ENGINEERING DRAWING (WELDER, FITTER, MECHANIC DIESEL)

NSQF LEVEL - 5

HANDBOOK FOR CRAFTS INSTRUCTOR TRAINING SCHEME

कौशलं बलम्
Directorate General of Training

DIRECTORATE GENERAL OF TRAINING
MINISTRY OF SKILL DEVELOPMENT & ENTREPRENEURSHIP
GOVERNMENT OF INDIA

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A Comprehensive Training Program
under Crafts Instructor Training Scheme (CITS)
for Instructors

HANDBOOK ON
TECHNICAL INSTRUCTOR TRAINING MODULES

अतुल कुमार तिवारी, I.A.S.
सचिव
ATUL KUMAR TIWARI, I.A.S.
Secretary

सत्यमेव जयते

भारत सरकार
कौशल विकास एवं उद्यमिता मंत्रालय
GOVERNMENT OF INDIA
MINISTRY OF SKILL DEVELOPMENT AND ENTREPRENEURSHIP

Foreword

In today's rapidly evolving world, the role of skilled craftsmen and women is more crucial than ever. The Craft Instructor Training Scheme (CITS) stands at the forefront of this transformation, shaping the educators who will train the next generation of artisans and technicians. This book aims to provide an in-depth understanding of the subject, exploring its significance, methodologies, and impact on vocational training.

The Craft Instructor Training Scheme was established with the objective of enhancing the quality of instruction in industrial training institutes and other vocational training institutions. By equipping instructors with advanced skills and knowledge, the scheme ensures that they are well-prepared to impart high-quality training to their students. This, in turn, contributes to the creation of a highly skilled workforce capable of meeting the demands of modern industry.

The initial chapters provide the importance of specialized instructor training. Following this, detailed chapters delve into the curriculum covering advanced techniques, safety protocols, and instructional strategies. Each section is designed to offer both theoretical insights and practical applications, ensuring a well-rounded understanding of the subject.

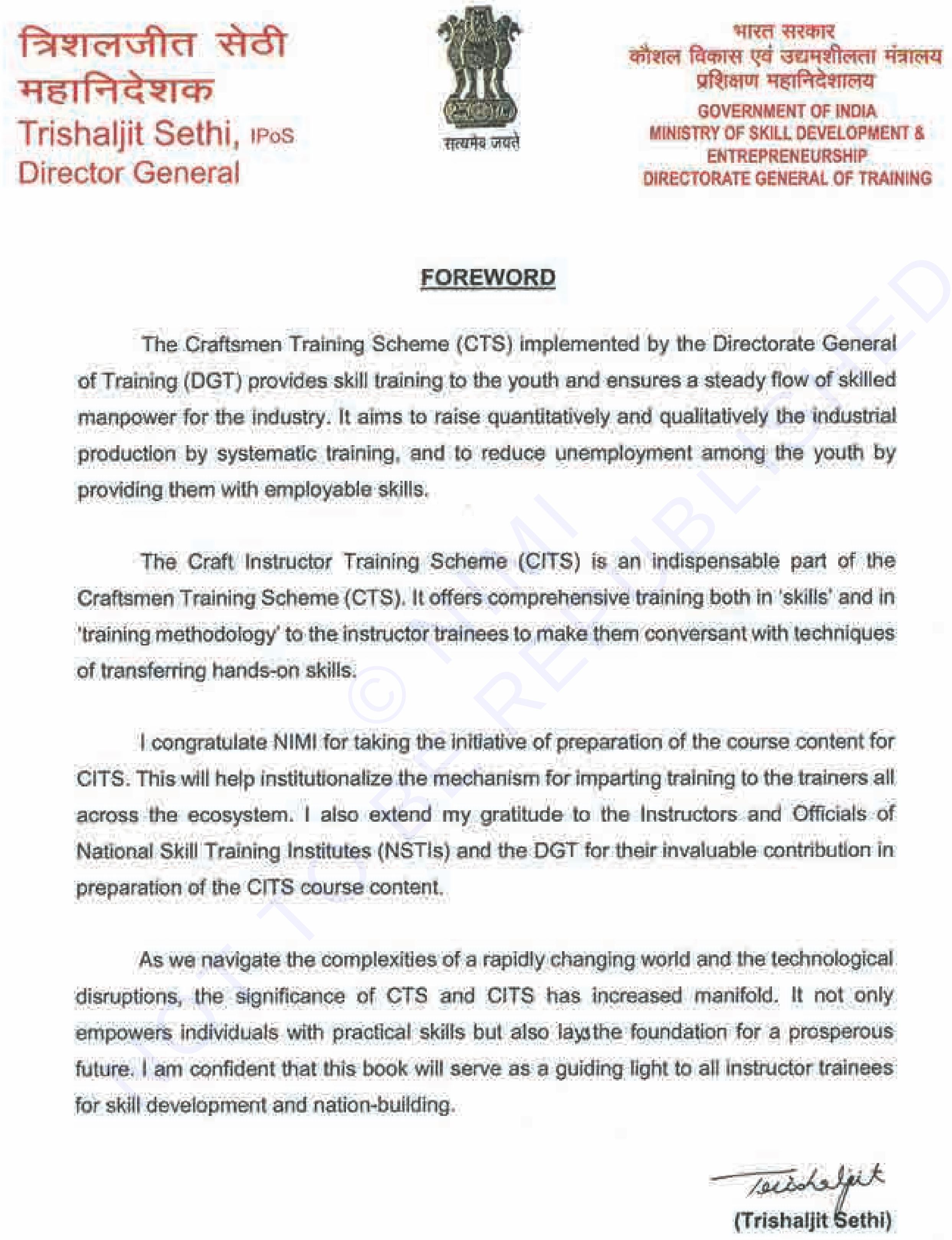
The book offers recommendations for overcoming obstacles and enhancing the effectiveness of the program, with the ultimate goal of producing highly skilled instructors capable of shaping the future workforce.

This book is intended for a diverse audience, including current and aspiring instructors, vocational training administrators, policymakers, and industry stakeholders. It serves as a valuable resource for understanding the intricacies of the subject and its pivotal role in vocational education.

I extend my heartfelt gratitude to all contributors who have shared their experiences and expertise, enriching this book with their valuable insights. Special thanks to the contribution of the development team, reviewers and NIMI that have supported this endeavor, providing essential data and resources.

It is my sincere hope that this book will inspire and guide readers in their efforts to enhance vocational training, ultimately contributing to the development of a skilled and competent workforce.

ATUL KUMAR TIWARI, I.A.S.
Secretary, MSDE



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महानिदेशक
Trishaljit Sethi, IPoS
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सत्यमेव जयते

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GOVERNMENT OF INDIA
MINISTRY OF SKILL DEVELOPMENT &
ENTREPRENEURSHIP
DIRECTORATE GENERAL OF TRAINING

FOREWORD

The Craftsmen Training Scheme (CTS) implemented by the Directorate General of Training (DGT) provides skill training to the youth and ensures a steady flow of skilled manpower for the industry. It aims to raise quantitatively and qualitatively the industrial production by systematic training, and to reduce unemployment among the youth by providing them with employable skills.

The Craft Instructor Training Scheme (CITS) is an indispensable part of the Craftsmen Training Scheme (CTS). It offers comprehensive training both in 'skills' and in 'training methodology' to the instructor trainees to make them conversant with techniques of transferring hands-on skills.

I congratulate NIMI for taking the initiative of preparation of the course content for CITS. This will help institutionalize the mechanism for imparting training to the trainers all across the ecosystem. I also extend my gratitude to the Instructors and Officials of National Skill Training Institutes (NSTIs) and the DGT for their invaluable contribution in preparation of the CITS course content.

As we navigate the complexities of a rapidly changing world and the technological disruptions, the significance of CTS and CITS has increased manifold. It not only empowers individuals with practical skills but also lays the foundation for a prosperous future. I am confident that this book will serve as a guiding light to all instructor trainees for skill development and nation-building.

Trishaljit
(Trishaljit Sethi)

PREFACE

The Craft Instructor Training Scheme is an indispensable module of the Craftsmen Training Scheme, which has been an integral part of the Indian skill development industry since its inception. This program aims to equip instructors with the necessary skills and teaching methodology to effectively transfer hands-on skills to trainees and promote a holistic learning experience. The first Craft Instructor Training Institute was established in 1948, followed by six more institutes across India in 1960. Today, these institutes, including the National Skill Training Institute (formerly Central Training Institute for Instructors), offer the CITS course, which is mandated by the Directorate General of Training (DGT).

The Craft Instructor training program is designed to develop skilled manpower for industries. The course aims to offer instructors an opportunity to improve their instructional skills, engage learners effectively, offer impactful mentoring, and make efficient use of resources, leading to a more skilled workforce in various industries. The program emphasizes collaborative and innovative approaches to teaching, resulting in high-quality course delivery. Overall, the Craft Instructor Training Scheme is a pivotal program that helps instructors grow in their careers and make a significant contribution to society. This program is essential for developing skilled manpower and promoting a robust learning environment that benefits both trainees and instructors alike.

ACKNOWLEDGEMENT

National Instructional Media Institute (NIMI) sincerely acknowledges with thanks for the co-operation and contribution extended by the following experts to bring out this Instructional material for **CITS Engineering Drawing (Welder, Fitter, Mechanic Diesel) (NSQF Level - 5)** for **Instructors**.

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NIMI records its appreciation of the Data Entry, CAD, DTP Operators for their excellent and devoted services in the process of development of this Instructional Material.

NIMI also acknowledges with thanks, the invaluable efforts rendered by all other staff who have contributed for the development of this Instructional Material.

NIMI is grateful to all others who have directly or indirectly helped in developing this IMP.

ABOUT THE TEXT BOOK

The Vocational Instructor Training Program is a comprehensive initiative designed to equip aspiring students with the necessary skills and knowledge to effectively teach in vocational education settings. This program encompasses a range of pedagogical strategies, instructional techniques, and subject-specific content tailored to the diverse vocational fields. Participants engage in coursework that covers curriculum development, assessment methods, classroom management, and the integration of industry-relevant technologies. Practical experience and hands-on training are emphasized, allowing participants to apply theoretical concepts in real-world teaching environments. Through collaborative learning experiences and mentorship opportunities, aspiring vocational instructors develop the confidence and competence to facilitate engaging and impactful learning experiences for their students. This training program aims to cultivate a new generation of educators who are not only proficient in their respective vocational fields but also adept at fostering the success and employability of their students in today’s competitive workforce.

This text book covers communication, self-management, information and communication technology, entrepreneurial and green skills. It has been developed as per the learning outcome-based curriculum.

G C Rama Murthy,
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EXERCISE 1 : Circles, Tangents and Ellipse

Circles, Tangents and Ellipse

Circle: Circle is a plane figure bounded by a curve, formed by the locus of a point which moves so that it is always at a fixed distance from a stationery point the "Centre".

Radius: The distance from the centre to any point on the circle is called the "Radius".

Diameter: The length of a straight line between two points on the curve, passing through the centre is called the "Diameter". (D: Dia or d) It is twice the radius.

Circumference: It is the linear length of the entire curve, equal to πD.

Arc: A part of the circle between any two points on the circumference or periphery is called an 'Arc'.

Chord: A straight line joining the ends of an arc is called the chord. (Longest chord of the circle is the diameter)

Segment: A part of the circle or area bound by the arc and chord is the segment of the circle.

Sector: It is the part of a circle bounded by two radii (plural of radius) meeting at an angle and an arc.

Quadrant: Part of a circle with radii making 90° with each other is a quadrant (one fourth of the circle).

Half of the circle is called as semi-circle.

Tangent: Tangent of a circle is a straight line just touching the circle at a point. It does not cut or pass through the circle when extended. The point where the tangent touches the circle is called the "point of tangency". The angle between the line joining the centre to the point of tangency and the tangent is always 90°.

Fig 1 shows all the above elements.

Concentric circles: When two or more circles (drawn) having common centre, they are called concentric circles. Ball bearing is the best example of concentric circles. (Fig 2)

Fig 1

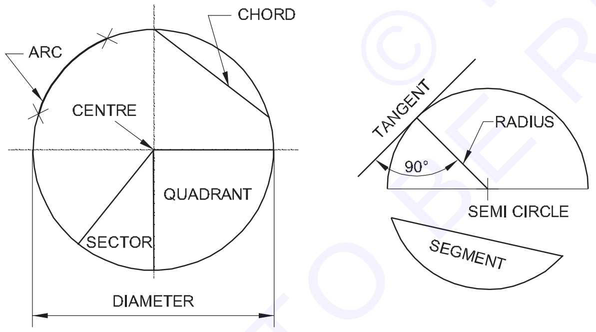
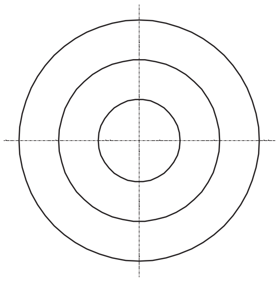


Fig 2

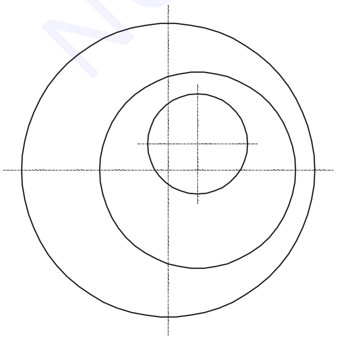


CONCENTRIC CIRCLES

Eccentric circles: Circles within a circle but with different centres are called eccentric circles. (Fig 3)

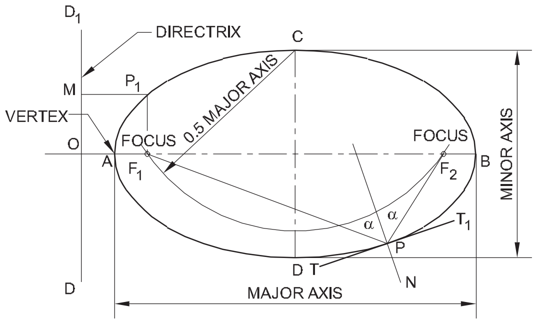
Elements of an ellipse (Fig 4)

Fig 3



ECCENTRIC CIRCLES

Fig 4



Major axis: It is the longest distance which passes through the centre, at right angle to the fixed lines called the directrix. AB is the major axis.

Minor axis: It is the maximum distance which bisects the major axis at right angle. It will be parallel to the directrix. CD is the minor axis.

Directrix: It is a straight line perpendicular to the major axis.

Focus: When an arc is drawn with C or D as centre and radius equal to half the major axis i.e $\frac{AB}{2}$, it is cut at two points F_1 and F_2 on themajor axis. F_1 and F_2 are the focal points of an ellipse F_1 or F_2 is the focus. The sum of the distances from F_1, F_2 to any point on the curve i.e., $F_1P + F_2P$ is always constant and equal to the major axis.

Focal radii: The distances from point P on the curve to the focal points F_1 and F_2 are called focal radii. Sum of the focal radii is equal to the major axis.

Eccentricity: The ratio between the distances from the vertex to focus and vertex to the directrix is called the eccentricity and is always less than one.

$AF_1/A0$ is less than one.

It can also be stated as the ratio of the distance from focus onto any point on the curve, say P_1 and the perpendicular distance of P_1 from the directrix. i.e P_1F_1/P_1M is a constant.

Vertex: The end points of the major axis on the curve are called vertex. (A, B)

Tangent and normal to an ellipse: Normal is the line bisecting the angle F_1PF_2 in Fig 4. Tangent in a line at 90° to the normal and touching the ellipse.

Directrix, axis, focus, vertex and tangent are the elements common to ellipse, parabola and hyperbola.

All ellipse can be constructed in different methods:

- Rectangle method (oblong)
- Concentric circle method
- Arcs method
- String and pins method
- Paper trammel method
- 4 centre method
- Conjugate diameters method
- Eccentricity method

Practical applications: In general, a circle in a pictorial drawing is represented by an ellipse. The use of elliptical shape is rarely used for engineering applications. Ellipse is dealt extensively in mathematical books. Elliptical shape is adopted for better asthetics.

Circles, arcs and ellipse

Follow the procedures and construct the following.

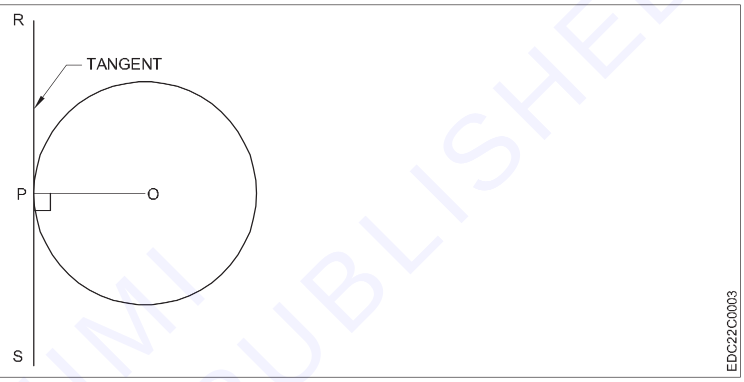
Procedure

Exercise 1.1

Draw a tangent to a given circle of ϕ 70 mm at any point `P' on it. (Fig 1)

- Draw a circle of ϕ 50 with `O' as centre.
- Mark the given point `P' on the circumference of the circle.
- Join OP.
- Draw a line RS perpendicular to PO through `P'.
- RS is the tangent at `P'.

Fig 1

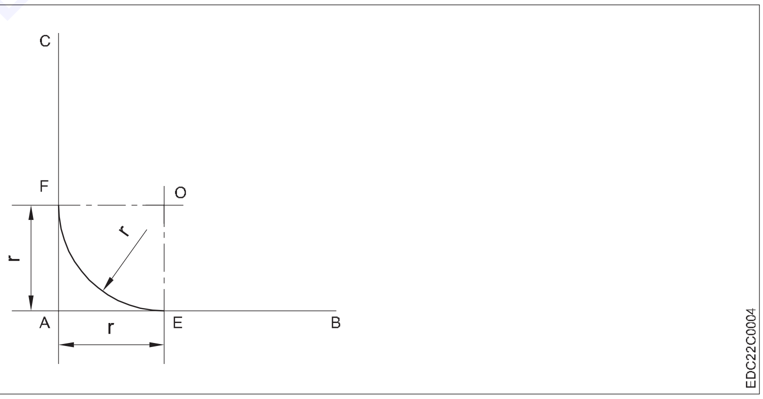


Exercise 1.2

Draw an arc of given radius (R 20 mm) to touch to two straight lines (50 mm each) at right angles. (Fig 2)

- Draw the lines AB and AC (50 mm each) at right angles.
- With `A' as centre and given radius (R 20 mm) draw an arc to cut lines AB and AC at E and F.
- With E and F as centres and the radius given (R 20 mm), draw arcs to intersect each other at `O'.
- Use `O' as centre and with same radius (R 20) draw a curve (arc) which will just touch the given lines AB and AC.

Fig 2

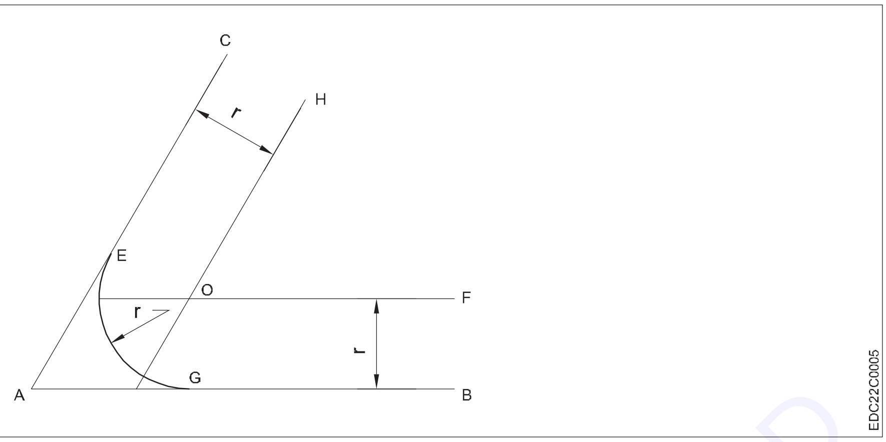


Exercise 1.3

Draw an arc of given radius (R 20 mm) to touch the given lines which make an acute angle between them (assume 60°). (Fig 3)

- Draw an acute angle BAC (60°).
- Draw a horizontal parallel line EF at a distance equal to the given radius (20 mm).

Fig 3



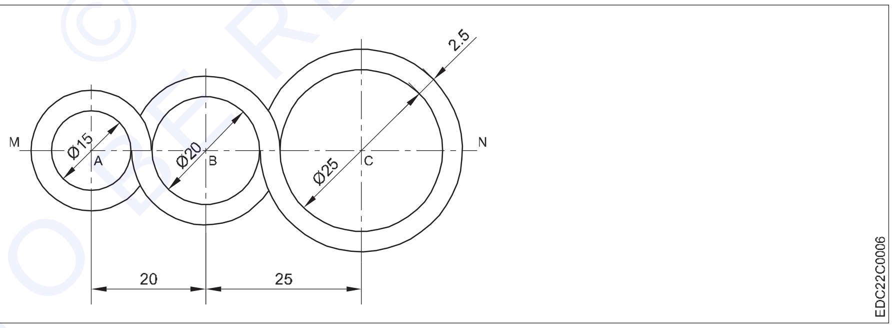
- Draw another angular parallel line GH at a distance of given radius 20 mm. Both the parallel lines drawn meet at `O'.
- With `O' as centre and `r' as radius (20 mm) draw an arc touching both lines AB and AC.

Exercise 1.4

Draw a loop of 3 circles pattern. (Fig 4)

- Draw any line MN and mark points A,B and C. So that AB = 20 mm and BC = 25 mm.
- With 'A' as centre draw concentric circles of dia 15 mm and dia 20 mm.
- With 'B' as centre draw concentric circles of ϕ 20 mm and ϕ 25 mm.
- With 'C' as centre draw concentric circles of ϕ 25 mm and ϕ 30 mm.
- Erase unwanted part of the circles to form the pattern.

Fig 4

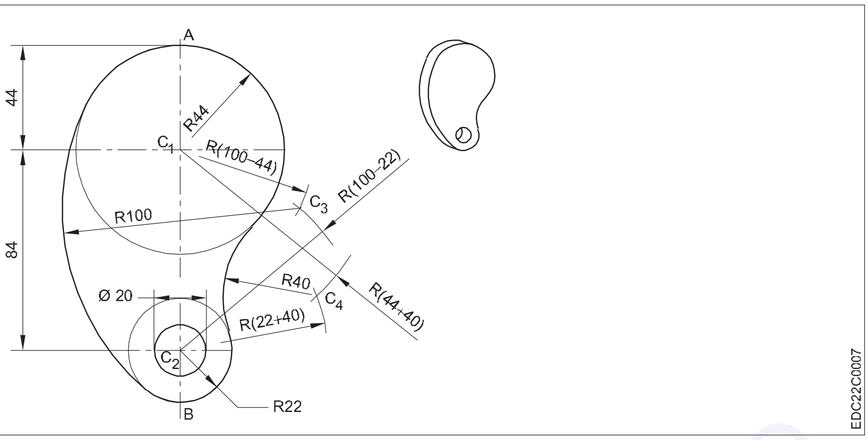


Exercise 1.5

Draw the cam as per dimensions given. (Fig 5)

- Draw a vertical line and mark the points C_1C_2 such that C_1C_2 = 84 mm.
- C_1 as centre, draw an arc of radius 56 mm (100-44) and C_2 as centre, draw another arc of radius 78 mm (100-22). Both arcs cut at C_3.
- Similarly obtain a point C_4 by drawing two arcs of radii 84 mm (44 + 40) and 62 mm (22 + 40) from points C_1 and C_2.
- Draw a circle of radius 44 mm with C_1 as centre and draw a circle or radius 22 mm with C_2 as centre.
- Produce C_1C_2 and get points A and B.
- C_3 as centre and radius BC_3 (100 mm) draw an arc.
- C_4 as centre and radius 40 mm draw an arc.
- Draw a circle of R10 with centre C_2.
- Rub off the unwanted lines and complete the pattern.

Fig 5

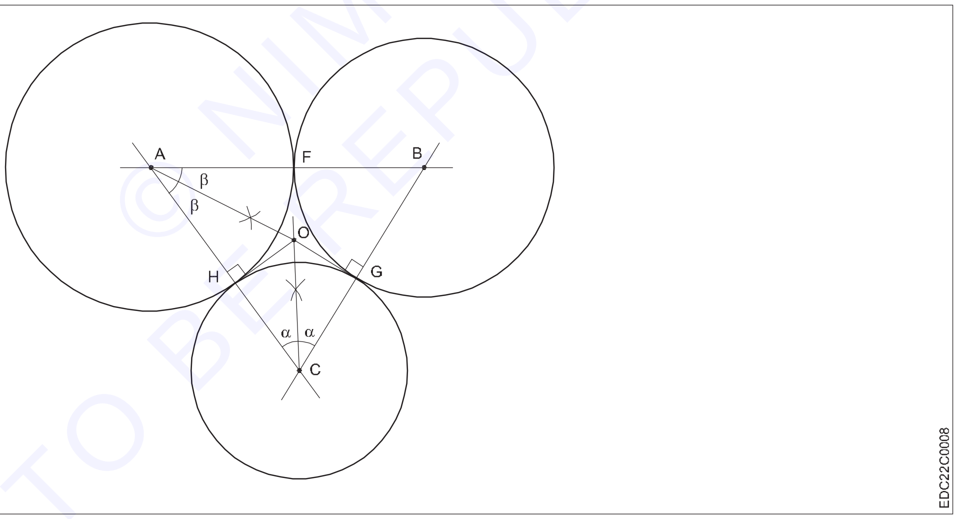


Exercise 1.6

Draw three circles tangential to each other if centres A,B & C are given. (Fig 6)

- Mark centres A,B & C.
- Join AB, BC & CA and form triangle ABC.
- Bisect any two angles of the triangle. Bisectors cut the opposite sides AB and BC at F and G.
- 'A' as centre and AF as radius draw a circle.
- 'B' as centre and BF or BG as radius draw another circle.
- 'C' as centre and CG as radius draw the third circle.

Fig 6

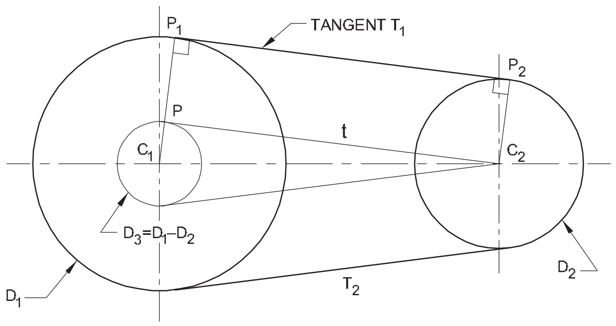


Exercise 1.7

Draw external tangents to circles of dia 40 and 30 and center distance 60 mm. (Fig 7)

- Draw a line and mark two points C_1 and C_2 at 60 mm.
- Draw two circles of dia 40 and dia 30 with center marked as C_1 and C_2.
- On dia 40 circle (D_1) draw a concentric circle of dia 10 (D_3) (dia 40 - dia 30).
- From center C_2 draw a line 't' touching circle D_3 at P.
- Join C_1 and P (angle P is right angle).
- Extend line C_1, P upto the circle D_1 meeting at P_1.
- Draw C_2P_2 parallel to C_1P_1.
- Join P_1 and P_2 forming the (common) tangent T_1 to circle D_1 and D_2.
- Similarly draw tangent T_2. Tangents T_1 and T_2 are called external tangents.

Fig 7

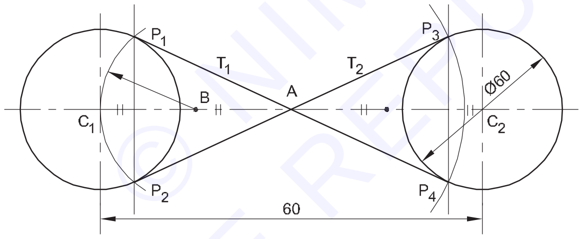


Exercise 1.8

Draw internal tangents to circles of same diameter 40 each and center distance 60 mm. (Fig 8)

- Mark two points C_1 and C_2 60 mm apart.
- Draw circles of ϕ 40 with centres C_1 and C_2.
- Mark 'A' the midpoint of C_1C_2 .
- Bisect C_1A and mark it as B.
- 'B' as centre C_1B as radius, draw an arc and get points P_1 and P_2.
- 'A' as centre AP_1 as radius, draw an arc and get points P_3 and P_4.
- Join P_1P_4 and P_2P_3.
- Mark them as T_1 and T_2, the internal tangents.

Fig 8

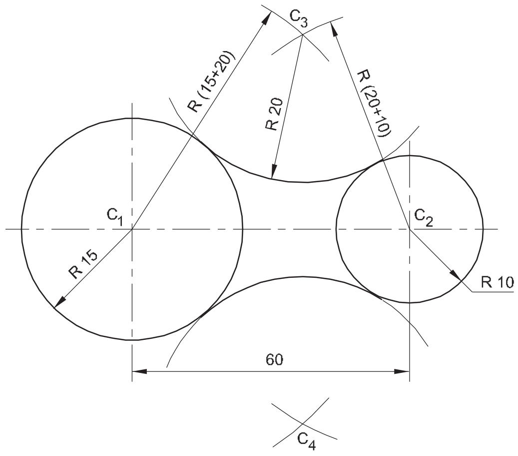


Exercise 1.9

Draw tangential curves for the circles of diameter 30 mm and 20 mm. (Fig 9)

- Draw centers C_1 and C_2 at a distance of 60 mm.
- Draw the given circles of dia 30 and dia 20 on C_1 and C_2.
- C_1 as center and radius 35 (R15 + R20), draw arcs on either side of horizontal center line.

Fig 9



- C_2 as centre and radius 30 (R10 + R20), draw arcs cutting the arcs R35 at C_3 and C_4.
- C_3 and C_4 as centers, with R20, draw arcs forming the arc tangential curves to the circles.

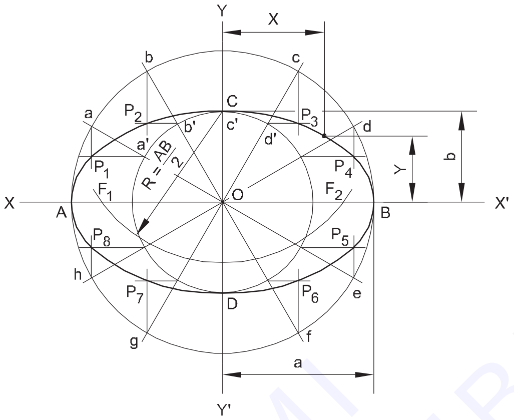
Exercise 1.10 to 1.12

Follow the procedures of the above exercises and draw the parts involving circles and arcs available in the work book. (Ex.1.10 to 1.12)

Exercise 1.13

Construct an ellipse by concentric circle method. Major axis 80 mm. Minor axis 40 mm. (Fig 10)

Fig 10



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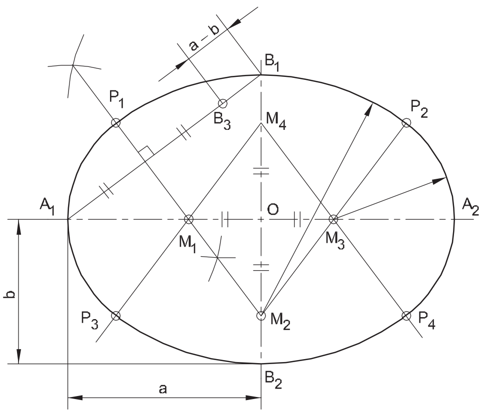
- Draw the major axis AB (80 mm) and minor axis CD (60 mm), bisecting at right angle at 0.
- '0' as centre OA and OC as radius, draw two concentric circles.
- Draw a number of radial lines through '0' (say 12) cutting the two circles.
- Mark the points on the outer circle as a,b,c.
- Similarly mark the corresponding intersecting points on inner circle as a',b',c'.
- From points such as a,b,c... draw lines parallel to minor axis.
- From points such as a', b',c'.... draw lines parallel to the major axis to intersect with the corresponding vertical lines at points P_1, $P_{2,}P_3$.... etc.
- Join all these points with a smooth curve by free hand or using "french curve" and form the ellipse.
- To find the 'Foci' - with half the major axis (a) as radius and with 'C' on the minor axis as centre, draw an arc cutting the major axis, at two points, mark them as F_1F_2 the focus points of the ellipse.

Exercise 1.14

Construct an ellipse by four centre method - Major axis = 80 mm and Minor axis = 40 mm - Type A. (Fig 11)

- Draw the major axis A_1A_2 and minor axis B_1B_2.
- Set off B_1M_2 and B_2M_4 equals to A_1B_1.
- Join A_1B_1 and set off B_1B_3 equal to a-b (a = OA_1, b = OB_1)
- Draw a bisector on A_1B_3 which intersects A_1A_2 at M_1.
- Similarly obtain M_3. M_2 & M_4 as centres and B_1M_2 as radius, draw arcs P_1P_2 & P_3P_4.
- M_1M_3 as centres and M_1P_1 as radius, draw arcs P_1P_3 & P_2P_4 and complete the ellipse.

