.
5. A cone of base diameter 40 mm and axis height 60 mm rests on the ground on a point of its base circle such that the axis of the cone is inclined at 40° to the HP and 30° to the VP. Draw its front and top views.

6. Draw the projections of a hexagonal pyramid of side of base 30 mm and axis 60 mm long resting on one of its base edges in HP with its axis inclined at 30° to H.P. and the top view of axis is 45° to VP.
7. Draw the projections of a pentagonal prism, base 25 mm side and axis 50 mm long resting on one of its rectangular faces on HP, with the axis inclined at 45° to VP.
8. An equilateral triangular prism 20 mm side of base and 50 mm long rests with one of its shorter edges on HP such that the rectangular face containing the edge on which the prism rests is inclined at 30° to HP. The edge on which prism rests is inclined at 60° to VP. Draw its projections.
9. Draw the top and front views of a rectangular pyramid of sides of base 40 × 50 mm and height 70 mm when it lies on one of its larger triangular faces on HP. The longer edge of the base of the triangular face lying on HP is inclined at 60° to VP in the top view with the apex of the pyramid being nearer to VP.
10. A pentagonal pyramid with 25 mm side base and 65 mm height has one of its slant faces on the horizontal plane and the edge of the base contained by that slant face makes an angle of 25° to the V.P. Draw the projections of the pyramid.

KNOW MORE

Floor Plan Online App: RoomSketcher

Draw floor plans online using RoomSketcher App., with RoomSketcher app, it's easy to draw floor plans. RoomSketcher works on PC, Mac and tablet and projects synch across devices so that user can access floor plans anywhere. Draw a floor plan, add furniture and fixtures, and then print and download to scale.

When floor plan is complete, create high-resolution 2D and 3D Floor Plans that can be print and download to scale in JPG, PNG and PDF. In addition to creating floor plans, app can also create stunning 360 Views, beautiful 3D Photos of designs, and interactive Live 3D Floor Plans that allow users to take a 3D walkthrough of floor plan.

DESIGN ACTIVITY

Developing the following solids with the help of cardboard/ thick paper and then draw orthographic views of those shapes—which are the views, such as top, front and side.

1. Cube and cuboid
2. Prisms & pyramids (triangular, square, pentagonal and hexagonal)
3. Right circular cylinder and cone

INTERESTING FACTS

- Increasing the focal length and distance of the camera to infinity in a perspective projection results in an orthographic projection.
- The tetrahedron is one of the types of pyramid, which is a polyhedron with triangular faces connecting the base to a common point and a flat polygon base. A tetrahedron has a triangular base, thus is also referred to as triangular pyramid.

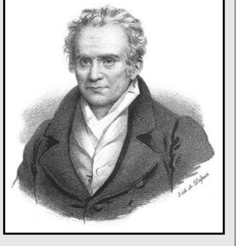
APPLICATIONS

The shape of a solid is described orthographically by drawing its two orthographic projections, usually, on the two principal planes of projection i.e., HP and VP.

Design and development involve creating working drawings and parts lists to enable a third party to manufacture the design. They are used to show an object from every angle to help manufacturers plan production.

CASE STUDY

Father of Orthographic Projection



Gaspard Monge (1746 – 1818) was a french mathematician and father of descriptive geometry. He was devised a system that could be used to communicate an object to anyone across the world. This system is called Orthographic projection and was quickly adopted by army engineers. However France was at war and Monge's system was kept top secret. It wasn't until many years later that Monge was allowed to publish and teach his system. Shortly this system spread across and revolutionized world industry.

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

1. https://youtu.be/4mxUshGia-s
2. https://youtu.be/Cw1qTI2f2nY
3. https://youtu.be/r4Ok0F41oM4?t=181
4. https://youtu.be/rgsD3C4uQoM

4 Sectional Views of Solids

UNIT SPECIFIC

This module, "Sectional View of Solids", will allow students to learn and understand various methods of sectioning, sectional views and development of surfaces of solids such as prism, cylinders, pyramids, and cones. The module also covers the sectional orthographic views of objects used in various industries.

RATIONALE

The knowledge of sections of solids and their development is very essential to instill a clear understanding of fabricated sheet metal parts and their practical application in technology and industry.

PREREQUISITE

Knowledge of basic geometric solids and their orthographic projections.

UNIT OUTCOMES

Upon successful completion of this module, the students will be able to:

- U4-O1: Create section of solids.
- U4-O2: Create sectional views of solids such as prisms, pyramids, cylinders and cones.
- U4-O3: Develop lateral surfaces of solids.
- U4-O4: Apply development technique for industrial objects.

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-4 Outcomes	Expected Mapping With Course Outcomes *(1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)*				
	CO-1	CO-2	CO-3	CO-4	CO-5
U4-O1	2	3			
U4-O2	2	2			
U4-O3	3	2			
U4-O4		2			3

4.1 INTRODUCTION TO SECTION OF SOLIDS

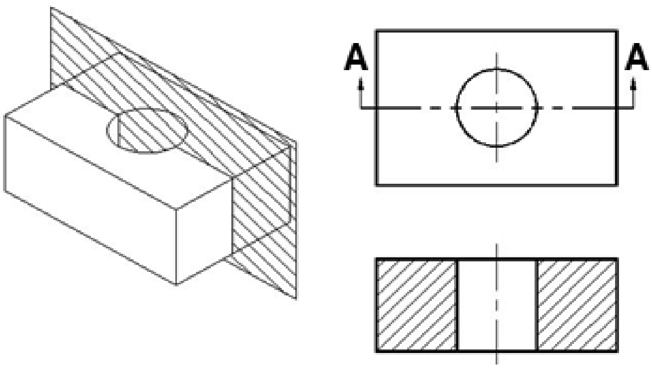
In orthographic views the outline of visible features are drawn in dark and continuous lines, whereas the features which are not seen are described by using a line of dashes (hidden lines). The part between the observer and the cutting plane is assumed to be removed and the interior details are visible. The imaginary plane is called a section *plane* or a *cutting plane.* The surface produced by cutting the object by the section plane, is called the *section.* The projection of the section along with the remaining portion of the object is called a *sectional view.* Sometimes, only the word section is also used to denote a sectional view.

4.2 TYPES OF SECTIONAL VIEWS

Sectional views reveal hidden details in an engineering drawing, these views assume that a cutting plane has removed portions of the object represented by the drawing, displaying the appropriate section of the interior. The different types of sectional views assist them in this process are as follows:

4.2.1 Full Sections

When a cutting plane A-A passes entirely through an object, the resulting section is called a full section as illustrates in figure 4.1.



Section Views

Figure 4.1: Full Section View

Features:

(a) The arrows at the end of the cutting plane line indicate the direction of sight for the sectional view.

(b) The cutting plane line may be omitted if the view is in orthographic position.

(c) All visible edges and contours behind the cutting plane should be drawn; otherwise, a section will appear made up of disconnected or unrelated parts.

(d) Hidden lines should be omitted in sectional views. In most cases, they add to confusion on a drawing. However, hidden lines may be used if necessary for clarity, or if reduce the need for an additional view.

(e) Section-lined areas are to be completely bound by visible lines-never by a hidden line.

(f) Never allow a visible line to cross over a section-lined area.

4.2.2 Half sections

If the cutting plane only passes halfway through an object, and one quarter of the object is removed, the resulting section is a half section as illustrated in figure 4.2.

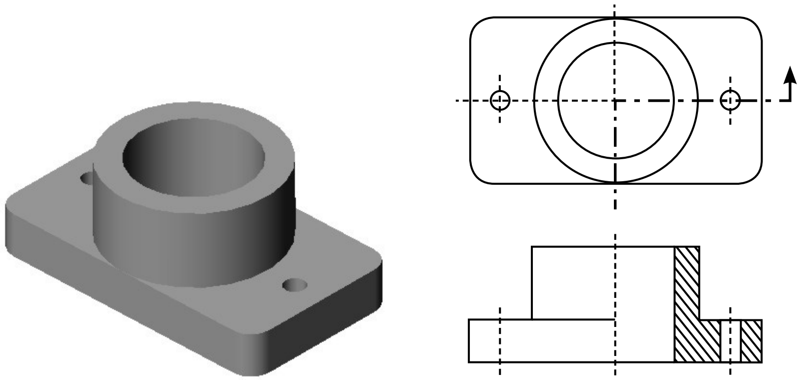


Figure 4.2: Half Section View

Features:

(a) Shows the internal and external features in the same view.
(b) Two cutting planes are passed at right angles to each other with the cutting plane passing only halfway through the object and one quarter of the object being removed.
(c) Is used when the object is symmetrical.
(d) Hidden lines can be omitted from the unsectioned half by a centerline.
(e) The sectioned half is separated from the unsectioned half by a centerline.

4.2.3 Offset Sections

In an offset section several features of an object that are not in a straight line, can be included in a single section. The cutting plane line is bent or "OFFSET" to pass through the features of the part (that need to be included) as illustrated in figure 4.3.

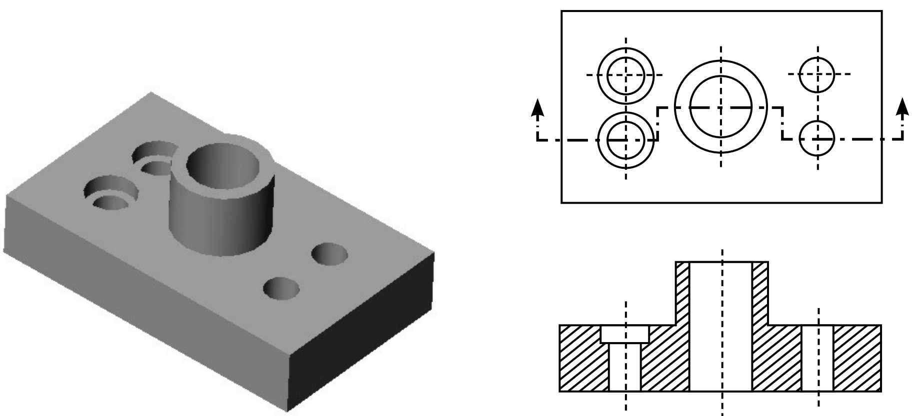


Figure 4.3: Offset Section

Features:

a. When sectioning through irregular objects, it is often desirable to show features that do not lie in a straight line by "offsetting" or bending the cutting plane. Such a section is called an offset section.

b. The cutting plane is bent at one or more 90-degree angles so that it will pass through important features.

c. The change of plane that occurs when the cutting plane is bent 90 degrees is not represented with lines in the sectional view.

4.2.4 Revolved Sections

A revolved section shows the shape of an object, by rotating a section 90 degrees to face the viewer. Figure 4.4 illustrates the revolved section.

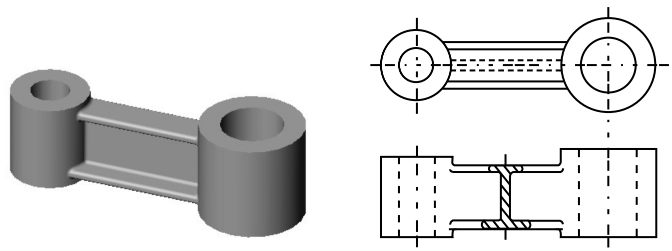


Figure 4.4: Revolved Section

Features:

a. Used to represent the cross sectional shape of a handle, spoke, or other elongated features.

b. Superimpose the sectional view on the orthographic view.

c. Make the view "stand out" by using short break lines or allow the orthographic lines to touch the sectional drawing.

d. The cross section is not distorted to fit, but rather is drawn true size and shape unless it is also removed.

4.2.5 Removed Sections

A section removed from its normal projected position in the standard arrangement of views is termed a "removed" section as illustrated in figure 4.5. Such sections are labeled SECTION A-A, SECTION B-B, etc., corresponding to the letter designated at the ends of the cutting plane line.

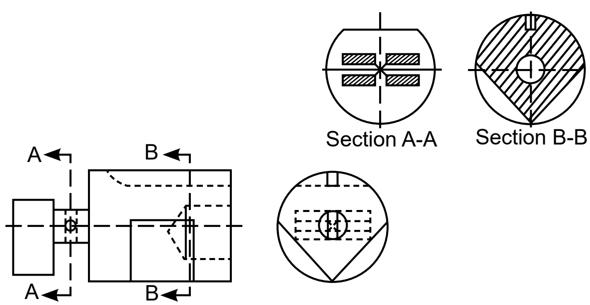


Figure 4.5: Removed Section

4.2.6 Broken-out Sections

A broken-out section is part of an existing drawing view and not a separate view. Material is removed to a specified depth to expose the inner details. In the figure 4.6, the broken-out section is removed by a freehand break line.

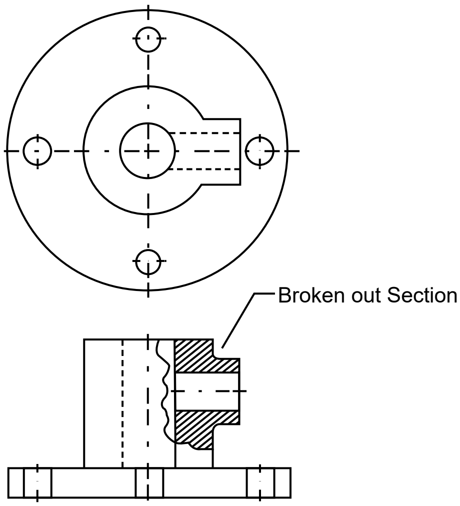


Figure 4.6: Broken-out Section

4.3 SECTIONAL PLANES

In orthographic views the section planes are represented by their traces. These section planes are so chosen that they are perpendicular to one of the reference planes and inclined to the other, thus indicative of the trace of the section which will cut the object in that view. It is important to note that the projection of a section plane, on the plane to which it is perpendicular, is a straight line. This line will be parallel, perpendicular or inclined to the reference line.

4.3.1 Representation of Sectional Plane

A section plane is represented on a drawing by a heavy long-short-short-long line terminated with arrows as shown in figure 4.7. The direction of the arrow indicates the line of sight. The sight arrows at the end of the cutting plane are always perpendicular to the cutting plane.

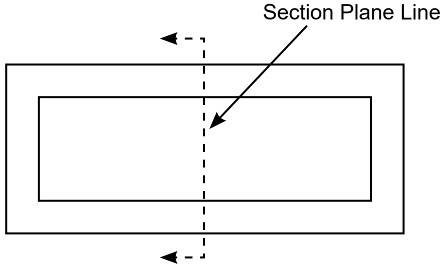


Figure 4.7: Representation of Sectional Plane

4.3.2 Section lining

Section lines *used to indicate where the cutting plane cuts the material. Section lines are thin and the symbols (type of lines) are chosen according to the material of the object as illustrated in figure 4.8. Section lines are generally drawn at a 45° angle.*

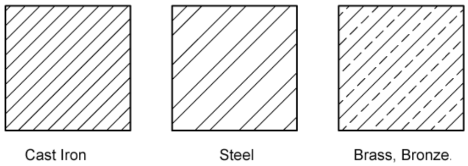


Figure 4.8: Section Lining

4.3.3 Types of Sectional Plane

There are mainly four types of section planes, depending on whether the section plane is parallel, perpendicular or inclined to the other reference plane.

1. **Section plane perpendicular to V.P. and parallel to H.P.**
   In this case the sectional plane is a vertical trace and will be a straight line parallel and above line XY as shown figure 4.9.

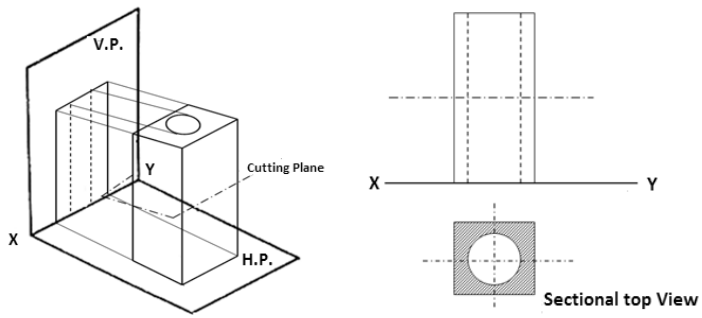


Figure 4.9: Section Plane Perpendicular to V.P. and Parallel to H.P.

2. **Section plane perpendicular to H.P. parallel to V.P.**
   In this case the sectional plane is a horizontal trace and is a straight line parallel and below to line XY as shown in figure 4.10.

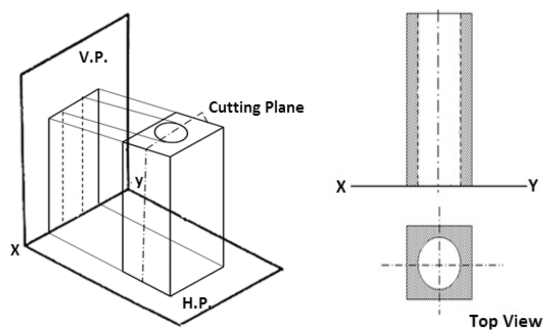


Figure 4.10: Section Plane Perpendicular to H.P. and Parallel to V.P.

3. **Section plane is perpendicular to H.P. and inclined to V.P.**
   In this case the sectional plane is a horizontal trace and will be a straight line inclined to line XY and lies below line XY as shown in figure 4.11.

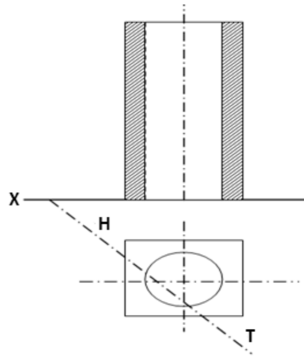


Figure 4.11: Section Plane Perpendicular to H.P. and Inclined to V.P.

4. **Sectional plane is perpendicular to V.P. and inclined to H.P.**
   In this case the sectional plane is a vertical trace and will be a straight line inclined to line XY and lies above line XY as shown in figure 4.12.

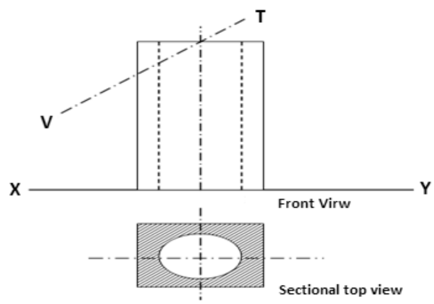


Figure 4.12: Sectional Plane is Perpendicular to V.P. and Inclined to H.P.

4.4 TRUE SHAPE OF THE SECTION

The projection of the section on a plane parallel to the section plane will show the true shape of the section. It is obtained by viewing the object normal to the cut surface and projecting it on a plane, parallel to the section plane. These are the methods used to find the true shape of a section:

1. When the section plane is parallel to the HP or the ground, the true shape of the section will be seen in a sectional top view.
2. When the section plane is parallel to the VP, the true shape will be visible in the sectional front view.
3. When the section plane is inclined, the section has to be projected on an auxiliary plane parallel to the section plane, to obtain its true shape.
4. When the section plane is perpendicular to both the reference planes, the sectional side view will show the true shape of the section.

4.5 SECTIONING TECHNIQUES

1. First draw the plan and elevation, and if required then draw the end elevation also.
2. Decide in which view the section plane will be seen as a line view.
3. Draw the sectional view at the desired position and inclination.
4. Mark the edges of the solid which are cut by the sectional plane. The points of the visible edges will be marked first on these edges, and then on the edges which are hidden.
5. Join the points, keeping in mind the sequence.
6. The portion of the object which comes between the plane of the section and the observer is assumed to be removed; hence it is represented by lighter lines.

SOLVED EXAMPLES

Example 4.1: A triangular prism edge of base 30 mm and axis 50 mm long is lying on HP on one of its rectangular faces with its axis inclined at 30° to VP. It is cut by a horizontal plane at a distance of 12 mm above HP. Draw its elevation and sectional plan.

Solution: Following steps are used to draw the elevation and sectional plan of given triangular prism as illustrate in figure 4.13:

1. First draw the orthographic view of triangular prism assuming prism in the simple position.
2. Draw the projection, by tilting plan by 30°.
3. The section plane is 12 mm above HP, so draw the section plane line above XY at a distance of 12 mm in the tilted elevation.
4. The section plane cuts the solid at 4 points on edges $a'_1\ c'_1$, $b'_1\ c'_1$, $1'_1\ 3'_1$ & $2'_1\ 3'_1$.
5. Draw projectors from $a'_1\ c'_1$, $b'_1\ c'_1$, $1'_1\ 3'_1$ & $2'_1\ 3'_1$ and get the corresponding points in the tilted plan.
6. Join the points appropriately and shade the sectional plan.

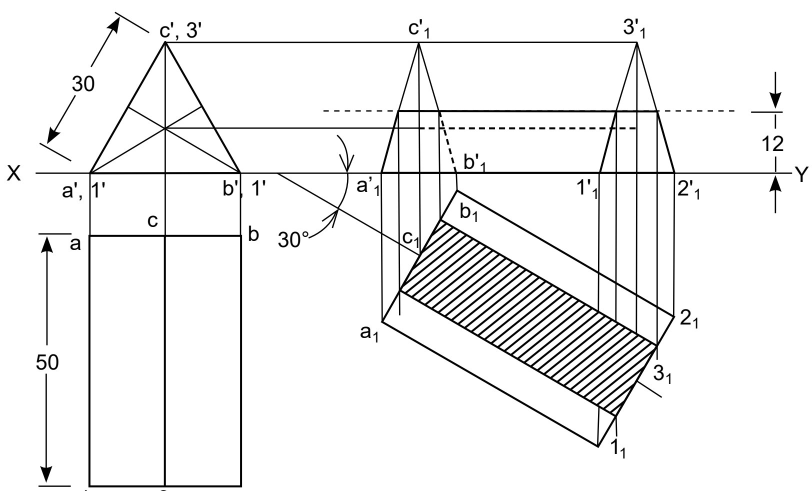


Figure 4.13: Sectional Views of Prism

Example 4.2: A square pyramid base 40 mm side and axis 65 mm long has its base on HP and all the edges of the base are equally inclined to VP. It is cut by a section plane perpendicular to VP and inclined at 45° to HP and bisecting the axis. Draw its sectional plan, sectional end elevation and true shape of the section.

Solution: Following steps are used to draw the sectional view and true shape of square pyramid, illustrate in figure 4.14:

1. First draw the orthographic view of square pyramid assuming it in the simple position.
2. Draw the section plane inclined at an angle 45° to XY and passing through the centre of the axis (i.e., 32.5 mm from XY).
3. The section plane cuts the solid at four points s1 in the edge AV, s2 at BV, s3 at CV and s4 at DV
4. Draw the projectors from s1, s2, s3, & s4 and get the corresponding points in the respective edges in the plan viz., s1, s2, s3, & s4
5. The section Point s2 & s4 are in the line perpendicular to XY. To transfer them take those points to the any one of the extreme edge" c'v' " (or a'v') and project it on "cv" (or av) and transfer that to points to the respective edges "bv" and "dv"
6. Join the points s1, s2, s3, & s4 in cyclic order and hatch the portion to show it as sectional plan
7. True shape of section:
   (a) Draw a line parallel to section plane in the front elevation at a suitable distance.
   (b) Draw projectors s1, s2, s3, & s4 perpendicular to X1Y1.
   (c) Measure the distance of s1, s2, s3, & s4 from XY in the plan and mark that distance in the respective projectors drawn perpendicular to X1Y1 and get the points s1, s2, s3, & s4.
   (d) Join the points in the cyclic order and hatch the portion.

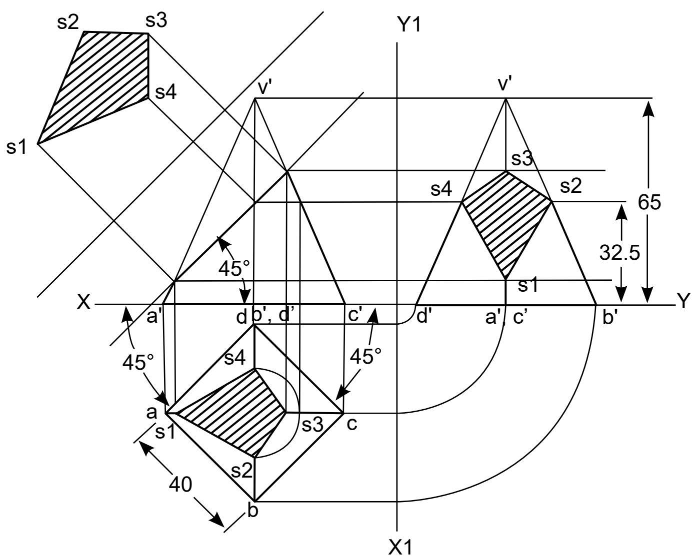


Figure 4.14: Sectional Views and True Shape of Square Pyramid

Example 4.3: A right circular cone of 40 mm diameter of the base and 60 mm altitude stands on HP. A plane normal to HP and inclined at 45° to VP cuts the cone at a distance of 10 mm from its axis. Draw the sectional elevation and true shape of the section.

Solution: Following steps are used to draw the elevation and true shape of right circular cone, illustrate in figure 4.15:

1. First draw the orthographic view of cone, assuming it in the simple position.
2. Draw the section plane inclined at an angle 45° to XY and passing at a distance of 10 mm from the centre of circle. With vertex v as centre and 10 mm as radius draw a circle and the section plane should be drawn as tangent to the circle.
3. The section plane will cut the base and generators of cone at point's s1, s2, s3, s4, & s5 as shown in the figure.
4. Get the section plan and true shape of section as explained in the previous example (4.2).

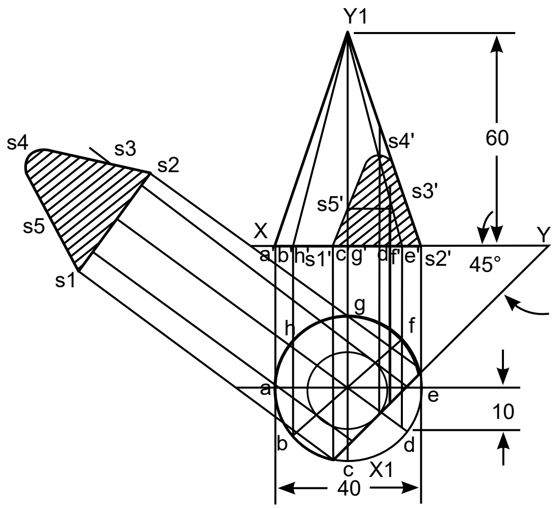


Figure 4.15: Sectional Views and True Shape of Right Circular Cone

Example 4.4: A cylinder of 40 mm diameter, 60 mm height and having its axis vertical, is cut by a section plane, perpendicular to the V.P., inclined at 45° to the H.P. and intersecting the axis 32 mm above the base. Draw its front view, sectional top view, sectional side view and true shape of the section.

Solution: Following steps are used to draw the section views of given cylinder as illustrated in figure 4.16:

1. As the cylinder has no edges, a number of lines representing the generators may be assumed on its curved surface by dividing the base-circle into, say 12 equal parts.
2. Name the points at which these lines are cut by the V.T. In the top view, these points lie on the circle and hence, the same circle is the top view of the section. The width of the section at any point, say c', will be equal to the length of the chord cc1 in the top view.
3. The true shape of the section may be drawn around the centre line *ag* drawn parallel to V.T. as shown. It is an ellipse the major axis of which is equal to the length of the section plane viz. *a'g'*, and the minor axis equal to the diameter of the cylinder viz. *dd1*.

4. Project the sectional side view as shown. The section will be seen as a circle because the section plane makes 45° angle with *xy.*

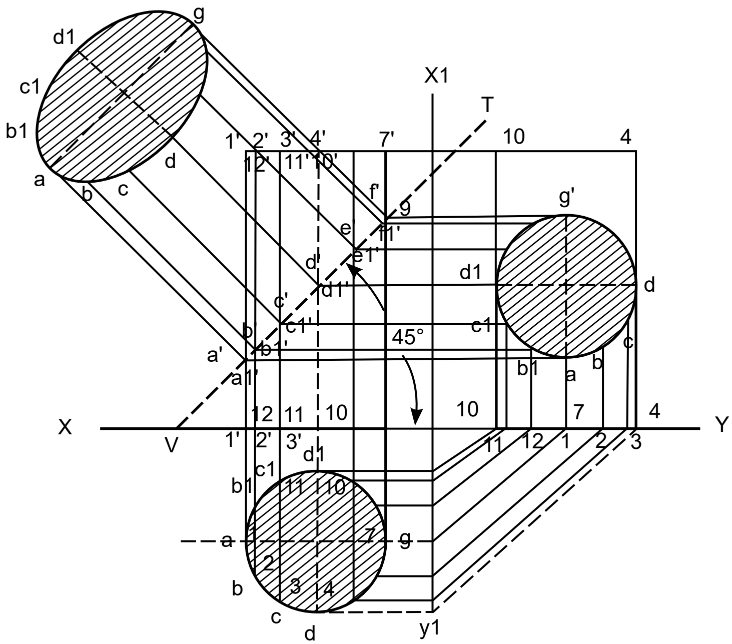


Figure 4.16: Sectional Views of a Cylinder

4.6 DEVELOPMENT OF SURFACES OF SOLIDS

A layout of the complete surface of a solid object on a plane is called the development of the surface or flat pattern of the object. The development consists of drawing successive surfaces of the object in its true size. The development of surfaces is very important in the fabrication of articles made of sheet metal. Objects such as containers, boxes, boilers, hoppers, vessels, funnels, trays etc, are made of sheet metal by using the principle of development of surfaces.

Development of Surfaces - I

4.6.1 Principle of Development

Every line on the development should show the true length of the corresponding line on the surface which is developed.

4.6.2 Methods of Development

The method to be followed for making the development of a solid depends upon the nature of its lateral surfaces. Based on the classification of solids, the following are the methods of development.

1. **Parallel-line Method:** This is used for developing prisms and single curved surfaces like cylinders, in which all the edges/generation of lateral surfaces are parallel to each other.
2. **Radial-line Method:** This is used for pyramids and single curved surfaces like cones in which the apex is taken as centre and the slant edge or generator as radius of its development.

3. **Triangulation Method:** This is used for developing transition pieces.
4. **Approximate Method:** This is used for double curved surfaces like spheres, as they are theoretically not possible to develop. The surface of the sphere is developed by approximate method. When the surface is cut by a series of cutting planes, the cut surfaces is called a zone.

4.6.3 Development of Right Regular Solids

If the surface of a solid is laid out on a plain surface, the shape thus obtained is called the development of that solid. In other words, the development of a solid is the shape of a plain sheet that by proper folding could be converted into the shape of the concerned solid. The development of different types of solids are as follows:

(i) Development of a right prism: A right prism consists of rectangular faces that are perfectly aligned to the side faces. The sides of the rectangle are equal to the length of the base edge and height of the prism.

To develop a right regular prism, follow these steps, as illustrated in figure 4.17:

1. Draw the front view and top view of prism in the required position.
2. Stretch out the horizontal line equal to the perimeter of the base and divide it into four equal parts.
3. Erect perpendiculars at each division, and mark their heights equal to the height of the prism.
4. Join the points and get the development of the right prism

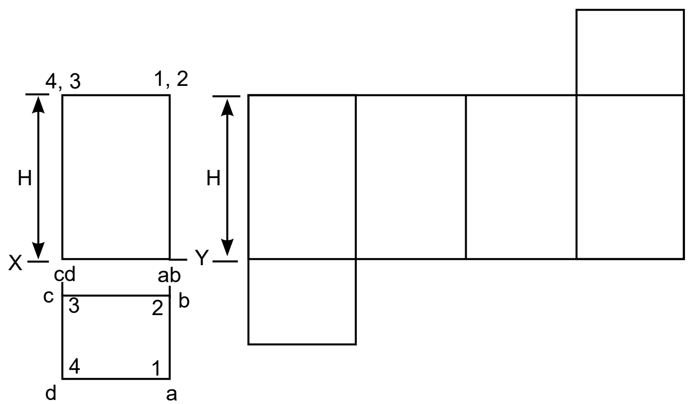


Figure 4.17: Development of Right Regular Prism

(ii) Development of a right regular cylinder: The development of a right regular cylinder consists of a rectangle with one side equal to the circumference of base circle and other side equal to the height of the cylinder.

Follow these steps to develop a right regular cylinder as illustrated in figure 4.18:

1. Draw the front view and top view of prism in the required position and divide the circle into 12 equal parts.
2. Stretch out the horizontal line equal to the perimeter of the circle and make its height equal to height of the cylinder.

3. Show the generators by dividing the circumference into 12 equal parts.

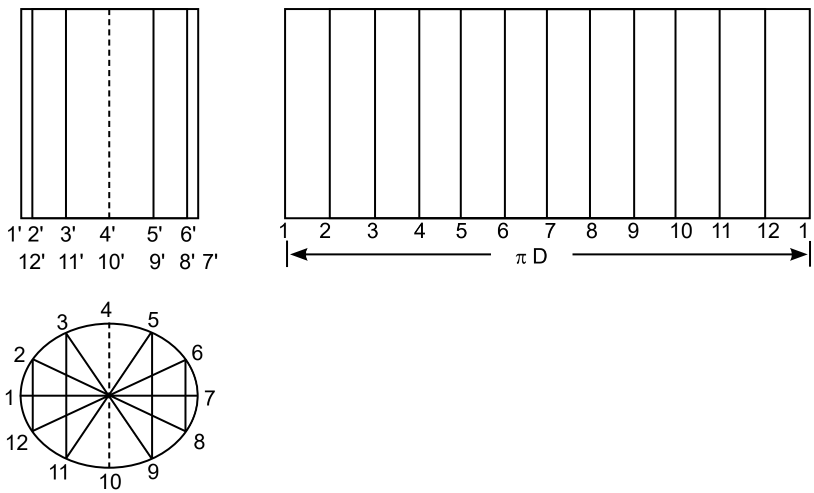


Figure 4.18: Development of Right Regular Cylinder

iii. Development of right regular pyramid: The following steps are used to develop a square pyramid resting in HP on its base and the edge of the base is perpendicular to VP, as illustrated in figure 4.19:

1. Draw the top view and front view of the pyramid.
2. None of the views will give the true shape. Taking o as center and oc as radius draw an arc to cut at c1 from c.
3. Project the point on line XY as c′1.
4. o′c′1 gives the true length of the slant edge. So taking o′c′1 as the radius and o as the centre, now draw an arc.
5. Mark four divisions as A-B,B-C,C-D,D-A.
6. Join them to the center making four triangles. These four isosceles triangles represent the lateral development of the pyramid.

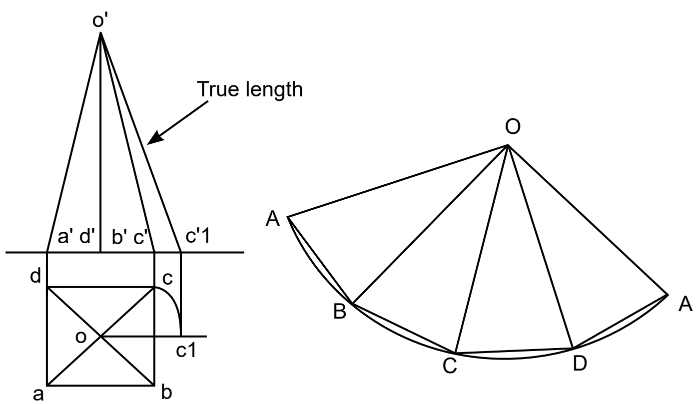


Figure 4.19: Development of Right Regular Pyramid

(iv) Development of right circular cone: The following steps are used to develop a cone, having base radius r and slant height l, resting in HP on its base, as illustrated in figure 4.20:

1. The true sizes of the triangles are found and then these triangles are drawn in order, adjacent to each other to produce the required pattern.
2. Calculate the angle of the sector, depending on the base radius and the slant height of the cone.
3. Let the radius of the base of the cone be R, the slant height of the cone be L, and the angle at the centre of the development be θ.

   θ = (Radius of the base circle / True slant length) × 360 = (R / L) × 360

   In this method of development, the surface of the object is divided into a number of triangles.

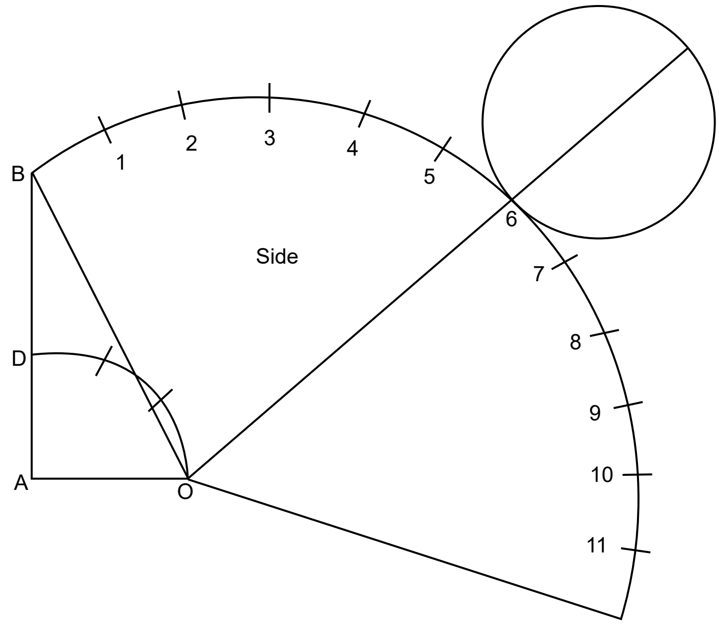


Figure 4.20: Development of Right Circular Cone

SOLVED EXAMPLES

Example 4.5: A hexagonal prism of 25 mm base and 50 mm axis is resting on H.P. with the base such that base edge is parallel to VP. The prism is cut by a plane perpendicular to V.P. and inclined at 50° to HP. The cutting plane bisects the axis. Draw the development of surface of the remaining part.

Solution: Following steps are used to develop a given hexagonal prism as illustrated in figure 4.21:

1. Draw the top view and front view of hexagonal prism.
2. Draw the section plane inclined at 50° to H.P. and name cutting points as l′, m′, n′ and o′.

3. Draw a stretch line 1-1 of length 150 mm (equal to perimeter of base) and divide it in six equal parts and marked as 1-2-3-4-5-6-1.
4. From each point draw perpendiculars of height equal to height of prism.
5. Draw projectors from l′, m′, n′, o′ parallel to base to the corresponding lines on development of prism surface.
6. Join all the points and the development of remainder prism will be formed.

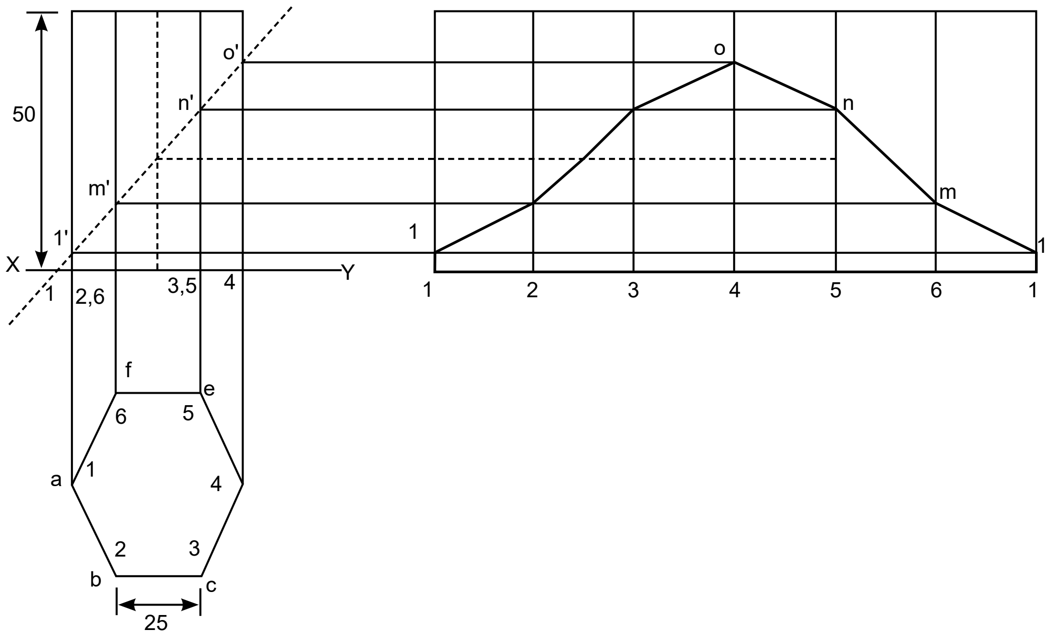


Figure 4.21: Development of Hexagonal Prism

Example 4.6: Draw the development of the lateral surface of the lower portion of a cylinder of diameter 50 mm and altitude 70 mm when it is cut by a plane perpendicular to VP and inclined at 40° to HP and passing through the midpoint of the axis.

Solution: Following steps are used to develop a given cylinder as illustrated in figure 4.22:

1. Draw the plan and elevation of the cylinder in the simple position.
2. Divide the circle (plan of cylinder) into 8 equal parts and name it as "1 to 8" to represent the plan of the base of the cylinder and "a to h" to represent the plan of the top of the cylinder.
3. Draw the section plane inclined at an angle 40° to XY and passing at a distance of 35 mm above XY (midpoint of the axis) in the front elevation and get the section points.
4. To the right side of elevation draw the development of cylinder. The rectangle 1AA1 is the development of the lateral surface of the cylinder.
5. 1A is equal to the axis length or height of the cylinder and 11 or AA is equal to the circumference of the cylinder.

6. Divide the rectangle (1AA1) into 8 equal parts and draw the generators 1A, 2B, 3C … 1A.
7. From each section points in the elevation draw lines parallel to XY to intersect the corresponding generators in the development.
8. Join all the points in the development with a smooth curve.

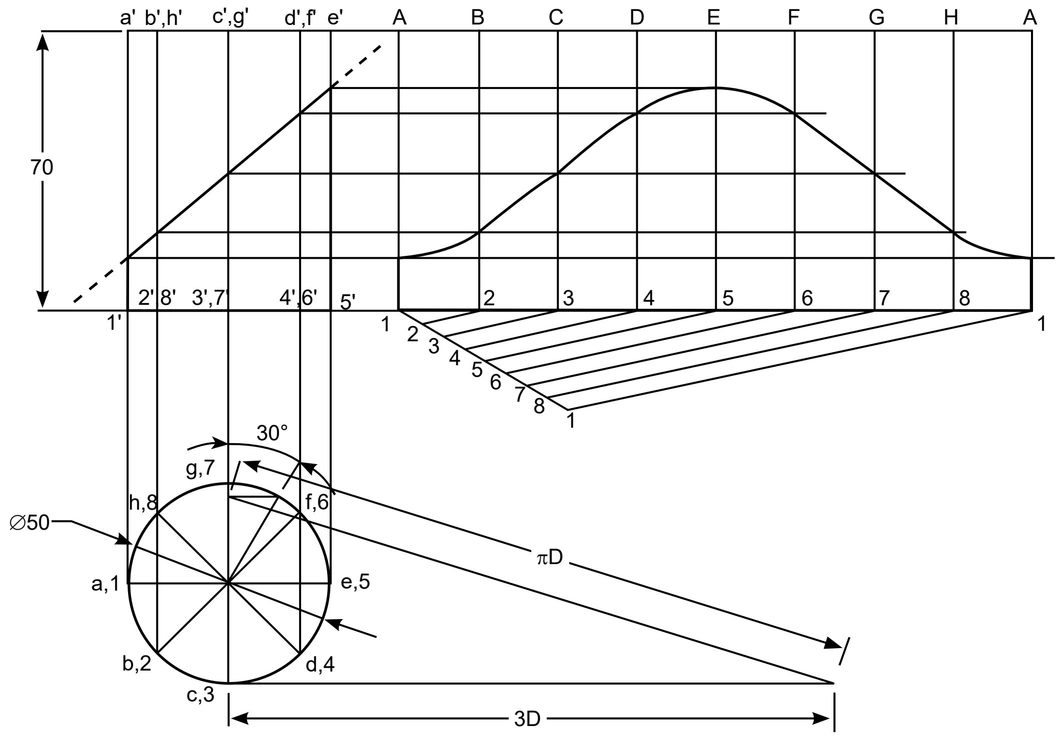


Figure 4.22: Development of Cylinder

Example 4.7: A pentagonal pyramid edge of base 30 mm and height 50 mm stands on its base on HP with an edge of base parallel to VP. A section plane cuts the pyramid at a point 30 mm above the base and makes an angle of 50° with axis. Draw the development of the truncated pyramid.

Solution: Following steps are used to develop a pentagonal pyramid as illustrated in figure 4.23:

1. First draw the pentagonal pyramid in the simple position.
2. Draw the section plane inclined at an angle 50° to axis and passing at a distance of 30 mm above XY (from the base) in the front elevation and get the section points.
3. To the right side of elevation draw the development of pyramid.
4. Draw a line "VA" parallel and equal to v′c′ the slant length of the pyramid.
5. With "v" as centre and "VA" as radius draw an arc and step off 30 mm from the point A 5 times to get the points B, C, D, E, A. Join all the points with "V" to get the slant edges of the pyramid in the development.
6. Take all section points to the extreme slat edge (v'c') and transfer all the points to the corresponding edges in the development.

7. Join all the points and complete the development.

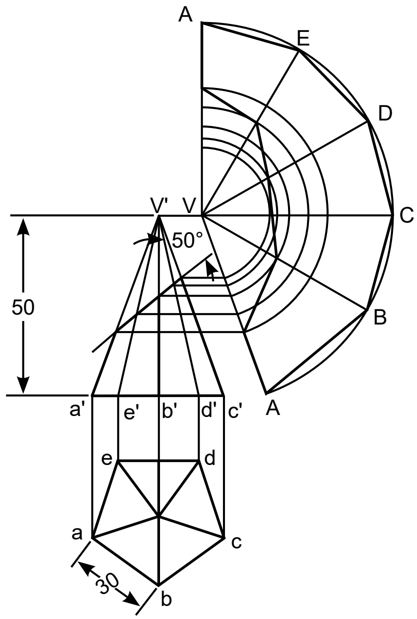


Figure 4.23: Development of Pentagonal Pyramid

Example 4.8: A cone of base 50 mm diameter and height 65 mm rests with its base on HP. A section plane perpendicular to VP and inclined at 30° to HP bisects the axis of the cone. Draw the development of the lateral surface of the truncated cone.

Solution: Following figure 4.24 illustrated the development of a cone:

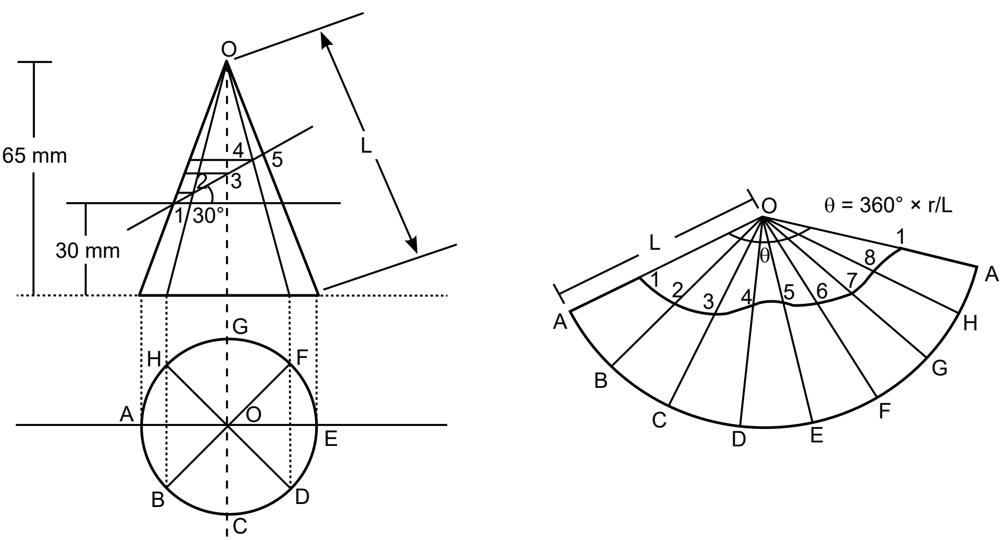


Figure 4.24: Development of Truncated Cone

UNIT SUMMARY

- A section view is drawn, cutting the object by an imaginary section or cutting plane, so as to reveal the interiors.
- The projection of the section on a plane parallel to the section plane will show the true shape of the section.
- When the section plane is parallel the HP or the ground, the true shape of the section will be seen in sectional top view.
- When the section plane is parallel to the VP, the true shape will be visible in the sectional front view.
- When the section plane is inclined, the section has to be projected on an auxiliary plane parallel to the section plane, to obtain its true shape.
- When the section plane is perpendicular to both the reference planes, the sectional side view will show the true shape of the section.
- A layout of the complete surface of a 3 dimensional object on a plane surface is called a development of pattern.
- Development of right cylinder is a rectangle of width equal to the circumference of cylinder.
- The objects such as containers, boxes, boilers, hoppers, vessels, funnels, trays etc, are made of sheet metal by using the principle of development of surfaces.
- A parallel line method is used for developing prisms and single curved surfaces like cylinders, in which all the edges/generation of lateral surfaces are parallel to each other.

EXERCISES

4.1 Multiple Choice Questions

1. A plane cut the cone in such a way that true shape of cutting portion is seen as triangle when cutting plane is cut the base and passed through _________
   (a) Midpoint of axis (b) Apex of cone
   (c) Generator of cone (d) Any point on axis
2. A cylinder standing on the HP is cut by a vertical plane parallel to the axis and away from it. The shape of the section will be.
   (a) Rectangle
   (b) Circle
   (c) Ellipse
   (d) Hyperbola
3. When base of a cone resting on V.P, is cut by a plane parallel to V.P., then the true shape is a __________ and can be seen in _________ view.
   (a) Circle, Front
   (b) Ellipse, Front
   (c) Ellipse, Top
   (d) Circle, Top

4. To obtain the true shape of the section of solid, an auxiliary plane is set __________
   (a) Inclined at an angle of 45 to a cutting plane
   (b) Inclined at an angle of 45 to H.P.
   (c) Parallel to a cutting plane
   (d) Perpendicular to cutting plane
5. A regular triangular prism is resting on HP and section plane is parallel to HP and cutting the prism the section would be a __________
   (a) Triangle (b) Rectangle
   (c) Trapezium (d) Parallelogram
6. A cube is rested on HP on one of its base such that base's diagonal is perpendicular to VP and section plane is making 45 degrees with VP and perpendicular to HP the section will be a __________
   (a) triangle (b) rectangle
   (c) trapezium (d) parallelogram
7. A triangular prism resting on one of its longest faces on HP and axis of prism is parallel to VP, the section plane is perpendicular to both VP and HP the section will be a __________
   (a) Triangle
   (b) Rectangle
   (c) Trapezium
   (d) Parallelogram
8. A pentagonal prism resting on one of its longest faces on HP and axis of prism is parallel to VP, the section plane is perpendicular to VP and inclined to HP the section will be a __________.
   (a) Pentagon
   (b) Irregular Pentagon
   (c) Rectangle
   (d) Trapezium
9. A hexagonal prism is resting on HP on one of its longest faces, axis is perpendicular to VP the section plane is perpendicular to HP and inclined to VP and cutting solid at approximately middle. The section will be like a __________
   (a) Hexagon
   (b) Irregular Hexagon
   (c) Rectangle
   (d) Trapezium
10. A hexagonal prism is resting on HP on one of its longest faces, axis is perpendicular to VP the section plane is parallel to VP and perpendicular to H.P. The section will be like a __________
    (a) Hexagon
    (b) Irregular Hexagon
    (c) Rectangle
    (d) Trapezium

11. Which method of development is employed in case of prisms?
    (a) Parallel-line development
    (b) Approximation method
    (c) Triangulation development
    (d) Radial-line development
12. Which method of development is employed in case of cones?
    (a) Parallel-line development
    (b) Approximation method
    (c) Triangulation development
    (d) Radial-line development
13. Which method is used to develop transition pieces?
    (a) Parallel-line development
    (b) Approximation method
    (c) Triangulation development
    (d) Radial-line development
14. The development of the lateral surface of a cylinder is a rectangle having one side equal to the _____________ of its base-circle and the other equal to its length.
    (a) Circumference (b) Area
    (c) Diameter (d) Radius
15. The development of the curved surface of a cone is a __________ of a __________.
    (a) Sector, Circle
    (b) Segment, Circle
    (c) Segment, Ellipse
    (d) Arc, Parabola
16. The development of the surface of a cube consists of ____ equal squares, the length of the side of the squares being equal to the length of the edge of the cube.
    (a) 4 (b) 6
    (c) 12 (d) 8
17. Knowledge of Development of surfaces of solids is required for fabrication of ________
    (a) Pipe work
    (b) Ducts
    (c) Containers
    (d) All of the above
18. The lateral surface of right regular pentagonal prism of each base edge 20 mm and height of axis 60mm is developed, we get ?
    (a) Square of each side 100 mm
    (b) Rhombus of each side 60 mm
    (c) Rectangle of length 100 mm and width 60 mm
    (d) None of the above

19. When a square prism is developed, a shape available is ________
    - (a) 1 rectangle
    - (b) 3 rectangles in series
    - (c) 5 rectangles in series
    - (d) None
20. Development of an object uses ________
    - (a) Apparent lengths
    - (b) True length
    - (c) Apparent & true lengths
    - (d) None

Answer Keys (Exercise: 4.1)

1. (b),	2. (a),	3. (a),	4. (c),	5. (b),	6. (b),	7. (a)
8. (b),	9. (b),	10. (a),	11. (a),	12. (d),	13. (c),	14. (a)
15. (a),	16. (b),	17. (d),	18. (c),	19. (d),	20. (b)	

4.2 Short and Long Answer Type Questions

Category I

1. Why do you section a solid?
2. Define Sectional View.
3. Discuss about sectional plane.
4. Define types of section plane.
5. What is the need for determining the true shape of section?
6. What are section lines?
7. What is apparent shape of section?
8. What will be the position of the cube and cutting plane to get the true shape of section as rhombus?
9. Name the methods of drawing the sectional views of a cone.
10. What is meant by development of surfaces?
11. State the practical applications of development of surfaces.
12. State the methods of development.
13. What will be the shape of the development of the lateral surface of a cone?
14. What will be the shape of the full development of a square pyramid of base side ‘a’ and slant height ‘h’?
15. An upright cone is cut by a plane which is inclined to the axis and is not parallel to a generator. The plane cuts all the generators. What is the shape of the Section?

Category II

16. Explain various types of sectional views and their applications.
17. Explain with neat sketches various types of sectional planes.
18. Explain the various methods used in development of surfaces.

19. Show the development of an equilateral triangular prism of base side A and height H.
20. Demonstrate with neat diagram, how to find the true shape of various solid objects.

4.3 Practical Exercises

1. A hexagonal prism, side of base 30 mm and axis 60 mm long rests with one of the edges of its base on HP; its axis is inclined at 45° to HP and parallel to VP. A section plane perpendicular to HP and inclined at 45° to VP passes through a point on the axis at a distance of 18 mm from one of its ends. Draw the sectional front view and the true shape of section.
2. A pentagonal pyramid, side of base 35 mm and axis 70 mm long lies with one of its triangular faces on HP and its axis is parallel to VP. A section plane perpendicular to HP and inclined at 30° to VP bisects the axis. Draw the sectional front view and the true shape of section assuming the portion containing the apex is removed.
3. A cone, base 60 mm diameter and height 80 mm stands on HP with its circular base. A section plane inclined at 45° to HP and perpendicular to VP bisects the axis of the cone. Obtain the front and sectional top views.
4. A cube of side base 30 mm rests on the HP on one of its faces with a vertical face inclined at 45° to the VP. A plane perpendicular to the HP and inclined at 45° to the VP cuts the cube. Draw the top view and sectional front view. Also draw the true shape of the section.
5. A square prism of base side 30 mm and height 75 mm rests on HP on one of its ends with two of its rectangular faces equally inclined to the VP. It is cut by a plane perpendicular to the VP and inclined at 50° to the HP meeting the axis at 20 mm from the top. Draw its elevation, sectional plan and the true shape of the section.
6. A hexagonal prism, side of base 30 mm and axis 70 mm long rests on the HP on one of its rectangular faces with its axis perpendicular to the VP. It is cut by a vertical plane inclined at 30° to the VP. Draw the top view, sectional elevation and the true shape of the section.
7. A pentagonal prism of base side 40 mm and length 80 mm is resting on one of its base edges on the HP with its axis inclined at 45° to the HP and parallel to the VP. It is cut by a plane perpendicular to the VP and inclined at 30° to the HP. The cutting plane meets the axis at 30 mm from the top end. Draw the front view, sectional top views and the true shape of the section.
8. A hexagonal prism, side of base 50 mm and axis 80 mm rests on one of its base edges on the HP. The end containing that edge is inclined at 30° to the HP and the axis is parallel to VP. It is cut by a plane perpendicular to the VP and parallel to the HP. The cutting plane bisects the axis. Draw the front and sectional top views.
9. A pentagonal pyramid of base side 25 mm and altitude 60 mm rest on its base on HP with one of the base edge perpendicular to the VP. It is cut by a plane inclined at 30° to the base. The cutting plane meets the axis at 15mm above the base. Draw the front view, sectional top views and the true shape of the section.
10. A square pyramid of base side 30 mm and altitude 45 mm rests on the HP on its base with the base edges equally inclined to the VP. It is cut by a plane perpendicular to the VP and inclined at 30° to the HP meeting the axis at 20 mm above the HP. Draw the sectional plan and the true shape of the section.

11. A cylinder of diameter 40 mm and axis length 65 mm lies on the HP on one of its generators with its axis parallel to the HP and inclined at 30° to the VP. It is cut by a plane perpendicular to the HP so that the true shape of the section is an ellipse of major axis 55 mm. Draw the sectional front view and true shape of the section.
12. A cylinder of diameter 60 mm and height 90 mm is lying on the HP with its axis parallel to both the HP and the VP. A vertical plane inclined at 25° to the VP cuts the cylinder and passes through the mid-point of the axis. Draw the sectional front view and the true shape of the section.
13. A cone of base diameter 50 mm and axis length 60 mm lies on the HP on one of its generators. The cone is cut by a vertical plane inclined at 75° to the VP and meeting the axis at 27 mm from the vertex in the plan. Draw the top view, sectional front view and the true shape of the section.
14. A cone of base diameter 50 mm and altitude 60 mm rests on its base on the HP. It is cut by a section plane perpendicular to the VP and inclined at 80° to the HP, passing through the apex. Draw the sectional plan and the true shape of the section.
15. A hexagonal prism of base side 20 mm and height 45 mm is resting on one of its ends on the HP with two of its lateral faces parallel to the VP. It is cut by a plane perpendicular to the VP and inclined at 30° to the HP. The plane meets the axis at a distance of 20 mm above the base. Draw the development of the lateral surfaces of the lower part of the prism.
16. A rectangular prism of cross section 30 x 50 mm and height 70 mm is resting on one of its ends on the HP with one of its longer edges of the base inclined at 45° to the VP. It is cut by a plane perpendicular to the VP and inclined at 45° to the HP. The plane meets the axis at a point 10 mm below the top end. Draw the development of the surfaces of the truncated lower part of the prism.
17. A square pyramid of base side 20 mm and altitude 50 mm rests on its base on the HP on its base with two sides of the base parallel to the VP. It is cut by a plane bisecting the axis and inclined at 40° to the base. Draw the development of the lateral surfaces of the lower part of the cut pyramid.
18. A cylinder of diameter of base 40 mm and height 50 mm is standing on its base on H.P., A cutting plane inclined at 45° to the axis of the cylinder passes through the left extreme point of the top base. Develop the lateral surface of the truncated cylinder.
19. A cylinder of diameter of base 60 mm and altitude 80 stands on its base. It is cut into two halves by a plane perpendicular to the V.P. and inclined at 300 to the H.P., Draw the development of the lower half.
20. A cylinder of diameter 40 mm and height 50 mm is resting vertically on one of its ends on the HP. It is cut by a plane perpendicular to the VP and inclined at 30° to the HP. The plane meets the axis at a point 30 mm from the base. Draw the development of the lateral surface of the lower portion of the truncated cylinder.

KNOW MORE

Development of transition pieces

Transition pieces are usually made to connect two different forms, such as round pipes to square pipes. These transition pieces will usually fit the definition of a non developable surface that must be developed by approximation. Triangulation development Employed to obtain the development of Transition Pieces. In this method, the lateral surfaces of the transition pieces are divided in to a number of triangles. By finding the true lengths of the sides of each triangle, the development is drawn by laying each one of the triangles in their true shapes adjoining each other. Figure 4.25 illustrate the different transition objects.

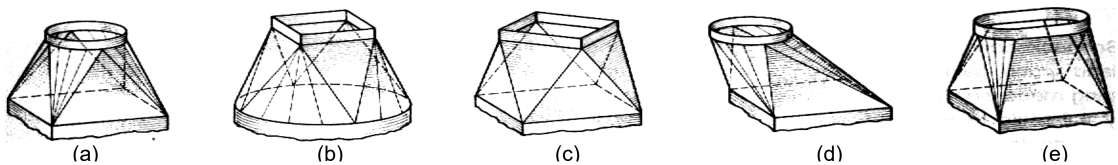


Figure 4.25: Transition pieces

DESIGN PROJECT/ACTIVITIES

Preparing the following sections of solids (prisms, pyramids, spheres, etc.) with clay, soap, thermocol, plastic, wax or any other material easily and economically available considering the cutting plane is: parallel to the base, perpendicular to the base and inclined to the base.

INTERESTING FACTS

- A primary reason for creating a section view is the elimination of hidden lines, so that a drawing can be more easily understood or visualized.
- There are so many different materials used in engineering design that the general symbol (i.e., the one used for cast iron) may be used for most purposes on engineering drawings.
- The angle at which lines are drawn is usually 45 degrees to the horizontal, but this can be changed for adjacent parts shown in the same section.

APPLICATIONS (REAL LIFE / INDUSTRIAL)

There are so many products or objects which are difficult to manufacture by conventional manufacturing processes, because of their shapes and sizes. Those are fabricated in sheet metal industry by using development technique, there is a vast range of such objects.

Some applications of development of surfaces are as follows:

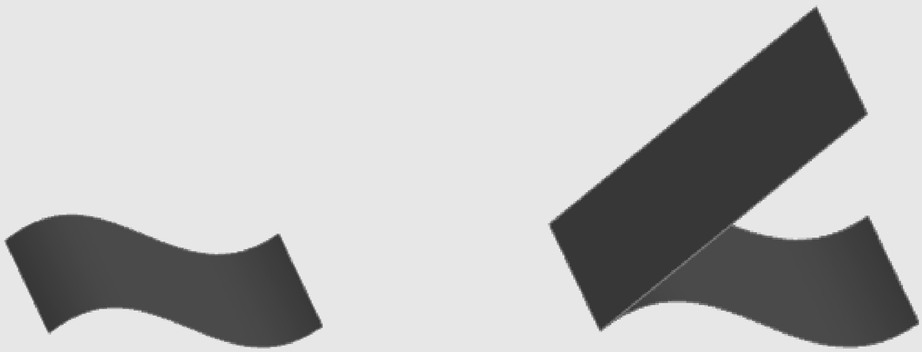
1. Boiler Shells & chimneys
2. Pressure Vessels, Shovels
3. Trays, Boxes & Cartons
4. Feeding Hoppers

5. Large Pipe sections
6. Automobiles
7. Ships
8. Air crafts

CASE STUDY

Flatten Surface using CAD Software

The CAD software can flatten developable and non-developable surfaces and faces. A developable surface is a surface that can be flattened onto a plane without distortion. Non-developable surfaces and faces deform when flattened. User can select curves and sketches on the surface to flatten and also select curves, sketches, and edges on the surface to be relief cuts or split lines. When user flattens a surface, CAD software generates a flattened surface body tangent to the edge it was flattened from as shown in below figure.



Folded surface   Flattened surface tangent to the folded surface

User can use the Measure tool to check the accuracy of flattened surface and can measure the surface area of the folded and flattened features. If there is a large discrepancy between the surface areas, the flattened feature might not be accurate then user can also use deformation plots to check for accuracy.

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

1. Bhatt N.D., Panchal V.M. & Ingle P.R., (2014), Engineering Drawing, Charotar Publishing House
2. https://youtu.be/yrUIF09uYD4
3. https://youtu.be/jzdUdQBk4Vo
4. http://www.iitg.ac.in/rkbc/ME111/Lecture11%20Sections%20of%20solids.pdf

5 Isometric Projection

UNIT SPECIFIC

This unit, "Isometric projection" covers various topics such as, principles of isometric projection, isometric scale, isometric views of planes and solids; conversion of isometric views to orthographic views and vice-versa.

RATIONALE

Isometric projection is a method for visually representing three-dimensional objects in two dimensions in technical and engineering drawings. Pictorial projections are utilized for presenting ideas that might be easily understood by persons, even without technical knowledge and training of multi-view drawing.

PREREQUISITE

Knowledge of orthographic projections.

UNIT OUTCOMES

Upon successful completion of this module, the student will be able to:

U5-O1: Understand the concept of isometric projection.
U5-O2: Create isometric views of planes and solids.
U5-O3: Draw isometric view from given orthographic views.
U5-O4: Convert 3D views to orthographic views

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-5 Outcomes	Expected Mapping With Course Outcomes *(1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)*				
	CO-1	CO-2	CO-3	CO-4	CO-5
U5-O1	2	3			2
U5-O2	2	3			2
U5-O3	2	3			2
U5-O4	2	3			2

INTRODUCTION

Isometric projection is a type of pictorial projection in which the three dimensions of a solid object are not only shown in one view but their actual sizes can be measured directly from it. "Iso" means "equal" and "metric" means measure so, in isometric projections, length, width and height are equally reduced (foreshortened).

The differences between a multi-view projection and an isometric projection are that, in a multi-view projection, only two dimensions of an object are visible in each view and more than one view is required to define the object; whereas, in an isometric projection, the object is rotated about an axis to display all three dimensions, and only one view is required.

5.1 PRINCIPLE OF ISOMETRIC PROJECTION

The principle of isometric projection can be explained by drawing orthographic projection of a cube, placed on one of its corners on the ground with solid diagonal perpendicular to the V.P. It will be seen that the front view so obtained will gives the required isometric projection of the cube.

Steps for construction of isometric projection as illustrated in figure 5.1:

1. Draw a square (a b c d) in the top view with its sides inclined at 45° to XY, the line (ac) representing the solid diagonal parallel to XY.
2. Project the front view (a′ e′ g′ c′) above XY.
3. Tilt the front view about corner (g′) so that the line (e′ c′) becomes parallel to XY.
4. Project the second top view. The solid diagonal (e′ c′) is now parallel to both H.P. and the V.P.
5. Reproduce the second top view so that the top view of the solid diagonal (e′1c′1) is perpendicular to XY.
6. Project the front view, which is the required isometric projection of the cube.

Isometric projection - Introduction

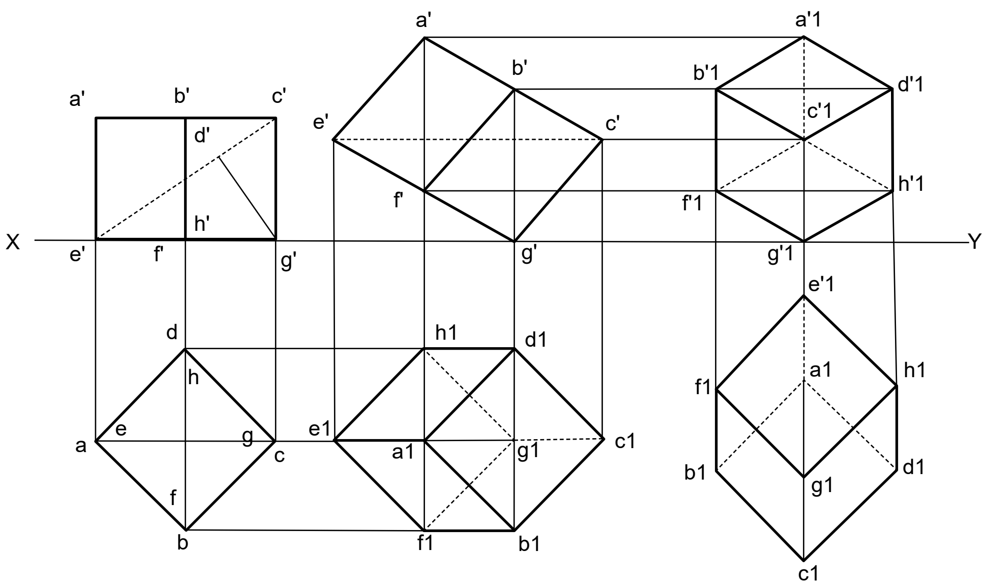


Figure 5.1: Principle of Isometric Projection

5.2 TERMINOLOGY OF ISOMETRIC PROJECTION

Figure 5.2 shows the same front view of a cube that is presented in figure 5.1, with corners named in capital letters. The rhombus ABCD represents the isometric projection of the top square face of the cube, in which BD is the true length of the diagonal. All of the edges of the cube are equally foreshortened.

Draw another square BPDQ on the same diagonal BD, the side BP being equal to the true edge length of the cube. This true length is shortened to isometric length represent by BA.

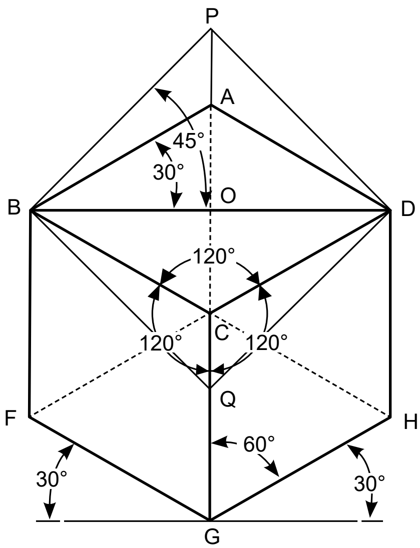


Figure 5.2: Front View of Cube

The three lines AB, AD and AE are meeting at A. These edges are mutually perpendicular to each other in the solid. Since all these edges are equally inclined to H.P, they are making and angle of 120° with each other in the plane of projection; also they are equally foreshortened and the diagonal BD retains its true length because it is parallel to VP.

- **Isometric axes:** The lines AB, AD and AE meeting at a point A and making an angle of 120° with each other are termed as isometric axes.
- **Isometric lines:** The lines parallel to the isometric axes are termed as isometric lines and lines which are not parallel to the isometric axes are known as non-isometric lines. In figure (5.2), lines AB, AD, GF, GH, BF and DH are isometric lines and the lines BD, AC, CF, BG are non-isometric lines.
- **Isometric planes:** The planes representing the faces of the rectangular prism as well as other planes parallel to these planes are termed isometric planes. In figure (5.2), ABCD, BCGF, CGHD are Isometric planes.
- **Isometric scale:** While drawing isometric projection, it is necessary to convert true lengths into isometric lengths for measuring and marking the sizes. Isometric scale is used to converts true lengths into isometric lengths (foreshortened).

5.3 CONSTRUCTION OF ISOMETRIC SCALE

Steps for construction of isometric scale as illustrated in figure 5.3:

1. Draw a horizontal line BD.
2. From B draw a line BP at 45° to represent actual or true length and another line BA at 30° to BD to measure isometric length.
3. On BP mark the point 0, 1, 2 etc to represent actual lengths.
4. From these points draw verticals to meet BA at 0′, 1′, 2′ etc.
5. The length B1' represents the isometric length of B1 and so on.

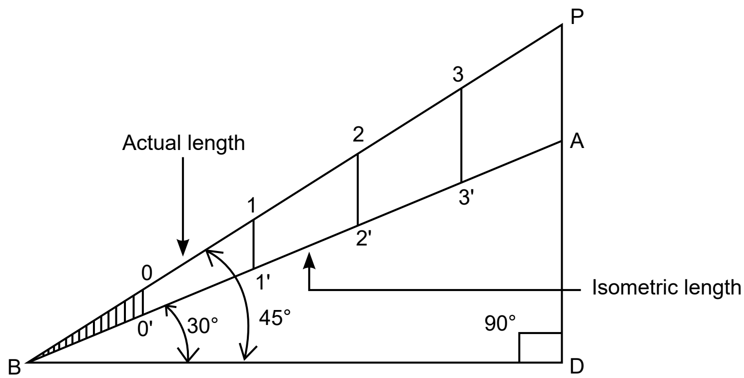


Figure 5.3: Construction of Isometric Scale

Calculation of isometric lengths:

From the above figure 5.3,

$$\frac{BD}{BP} = \cos(45) = \frac{1}{\sqrt{2}}$$

$$\frac{BD}{BA} = \cos(30) = \frac{\sqrt{3}}{2}$$

Therefore;

$$\frac{BD}{BP} \times \frac{BA}{BD} = \frac{1}{\sqrt{2}} \times \frac{2}{\sqrt{3}}$$

$$\frac{BA}{BP} = \frac{\sqrt{2}}{\sqrt{3}} = .815$$

BA = .815 *BP

Therefore, Isometric length = .815 * True length

5.4 ISOMETRIC VIEW

An isometric view is a type of 3D view that is set out using 30-degree angles. It is a type of axonometric view in which the same scale is used for every axis, resulting in a non-distorted image. Instead, a convenient way in which the foreshortening of lengths is ignored and actual or true lengths are utilized to obtain the projections, known as isometric view, or isometric perspective is normally used. The isometric view is advantageous because the measurement could be made directly from it. The isometric view of any given figure is slightly larger (approximately 22%) than the isometric projection. Since the proportions are the same, the increased size does not affect the pictorial value of this representation, and at the same time, it might be done quickly.

5.5 DIFFERENCE BETWEEN ISOMETRIC PROJECTION AND ISOMETRIC VIEW

The difference between isometric projection and isometric view is presented in following table 5.1

Table 5.1: Difference between isometric view and isometric projection

S. No.	Isometric View	Isometric Projection
1.	Drawn to actual scale	Drawn to isometric scale
2.	When lines are drawn parallel to isometric axes, the true lengths are laid off.	When lines are drawn parallel to isometric axes, the lengths are foreshortened to approximate 0.81 times the actual lengths.
3.	The measurement could be made directly from a drawing.	The isometric scale is to be used for measurement.

5.6 ISOMETRIC VIEWS OF PLANE FIGURES

The planes may be horizontal or vertical; if a plane is horizontal its sides in isometric projection will be parallel to the two sloping axes. If the plane is vertical, its one side will be parallel to sloping axis and the other parallel to the vertical axis.

5.6.1 Isometric View of a Square [Figure 5.4 (a)]

Following steps are used to draw isometric view of a given square:

1. Through any point (a), draw a vertical line such that, ad = AD.
2. From point (a), draw a line ab = AB, inclined at 30° to the horizontal and 60° to ad.
3. Complete the rhombus (abcd), which is the required isometric view of square [figure 5.4 (b)].
4. The isometric view can also be drawn in direction of another sloping axis [figure 5.4 (c)].