Fig 11



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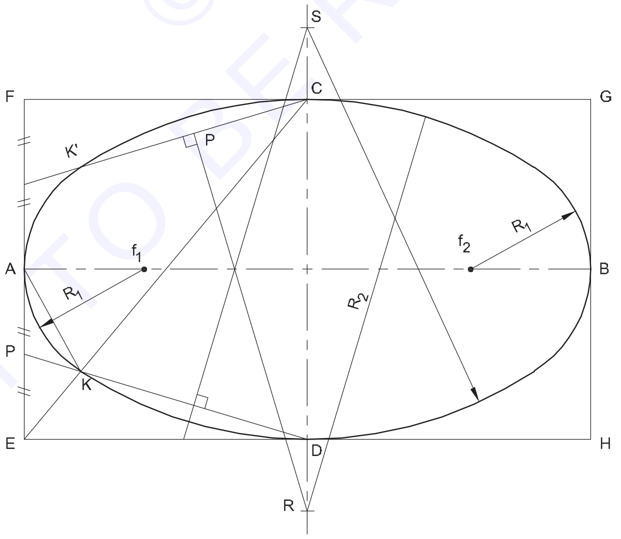
Exercise 1.15

Construct an ellipse by four centre method - Major axis = 80 mm and Minor axis = 40 mm - Type B. (Fig 12)

- Draw the rectangle EFGH (80 x 40) and draw AB & CD represent major and minor axis.
- Join EC
- Bisect AE and mark P the mid-point.
- Join DP meeting EC at K.
- Draw perpendicular bisectors of KD and extend DC and locate point `S'.
- 'S' as centre SD as radius draw the arc KD.
- Similarly get the point 'R'.
- Join AK and draw perpendicular bisector on it, and meet AB at f_1.

Fig 12



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EXERCISE 2 :Parabolic curves, Hyperbola, Involute and Helix

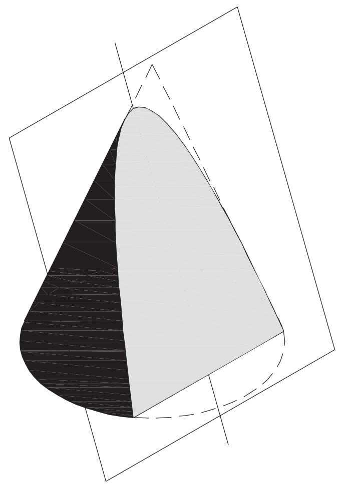
Parabolic curves, Hyperbola, Involute and Helix

When the cutting plane is parallel to the generators (slant line) of the cone, (and inclined to the axis) the section obtained is called "Parabola". (Fig 1)

Properties: Parabola is defined as the locus of a point which moves so that the ratio of its distance from a fixed point F (called the focus) and a directrix bears a constant and equal to 1 (Unity).

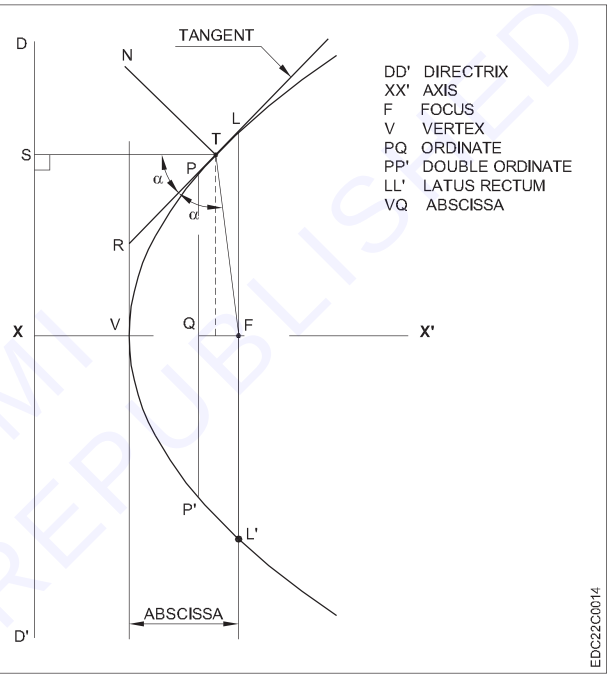
In other words if the perpendicular distance of any point on the curve from a fixed line called directrix is equal to its distance from focus, the curve is called "Parabola". (Fig 2)

Fig 1



PARABOLA

Fig 2



Elements of Parabola

Axis: It is a line (XX') perpendicular to the directrix and passing through the focus.

Vertex (V)**:** It is the mid point of the perpendicular line drawn from focus to directrix.

Ordinate: Perpendicular distance of any point P on the curve to the axis line PQ.

Double ordinate: When the ordinate is extended to meet the curve on the other side. Crossing the axis, it is twice the ordinate line P-Q-P' is the double ordinate.

Latus rectum: The double ordinate which passes through the 'Focus' is called latus rectum. (LFL')

Abscissa: The distance along the axis XX' from vertex (V) and a point through which the double ordinate passes is called the "Abscissa" VQ is the abscissa corresponding to the ordinate PQ.

Tangent and normal for the point T

- Draw TS perpendicular to directrix
- Draw TF
- Bisect angle STF, it will be tangential to parabola at P.
- Draw TN perpendicular to tangent will be normal at P.

A parabola can be constructed by any one of the following methods:

- ordinate method
- rectangle method
- tangent method

- parallelogram method
- offset method

Practical application: Search lights, reflecting surfaces for light and sound, bridge arches, wall brackets and largely used in graphic methods for determining the stress upon beams and girders etc.

Parabolic curves

Follow the procedures given below and construct parabolic curves.

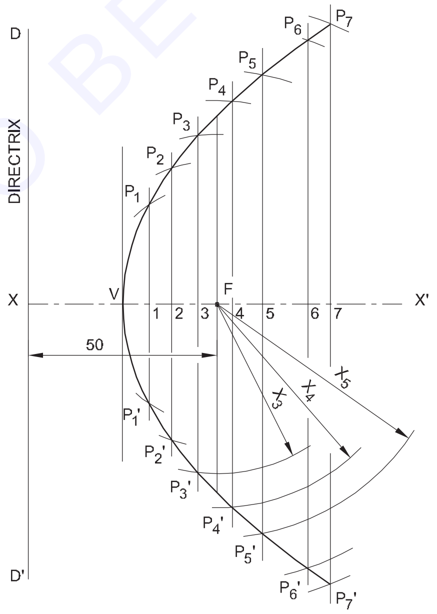
Procedure

Exercise 2.1

Construct a Parabola from a given focus is at 50 mm from the directrix. (Fig 1) - Ordinate method.

- Draw a vertical line D-D^1 the directrix.
- Draw horizontal line XX^1, the axis through any point X on the directrix.
- Mark the focus 'F' on XX^1 = 50 mm from X (on the directrix).
- Mark the mid point of XF, as V.
- Mark a number of points from V towards right side on the axis as 1,2,3,4.....
- Draw vertical lines through these points as shown, forming double ordinates.
- Point 'F' as centre, X-1 as radius, draw arcs on the co-ordinates (vertical lines) passing through 1, mark points P^1 & P_1^1.
- X-2 as radius, F as centre, draw arcs on the 2nd ordinate, mark P^2 & P_2^1.
- Similarly get point P_3, P_4.... P_3^1, P_4^1 etc. on the axis as above.
- Join all the points with a smooth curve, by free hand and form the parabola curve.

Fig 1



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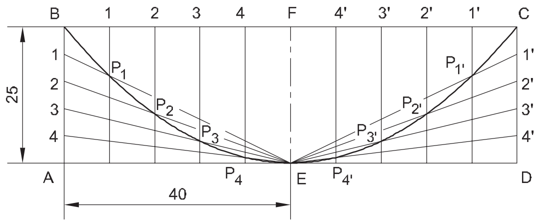
Exercise 2.2

Construct a Parabola given the base and axis of a rectangle; base 80 mm, axis 25 mm - Rectangle method. (Fig 2)

- Draw a rectangle ABCD of sides 80 mm & 25 mm.
- Mark centre points of AD and BC, as E and F, join EF.
- Divide AB & CD and into any number of equal parts say 5. Also divide AE and ED into the same number of equal parts and number them as shown.
- From point E on AD, draw lines to the divisions on AB & CD.
- From the points on AED, draw parallel lines to EF.
- Mark the intersecting points P_1, P_2, P_3, P_4 on either side of axis.
- Form the parabola by joining the points BEC and intersecting with a smooth curve, passing through P_1, P_2

Fig 2



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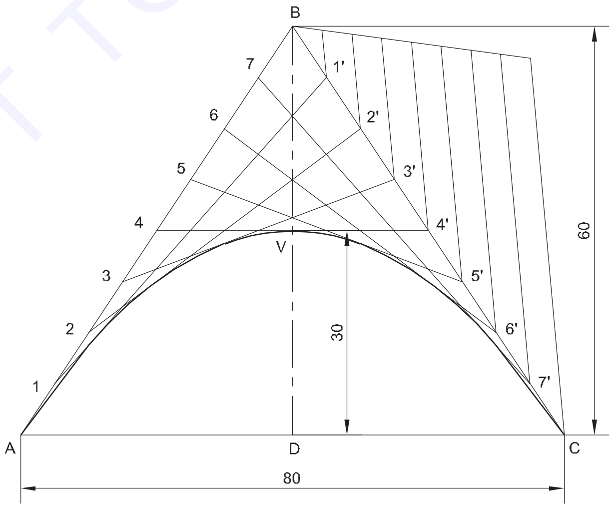
Exercise 2.3

Construct a Parabolic curve with base as 80 mm and axis 30 mm - Tangent method. (Fig 3)

- Draw an isosceles triangle of base 80 mm and altitude 60 mm (double the abscissa).
- Join BD and mark mid point V, the vertex.
- Divide AB and BC into same number of equal parts using divider/other methods.
- Mark the points on AB as 1,2,3 etc in ascending order..
- Similarly mark 1', 2', 3' etc on CB but in descending order.
- Draw lines 1-1', 2-2'..... 7-7'.
- Join the points with A, V and C with a smooth curve. AVC is tangential to line 1 1', 2 2' etc and form the required parabola.

Fig 3



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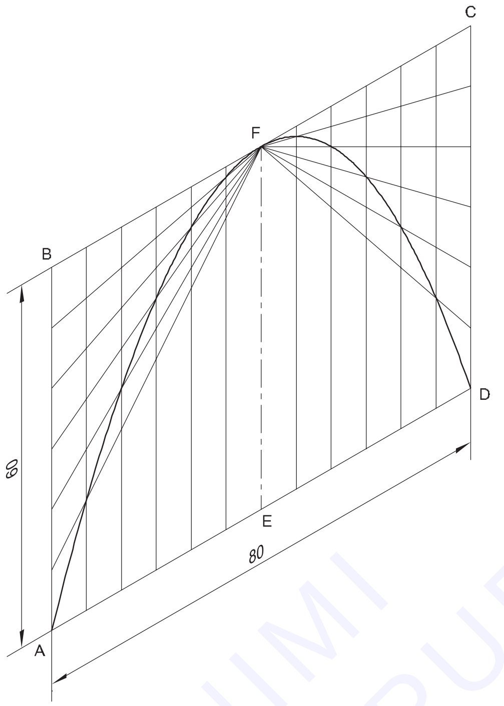
Exercise 2.4

Parabolic curve of sides 80 and 60 making 60°/120° Parallelogram method. (Fig 4)

- Procedure is similar to the exercise (7.2) rectangle method of drawing a Parabolic curve.

Fig 4

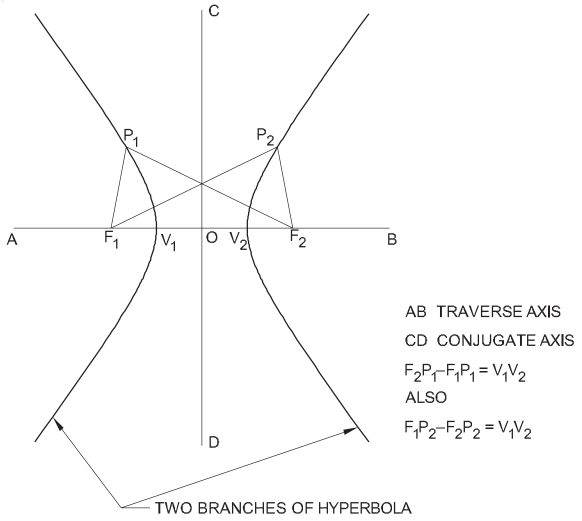


Hyperbola

Hyperbola (Fig 1)**:** It is a conic section, formed by cutting plane inclined to the axis (not parallel to the generator) and passes through the base. It is the locus of the point which moves so that its distance from a fixed point, the 'Focus' (F) bears a constant ratio (this ratio is called eccentricity and it is always greater than 1) to its perpendicular distance from a straight line called the Directrix.

It is the path of a point moving in such a way that the difference of its distances from the fixed points is a constant and is equal to the distance between the vertices of the two branches of the hyperbola. This distance is also known as major axis of the hyperbola. (Fig 1)

Fig 1



Transverse axis (Fig 1)**:** It is the (horizontal axis) line passing the two vertices (V_1,V_2)of the pair of hyperbola.

FOCI (F_1,F_2)**:** The two fixed points used for defining a hyperbola are called the foci and they lie on the traverse axis.

Ordinate: It is the perpendicular distance from any point on the curve to the transverse axis.

Double ordinate: The distance between the two (similar) points PP' (Fig 2) perpendicular to the axis.

Abscissa: The distance from vertex to the point on the axis where the double ordinate cuts the axis. (AQ in Fig 2)

Conjugate axis: It is the perpendicular XX' to the transverse axis passing through the mid point of the transverse axis AB.

Asymptotes: These are lines passing through the center and tangential to the curve at infinity.

Rectangular hyperbola: When the angle between the asymptotes is 90° the curve is called a rectangular or equilateral hyperbola. (Fig 3)

Practical application: Rectangular hyperbola and its application in design of water channels. Further it also represents Boyle's law graphically and in design of Electronic transmitter receiver and Radar antenna.

Fig 2

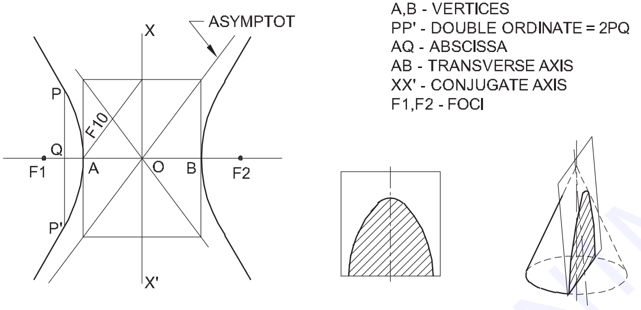
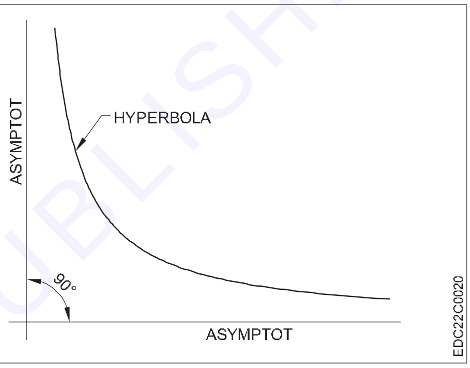


Fig 3



Hyperbola

Follow the procedure and construct hyperbola

Procedure

Exercise 2.5

Construct a hyperbola for that eccentricity and the distance of focus from the directrix are given. (Fig 1)

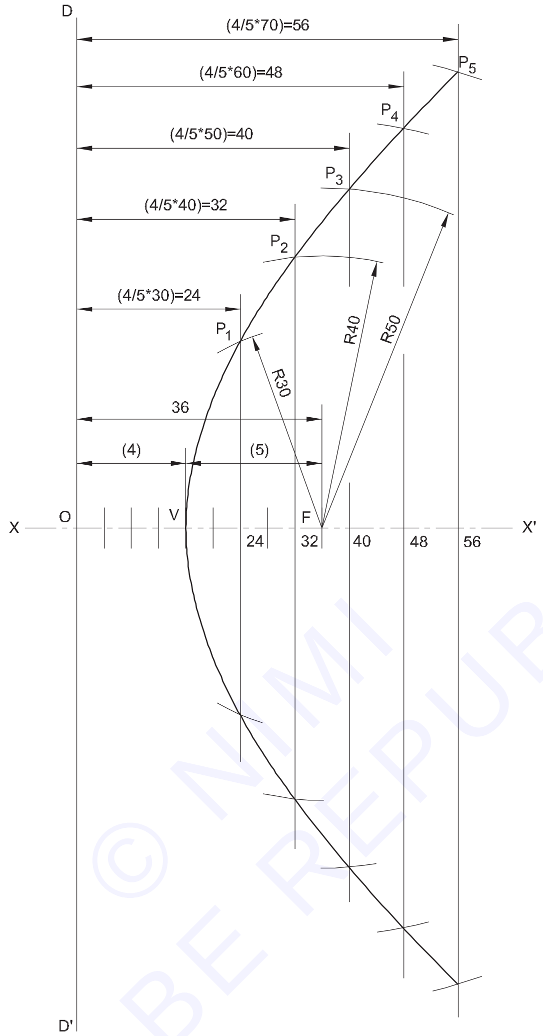
- Draw the locus of a point which moves so that its distance from a fixed point (Focus) and a line (directrix) bears a constant ratio of 5/4 (i.e eccentricity). Assume the focus at a distance of 36 mm from the directrix.
- Draw a directrix DD^1 and perpendicular XX^1 to it at 0.
- Mark F on XX^1 at a distance of 36 mm from 0.
- Divide 'OF' into nine equal parts and mark 4th division as 'V'.
- Prepare a table as shown below such that the ratio of the value in each column is 4:5

From directrix	24	32	40	48	56
From focus	30	40	50	60	70

- Draw parallels to directrix at distances 24,32 etc.
- F as centre and radius equal to 30, 40 etc. draw arcs to intersect the corresponding lines drawn in the previous step.

- Mark the intersecting points as P_1, P_2...etc.
- Complete the hyperbola by a smooth curve passing through these points.

Fig 1



HYPERBOLA

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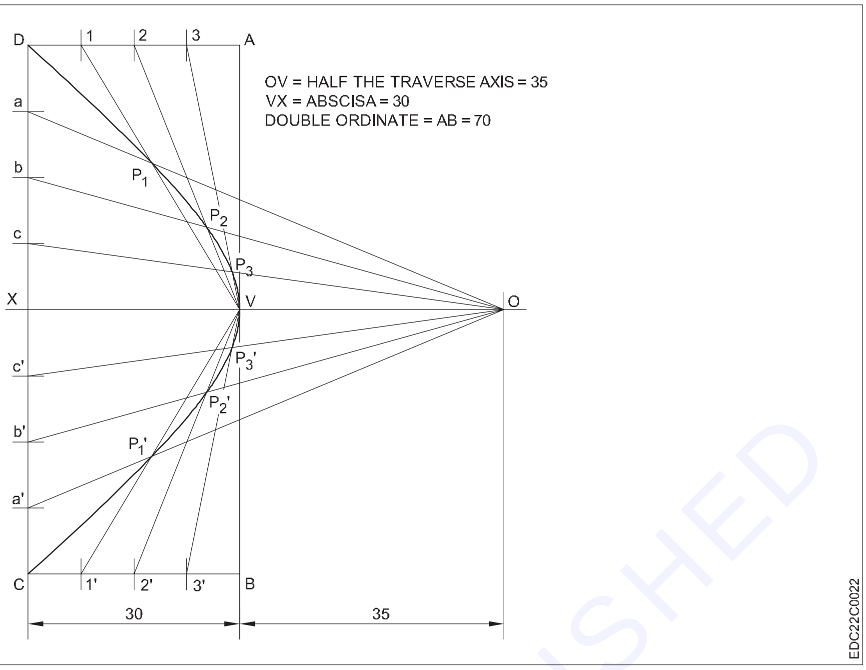
Exercise 2.6

Given the double ordinate, abscissa and distance between vertices (Fig 2)

- Draw a hyperbola with double ordinate 70 mm and abscissa 30 mm (rectangle method) and the traverse axis 70 mm.
- Draw the rectangle ABCD 70 x 30 mm.
- Mark the mid points V, X on AB and CD.
- Join VX and extend it outside to '0' 35 mm (70/2) from V.
- Divide AD and BC into 4 equal parts. Mark them as 1,2,3,1^1,2^1,3^1.
- Join these points to V by straight lines.
- Divide DX;XC into 4 equal parts, each mark them as a,b,c,c^1,b^1,a^1.
- Join '0' to these points by straight lines.
- Mark the intersecting points as P_1,P_2 etc as shown.
- Join V-D-C with a smooth curve through P_1,P_2,P_3 etc.

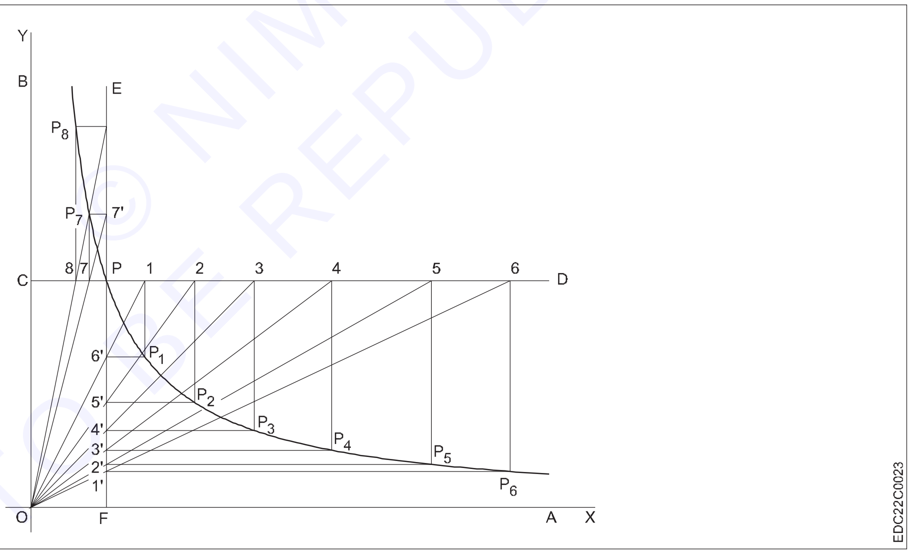
Fig 2



Exercise 2.7

Draw a rectangular hyperbola, given the position of a point on the curve. (Fig 3)

Fig 3



- Draw a rectangular hyperbola given a symptotes as OX and OY are at right angles and a point P on the curve is 30 mm and 10 mm from OX and OY respectively.
- Draw the asymptotes OA and OB at right angles to each other and locate the given point P. (10 mm from OX and 30 mm from OY)
- Draw the lines CD and EF passing through P and parallel to OA and OB respectively.
- Locate number of points 1,2,3 etc. (not necessarily equidistant) along the line CD.
- Join 1,2,3 etc to O and extend if necessary till these lines meet the line EF at points 1', 2', 3' etc.
- Draw vertical lines through 1,2,3 etc. parallel to EF and through 1', 2', 3' etc. parallel to CD to intersect at P_1, P_2, P_3 etc.

  A smooth curve passing through P_1, P_2, P_3 etc. is the required rectangular hyperbola.

Note: Rectangular hyperbola is a graphical representation of Boyle's law, PV = constant. This curve also finds application in the design of water channels.

Exercise 2.8

Construct a hyperbola passing through a given point between the asymptotes making any angle other than 90°.

Involute

Involutes (Fig 1)**:** It is yet another geometrical curve. Although they are defined in more than one way, perhaps the simplest one is as follows.

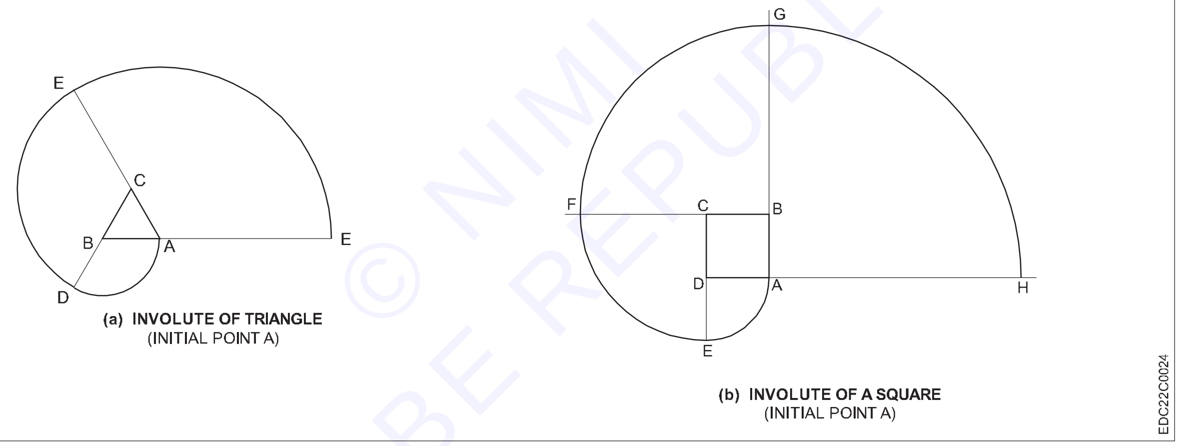
It is the curve traced by a point on a cord as it unwinds (but remains taut) around a circle or polygon.

Alternatively an involute may be defined as the curve traced by a point on a straight line which rolls around a circle or polygon without slip. Depending on the plane shape around which the line rolls, the involutes are named as involute of a triangle, involute of a square, involute of a polygon, involute of a circle etc.

Involutes are not expressed in terms of their basic shape like square etc refers only the involute of a circle.

The most common application of involute is seen in the manufacture of gears. The profile of a gear tooth is the shape of an involute.

Fig 1



(a) INVOLUTE OF TRIANGLE
(INITIAL POINT A)

(b) INVOLUTE OF A SQUARE
(INITIAL POINT A)

Involute

Follow the procedures and construct involutes.

Procedure

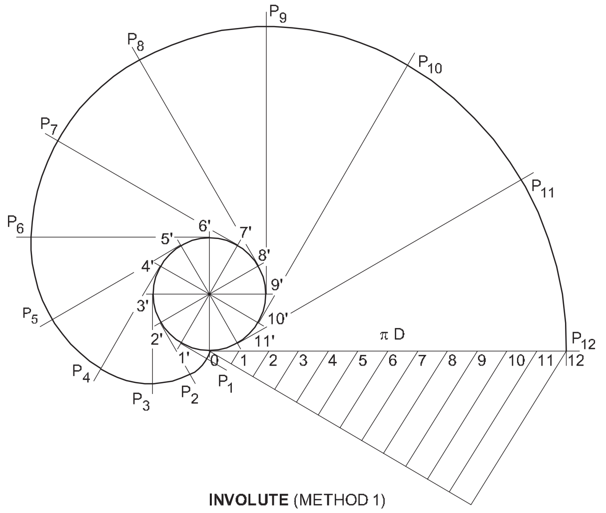
Exercise 2.9

Construct an involute of a circle of diameter 30 mm

Radial line method (Fig 1)

- Draw a circle of diameter 40 mm.
- Divide the circle into a number of (say 12) equal parts and number than as 1', 2', 3'......12'.
- Draw a tangent through any of the points 1' , 2' etc and set a length equal to πD (graphically) on it, preferable draw the tangent from the point `O'.
- Divide the circumference (πD) into equal parts as was done for circles and number them as 1,2,3....12.
- Draw tangents from points 1'2'3' etc and mark their lengths respectively equal to 01, 02, 03....011 etc and get points such as P_1,P_2...P_{12} .

Fig 1



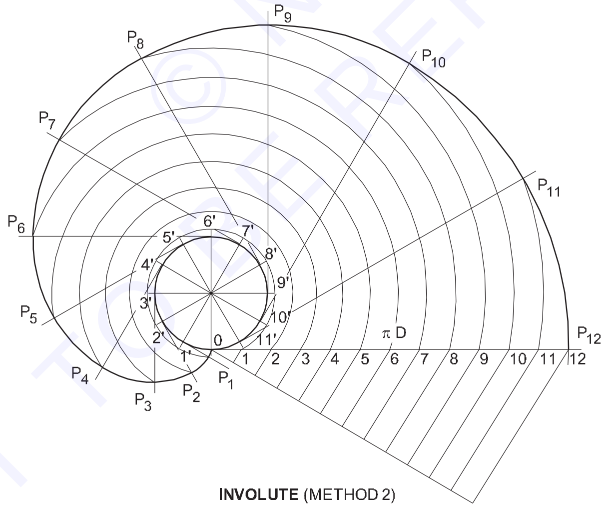
INVOLUTE (METHOD 1)

Exercise 2.10

Construct an involute of a circle of diameter 30 mm by Concentric method (Fig 2)

- Follow the first four steps of the Radial line method. (Ex.2.1.1)
- Draw the concentric arcs, from the centre of the circle through the divisions 1,2,3.... etc to cut the tangential lines drawn from 1',2',3' etc at points P_1,P_2,P_3 etc.
- Join points P_1, P_2, P_3 by a smooth curve and form the involute of the circle.

Fig 2



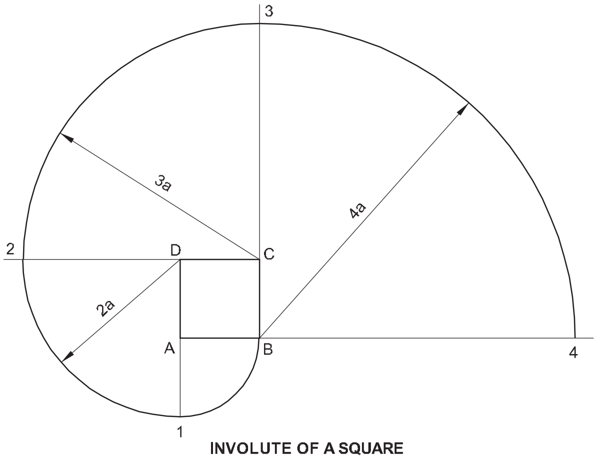
INVOLUTE (METHOD 2)

Exercise 2.11

Construct an involute of square of side 25 mm (Fig 3)

- Draw a square ABCD and extend the sides.
- 'A' as centre and radius 25 mm, draw the quadrant B.
- 'D' as centre and radius D-1, draw a second quadrant 1-2.
- 'C' as centre and radius C-2, draw a third quadrant 2-3.
- 'B' as centre and radius B-3 and draw forth quadrant 3-4.
- Now the curve 1-2-3-4 is the involute of the square.

Fig 3



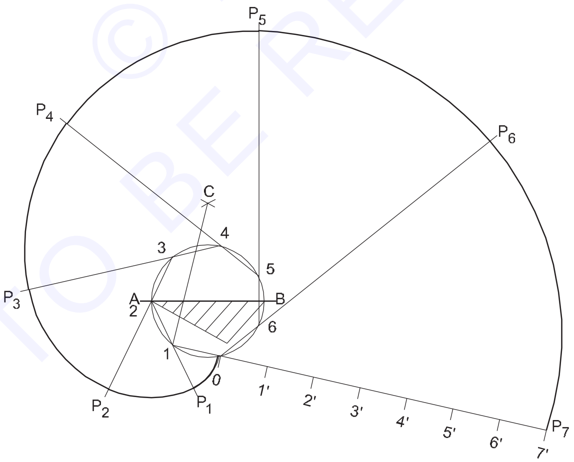
INVOLUTE OF A SQUARE

Exercise 2.13

Construct an involute of a heptagon (Polygon) in a circle of diameter 40 mm (Fig 4)

- Draw a polygon (heptagon) in a circle of ϕ 40 and mark the corners 0,1,2,.....6 etc.
- Extend each side of the polygon (heptagon) as shown.
- '1' as centre and 1-0 as radius, draw an arc OP_1
- '2' as centre and 2-P_1 as radius, draw an arc P_1P_2
- Proceed in the similar way and draw arcs P_3 P_4 P_5 P_6 & P_7
- Now the curve `O' $P_1P_2P_3....P_7$ is the required involute.

Fig 4



INVOLUTE OF A HEPTAGON

Helix

Helix (Fig 1)**:** When a point moves at a constant speed along a line which revolve at a constant rate around a fixed axis. The point traces a curve similar to the coil of a spring, such a curve is called the helix.

The fixed distance through which the point moves parallel to the axis for each revolution of the line is called the "Pitch" of the helix. (or lead)

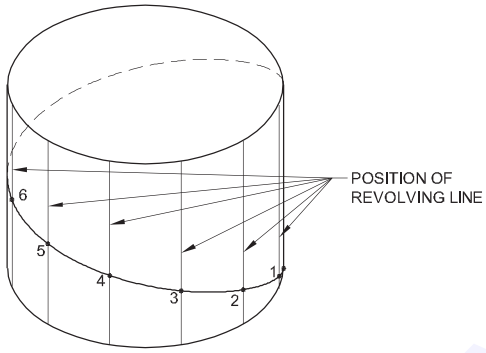
The helix is called as cylindrical helix when the revolving line is parallel to the axis of revolution.

If the revolving line is inclined to the axis of revolution the resulting helix is called conical helix. (Fig 2)

The helix may be either right handed or left handed. The right handed helix climbs from the base towards right side as it rises along the axis.

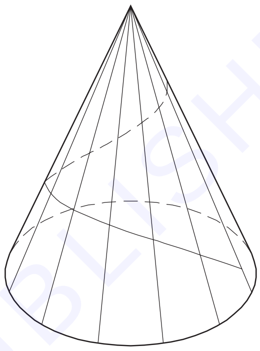
Practical application: Threads on bolts, screws, nuts, springs, spiral staircase etc.

Fig 1



1,2,3,4 etc POSITION OF THE MOVING POINT

Fig 2



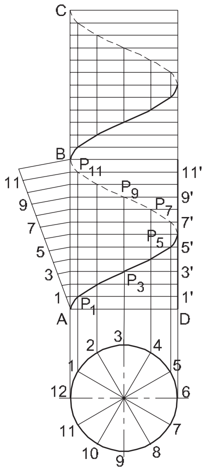
Helix

Procedure

Exercise 2.14

Construct a helix for two revolutions of a point on a cylinder dia 40 and pitch of helix 30 mm. (Fig 3)

Fig 3



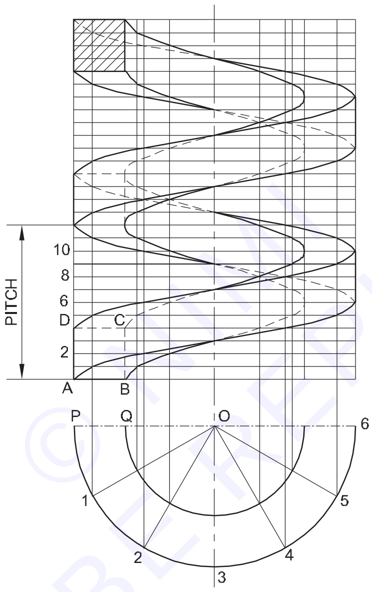
HELIX

- Draw a circle of diameter 40 mm, divide it into 12 equal parts and number them as 1,2.....12.
- Draw vertical projectors of sufficient length (more than 60 mm) from points 1,2.....12, such that the points 1 and 11, 2 and 10, 3 and 9 etc lie on same vertical line.
- Mark the points A,B and C such that AB = BC = 30 mm. (Pitch of the helix)
- Divide AB into the same number (12) of equal parts as there are similar divisions in the circle.
- Draw horizontal lines from 0, 1',2'..... etc to intersect with the projections from 1,2,3 etc at points P_1, P_2, P_3...... P_{12}.
- Join the points P_1, P_2 etc with a smooth curve and form helix.
- Repeat the above procedure in portion BC and complete the helix for two revolutions.

Exercise 2.15

Construct a helical spring of 10 mm square. Cross-section has an outside dia of 60 mm and pitch 30 mm. Draw two pitches of the spring. (Fig 4)

Fig 4



- To draw the spring with one pitch we have to draw four helixes.
- Helixes starting from A is to be drawn using procedure of the previous exercise with outside diameter 60 mm. For helix 'D' reproduce same thing at distance 10 mm above helix A (thickness of spring 10 mm).
- Follow the above said procedure complete two helixes B&C with inside diameter 40 mm.
- Repeat four more curves and complete the two pitches of the spring.

Exercise 2.16

On a cylindrical rod of dia 80 mm a square helical groove of depth 12 mm is cut forming square thread. Assuming the pitch equal to 24 mm, draw 2 complete turns of the thread. Find the helix angle. Assume width/thickness of thread = depth of thread. (Fig 5)

Outside dia = 80 mm

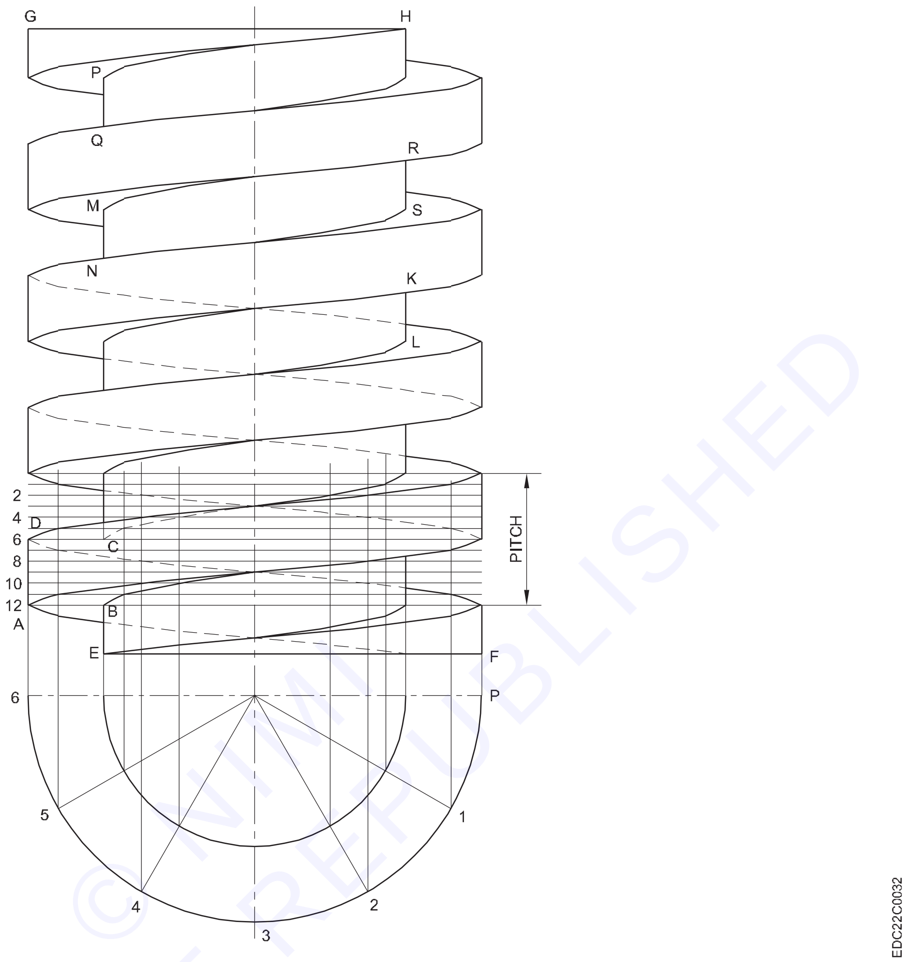
Depth of thread = 12 mm

Pitch of thread = 24 mm

- Starting from ABCD&E, draw four helixes using the method followed in the previous exercises and complete 5 pitches by drawing parallel helical curves.
- Draw vertical lines such as PQ, RS, MN etc to represent the core of the all five pitches.
- Draw parallel lines EF and GH at any convenient distance.

- Rub of the invisible portions of the helixes and complete the square thread.

Fig 5



EXERCISE 3 : Technical drawing / Sketching of components/ Parts

Technical drawing/Sketching of components/Parts

Views of objects: A pictorial representation of a drawing or a photograph shows an object as it appears to the observer. Hence such a representation cannot describe the object fully and also it does not show the exact shapes and sizes of the several parts.

In industry for the purpose of production, a complete clear description of the shape and size of an object to be made is essential, to sketch certain objects that are to be manufactured as intended by the designer. So to provide such information perfectly and correctly a number of views and arranged in multi-views method of projection. In this method each view supplies a certain definite information.

In Fig 1 the observer looks perpendicularly to the face of the object and obtains true shape and size of that side. It is known that an object has three principle dimensions such as height, width and depth.

Note: In any one view of a three dimensional object can show only two dimensions, the third dimensions will be seen in an adjacent view.

To get the additional views, the object is revolved as shown in Fig 2.

Fig 1

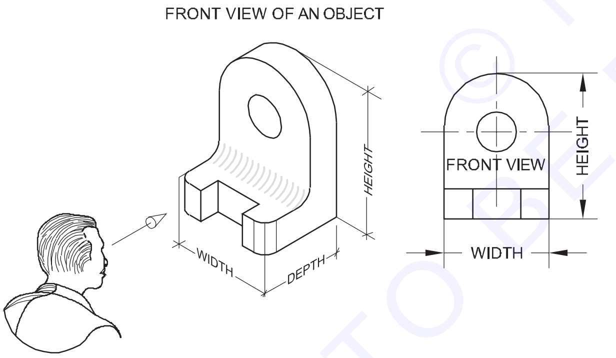
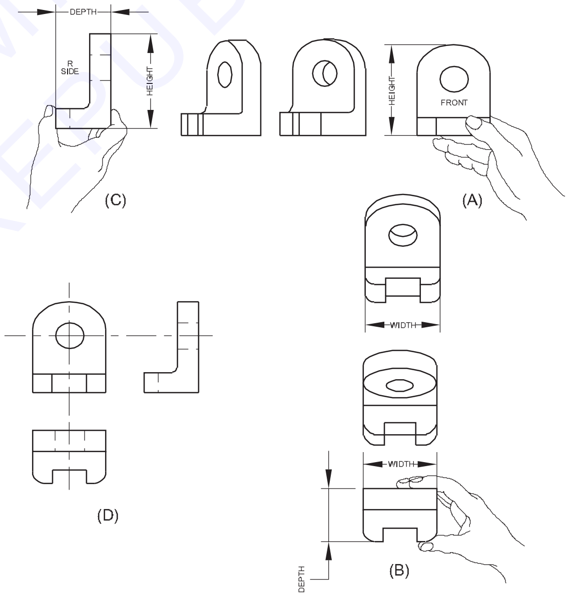


Fig 2



Any object can be visualised from six mutually perpendicular directions as shown in Fig 3 and the arrangement of the views shown in Figure. The method of obtaining the views of the objects is discussed in earlier lessons.

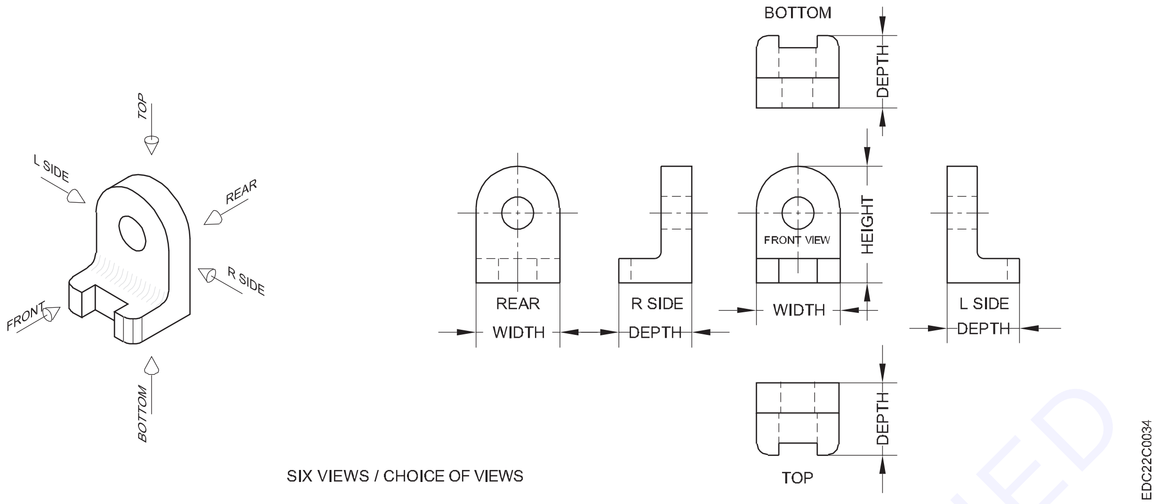
Choice of views: It is more important that drawing of the object for use of production should have only such of those views needed for a clear and complete shape description of the object. The required distinctive features are rounded top hole, seen from front.

- Rectangular notch and rounded corners seen from the top. (Fig 4)
- Right angle with filleted corner, seen from the side.

The above features can be shown in three views by eliminating the unnecessary views which has shown in Fig 3.

In some cases two views will be sufficient to furnish the features of the object. Fig 5A,B,C are the some examples for such cases.

Fig 3



SIX VIEWS / CHOICE OF VIEWS

Fig 4

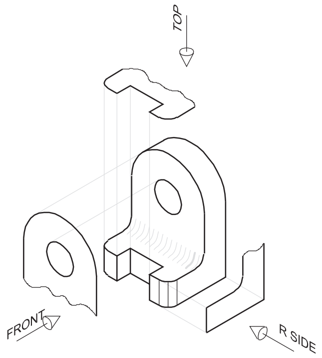
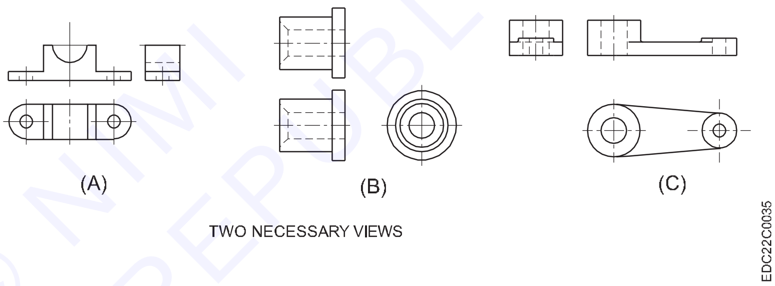


Fig 5

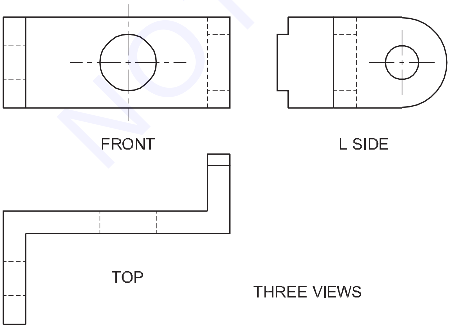


TWO NECESSARY VIEWS

In some cases three views are compulsary. For example Fig 6 shows the three views of a bracket. In this case if the top view is omitted, providing the front and side view alone, it is difficult to understand the two views or visualise the object, because the characteristic `Z' shape of the top view is omitted and also the corners are squares and not filleted. Hence in this example three views are necessary.

One view drawings: Often single view with a note or lettered symbols is so sufficient to explain the features of the object. Fig 7A one view of the thin component, adding a note indicating the thickness is sufficient to explain the features. At Fig 7B one view of a stepped turning piece shown with a threaded portion at the end with additional notes, is sufficient to explain the features of the object.

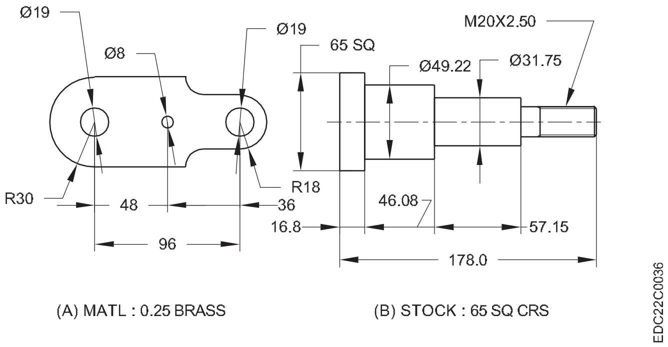
Fig 6



THREE VIEWS

Fig 7

ONE - VIEW DRAWINGS

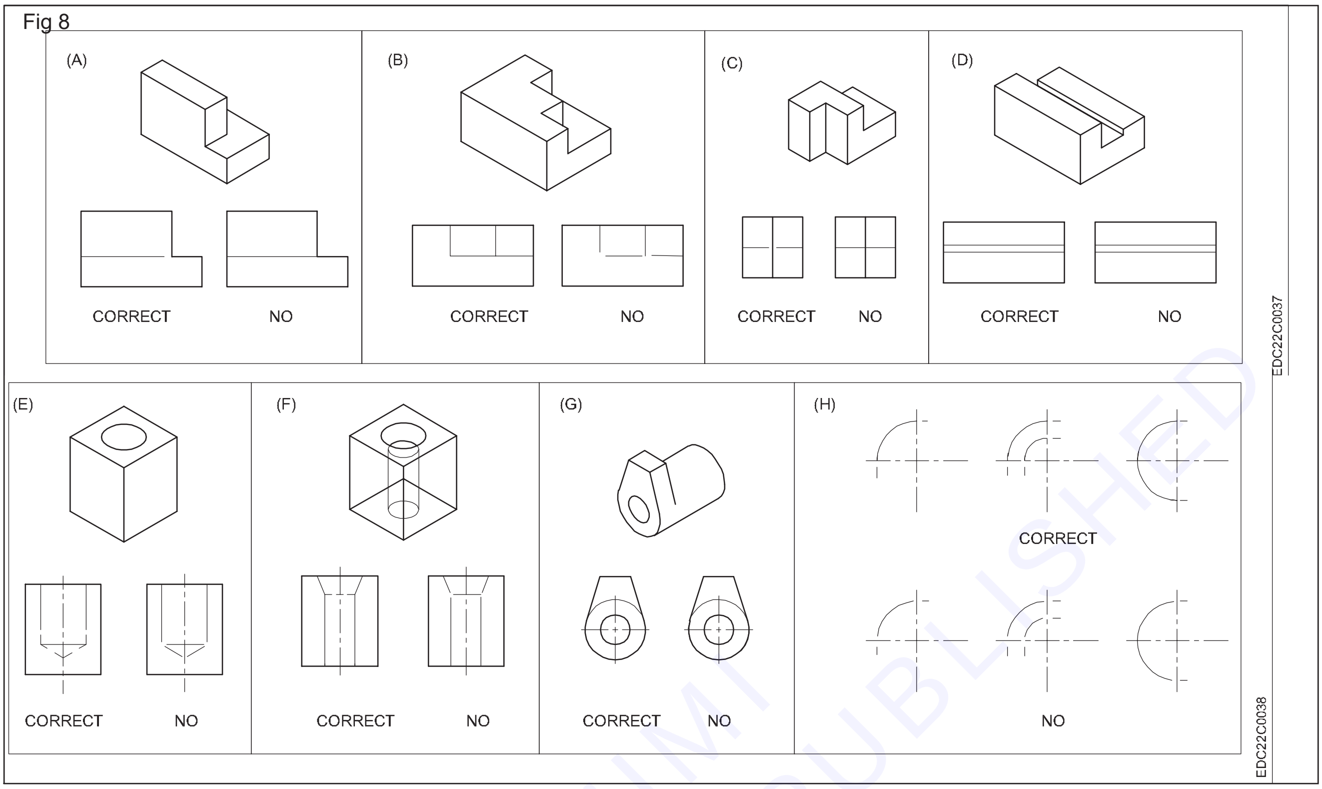


(A) MATL : 0.25 BRASS

(B) STOCK : 65 SQ CRS

Hidden lines: Correct and incorrect method in sketching/drawing hidden lines are shown in Fig 8.

Fig 8



The following rules can be adopted for the drawing of hidden lines.

- Leave a gap wherever a hidden line dash forms a continuation of a visible line. (Fig 8A)
- A hidden line should intersect to form `L' or `T' corners. (Fig 8B)
- A hidden line preferably should jump a visible line. (Fig 8C)
- Parallel hidden lines should be staggered. (Fig 8D)
- When two or three hidden lines meet at a point, the dashes should join. (Fig 8E)
- For showing the counter sunk hole. (Fig 8F)
- Correct and incorrect methods of hidden arcs. (Fig 8G & H)

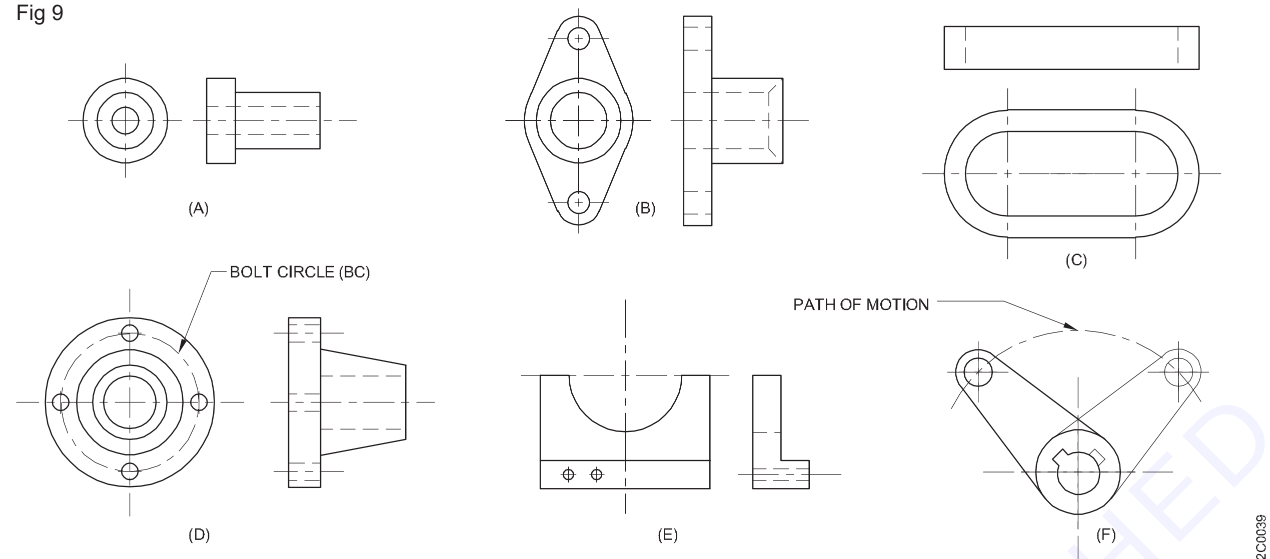
Centre lines: Centre lines are used to indicate arcs of symmetrical objects or features. Fig 9 shows the typical applications of the centre lines in various features.

Meaning of lines: A hidden line or a visible line has the following meanings

- Intersection of two surfaces
- Edge view of a surface
- Contour view of a curved surface. (Fig 10)

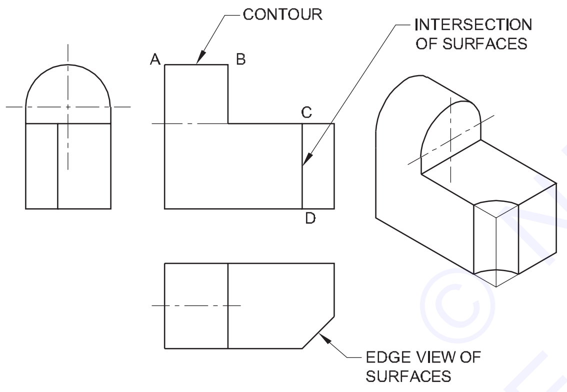
Procedure line: Hidden lines, centre lines and visible lines often coincide with each other and it is needed for craftsman to know which line should be shown. Fig 11 explains the same. A visible line is always precedence over (corners up) a centre line or hidden line A & B of Fig 11. A hidden line is always takes precedence over a centre line at c of Fig 11. (3rd angle projection)

Fig 9



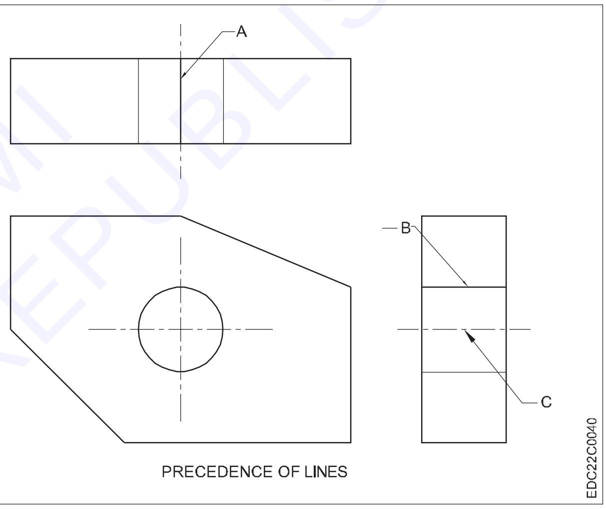
CENTER - LINE APPLICATIONS

Fig 10



MEANING OF LINES

Fig 11



PRECEDENCE OF LINES

Note that at A and C of 11 the centre line are shown, but separated from the view by a short gap.

Information given earlier in this lesson concerning views objects, choice of views, hidden lines, centre lines etc are applicable for both by drawing with instruments and free hand sketching.

Importance of technical sketching: The importance of free hand technical sketching of machine parts and components in engineering field cannot be over estimated. Free hand technical sketching is a drawing drawn with/without the use of any drawing instruments and drawn not to scale.

The presentation of the views should be in good proportion to the extent possibly and by visual identification. Free hand technical sketching helps the designer in reflecting his thoughts and recording his ideas. Most original ideas and thoughts are expressed first through the medium of free hand sketching. For verbal explanation free hand sketching plays a very important role.

Free hand sketching contains all necessary details such as shape and size description.

The Fig 12 shows a chuck key which records the importance of free hand sketching.

Fig 12A shows the `object' for layman's easy understanding and Fig 12B gives a free hand sketch for easy production with necessary dimensions.

The perfection and proficiency in free hand technical sketching can be obtained only by more practice.

Type of sketches

In technical drawing sketching the figures are prepared from three dimensional objects. The form of the drawing sketch determines approximately to any one of the following standard types of projections. (Fig 13)

- Multi-view
- Oblique
- Isometric
- Perspective

In multi-view (Fig 13A) the object is explained by its required views, which has been already discussed in earlier lesson.

The object may be shown in a single view by isometric (Fig 13B), oblique (Fig 13C) and by perspective (Fig 13D) projections.

Fig 12

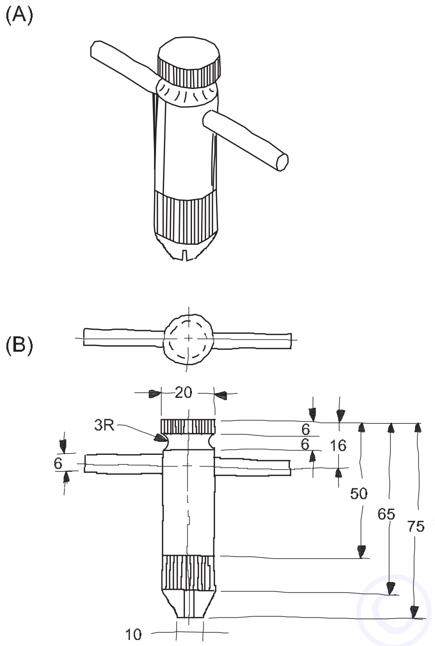
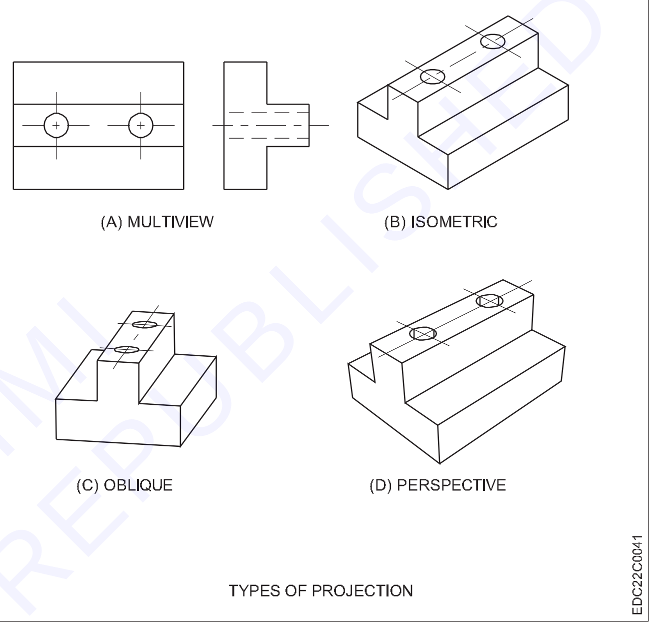


Fig 13



TYPES OF PROJECTION

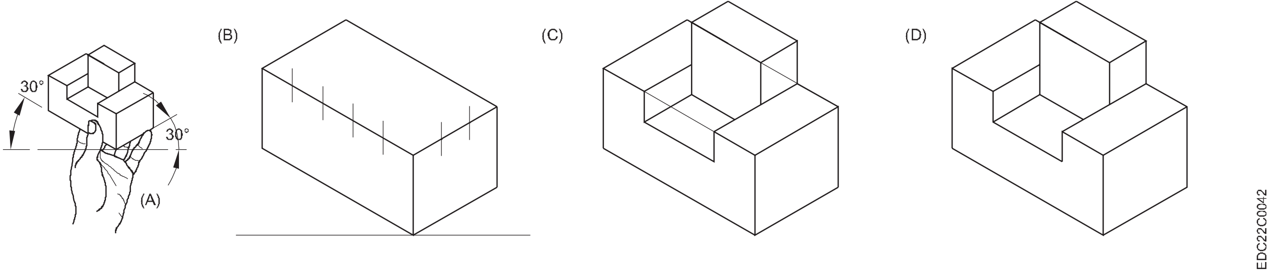
Scale: Sketches, generally are made not to scale. In technical sketching object should be sketched in proportion to the nearest possible size.

Grid (Cross-section) paper provides a ready scale that may be employed to help in drawing/sketching to correct proportions. The size of the drawing sketch depends on the complexity of the object and the size of the paper.

Isometric drawing/sketching

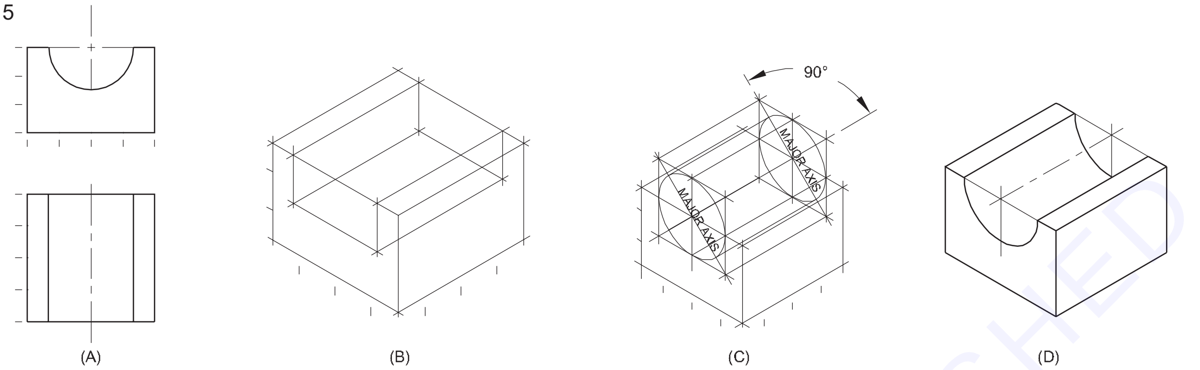
- To make an isometric drawing/sketch from an actual object/component hold the object/component in hand and tilt towards you as shown in Fig 14A.
- In this position, the front corner will be vertical and in the other two receding bottom edges will be inclined about 30° to the horizontal.
- Sketch the enclosing box lightly as shown in Fig 14B.
- Block the recess and projecting block. (Fig 14C)
- Erase all construction lines with soft eraser. (Fig 14D)

Fig 14



- Sketching an isometric view of a block when two views are given. (Fig 15A)
- Block the given object, including the rectangular space for semi-cylinder. (Fig 15B)
- Block in the box enclosing the full cylinder lightly as shown in Fig 15C.
- Remove all construction lines and dark the all final lines showing only the lower half of the cylinder. (Fig 15D)

Fig 15



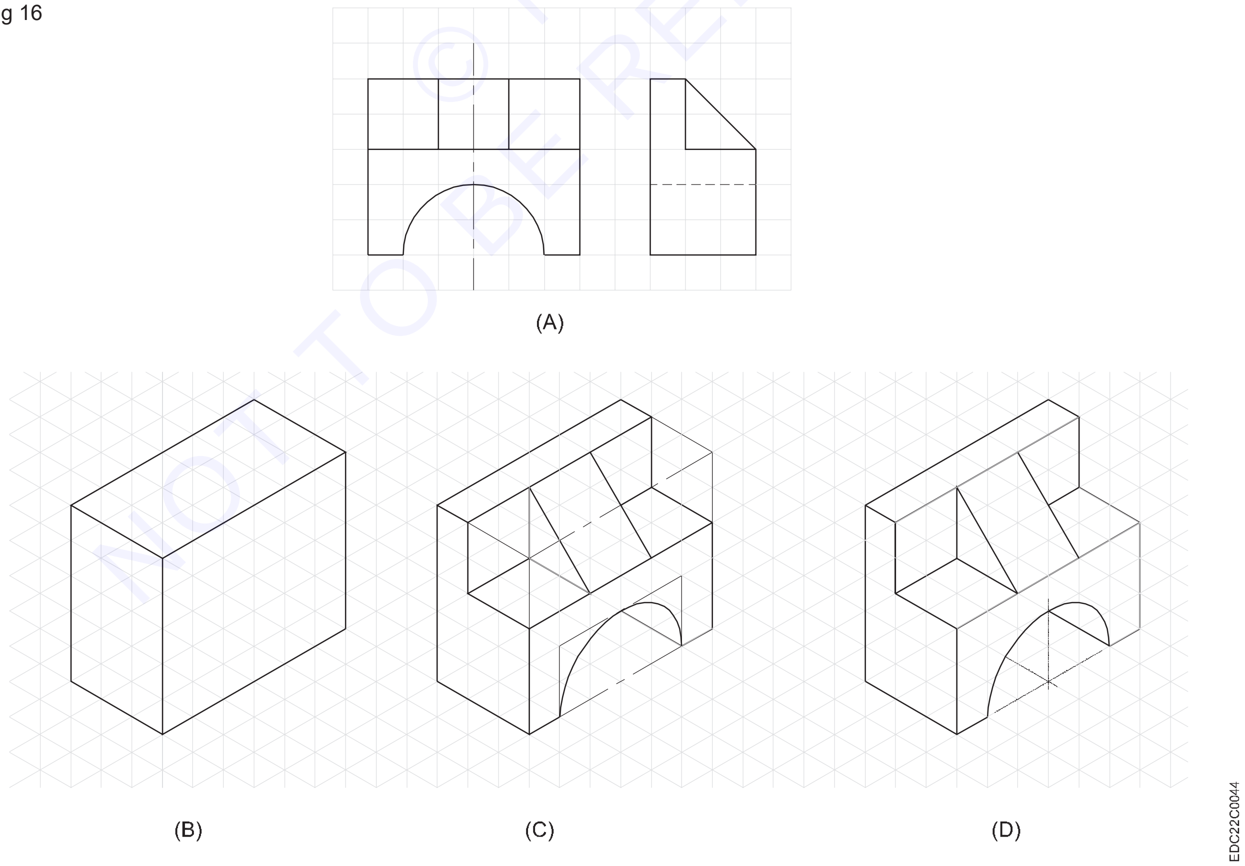
Drawing Sketching on isometric paper: Two views (plan and elevation) of a object are shown in Fig 16A.