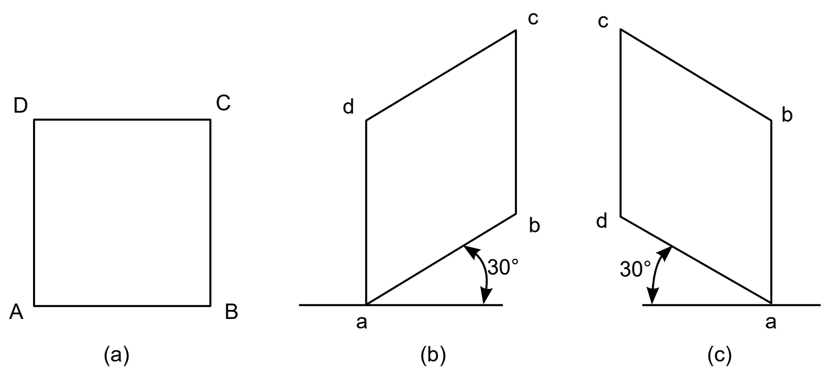


Figure 5.4: Isometric View of Square

5.6.2 Isometric View of Triangle [Figure 5.5 (a)]

Following steps are used to draw isometric view of a given triangle:

1. Enclose the triangle ABC in the rectangle ABQP.
2. Draw the isometric view of rectangle abqp [figure 5.5 (b)].
3. Mark a point ‘c’ in ‘pq’ such that (pc = PC) and (cq = CQ).
4. Draw the triangle ‘abc’, which is the required isometric view [figure 5.5 (b)].
5. Draw other direction of isometric view [figure 5.5 (c)].

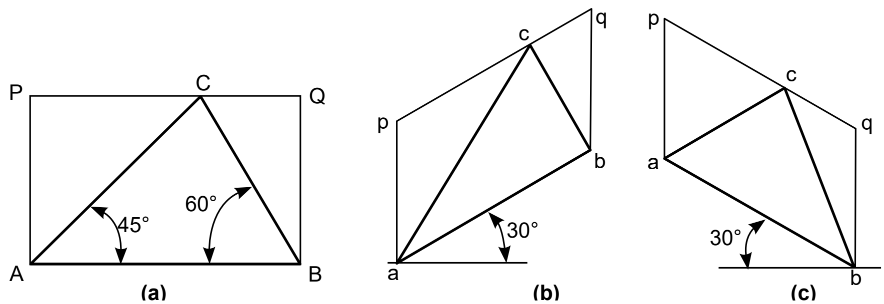


Figure 5.5: Isometric View of Triangle

5.6.3 Isometric View of Circle [Figure 5.6(a)]

Following steps are used to draw isometric view of a given circle:

1. Draw a circle with 60 mm diameter and enclose it in a square ABCD as shown in figure 5.6 (a).
2. Mark midpoints of the sides 1, 2, 3 and 4, where the square touches the circle tangentially.
3. Draw the diagonals of the square which cut in the circle at points 5, 6, 7 and 8 as shown in figure 5.6 (a).
4. Draw a rhombus ABCD to represent isometric view of a square ABCD.
5. Mark points 1, 2, 3 and 4 on it as the midpoint of the sides.
6. Mark points 5, 6, 7 and 8 on it, such that they are at a distance equal to Ax from the side of the square.
7. Join points to obtain isometric view as shown in figure 5.6 (b).
8. Draw other direction of isometric view [figure 5.6 (c)].

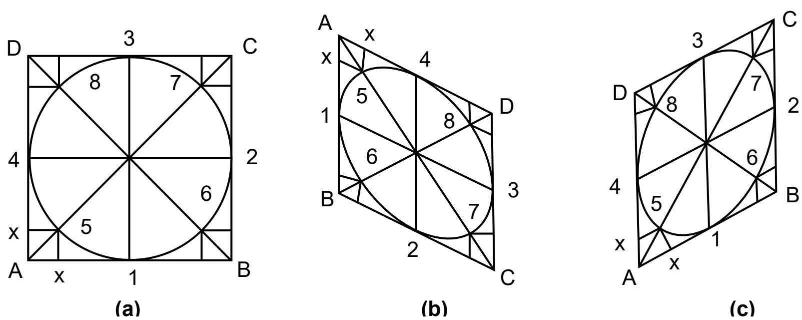


Figure 5.6: Isometric View of Circle

SOLVED EXAMPLES

Example 5.1: Draw the isometric view of a circle lamina with a 60 mm diameter on all three principle planes using for center method.

Solution: Following steps are used to draw isometric view of a given circle as illustrated in figure 5.7:

1. Draw a rhombus ABCD of 60 mm side to represent isometric view of a square.
2. Mark 1,2,3 and 4 as a midpoints of the sides AB, BC, CD and DA respectively and join (the ends of the minor diagonals) B to meet points 3 & 4 and D to meet points 1 & 2. Let B4 and D1 intersect at point E and B3 and D2 intersect at a point F, then B, E, D and F are the four centers for drawing the ellipse.
3. With center B and radius B3 draw arc 3-4 and with center D and radius D1 draw arc 1-2. With center E and radius E1 draw arc 1-4 and with centre F and radius F2 draw arc 2-3.
4. These arcs join in the form of an ellipse which represents the required isometric view.

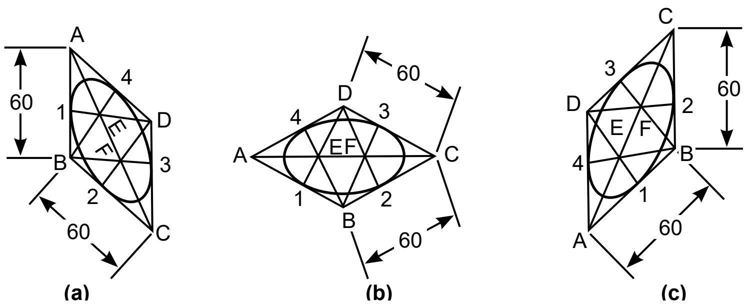


Figure 5.7: Isometric View of Circle Using Centre Method

Example 5.2: Draw isometric view of pentagon.

Solution: Following steps are used to draw isometric view of a given pentagon as illustrated in figure 5.8:

1. Enclose the given pentagon in a rectangle and obtain the parallelogram as shown in figure 5.8 (a).
2. Locate points 1, 2, 3, 4 and 5 on the rectangle and mark them on the parallelogram.
3. Mark the distances *A*–1, *B*–2, *C*–3, *C*–4 and *D*–5 in isometric drawing are same as the corresponding distances on the pentagon enclosed in the rectangle.
4. Figure 5.8 (b) and 5.8 (c) represent the required isometric view of pentagon.

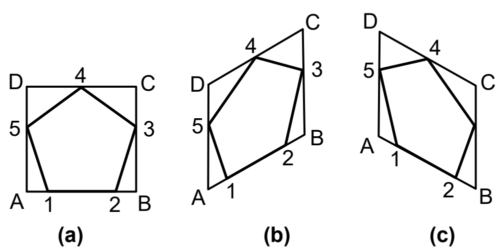


Figure 5.8: Isometric View of Pentagon

Example 5.3: Draw isometric view of irregular shape.

Solution: Following steps are used to draw isometric view of any irregular shape as illustrated in figure 5.9.

1. First enclosed the irregular shape in a rectangle as shown in figure 5.9 (a).
2. The parallelogram is obtained in isometric for the rectangle as shown in figure 5.9 (b)
3. The iso-lines A-1, B-2, B-3, C-3, C-4, D-4, D-5, A-5 has the same length as in original shape.
4. Figure 5.9 (b) and 5.9 (c) represent the required isometric view of given irregular shape.

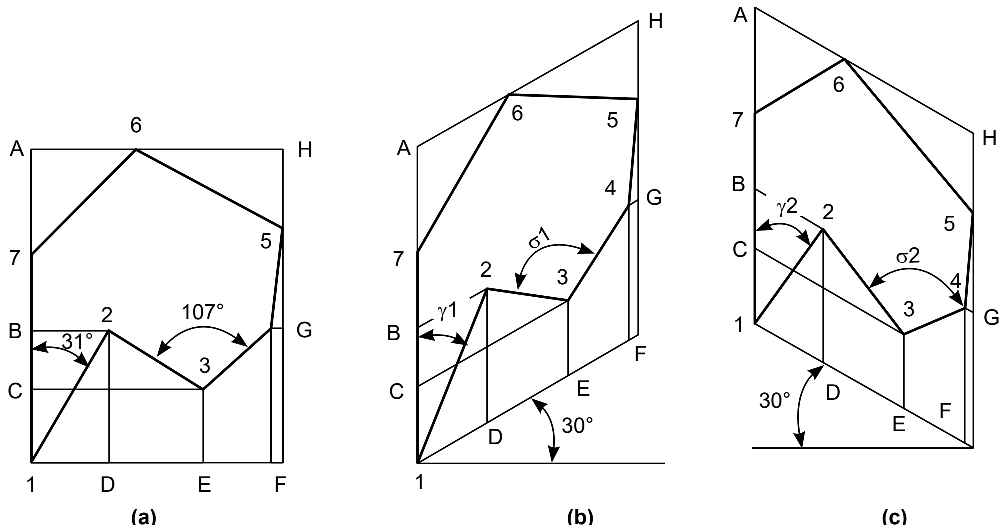


Figure 5.9: Isometric View of a Irregular Shape

5.7 ISOMETRIC VIEWS OF SOLIDS

Following are the two methods of drawing isometric views of solids:

1. Box Method.
2. Off-set Method.

5.7.1 Box Method

This method of drawing isometric view is easy and intelligible. However, it takes much time for the drawing. In this method the isometric view of solids like cubes, squares and rectangular and prisms are drawn directly when their edges are parallel to the three isometric axes. The isometric view of all other types of prisms and cylinders are drawn by enclosing them in a rectangular box. This method is called box method.

1. In this method, the maximum length, breadth, and height of an object are noted.
2. A box is constructed in accordance with these dimensions.
3. These dimensions of the box are represented according to isometric projection.
4. These make angles of 30°, 30° and 90° with the horizontal line.
5. After this other parts of the object are shown.

6. For this purpose, isometric lines are drawn parallel to the isometric axis.
7. After this, non-isometric lines, circles, and other curves are drawn.

5.7.2 Off-set Method

1. In this method, for preparing isometric view, by selecting the side, the length, and breadth of the object are drawn parallel to the isometric axis.
2. After this, the other isometric and non-isometric lines are drawn according to the dimensions.
3. Finally, circles and curves are drawn and the drawing is completed by erasing extra lines.

SOLVED EXAMPLES

Example 5.4: Draw the isometric projection of a pentagonal prism of edge of base 30 mm and axis 70 mm long when the axis is horizontal**.**

Solution: Following steps are used to draw isometric view of a given pentagonal prism as illustrated in figure 5.10:

1. Draw the pentagonal prism in the simple position as shown in below figure 5.10 (a).
2. To the right side of elevation draw iso axis as shown below figure 5.10 (b).
3. Enclose the pentagon in the plan in a square.
4. Get the corresponding box of length equal to iso-scale of 70 mm, width equal to iso-scale of "5-3" in the plan and height of the box equal to iso-scale of height of rectangle in the plan.
5. Complete the pentagon in the front and back of the rectangular box in the isometric projection.
6. Join all the visible edges with continuous line and invisible edges with dotted lines to complete the isometric projection.

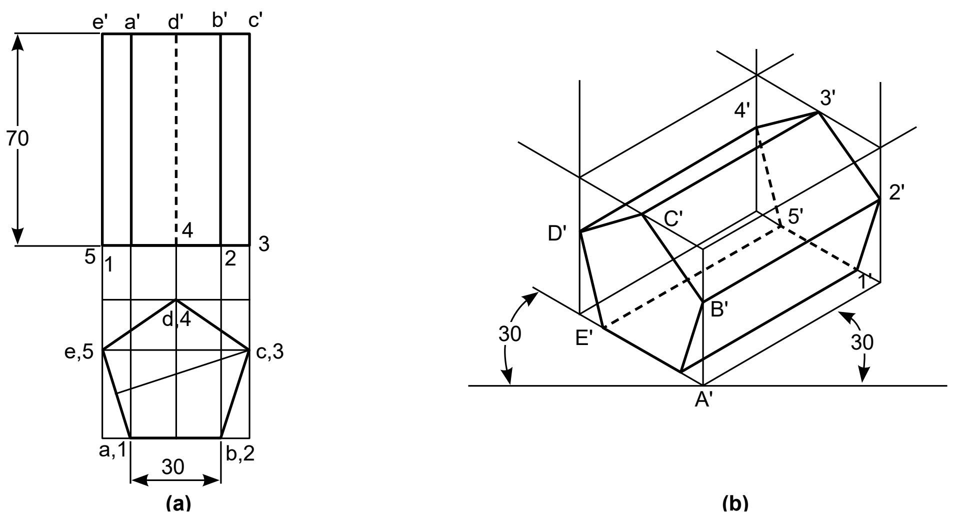
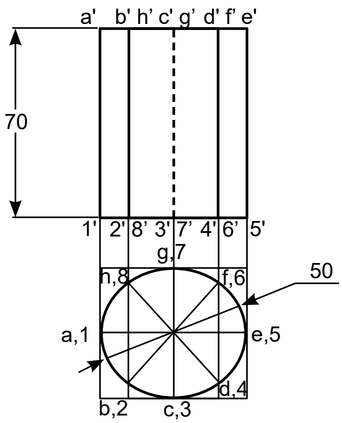


Figure 5.10: Isometric View of a Pentagonal Prism

Example 5.5: Draw the isometric projection of a cylinder of diameter 50 mm and axis 70 mm long when the axis is vertical.

Solution: Following steps are used to draw isometric view of a given cylinder as illustrated in figure 5.11:

1. Draw the cylinder in the simple position as shown in below figure 5.11 (a).
2. To the right side of elevation draw iso axis as shown in below figure 5.11 (b).
3. Inscribe the circle in the plan in a square.
4. Get the corresponding box of height equal to iso-scale of 70 mm, width and breath equal to iso-scale of side of the square.
5. Complete the ellipse in the top and bottom of the rectangular box in the isometric projection as explained by four center method.
6. Join the top and bottom of the ellipse by straight line as shown below and rear part of the bottom face of the cylinder by dotted line as it will not be visible to complete the isometric projection.



(a) Projection

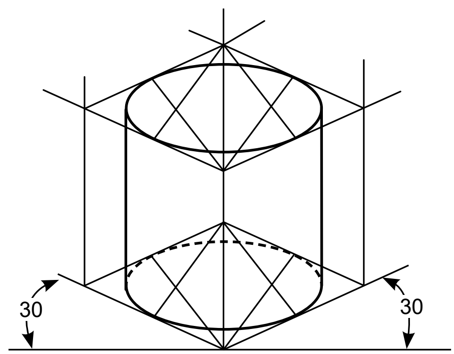

(b) Isometric Projection

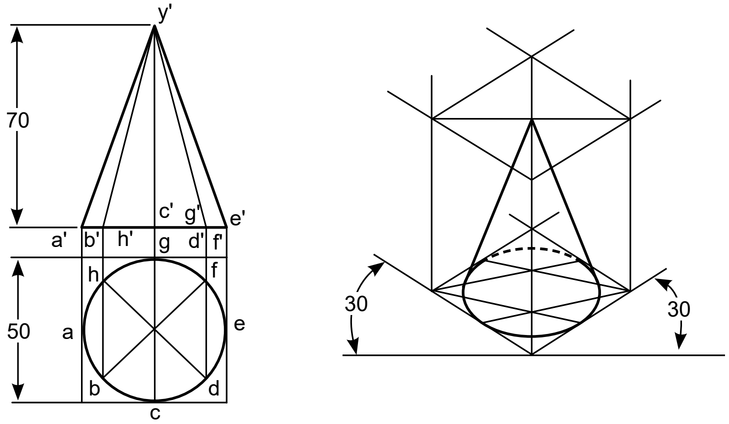
Figure 5.11: Isometric View of a Cylinder

Example 5.6: Draw the isometric view of a cone of base diameter 50 mm and altitude 70 mm when the base is on HP.

Solution: Following steps are used to draw isometric view of a given cone as illustrated in figure 5.12:

1. Draw the cone in the simple position as shown in below figure 5.12 (a).
2. To the right side of elevation draw iso axis as shown in below figure 5.12 (b).
3. Inscribe the circle in the plan in a square.
4. Get the corresponding box of height equal to 70 mm, width and breath equal to side of the square.
5. Complete the ellipse in the bottom of the rectangular box in the isometric view as explained by four center method.
6. Get the centre of the top face of the rectangular box to get the vertex point.

7. Join the bottom of the ellipse with the vertex by straight line as shown below and rear part of the bottom face of the cone by dotted line as it will not be visible to complete the isometric view as shown in figure 5.12 (b).



(a) **(b)**

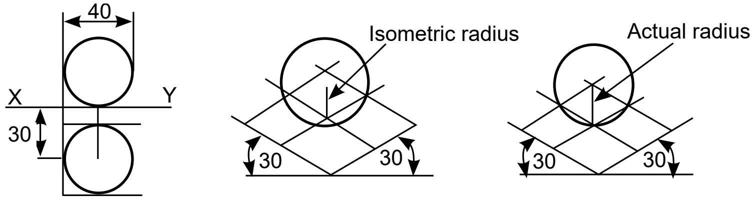
Figure 5.12: Isometric View of a Cone

Example 5.7: Draw the isometric projection and isometric view of a sphere of diameter 40 mm and mark the lowest point on its surface.

Solution: The isometric projection of the sphere is a circle whose diameter is (3/2) times that of the actual diameter of the sphere. The isometric projection view of the sphere is simply a circle whose diameter is equal to the true diameter of the sphere.

Following steps are used to draw isometric view of a given sphere as illustrated in figure 5.13:

1. Draw the sphere in the simple position as shown in below figure 5.13 (a).
2. To the right side of elevation draw iso axis as shown below figure 5.13 (b).
3. Inscribe the circle in the plan in a square.
4. Gets the rhombus corresponding to iso-scale of square in the plan.
5. Fix the centre of the rhombus and fix the centre of the circle equal to iso scale of radius of the sphere, with that centre and radius equal to isometric radius draw a circle.
6. Repeat the above procedure only with a difference that the radius of the circle is equal to actual radius of sphere to get isometric view of sphere.



(a) Projection **(b)** Isometric Projection **(c)** Isometric View

Figure 5.13: Isometric View of a Sphere

Example 5.8: Draw the isometric view of the pentagonal pyramid, the projections of which are given in figure 5.14 (a).

Solution: Following steps are used to draw isometric view of a given pentagonal pyramid as illustrated in figure 5.14:

1. Enclose the base (in the top view) in an oblong.
2. Draw an offset *oq* (i.e. *pq)* on the line *ab.*
3. Draw the isometric view of the oblong and locate the corners of the base in it.
4. Mark a point Q on the line *AB* such that *AQ* = *aq.* From Q, draw a line *QP* equal to *qo* and parallel to 2C. At *P,* draw a vertical *OP* equal to *o'p'.*
5. Join O with the corners of the base, thus completing the isometric view of the pyramid as shown in figure 5.14 (b).
6. Isometric view of the same pyramid with its axis in horizontal position as shown in figure 5.14 (c).

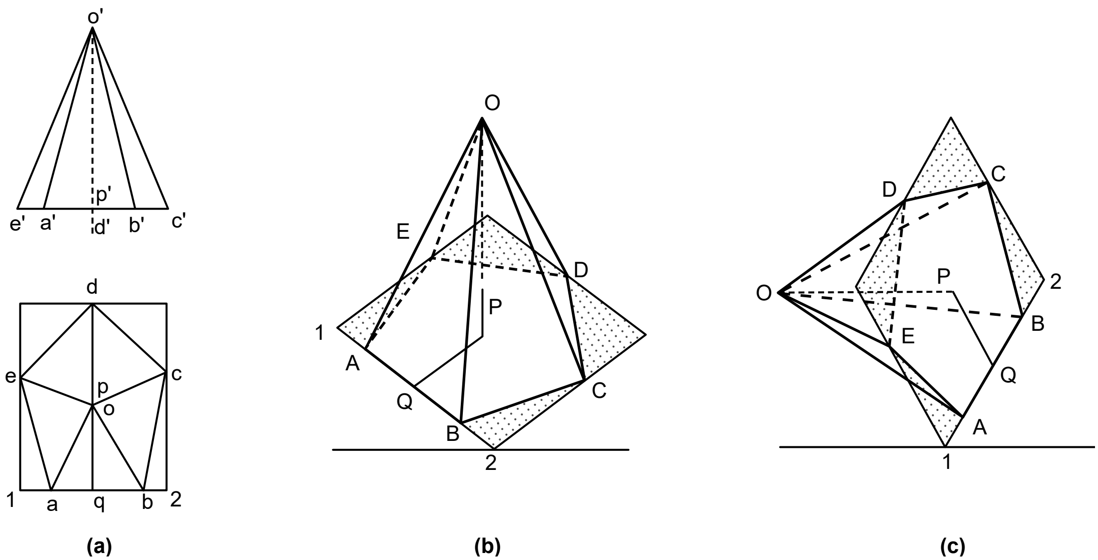


Figure 5.14: Isometric View of a Pentagonal Pyramid

Example 5.9: Draw the isometric projection of a hexagonal prism of side of base 40 mm and height 60 mm with a right circular cone of base 40 mm as diameter and altitude 50 mm, resting on its top such that the axes of both solids are collinear.

Solution: Following steps are used to draw isometric projection of composite solid hexagonal prism and right circular cone as illustrated in figure 5.15:

1. Draw the TV and FV of the prism using isometric scale.
2. A rectangular box of exact size is used to enclose the prism.
3. Construct the box in isometric.
4. Measure the corners from TV and mark them on the sides of the box.
5. Repeat the procedure for cone but construct it above the prism.

6. Darken the top, left and right edges of the prism and the visible portion of the objects to complete the isometric projection.

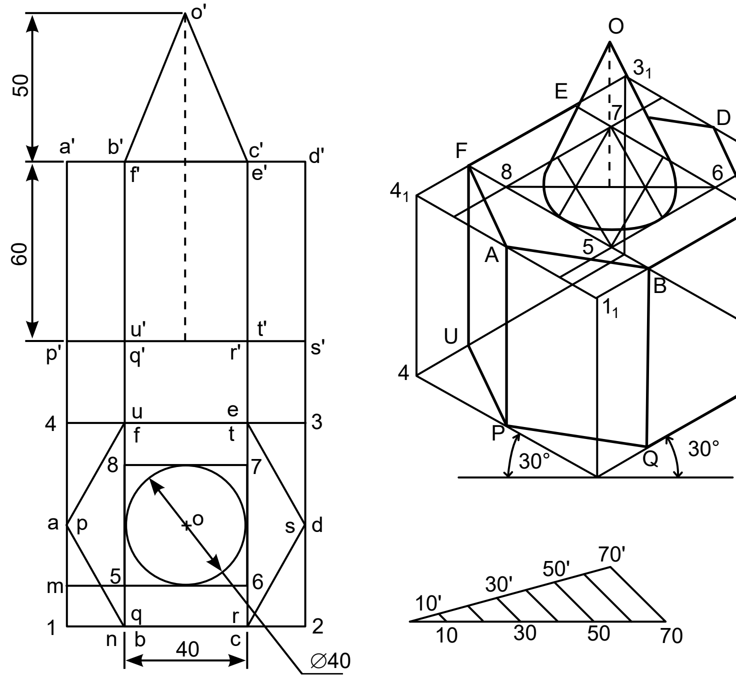


Figure 5.15: Isometric View of Compound Solid

5.8 CONVERSION OF ORTHOGRAPHIC VIEWS TO ISOMETRIC VIEWS

Following steps are used to make an Isometric drawing from orthographic views of given object:

1. First study the given orthographic views of the object and note the principal dimensions and other features of this object as shown in figure 5.16 (a).

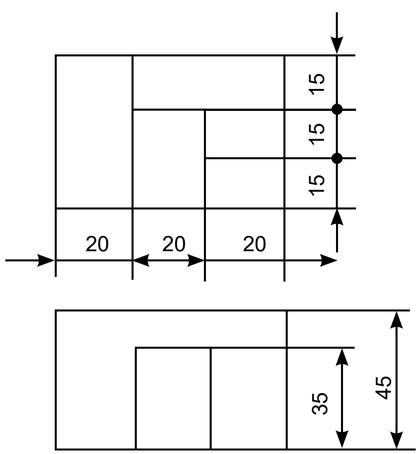


Figure 5.16 (a): Orthographic Views

2. Draw the isometric axes and mark the principal dimensions to-their true values along the isometric axes, as shown in figure 5.16 (b).

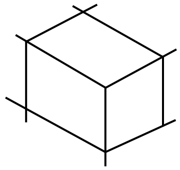


Figure 5.16 (b): Isometric Axes

3. Complete the object by drawing lines parallel to the isometric axes and passing through the above markings as shown in figure 5.16 (c).

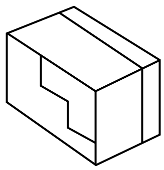


Figure 5.16 (c): Drawing Line Parallel to Isometric Axes

4. Locate the principal corners of all the features of the object on the three faces of the object as shown in figure 5 .16 (d).

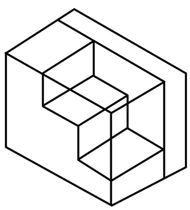


Figure 5.16 (d): Locate the Principal Corners

5. Draw lines parallel to the axes and passing through the above points and obtain the isometric drawing of the object by darkening the visible edges as shown in figure 5.16 (e).

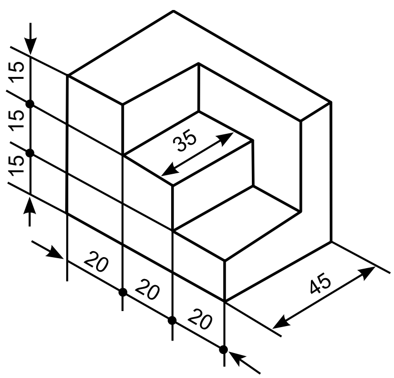


Figure 5.16 (e): Isometric Drawing

5.9 CONVERSION OF ISOMETRIC VIEW TO ORTHOGRAPHIC PROJECTIONS

Following steps are used to creation of an orthographic projection.

1. Select a front view that shows the most about the object.
2. Decide how many views are needed to completely describe the object. If unable to determine which views will be needed then draw the standard views (front, top and right side).
3. Draw the visible features of the front view.
4. Draw projectors off of the front view horizontally and vertically in order to create the boundaries for the top and right side views.
5. Draw the top view and use the vertical projectors to fill in the visible and hidden features.
6. Project from the top view back to the front view and use the vertical projectors to fill in any missing visible or hidden features in the front view.
7. Draw a 45° projector off of the upper right corner of the box that encloses the front view.
8. From the top view, draw projectors over to the 45° line and down in order to create the boundaries of the right side view.
9. Draw the right side view.
10. Project back to the top and front view from the right side view as needed.
11. Draw center lines where necessary.
12. Example of conversion of isometric view to orthographic projection as shown in figure 5.17.

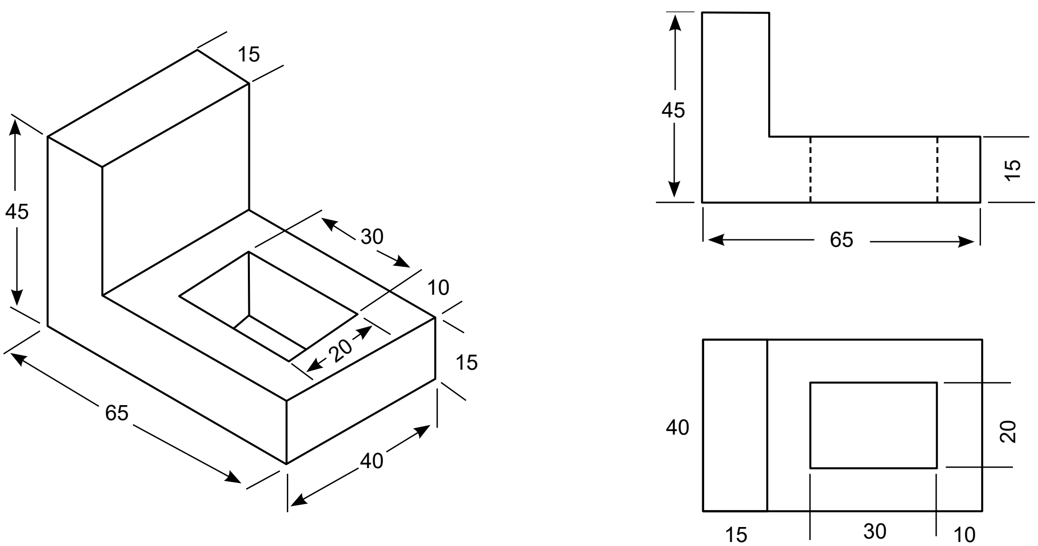


Figure 5.17: Conversion of Isometric View to Orthographic Projections

UNIT SUMMARY

- Isometric scale is used to converts true lengths into isometric lengths.
- Isometric projection is a type of pictorial projection in which the three dimensions of a solid object are not only shown in one view but their actual sizes can be measured directly from it.
- An isometric view is a type of 3D view that is set out using 30-degree angles.
- Methods used to drawing isometric views of solids are: Box Method and Off-set Method.
- Off-set method of making an isometric drawing is preferred when the object contains irregular curved surfaces.
- When an object contains a number of non-isometric lines, the isometric drawing may be conveniently constructed by using the box method.

EXERCISES

5.1 Multiple Choice Questions

1. Front view of the square is given and has to draw its isometric view which angle the base has to make with horizontal?
   (a) 90 degree (b) 15 degree
   (c) 30 degree (d) 60 degree
2. Front view of triangle is given and isometric view is to be drawn which of the following is correct procedure in drawing isometric view.
   (a) Turning the triangle such that base is making 30 degrees with horizontal
   (b) By increasing or decreasing angles at required proportions
   (c) Drawing parallel to isometric axes
   (d) Drawing rectangle with base and height of triangle and the drawing rectangle parallel to isometric axes and pointing triangle in it
3. Isometric view of equilateral triangle will be _____________
   (a) Equilateral triangle
   (b) Scalene triangle
   (c) Isosceles triangle
   (d) Right angled triangle
4. Isometric view of rhombus will become __________
   (a) Parallelogram (b) Rhombus
   (c) Rectangle (d) Square
5. Isometric view of rectangle will become __________
   (a) parallelogram
   (b) Rhombus
   (c) Rectangle
   (d) Square

6. If Axonometric Plane is equally inclined to all the three principal planes then that plane is known as
   (a) Dimetric plane
   (b) Isometric plane
   (c) Trimetric plane
   (d) Orthographic plane

7. Isometric projection of a sphere is a(n) ___ .
   (a) Ellipse (b) Semi-circle
   (c) Circle (d) Arc

8. Box method is used to draw isometric view of
   (a) Pyramid (b) Prism
   (c) Cylinder (d) All of the above

9. Isometric view of a circle is an
   (a) Circle (b) Ellipse
   (c) Parabola (d) Line

10. Isometric view of square is a
   (a) Square (b) Rectangle
   (c) Rhombus (d) Circle

11. While drawing isometric view/drawing, scale is reduced by
   (a) 61.5% (b) 71.5%
   (c) 81.5% (d) No size reduction is required

12 Dimension of isometric projection is reduced by _______ approximately to the actual dimension.
   (a) 51% (b) 61.5%
   (c) 71.5% (d) 81.5%

13. In preparing isometric scale, true or actual scale is drawn at ___ to the horizontal.
   (a) 15° (b) 30°
   (c) 45° (d) 60°

14. In isometric projection, direction of sight is ____ to the plane of projection. i.e. Axonometric plane.
   (a) Parallel (b) Perpendicular
   (c) Inclined (d) None of The Above

15. Isometric projection is reduced volume wise by ____ to the actual size.
   (a) 34.13% (b) 44.13%
   (c) 54.13% (d) 64.13%

Answer Keys (Exercise: 5.1)

1. (c), 2. (d), 3. (b), 4. (a), 5. (a), 6. (b), 7. (c), 8. (d),
9. (b), 10. (c), 11. (d), 12. (d), 13. (c), 14. (b), 15. (c)

5.2 Short and Long Answer Type Questions

Category I

1. What are the three types of axonometric projection?
2. Define Isometric View.
3. State necessary steps followed to make an isometric view.
4. What is the limitation of isometric projection?
5. Write the difference between isometric projection and isometric drawing.
6. Explain the principle of isometric projection.
7. Why isometric projection is important?
8. What is the use of isometric scale?
9. How to convert orthographic views in isometric view?
10. Differentiate isometric and perspective projection.

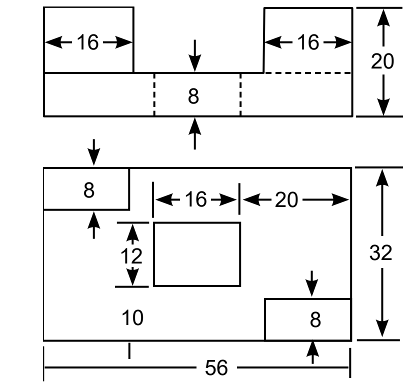
Category II

11. Draw isometric projections for the all prismatic objects.
12. Explain the methods of drawing isometric view of solid objects
13. Construct the isometric scale and derive the equation for calculation of the isometric length.
14. Explain Box method and Off set method.
15. Write the steps used to convert isometric view to orthographic views.

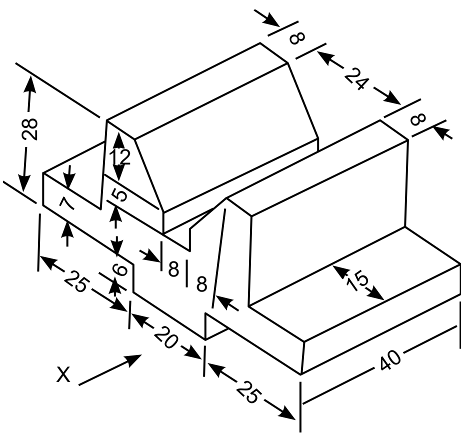
5.3 Practical Exercises

1. Draw the isometric view of a circle lamina with a 60 mm diameter on all three principal planes using four center methods.
2. Draw the isometric view of a hexagon with 40 mm side such that its surface is parallel to the HP and a side parallel to the VP.
3. A cylindrical block of base, 60 mm diameter and height 90 mm, standing on the HP, with its axis perpendicular to the HP. Draw its isometric view.
4. Draw an isometric view of a square prism having a base with 30 mm side and a 60 mm long axis, resting on the HP when (a) On its base with axis perpendicular to the HP (b) On its rectangular faces with axis perpendicular to the VP and (c) on its rectangular face with axis parallel to VP.
5. Draw isometric view of a hexagonal prism having a base with 40 mm side and a 80 mm long axis resting on its base on the HP. With an edge of the base parallel to the VP (a) using Box Methods (b) using Off-set Method.
6. Draw an isometric view of a cylinder, with a 60 mm base diameter and a 75 mm long axis when (a) The base is on the HP (b) when one of the generators is on the HP?
7. A cylinder of base diameter 35 mm axis 70 mm is resting centrally on a slab of 60 mm square and thickness 20 mm. Draw the isometric projection of the combination of the solids.

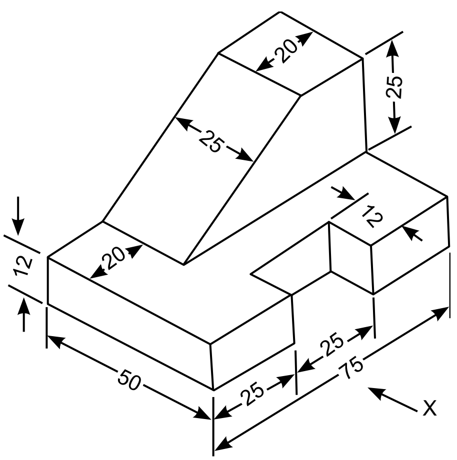
8. Draw an isometric view of given figure below. (All dimensions are in mm).



9. Draw the front view, top view and side view of the object whose isometric view is shown in the figure below (All dimensions are in mm).



10. Draw the Orthographic views of the Isometric view shown in the following figure:



INTERESTING FACTS

- Isometric projection is a method of graphic representation and the technique is intended to combine the illusion of depth.
- The biggest visual difference between isometric drawings and isometric projections is the size of the two images. The isometric drawing is drawn using 100% true length measurements on the height, width, and depth axes.
- However, in isometric projections the height, width and depth are displayed at 82% of their true length. When drawing a pictorial on a computer, the image that appears on the screen is a projection, and therefore it is foreshortened.
- The big advantage of drawing these pictorials on the screen is that the object may be easily rotated about the axes in a countless number of positions once the data for the object's features have been entered.

KNOW MORE

Isometric drawing v/s One-point perspective

Both isometric drawings and one-point perspective drawings use geometry and mathematics to present 3D representations on 2D surfaces. One-point perspective drawings mimic what the human eye perceives, so objects appear smaller when they move away from the viewer. In contrast, isometric drawings use parallel projection, which means objects remain at the same size, no matter how far away they are.

Basically, isometric drawing does not use perspective in its rendering (i.e. lines do not converge as they move away from the viewer). Isometric drawings are more useful for functional drawings that are used to explain how something works, while one-point perspective drawings are typically used to give a more sensory idea of an object or space.

APPLICATIONS

Isometric views are used in component and assembly drawings. Isometric component pictorials presented in two formats: isometric detail drawing or isometric insert view. Isometric detail drawing is a single view with proper annotations. Isometric insert view is an isometric view of a component that is added to necessary orthographic views principally to aid visualization and sometimes for completeness of documentation.