By using a isometric paper which gives perfect visual effect of the object. Isometric grid paper placed in work book.

By counting off the isometric grid spaces equal to the squares on the corresponding given views sketch in the enclosing box and also the surface A. (Fig 16B)

Sketch the other surfaces B,C,E etc to complete the isometric views. (Fig 16C & 16D)

Fig 16



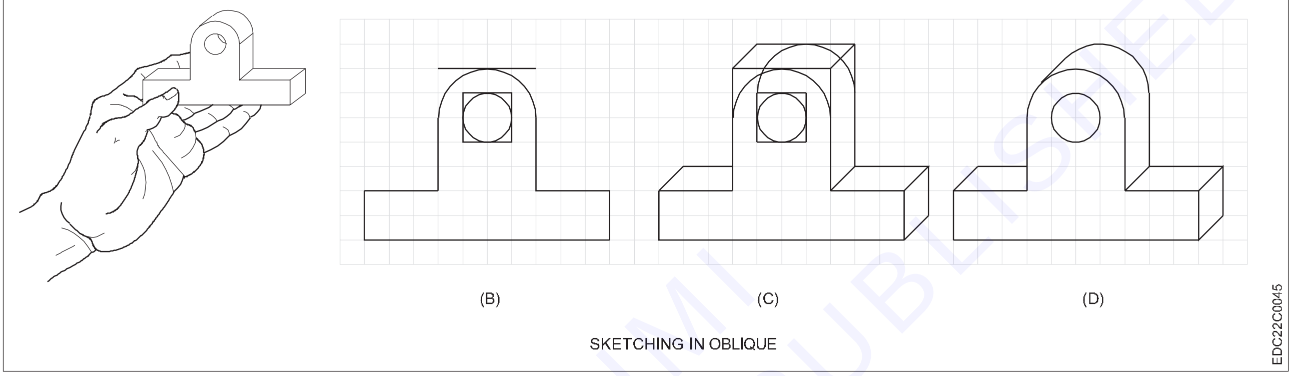
Oblique drawing/Sketching on square grid paper: Fig 17 explains the method of creating oblique on grid paper.

- Hold the object in hand. (Fig 17A)
- Sketch the front view. (Fig 17B)
- Sketch the receding lines at 45° diagonally through squares and complete the object. (Fig 17C)
- Erase the construction lines. (Fig 17D)

In this method dimensions are determined by counting squares.

The receding lines may be equal to the original length of the object or half the original length. If the length of receding line is equal to the original length, sketch or view is a cavelier. If half size it is cabinet.

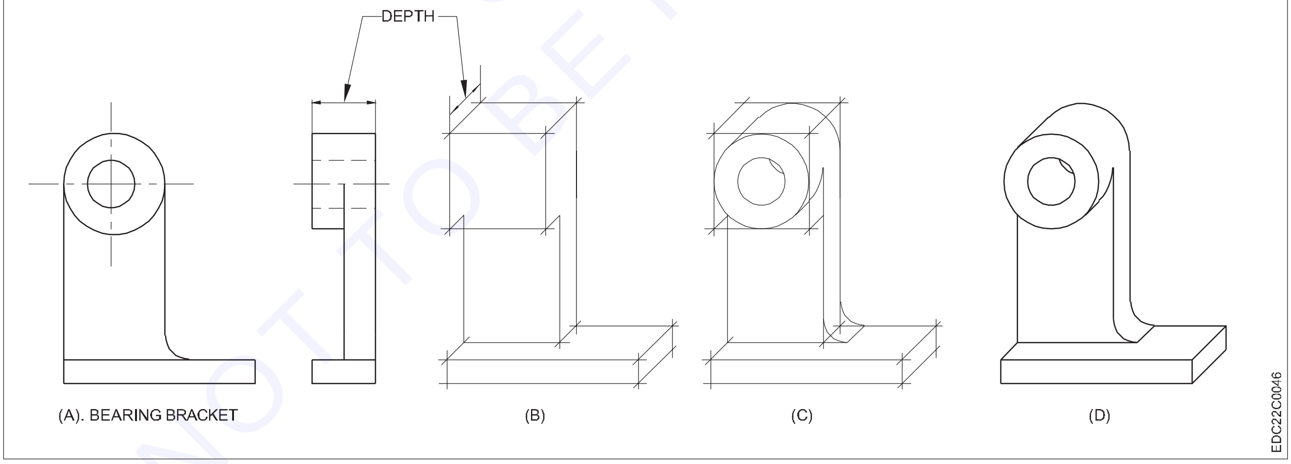
Fig 17



Oblique drawing/Sketch - Another method (Fig 18)**:** Two views of a bearing bracket given. (Fig 18A)

Figs 18B, 18C & 18D are the different stages of doing oblique drawing/sketch.

Fig 18



Drawing sketching multi-views of object: In sketching multi-views of the object the following steps are most important in this endevour.

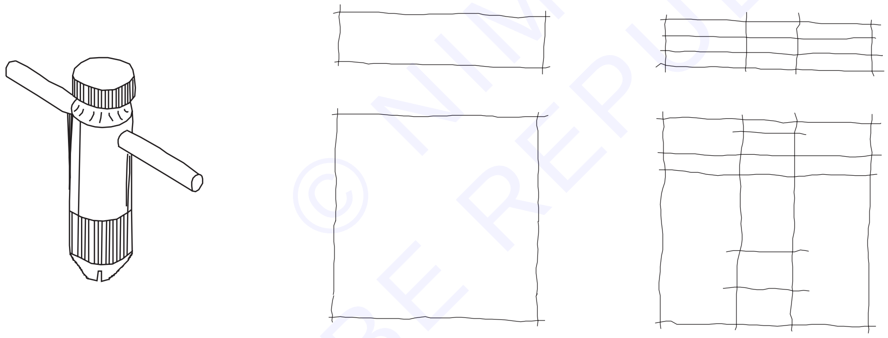
- Visualisation of the object
- Determination of the views (whether 1st andle or 3rd angle)
- Determination of the size of the sketch.
- Location of centre lines
- Allocation of dimensions with proper dimension lines an arrow heads.
- Blocking the main outlines

- Writing titles and any other notes/details with dates
- Checking the drawings.

Before making a free hand technical sketch of a machine part/component the following points is to bear in the mind.

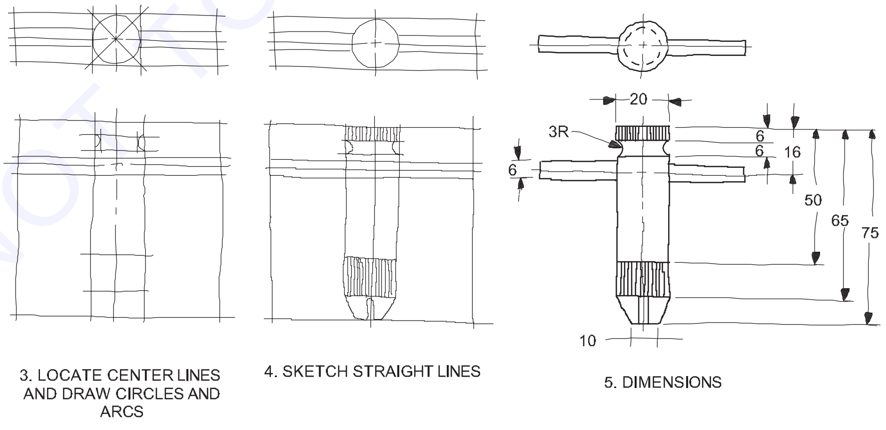
- Before a graphic idea can be developed is essential that the mental image of it to be defiinite and clear.
- When making free hand multi-views sketches, it has to begin by blocking the overall size of each view by using very light lines.
- Establish the length, height and width of each view.
- Locate the centre line of the cylindrical features.
- Locate the centres for arcs and circular parts.
- Circular parts and arcs are sketched first followed by horizontal and vertical lines.
- Additional separate views may be sketched rather than complicating views with added lines.
- A machine component can be represented right side up in its natural working position.
- If symmetrical about an axis mostly one half may be sketched.
- Choose a correct scale for the sketch, so that enough space is provided to show all details.
- The rules and methods are same as applicable to multi-views drawn by using instruments.
- Fig 19 shows a model of free hand sketch for the reference.

Fig 19



1. BLOCK IN THE AREA NEEDED FOR EACH VIEW

2. BLOCK IN MAJOR FEATURES TO GET THEM IN PROPER PROPORTION



3. LOCATE CENTER LINES AND DRAW CIRCLES AND ARCS

4. SKETCH STRAIGHT LINES

5. DIMENSIONS

EXERCISE 4 : Projection

Orthographic projection

Graphics are preferred by engineer's and craftsman to communicate their ideas. When graphics are used for communication it is called graphical language. Those who donot have the knowledge of this language are professionally illiterate.

The saying that "A picture is worth a thousand words" is very much relevant in technical work.

An engineering drawing conveys many different types of information of which the most important thing is the shape of the object. Fig 1 shows a sample drawing. In this drawing the shape of the part is represented by three views.

For an untrained person it will be very difficult to conceive the shape of the object from the above drawing.

But in Fig 2, the same object is shown pictorially in a different ways and the shape is easily understood even by a layman.

From Fig 1 & 2, it is clear that there are different ways of describing the shape of a part on a paper. Figure 1 is called as Multiview drawing or Orthographic drawing and the method adopted in figure 2 is called pictorial drawing. The different views in a multiview drawing are called as 'Orthographic views' or Orthographic projections.

Fig 1

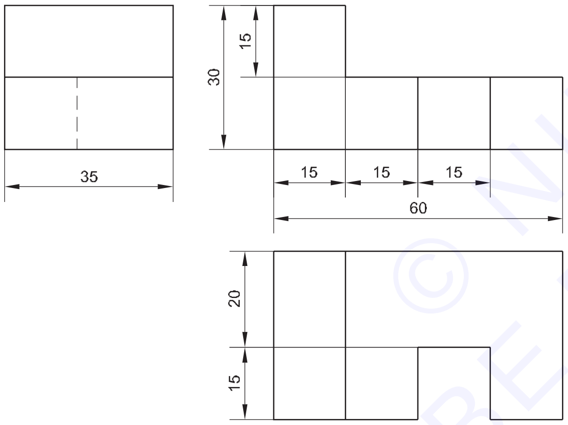
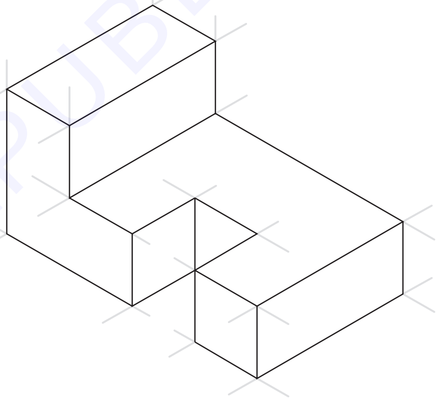


Fig 2



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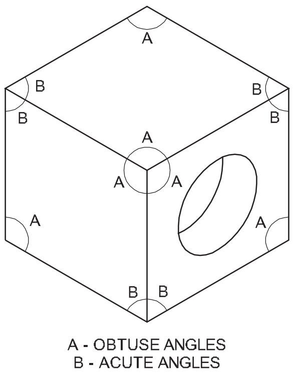
To describe the shape of a part in engineering drawings, multiview or orthographic view method is preferred as only Orthographic view can convey the true shape of the object. Whereas in pictorial drawing through this shape is easily understood and it is distorted. To emphasise this point, see Fig 3, wherein a cube with a circular hole is represented pictorially. We know that all corners of the cube are of 90°. But in the pictorial drawing in Fig 3, the same 90° is represented at some places by acute angles and at some other places by obtuse angles.

Fig 3



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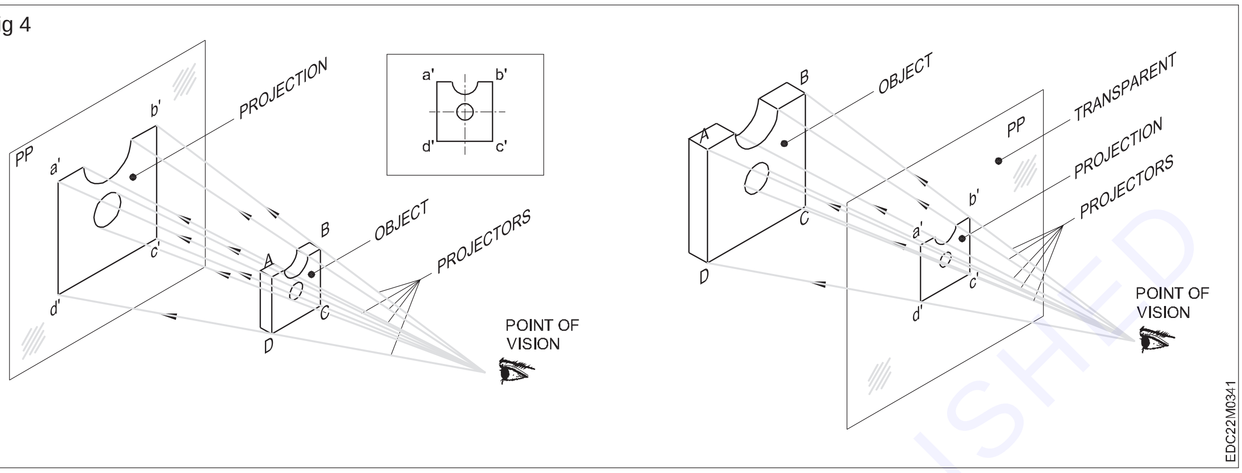
Projection: Projection is commonly used term in draughtsmans vocabulary. In the context of engineering drawing, projectors means image and it is comparable to the image formed on the retina of the eyes. (Projection can also be compared to the image of the object on the screen, where the film is projected (by the cinema projector) by the light rays.

Projection or images can also be formed inbetween the eyes and the object by keeping a transparent plane. (Fig 4)

Fig 4



In this figure 4 the rays from the object converge to the eyes and this image (Projection) is smaller than the object. However if the rays are parallel as in the case of rays coming from the sun, the image (Projection) will be of the same size as that of the objects. Such a projection is called orthographic projection. The parallel lines/rays drawn from the object are called projectors and the plane on which image is formed is called plane of projection. In orthographic projection, the projectors are perpendicular to the plane of projection. (Fig 5)

Orthographic projection: The term orthographic is projection derived from the words, Ortho means straight or at right angles and graphic means written or drawn. The projection comes from the Old Latin words PRO means forward and Jection means to throw. The orthographic projection literally means "Throw to forward", "drawn at right angles" to the planes of projection.

An orthographic system of projection is the method of representing the exact shape and size of a three dimensional object on a drawing sheet or any other plain surface such as drawing board.

A single orthographic view of an object will show only two of its three dimensions. The view in figure 6 shows only the length and height of the object only.

Therefore, it becomes necessary to have an additional view to show the missing dimensions (width). Therefore, we have to make two views to represent the three dimensions of an object.

Fig 5

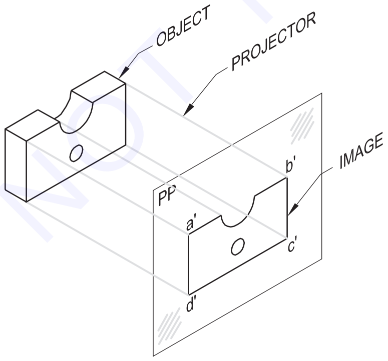
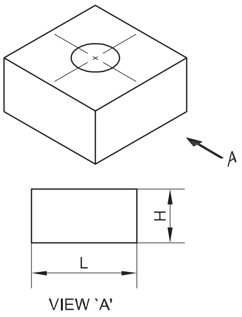


Fig 6



The two views thus required are to be obtained on two different planes which are mutually perpendicular (one HP and one VP) with the object remaining in the same position. The projection or the view obtained on the horizontal plane is called the top view or plan and the view obtained on the vertical plane is called elevation.

First angle and third angle projection: One vertical plane (VP) and one horizontal plane (HP) intersect at right angles to each other. (Fig 7)

All the four quadrants have one HP and one VP formation. As per convention in mathematics, the quadrants are numbered as 1st, 2nd, 3rd and 4th. These four quadrants are called four dihedral angles, namely 1st angle, 2nd angle, 3rd angle and 4th angle.

To draw two views of an object, we assume that the object is placed in any one of the quadrant/angles, 1st angle & 3rd angle Fig 8a, 9a and its plan and elevation projected to the respective planes. Now tomake it possible to draw the two views (Plan & elevation) in one plane i.e the plane of the drawing paper, the horizontal plane is assumed to be unfolded in clockwise direction through 90° Fig 8b & 9b. We proceed this way, when the views are made. When the object is placed in the 2nd or fourth quadrant the plan and elevation will get super imposed (one up on the other) Fig 10a & b. Due to this reason the 2nd and 4th angle are not used for making engineering drawings as the three dimensions cannot be easily identified. Hence for representing the three dimension of the object, we assume the object is placed either in 1st angle and in 3rd angle (Fig 11 & 12) respectively

Fig 7

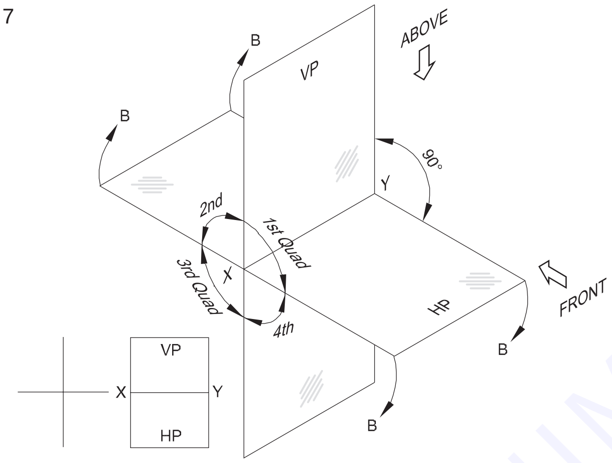
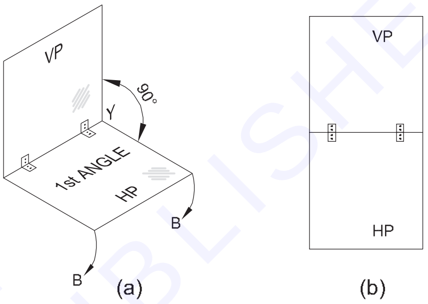
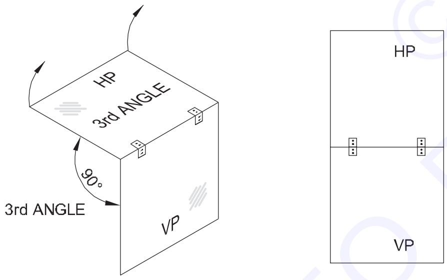


Fig 8



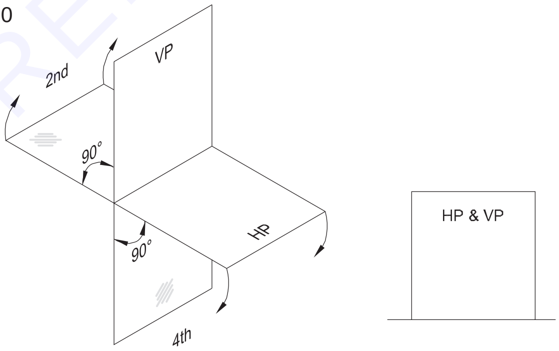
(a) (b)

Fig 9



(a) (b)

Fig 10



(a) (b)

Fig 11

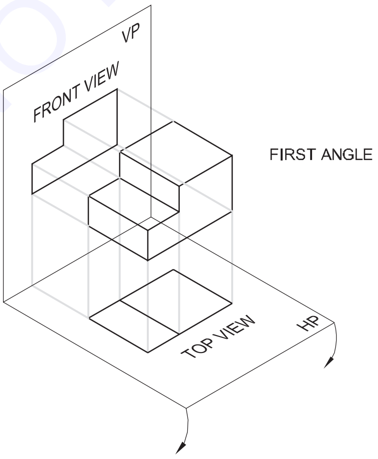
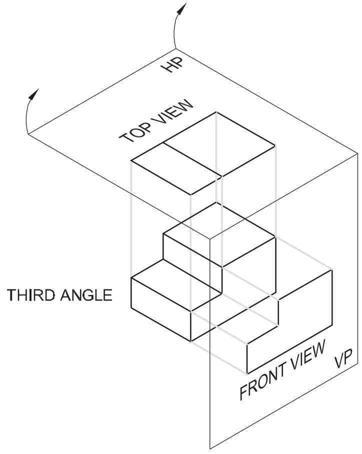


Fig 12



The placement of plan and elevation when the horizontal plane is unfolded will be different in these two systems. It may be observed in Fig 13 that in the first angle projection plan (top views) will be directly below the elevation, whereas in 3rd angle projection plan lies directly above the elevation. (Fig 14)

Views can be drawn in any one of these two methods. However Indian STandard (BIS) has recommended the first angle method to be used in our country.

Fig 13

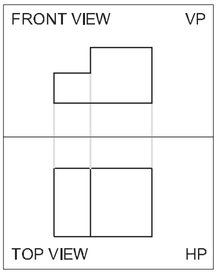
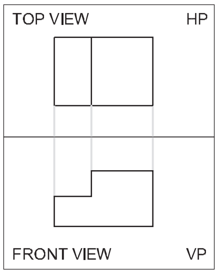


Fig 14



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Geometrical solids

Solids: Solids are the objects which have definite shape, size and occupies certain space. They have three dimensions viz., length, breadth or width and height. According to their shapes. They are classified into two groups.

– Polyhedra
– Solid of revolution

Polyhedra: are solids having (poly-many) more than three flat surfaces called faces. The ends of surfaces meeting with each other are called edges. When the faces are identical to each other, they are called `Regular Polyhedra'. Depending on the number and shape of faces regular polyhedrons are named. Of the many regular polyhedrons three are defined below:

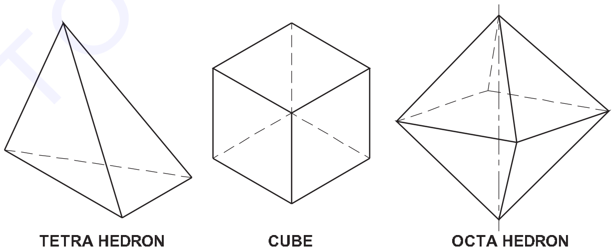
Tetrahedron: A solid having four equilateral triangular faces solid having least number of flat surfaces.

Cube or Hexahedron: A solid having six regular square faces.

Octahedron: A solid having eight equilateral triangular faces. (see Fig 1)

When solids are not composed of identical surfaces, such polyhedrons are either Prisms or Pyramids.

Fig 1



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Prism: Prism is `Polyhedron' having two identical end faces. The top and bottom base surfaces are joined by parallelograms or rectangular surfaces. Imaginary line joining the centre of the end faces is called the axis. The axis is right angles to the end faces. Prisms are in general designated according to the shape of the end faces. Eg. Square, rectangular, triangular, hexagonal, pentagonal, octoganal (Prisms) etc. Prisms are right or oblique, the axis of regular prisms is at right angles to the face. Whereas in oblique prisms the axis is inclined to the end face. (Fig 2)

Pyramids: Pyramids are polyhedra solids having a base surface whose shape may be triangular, square or polygon and as many slant triangular faces as there are sides in the base. All the slant triangular faces join at a common point called APEX.

Similar to prisms, pyramids also are known by the shape of their base viz triangular, square, rectangular, pentagonal, hexagonal etc. The imaginary line joining the centre of the base to the apex is called the AXIS.

Fig 3 shows some pyramids and their views.

Fig 2

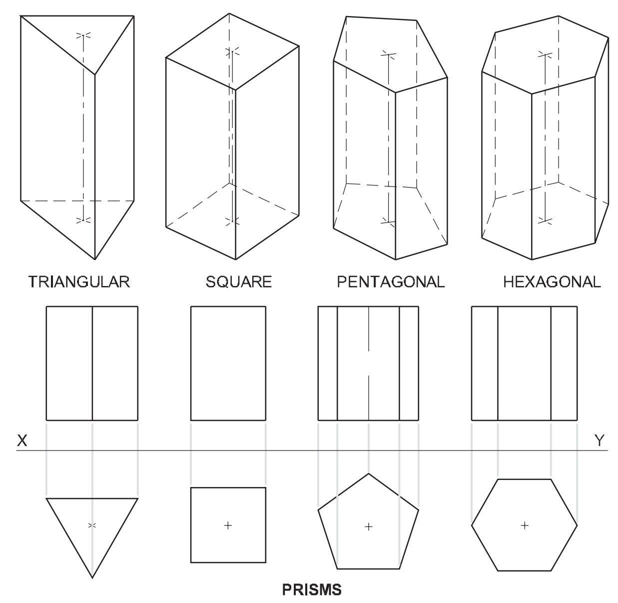
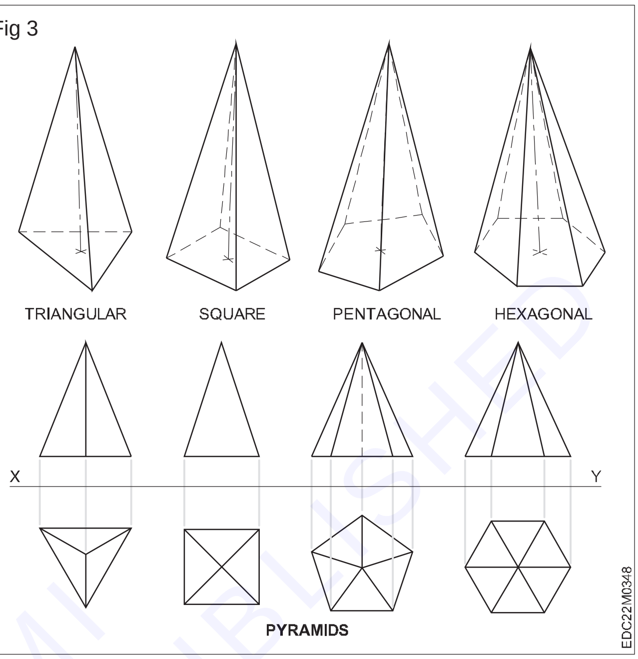


Fig 3



Solids of revolution: When a plane figure revolves about an axis a solid is generated.

Example

The solid shown in the Fig 4 is formed by the revolution is formed by the revolution of the plane (Fig 4A) abc about the axis ab.

Geometrical solids like cylinder, cone and sphere are solids of revolutions.

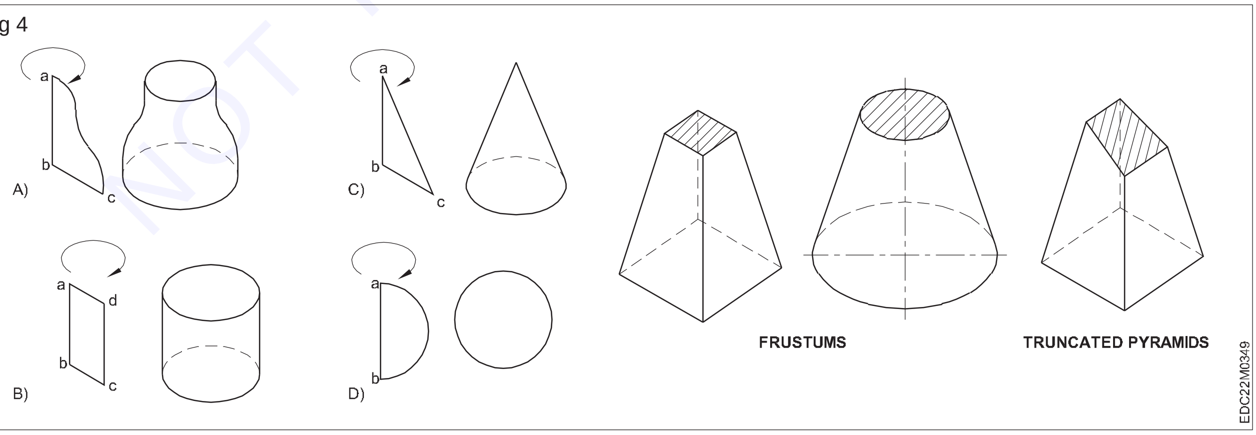
Cylinder: When a rectangle rotates about one of its sides a cylinder is generated.

Cylinder has two flat circular faces and a curved surface. (Fig 4B)

Cone: When a right angled triangle revolves about one of its side forming the right angle, a cone is generated. Cone forming has a circular face and a slant curve surface. (Fig 4c)

Sphere: When a semi-circle revolves about its diameter a sphere is generated. A sphere has no flat surface. (Fig 4D)

Fig 4



The term solids of revolution is a mathematical concept and a physical requirement in geometry.

Frustums: Pyramid/cone is cut parallel to the base and the top portion is removed. The remaining bottom portion is called frustum of a pyramid/cone.

If the cutting plane is at an angle to the axis/base, of the pyramids or cone they are called "Truncated pyramids or cones".

Fig 5 shows frustums and truncated pyramids.

All items we use are solids. Their shapes may confirm to individual geometrical solids like prisms, cones or other combination.

Figure 6 shows some such items.

Fig 5

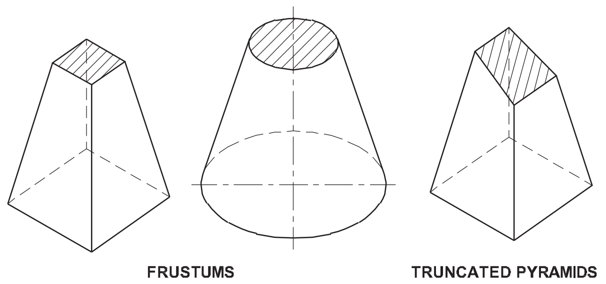
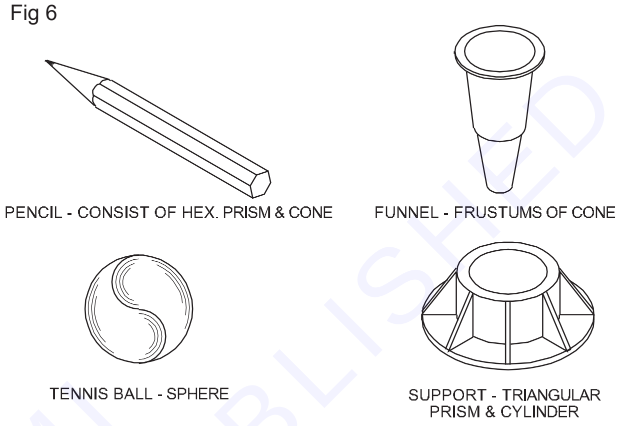


Fig 6



Views of solids: When dealing with projection of plane figures earlier was stated that solids are enveloped by planes and therefore drawing the views of solids would actually mean drawing the views of planes the solids are composed of.

The faces of solids which are parallel to the planes of projection will be seen in true shape in the respective planes. When planes are not parallel to the plane of projection the views will have a disorted look.

Figures 7,8,9,10,11,12 & 13 indicate the plan, elevation and end view of some solids for the position defined against each.

Fig 7

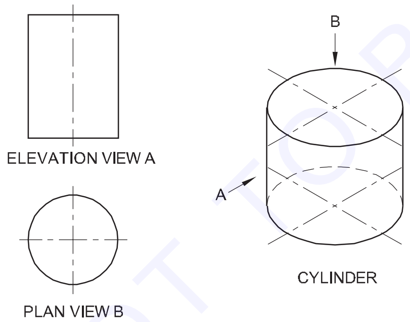


Fig 8

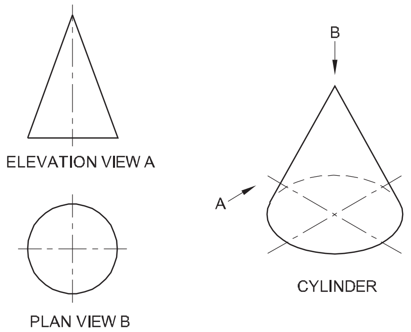


Fig 9

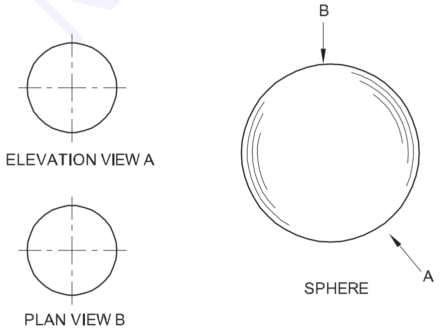
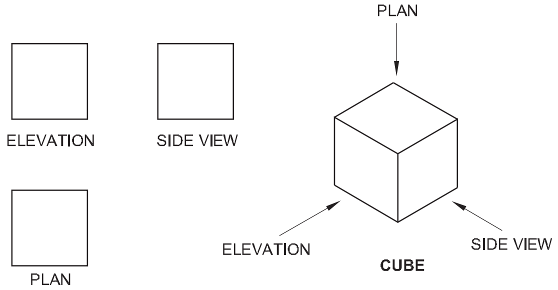


Fig 10



While drawing the views of solids all the edges of solids may not be visible in the views concerned. For example in the figure shown the edge will not be visible in the front view. Such edges are referred as hidden or invisible edges. All visible edges in a view are drawn usually thick lines. But, invisible edges are drawn using dotted lines of medium thickness. (The thickness of dotted lines is inbetween thickness of thick lines and construction lines) Dotted lines are short dashes.

In some cases it is required to show the axis of solids. Axis is represented by another type of line called centre line. Centre lines are thin lines consisting of alternating, alternate long and short dashes. (Figs 14,15 & 16)

Fig 11

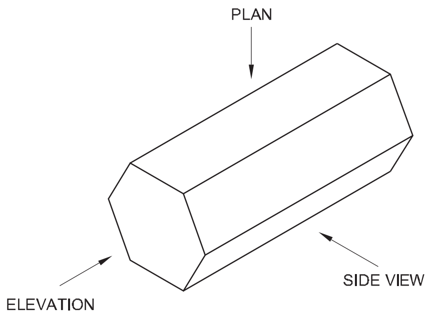


Fig 12

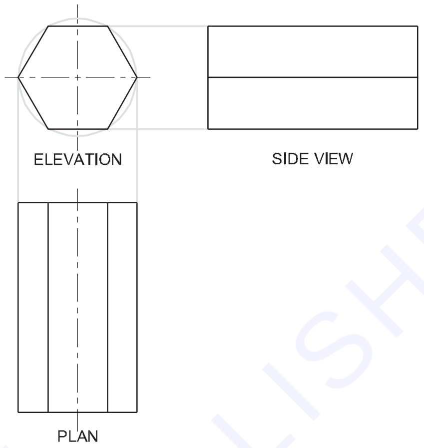


Fig 13

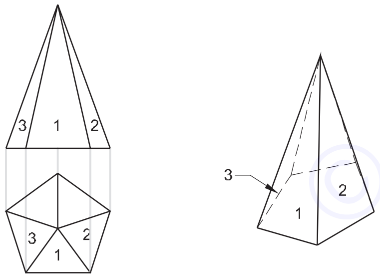


Fig 14

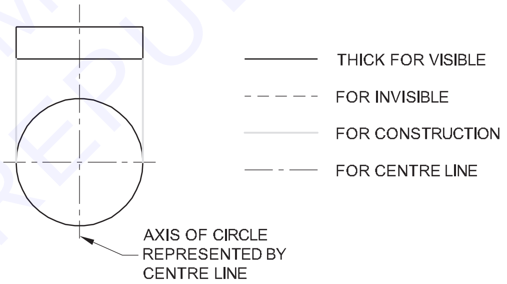


Fig 15

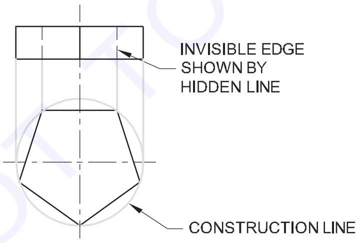
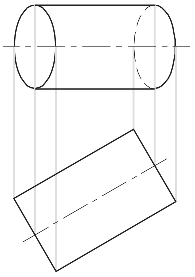


Fig 16



PLAN AND ELEVATION OF THE CYLINDER IN INCLINED POSITION. THE SURFACES ARE SEEN ELIPTICAL IN THE ELEVATION

Projection of plan figures

Two dimensional and three dimensional figures: We know that solid object are enveloped by surfaces, while solids are classified as three dimensional surfaces and implies volume and two dimension implies area.

When we draw orthographic views to represent solids in effect, we are drawing the projection of the solids.

Types of surfaces (Fig 1)**:** Surfaces may be flat or curved. Flat surfaces are also referred as planes. (Plain surfaces) Flat surfaces, depending on their orientation, may be vertical, horizontal or inclined. Fig 1 shows a solid and it has flat surfaces and curved surfaces. Flat surfaces are marked as F_1,F_2 etc.

Surfaces F_1,F_4,F_6,F_8,F_9,F_{10},F_{12},F_{13} and F_{14} are vertical surfaces.

F_2, F_7 and F_{15} are the horizontal surfaces.

C_1,C_2 and C_3 are the curved surfaces.

F_3, F_5 and F_{11} are inclined or oblique surfaces or their combination.

For example in F_3 is rectangular while F_{13} is circular. But surface F_1 is a combination of several plane figure.

Projection of Flat surfaces: While drawing the projection of surfaces (plane figures) the following points should be noted.

If the surface is parallel to the plane of projection, the resulting projection will be the true shape of the surface. (Fig 2)

Fig 1

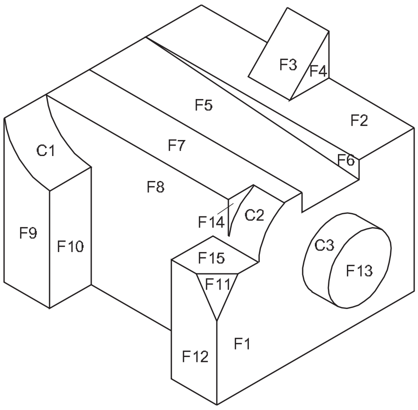
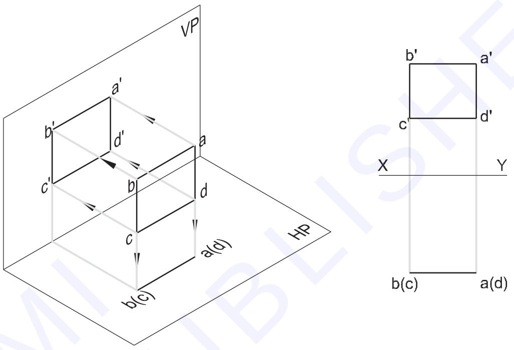


Fig 2



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True shape: When the projection of a surface is identical to the surface projected, the projection is said to be of true shape.

When the surface is perpendicular to the plane of projection, the resulting projection will be a straight line. (Fig 3)

If the surface is inclined to the plane of projection, its projection will not have the true dimensions. They are fore shortened. (Fig 4)

Foreshortened view: Where the projection of a surface is not identical to the surface projected, the projection is said to be foreshortened.

In figure 4, the length pq or the length on is of true length in plan, but in front elevation and in side view same is foreshortened in a different way according to the inclination of the surface to the plane of projection.

If a surface is inclined to a vertical plane, the angle of inclination will be seen on HP and vice-versa. (Fig 4)

Fig 3

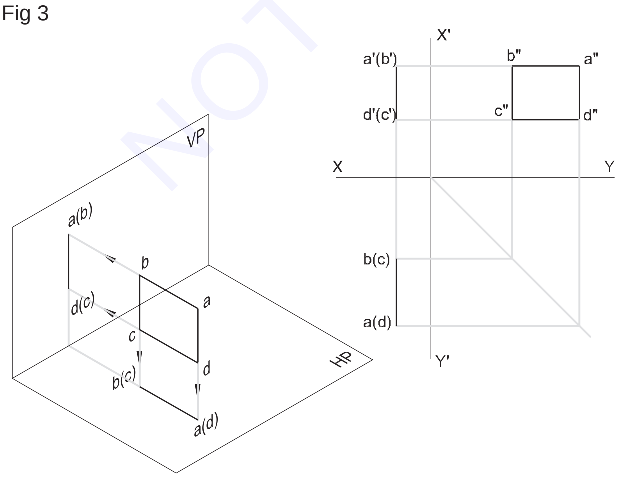
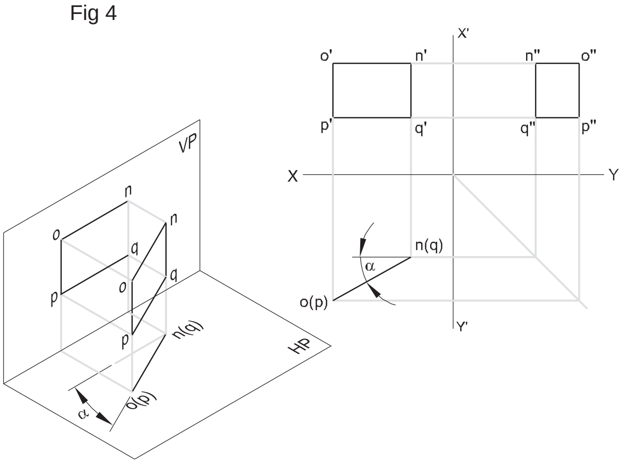


Fig 4



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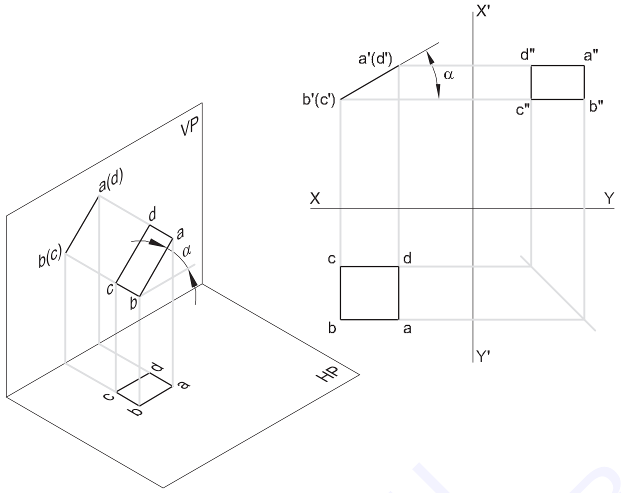
If a surface is inclined to horizontal plane the angle of inclination will be seen on VP and vice-versa. (Fig 5)

Guidelines to be followed: The intersection (folding lines) between HP and VP is marked as XY whereas the intersection between VP and AVP is marked as X'Y'. (Fig 5)

In exercises/problems wherein the distances of the object (point, line, surface) from HP, VP and AVP are not given but a convenient distances may be assumed and followed.

Fig 5



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Terminology of views/projections:

- The view projected on HP is termed as plan or top view.
- The view projected on VP is termed as elevation or front elevation or front view.
- The view projected on AVP is termed as side view or end view or side elevation or end elevation.

Visualisation

We had done the exercises on projection of plane surfaces and solids with the ultimate objective of applying it to describe the shape of an object on plain paper. We know that a surface parallel to VP will be seen in true shape in front view and as lines in side view and plan. Similarly the surfaces which are inclined to a plane will be seen foreshortened.

The example shown in figure illustrates the above points in a practical situation. The object in Fig 1 has eleven surfaces. Of them A,B,C,D,G,L & M are rectangular in shape where as E,F,H,K are combination of basic geometrical shape (rectangle and triangle). (Fig 2)

Fig 1

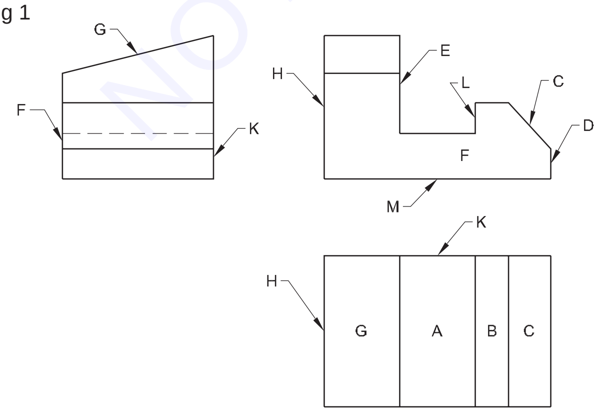
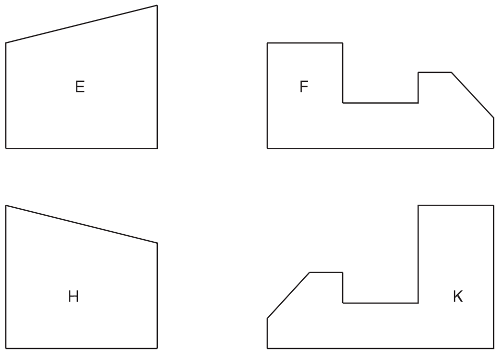


Fig 2



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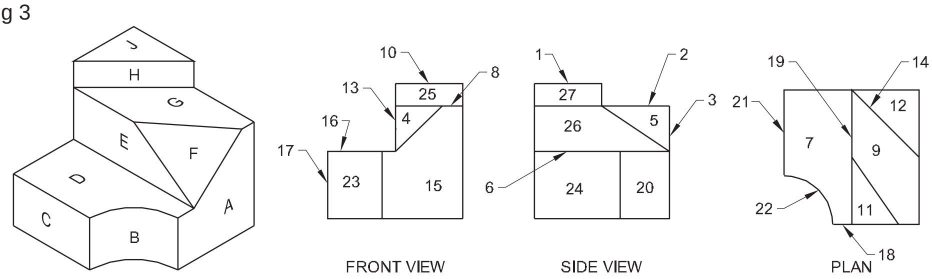
As per the principle explained in orthographic projection, all the 11 surfaces of this object will have its representation in all the three views. For example surface will be seen as surface in front view where as it will be seen as single line in end view and plan and so on. These principles are further illustrated. In Fig 3 both the isometric view and multi views of a block of an object are given. The table given along is to identify the line that indicates the various surface. Note that the surface 'F' in isometric view is as surface itself in all the three views. The inclined surface 'H' is seen as a surface in end view and front view. Similarly the curved surface 'B' is seen as flat surface in front view and end view. Only in plan it is seen as a curved line.

Fig 3



	A	B	C	D	E	F	G	H	J
F	15	23	17	16	13	4	8	25	10
P	18	22	21	7	19	11	9	14	12
S	3	20	24	6	26	5	2	27	1

EDC22M0360

Visualisation

In the context of engineering drawing visualisation means studying a multi view drawing and forming a mental image of the shape of the object.

It has been mentioned before, multi view drawing has no 3D effect and do not convey the shape in the first sight itself. One has to recall the principles of projection and analyse each view and do his own reasoning in order to conceive the shape.

Visualisation is definately not that easy when it comes to difficult shape. A craftsman makes simple drawing, but it is not an indication of his ability to visualise the shape of the object.

In order to manufacture a component, the shape of the component must be understood in unmistakable terms and all those who are associated with manufacturing must be able to visualise the part from its multiview drawing. Ability to read drawing quickly is an added advantage to any engineer/technician and it will help them to go up in professional ladder.

Visualisation procedure: There is no one set of rules to read and visualise the shape of an object from the given multiview drawing. Visualisation is essentially the reverse of the mental process that is gone through for obtaining the shape of the object.

To begin with, choose the view that tells the shape of object most. If no view could be set to be choosen, start from any view. Consider the Figs 4 & 5.

Fig 4

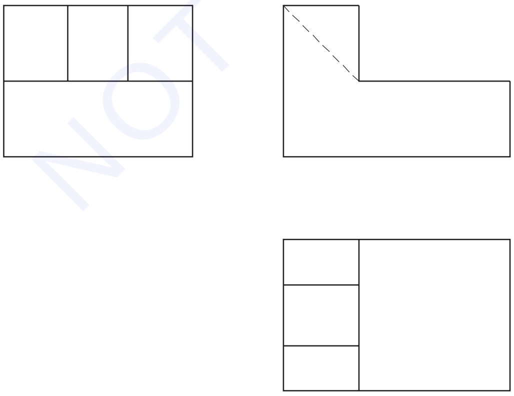
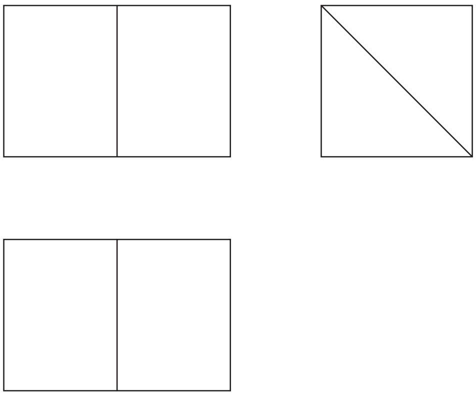


Fig 5



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In Fig 4 we can choose the front view first. In this view overall shape of the object is reasonably reaches the mind. Whereas in Fig 5 we start from any view. None of the view is specifically telling anything special. Using the detail available in the view so choosen imagine a shape that fits the view most. For example the front view in Fig 4 but for the inclined hidden line the shape would have been as at Fig 6a.

The presence of the hidden line in the front view gives rise to three different possibilities as shown at Fig 6.b,c,d. Now on comparison with the other two views we can eliminate the possibilities that do not match with side view and plan of the object. After the elimination exercise Fig 6b matches with the side view and plan and hence the figure at 6b is the isometric view of the object.

Modelling: When it is difficult to form a mental image one can resort to modelling by means of isometric sketching. This can be done in two ways. Get the area dimensions of a prism Fig 7a that encompasses the object represented by multiview and then draw its isometric view by cutting away the portion of the prism as required till we satisfy with the given views. (Fig 7 b & c).

Instead of cutting away, we can also form the object by adding or joining method as shown in (Fig 8 a,b,c)

Yet another way to assist in visualisation is to model the object using suitable material. For example thermocole, soap, modelling clay etc. are very handy for this purpose. Modelling can also be done either by cutting away technique or by joining technique.

Constant practice is the only key to improve the visualisation ability or skill and hence there is no limit to the number of exercise one has to do to achieve this goal. It may noted that sometimes a well experienced person may get stuck while reading some complicated drawings.

Taking into account the importance of this aspect different types of exercises are devised. Some of them are

- identifying lines and surfaces
- adding views to the given views
- drawing isometric views using multiviews
- identifying missing lines
- finding number of views to the given one or two views
- question answer type

You will not come across the problems given in this exercise in any job drawings/industries. But these are given purely to improve the capability of visualising the shape of any object from the given multiviews.

Fig 6

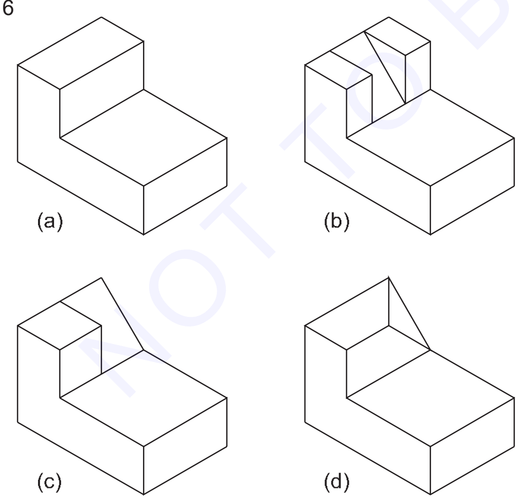


Fig 7

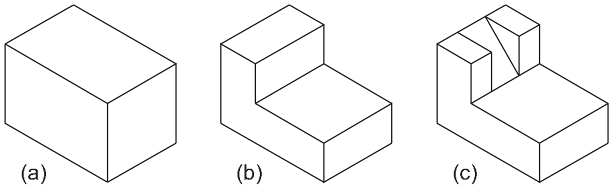
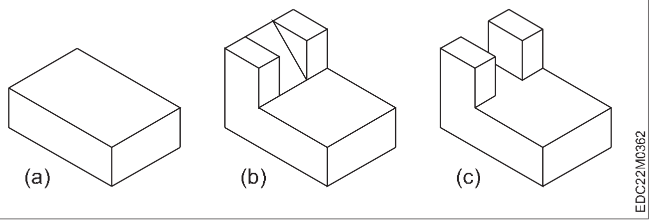


Fig 8



Orthographic projection

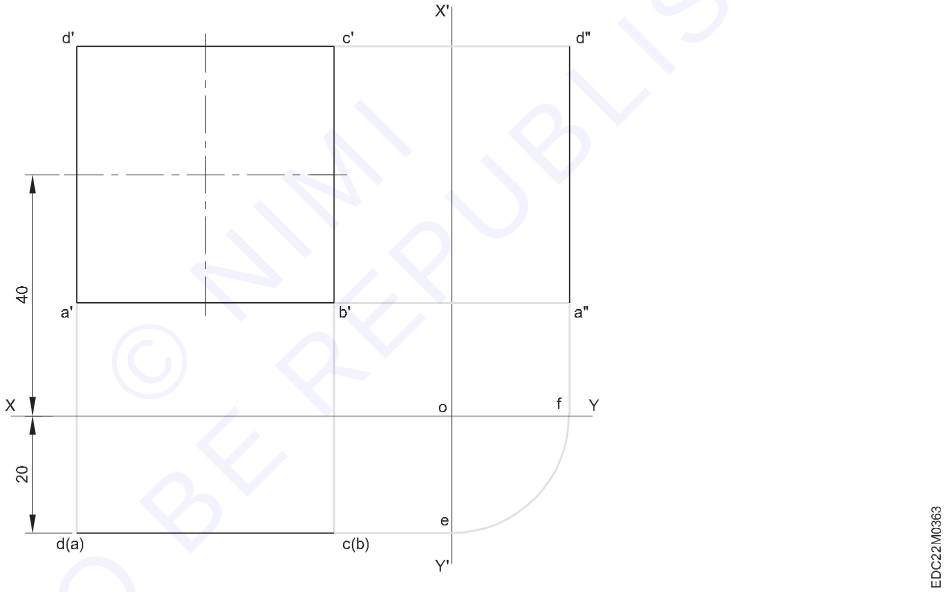
Procedure

Exercise 4.1

Draw the orthographic projection (elevation, plan and side view) of the square sheet (40 mm side) kept perpendicular to HP and parallel to VP. (I angle) (Fig 1)

- Draw the xy line.
- Draw the square with its centre 40 mm above the xy line and one edge parallel to xy line.
- Mark the corners of the figure a', b', c' & d'. This will be the elevation of the square.
- Draw the vertical projectors from a'b' downward beyond the xy line.
- Draw a horizontal line dc at a distance of 20 mm below the xy line. Line dc will be the plan.
- Draw a X'Y' line at a convenient distance from b'c', intersecting the xy line at `0'.
- Project the plan to the X Y line meeting at e.
- By arc method transfer Oe to xy and mark the point `f' at a" and d" respectively. Now the line a"d" is the side view.

Fig 1



Exercise 4.2

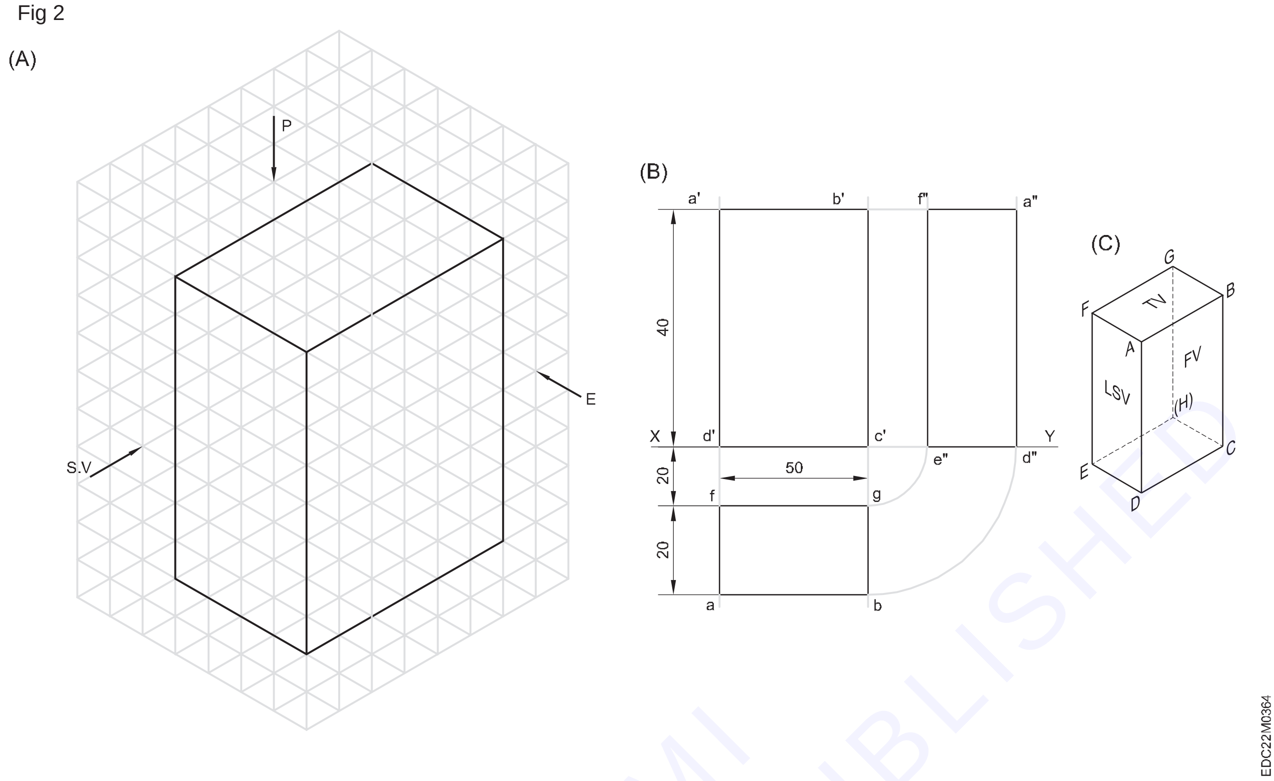
Draw the plan, elevation and side view of the rectangular prism of base 30 x 20 mm and height 40 mm. (Fig 2A)

Note: In this exercise the faces of prism are parallel to the planes of projection. Therefore all the lines orthographic projection are vertical and horizontal lines only.

Visualise the shape of the object and imagine the shape description of views. Surface ABCD (Fig 2B) only visible from the elevation. At the same time all the four sides of ABCD are isometric lines. Therefore in the elevation a rectangle of 40 x 30 mm is seen.

- Draw xy line of convenient length. (Fig 2B)
- Draw a rectangle a'b'c'd' on the xy line. This will be the elevation of the prism.
- Project the vertical sides of the elevation (a'd' and b'd') downwards beyond xy line.
- Draw a horizontal line fg approximately 20 mm below xy line.
- Draw a rectangle fgba of 30 x 20 mm size. This will be the plan of the prism.
- Project points b' and c' horizontally a convenient length to the right side of the elevation.
- Transfer the width of plan gb by arc and locate points e"d" on the xy line.
- Project e"d" vertically up and locate points f"a"d"e" is the left side view of the prism.

Fig 2



Exercise4.3

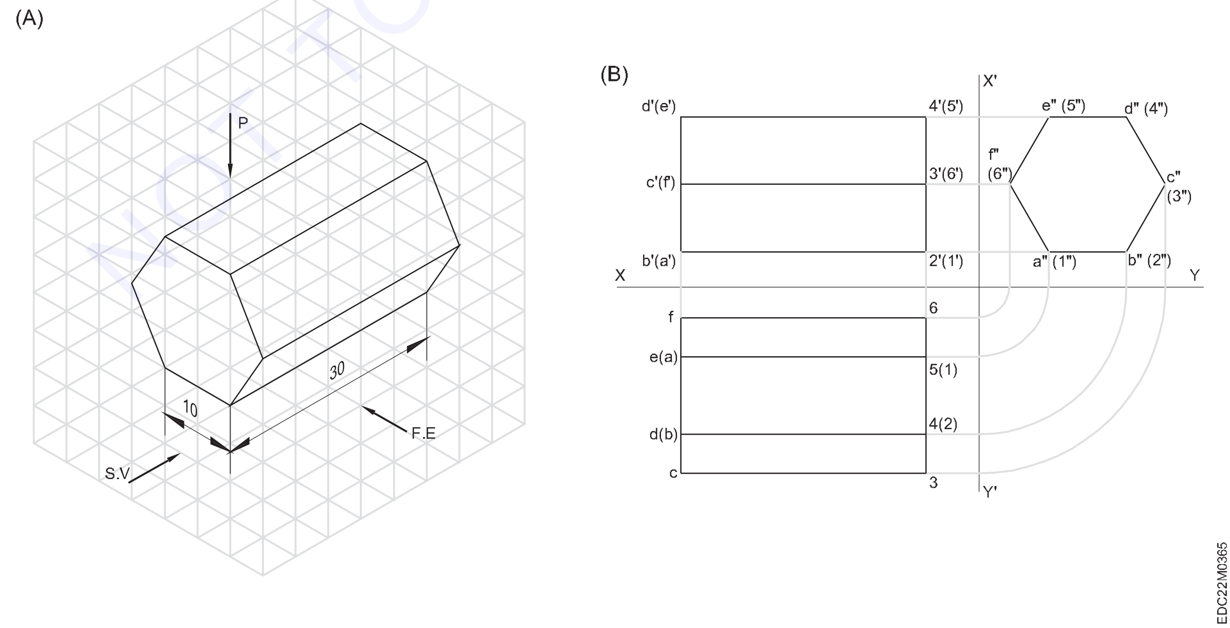
Draw the three views of the hexagonal prism shown in Fig 3A.

Note: From the position described above, it is clear that the hexagonal face of the prism is parallel to AVP. Therefore the end view is a true hexagon and hence this view should be drawn first.

- Draw the end view (hexagonal of side 25 mm) with one side on HP line. (Fig 3B)
- Draw horizontal projectors from side view and complete the elevation. (in the elevation two lateral faces are visible, but they are fore shortened)
- Draw projectors from elevation and side view and complete the plan.

(Three lateral faces are visible of which one is of true shape and the other two are fore shortened)

Fig 3



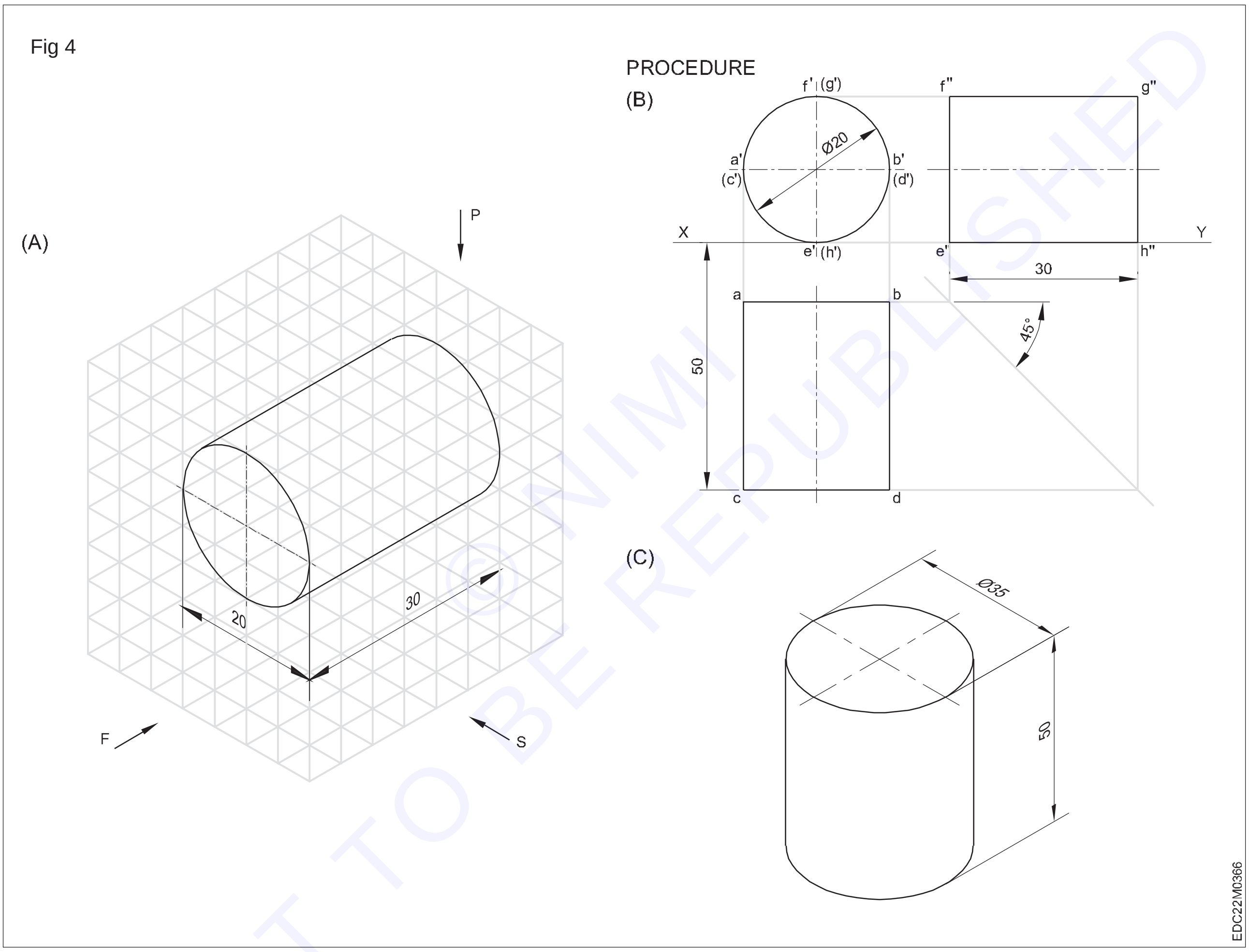
Exercise 4.4

Draw the plan, elevation and side view of a cylinder of diameter 20 mm and length 30 mm. (Fig 4A)

Note: In this problem the circular faces are parallel to VP. Therefore the elevation is a circle resting on XY line. Plan an end views are rectangles of size 30 mm x 20 mm.