In assembly drawings, isometric views provide general graphic description of each component in outline, exploded and section views.

Section pictorial views show all internal components in mating position at a plane defined by the cutting plane line. Broken section isometric views are used in assembly and detail drawings.

ACTIVITY

1. Sketch isometric view of furniture available in graphics lab and convert into orthographic projection.
2. Sketch the isometric view of desktop computer and its peripherals.

CASE STUDY

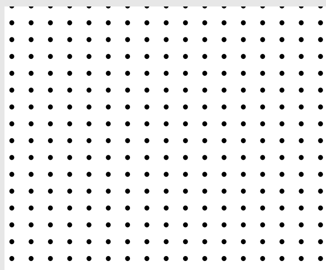
Isometric Sketch

Isometric Sketching or isometric drawing is a pictorial representation of an object in which all the three-dimensions are drawn at the full scale. It looks like an isometric projection. In this case, all the lines parallel to its major axes are measurable.

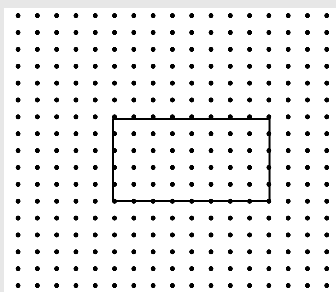
Isometric Sketching Example

An example to draw the isometric sketching is given below. Following steps are used to draw an isometric sketch of an $l \times b \times d$ cuboids..

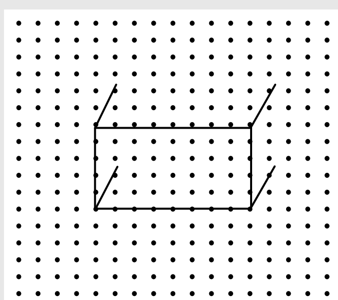
1. To draw an isometric sketch of a cuboids with dimension $l \times b \times d$, take an isometric dot paper as shown below:



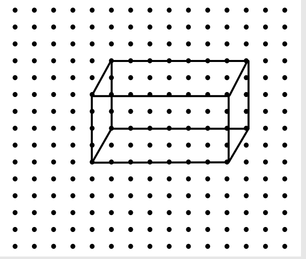
2. To draw the front face, join *l* dots to form the length of the cuboid and *b* adjacent dots to form its breadth as shown:



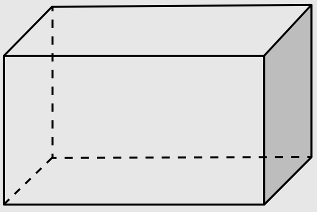
3. From the corners of the rectangle drawn above, draw 4 parallel line segments as shown below:



4. Join the corners of the image together as shown below:



5. According to the convention redraw the hidden edges as dotted lines as shown:



This isometric sketch representing a cuboid of dimension $l \times b \times d$. Thus, using isometric dot sheet, three-dimensional shapes of exact measurements or dimensions can be draw without any ambiguity.

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

1. Engineering Drawing with an Introduction to AutoCAD Dhananjay A Jolhe, McGraw Hill Education
2. https://www.creativebloq.com/features/isometric-drawing
3. https://youtu.be/kYqn4QhUqe4
4. https://youtu.be/RqbzYCttHfM
5. https://nptel.ac.in/courses/105/104/105104148/

6 Overview of Computer Graphics

UNIT SPECIFIC

This unit covers use of computers for engineering drawing and drafting explaining different aspects like Interface, menu system, toolbars, status bar, different methods used for object selection and AutoCAD commands ZOOM and ERASE.

RATIONALE

The study of computer graphics enables engineers to understand the working and capability of computer aided design software. CAD software helps users creating designs in either 2D or 3D so that they can create and manipulate.

PREREQUISITE

Knowledge of computer operations, windows and sketching basics.

UNIT OUTCOMES

Upon successful completion of this module, the student will be able to:

U6-O1: Demonstrate use of hardware for CAD application.
U6-O2: Setup workspace for CAD
U6-O3: Generate simple 2 D drawings using AutoCAD
U6-O4: Edit 2 D drawings in AutoCAD

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-6 Outcomes	**Expected Mapping With Course Outcomes** *(1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)*				
	CO-1	**CO-2**	**CO-3**	**CO-4**	**CO-5**
U6-O1			3	3	
U6-O2			3	3	
U6-O3			3	3	
U6-O4			3	3	

6.1 COMPUTER AIDED DESIGN (CAD)

Computer-aided design (CAD) can be defined as the use of computer systems to assist in the creation, modification, analysis or optimization of a design. It is a modern digital technology which replaces manual drafting with an automated process to create 2D drawings and 3D models of real-world products. CAD systems are based on user oriented system in which the computer is employed to create, transform, and display data in the form of pictures or symbols. CAD is also known as Computer-Aided Design and Drafting (CADD), Computer Aided Engineering Graphics (CAEG).

Benefits of CAD

CAD system is much more than a replacement for manual drawings, it brings about a whole new level of possibilities that engineers can utilize for creating better products with less mistakes and faster pace. The important benefits of CAD are as follows:

1. **Short Lead Times**

   The computer aided design and drafting process is inherently faster than the traditional process. Making the drawings, preparing reports of assembly and part drawings, making specifications, preparing bill of materials etc, are all much easier and faster by using the CAD systems.

2. **Increases Productivity**

   Time saved translates directly into augmented productivity. The same amount of time can result in a higher number of completed projects. CAD software allows designers to lower production costs, work faster and smarter, ultimately leads to quicker project completion.

3. **Better Quality**

   Traditionally, when a design did not function as expected, the team had to go back to the drawing board. The use of CAD allows design teams to control the quality of the final engineered product. It's easy to investigate an error, diagnose the problem, and solve it all using the software before any prototypes are made.

4. **Ease of Understanding**

   The availability of 3D models to accompany drawings can make even the most difficult drawings easy to comprehend. This cannot be done in physical sketches as a minimum of three sketches (plan, elevation, and side view) would be required to get a general idea.

5. **Saving of design data and drawings**

   All the data used for designing can easily be saved and used for the future reference, thus certain components do not have to be designed again and again. Similarly, the drawings can also be saved and any number of copies can be printed whenever required. Some of the component drawings can be standardized and used in any future drawings.

6. **Quick Sharing for Collaboration**

   CAD drawings, being digital files, are easy to share among team members who are working on the same product. No bulky drawings need to be transported. Sharing is instantaneous and a remote employees can stay in the loop about any developments without any issue.

6.2 COMPUTER GRAPHICS SYSTEM

Computer Graphics is concerned with all aspects of producing pictures or images in computer by using specialized graphics hardware and software. Computer Graphics plays a key role in computer aided design (CAD) to create virtual world for digital prototyping and enable designers to interactively synthesize various shapes, visualize and analyze their functional performance. Typically, the term *computer graphics* refers to several different things:

- The representation and manipulation of images by a computer.
- The various technologies used to create and manipulate images.
- Methods for digitally synthesizing and manipulating visual content.

The typical computer graphics system is a combination of hardware and software as shown in figure 6.1. The hardware includes a central processing unit, one or more workstations (including the graphics display terminals), peripheral devices such as printers, plotters and drafting. The software consists of the computer programs needed to implement graphics processing on the system.

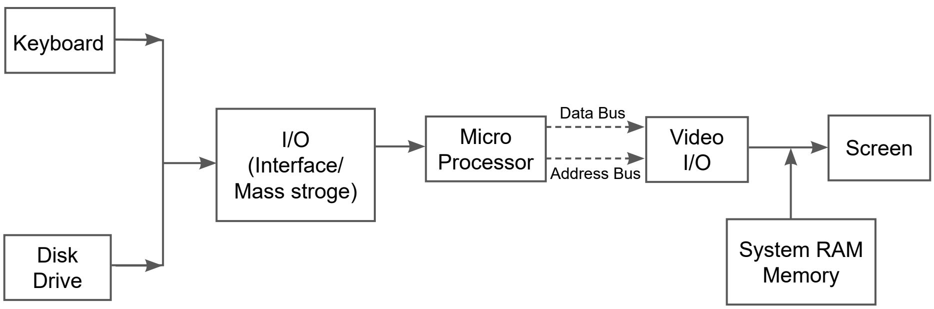


Figure 6.1: Computer Graphic System

CAD Workstation

CAD Workstation is a high-performance computer system that is basically designed for a single user, comprises of the devices that allow the user to create and design objects, using both graphic and non-graphic instructions and data. Figure 6.2 shows a typical CAD workstation, equipped with powerful graphics processing capabilities and specialized input devices for user interaction.

Introduction to Computer Graphics

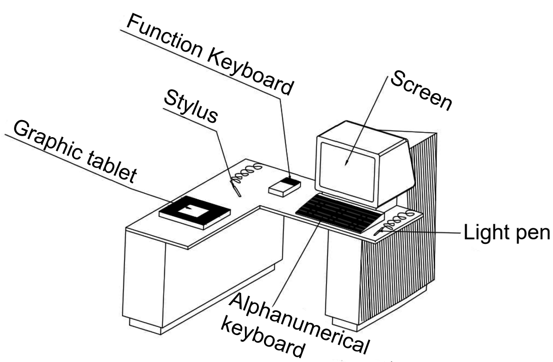


Figure 6.2: CAD Workstation

CAD Software

Computer-Aided Design (CAD) software is an application used to create or optimize a design. Therefore, CAD software allows engineers, architects, designers and others to create precision drawings or technical illustrations in 2D or 3D. CAD software can increase productivity, improve quality, and maximize organization by creating a documentation database for manufacturing. CAD software has a host of applications including the design of manufacturing parts, electronic circuit boards, 3D printers and buildings.

There is a variety of CAD software available, each with special features that sometimes make them suitable for a particular application; these are mainly classified as 2D and 3D software.

2D CAD Software (2D CAD)

2D CAD is the pioneer of CAD software, and was developed in the early 70s. At that time, major automobile, aerospace and other engineering companies developed in-house tools to automate repetitive drafting requirements. 2D CAD relies on basic geometric shapes like lines, rectangles, circles, etc. to produce flat drawings. Autodesk is one of the pioneering company that have played a significant role in developing CAD software.

3D CAD Software (3D CAD)

3D CAD software is more advanced and a step up from the 2D CAD software. As the processing power of computers increased and the graphic display capabilities improved, 3D CAD has become an increasingly popular design tool. The images created by 3D software are realistic and can be viewed and rotated in any direction. 3D CAD tools were introduced in 1980’s by a partnership between IBM-Dassault and quickly became popular because of enhanced visual capability.

6.3 AUTOCAD SOFTWARE

AutoCAD is a software application developed by Autodesk that enables computer-aided design (CAD) and drafting. The software is used to produce 2D and 3D drawings and allows users to conceptualize ideas, produce designs and drawings to the required levels of technical accuracy and even perform rapid design calculations and simulations; across a wide range of industries. AutoCAD is first very effective computer aided design and drafting software has been available on the market since 1982, making it the first CAD system developed for PCs. AutoCAD is accepted as the industry standard and it is preferred by a large community of CAD users in the world. The software has evolved massively over the years and now offers various ‘themed’ versions catering for the specific requirements of different engineering disciplines. There is virtually no limit to the kinds of line drawings using AutoCAD. If a drawing can be created by hand, it can be generated by AutoCAD. Here are a few of the applications of the AutoCAD:

- Drawings for electronic, chemical, civil, mechanical, automotive and aerospace engineering.
- Architectural drawing of all kinds.
- Interior design and facility planning.
- Work-flow charts and organizational diagrams.
- Graphs of all kinds.
- Topographic maps and nautical charts.
- Plots and other representations of mathematical and scientific functions.

- Technical illustrations and assembly diagrams.
- Company logos.
- Greeting cards.
- Line drawings for the fine art.

AutoCAD Workspace

Workspaces are sets of menus, toolbars, palettes and ribbon control panels that are grouped and organized in a custom task-oriented drawing environment. AutoCAD offers following predefined workspaces:

1. 2D Drafting & Annotation - Common ribbon tools for 2D drawings.
2. 3D Basics - Basic ribbon tools for creating and viewing 3D models.
3. 3D Modeling -A full set of ribbon tools for 3D modeling, viewing and rendering.

User can select or switch to another workspace, whenever to work on a different task, from the workspace switching button on the status bar at the bottom-right of the application window as shown in figure 6.3.

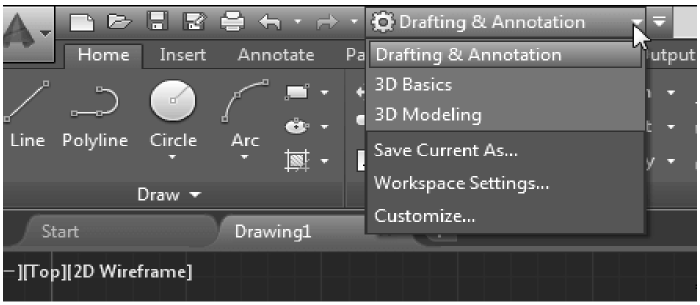


Figure 6.3: AutoCAD Workspace Selection

6.4 AutoCAD USER INTERFACE

The Graphical User Interface (GUI) window of AutoCAD is shown in figure 6.4. The standard **features** of AutoCAD layout screen are listed as follows:

1. Application Menu
2. Pull-Down Menus
3. Shortcut Menus
4. Standard Toolbar
5. User Coordinate System / UCS Icon
6. Crosshairs, Pick-box and Cursor
7. Command window
8. Navigation Bar
9. Quick access toolbar
10. Status bar
11. Text window

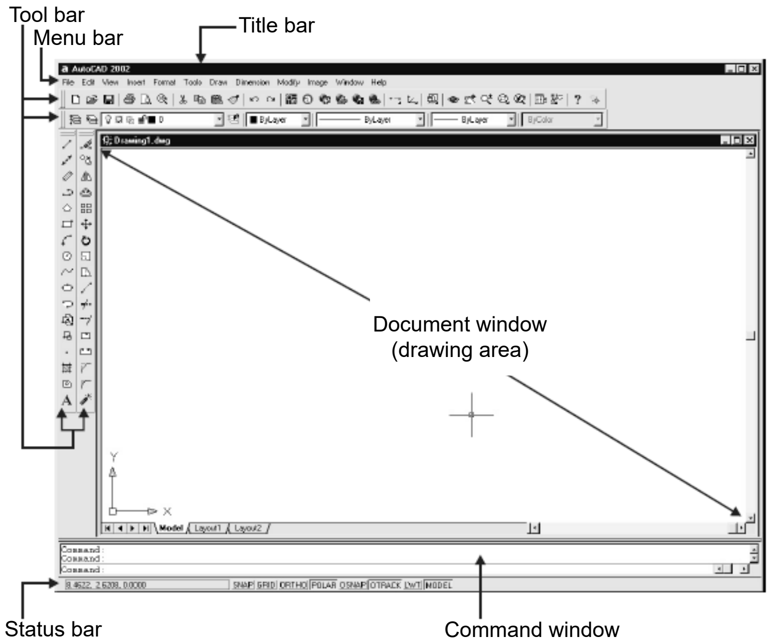


Figure 6.4: GUI of AutoCAD

6.4.1 Application Menu

Menus are available through the application button in the upper left corner of the drawing window. Application menu contains the commands used to create, save, print, and manage drawings. Figure 6.5 shows the layout of application menu.

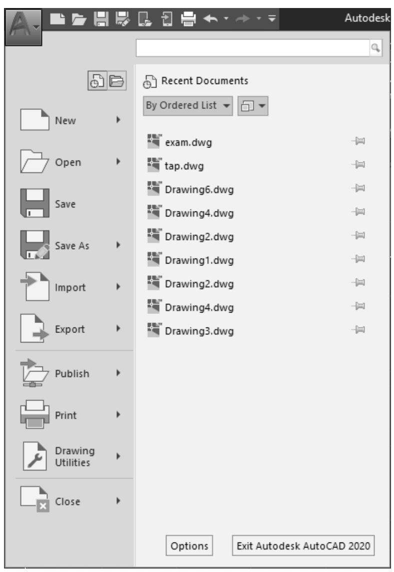


Figure 6.5: Layout of Application Menu

6.4.2 Pull-Down Menus

Most AutoCAD commands, as well as numerous standard windows functions, are available from pull-down menus on the menu bar. These menus are arranged in a hierarchical fashion. Commands for drawing new AutoCAD objects are found in the Draw (pull-down) menu and commands to modify drawing are found in the Modify (pull-down) menu. Figure 6.6 shows a typical layout of pull-down menu.

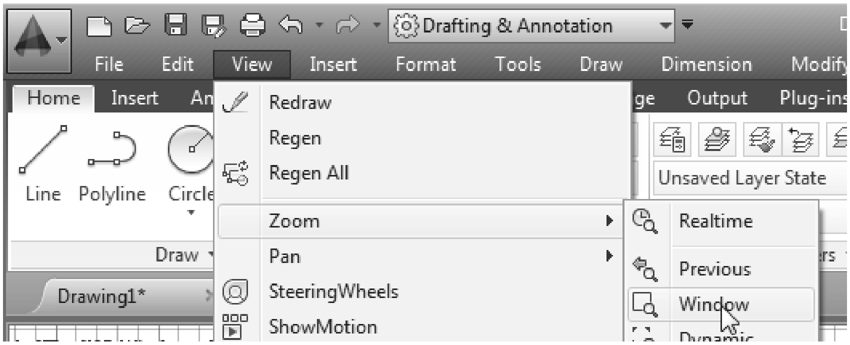


Figure 6.6: Layout of Pull-down Menus

6.4.3 Shortcut Menus

The shortcut menus are special menus that display at the cursor position when user presses the right-mouse button. Shortcut menus are completely context-sensitive. The functions displayed in the menu vary, depending on the location of the cursor when user right-click, the type of object selected, and whether an AutoCAD command is active. Figure 6.7 shows typical layout of short cut menu, when the cursor is on line and user presses right mouse button.

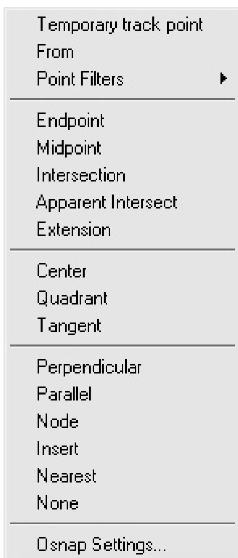


Figure 6.7: Layout of Short Cut Menu

6.4.4 Standard Toolbar

When user starts AutoCAD first time, the Standard, Object Properties, Draw, and Modify toolbars are displayed. AutoCAD's standard menu provides 26 toolbars, each of which contains a group of related commands. User can have any of these toolbars visible at any time and control where they are placed on the desktop. All of these toolbars can be customized by adding and deleting buttons. Figure 6.8 shows the Standard Tool Bars:

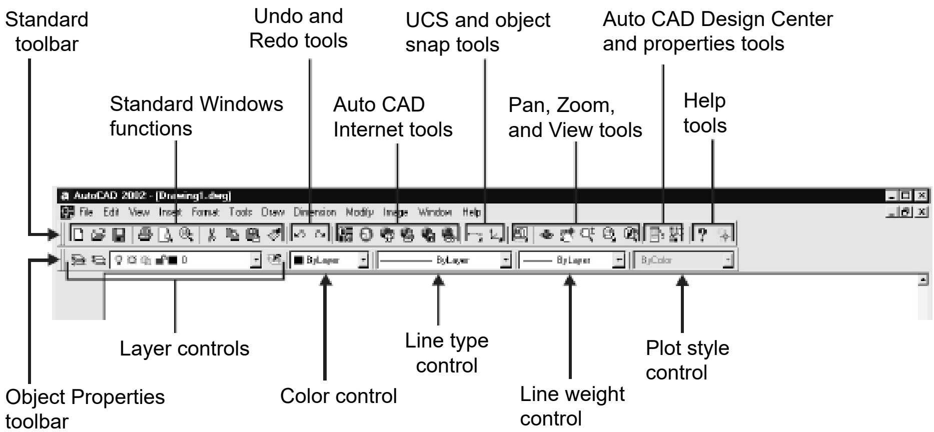


Figure 6.8: Layout of Standard Tool Bars

6.4.5 Coordinate System

When a user creates 2D drawings, data input is ultimately passed to the software in the form of cartesian or polar coordinates. User can either manually enter these coordinates or infer them by picking a point in the drawing window. The coordinate system icon helps to understand how drawing is oriented. The icon consists of two arrows, one pointing to the right and one pointing to the top of the drawing area. Notice that one arrow is labeled X and the other Y. These labels indicate the current orientation of the drawing's X and Y axes. Figure 6.9 shows the coordinate system Icon.

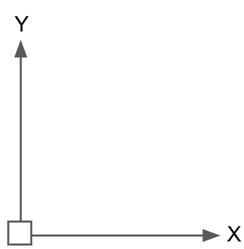


Figure 6.9: Coordinate System Icon

Cartesian coordinate system

The Cartesian coordinate system is used to determine points in space that are a specified distance from a set of perpendicular axes that intersect at the origin of the system. In the World coordinate system, the X axis represents the horizontal direction, the Y axis represents the vertical direction and the origin is located at 0, 0. Positive X moves to the right, positive Y moves up, and the Z axis moves in the positive direction directly towards, the viewer. The following figure 6.10 illustrates a line drawn from the origin of the coordinate system 0, 0 with its endpoint at the coordinate 4, 6.

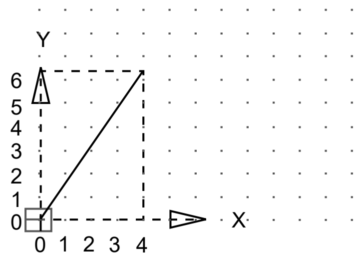


Figure 6.10: Cartesian Coordinate System

To specify cartesian coordinate, type the X and Y coordinates and press enter key.

Polar Coordinates

A polar coordinate is a point in the coordinate system that is determined by a distance and an angle. The following figure 6.11 illustrates a line drawn from the origin of the coordinate system with a length of 7 units and an angle of 45 degrees.

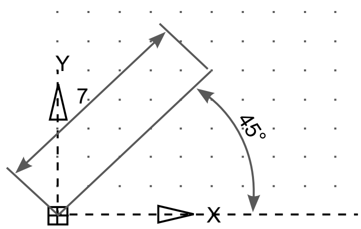


Figure 6.11: Polar Coordinate System

To specify a polar coordinate, type the distance < angle, example (7<45) and press enter key.

Where distance equals the distance traveled from the specified origin point and angle equals the angle from the X axis. The default polar angle is measured counterclockwise from the zero angle position and the default zero is in the east compass direction. The following figure 6.12 shows how angles are defined with a polar coordinate. This angle measurement applies to entering coordinates, working with arcs, and rotating objects.

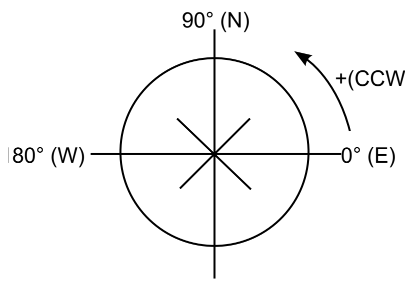


Figure 6.12: Polar Coordinate System Angle Direction

Absolute and Relative Coordinates

Coordinates, can be in the form of an absolute or a relative coordinate:

- An absolute coordinate represents a specific point in the current coordinate system relative to the origin point (0,0).To enter an absolute coordinate, type the values as a Cartesian coordinate (x, y) or Polar coordinate (distance angle).
- A relative coordinate is a point located from a previously selected point. To enter a relative coordinate, select first point, then precede the next coordinate point with the @ symbol, as illustrate in figure 6.13:

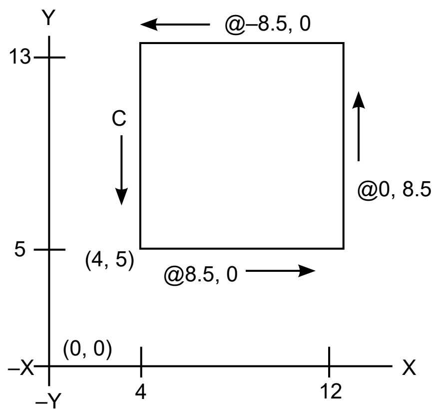


Figure 6.13: Relative Coordinate System

6.4.6 Crosshairs, Pick-box and Cursor

When user moves the mouse, accordingly the drawing cursor moves around the screen. The cursor can be used for selecting points or objects within the drawing area. The appearance of the cursor changes with command. By default, the cursor appears as a small plus sign with a box at its center. The point at which the *crosshairs* meet is the actual cursor position and corresponds to a specific point within the AutoCAD drawing. The box, called a pick-box, is used to select objects within the drawing. Figure 6.14 shows Crosshairs, Pick-box, and Cursor Icon.

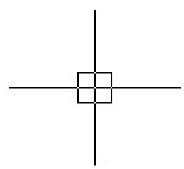


Figure 6.14: Crosshairs, Pickbox, and Cursor Icon

6.4.7 Command window

The Command window is where user type AutoCAD commands and view prompts and messages. When initially displayed, the command window is docked at the bottom of the screen, between the drawing area and the status bar. The command window initially displays the three most recent lines of prompts, but user can change the number of lines displayed. User can undock and move this window by dragging it, and also dock it at the top of the drawing area. Figure 6.15 shows the command window.

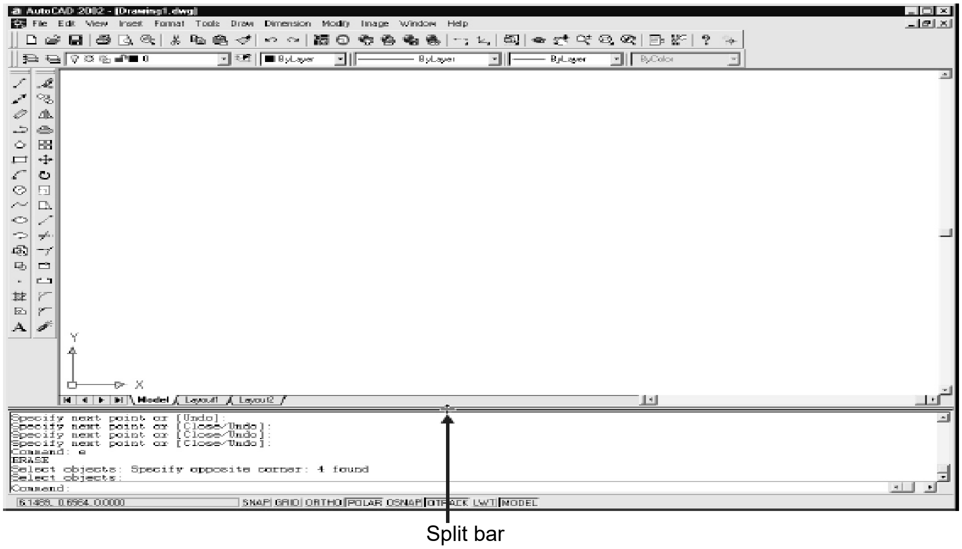


Figure 6.15: Command Window

6.4.8 Navigation Bar

The navigation bar is a user interface element where user can access both unified and product-specific navigation tools. Unified navigation tools are those that can be found across many Autodesk products. Product-specific navigation tools are unique to a product. Figure 6.16 shows the navigation tool bar.

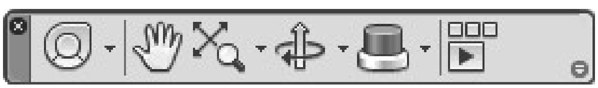


Figure 6.16: Navigation tool bar

6.4.9 Quick Access Toolbar

The Quick Access toolbar 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. Figure 6.17 shows a quick access toolbar.

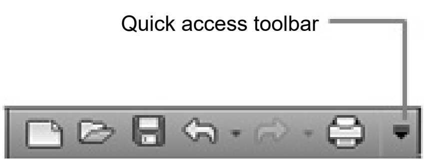


Figure 6.17: Quick Access Toolbar

6.4.10 Status bar

The status bar 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 it can be used to toggle the settings such as grid, snap, polar tracking and object snap. Figure 6.18 shows a typical status bar.

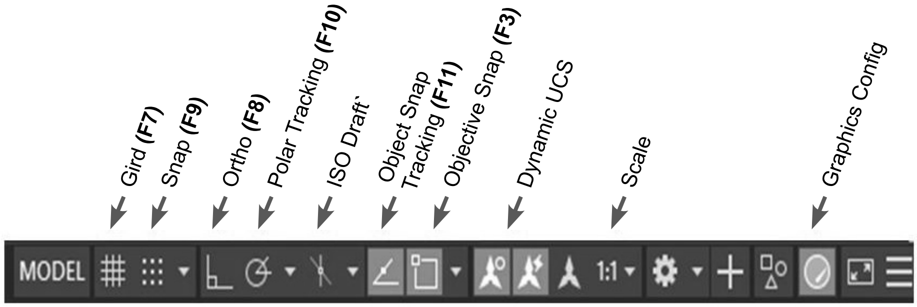


Figure 6.18: Status Bar

6.4.11 Text window

In addition to the AutoCAD drawing *graphics window*, the text window is another important element to the interface. The *Text window* is a second window in which user can type commands and view prompts and messages. Figure 6.19 shows a typical text window.

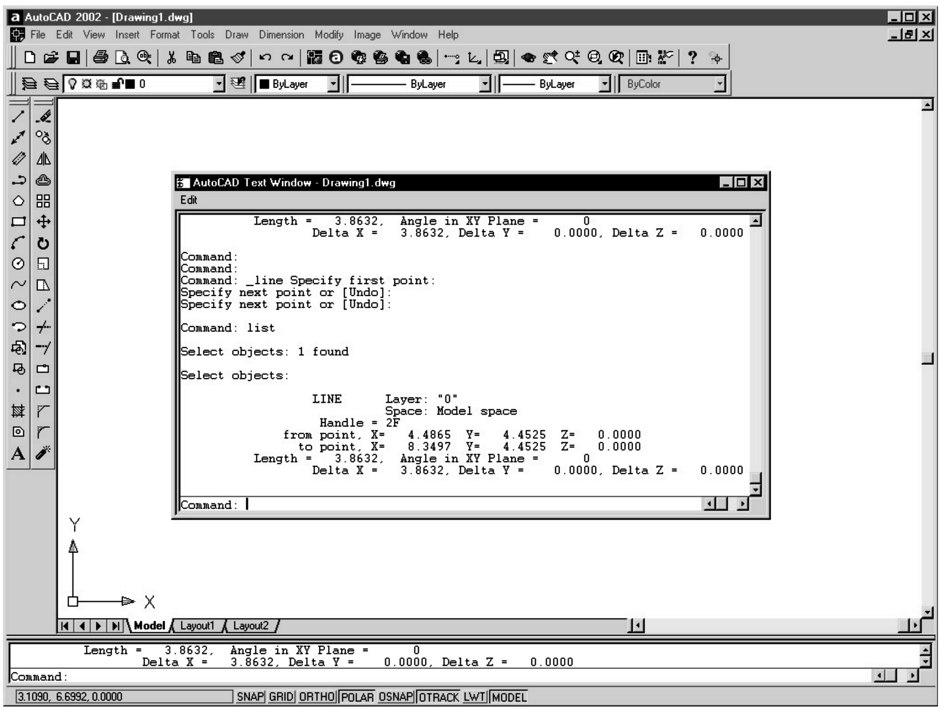


Figure 6.19: A typical Text Window

6.5 FUNCTION KEYS

In AutoCAD, function keys provide some shortcuts, the computer keyboard function keys "F1-F12" can be used to control several AutoCAD settings as given in table 6.1:

Table 6.1: Function Keys

Keys	Features	Description
F1	Help	Displays Help for the active tooltip, command, Palette or dialog box.
F2	Expanded History	Displays an expanded command history in the Command window
F3	Object Snap	Turns object snap ON and OFF
F4	3D Object Snap	Turns additional object snaps for 3D ON and OFF
F5	Iso-plane	Cycles through 2D iso-plane settings (Top, Right and Left)
F6	Dynamic UCS	Turns automatic UCS alignment with planar surfaces ON and OFF
F7	Grid display	Turns the grid display ON and OFF
F8	Ortho	Locks cursor movement to horizontal or vertical
F9	Grid Snap	Restricts cursor movement to specified grid intervals
F10	Polar Tracking	Guides cursor movement to specified angles
F11	Object Snap Tracking	Tracks the cursor horizontally and vertically from object snap locations
F12	Dynamic input	Displays distances and angles near the cursor and accepts dynamic input.

6.6 SELECT COMMAND

In order to manipulate one or more objects with an editing command, the object or group of objects must first be selected. The selection process is accomplished by drawing a selection figure with the pointing device, then clicking to lock in the selection. The following are useful selection figures, also called selection modes.

1. **Point Selection**

   An object can be selected by a pointing device, the selected object's boundary (outline) becomes dotted after clicking the button at a point on object as shown in figure (6.20). This is the most common method used in AutoCAD for selection of objects.

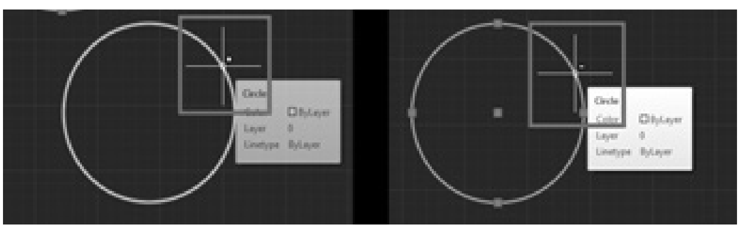


Figure 6.20: Point Selection

2. **Object Selection Using Windows**

   This mode is used to select a group of objects. When the pointing device is clicked for the first time, a corner of a selection window or box appears. By dragging the cursor, one establishes the window perimeter. The window mode selects only those objects that are entirely contained within the window, as shown in Figure 6.21.

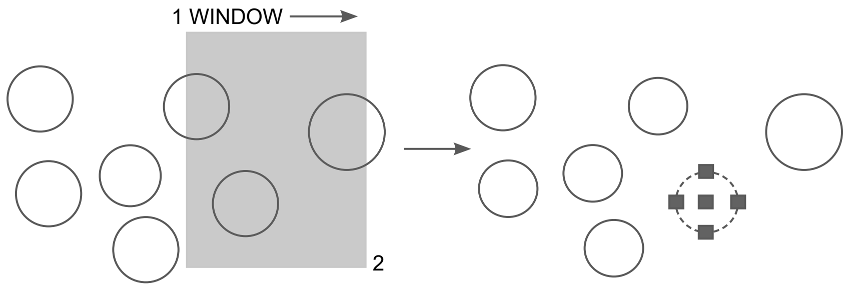


Figure 6.21: Object Selection Using Window

3. **Crossing Selection**
   Similar to the Window mode, this figure is also a box or rectangle, shown in figure 6.22. However, all objects that are partially or entirely within the box will be selected.

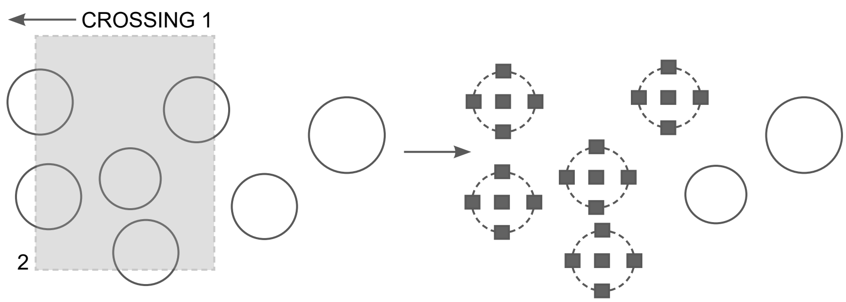


Figure 6.22: Object selection using crossing window

4. **Selecting Objects with a Crossing Line or Fence**
   A line consisting of a sequence of line segments is drawn as illustrated in figure 6.23. All objects touched by the line are selected when the pointing device is clicked as.

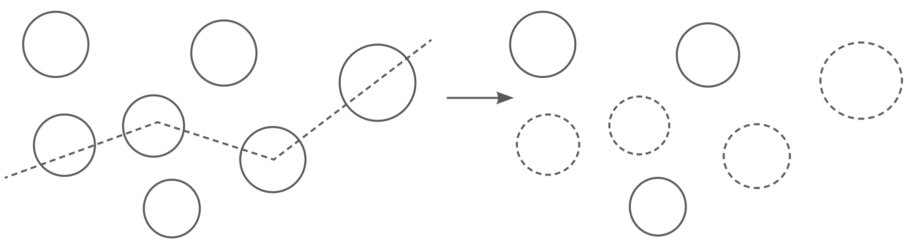


Figure 6.23: Selecting Objects with a Crossing Line

5. **Select all objects**
   This command is used to select all the objects present on the viewport. Type **ALL** on the command line being in the selection mode to activate the selection process.

6. **Select Last or pervious object**

   When prompted to select objects, type [**L**] and last created object will be selected. If user erase last object in the meantime, the one that was created before it will be selected. The most recently created object is selected.

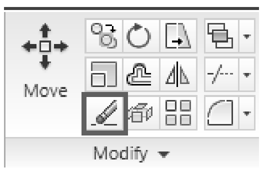
6.7 ERASE COMMAND

One of the advantage of using a CAD system is the ability to select objects from drawing without leaving any mark. One enters the ERASE command or selects the ERASE icon (looks like a pencil eraser), then selects objects, and then depresses the carriage return (Enter key). The selected object(s) will disappear from the screen but can be recalled immediate after the Erase command by entering the OOPS command on the command Line.