- Draw the circle of diameter 40 mm touching XY line. (Fig 4B)
- Draw the plan projecting it from the elevation.
- Draw the end view by drawing projection on it, from the plan and elevation.
- Draw the plan, elevation and side view of a cylinder whose base diameter 30 mm and height 50 mm when its position is as shown in Fig 4C.

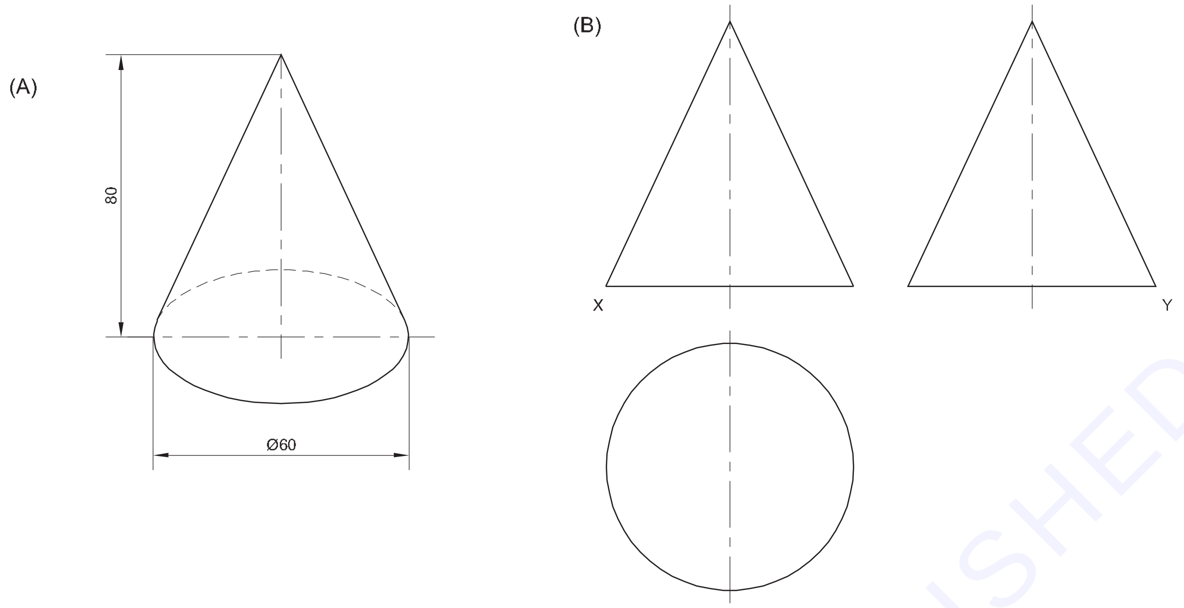
Fig 4



Exercise 4.5

Draw the multi-views of the cone shown in the Fig 5A. Follow the procedures of the earlier exercises and draw the multi-views. (Fig 5B)

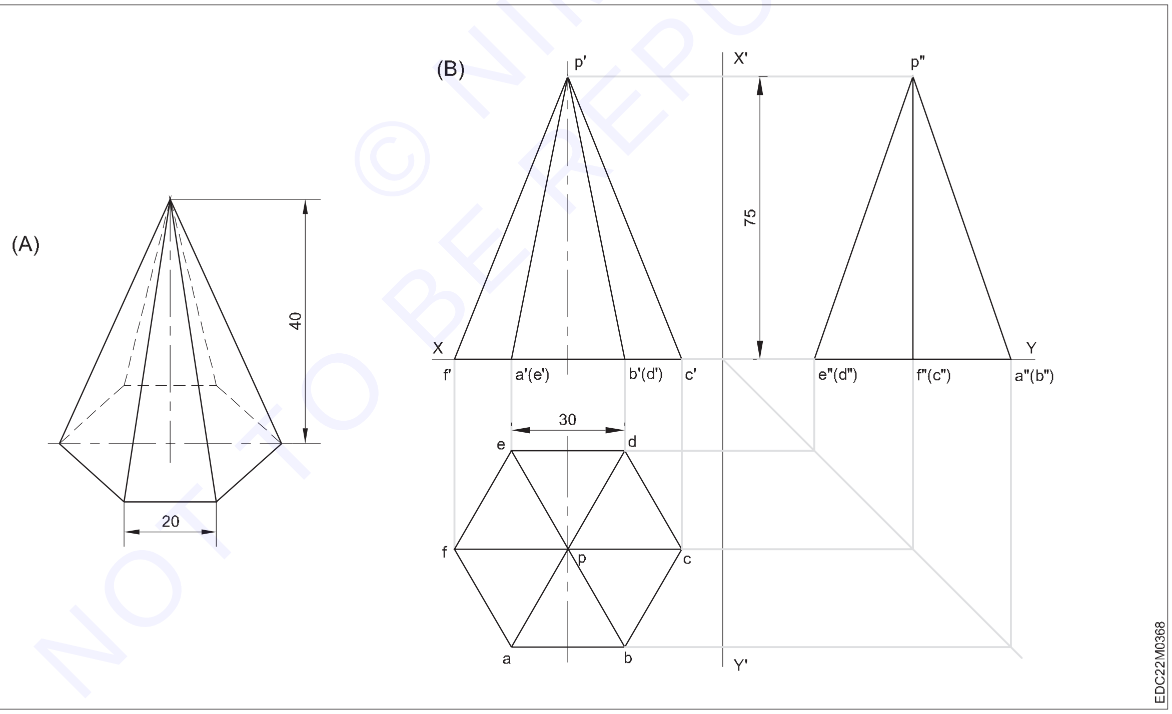
Fig 5



Exercise 4.6

Draw the Orthographic views of a regular hexagonal pyramid of side 20 mm and height 40 mm given its position as below. (Fig 6A)

Fig 6



– standing vertically with its base on HP and one side of the hexagonal base parallel to VP.

Note: The pyramid has 6 triangular faces and one hexagonal base. The plan will show the true shape of the base and other six triangular faces are fore-shortened.

In this elevation, three triangular faces are seen and all of them are fore-shortened.

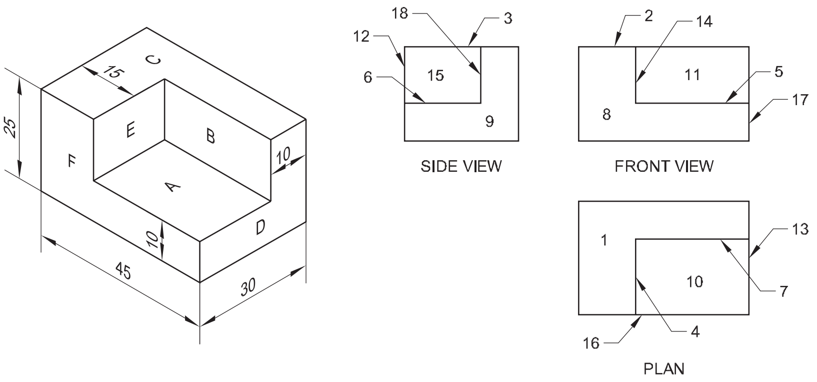
- Mark the centre of hexagon (Point P) and draw lines from P to the six corners of the hexagon. Now this is the required plan. (Fig 6B)
- Project this P from plan upwards and mark P' at a distance of 75 mm from XY line.

- Mark the points f', a'b'c' etc... on XY line by projecting the corresponding points from plan.
- Join the P' with f', a',b',c' etc and complete the required elevation.
- Draw projectors from elevation and plan to complete the required side view.

Exercise 4.7

Identify the surfaces of the block shown in the isometric view with the corresponding multi-views and fill the numericals in the given tabulation column. (Fig 7)

Fig 7



	F	S	P
A	5	6	10
B			
C			
D			
E			
F			

Note: The surfaces are parallel and perpendicular to the plane of projection.

- Study the isometric view and the corresponding multi-views carefully.
- You may observe that the surfaces seen in one view are represented by lines in other two views.
- In this exercise the surface `A' shown in isometric view is seen as a line in the front elevation and numbered as `5' in the front view of the corresponding multi-views.
- The same surface `A' in the side view is seen as a line and numbered `6' in the side view of the multi-views.
- Similarly the surface `A' seen from the top of the isometric views is numbered as `10' in the plan of the multi-views, whereas the full surface area is visible.

Note: When a surface is parallel or perpendicular to the plane of projection vice-versa in a multi-views drawing, full area of the surface will be seen in any one the three views (plan, elevation and side view) and in other two views the corresponding line of the surface will be seen.

Exercise 4.8

Follow the method of Ex.No.4.7 and fill the tabulation column by refering the isometric view and multi-views available in the work book Ex.No.4.8.

Exercise 4.9

Identify the surfaces of the block with slope cuts shown in isometric view with the corresponding multi-views and fill the tabulation column. (Fig 8)

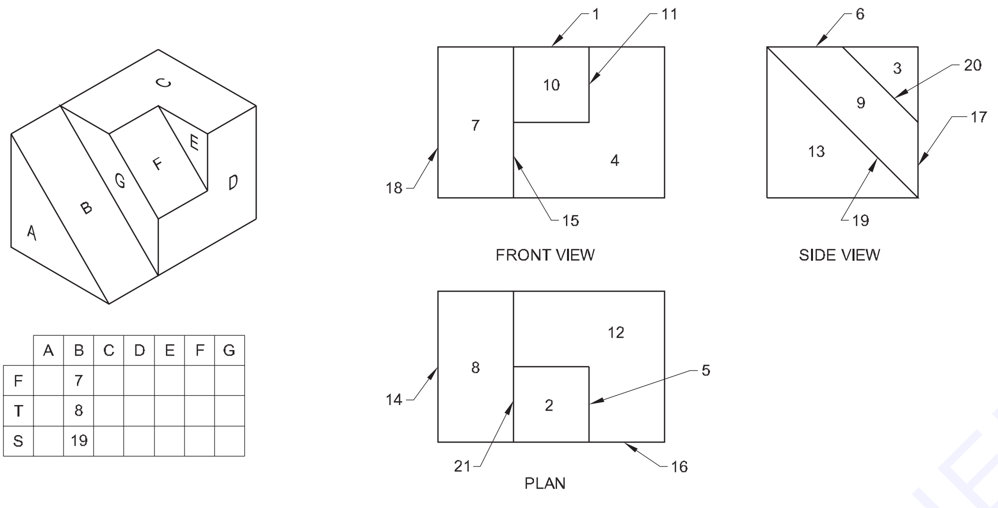
Note: Surfaces B and F are inclined to HP and parallel to VP.

- You may observe that the surface inclined to one plane is seen in other two views as fore-shortened area of the surface and in other view the corresponding line of the surface is visible.
- In this exercise, the surface `B' is seen as a surface in front view and top view, numbered in multi-views as 7 & 8 respectively.
- The same surface `B' in the side view is seen as a line and numbered as 19 in the multi-views which are shown in tabular column. Study the drawing carefully and fill up the other columns.

Exercise 4.10 & 4.11

Follow the procedure of Ex.No.4.9 and fill columns by refering isometric views and multi-views available the work book Ex.No.4.10 & 4.11.

Fig 8



	A	B	C	D	E	F	G
F		7					
T		8					
S		19					

Exercise 4.12 (Fig 9)

Identify the surfaces of the block shown in Isometric view with the correspoding multi-views and fill in the tabulation column. (Fig 9)

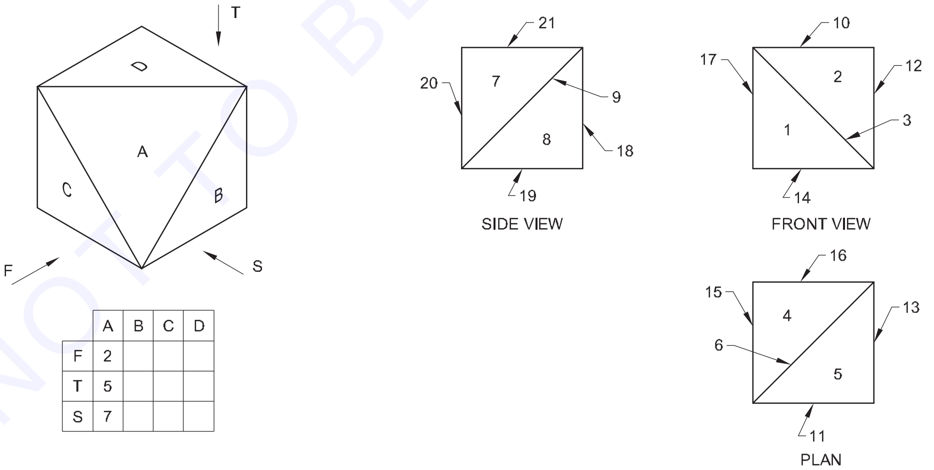
Note: The surface is inclined to three planes HP, VP & AVP.

- In this exercise you may observe that the surface `A' is inclined to all the three planes.
- When you visualise the surface `A' in the front view of the isometric view, the fore-shortened area of the surface is seen and numbered as 2 in the multi-view.
- Seen from top of the isometric view the fore-shortened area of the surface is seen and numbered as 5 in the top view of the multi-view. Similarly in side view it is numbered as 7 in multi-view.

Note: When a surface of the object is inclined to all the planes, the complete fore-shortened surface will be seen in all the three views.

- Fill the other columns and complete the exercise.

Fig 9



	A	B	C	D
F	2			
T	5			
S	7			

Exercise 4.13

Follow the method of Ex.No.4.12 and complete column kept in Ex.No.4.13 of work book.

Exercise 4.14 to 4.31

Draw the exercises available in the workbook 4.14 to 4.31.

Exercise 4.14

Reproduce the isometric view (Fig 10) and also draw the three views in the work book.

Orthographic projection shows the shape of a component by drawing number of views each looking at different side of the component.

A minimum of two views are required to represent a component. In order to clarify clearly the internal and external details a minimum of three views are to be drawn. They are:

- Elevation or Front view or Front elevation. (F)
- Plan or Top view. (P)
- Side view or side elevation or end elevation. (S)

In the Fig 10, surface `A' is only seen when looking at the front of the figure. All the lines in the isometric view are isometric lines. Therefore in the orthographic projection, the front view will be like this. (Fig 11)

In the plan surfaces `C', `D' and `F' are visible and the bottom surface will not visible. The line joining the two surfaces will not visible. The line joining the surfaces will appear in the plan by hidden line. (Fig 12)

Fig 10

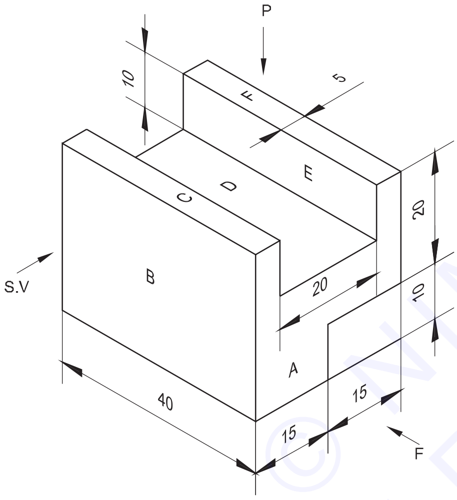


Fig 11

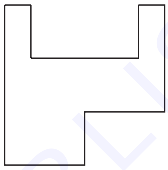
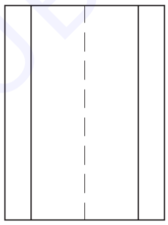


Fig 12



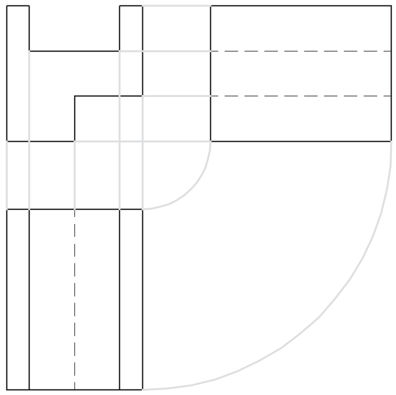
In the side view surface `B' is visible and surfaces `E' and back bottom surfaces are invisible. Due to this reason the line joining `D' and `E', also the line joining back bottom surfaces are appearing in the side view by hidden line. (Fig 13)

Arrange views as stated earlier with uniform gap between views. (Fig 13)

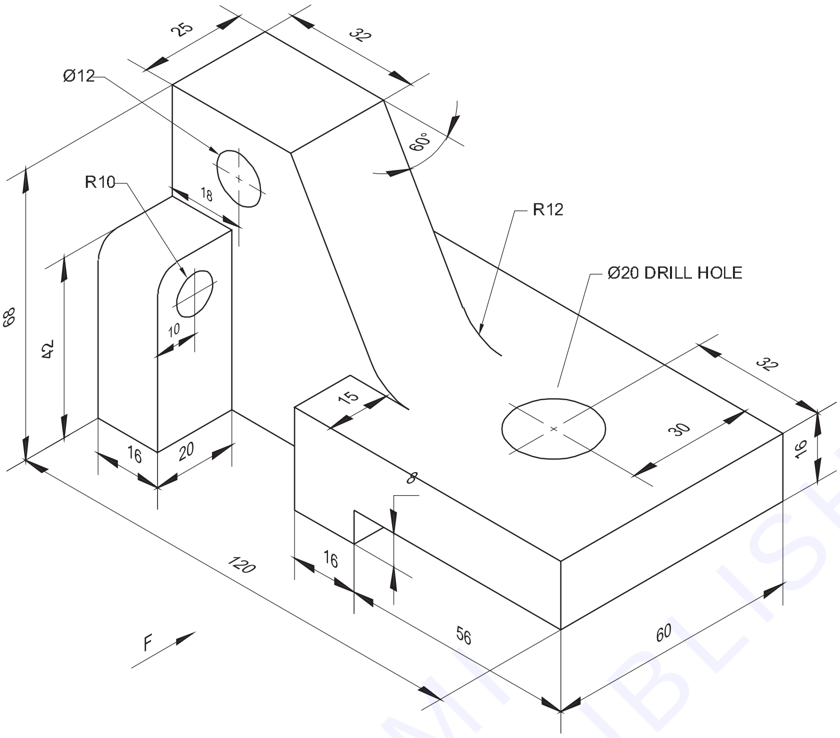
Exercise 4.32 to 4.41

Draw the three views of the following ten exercises in A3 drawing paper separately. Also prepare the title block on each paper as given the theory of earlier exercise. Assume the missing dimensions if any suitably.

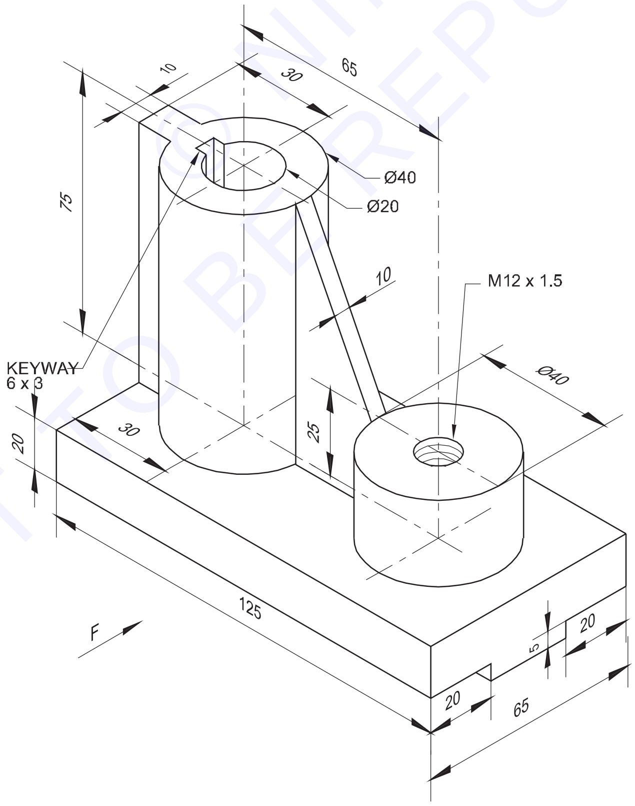
Fig 13



Ex.4.32



Ex.4.33



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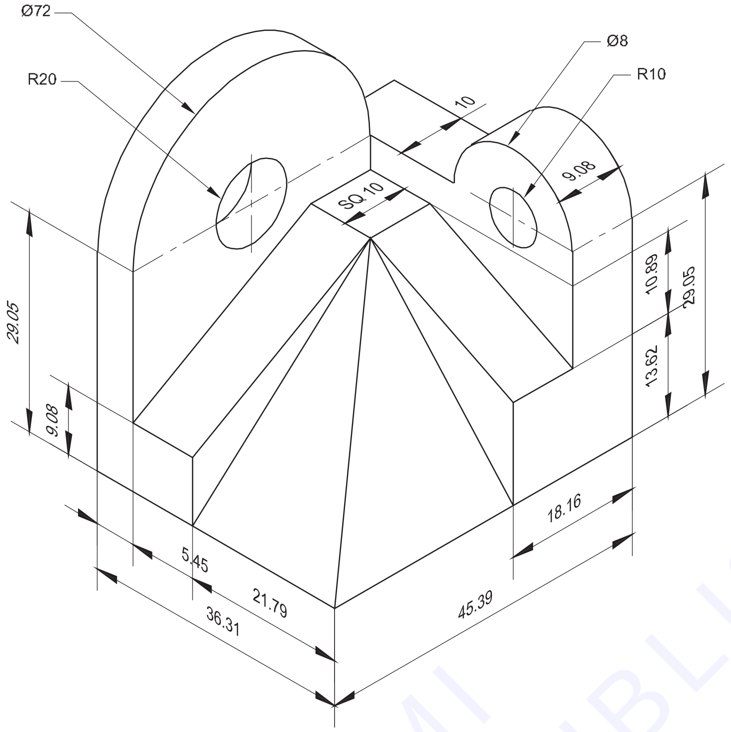
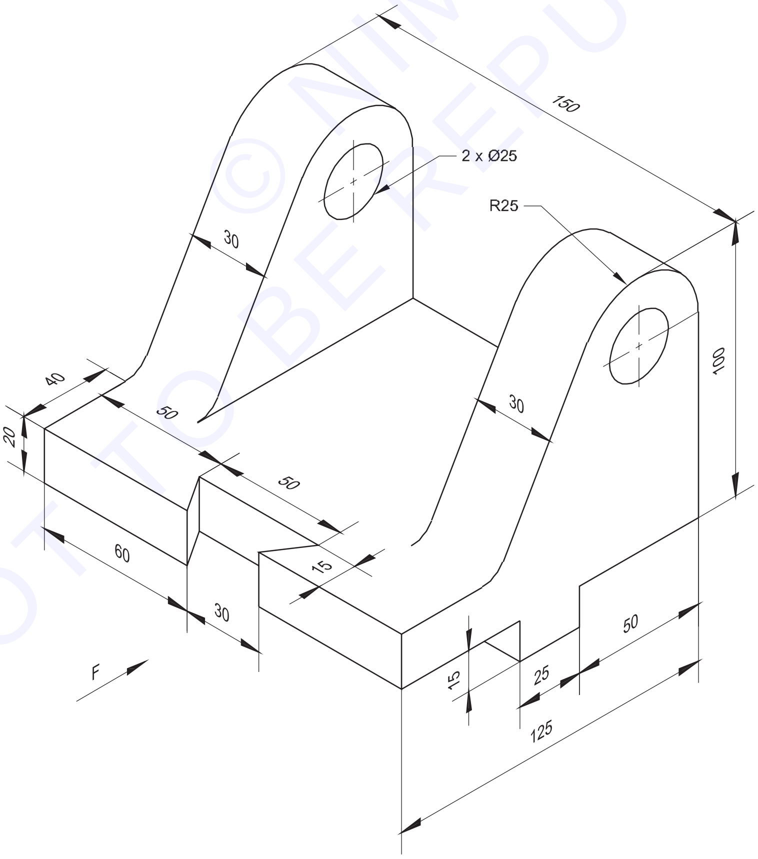
Ex.4.34



Ex.4.35



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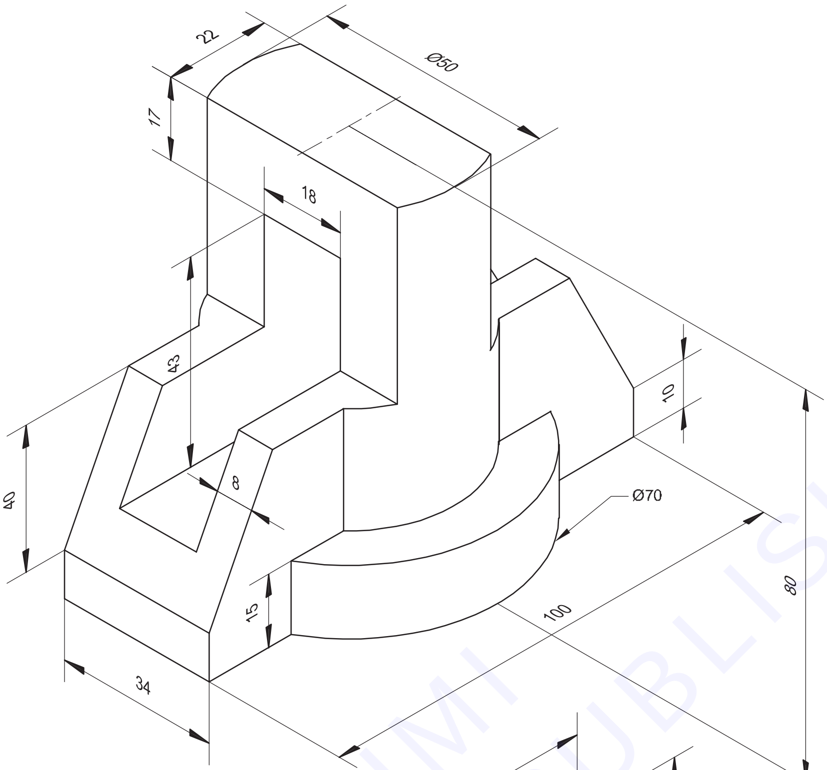
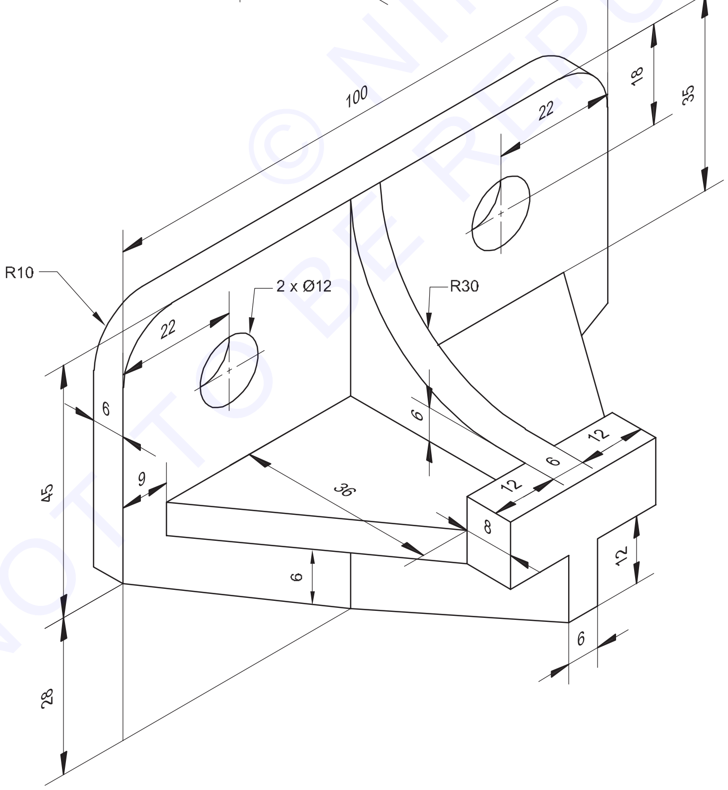
Ex.4.36



Ex.4.37



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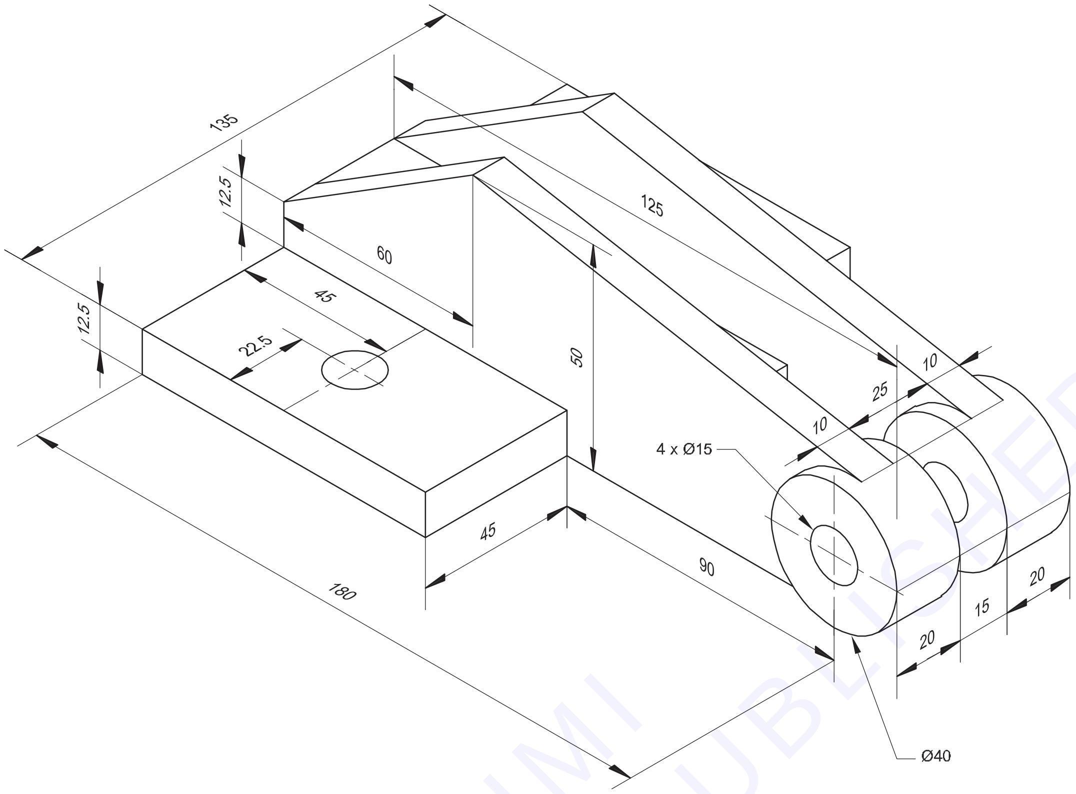
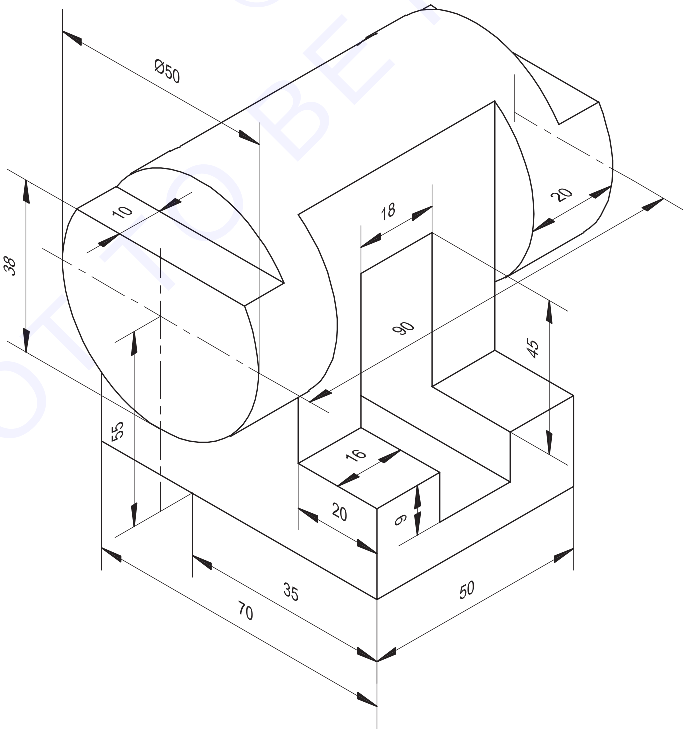
Ex.4.38