Command Access

Erase

- Command Line: ERASE, E
- Menu Bar: Modify > Erase
- Ribbon: Home tab > Modify panel > Erase



Pick Erase in the Modify toolbar. (The icon is a picture of an eraser at the end of a pencil.) The message "Select objects" is displayed in the command prompt area and AutoCAD awaits user to select the objects to erase.

6.8 ZOOM COMMAND

Zoom command is used to Increases or decreases the magnification of the view in the current viewport. Using ZOOM does not change the absolute size of objects in the drawing. It changes only the magnification of the view. The Zoom command has 11 different options; the most important ones are described in this list:

- **Extents and All:** The Zoom Extents button zooms out just far enough to show all objects in the current drawing. The Zoom All button does almost the same thing: It zooms to show the rectangular area defined by the drawing limits set with the LIMITS command, or it zooms to show the extents — whichever is larger.
- **Window:** This option, which is useful for zooming in quickly and precisely, zooms to a section of the drawing that specify by clicking two points. The two points define the diagonal of a window around the area user want to see.
- **Realtime:** Enables user to zoom in and out by starting a realtime zoom and then dragging the magnifying-glass cursor up (to zoom in) or down (to zoom out).

- **Previous:** Use this option to undo the last zoom, returning to where started. User can repeat this option to step through previous views, even if user create or edit objects after changing the view.
- **Object:** This option zooms in close enough to show selected objects as large as they can be displayed onscreen. Using Zoom Object is similar to examining selected objects under the AutoCAD microscope.

UNIT SUMMARY

- Computer-aided design (CAD) can be defined as the use of computer systems to assist in the creation, modification, analysis and optimization of a design.
- Computer Graphics is concerned with all aspects of producing pictures or images in computer by using specialized graphics hardware and software.
- WORKSTATION is a high-performance computer system equipped with powerful graphics processing capabilities and specialized input devices for user interaction.
- CAD software is a computer program that allows engineers, architects, designers, and others to create precision drawings and technical illustrations in 2D or 3D.
- AutoCAD is first very effective computer aided design and drafting software has been available on the market since 1982.
- AutoCAD offers three predefined workspaces for different applications.
- The object selection process is accomplished by drawing a selection figure with the pointing device, then clicking to lock in the selection.
- Zoom command is used to Increases or decreases the magnification of the view in the current viewport.

EXERCISES

6.1 Multiple Choice Questions

1. Which of the organization developed AutoCAD Software?
   (a) Microsoft Corp.
   (b) Apple Inc.
   (c) Autodesk Inc.
   (d) Dassault System
2. What are the Advantages of AutoCAD?
   (a) Improve the quality of designs
   (b) Increase the productivity of the designer
   (c) Creates drawings with accuracy and quickly
   (d) All of the above
3. _________ Key automatically activates the Osnap feature of AutoCAD.
   (a) F12
   (b) F3
   (c) F11
   (d) F9

4. Computer-aided design software is used by __________
   (a) Architects
   (b) Engineers
   (c) Drafters & artists
   (d) All of the above
5. Which of the following workspaces are available in AutoCAD?
   (a) 3D basics
   (b) 3D Modeling
   (c) Drafting and Annotation
   (d) All of the above
6. AutoCAD was first released in the year:
   (a) 1972
   (b) 1982
   (c) 1992
   (d) 2002
7. GUI stands for –
   (a) Graphics uniform interaction
   (b) Graphical user interaction
   (c) Graphical user interface
   (d) None of the above
8. The components of Interactive computer graphics are -
   (a) A monitor
   (b) Display controller
   (c) Frame buffer
   (d) All of the above
9. GKS stands for __________
   (a) General Kernel System
   (b) Graphics Kernel System
   (c) Graphics Kernel Software
   (d) General Kernel Software
10. PHIGS stands for ___________
    (a) Programmer's Hierarchical Interface Graphics Standard
    (b) Programmer's Hierarchical Interactive Graphics Standard
    (c) Programmer's Hierarchical Interest Graphics Standard
    (d) None of these

Answers Keys (Exercise: 6.1)

1. (c), 2. (d), 3. (b), 4. (d), 5. (d), 6. (b), 7. (c), 8. (d), 9. (b), 10. (b)

6.2 Short and Long Answer Type Questions

Category I

1. What do you understand by CAD System?
2. What are the benefits of Computer Aided Design (CAD).
3. What are the applications of CAD?
4. Differentiate 2D and 3D CAD.
5. What are the uses of AutoCAD?
6. What is the important hardware system used in CAD.
7. What do you understand by GUI?
8. Name the main components of AutoCAD user interface.
9. What do you understand by graphics standard?
10. What is GKS?

Category II

1. Name and explain the specialized input devices used in CAD workstation.
2. Explain the features of latest version of AutoCAD software.
3. Write and specify the minimum hardware requirement to run the CAD software on desktop PC.
4. Explain with example different methods used for object selection in AutoCAD.
5. Study and compare the various 2D software available in the market.

KNOW MORE

CAD Standards and Formats:

The purpose of CAD standard is that the CAD software should not be device-independent and should connect to any input device via a device driver and to any graphics display via a device drive. The graphics system is divided into two parts: the kernel system, which is hardware independent and the device driver, which is hardware dependent as shown in figure 6.24.

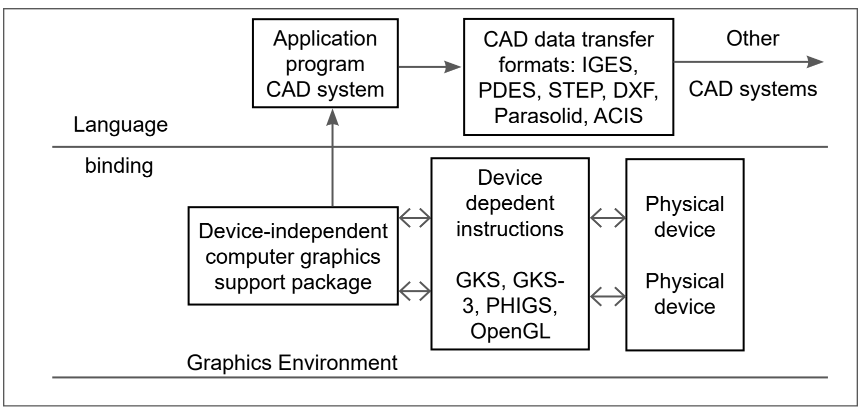


Figure 6.24: Graphics Standards

Important graphics standards and formats

- IGES (Initial Graphics Exchange Specification) enables an exchange of model data basis among CAD system.
- DXF (Drawing / Data Exchange Format) file format was meant to provide an exact representation of the data in the standard CAD file format.
- STEP (Standard for the Exchange of Product model data) can be used to exchange data between CAD, CAM , CAE.
- GKS (Graphics Kernel System) is an international standard for CAD files that define how graphics are handled by software. This standardization allows for CAD files to be created on one computer and moved to a different computer or software program.
- PHIGS (Programmer's Hierarchical Interactive Graphic System) is an application programming interface (API) standard for rendering 3D graphics. A set of functions and data structures to be used by a programmer to manipulate and display 3-D graphical objects.
- VDI (Virtual Device Interface) lies between GKS or PHIGS and the device driver code. VDI is now called CGI (Computer Graphics Interface).
- VDM (Virtual Device Metafile) can be stored or transmitted from graphics device to another. VDM is now called CGM (Computer Graphics Metafile).

DESIGN PROJECT/ACTIVITIES

Prepare a report on the computer hardware, peripherals and CAD software available in your laboratory.

INTERESTING FACTS

- The program that became AutoCAD was created by Mike Riddle, and was originally called "Interact". Its name was changed to "Micro-CAD," but when another company had already created a program with that same name, it was renamed " AutoCAD".
- The company known today as Autodesk was founded by 17 friends and acquaintances in John Walker's home in Mill Valley, CA.
- Since 2010, AutoCAD was released as a mobile application marketed as AutoCAD 360.

APPLICATIONS (AUTOCAD SOFTWARE)

A lot of industries use AutoCAD for various purposes. Some of the main industries where AutoCAD finds its applications are as follows: -

- Architecture
- Civil engineering
- Interior design
- Automobile Industry
- Aerospace Industry
- Mechanical engineering
- Fashion designing
- Jewellery designing
- 3D printing

INQUISITIVENESS AND CURIOSITY TOPICS

Need For Graphic Standards

- Portability of the geometric model among different hardware platforms.
- Exchange drawing database among software packages.
- Exchanging graphic data between different computer systems.
- Requirement of graphic data exchange formats and their details such as IGES, DXF and STEP.

CASE STUDY

Following are the some free / open source CAD softwares:

LibreCAD

LibreCAD is a good free CAD software to start creating CAD drafting. It is open-source software available with Mac Windows and Linux.

Solid Edge 2D Drafting Siemens

2D drafting Solid edge from siemens is a free CAD software program that can help with a familiar user interface, easy to use tools that comply with most popular drafting standard capabilities to simplify user experience.

QCAD

QCAD is a free, open source application for computer aided drafting (CAD) in two dimensions (2D). With QCAD students can create technical drawings such as plans for buildings, interiors, mechanical parts or schematics and diagrams. QCAD works on Windows, Macand Linux.

DraftSight

DraftSight is a free design and drafting software program that students can use for CAD projects and 2D drawings. Designed by Dassault Systemes, the Windows application offers several features like automatic formatting, multiple design templates, and comparison preview.

NanoCAD

NanoCAD, is a free drafting software providing high performance and advanced CAD tools. It has a classic and friendly interface and a native .dwg format support. It offers various features such as parametric 2D design and open API.

FreeCAD

FreeCAD is a multiplatform (Windows, Mac and Linux), highly customizable and extensible software. It reads and writes to many open file formats such as STEP, IGES, STL, SVG, DXF, OBJ, IFC, DAE and many others, making it possible to seamlessly integrate it into your workflow. FreeCAD allows students to sketch geometry constrained 2D shapes and use them as a base to build other objects.

AutoCAD

Developed by Autodesk, AutoCAD is both 2D CAD software and a 3D modeling program. Autodesk can be used for architecture or mechanical projects, to make blueprints or engineering plans. Keep in mind that AutoCAD is not free 2D software, but Autodesk is offering a free version of this 2D CAD program for students.

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

1. CAD/CAM: Theory and Practice: by Ibrahim Zeid TMH Publication
2. Computer Graphics By Donald Hearn and M.Pauline Baker
3. https://knowledge.autodesk.com/support/autocad

7 Customization & CAD

UNIT SPECIFIC

The unit consists of setting up of the drawing space, units and drawing limits. The orthographic constraints snap to objects manually and automatically. Creating drawings by using various coordinate input entry methods to draw straight lines and circles.

RATIONALE

To create a drawing using AutoCAD software, it is essential to know how to setup drawings, and draw lines and circles using different input methods. The expertise to draw using AutoCAD software enables engineers to produce engineering drawings very efficiently and in less time.

PREREQUISITE

Knowledge of AutoCAD user interface and drawing standards.

UNIT OUTCOMES

Upon successful completion of this module, the student will be able to:

U7-O1: Set up Graphical User Interface (GUI).
U7-O2: Draw Lines and Circles using AutoCAD.
U7-O3: Modify drawings using AutoCAD.
U7-O4: Apply various geometrical constraints.

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-7 Outcomes	Expected Mapping With Course Outcomes *(1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)*				
	CO-1	CO-2	CO-3	CO-4	CO-5
U7-O1			3	3	
U7-O2			3	3	
U7-O3			3	3	
U7-O4			3	3	

7.1 CREATING BASIC DRAWINGS USING AUTOCAD

Every drawing begins with the creation of basic geometry objects such as lines, circles, arcs, and rectangles. Normally, there is more than one way to accomplish a task using the software. The 2D Drafting & Annotations workspace with the toolbar ribbon at the top of the AutoCAD window is used to create 2D drawings. It is crucial to set-up a drawing page first in order to produce a drawing that makes sense and is visually attractive. These are the steps used for the setup of a drawing in AutoCAD software:

7.1.1 START AutoCAD Program

2D Drafting Using AutoCad Software

The AutoCAD program can be started by either double clicking on the program icon on the desktop or by clicking on it in the START menu.

On startup, a dialog box will appear showing four ways to start a drawing as shown in figure 7.1.

- Open an existing drawing
- Start a drawing from scratch
- Start a drawing based on a template
- Use wizards to set up drawing

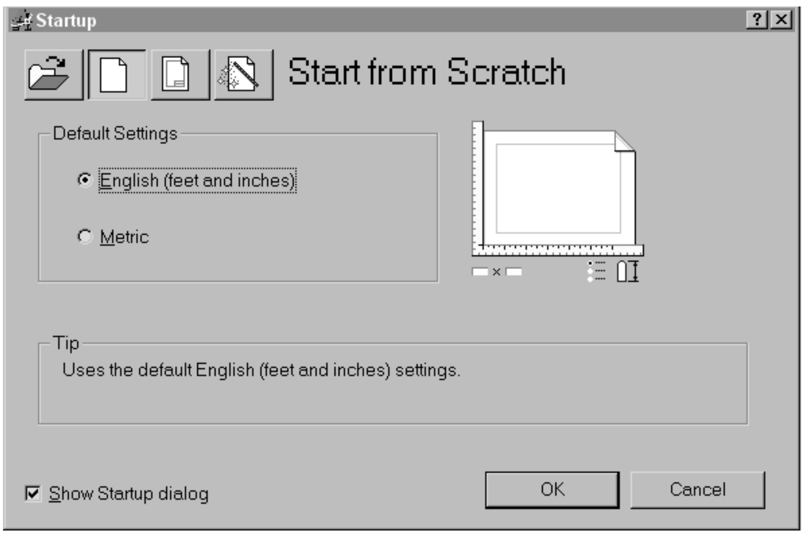


Figure 7.1: Startup Dialog Box

7.1.2 Setup Layout

There are two views in AutoCAD: a model space and a paper space. The drawing should always be defined (outlined) using the model space, and the dimensions that are added later should be represented using the paper space. To switch between the model space and paper space, one can toggle the tabs located at the bottom of the screen. The first tab is labeled "model space" and the other tabs are labeled either as "sheet" or "layout". The "sheet" or "layout" tabs indicate paper space. In model space, the background of the screen appears black and in paper space, the background appears white.

7.1.3 Set Drawing Units

Units represent the baseline of all the geometrical shapes that are created in the drawing. It is left to discretion of the user to determine what unit of measurement will be used in the drawing. The units (i.e. inches, millimeters, feet) used to draw objects in the drawing area can be selected using the UNITS command. The following dialog box will appear, allowing the user to select the units and the number of decimals displayed in the commands as shown in figure 7.2.

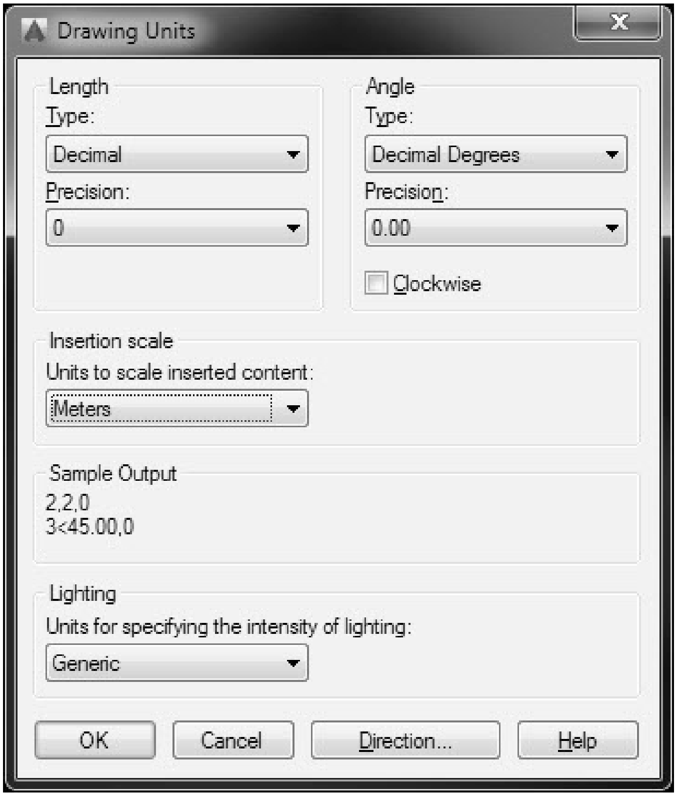


Figure 7.2: Drawing Units

7.1.4 Customize User Interface

The customize user interface (CUI) in AutoCAD allows tailoring the drawing environment to suit the user needs. These are the steps to be followed:

- Open the CUI editor by clicking the manage tab of the ribbon, customization panel.
- The user interface as shown in figure 7.3 will appear.
- In the CUI editor, the user can view the contents of the loaded customization files by expanding the elements in the tree structure and viewing the properties of the elements by selecting them.
- A long list of options will show up, displaying various toolbars, containing different commands. The most popular toolbars that are used for AutoCAD 2D drawings are the DRAW, MODIFY, and OBJECT PROPERTIES toolbars.

- Select these toolbars, as they pop-up on the screen and move them to the side to create space for the drawing.

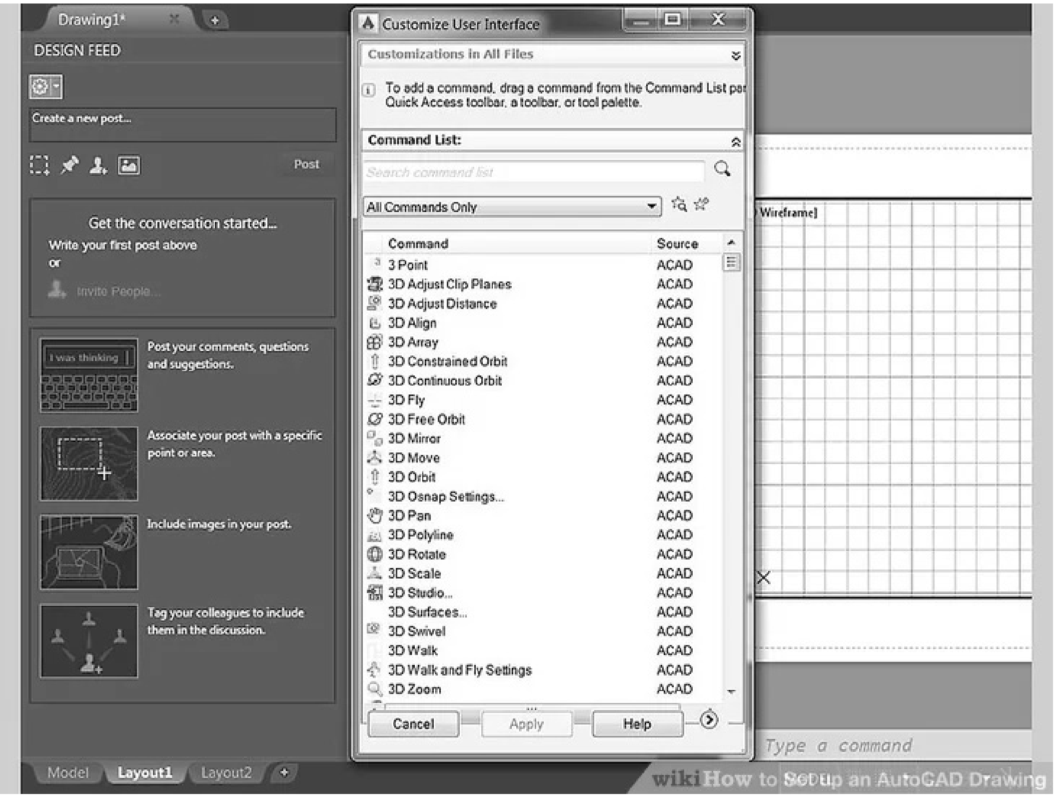


Figure 7.3: Customize User Interface

7.1.5 Set Drawing Limits

Drawing limits are used to set the boundaries of the drawing. The drawing boundaries are usually set to match the size of a sheet of drawing paper. The limits command in AutoCAD is used to create an invisible rectangular boundary in the drawing area or viewport. This restricts the grid display and the point locations. Users are then required to specify the coordinates of the opposite corners of the rectangular window.

Command: Limits

Specify lower left corner or [ON/OFF] <0, 0>: Specify a point

Specify upper right corner or <12, 9>: Specify a point

7.2 GEOMETRICAL CONSTRAINTS

Geometric constraints are used to control the relationships of objects in respect to each other. Geometric constraints associate geometric objects together or specify a fixed location or angle. Geometric constraints can be found on the parametric tab of the AutoCAD ribbon menu and as shown in figure 7.4. Constraint options include coincident, collinear, concentric, fix, parallel, perpendicular, horizontal, vertical, tangent, smooth, symmetric, and equal. One can apply geometric constraints to almost any object.

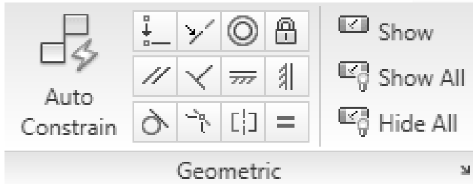
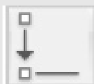
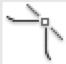
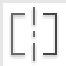


Figure 7.4: Geometric constraint options

There are 12 geometric constraints that can be applied to the drawing objects, and the easiest way to apply them is by clicking buttons on the geometric panel of the ribbon or toolbar on the parametric tab. Table 7.1 presents a list of the geometric constraints and the description of each constraint.

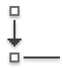
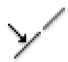
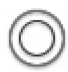
Table 7.1: Geometric Constrained and Description

S. No.	Symbol	Constraint	Description
1.		Coincident	Force to make coincident two or more points.
2.		Concentric	Forces the centers of two or more objects to coincident.
3.		Collinear	Forces two or more lines to be collinear.
4.		Parallel	Forces two or more lines to be parallel.
5.		Perpendicular	Forces two or more lines to be perpendicular to each other.
6.		Horizontal	Forces a line or two points to be horizontal relative to current coordinate system.
7.		Vertical	Forces a line or two points to be vertical.
8.		Fix	A fix geometric constraint forces a point or an entity to a fixed location.
9.		Tangent	Forces objects like line and circle to be tangent to each other.

S. No.	Symbol	Constraint	Description
10.		Smooth	A smooth constraint forces a spline to maintain geometric continuity with another spline, line, arc, or polyline.
11.		Symmetric	The Symmetry constraint enables to constrain two sets of entities in a 2D profile so that they are symmetric to each other with respect to a symmetry axis.
12.		Equal	Force two objects or segments to be the same dimension.

Applying Geometric Constraints

The following examples are used to demonstrate the procedure of applying different constraints.

1. **To apply a coincident constraint**
   - Click Parametric tab → Geometric panel → Coincident. 
   - Select the first and second points on two different objects.
   - The second point is made coincident to the first.
2. **To apply a collinear constraint**
   - Click Parametric tab → Geometric panel → Collinear. 
   - Select the first and second object. You can select either a line object or a polyline sub object.
   - The second object is made collinear to the first.
3. **To apply a concentric constraint**
   - Click Parametric tab → Geometric panel → Concentric. 
   - Select the first and second arc or circle object.
   - The second arc or circle moves to have the same center point as the first object.
4. **To apply a fix constraint**
   - Click Parametric tab → Geometric panel → Fix. 
   - Select a point on an object.
   - Applying the Fix constraint to a point on the object locks the node in place.
5. **To apply a parallel constraint**
   - Click Parametric tab → Geometric panel → Parallel. 
   - Select the two objects to be made parallel , either a line object or a polyline sub object.
   - The second object is made parallel with the first object.
6. **To apply a perpendicular constraint**
   - Click Parametric tab → Geometric panel → Perpendicular. 

- Select the two objects to be made perpendicular. You can select either a line object or a polyline sub object.
- The second object is made perpendicular to the first object.

7. To apply a horizontal constraint

- Click Parametric tab → Geometric panel → Horizontal. 
- Select the line object or polyline sub object to be made horizontal.

8. To apply a vertical constraint

- Click Parametric tab → Geometric panel → Vertical. 
- Select the line object or polyline subobject to be made vertical.

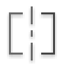
9. To apply a tangent constraint

- Click Parametric tab → Geometric panel → Tangent. 
- Select the two objects to be made tangential.
- The second object maintains a point of tangency with the first object.

10. To apply a smooth constraint

- Click Parametric tab → Geometric panel → Smooth. 
- Select the first spline curve.
- Select the second spline, line, polyline (subobject), or arc object.
- The two objects are updated to be continuous with one another.

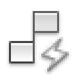
11. To apply a symmetric constraint

- Click Parametric tab → Geometric panel → Symmetric. 
- Select the first and second object.
- Select the symmetry line.
- The selected objects become symmetrically constrained about the selected line.

12. To apply an equal constraint

- Click Parametric tab → Geometric panel → Equal.
- Select the first and second object.
- The second object is made equal to the first object.

13. To apply multiple geometric constraints to an object

- Click Parametric tab → Geometric panel → AutoConstraint. 
- Select the objects that you want to constrain.
- Press Enter when you select the objects to be automatically constrained.
- The Command prompt displays the number of constraints applied.

7.3 OBJECTS SNAPS (OSNAP)

The Object Snaps (OSNAPS) are drawing aids which are used in conjunction with other commands to help draw accurately. OSNAPS allow users to *snap* onto a specific object location when picking a point. Using OSNAPS users can draw conveniently and accurately.

Given below are four basic methods of accessing the OSNAPS:

- The OSNAPS are available from a *flyout* button on the standard toolbar, as illustrated in figure 7.5.
- The OSNAPS are available on the Object Snap toolbar.
- To access the OSNAP from the cursor menu – just hold the *Shift* key down on the keyboard and right-click the mouse to bring up the cursor menu.
- To access the OSNAP from the command prompt, one can do so by typing OSNAP.

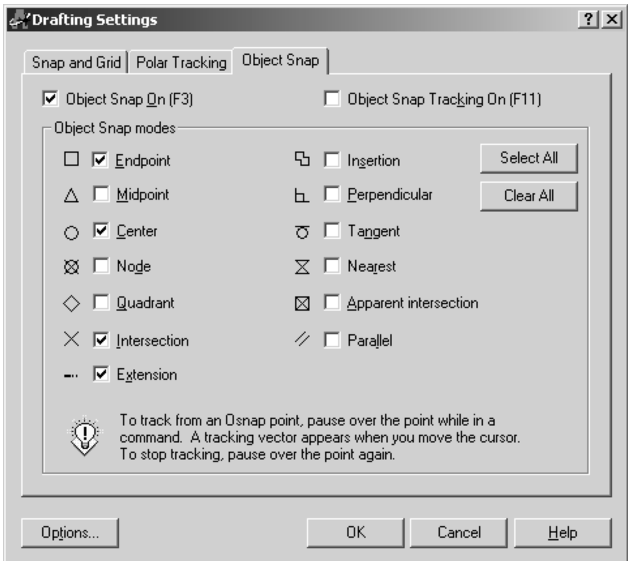


Figure 7.5: OSNAP Toolbar

OSNAP Modes

OSNAPS options not only facilitate the making of drawings precisely, but they also speed up the drawing workflow. Table 7.2 presents a list of thirteen OSNAP options with a description of each option.

Table 7.2: OSNAPS MODES

S. No.	OSNAP Mode	Description
1.	ENDpoint	Snaps to the closest endpoint or corner of a geometric object
2.	MIDpoint	Snaps to the midpoint of a geometric object
3.	CENter	Snaps to the center of an arc, circle, ellipse, or elliptical arc
4.	NODe	Snaps to a point object, dimension definition point, or dimension text origin
5.	QUAdrant	Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc
6.	INTersection	Snaps to the intersection of geometric objects
7.	EXTension	Causes a temporary extension line or arc to be displayed when you pass the cursor over the endpoint of objects, so you can specify points on the extension

S. No.	OSNAP Mode	Description
8.	INSertion	Snaps to the insertion point of objects such as an attribute, a block, or text.
9.	PERpendicular	Snaps to a point perpendicular to the selected geometric object
10.	TANgent	Snaps to the tangent of an arc, circle, ellipse, elliptical arc, polyline arc, or spline.
11.	NEArest	Snaps to the nearest point on an object such as an arc, circle, ellipse, elliptical arc, line, point, polyline, ray, spline, or xline.
12.	APParent intersection	Snaps to the visual intersection of two objects that do not intersect in 3D space but may appear to intersect in the current view.
13.	PARallel	Constraints a new line segment, polyline segment, ray or xline to be parallel to an existing linear object.

7.4 SNAP AND GRID

Snap restricts the movement of the crosshairs cursor to an interval that one defines. When the snap mode is on, the cursor adheres or "snaps" to an invisible grid. The snap mode is useful for specifying precise points with the cursor.

Grid shows a rectangular pattern of lines over the drawing area. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps to align and visualize the distance between objects. The grid does not appear in the plotted drawing.

To use the snap mode grid display – follow these steps. Right click the Snap Mode or Grid Display button on the status bar and choose settings from the menu that appears. The Drafting Settings dialog box will appear with the Snap and Grid tab selected as shown in figure 7.6.

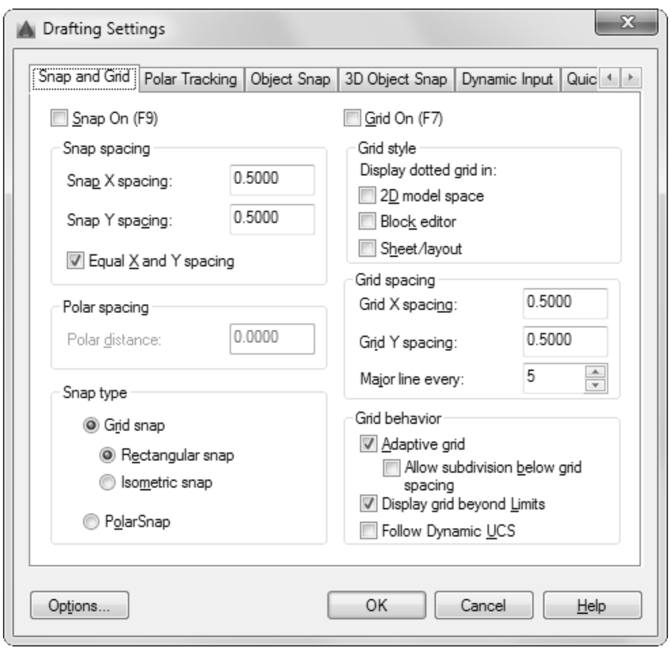


Figure 7.6: Snap and Grid in Drafting Setting

Steps to set grid spacing

1. In the Drafting Settings dialog box, Snap and Grid tab, ensure that Grid On (F7) is selected as illustrated in figure 7.7.

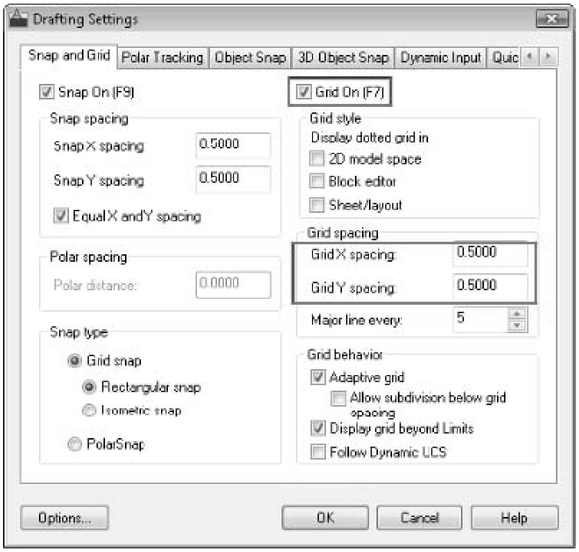


Figure 7.7: Grid Setup

2. Under Grid Spacing, do the following:
   - In the Grid X Spacing box, enter 0.5000 to set the horizontal grid spacing in units.
   - In the Grid Y Spacing box, enter 0.5000 to set the vertical grid spacing in units.
   - Click OK. The grid will appear as shown in figure 7.8.

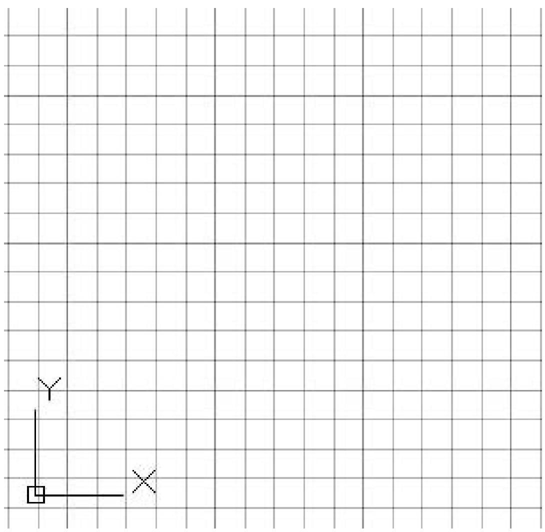


Figure 7.8: Grid Spacing

Steps to set snap spacing

1. In the Drafting Settings dialog box, Snap and Grid tab, ensure that Snap On (F9) is selected as illustrated in figure 7.9.

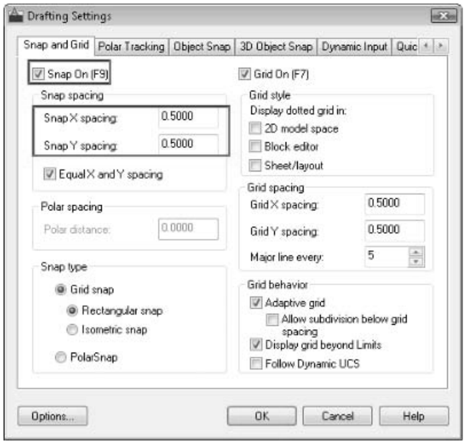


Figure 7.9: Snap On

2. Under Snap Spacing, do the following:
   - ❑ In the Snap X Spacing box, enter 0.5000 to set the horizontal snap spacing value in units.
   - ❑ In the Snap Y Spacing box, enter 0.5000 to set the vertical snap spacing value in units.
3. Click OK.

7.5 ORTHO MODE

Ortho is short for *orthogonal*, which means either vertical or horizontal. The ortho mode in AutoCAD is used to restrict cursor movement to specific directions. It allows the cursor movement only in the vertical and horizontal direction. Ortho mode can be toggled on or off in one of the following three ways. The quickest way is just to click "on" the ORTHO button on the status bar.

Ortho Mode Access Options

- Status Bar: 

- Command Prompt : ORTHO
- Keyboard : F8

In the illustration 7.10, a line is drawn using ortho mode on/off. Point 1 is the first point specified, and point 2 is the position of the cursor when the second point is specified.

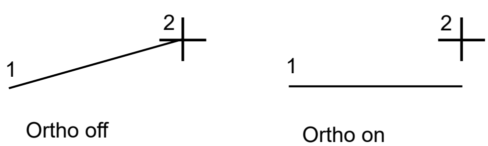
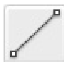


Figure 7.10: Ortho Mode (ON / OFF)

7.6 DRAW COMMANDS

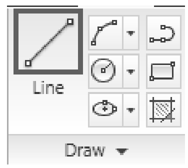
The draw commands can be used to create basic objects such as lines and circles. Most AutoCAD drawings are composed purely and simply from these basic components. A good understanding of the draw commands is fundamental to the efficient use of AutoCAD. In common with most AutoCAD commands, the draw commands can be started in a number of ways. Command names or short-cuts can be entered at the keyboard; commands can be started from the draw pull-down menu, shown on the right or from the draw toolbar.The sections below cover the most frequently used draw commands such as lines, circles, arcs, rectangles, and polygons.

7.6.1 Line Command 

Use the Line command to create a single line or multiple line segments from a start point to an end point.

Command Access

- Command Line: Line, L
- Menu Bar: Draw → Line
- Ribbon: Home tab → Draw panel → Line

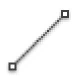


Command Options:

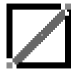
Option	Description
First point	(default) Specifies the start point of the line segment.
Next point	(default) Specifies the endpoint of the line segment. Continue to specify next points for additional line segments.
Undo	Removes the previous line segment without exiting the Line command. Select or enter the capitalized letter only.
Close	Appears only after you have drawn two line segments. Uses the first point of the line segments as the next point for the current segment to create a closed boundary of line segments. Select or enter the capitalized letter only.

Examples: Line Command

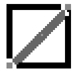
i. Draw a line using Specific Coordinates

1. Click Home tab → Draw panel → Line. 
2. Specify the start point.
   Type the coordinate value for the first point by typing the X value, a comma, then the Y value, for example 1.65,4.25.
3. Complete the first line segment by specifying the endpoint.
   Type the X value, a comma, then the Y value, for example 4.0, 6.75.
   To undo the previous line segment during the LINE command, enter u or click Undo on the toolbar.
4. Press ENTER to end or c to close a series of line segments.

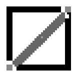
ii. Draw a line using Relative Coordinate

1. Click Home tab → Draw panel → Line.  Find
2. Specify the first point.
3. To specify the second point relative to the first point, do one of the following:
   Type the sign (@) followed by the X-value, a comma, then the Y-value, for example @4.0, 6.75.
   Press the Spacebar or Enter.

iii. Draw a line of specific Length

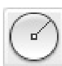
1. Click Home tab → Draw panel → Line.  Find
2. Specify the start point.
3. Do one of the following to specify the length:
   - ➢ Move the cursor to indicate the direction and angle, and enter the length, for example 6.5.
   - ➢ Enter the at symbol (@) and the length, followed by the left angle bracket (<) and the angle, for example @6.5<45.

iv. Draw a line at a specific Angle

1. Click Home tab → Draw panel → Line.  Find
2. Specify the start point.
3. Do one of the following to specify the angle:
   - ➢ Enter the left angle bracket (<) and the angle, for example <45, and move the cursor to indicate the direction.
   - ➢ Enter polar coordinates, for example 2.5<45.
   - ➢ Move the cursor to indicate an approximate angle.

   Do one of the following to specify the length:
   - ➢ Click a point to specify the endpoint with or without using object snaps.
   - ➢ Enter the length of the line, for example 2.5.

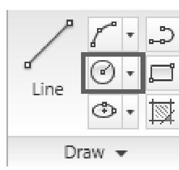
   Press the Spacebar or Enter.

7.6.2 Circle Command 

Use the circle command to create circles in the drawing. When the circle command is started, one is prompted to select a center point, then specify the radius. Use the data input methods discussed earlier to input these values.

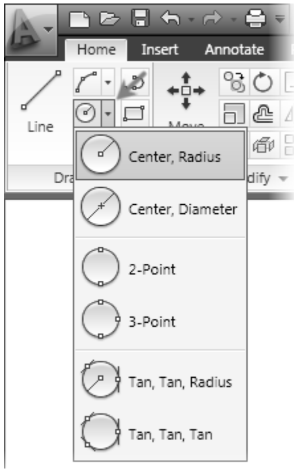
Command Access

- Command Line: CIRCLE, C
- Menu Bar: Draw → Circle → choose option
- Ribbon: Home tab → Draw panel → Circle



Circle Command Options

Circle options can be accessed from the drop down menu next to the Circle button.



Option	Description
Specify centre point	(default) Click a point or enter a coordinate for the center of the circle.
D	After specify a center point, user have the option to specify a Diameter instead of the radius.
3P	Create the circle based on three points specify to define the circle's diameter.
2P	Create the circle based on two points specify to define the circle's diameter.
Ttr (tan tan radius)	Create a circle tangent to other objects at a radius specify.

Examples: Circle Command

1. **Draws a circle based on a center point and a diameter or a radius.**

   Specify center point for circle or [_3P/ 2P/ Ttr (tan tan radius)]: *Specify a point or enter an option*

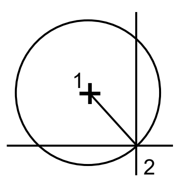
   Centre Point

   Draw a circle based on a center point and a diameter or a radius.

   Specify radius of circle or [Diameter]: *Specify a point, enter a value, enter **d**, or press* Enter

   Radius

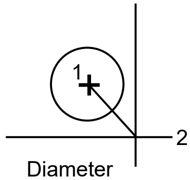
   Defines the radius of the circle. Enter a value, or specify a point.



Radius

Diameter

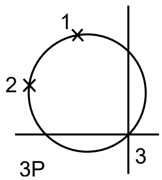
Defines the diameter of the circle. Enter a value, or specify a second point.



Diameter

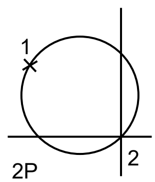
Specify diameter of circle <*current*>: *Specify a point (2), enter a value, or press* Enter

2. **3P (three point)**
   Draws a circle based on three points on the circumference.
   Specify first point on circle: *Specify a point (1)*
   Specify second point on circle: *Specify a point (2)*
   Specify third point on circle: *Specify a point (3)*



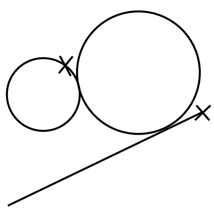
3P

3. **2P (two point)**
   Draws a circle based on two endpoints of the diameter.
   Specify first endpoint of circle's diameter: *Specify a point (1)*
   Specify second endpoint of circle's diameter: *Specify a point (2)*

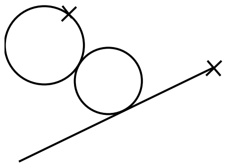
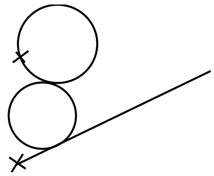


2P

4. **TTR (Tangent, Tangent, Radius)**
   Draw a circle with a specified radius tangent to two objects.
   Specify point on object for first tangent of circle: *Select a circle, arc, or line*
   Specify point on object for second tangent of circle: *Select a circle, arc, or line*
   Specify radius of circle <*current*>:

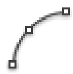

tan, tan, radius

Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.

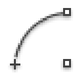



7.6.3 Arc Command

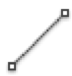
i. To draw an arc by specifying three points

1. Click Home tab → Draw panel → 3-Point. 
2. Specify the start point.
3. Specify a point on the arc.
4. Specify the endpoint.

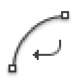
ii. To draw an arc using a start point, a center point, and an endpoint

1. Click Home tab → Draw panel → Start, Center, End. 
2. Specify a start point.
3. Specify the center point.
4. Specify the endpoint.

iii. To continue an arc with a tangential line

1. Complete the arc.
2. Click Home tab → Draw panel → Line. 
3. Press ENTER at the first prompt.
4. Enter the length of the line and press ENTER.

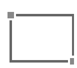
iv. To continue an arc with a tangential arc

1. Complete the arc.
2. Click Home tab → Draw panel → Continue. 

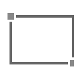
   Specify the second endpoint of the tangent arc.

7.6.4 Rectangle Command

i. Draw rectangle by two corner points

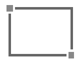
1. Click Home tab → Draw panel → Rectangle. 
2. Specify the first corner of the rectangle.
3. Specify the other corner of the rectangle.

ii. Draw rectangle by Length and Width

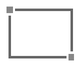
1. Click Home tab → Draw panel → Rectangle. 
2. Specify the first corner of the rectangle.
3. Enter D for Dimensions.
4. Enter the length.

5. Enter the width.
6. Specify the other corner.

iii. Draw rectangle by Area

1. Click Home tab → Draw panel → Rectangle.  Find
2. Specify the first corner of the rectangle.
3. Enter A for Area.
4. Enter the area.
5. Do one of the following:
   - Enter L to enter the length. The width is calculated based on the length and the area.
   - Enter W to enter the width. The length is calculated based on the width and the area.
   - Enter the value for the length or width.

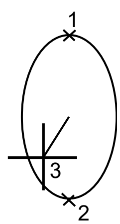
iv. Draw rectangle with Rotation

1. Click Home tab → Draw panel → Rectangle.  Find
2. Specify the first corner of the rectangle.
3. Enter R for Rotation.
4. Enter the rotation value or enter P to pick two points to define the angle of rotation.
5. Specify the other corner.

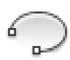
7.6.5 Draw Ellipse and Elliptical Arc Command

i. To draw a true ellipse using endpoints and distance

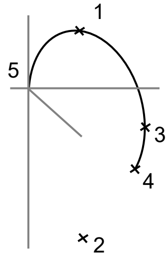
1. Click Home tab → Draw panel → Axis, End. 
2. Specify the first endpoint of the first axis (1).
3. Specify the second endpoint of the first axis (2).
4. Drag the pointing device away from the midpoint, and click to specify a distance (3) for half the length of the second axis.



ii. To draw an elliptical arc using start and end angles

1. Click Home tab → Draw panel → Arc. 
2. Specify endpoints for the first axis (1 and 2).
3. Specify a distance to define half the length of the second axis (3).
4. Specify the start angle (4).
5. Specify the end angle (5).

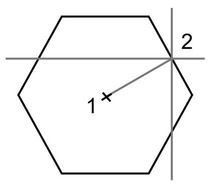
The elliptical arc is drawn counterclockwise between the start point and endpoint.



7.6.6 Polygon Command

i. To draw a circumscribed polygon

1. Click Home tab → Draw panel → Polygon. 
2. At the command prompt, enter the number of sides.
3. Specify the center of the polygon (1).
4. Enter **c** to specify a polygon circumscribed about a circle.
5. Enter the radius length (2).



ii. To draw a polygon by specifying one edge

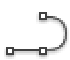
1. Click Home tab → Draw panel → Polygon. 
2. At the command prompt, enter the number of sides.
3. Enter **e** (Edge).
4. Specify the start point for one polygon segment.
5. Specify the endpoint of the polygon segment.

iii. To draw an inscribed polygon

1. Click Home tab → Draw panel → Polygon. 
2. At the command prompt, enter the number of sides.
3. Specify the center of the polygon.
4. Enter **i** to specify a polygon inscribed within a circle of specified points.
5. Enter the radius length.

7.6.7 Polyline Command

i. Draw a Polyline with Straight Segments

1. Click Home tab → Draw panel → Polyline.  Find
2. Specify the first point of the polyline.
3. Specify the endpoint of the first segment.

4. Continue specifying segment endpoints as needed.
5. Press Enter to end, or enter c to close the polyline.

> **Important observation**
> To start a polyline at the endpoint of the last polyline drawn, start the command again and press Enter at the Specify Start Point prompt.

ii. Draw a Wide Polyline

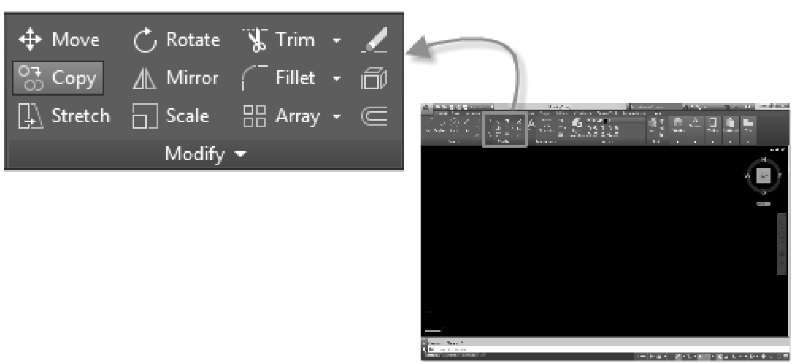
1. Click Home tab → Draw panel → Polyline. Find
2. Specify the first point of the polyline.
3. Enter w (Width)
4. Enter the starting width of the segment.
5. Specify the ending width of the segment using one of the following methods:
   - ➢ To create a segment of equal width, press Enter.
   - ➢ To create a tapering or increasing segment, enter a different width.
6. Specify the endpoint of the segment.
7. Continue specifying segment endpoints as needed.
8. Press Enter to end, or enter c to close the polyline.

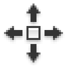
iii. Draw a Polyline with Straight and Curved Segments

1. Click Home tab → Draw panel → Polyline. Find
2. Specify the first point of the polyline.
3. Specify the endpoint of the first segment.
4. Switch to Arc mode by entering a (Arc) at the Command prompt.
5. Return to Line mode by entering L (Line).
6. Specify additional segments as needed.
7. Press Enter to end, or enter c to close the polyline.

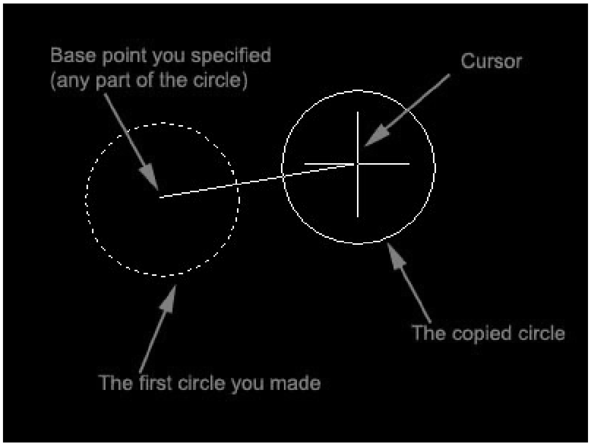
7.7 MODIFY COMMANDS

Modify commands are used to perform editing operations such as move, copy and trim on the objects in a drawing. The most common of these tools are located on the Modify panel of the Home tab as shown below.



7.7.1 Move command 

Moves objects by a specified distance in a specified direction. Use coordinates, grid snaps, object snaps, and other tools to move objects with precision.



The following prompts are displayed.

Select objects

Specifies which objects to move.

Base point

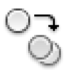
Specifies the start point for the move.

Second point

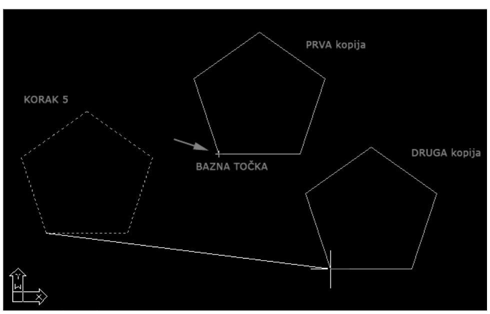
In combination with the first point, specifies a vector that indicates how far, and in what direction, the selected objects are moved.

Displacement

Specifies a relative distance and direction.

7.7.2 Copy Command 

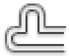
Copies objects a specified distance in a specified direction.



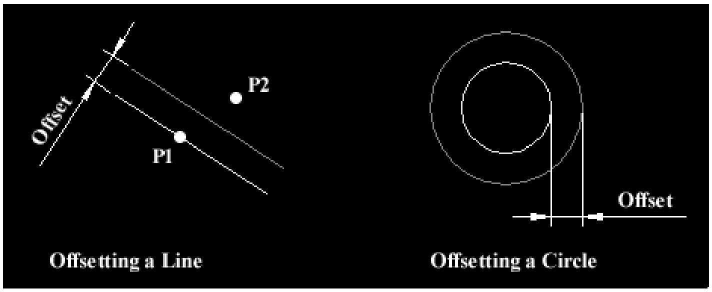
With the COPYMODE system variable, user can control whether multiple copies are created automatically.

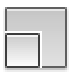
The following prompts are displayed:

- Select objects: *Use an object selection method and press Enter*
- Specify base point or [Displacement/mOde/Multiple] <Displacement>: *Specify a base point or enter an option*
- Specify second point or [Array] <use first point as displacement>: *Specify a second point or enter an option*

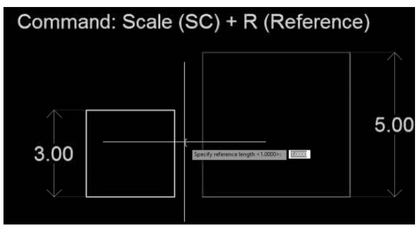
7.7.3 Offset Command 

Creates concentric circles, parallel lines, and parallel curves. User can offset an object at a specified distance or through a point.



7.7.4 SCALE (SC) Command 

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



The following prompts are displayed.

Select objects

Specifies which objects to resize.

Base point

Specify a base point for the scale operation.

The base point specify identifies the point that remains in the same location as the selected objects change size (and thus move away from the stationary base point).

> **Note**
> When you use the SCALE command with *annotative* objects, the position or location of the object is scaled relative to the base point of the scale operation, but the size of the object is not changed.

Scale Factor

Multiplies the dimensions of the selected objects by the specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects. User can also drag the cursor to make the object larger or smaller.

Copy

Creates a copy of the selected objects for scaling.

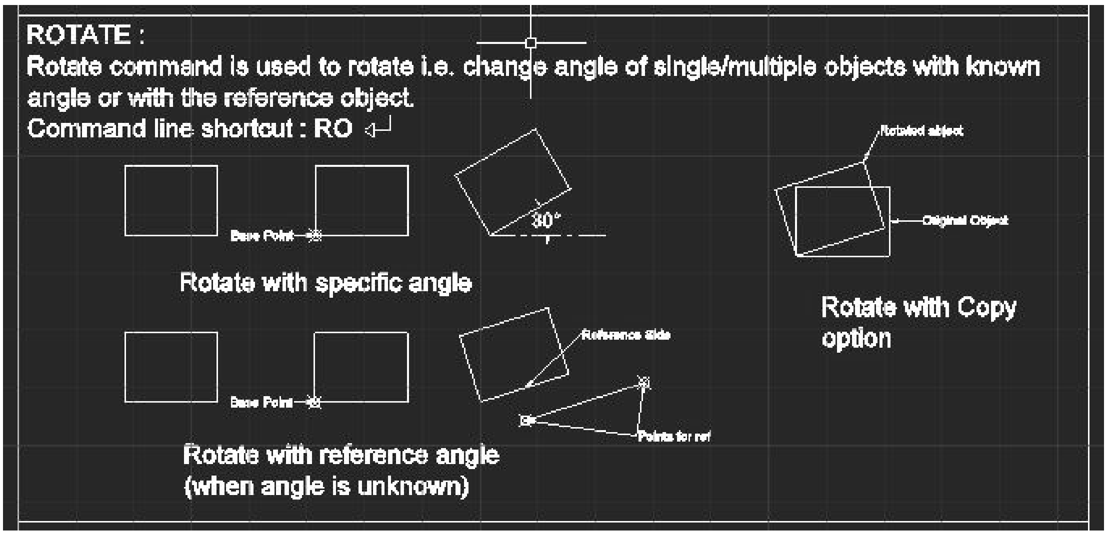
Reference

Scales the selected objects based on a reference length and a specified new length.

7.7.5 Rotate Command

Rotates objects around a base point.

User can rotate selected objects around a base point to an absolute angle.



The following prompts are displayed.

Select objects

Use an object selection method and press Enter when finish.

Specify base point

Specify a point.

Specify rotation angle

Enter an angle, specify a point, enter ***c*** , or enter **r**.

- **Rotation Angle:** Determines how far an object rotates around the base point. The axis of rotation passes through the specified base point and is parallel to the *Z* axis of the current UCS.
- **Copy:** Creates a copy of the selected objects for rotation.
- **Reference:** Rotates objects from a specified angle to a new, absolute angle. When rotate a viewport object, the borders of the viewport remain parallel to the edges of the drawing area.

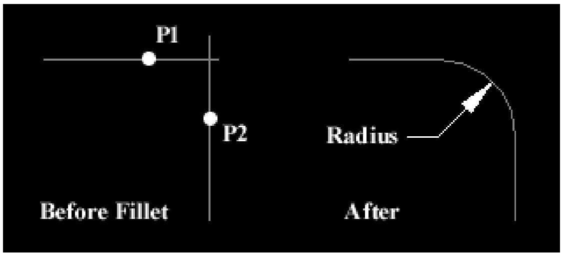
7.7.6 Fillet Command 

Rounds or fillets the edges of two 2D objects or the adjacent faces of 3D solid.

A round or fillet is

- An arc that is created tangent between two 2D objects.
- An curved transition between two surfaces or adjacent faces on a 3D solid.

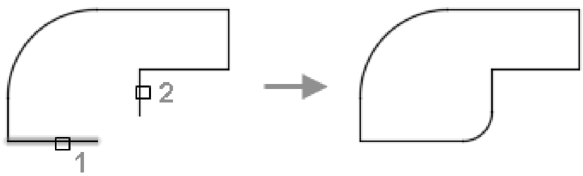
In this example, an arc is created tangent to the selected lines, which are trimmed to meet the endpoints of the arc.



The following prompts are displayed when creating a 2D fillet.

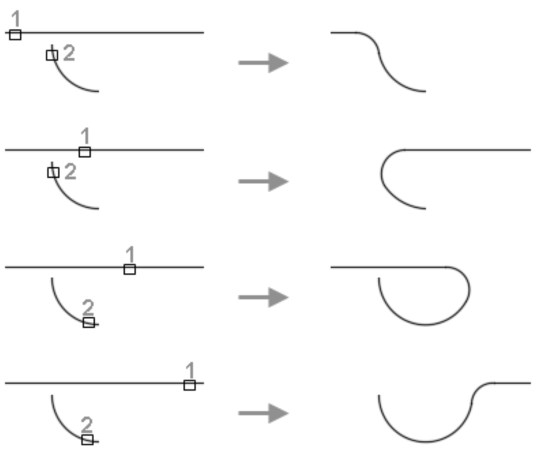
First Object

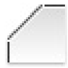
Select the first of two objects or the first line segment of a 2D polyline to define the fillet.



Second object or shift-select to apply corner

Select the second object or line segment of a 2D polyline to define the fillet.

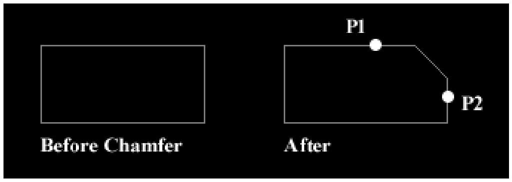


7.7.7 Chamfer Command 

Bevels or chamfers the edges of two 2D objects or the adjacent faces of a 3D solid.
A bevel or chamfer is

- An angled line that meets the endpoints of two straight 2D objects.
- A sloped transition between two surfaces or adjacent faces on a 3D solid.

The distances and angles applied in the order that user select the objects.



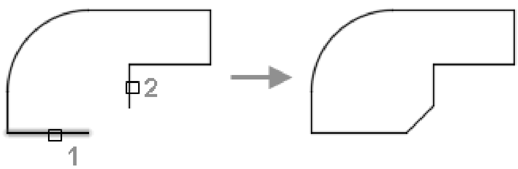
Create 2D Chamfer

A bevel or chamfer can be defined by selecting two objects of the same or different object types: lines, polylines, rays, and xlines.

The following prompts are displayed when creating a 2D chamfer.

First Line

Select the first of two objects or the first line segment of a 2D polyline to define the chamfer.



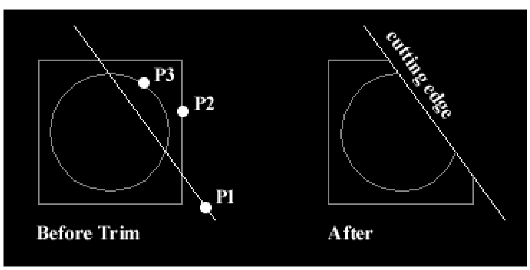
Second line or shift-select to apply corner

Select the second object or line segment of a 2D polyline to define the chamfer.

7.7.8 Trim Command 

Trims objects to meet the edges of other objects.

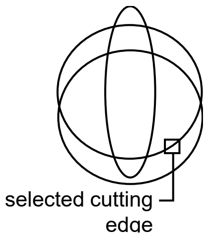
To trim objects, select the boundaries and press Enter. Then select the objects that to trim. To use all objects as boundaries, press Enter at the first Select Objects prompt.



The following prompts are displayed.

Select cutting edges

Specifies one or more objects to be used as a boundary for the trim. TRIM projects the cutting edges and the objects to be trimmed onto the *XY* plane of the current user coordinate system (UCS).



Select objects

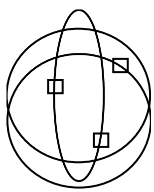
Specifies objects individually.

Select all

Specifies that all objects in the drawing can be used as a trim boundary.

Object to Trim

Specifies the object to trim. If more than one trim result is possible, the location of the first selection point determines the result.



7.8 DIMESIONING

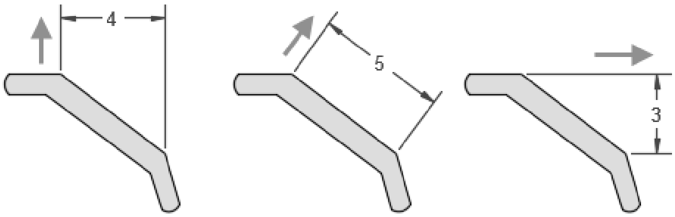
Dimensions are used to indicate the measurements of design components. It is very important to use the correct dimensions when drawing. The dimensions will be in the units set in the drawing setup page. When making drawings with AutoCAD software users can add dimensions either from the HOME or the ANNOTATE tab. The appearance of dimension can be control by setting up dimension styles or by editing individual dimensions in special cases.

In AutoCAD user can create several types of dimensions for a variety of object types in many orientations and alignments. The basic types of dimensioning are:

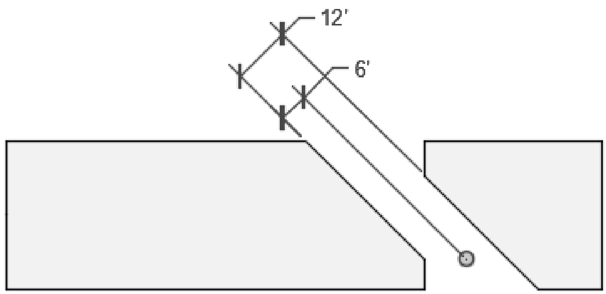
1. Linear
2. Radial,
3. Angular,
4. Ordinate, and
5. Arc length.

7.8.1 Linear Dimensions

Linear dimensions can be horizontal, vertical, or aligned. user can create an aligned, horizontal, or vertical dimension with the DIM command depending on how user move the cursor when placing the text.

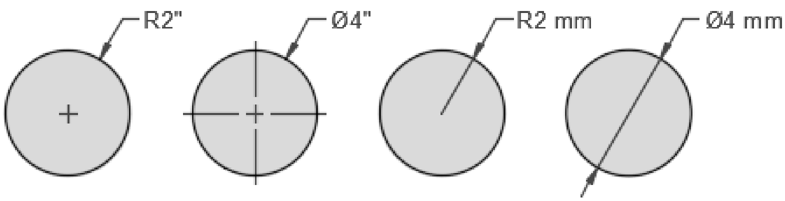


In rotated dimensions, the dimension line is placed at an angle to the extension line origin points. In this example, the angle specified for dimension rotation is equal to the angle of the slot.



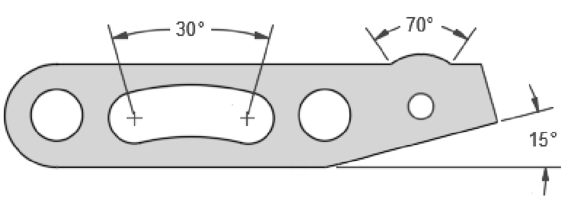
7.8.2 Radial Dimensions

A radial dimension measures the radius or diameter of arcs and circles with an optional centerline or center mark. Several options are displayed in the illustration.



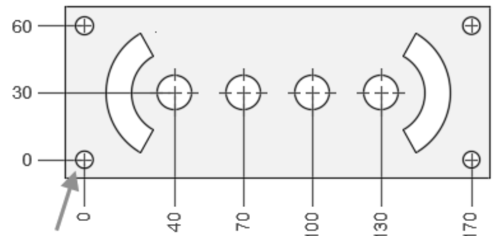
7.8.3 Angular Dimensions

Angular dimensions measure the angle between two selected geometric objects or three points. From left to right, the example shows angular dimensions created using a vertex and two points, an arc, and two lines.



7.8.4 Ordinate Dimensions

Ordinate dimensions measure the perpendicular distances from an origin point called the *datum*, such as a hole in a part. These dimensions prevent escalating errors by maintaining accurate offsets of the features from the datum.

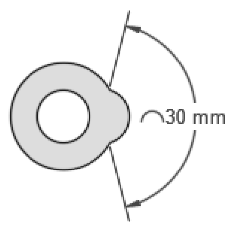


In this example, the datum (0,0) is indicated as the hole in the lower-left corner of the illustrated panel.

7.8.5 Arc Length Dimensions

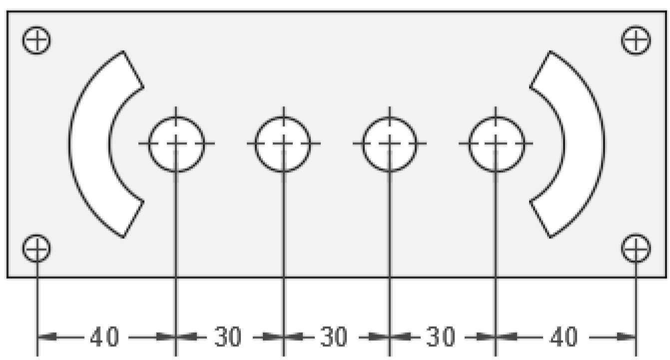
Arc length dimensions measure the distance along an arc or polyline arc segment. Typical uses of arc length dimensions include measuring the travel distance around a cam or indicating the length of a cable.

To differentiate them from linear or angular dimensions, arc length dimensions display an arc symbol by default. The arc symbol, also called a hat or cap, is displayed either above the dimension text or preceding the dimension text.

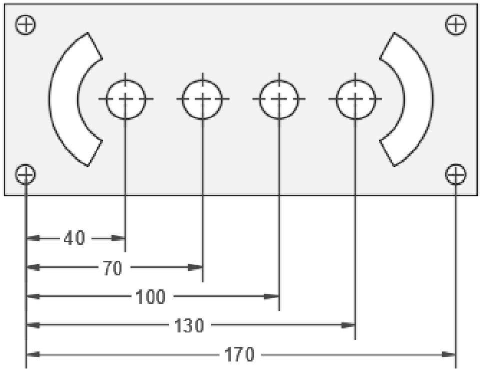


7.8.6 Baseline and Continued Dimensions

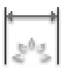
Continued dimensions, also called chained dimensions, are multiple dimensions placed end-to-end.



Baseline dimensions are multiple dimensions with offset dimension lines measured from the same location.



Creates multiple types of dimensions within a single command session.



When hover over an object for dimensioning, the DIM command automatically previews a suitable dimension type to use. Select objects, lines, or points to dimension and click anywhere in the drawing area to draw the dimension.

The supported dimension types range from vertical, horizontal, aligned, and rotated linear dimensions, to angular dimensions, to radius, diameter, jogged radius, and arc length dimensions, to baseline and continued dimensions. If required, user can change the dimension type using command line options.

The following prompts are displayed.

Select objects

Automatically selects an applicable dimension type for the objects user select and displays the prompts corresponding to that dimension type.

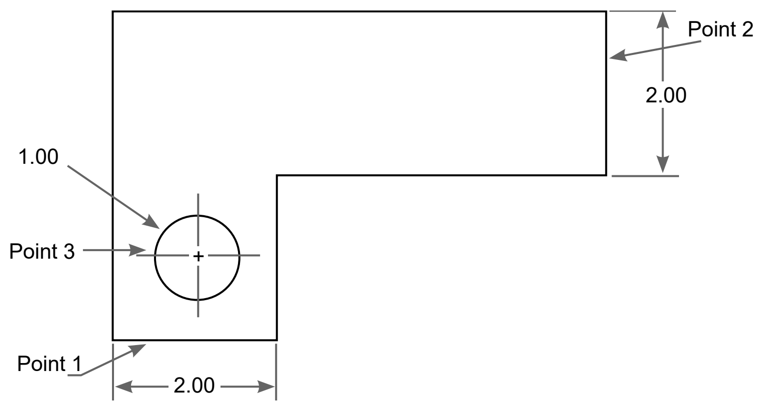
Selected object type	Action
Arc	Defaults the dimension type to radius dimensions.
Circle	Defaults the dimension type to radius dimensions.
Line	Defaults the dimension type to linear dimensions.
Dimension	Displays options to modify the selected dimension.
Ellipse	Defaults to options set for selecting a line.

Example: Creating Dimensions

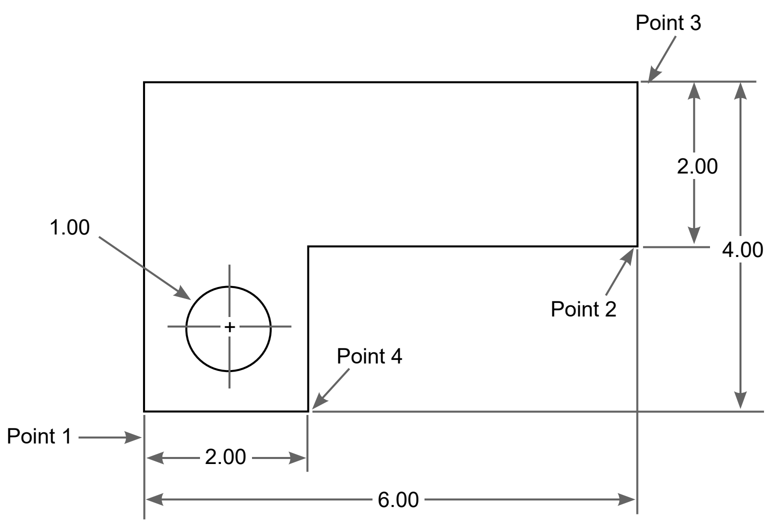
Select the Annotate tab of the ribbon to switch to the annotation tools.

1. Select the Dim Layer Override drop-down list on the Dimensions panel and set the Dim layer current.
2. Choose the Dimension tool from the Dimensions panel to start the DIM command. AutoCAD prompts to *Specify objects or specify first extension line origin or*  .

3. Pick the line at the bottom of the drawing. AutoCAD starts dragging a dimension from that line and prompts to *Specify dimension line location or second line for angle* ↓.



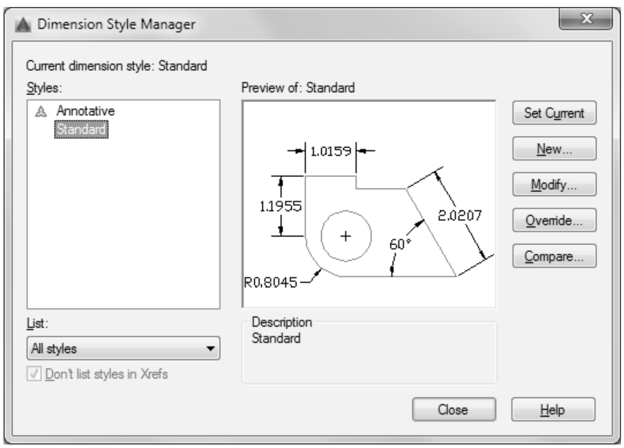
4. Pick a point below the line. The dimension is placed and AutoCAD prompts to *Select objects or specify first extension line origin or* ↓.
5. Pick the line at the right side of the drawing point 2.
6. Drag the dimension to the right and pick a point to place it. The dimension is placed and AutoCAD prompts to *Select objects or specify first extension line origin or* ↓.
7. Select the circle and pick a point above and to the left of the circle point 3 in . The dimension is placed and AutoCAD prompts to *Select objects or specify first extension line origin or* ↓.
8. Press the down arrow and choose the Baseline option from the menu. AutoCAD prompts to *Specify first extension line origin as baseline or* ↓.
9. Pick the left dimension line of the first dimension created point 1. A dimension line rubber-band from the dimension selected.



10. Make sure Object Snap toggle is turned on the status bar, and move cursor near point 2. When the Endpoint object snap appears, pick that point to select the endpoint of that line. AutoCAD will place the dimension and start dragging a new dimension.
11. AutoCAD prompts to *Specify a second extension line origin or* . Press the down arrow and choose the Select option from the menu. AutoCAD prompts to *Specify first extension line origin as baseline or*  .
12. Pick the upper dimension line of the vertical dimension point 3.
13. Move cursor near point 4. When the Endpoint object snap appears, pick that point to select the endpoint of that line. AutoCAD will place the dimension and start dragging a new dimension.
14. Press <Esc> twice to end the DIM command.
15. Save the drawing.

7.9 DIMENSION STYLE MANAGER

Creates new styles, sets the current style, modifies styles, sets overrides on the current style, and compares styles.



List of Options

The following options are displayed.

Current Dimension Style

Displays the name of the dimension style that is current. The default dimension style is STANDARD. The current style is applied to dimensions user create.

Styles

Lists dimension styles in the drawing. The current style is highlighted. Right-click in the list to display a shortcut menu with options to set the current style, rename styles, and delete styles. User cannot delete a style that is current or in use in the current drawing.

The  icon before the style name indicates that the style is annotative.

Unless you select Don't List Styles in Xrefs, dimension styles are displayed in externally referenced drawings using the syntax for externally referenced named objects. Although user cannot change, rename, or make current externally referenced dimension styles, user can create new styles based on them.

The item selected in List controls the dimension styles displayed.

List

Controls the display of styles in the Styles list. Select All Styles if user want to see all dimension styles in a drawing. Select Styles in Use if user want to see only the dimension styles currently used by dimensions in the drawing.

Preview

Shows a graphic representation of the style selected in the Styles list.

Description

Describes the style selected in the Styles list relative to the current style. If the description is longer than the space provided, user can click in the pane and use arrow keys to scroll down.

Set Current

Sets the style selected under Styles to current. The current style is applied to dimensions user created.

New

Displays the Create New Dimension Style dialog box, in which user can define a new dimension style.

Modify

Displays the Modify Dimension Styles dialog box, in which user can modify dimension styles. Dialog box options are identical to those in the New Dimension Style dialog box.

Override

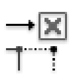
Displays the Override Current Style dialog box, in which user can set temporary overrides to dimension styles. Dialog box options are identical to those in the New Dimension Style dialog box. Overrides are displayed as unsaved changes under the dimension style in the Styles list.

Compare

Displays the Compare Dimension Styles dialog box, in which user can compare two dimension styles or list all the properties of one dimension style.

7.10 DIMENSION ASSOCIATE

Associates or re-associates selected dimensions to objects or points on objects.



Each selected dimension is highlighted in turn, and prompts for association points appropriate for the selected dimension are displayed.

A marker is displayed for each association point prompt.

- If the definition point of the current dimension is not associated to a geometric object, the marker appears as an X
- If the definition point is associated, the marker appears as an X inside a box.

Press Esc to end the command without losing the changes that were already specified. Use UNDO to restore the previous state of the changed dimensions.

> **Note**
> The marker disappears if user pan or zoom with a wheel mouse.

The following prompts are displayed.

Select Objects

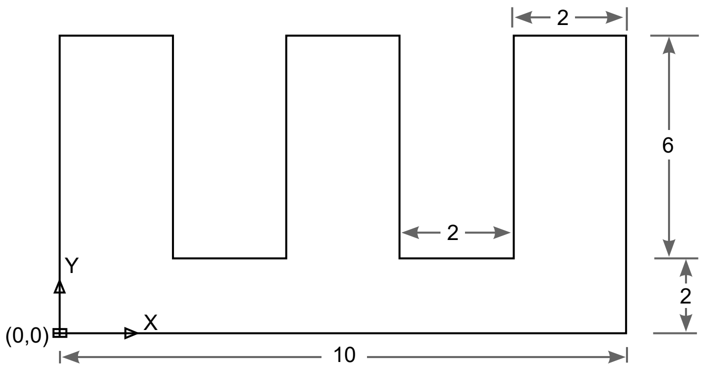
Specifies one or more non-associative dimension or leader objects to reassociate them manually to objects or points on objects.