Ex.4.39



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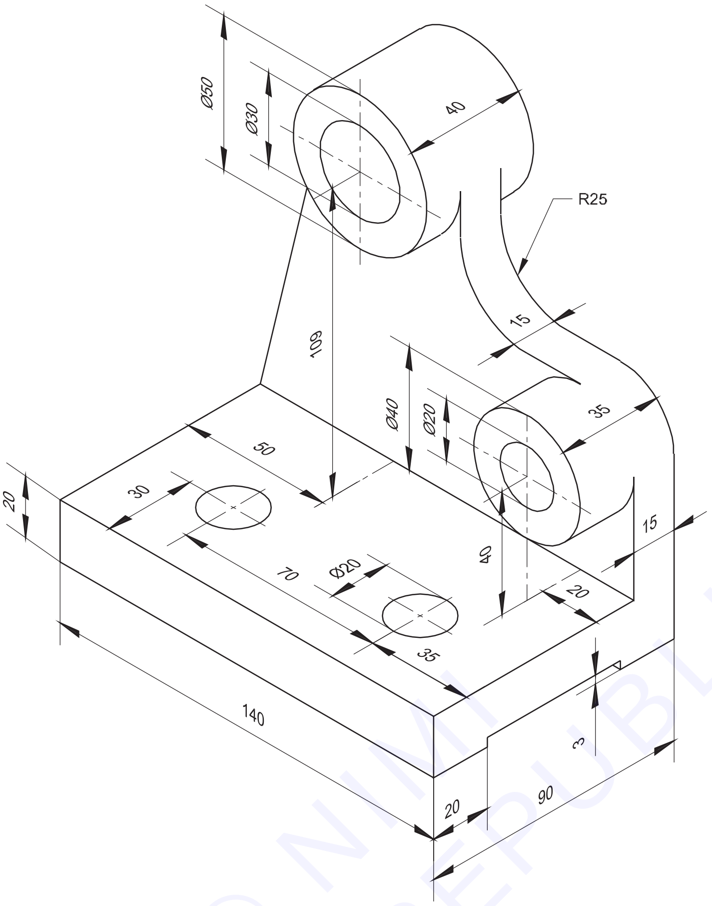
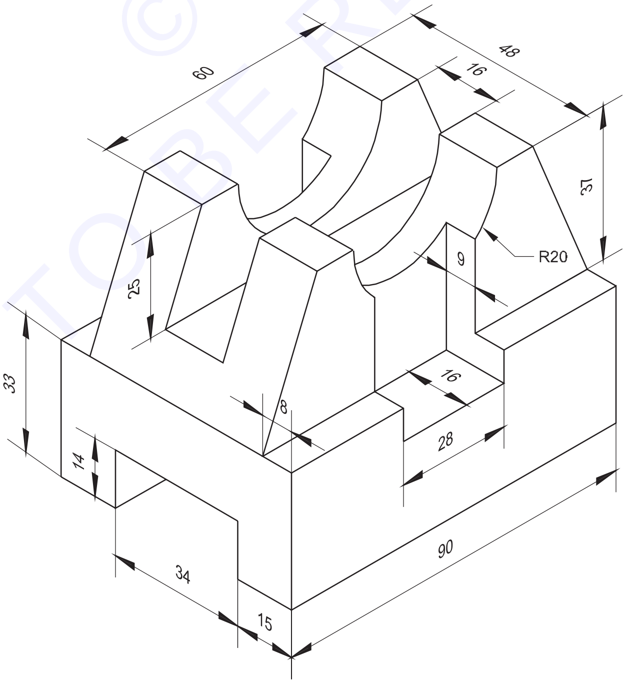
Ex.4.40



Ex.4.41



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Pictorial projection

Pictorial projection: It is possible to show all the 3 faces/dimensions of an object in one view itself. Such orthographic views are called pictorial drawings or pictorial projections. To get the pictorial drawing the object (say a cube) has to stand on one corner such that 3 of its mutually perpendicular faces are inclined to the plane of projection. (Fig 1)

Fig 1

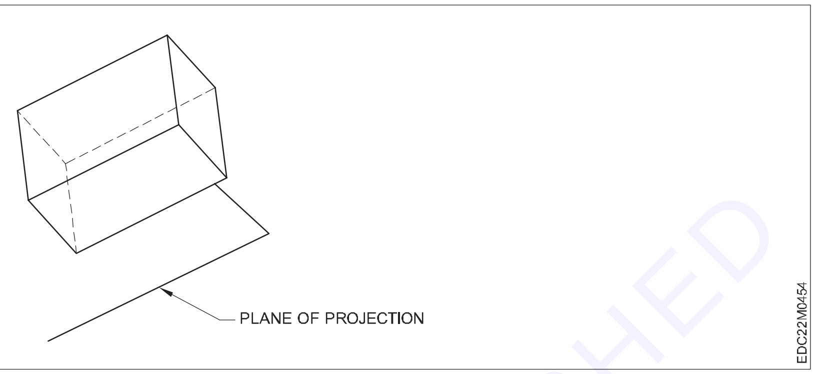
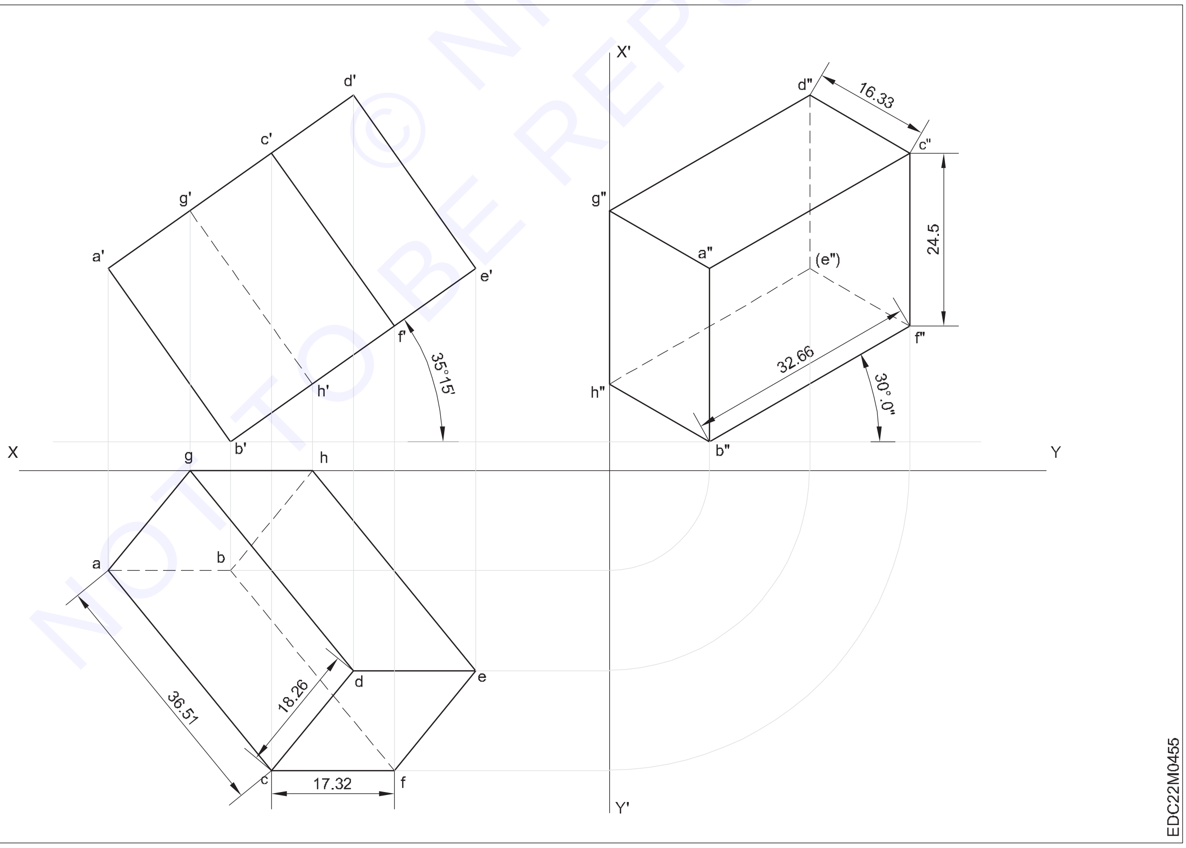


Fig 2 shows the front view, plan and side view of a rectangular prism positions in the manner stated above. Here notice that two of the views (Plan and side view) lock like solids, the reason being that in each of these two views we can see the three faces of the prism. So in this example both the plan and side view are pictorial views in its own right.

Depending on the angle of inclination of the faces with the plane of projection, pictorial projection are classified as Timetric, Dimetric or Isometric.

Fig 2



In trimetric projection (Fig 3a) the three faces make unequal angles with the plane of projection whereas in dimetric (Fig 3b) projection 2 faces make equal angles. In isometric projection all the three faces make equal angles. (Fig 3c)

The projections - trimetric dimetric and isometric projection are generally grouped in one heading called "Axanometric" projection.

In the three types (trimetric, dimetric and isometric) of pictorial projections mentioned above, because the faces of the object are not parallel to the plane of projection the views will not show the true size and shape of the object. The shapes are distorted and lengths of edges are fore shortened. Referring to Fig 2 it may be seen that the true dimensions of the prims is 40 x 30 x 20. But in the front view these dimension measure 32.66 x 24.5 x 16.33 and in plane this corresponding measurements are 36.51 x 17.32 x 18.26. The reason for different lengths in front view and plane is that individual faces make different angles to their respective plane of projection.

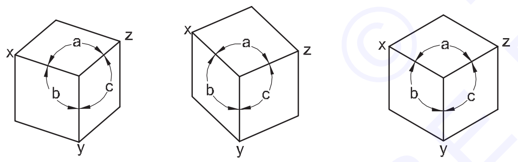
Pictorial projections will enable even a common man to understand the shape quickly, even though these pictorial views have a distorted look. In any case, these views are very useful for describing the shapes.

Out of the three types of axanometric projections, isometric views are preferred due to an advantage and hence it is dealt in more detail.

Isometric projection: In an isometric projection the three mutually perpendicular faces make the equal angles with the plane of projection The term isometric is derived from the Greek word ISO means equal and metra means measurement.

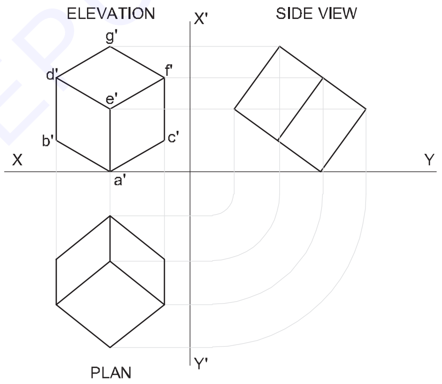
The projection of a cube, the three faces which make equal angles with vertical plane is shown in Fig 4. Here the front view is the isometric projection. Notice that the a'b', a'e', c'f', e'f', c'd', d'g', g'f', b'd', a'c' which represent the various edges of the cube are of equal lengths meaning that all have the same amount foreshortening. Because of this reason isometric projection will give a more natural appearance than trimetric and dimetric and this is extra advantage of isometric projection.

Fig 3



(a) TRIMETRIC (b) DIMETRIC (c) ISOMETRIC

Fig 4



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Isometric projection - Method of construction: We can make the isometric projection of any object using the principle of orthographic projection. But the method is best understood by constructing the isometric projection of cube or rectangular prism. The position required for isometric projection may be brought about as follows.

Place the cube on HP such that two of its mutually perpendicular faces make 45° will VP (the plan and side view elevation in the position will be as in Fig 5).

Next tilt the cube towares you with the corner b on HP. Tilt the solid diagonal DE will be at right angles to VP. Now the 3 mutually perpendicular faces will make angles (35°16') with HP. The three views of the cube in this position are shown in Fig 6. Now the elevation will be the isometric projection. To obtain this proceed as follows:

– First reproduce the side view in Fig 5. Such that DE is parallel to XY line.

– Project from the above side view and the plan in Fig 5 shall be reproduced in Fig 6.

– Draw the elevation

Note: In figure 6, a plan for the tilted position is also drawn. But this is not an isometric projection. Actually it is a dimetric projection.

It may be observed from the above construction that the isometric projection gives 3d (3 dimensional) effect as we are able to show the length, breadth and thickness in the same view. However, making isometric projections this way is complicated and time consuming. Therefore, simpler method have been deviced to make pictorial drawing that are identical isometric projection and these methods are discussed below.

Fig 5

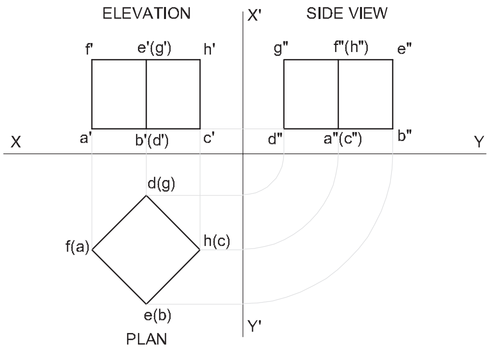
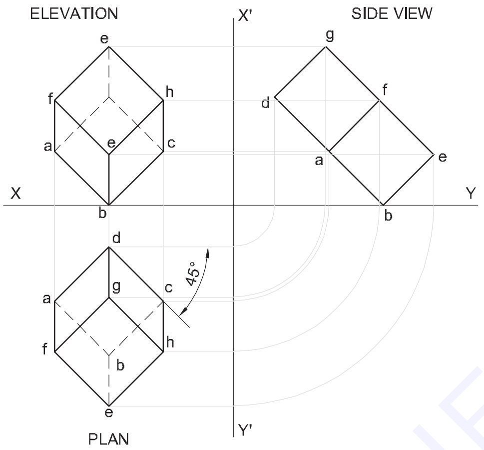


Fig 6



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Simpler method of isometric projection: On analysing the isometric projection in Fig 6, it will be seen that three mutually perpendicular edges of the cube are at an angle of 120° to each other. These three lines which represent the mutually perpendicular edges are called isometric axes. (Fig 7)

So to draw the isometric projection say of a cube, we firstdraw the three mutually perpendicular edges as in figure and set other lengths. Since of the lengths are foreshortened in isometric projections we must use an “isometric scale” to find out the foreshortened lengths. Thereafter, other edges are drawn parallel to the respective isometric axes to complete the figure.

Instead of drawing the isometric axes, first we can also start from the point `a’ .(Fig 7) At this point also 3 mutually perpendicular edges meet. While two of these edges make 30° to the horizontal, the other edge is vertical. (90° to horizontal) After drawing the two 30° lines one vertical line the parallel lines are drawn to complete the cube. Few other objects drawn this way are shown in Fig 8. The length of each edge of corner will be less than the true dimensions and it can be determined by using an isometric scale.

Fig 7

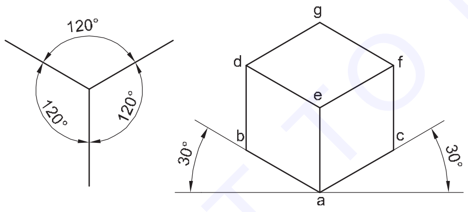
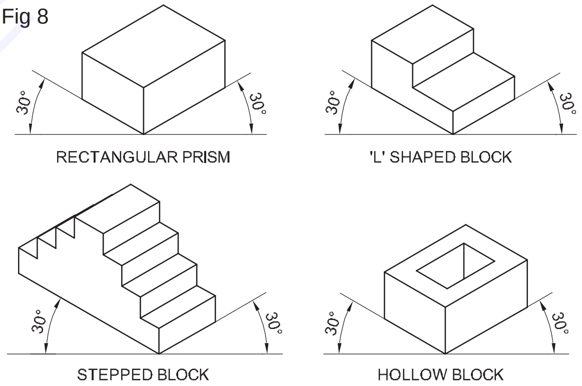


Fig 8



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Isometric scale

Isometric scales are used to get the foreshortened lengths required for isometric projection.

Before constructing an isometric scale, you must understand is the relationship between the true length of an edge and the length of the same in isometric projection.

To determine the relationship between the true length and corresponding length in isometric projection, proceed as follows:

Consider the isometric projection of a cube. (Fig 1)

Separately draw the top face of the cube adbc and join the longer diagonal ab. (Fig 2)

Note that the diagonal ab is of same length both in the isometric view of the face and the true face. Assume the top true face of the cube as afbc.

Now superimpose the true top face afbg keeping the diagonal ab common. (Fig 2)

Fig 1

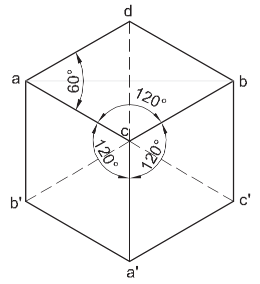
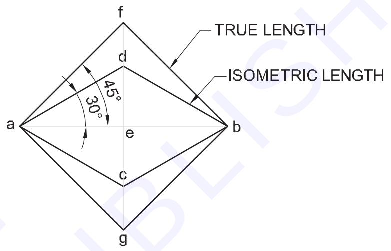


Fig 2



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$\angle FAE = 45° and \angle DAE = 30°$

AE = AF x Cos 45° and AD = AE $\div$ Cos 30° = AF x $\frac{\text{Cos} 45°}{\text{Cos} 30°}$

$$\frac{AD}{AF} = \frac{AF \times \text{Cos} 45°}{AF \times \text{Cos} 30°} = \frac{\text{Cos} 45°}{\text{Cos} 30°}$$

$$= \frac{1}{\sqrt{2}} \times \frac{2}{\sqrt{3}} = \frac{\sqrt{2}}{\sqrt{3}} = 0.8165$$

AD = 0.82 AF. This means that the length of a line in isometric projection is 0.82 times of it true length. While drawing an object in isometric projection, the dimensions on or parallel to isometric axes are reduced to this proportion. To make things easier we can construct a scale to the above ratio. Such a scale is called as isometric scale.

Procedure to construct

Isometric scale (Fig 3)

- Draw a horizontal line OA.
- Draw lines OB and OC making 30° and 45° with OA respectively.
- Mark 5 mm, 10 mm, 15 mm upto 100 mm on line OC.
- From the marked points on the regular scale OC, draw perpendiculars to OA meeting at OB.
- Print the corresponding values on the line OB resulting in the isometric scale.

Orientation of isometric axes: While the isometric axes make 120° to each other they may have different orientation as shown in Fig 4. Each of the orientation show3 of the 6 faces (left, right, top, bottom, front and rear) are shown in different combinations.

Fig 3

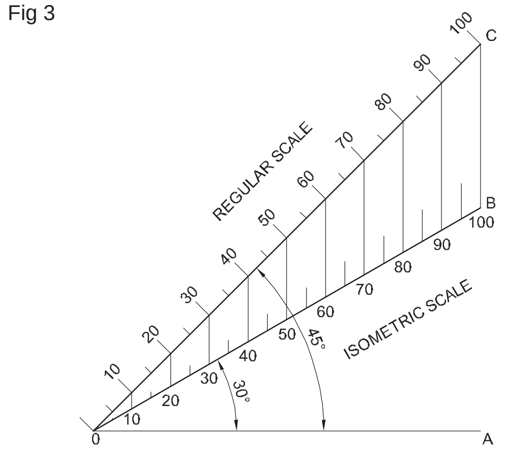
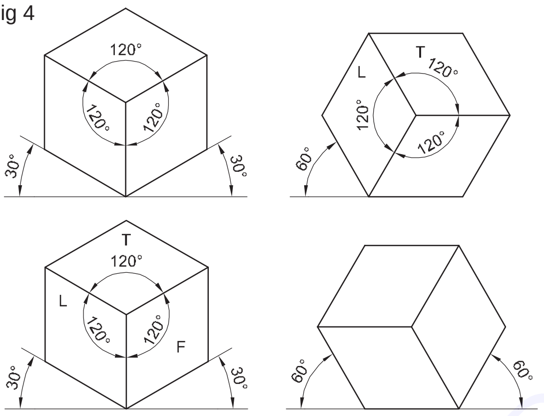
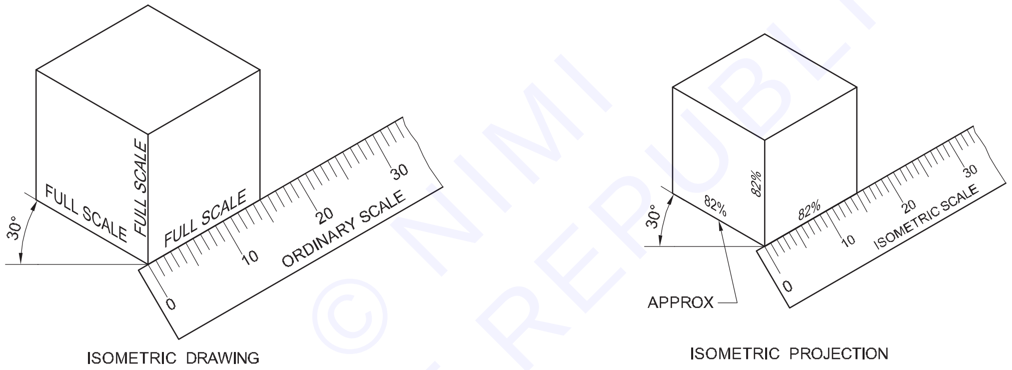


Fig 4



Isometric view and Isometric projection: A drawing is made with true lengths (dimensions) is called isometric view or isometric drawing. Whereas the same drawing made with isometric lengths is termed as isometric projection. (Fig 5)

Fig 5



Isometric and non-isometric lines: Fig 6 shows the isometric view of a shaped block. Here all lines except AB, BC and DE are parallel to isometric axis. Lines such as then which are parallel to isometric axes are called isometric lines whereas such as lines AB, BC and DE which are not parallel to isometric axes are called non-isometric lines.

The length of non-isometric lines will not follow the scale used for isometric lines. To proove this point consider the non-isometric lines AB or BC. The true length of both AB and BC is 5 cm while BC will be longer. Because of this reason non-isometric lines are drawn first by locating their starting and end points on isometric lines.

To locate the end points and to draw the non-isometric lines two methods are employed. They are

- Box method
- Off-set method

Box method: The object is assumed to be inside a rectangular box. Starting and end points are located and marked. By joining the points isometric view is drawn.

Off-set method: This method is most suited for the objects consisting of number of planes at a number of different angles.

These methods are not only useful for isometric views involving non-isometric lines but also for the isometric views involving isometric lines.

Box method of drawing a pyramid

Example

Draw an isometric view for the triangular pyramid shown in Fig 7 using a box method.

Fig 6

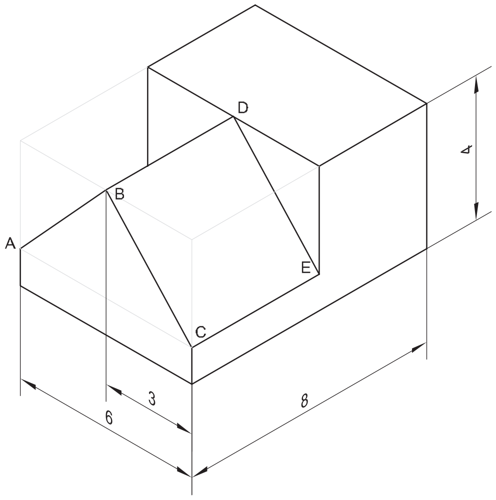
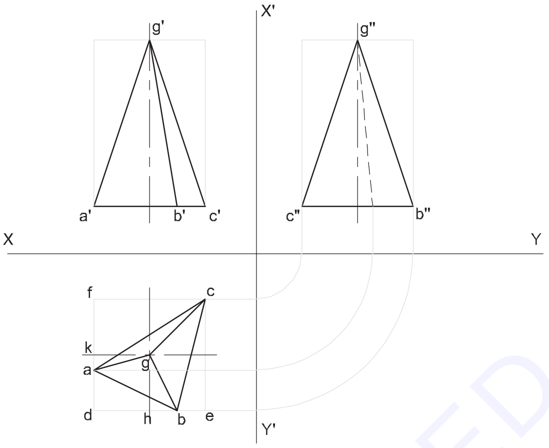


Fig 7



- Construct a rectangular box to the overall size of the pyramid (Fig 8a)
- Mark the distances ad and be from the plan of Fig 7 in the base of the box.
- Mark the distances kg and dh on the top face of box. (Fig 8a)
- Join the points AB, BC, CA, AG, BG and CG and complete the isometric view of the pyramid in box method. (Fig 8b)

Off-set method of drawing a pyramid

Example

Same triangular pyramid (Fig 7) is considered for drawing isometric view using offset method.

- Draw an isometric square/rectangle considering the corners of the base of the pyramid. (Fig 9a)
- Locate the corners 1,2 & 3 with help of offsets P and Q.
- Locate the projection of the vertex O_1 on the base by offsets x and y and draw the vertical centre line O_1O to the height of the pyramid. (Fig 9b)
- Join 1-2, 2-3, 1-3, 0-1, 0-2, 0-3 and complete the isometric view of the pyramid. (Fig 9c)

Fig 8

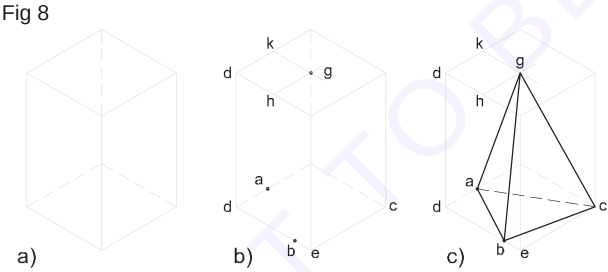
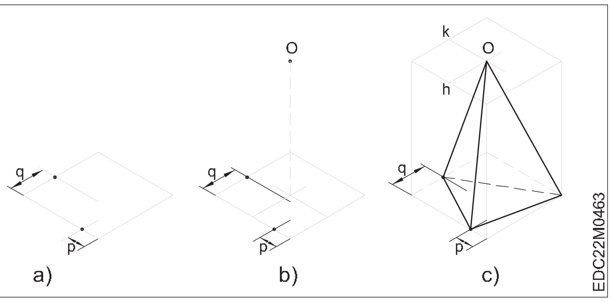


Fig 9



Angles in isometric projection: The angles of inclined surfaces will not have true value in the isometric projection, but will be more in some cases and less in other cases.

For example, in the isometric view of prism shown in Fig 10 the true value of all the angles is 90°. But in isometric projection the angles are 60° in some cases and 120° in others.

Isometric circles: The term isometric circle refers to the shape of circle in isometric view. An isometric circle will be elliptical in shape as shown in Fig 11 while drawing isometric view of cylindrical features isometric circles will have to be used. (Fig12)

An isometric circle can be drawn either by plotting / offset method or by arc method.

Plotting method (Fig 13)

- Draw a square of side equal to the dia of circle and inscribe the circle.
- Divide the circle into any number of equal parts and mark points such as 1,2,3,4,5,6,7,8 on the circle.

Fig 10

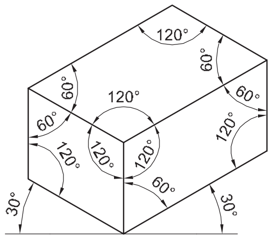


Fig 11

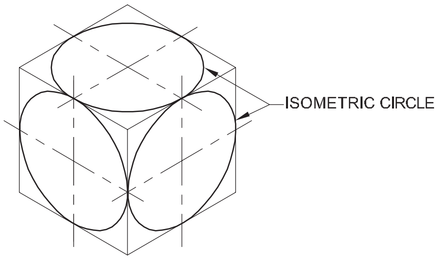


Fig 12

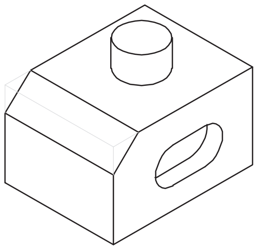
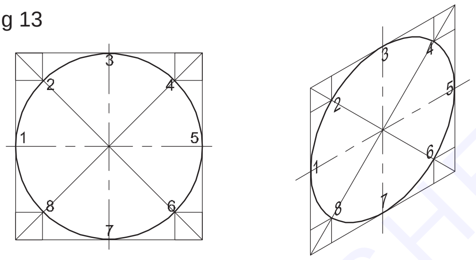


Fig 13



- Through the points 1,2,3 etc draw lines parallel to the both the axis of cylinder.
- Draw isometric view of the square.
- Mark points corresponding to 1,2,3....8 with isometric view of the square as points 1’,2’,3’....8’.
- Join these points with a smooth curve to form an ellipse.

Note: The orientation of the isometric circle will depend upon the plane on which the circular feature exists.

Arc method: Isometric circles drawn by offset method is the ideal method of making isometric circles as the ellipse obtained this way is geometrically true. But by free hand we cannot get a clear line.

Fig 14 shows the construction of isometric circle in 3 different orientation by arc method. Four arcs are to be drawn and the centres an C_1, C_2, B & D. While centre B and D are the corner of the rhombus C_1 and C_2 are intersection points of the longer diagonal with lines from points B or D to the mid point of the side of the rhombus.

Note: The arc method gives a clean ellipse, but this ellipse drawn this way will slightly deviate from true ellipse. It does not matter for our purpose.

The isometric circles can also be drawn using templates which can be bought from stationary shops.

Isometric views profiles: The profile MN of the block shown in Fig 15 is irregular in nature. The isometric views of such lines may be drawn by offset method described earlier. The points 1,2,3 and 4 lie on the profile. Lines A-1, B-2, C-3, D-4 are isometric lines and their length are same both in Fig 15 & Fig 16. After getting the points 1,2,3 & 4, they joined by smooth curve.

Fig 14

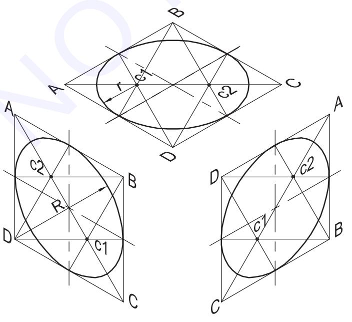
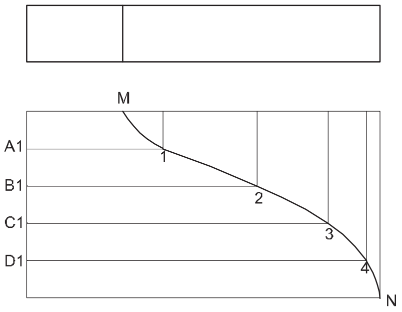


Fig 15



Note: In offset method more the number of points, better will be the accuracy of the curve.

Isometric projection of sphere: The Orthographic view of a sphere seen from any direction is a circle of diameter equal to the diameter of the sphere. Hence, the isometric projection of a sphere is also a circle of the same diameter.

The front view and the top view of a sphere resting on flat surface are shown in Fig 17a.

O as its centre, D is the diameter and P is the point of contact with the surface.

Assume a vertical section the centre of the sphere. Its shape will be a circle of diameter D. The isometric projection of this circle are ellipses 1 & 2 Fig 17(b) drawn in two different vertical positions around the same centre O. The major axis in each case is equal to D. The distance of the point P from the centre O is equal to the isometric radius of the sphere.

Fig 16

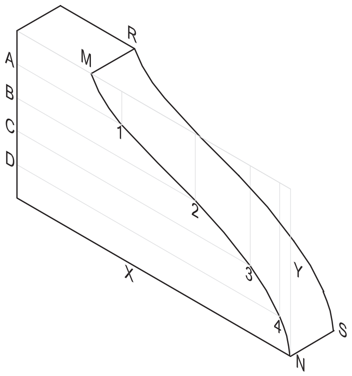
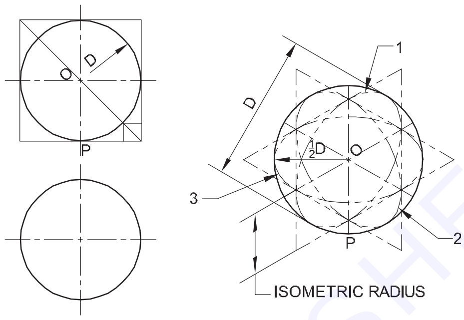


Fig 17



Again, assume a horizontal section through the centre of the sphere.

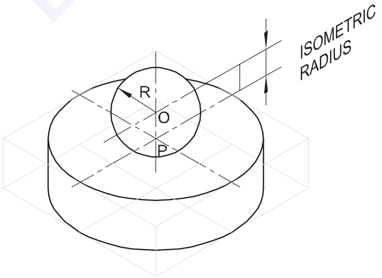
The isometric projection of this circle is shown by the ellipse 3, drawn in a horizontal position around the same centre O. In all the three cases 1,2 & 3 the outermost points on the ellipse from the centre O is equal to 1/2 D.

Thus, it can be seen that in an isometric projection, the distances of all the points on the surface of a sphere from its centre are equal to the radius of the sphere. Hence, the isometric projection of a sphere is a circle whose diameter is equal to the true diameter of the sphere.

Also the distance of the centre of the sphere from its point of contact with the flat surface is equal to the isometric radius OP of the sphere.

It is therefore of the utmost importance to note that isometric scale must invariably be used while drawing isometric projection of solids in conjunction with spheres or having spherical parts.

Fig 18



EXERCISE 5 : Isometric Views

Pictorial drawing (Isometric drawing)

Pictorial drawing: Even a common man can understand easily the shape of an object quickly by a picture or by a pictorial drawing. It is also called as three dimensional drawing.

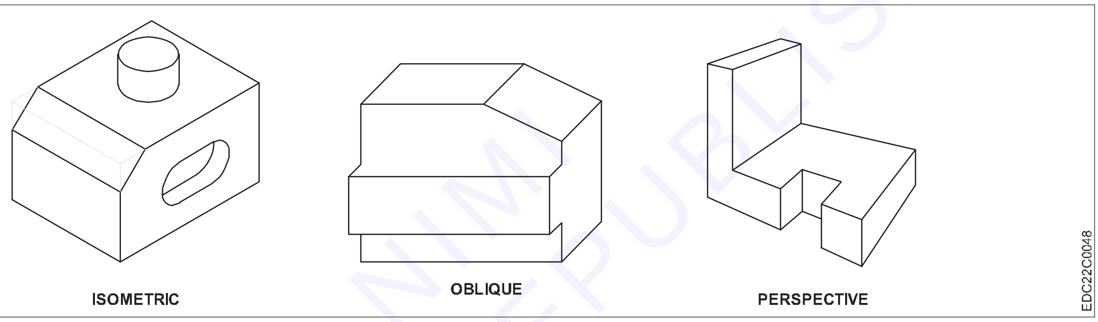
Pictorial drawings are very useful for describing the shape of a piece part or component, even though they have a distorted look.