Disassociated

Specifies all non-associative dimension or leader objects for manual re-association.

SOLVED EXAMPLES

Example 7.1: Draw the following figure using (i) Absolute coordinate Entry and (ii) Relative Coordinate entry.



(i) Absolute coordinates Entry

Following are the commands used to draw given figure using absolute coordinate entry:

Command: Line

Specify first point:0,0

Specify next point:10,0

Specify next point:10,8

Specify next point:8,8

Specify next point:8,2

Specify next point:6,2

Specify next point:6,8

Specify next point:4,8

Specify next point:4,2

Specify next point:2,2

Specify next point:2,8

Specify next point:0,8

Specify next point:0,0 or C

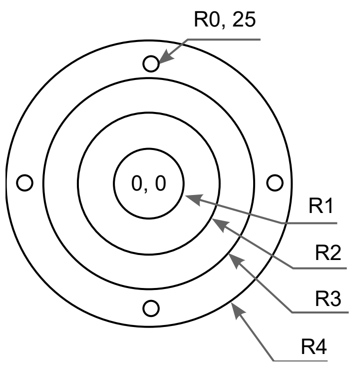
(ii) Relative Coordinate Entry

Following are the commands used to draw given figure using relative coordinate entry:

Command: Line

Specify first point: 0,0
Specify next point:@10<0
Specify next point:@8<90
Specify next point:@2<180
Specify next point:@6<270
Specify next point:@2<180
Specify next point:@6<90
Specify next point:@2<180
Specify next point:@6<270
Specify next point:@2<180
Specify next point:@6<90
Specify next point:@2<180

Example 7.2: Draw the following figure using AutoCAD, assuming the point (0, 0) is the center of the drawing.



Solution: Following are the commands used to draw given figure:

Command:C

Specify center point for circle:0,0
Specify radius of circle:1

Command:C

Specify center point for circle:0,0
Specify radius of circle:2

Command:C

Specify center point for circle:0,0
Specify radius of circle:3

Command:C

Specify center point for circle:0,0
Specify radius of circle:4

Command:C

Specify center point for circle:0,3.5
Specify radius of circle:0.25

Command:C

Specify center point for circle:3.5,0
Specify radius of circle:0.25

Command:C

Specify center point for circle:-3.5,0
Specify radius of circle:0.25

Command:C

Specify center point for circle:0,-3.5
Specify radius of circle:0.25

UNIT SUMMARY

- The 2D Drafting & Annotations workspace with the toolbar ribbon at the top of the AutoCAD window is used to create 2D drawings.
- There two views in AutoCAD: a model space and a paper space. The drawing should always be defined (outlined) using the model space, and the dimensions that are added later should be represented using the paper space.
- The customize user interface (CUI) in AutoCAD architecture allows tailoring the drawing environment to suit the user needs.
- Drawing limits are used to set the boundaries of the drawing. The drawing boundaries are usually set to match the size of a sheet of drawing paper.
- Geometric constraints are used to control the relationships of objects in respect to each other.
- The Object Snaps (OSNAPS) are drawing aids which are used in conjunction with other commands to help draw accurately. OSNAPS allow users to snap onto a specific object location when picking a point.
- Ortho is short for orthogonal, which means either vertical or horizontal. The ortho mode in AutoCAD is used to restrict cursor movement to specific directions. It allows the cursor movement only in the vertical and horizontal direction.
- The draw commands can be used to create basic objects such as lines and circles. Most AutoCAD drawings are composed purely and simply from these basic components.
- Modify commands are used to perform editing operations such as move, copy and trim on the objects in a drawing.
- The appearance of dimension can be control by setting up dimension styles or by editing individual dimensions in special cases.

EXERCISES

7.1 Multiple Choice Questions

1. How many UNIT options available in AutoCAD?
   (a) 4
   (b) 5
   (c) 7
   (d) 6
2. Which mode allows the user to draw 90^o straight lines?
   (a) OSNAP
   (b) ORTHO
   (c) LINEAR
   (d) GRID
3. A user can adjust the drawing __________ to control the size of the drawing area.
   (a) Units
   (b) Limits
   (c) Snap
   (d) None of the above
4. The primary difference between the Model tab and the Layout tab(s) is ____.
   (a) The Model tab is used for drawing in 3D and a layout is used for drawing in 2D.
   (b) The Model tab is used create drawing and a layout tab represents the sheet to print on.
   (c) The color of the background.
   (d) The Model tab displays the drawing copying from and the layout tab is used to lay out the new drawing.
5. In AutoCAD object snap is used to pick accurately _____________ of a line.
   (a) End point
   (b) Mid point
   (c) Nearest point
   (d) All of the above
6. In the AutoCAD a circle can be drawn by:
   (a) Center, radius
   (b) Center, diameter
   (c) 3 points
   (d) All of the above
7. In which Auto CAD tool bar do you find the "Grid & Snap" icon?
   (a) Draw toolbar
   (b) Modify toolbar
   (c) Status bar
   (d) Object properties toolbar

8. How long will line from 1, 5 to @5<10 be?
   (a) One unit (b) Four units
   (c) Five units (d) Ten units

9. Which of the following is the default coordinate system?
   (a) User Coordinate System
   (b) World Coordinate System
   (c) Screen Coordinate System
   (d) None of the above

10. If a line is drawn between points 1,5 and -3,5 its absolute length is
   (a) Three unit
   (b) Four units
   (c) Five units
   (d) Insufficient data

Answers Keys (Exercise: 7.1)

1. (b), 2. (b), 3. (b), 4. (b), 5. (d), 6. (d), 7. (c), 8. (c), 9. (b), 10. (b)

7.2 Short and Long Answer Type Questions

Category I

1. What are the advantages and disadvantages of using CAD systems to create engineering drawings?
2. How do the GRID and SNAP options assist in sketching?
3. List and describe the different coordinate entry methods available in AutoCAD?
4. How can you build a user interface using AutoCAD?
5. How to set unit of drawing in AutoCAD.
6. Differentiate GRID and SNAP command used in AutoCAD.
7. Name the difference options available in OSNAP command.
8. What is the use of ORTHO command in AutoCAD?
9. How many ways a circle can be draw in AutoCAD?
10. How many geometric constraint options available in AutoCAD?
11. What is modify commands in AutoCAD?
12. Differentiate COPY and MOVE command used in AutoCAD.
13. What is the use of OFFSET command in AutoCAD.
14. Differentiate FILLET and CHAMFER command used in AutoCAD
15. Explain Trim command used in autoCAD.

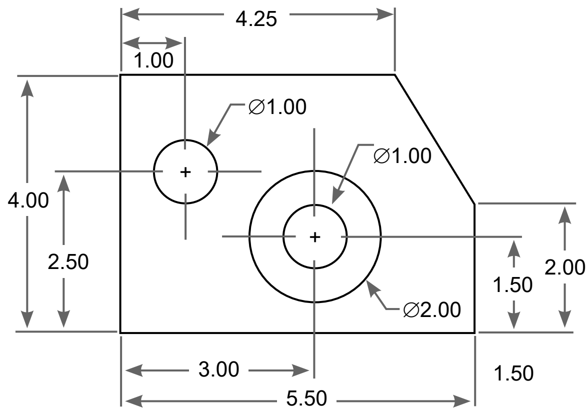
Category II

16. Describe the usefulness of Grid and Snap commands to create a drawing.
17. Customize user interface to draw a circular disk of diameter 100 mm.
18. Write the commands to draw a circle of radius 50 mm using various methods.

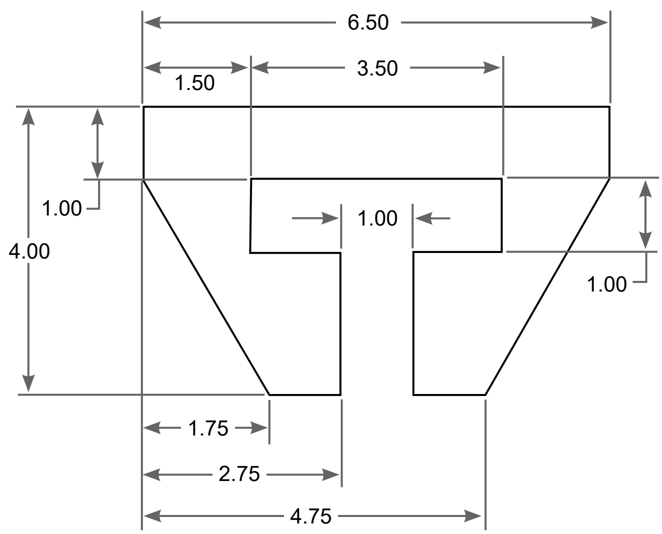
19. Explain any five geometric constraint available in AutoCAD.
20. Explain the ways of start a drawing.

7.3 Practical Exercises

1. Draw the following figure using CAD Software:



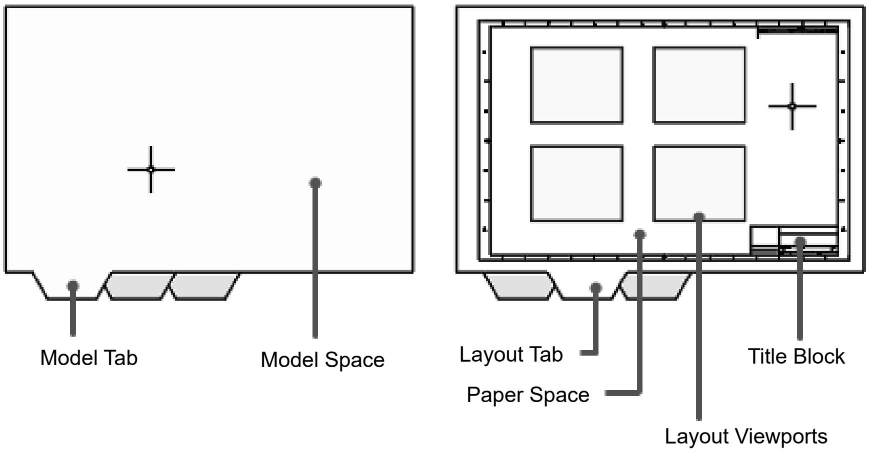
2. Draw following figure using CAD software:



KNOW MORE

Working with Layouts:

A *layout* is a 2D working environment for creating drawing sheets.The area within a layout is called *paper space*, where user can add a title block, display scaled views of model space within *layout viewports*, and create tables, schedules, notes, and dimensions for your drawing.Users can access one or more layouts from the tabs located at the bottom-left corner of the drawing area to the right of the Model tab. Users can use multiple layout tabs to display details of the various components of the model at several scales and on different sheet sizes.



There are several ways in which users can add new layouts or copy existing layouts.

- Use the LAYOUT command
- Right-click a layout tab
- Step through the Create Layout wizard
- Use DesignCenter

Each layout stores its own *page setup* which controls the appearance and format for displaying and printing each layout. For example, User would use the page setup to specify the sheet size and orientation. The Page Setup Manager is accessible from the PAGESETUP command, the Application menu, and the ribbon.

DESIGN PROJECT/ACTIVITIES

Practice and apply CAD software for 2D drawing of various surrounding objects.

INTERESTING FACTS

- AutoCAD 2.5, released in June 1986, was the first version in the US and Canada to require a hardware lock. Responding to user complaints, Autodesk sent all customers AutoCAD 2.52, which removed the lock.
- AutoCAD 2009 marked a major milestone with the introduction of the ribbon interface, the InfoCenter, Quick Properties, the ViewCube and Steering wheels, the Action Recorder and a modeless layer Manager.

INQUISITIVENESS AND CURIOSITY TOPICS

AutoCAD Mobile App

The AutoCAD mobile app is a fully enabled 2D drawing and drafting tool that lets you create, edit, and share your DWG files wherever you go. AutoCAD mobile app bridges the gap between office and work site. Download the latest drawings while in the field. Make edits and add precise measurements with a Bluetooth enabled DISTO device and your updated drawing is automatically synced across devices.

APPLICATIONS (REAL LIFE / INDUSTRIAL)

A lot of industries use AutoCAD for various purposes. Some of the main industries where AutoCAD finds its applications are as follows: -

- Architecture
- Civil engineering
- Interior design
- Automobile Industry
- Aerospace Industry
- Mechanical engineering
- Fashion designing

CASE STUDY

Virtual Reality

Virtual Reality (VR) is the use of computer technology to create a simulated environment. Unlike traditional user interfaces, VR places the user inside an experience. Instead of viewing a screen in front of them, users are immersed and able to interact with 3D worlds. By simulating as many senses as possible, such as vision, hearing, touch, even smell, the computer is transformed into a gatekeeper to this artificial world. The ability to create realistic virtual prototypes and models reduces the need for prototyping, helping to reduce the cost of production. In turn, clients no longer need to wait for a physical model to be produced before they can see—and thus approve—a design. There is therefore huge potential for VR in CAD for product design, architecture, and engineering, amongst a range of other fields.

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

- https://knowledge.autodesk.com/support/autocad/learn-explore
- https://www.cadtraininginstitute.com/essential-modify-panel-commands-in-autocad/

8 Annotation Layers and 3D Modelling

UNIT SPECIFIC

This unit covering concept of creating annotation and tolerance; Creating, modifying and customizing of layers; Computer-aided design (CAD) software modeling of parts and assemblies. Parametric and non-parametric solid modeling, surface, and wireframe models; 2-D documentation and Plotting/Printing.

RATIONALE

A solid model is a more complete representation of a surface (or wireframe model). A solid model communicates both geometric and topological information, this represents a solid unambiguously. Geometric information addresses shapes, sizes and positioning in a 3D CAD environment or coordinate system.

PREREQUISITE

Knowledge of creating drawings using CAD software.

UNIT OUTCOMES

Upon successful completion of this module, the student will be able to:

- U8-O1: Create dimensional annotation.
- U8-O2: Create layers and customization.
- U8-O3: Understand solid modeling.
- U8-O4: Create 3D models using CAD software.
- U8-O5: Plot drawings and documents.

Mapping of the Unit Outcomes with the Course Outcomes.

Unit-8 Outcomes	Expected Mapping With Course Outcomes *(1 – Weak Correlation; 2 – Medium Correlation; 3 – Strong Correlation)*				
	CO-1	**CO-2**	**CO-3**	**CO-4**	**CO-5**
U8-O1	-	-	3	3	-
U8-O2	-	-	3	3	-
U8-O3	-	-	3	3	2
U8-O4	3	-	3	3	-
U8-O5	-	-	3	3	-

8.1 ANNOTATION

Annotation put written information on the design making it easier to vision and to elucidate. Annotation objects includes dimensions, notes, and other types of explanatory symbols or objects commonly used to add information to the drawing. Annotation objects provide information about a feature, such as the length of a wall, the diameter of a fastener, or a detail callout.

In AutoCAD, a few of the many ways to annotate a draft are to use DIMENSIONS pallet tools and the TEXT pallet tools.

8.1.1 TEXT Pallet

This pallet provides tools that help user to write text in the drawing area. A text after creation will be just behaving like an ordinary object in AutoCAD. User will be able to move it; to duplicate it or do any other things with an image. Figure 8.1 shows the Text pallet window:

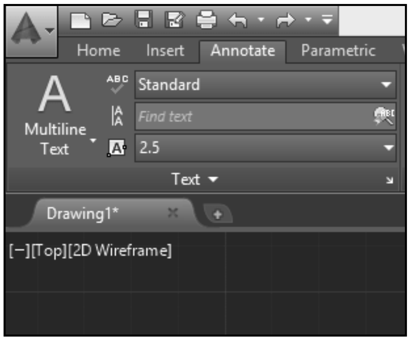


Figure 8.1: Text Pallet Window

8.1.2 Multiline TEXT (MTEXT command)

This command will simply allow user to create a multiline text object in AutoCAD as the user would, if user are using Microsoft word. To use it, user have to first specify the area in the drawing window where user would like the text to appear, then user will be able to enter the text. The following figure 8.2 shows a result of a text created using MTEXT

Figure 8.2: MTEXT Command

After creating a text, user will be able to control its style and formatting, and will be able to create paragraphs, insert symbols, break the text into columns and more.

8.1.3 Single Line (TEXT command)

Contrary to the latter, TEXT allows users to create a single line text object in AutoCAD. The major difference between the TEXT command and the MTEXT command is that while creating a text object with the TEXT command, each time user press the ENTER key, user are creating a piece of text not in the same entity with the previous one. Both can be selected separately and move independently one from another. Figure 8.3 shows the TEXT command.

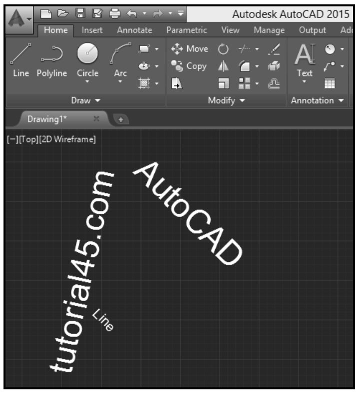
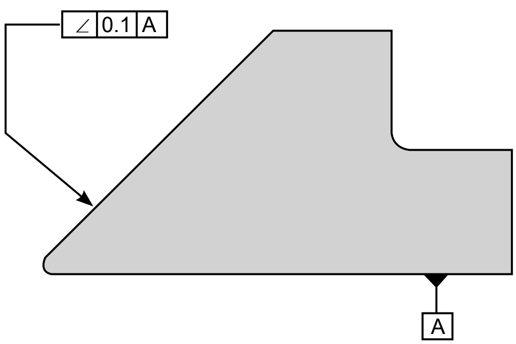


Figure 8.3: Single Line TEXT Command

8.2 TOLERANCE IN AUTOCAD

Geometric tolerances show acceptable deviations of form, profile, orientation, location, and run-out. Feature control frames can be created with leader lines using TOLERANCE, LEADER, or QLEADER.



For adding tolerance in AutoCAD dimensions, user can create a new dimension style with tolerance values and settings. This dimension style containing tolerance values can be applied wherever tolerances are required.

To make a dimension style with tolerances, type “D” or “DIMSTYLE” on the command line and press Enter.

The Dimension Style Manger window will pop up.

Click on the New button from this window, give new dimension style a name and click on the Continue button.

A new Dimension Style window will pop up.

Select the Tolerances tab from this window.

User will notice that all settings related to tolerances can be found on this tab.

Let's assume that user want to create a dimension with a "Deviation" tolerance format as shown in figure 8.4.

Figure 8.4: Deviation Type Tolerance

This type of tolerance is called deviation type because the upper and lower limits of tolerance are different. To create this tolerance type, select Deviation from the Method drop-down menu of the Tolerance format panel. Select a precision of 0.00 and specify the upper and lower limits of tolerance in the specified fields. Keep all settings unchanged and click on OK to apply these settings. See Figure 8.5 for reference.

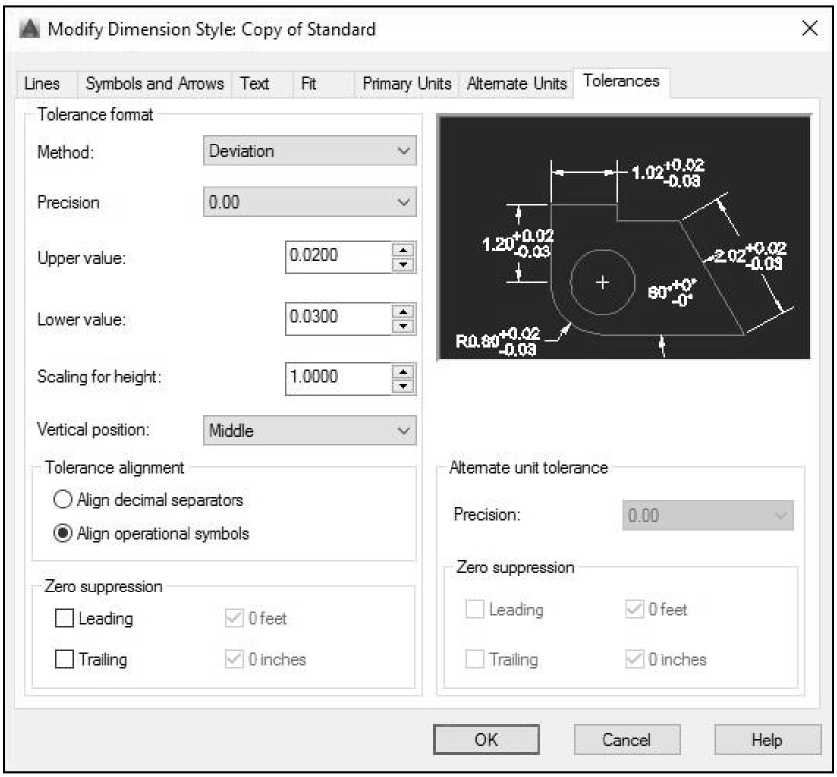


Figure 8.5: Tolerance Format Panel

8.3 LAYERS

Layers are the primary method for organizing the objects in a drawing by function or purpose. Layers can reduce the visual complexity of a drawing and improve display performance by hiding information that are not required to see at the moment.

Before start drawing, create a set of layers that are useful to work. In a house plan, user might create layers for the foundation, floor plan, doors, fixtures, electrical, and so on.

8.3.1 Create a Layer

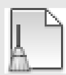
1. In the Layer Properties Manager, click New Layer. 
   A layer name is added to the layer list.
2. Enter a new layer name by typing over the highlighted layer name.
   - Layer names can be up to 255 characters long (double-byte or alphanumeric), and include letters, numbers, spaces, and several special characters.
   - Layer names cannot include the following characters: < > / \ " : ; ? * | = '
3. For complex drawings with many layers, enter descriptive text in the Description column.
4. Specify the settings and default properties of the new layer by clicking in each column.

8.3.2 Rename a Layer

1. In the Layer Properties Manager, click to select a layer.
2. Click the layer name or press F2.
3. Enter a new name.

8.3.3 Remove a Layer

1. In the Layer Properties Manager, click to select a layer.
2. Click Delete Layer. 

> **Note**
> To remove all unused layers, use PURGE. 

8.3.4 Set the Current Layer

1. In the Layer Properties Manager, click to select a layer.
2. Click Set Current. 

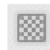
8.3.5 Change the Properties Assigned to Layers

1. If user want to change multiple layers, use one of the following methods in the Layer Properties Manager:
   - Press and hold Ctrl, and choose several layer names.
   - Press and hold Shift, and choose the first and last layers in a range.
   - Right-click, and click Show Filters in Layer List. Choose a layer filter from the list of layers.
2. Click the current setting in the column that user want to change.
   The dialog box for that property displays.

3. Choose the setting that user want to use.

   When changing layer properties:

   - If the linetype user want is not displayed, click Load and use one of the following methods:
   - In the Load or Reload Linetypes dialog box, choose the linetypes to load.
   - In the Load or Reload Linetypes dialog box, click File to open an additional linetype definition (LIN) file. Choose the linetypes to load and click OK.
   - Lineweights are not displayed automatically. If user want to display or hide lineweights, click Show/Hide Lineweight on the status bar. 

     If no change is visible, it's probably due to a combination of the thickness of the line compared to the display resolution of monitor.
   - Transparency is not displayed automatically. If user want to display or hide the transparency of objects, click Show/Hide Transparency on the status bar. 

8.3.6 Layer Properties Manager

User can add, delete, and rename layers, change their properties, set property overrides in layout viewports, and add layer descriptions as shown in figure 8.6.

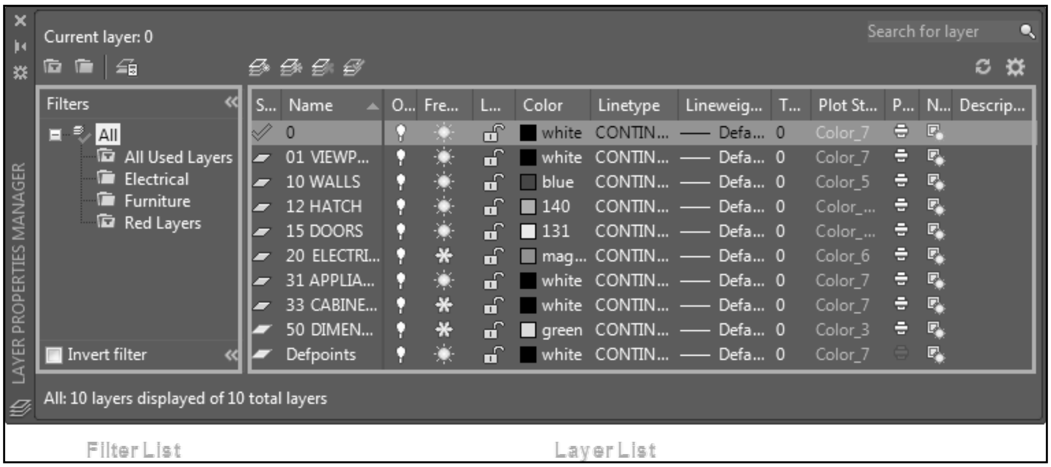
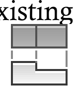


Figure 8.6: Layer Properties Manager

8.4 ORTHOGRAPHIC PROJECTION VIEWS

Followings are the steps used to create orthogonal views from an existing model:

1. Click Layout tab → Create View panel → Projected View. 
2. Click the drawing view to use as the parent view. A preview of a projected view appears at the cursor.
3. Move the preview to the desired location and click to place the view.
4. Repeat step 3 until all the required projected view are created.
5. Press ENTER.

To specify the projection angle for drawing view

User can set up the defaults for model documentation drawing views to use either the first angle or third angle projection.

1. Click Layout tab → Styles and Standards panel → Dialog box launcher. 
2. In the Drafting Standards dialog box, in the Projection type section, click the desired projection angle.

> **Note**
> The settings specify have no effect on any existing drawing views. They apply to the next drawing view create.

8.5 SECTION VIEWS

When users create a section view, it uses the settings of the current section view style. User can also apply a different section view style to it after the section view is created.

1. In the drawing area, select the section line corresponding to the section view to which user want to apply the style.
2. Click View tab → Palettes panel → Properties. 
3. In the Properties palette, under Annotation, in the Style list, select the section view style to apply.

   The section view style is applied to the section line and the corresponding section view.

Section View Style Manager

Section view styles control the appearance of model documentation section views. The style specifies the formatting of the section line, view label and hatching as shown in figure 8.7.

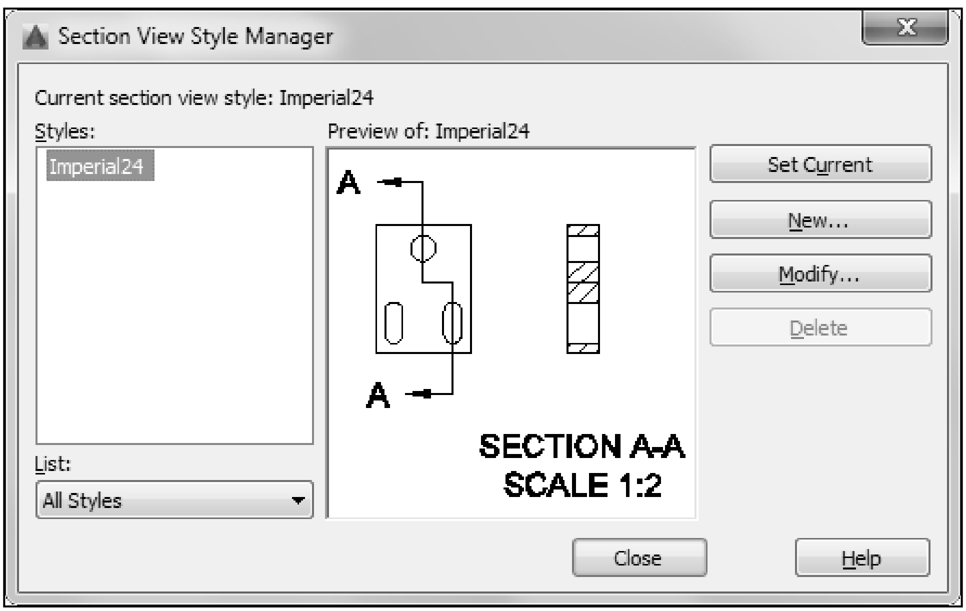


Figure 8.7: Section style manager

8.6 PLOT COMMAND

1. Click Output tab → Plot panel → Plot.  Find
2. Select a plotter.
3. Select paper size, plot area, plot scale, orientation and other options.
4. For additional options, click the More Options button as shown in figure 8.8.

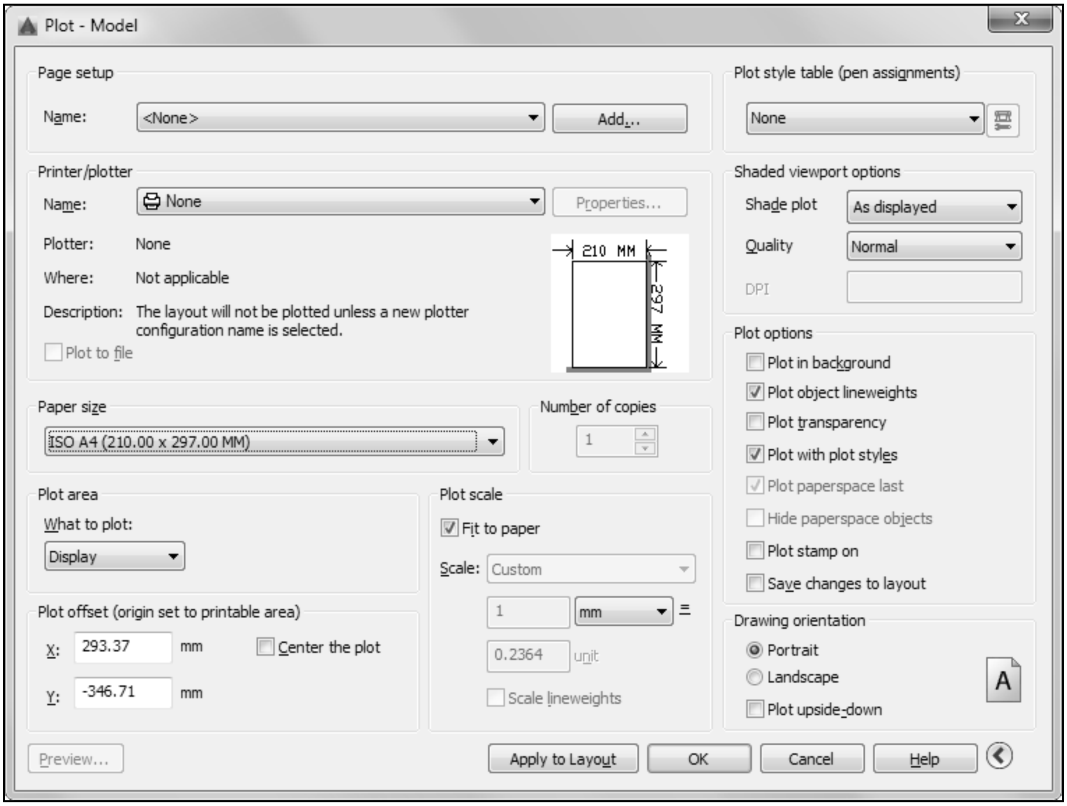
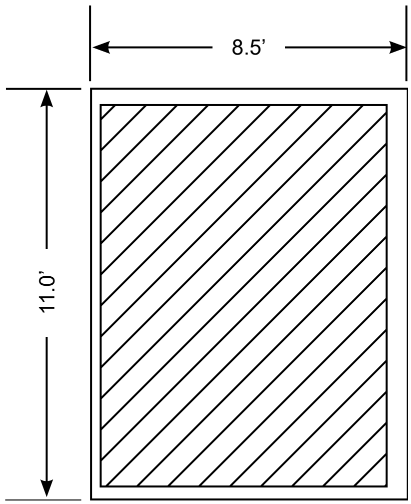


Figure 8.8: Plot Panel AutoCAD

List of Options: The following options are displayed.

- **Page Setup :**Displays a list of any named and saved page setups in the drawing. User can base the current page setup on a named page setup saved in the drawing, or user can create a new named page setup based on the current settings in the Plot dialog box by clicking Add.
- **Name :**Displays the name of the current page setup.
- **Add:** Displays the Add Page Setup dialog box, in which user can save the current settings in the Plot dialog box to a named page setup. User can modify this page setup through the Page Setup Manager.
- **Printer/Plotter :**Specifies a configured plotting device to use when plotting layouts. If the selected plotter doesn't support the layout's selected paper size, a warning is displayed and user can select the plotter's default paper size or a custom paper size.

- **Plotter:** Displays the plot device specified in the currently selected page setup.
- **Where:** Displays the physical location of the output device specified in the currently selected page setup.
- **Description:** Displays descriptive text about the output device specified in the currently selected page setup. User can edit this text in the Plotter Configuration Editor.
- **Plot to File:**Plots output to a file rather than to a plotter or printer. The default location for plot files is specified in the Options dialog box, Plot and Publish tab, under Default Location for Plot-to-File Operations.
- **PDF Options:** Displays the PDF Options dialog box, which gives user the ability to optimize a PDF file for the specific purpose user creating it.
- **Partial Preview:** Shows an accurate representation of the effective plot area relative to the paper size and printable area. The tooltip displays the paper size and printable area.



- **Paper Size:** Displays standard paper sizes that are available for the selected plotting device. If no plotter is selected, the full standard paper size list is displayed and available for selection. The actual printable area of the page, which is determined by the selected plotting device and paper size, is indicated in the layout by a dashed line. If plotting a raster image, such as a BMP or TIFF file, the size of the plot is specified in pixels, not in inches or millimeters.
- **Number of Copies:** Specifies the number of copies to plot. This option is not available when user plot to file.
- **Plot Area:** Specifies the portion of the drawing to be plotted. Under What to Plot, user can select an area of the drawing to be plotted.
- **Layout/Limits:** When plotting a layout, plots everything within the printable area of the specified paper size, with the origin calculated from 0, 0 in the layout. When plotting from the Model tab, plots the entire drawing area that is defined by the grid limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option.

- **Extents:** Plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted. The drawing may be regenerated to recalculate the extents before plotting.
- **Display:** Plots the view in the current viewport in the selected Model tab or the current paper space view in the layout.
- **View:** Plots a view that was previously saved with the VIEW command. User can select a named view from the list. If there are no saved views in the drawing, this option is unavailable.
- **Window:** Plots any portion of the drawing that user specify. When user select Window, the Window button becomes available. Click the Window button to use the pointing device to specify the two corners of the area to be plotted, or enter coordinate values.
- **Plot Offset:** Specifies an offset of the plot area relative to the lower-left corner of the printable area or to the edge of the paper, depending on the setting made in the Specify Plot Offset Relative To option (Options dialog box, Plot and Publish tab). The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.
- **Center the Plot:** Automatically calculates the *X* and *Y* offset values to center the plot on the paper. This option is not available when Plot Area is set to Layout.

  X: Specifies the plot origin in the *X* direction relative to the setting of the Plot Offset Definition option.

  Y: Specifies the plot origin in the *Y* direction relative to the setting of the Plot Offset Definition option.
- **Plot Scale:** Controls the relative size of drawing units to plotted units. The default scale setting is 1:1 when plotting a layout. The default setting is Fit to Paper when plotting from the Model tab.
- **Fit to Paper:** Scales the plot to fit within the selected paper size and displays the custom scale factor in the Scale, Inch =, and Units boxes.
- **Scale:** Defines the exact scale for the plot. *Custom* defines a user-defined scale. User can create a custom scale by entering the number of inches (or millimeters) equal to the number of drawing units.
- **Units:** Specifies the number of units equal to the specified number of inches, millimeters, or pixels.
- **Scale Lineweights:** Scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of plotted objects and are plotted with the linewidth size regardless of the plot scale.
- **Preview:** Displays the drawing as it will appear when plotted by starting the PREVIEW command. To exit the preview and return to the Plot dialog box, press ESC, press ENTER, or right-click and then click Exit on the shortcut menu.
- **Apply to Layout:** Saves the current Plot dialog box settings to the current layout.

8.7 WIREFRAME MODELING

A wireframe model is a skeletal description of a 3D object as shown in figure 8.9. There are no surfaces in a wireframe model; it consists only of points, lines, and curves that describe the edges of the object. With AutoCAD user can create wireframe models by positioning 2D objects anywhere in 3D space. AutoCAD also provides some 3D wireframe objects, such as 3D polylines (that can only have a CONTINUOUS linetype) and splines. Because each object that makes up a wireframe model must be independently drawn and positioned, this type of modeling can be the most time-consuming.

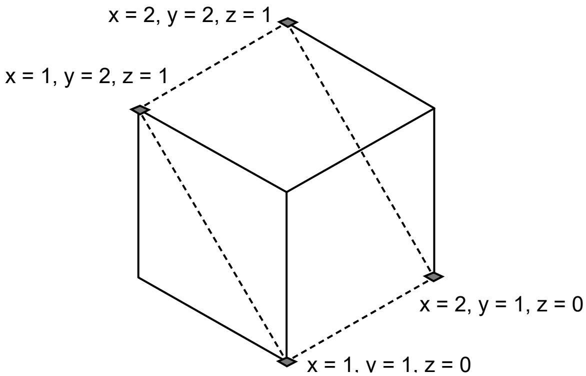


Figure 8.9: Wireframe Model

8.8 SURFACE MODELING

Surface modeling is more sophisticated than wireframe modeling in that it defines not only the edges of a 3D object, but also its surfaces. The AutoCAD surface modeler defines faceted surfaces using a polygonal mesh as shown in figure 8.10. Because the faces of the mesh are planar, the mesh can only approximate curved surfaces.

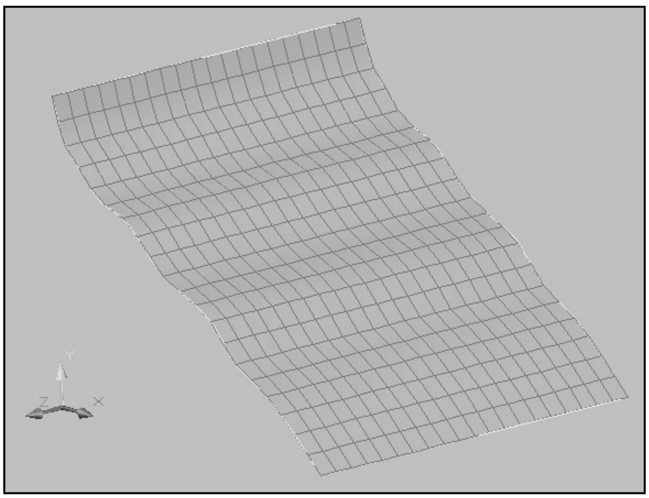


Figure 8.10: Surface Modeling

8.9 SOLID MODELING

Solid modeling is the easiest type of 3D modeling to use. With the AutoCAD solid modeler, user can make 3D objects by creating basic 3D shapes known as primitives as shown in figure 8.11:

boxes, cones, cylinders, spheres, wedges, and tori (do- nuts). User can then combine these shapes to create more complex solids by joining or subtracting them or finding their intersecting (overlapping) volume. User can also create solids by sweeping a 2D object along a path or revolving it about an axis.

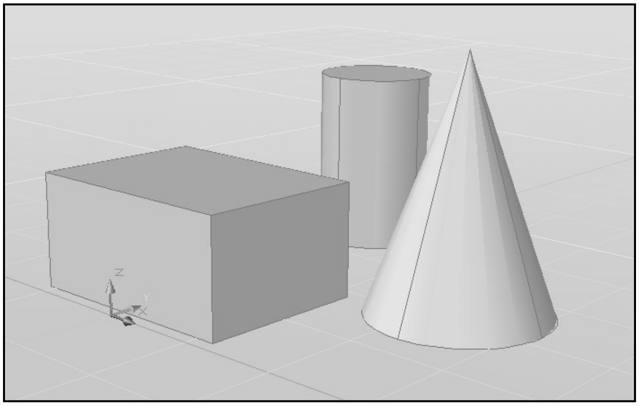


Figure 8.11: Solid primitives

Solid Modelling

8.9.1 Create Solid Object using EXTRUDE Command

Creates unique solid primitives by extruding existing two-dimensional objects as shown in figure 8.12. User can extrude multiple objects with EXTRUDE.

1. Type EXTRUDE at the command prompt.
   Command: **extrude**
   Current wire frame density: ISOLINES=4
   Select objects: **pick objects**
   Select objects: **enter**
   Specify height of extrusion or [Direction/Path/Taper angle]: **2**

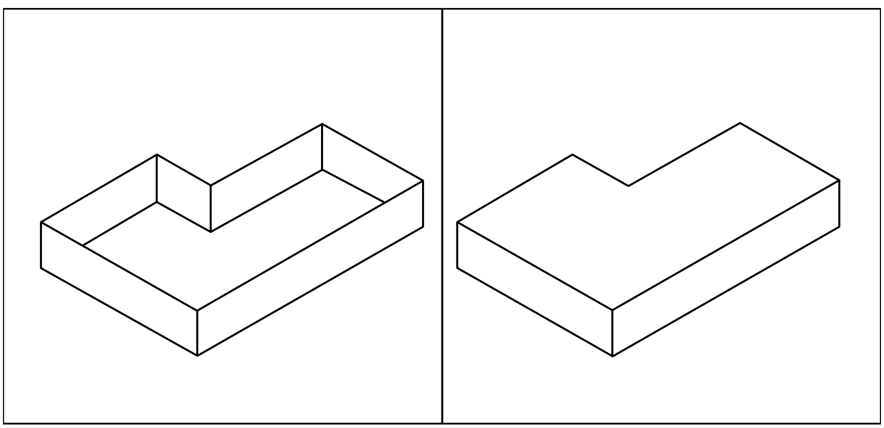


Figure 8.12: EXTRUDE in AutoCAD

8.9.2 Create Solid Objects using REVOLVE Command

Creates unique solid primitives by revolving existing two-dimensional objects about a axes as shown in figure 8.13.

1. Open a drawing with 3D objects and display in a 3D view.
2. Type REVOLVE at the command prompt.

   Command: **revolve**
   Current wire frame density: ISOLINES=4
   Select objects: **pick profile**
   Select objects: **enter**
   Specify start point for axis of revolution or define axis by [Object/X (axis)/Y (axis)]: **o**
   Select an object: **pick axis**
   Specify angle of revolution <360>: **enter**

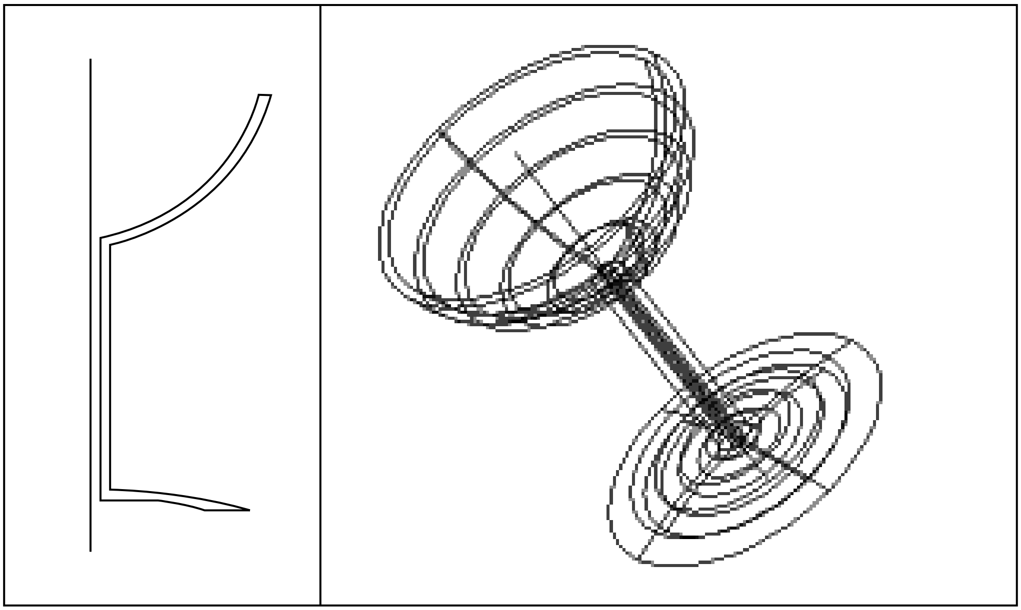


Figure 8.13: REVOLVE Command in AutoCAD

8.9.3 Create solid objects using SWEEP Command

Sweep command is used to create an solid object by sweeping an existing two-dimensional objects along a defined path as shown in figure 8.14.

1. Open a drawing with 2D objects to sweep and display in a 3D view.
2. Type SWEEP at the command prompt.

   Command: **sweep**
   Current wire frame density: ISOLINES=4
   Select objects to sweep: **pick arc**
   Select objects to sweep:

Select sweep path or [Alignment/Base point/Scale/Twist]: pick path

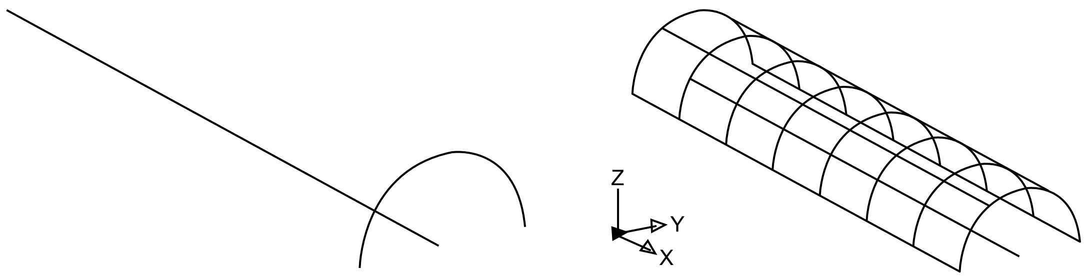
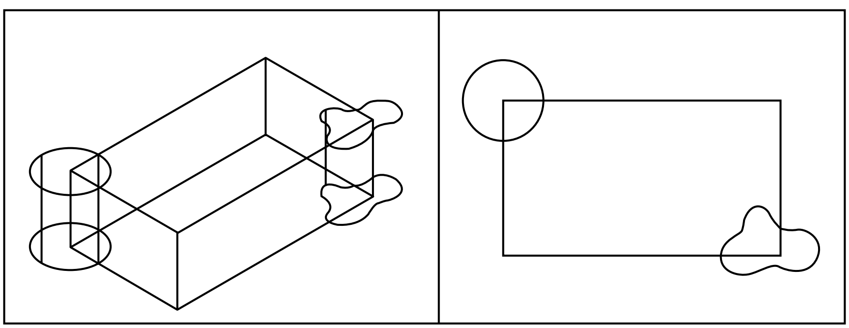


Figure 8.14: Sweep Command in AutoCAD

8.9.4 Create solid composites

Union (Figure 8.15**)**

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, Solids Editing, Union.
3. Type UNION at the command prompt.
   Command: UNION
   Select objects: pick objects to union
   Select objects: ENTER



Solid Objects Union Together

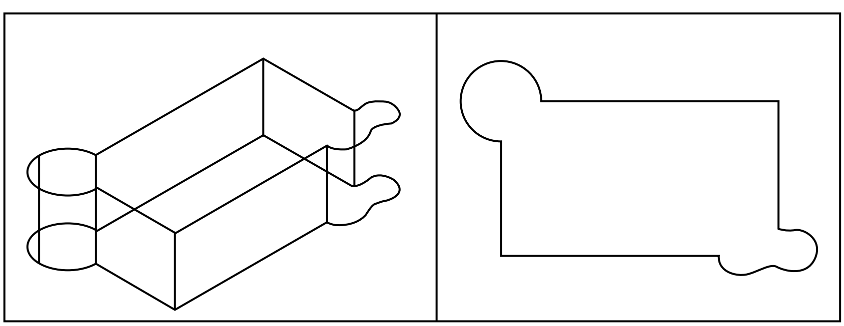
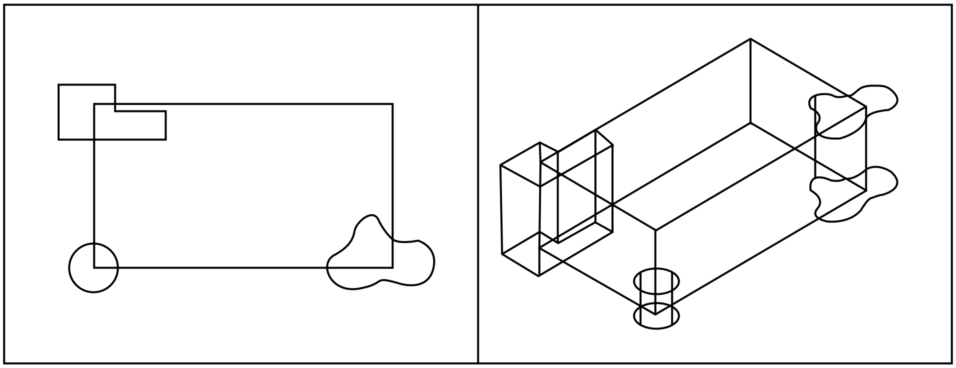


Figure 8.15: UNION of Objects

Subtract (Figure 8.16)

1. Open a drawing with 3D objects and display in a 3D view.
2. Choose Modify, Solids Editing, Subtract.
   or
3. Type SUBTRACT at the command prompt.
   Command: **SUBTRACT**
   Select solids and regions to subtract from...
   Select objects: **pick the main box**
   Select objects: **(press enter)**
   Select solids and regions to subtract... Select objects: **pick the other solids** Select objects: **enter**



Objects Subtracted from Box

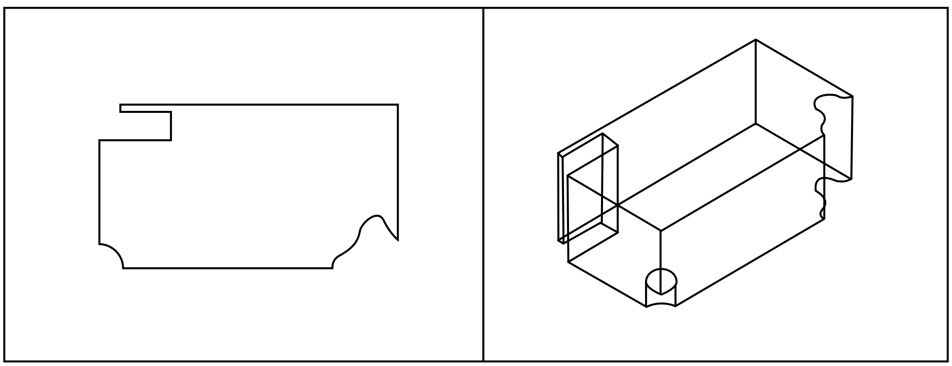
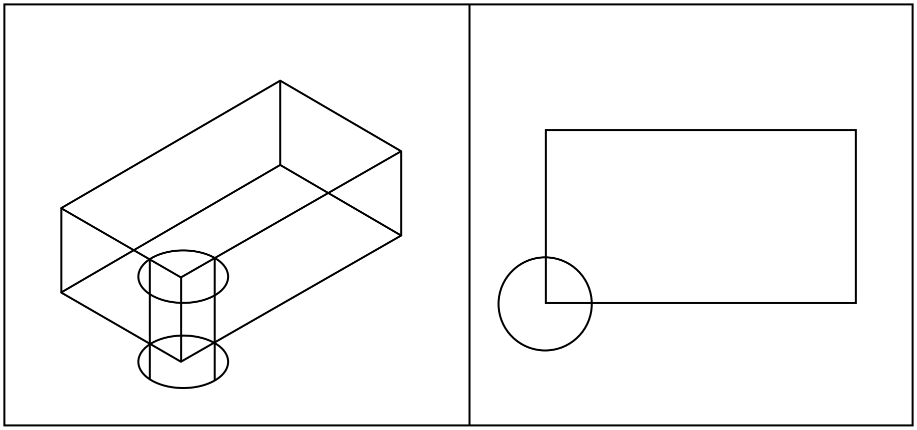


Figure 8.16: Subtraction of Objects

Intersect (Figure 8.17)

1. Choose Modify, Solids Editing, Intersect or
2. Type INTERSECT at the command prompt.
   Command: **INTERSECT**
   Select objects: pick objects

Select objects: **enter**



Object after Intersection

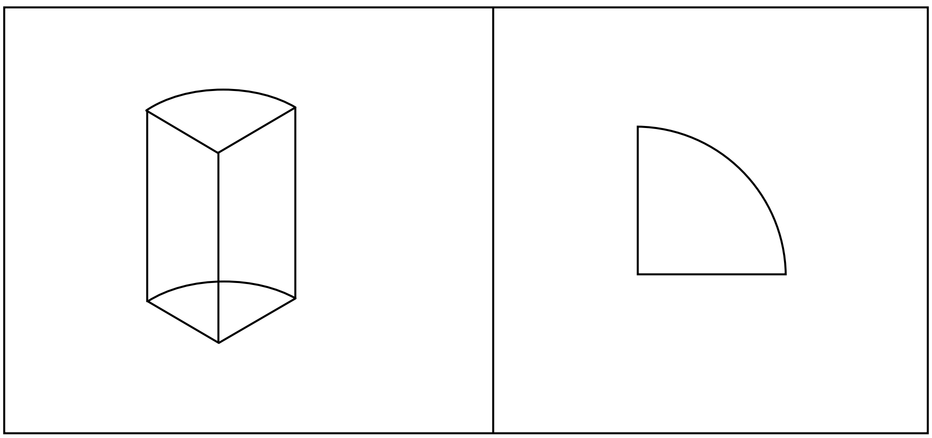


Figure 8.17: Intersection of Objects

8.10 PARAMETRIC MODELING AND NON PARAMETRIC MODELING

Parametric modeling is a computer aided design (CAD) software design tool that saves time—it eliminates the need for a design engineer to constantly redraw a design every time one of the design's dimensions change. A parametric model contains information like dimensions, constraints, and relationships between various entities like edges, sketches and features. User can easily make changes to the design, and it updates and responds to those changes.

A non-parametric model does not contain such relationships. It is essentially a "dumb model" which often happens when a CAD model is imported from another program. Dumb models can be modified, but they do not have the additional constraints and relationships to allow the update to affect other design elements.

UNIT SUMMARY

- Annotation put written information on the design making it easier to vision and to elucidate.
- Geometric tolerances show acceptable deviations of form, profile, orientation, location, and run-out.
- A wireframe model is a skeletal description of a 3D object
- Surface modeling is more sophisticated than wireframe modeling in that it defines not only the edges of a 3D object, but also its surfaces.
- Solid modeling describes object as finite , closed , regular sets of points. A solid model provides a complete characteristic and concise description of the object.
- Parametric modeling is a computer aided design (CAD) software design tool that saves time—it eliminates the need for a design engineer to constantly redraw a design every time one of the design's dimensions change.

EXERCISES

8.1 Multiple Choice Questions

1. If a layer is locked:
   (a) Details can be added to a locked layer
   (b) Details can be erased from a locked layer
   (c) Details cannot be added or erased from a locked layer
   (d) Details can be added to a locked layer but they disappear when the drawing file is saved.
2. All of the following are the example of solid primitives except the:
   (a) Sphere (b) Cone
   (c) Cylinder (d) Dome.
3. All of the following objects can be extruded except:
   (a) Arc (b) Polyline
   (c) Ellipse (d) Region
4. To create a revolved solid use the command:
   (a) Edgesurf (b) Revsurf
   (c) Revolve (d) Extrude
5. The following are some of the properties and settings that you can save in a layout, except ______________.
   (a) Plot scale
   (b) Paper size
   (c) Line weight
   (d) Drawing orientation

Answer Keys (Exercise: 8.1)

1. (c), 2. (d), 3. (a), 4. (c), 5. (d)

8.2 Short and Long Answer Type Questions

Category I

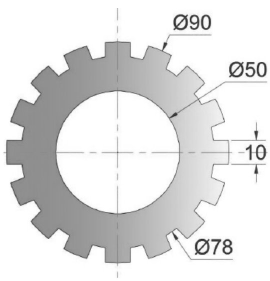
1. What is the use of annotation in engineering drawing?
2. What is the significance of layer in a drawing?
3. Explain the extrude command used in AutoCAD.
4. What do you understand by sweep command in AutoCAD?
5. What is parametric modeling?
6. Differentiate wireframe modeling and surface modeling.
7. Explain the MTEXT command.
8. How orthographic view generated in AutoCAD.

Category II

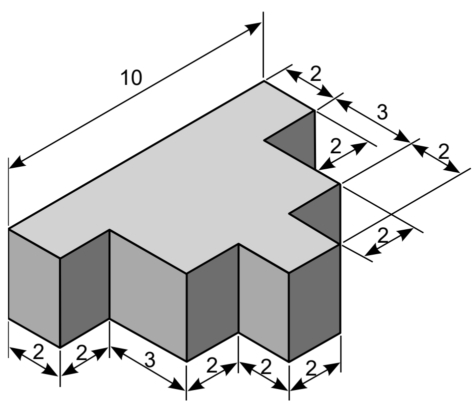
9. What is geometric tolerance and how it can be used in AutoCAD.
10. What are the different options available to set layers in AutoCAD?
11. Explain the primitives used in AutoCAD for solid modeling.
12. Define UNION, SUBTRACT and INTERSECTION command in AutoCAD.

8.3 Practical Problems

1. Draw the following objects in CAD software and convert it into solid object.



2. Create the following object using 3D software and generate orthographic drawings of the same.



KNOW MORE

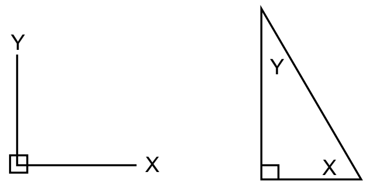
User Coordinate System (UCS)

The user coordinate system (UCS) establishes the location and orientation of a movable Cartesian coordinate system. The UCS is an essential tool for many precision operations.

The UCS defines

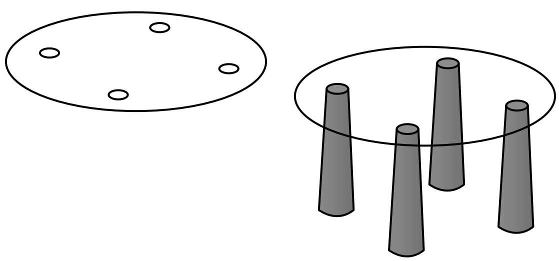
- The horizontal and vertical directions used for features like Ortho mode, polar tracking, and object snap tracking
- The alignment and angle of the grid, hatch patterns, text, and dimension objects
- The origin and orientation for coordinate entry and absolute reference angles

By default, the UCS icon appears in the lower-left corner of the drawing area for the *current* model viewport. The UCS in each paper space layout is displayed as a drawing triangle.

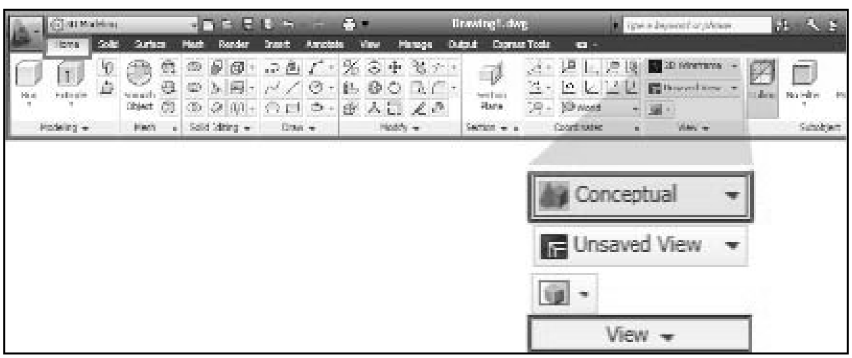


DESIGN PROJECT/ACTIVITIES

Draw a table stand with simple 3D solid primitives

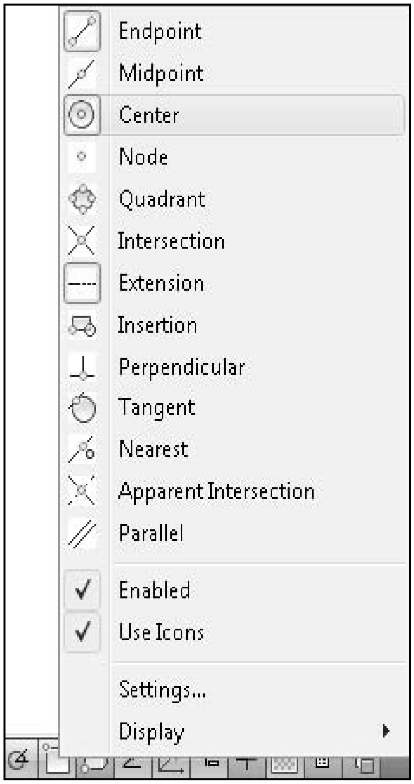


1. Click → Open.
2. In the Select File dialog box, browse to *C:\My Documents\Tutorials*. Open *create_table.dwg*.
3. On the ribbon, click Home tab → View panel → Visual Styles drop-down list → Conceptual.

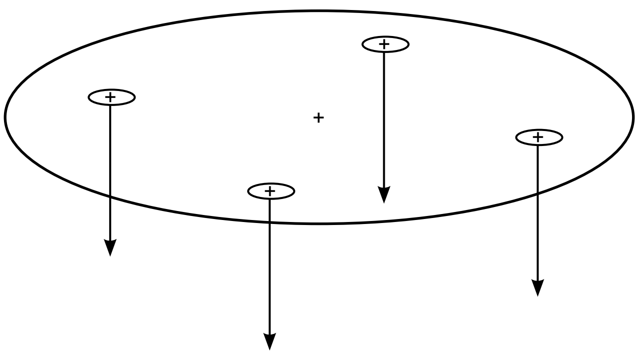


4. On the left half of the status bar, click the Object Snap button to enable object snap mode. Right-click the Object Snap button.

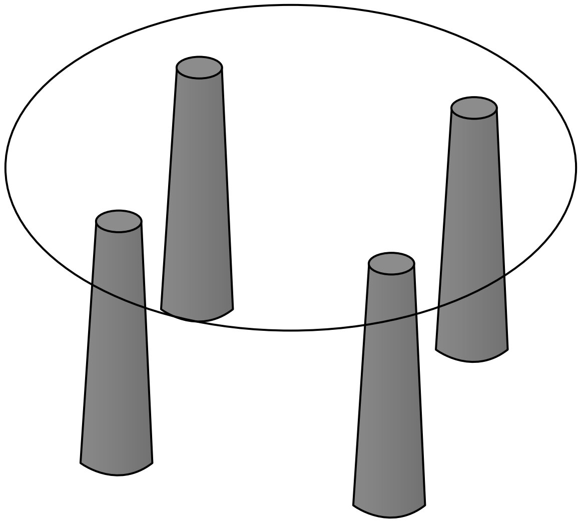
5 On the shortcut menu, click Center to turn it on. The Center option should now have a box around its icon which indicates the object snap is enabled.



6. On the ribbon, click Home tab → Modeling panel → Solid Primitives drop-down → Cone.
7. At the prompt, move the cursor over one of the smaller circles. The center point of the circle is displayed. Click the center of the circle in the drawing.



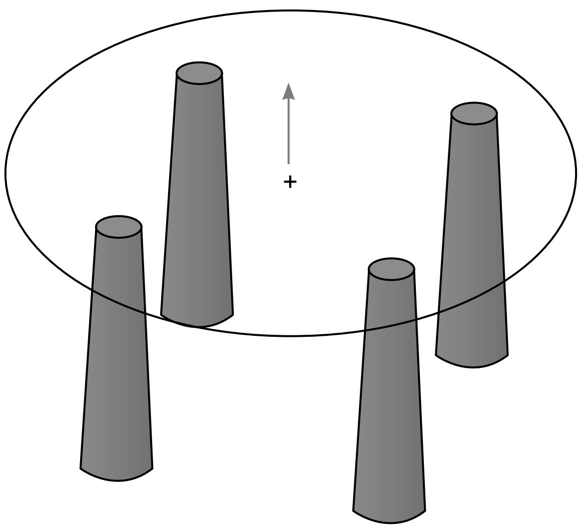
8. At the prompt, enter **T** for top radius and press Enter.
9. At the prompt, enter **0.5** for radius and press Enter.
10. At the prompt, enter **-4** for height and press Enter.
11. Repeat the process on the other smaller circles in the drawing to create four table legs.



Draw a table top with simple 3D solid primitives

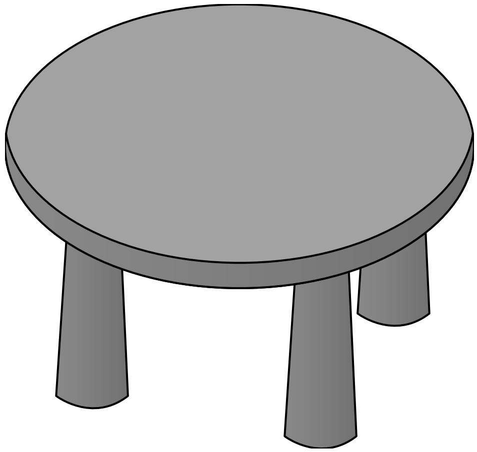
In the same drawing file, do the following:

1. On the ribbon, click Home tab → Modeling panel → Solid Primitives drop-down → Cylinder.
2. At the prompt, select the center point of the circle that has a radius of 4.



3. At the prompt, enter **4** for radius and press Enter.

4. At the prompt, enter **0.5** for height and press Enter.



INTERESTING FACTS

3D modeling software is no longer just for engineers. Aside from being able to design products, the software is now being applied to interior design, 3D printing and media, like video games and movies.

APPLICATIONS (REAL LIFE / INDUSTRIAL)

Applications of Solid Modeling

Solid modeling is used not only for creating solid models of machine parts, but also the buildings, electric circuits and even of the human beings. The solid modeling software are being used for a large variety of applications, here are some of them:

- **Engineering:** The engineering design professionals use solid modeling to see how the designed product will actually look like. The architects and civil engineers use it to use the layout of the designed building.
- **Entertainment industry:** The animation industry has been using solid modeling to create various characters and the movies out of them.
- **Medical industry:** Modern imaging scanners are being used to create the solid models of the internal parts of the body. This helps the doctors to visualize specific tissues of the body, designing various medical devices etc.

INQUISITIVENESS AND CURIOSITY TOPICS

Key Elements of a Solid Object

Solid modeling can be further defined by four differentiating factors, compared to wireframe and surface modeling:

- Complete → various points of an object within the modeling environment can be classified as inside or outside (giving a more accurate representation of the object and its surfaces)
- Valid → Vertices, faces and edges are properly connected for a complete view of the object

- Unambiguous → singular interpretation of all design aspects of the object (this means that there is certainty and clarity in the design of the object - user can see what it will actually look like in reality)
- Solid → consists of geometric and topological data (weight, shape, size, connectivity of nodes/edges/faces)

CASE STUDY

Environmental/ Sustainability/ Social/Ethical Issues

3D CAD: Features

- Create sophisticated part and assembly designs quickly and efficiently using powerful, easy-to-use software.
- Easily find and leverage existing engineering data to create new designs and speed up product development.
- Flexible 3D modeling tools cover the full range of design tasks to quickly develop product concepts.
- Verify that components can be assembled properly before going into production.
- Continuously check designs against cost targets with integrated automated manufacturing cost estimation tools.
- Create production-ready 2D drawings that communicate how design should be manufactured and assembled.
- Verify operation and performance while creating design with fully integrated simulation and analysis tools.
- Quickly and easily create powerful images and animations to communicate design intent and functionality.

SUGGESTED READINGS / VIDEO RESOURCES / LEARNING WEBSITES

1. CAD/CAD Theory and Practice By Ibrahim Zied , TMH publication.
2. https://www.usb.ac.ir/FileStaff/1365_2018-11-17-9-30-19.pdf
3. https://www.peachpit.com/articles/article.aspx?p=2982117&seqNum=13
4. https://tutorial45.com/autocad-tutorial-18-basics-of-annotation-in-autocad/

9 Projects

PROJECT - 01

OBJECTIVE

Create a simple floor plan for building: draw external walls, internal walls, and windows.

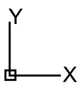
PREREQUISITES

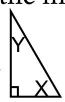
Basic AutoCAD draw and modify commands and

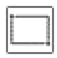
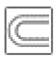
- Use **dline** to create a double line using straight line segments and arcs.
- Use **rectang** to create a rectangular polyline.

PROCEDURE

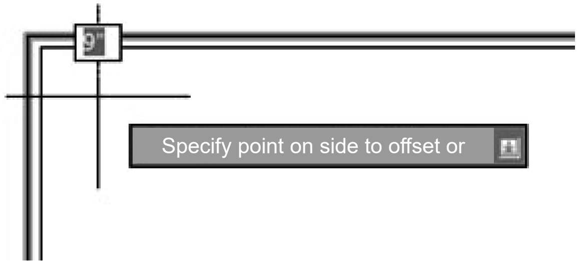
i. Draw external walls

1. **New drawing**. In Start Drawing templates, click on the *new* button in the top toolbar and select the *Tutorial i-Arch* template.
2. **Mspace.** In the new drawing, start out in the paper space. Click *Paper* in the status bar at the bottom of the screen to switch the model space. In the model space, a UCS icon 

displays instead of a triangle. 

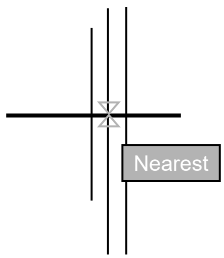
3. **Rectang.** In the ribbon, click the Home tab. In the Draw panel, click *Rectangle*.  Specify the lower left and upper right points of the rectangle with your mouse to form the outside of the exterior wall.
4. **Offset.** In the Modify panel, click *Offset*.  Specify 9" by typing **9** and pressing Enter. Select the rectangle.

5. **Specify** an inside point to create the other side of the wall. Then, hit Enter to escape the *Offset* command.

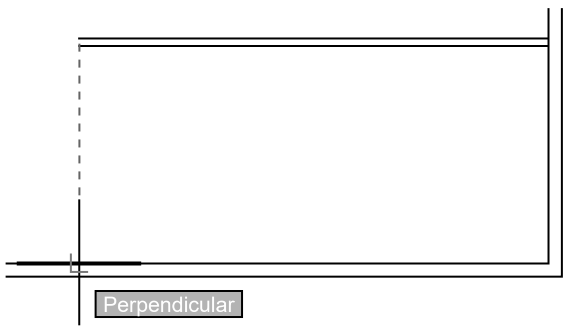


ii. Draw internal walls

1. There is no double-line tool on the ribbon, so enter DLINE and press Enter to start the command.
2. Type w and press space for the Width option. Enter 4 to specify a width of 4" for the interior walls.
3. Right-click and select *Osnap Overrides*, then select *Nearest*.
4. Click to specify a point on the inside rectangle of the wall on the East side.

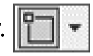


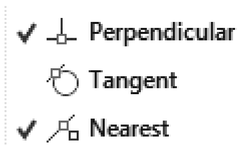
5. Click to specify a point inside the building.
6. Right-click and select *Osnap Overrides*, then select *Perpendicular*.
7. Click on the inside of the South outer wall. Press Enter to escape the dline command.



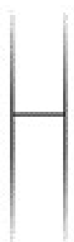
Note
The dline command is available only in AutoCAD LT. If following these steps in AutoCAD, use mline instead.

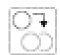
iii. Draw a simple window

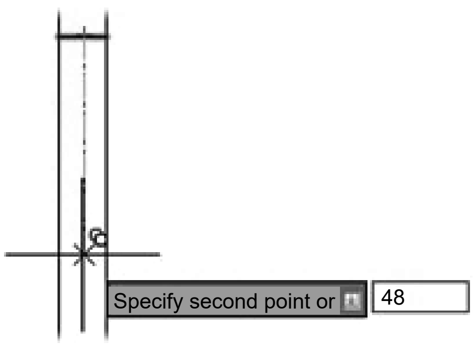
1. Osnap. Click the small down arrow for the object snap menu in the status bar.  Click on *Midpoint*, *Nearest,* and *Perpendicular* to enable these modes.



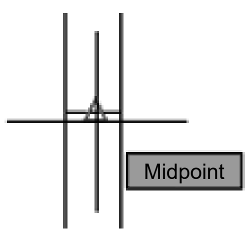
2. Line. In the ribbon, click *Line*. Draw a small 9" line through the outer wall on the East side.



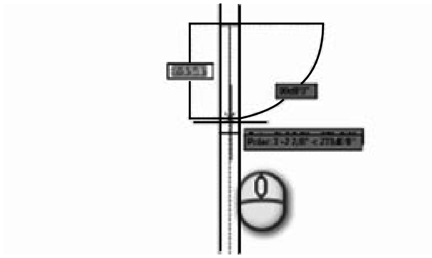
3. Copy. Select the line you just created. In the ribbon, click *Copy*.  For the base point, click on the selected line.
4. Displacement. Move your mouse to specify a direction along the wall, but do not click. Type in 48 and press Enter.



5. Glass pane. In the ribbon, click *Line*. Click to specify the *Midpoint* of one of the lines you created, then click to specify the Midpoint of the second line.

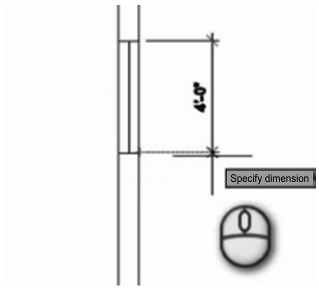


6. Press Esc to stop the line command. Your simple window is complete.



iv. Dimension the window

1. **Dimension.** In the ribbon, in the Annotation pane, click *Dimension.* 
2. **Place the dimension.** Specify both sides of the window by clicking on the outside edges with your mouse, then dragging away from the wall.



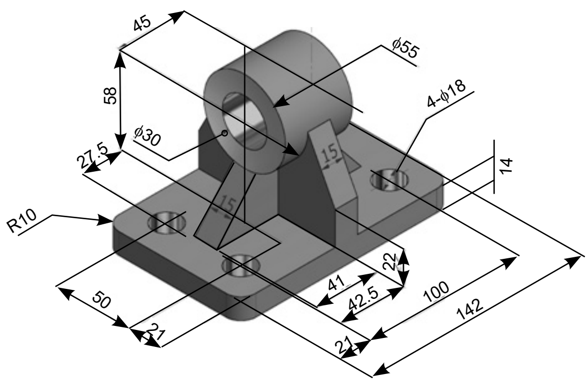
v. Conclusion

In this project students learned how to create a simple floor plan in AutoCAD...

PROJECT - 02

OBJECTIVE

Develope the following part using 3D CAD software and create 2D orthographic drawings of the same object.



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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 analyse 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 Outcomes	Expected Mapping with 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	PO-8	PO-9	PO-10	PO-11	PO-12
CO-1												
CO-2												
CO-3												
CO-4												
CO-5												

The data filled in the above table can be used for gap analysis.

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