Three type of pictorial drawings are (Fig 1)

- Isometric drawing
- Oblique drawing
- Perspective drawing

Out of the above three types, isometric drawings are very much prefered by machine shop and metal working trades group. But perspective drawings are popular in civil engineering group of trades.

Fig 1



Isometric drawing: In an isometric drawing the three mutually perpendicular edges of a cube are at an angle of 120° with each other. Instead of drawing the edges in the above said way, first we can also start from point `a'. At this point also three mutually perpendicular edges met while two of these edges make 30° to horizontal, the other edge is vertical. After drawing two 30° lines and one vertical line, parallel lines are drawn to complete cube. (Fig 2)

These three lines which represent the mutually perpendicular edges are isometric axes. Generally those axes are kept in four positions. (Fig 3)

Fig 2

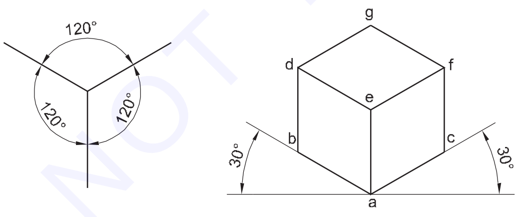
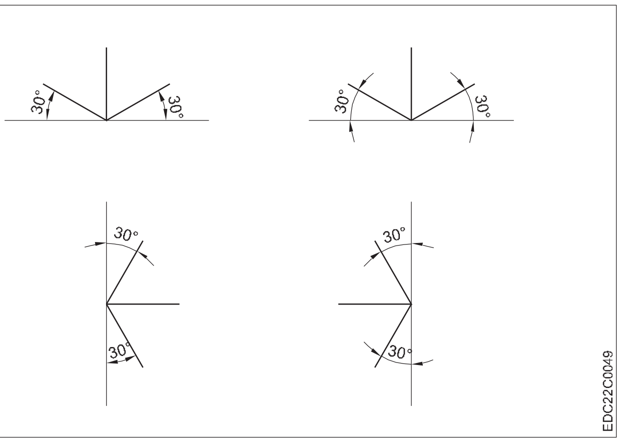


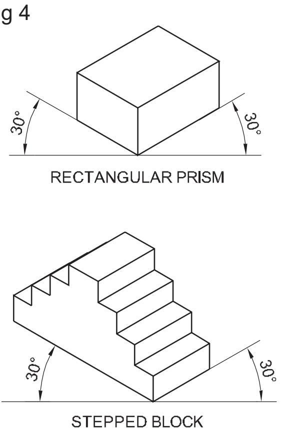
Fig 3



So to draw the isometric drawing, first draw the three mutually perpendicular edges, set other linear dimensions and complete the figure. (Fig 4)

Isometric and non-isometric lines: Fig 5 shows the isometric view of a shaped block. Here all lines except AB, BC and DE are parallel to isometric axis. Such lines which are parallel to isometric axes are called isometric lines whereas such lines AB, BC and DE which are not parallel to isometric axes are called non-isometric lines.

Fig 4



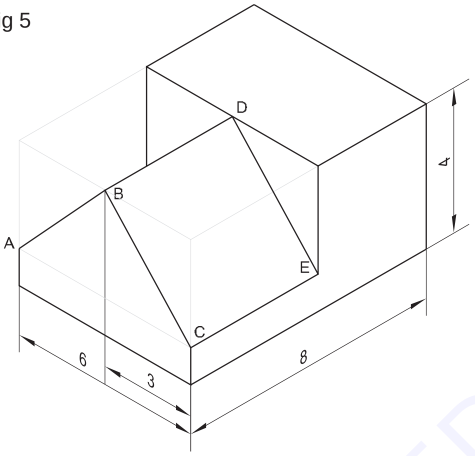
RECTANGULAR PRISM

'L' SHAPED BLOCK

STEPPED BLOCK

HOLLOW BLOCK

Fig 5



The length of non-isometric lines will not follow the scale used for isometric lines. To prove this point consider the non-isometric lines AB or BC. The true length of both AB and BC is 5 cm while BC will be longer. Because of this reason non-isometric lines are drawn first by locating their starting and end points on isometric lines.

To locate the end points and to draw the non-isometric lines two methods are employed. They are

- Box method
- Off-set method

Box method: The object is assumed to be inside a rectangular box. Starting and end points are located and marked. By joining the points isometric view is drawn.

Off-set method: This method is most suited for the objects consisting of number of planes at a number of different angles.

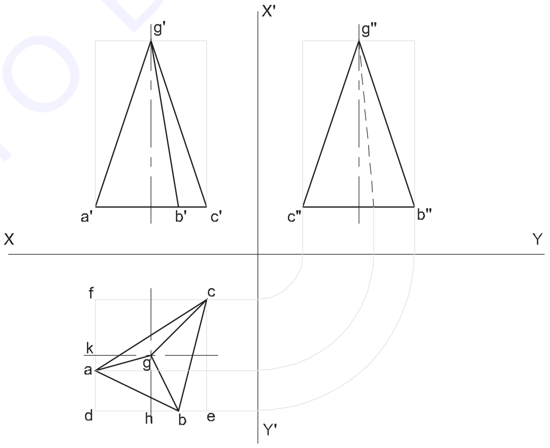
These methods are not only useful for isometric views involving non-isometric lines but also for the isometric views involving isometric lines.

Box method of drawing a pyramid

Example

Draw an isometric view for the triangular pyramid shown in Fig 6 using a box method.

Fig 6



- Construct a rectangular box to the overall size of the pyramid (Fig 7a)
- Mark the distances ad and be from the plan of Fig 7b in the base of the box.
- Mark the distances kg and dh on the top face of box. (Fig 7c)
- Join the points ab, bc, ca, ag, bg and cg and complete the isometric view of the pyramid in box method. (Fig 7c)

Off-set method of drawing a pyramid

Example

Same triangular pyramid (Fig 6) is considered for drawing isometric view using offset method.

- Draw an isometric square/rectangle considering the corners of the base of the pyramid. (Fig 8a)
- With the help of Fig 6 (Plan) locate all the three corners of the base P,Q and r using offset method.
- Locate the position of vertex `O' on base by referring the Fig 6 (Plan) using the same offset method. (Fig 8b)
- Draw the vertical line 0'-0 to the height of the pyramid.
- Join the corners of the base.
- Join the vertex 0' with the corners of the base and complete the pyramid. (Fig 8c)

Angles in isometric drawing: The angles of inclined surfaces will not have the value in the isometric drawing, but will be more in some cases and less in other cases.

Fig 7

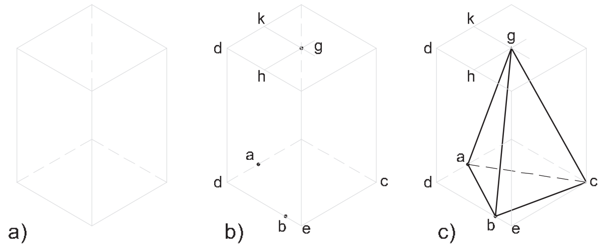
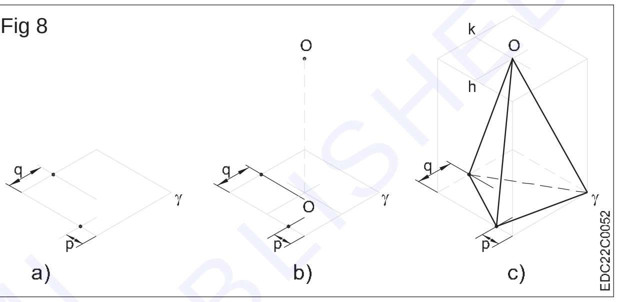


Fig 8



For example, in the isometric view of prism shown in Fig 9 the true value of all the angles is 90°. But in isometric drawing the angles are 60° in some cases and 120° in others.

Isometric circles: The term isometric circle refers to the shape of circle in isometric view. An isometric circle will be elliptical in shape as shown in Fig 10. While drawing isometric view of cylindrical features isometric circles will have to be used. (Fig11)

Fig 9

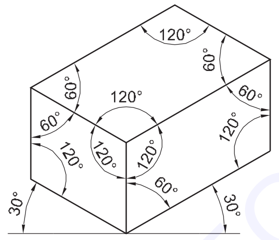


Fig 10

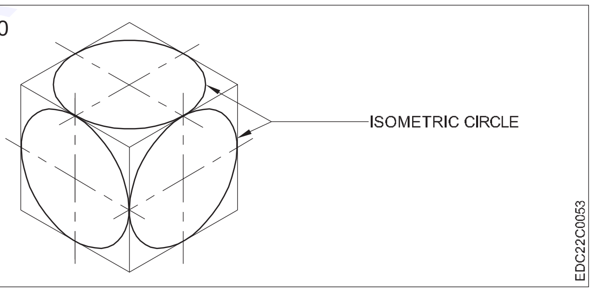
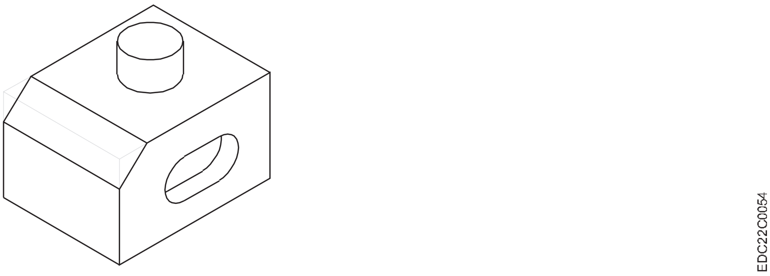


Fig 11

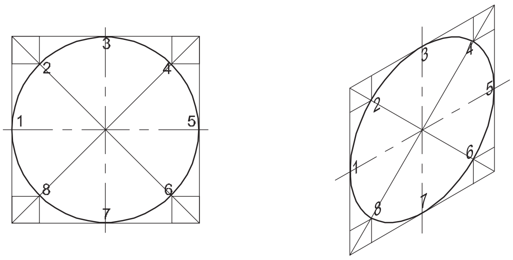


An isometric circle can be drawn either be plotting offset method or by arc method.

Plotting method (Fig 12)

- Draw a square of side equal to the dia of circle and inscribe the circle.
- Divide the circle into any number of equal parts and mark points such as 1,2,3,4,5,6,7,8 on the circle.
- Through the points 1,2,3 etc draw lines parallel to the both the axis of cylinder.

Fig 12



– Draw isometric view of the square.

– Mark points corresponding to 1,2,3....8 with isometric view of the square as points 1',2',3'....8'.

– Join these points with a smooth curve to for an ellipse.

Note: The orientation of the isometric circle will depend upon the plane on which the circular feature exists.

Arc method: Isometric circles drawn by offset method is the ideal method of making isometric circles as the ellipse obtained this way is geometrically true. But by free hand we cannot get a clear line.

Fig 13 shows the construction of isometric circle in 3 different orientation by arc method. Four arcs are to be drawn and the centres an C_1, C_2, B & D. While centre B and D are the corner of the rhombus C_1 and C_2 are intersection points of the longer diagonal with lines from points B or D to the mid point of the side of the rhombus.

Note: The arc method gives a clean ellipse, but this ellipse drawn this way will slightly deviate from true ellipse. It does not matter for our purpose.

The isometric circles can also be drawn using templates which can be bought from stationary shops.

Isometric view of profiles: The profile M'N' of the block shown in Fig 14 is irregular in nature. The isometric views of such lines may be drawn by offset method described earlier. The points 1',2',3' and 4' lie on the profile. Lines A'-1', B'-2', C'-3', D'-4' are isometric lines and their length are same both in Fig 14 & Fig 15. After getting the points 1,2,3 & 4, they joined by smooth curve.

Note: In offset method more the number of points, better will be the accuracy of the curve.

Fig 13

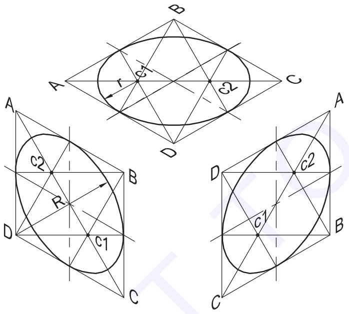


Fig 14

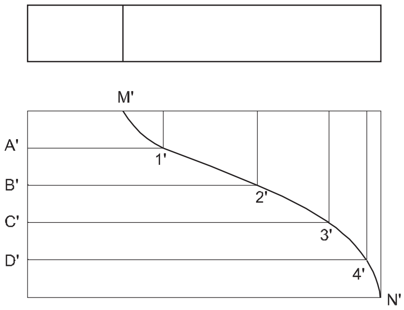
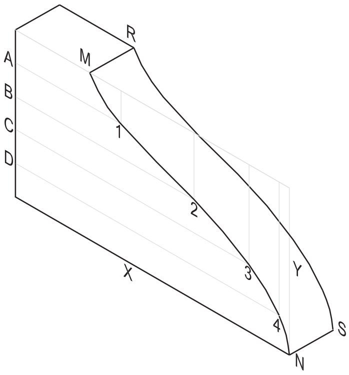


Fig 15



Isometric drawing of sphere: The Orthographic view of a sphere seen from any direction is a circle of diameter equal to the diameter of the sphere. Hence, the isometric drawing of a sphere is also a circle of the same diameter.

The front view and the top view of a sphere resting on flat surface are shown in Fig 16a.

`O' as its centre, D is the diameter and P is the point of contact with the surface.

Assume a vertical section the centre of the sphere. Its shape will be a circle of diameter D. The isometric drawing of this circle are ellipses 1 & 2 Fig 16(b) drawn in two different vertical positions around the same centre `O'. The major axis in each case is equal to D. The distance of the point P from the centre `O' is equal to the isometric radius of the sphere.

Again, assume a horizontal section through the centre of the sphere.

The isometric drawing of this circle is shown by the ellipse 3, drawn in a horizontal position around the same centre `O'. In all the three cases 1,2 & 3 the outermost points on the ellipse from the centre `O' is equal to 1/2 D.

Thus, it can be seen that in an isometric drawing, the distances of all the points on the surface of a sphere from its centre are equal to the radius of the sphere. Hence, the isometric projection of a sphere is a circle whose diameter is equal to the true diameter of the sphere. (Fig 17)

Fig 16

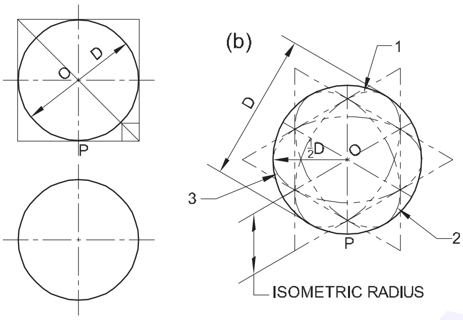
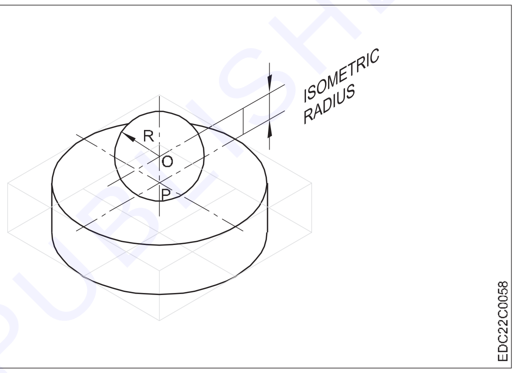


Fig 17



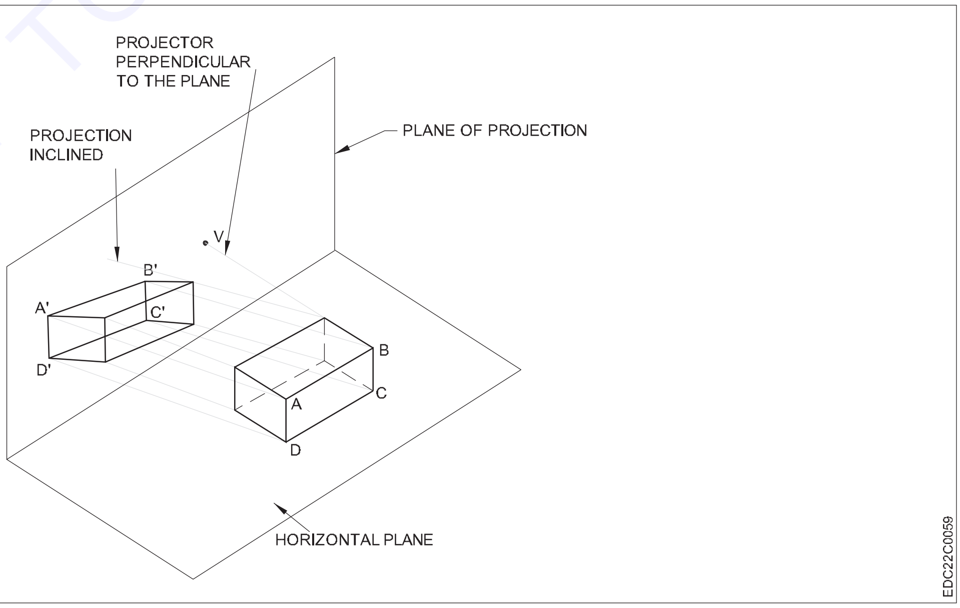
Also the distance of the centre of the sphere from its point of contact with the flat surface is equal to the isometric radius OP of the sphere.

It is therefore of the utmost importance to note that isometric scale must invariably be used while drawing isometric projection of solids in conjunction with spheres or having spherical parts.

Oblique drawings are yet another type of pictorial projections, they differ from isometric drawings in two ways.

– In oblique drawing, projections are oblique (inclined) to the plane of projection. whereas in isometric projections projectors are perpendicular to the plane of projection. (Fig 18)

Fig 18



- In an oblique drawing one of the principal faces of the object is kept parallel to the plane of projection, but in isometric, none of the faces of the object is parallel to the plane of projection.

Even though one of face of the object is positioned parallel to the picture plane, we still get a pictorial view and the projections are inclined to both HP and VP.

Because one of the principle faces of the object is parallel to the plane of projection, in the oblique projection, the projection of this face and faces parallel to it will appear in true size and shape. In the oblique projection of a prism is shown in Fig 19, the faces ABCD and EFGH are parallel to the plane of projection and they appear to be true in size and shape. The other four faces which are perpendicular to the plane of projection do not appear in true shape. (all these four faces are seen as parallelogram) However the vertical edges of these faces are parallel to the plane of projection and hence the projection of these edges will measure to their true lengths.

Projection of edges such as AE, DH, BF and CG which are perpendicular to the plane of projection will measure differently depending on the angle of inclination of the projectors. If the inclination of the projectors is 45° the projections of these edges measure to their true lengths. If the angle is less than 45° the projection of such perpendicular edges will measure less than the true length, if the angle of inclination of the projectors is greater than 45°. Projection of such perpendicular edges will measure more than the true length.

Oblique projections nevertheless have an unique advantage what we want to make pictorial drawings of object having curved features. For making isometric views of a curved feature we have first to draw their orthographic views in order to find out the offsets of points lying on the curve. But this difficult procedure is not necessary in the case of oblique views.

For example the component shown in Fig 20 has several curved features. While drawing oblique view of this component the curved features are drawn to true shape using compass. This is relatively easier method in comparison to the drawing of the same component in isometric view.

Fig 19

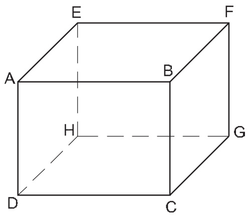
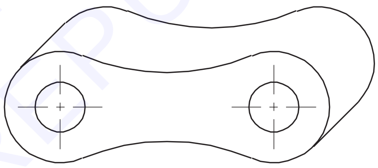


Fig 20



Inclination of projectors: While projectors can have any angle of inclination, usually oblique views are drawn to either 45° or 30°. The inclined lines in the oblique drawings are called receding lines (Fig 21) and they are more commonly drawn to 45°. The scale of lines along receding may be 1:1 (true lengths) or 1:2 (half of the true length). Oblique drawing drawn to 1:1 (Fig 22) are called as **cavalier** projections and those drawn to 1:2 (Fig 23) are called as **cabinet** projections. These two terms are of academic interest only and mostly we refer these views as oblique projections/drawings only.

In comparisn a cabinet projection will look less distorted than a cavalier projection.

Fig 21

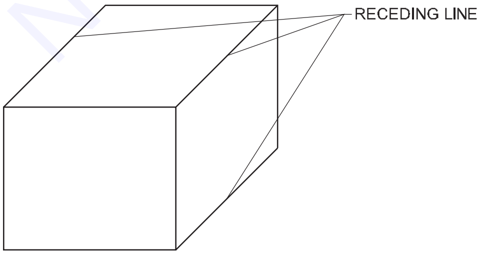
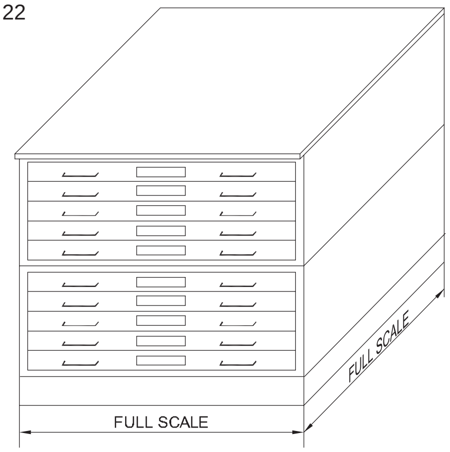


Fig 22



CAVALIER PROJECTION

Procedure for drawing oblique views: The procedure for creating oblique drawing is very much similar to that for drawing isometric views. To make isometric view we start from drawing three isometric axes at the desired orientation. In oblique drawing also we start by drawing three axes at the desired orientation. But here two of the axes are perpendicular to each other while the third axis (receding axis). Make 45° or 30° to the horizontal. The orientation of the axes may be any one of the four possibilities given in the figure 24.

Fig 23

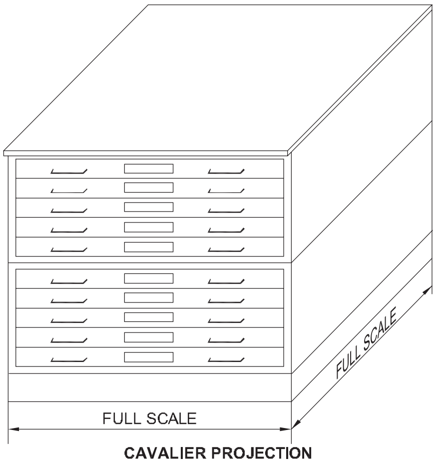
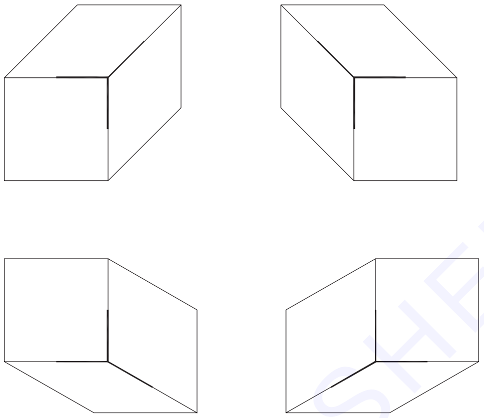


Fig 24



Object positioning in oblique drawing: Remember to position the object in such a way as to make best use of the advantage offered by oblique projection. The face that has the maximum curved details should be placed parallel to the plane of projection. See example in Fig 25.

Another point to note is as far as possible, place the longest dimension parallel to the plane of projection. (Fig 26a & b)

Few examples of oblique drawings are shown in Fig 27.

Draw the isometric figures (Ex.10.1 to 10.29) in the work book Ex.No.10. Follow the procedure given wherever necessary.

Fig 25

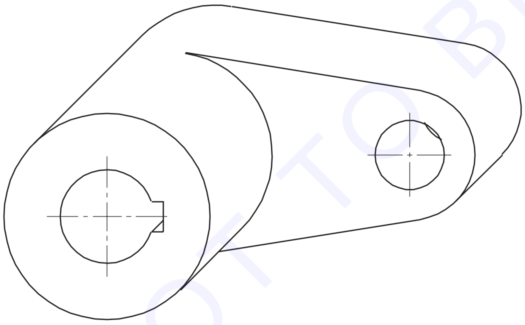


Fig 26

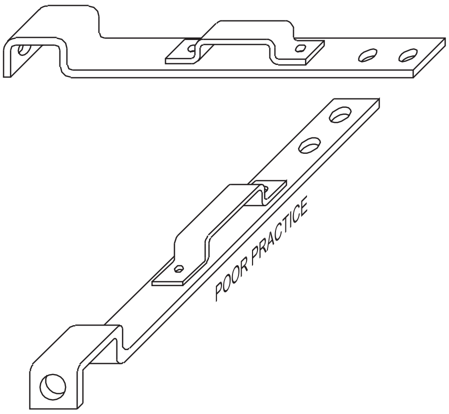
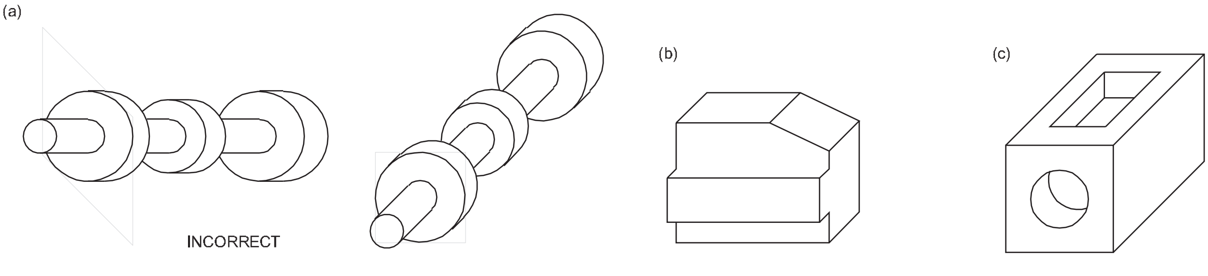


Fig 27



Pictorial drawing (Isometric drawing)

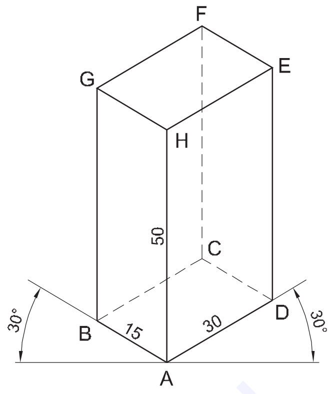
Draw the isometric figures

Procedure

Exercise 5.1

Draw the isometric drawing of a rectangular prism of base 30 mm x 40 mm and the height 60 mm. (Fig 1)

Fig 1



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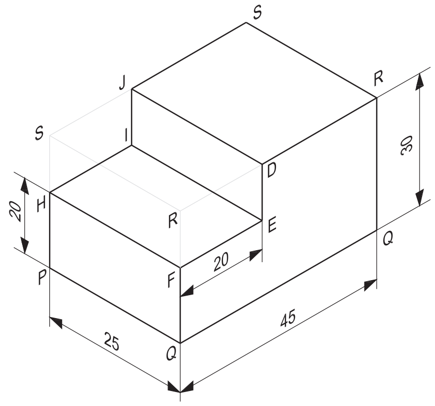
- Draw the three isometric axes through point `A'.
- Mark AB = 15 mm, AD = 30 mm and AH = 50 mm representing the three sides of prism.
- Draw two vertical lines parallel to the line AH through points B and D.
- Similarly draw two more lines parallel to AB and AD through point H.
- Mark G and E the intersecting points.
- Draw lines parallel to GH and HE through points G and E intersecting point is F.
- Draw lines parallel to AB & AD through points D and B respectively intersecting at C.
- Join CB & CD with dash lines.
- Join F and C also with dash lines.
- Rub off the construction lines and complete the prism.

Exercise 5.2

Draw the isometric view of the stepped block given in Fig 2.

Fig 2



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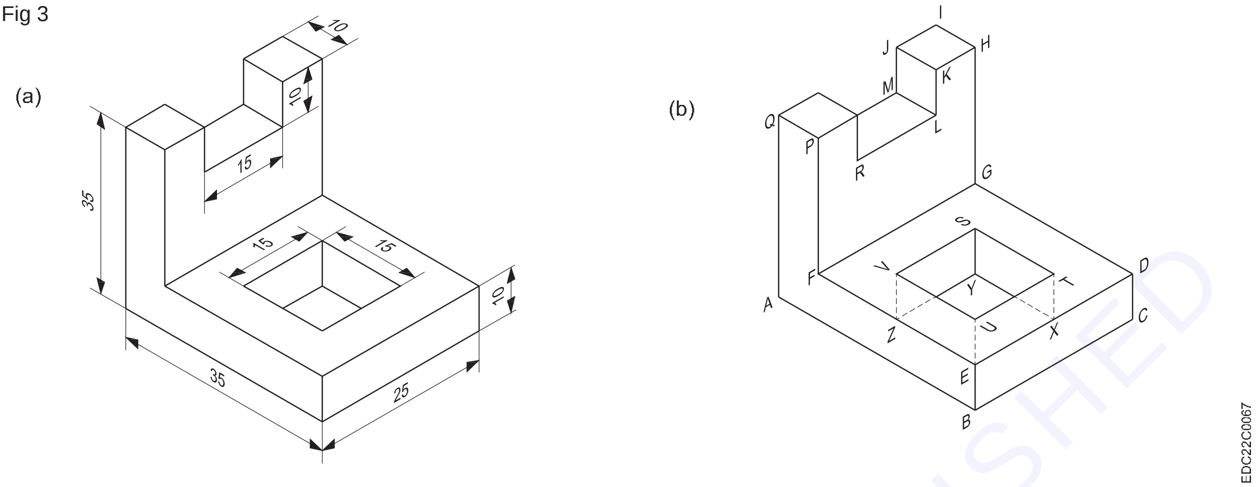
- Draw the isometric view of a rectangular prism of dimensions equal to the overall size of the block 45 x 25 x 30 mm.
- Draw the lines JD, DE, EF, FH, HI and IJ using the measurements given in the figure.
- Rub off SR, RD, SJ, SH and RF.

- Darken the remaining lines of the stepped block.

Exercise 5.3

Draw the isometric view of the components shown. (Fig 3)

Fig 3



- Draw the stepped block as per dimension. Follow the procedure given in the previous Ex.No.2.
- Mark points UTSV as per dimension on the top of the surface EDGF (Fig 3b)
- Join points UTSV.
- Project vetically downwards from the points UTSV and obtain the point WXYZ at bottom surface such that UW, TX, SY & VZ are equal to 10 mm. Join the point WXYZ and draw the thick lines which are all visible and dotted lines which are not visible.

Note: All construction lines should be in thin lines. After completion of the isometric views, in each case erase the unwanted lines and construction lines.

With the experiences gained in previous exercises of drawing isometric views, draw the following exercises 10.4 & 10.5 and complete the same.

Exercise 5.4 (Fig 4)

Exercise 5.5 (Fig 5)

Fig 4

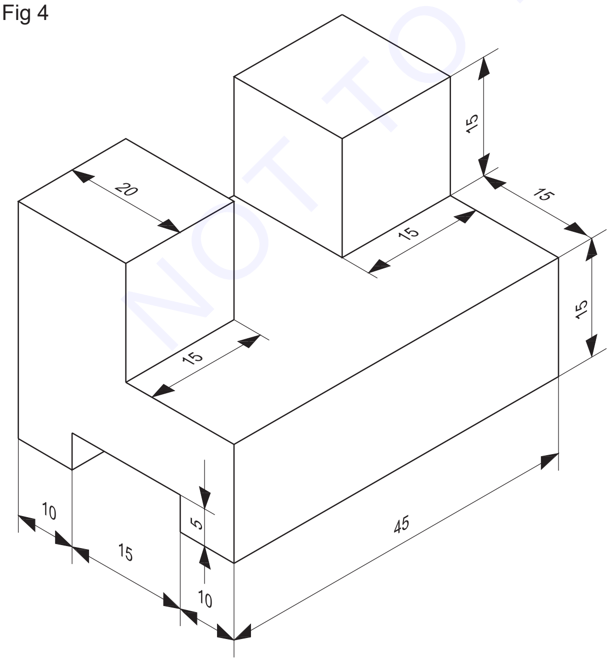
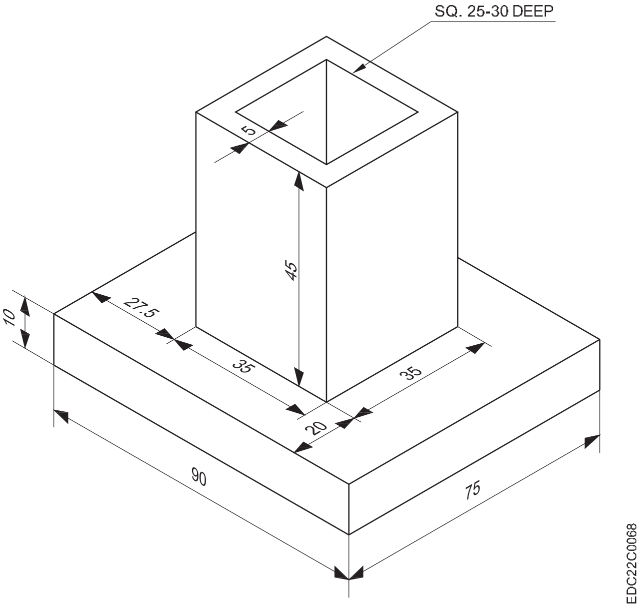


Fig 5

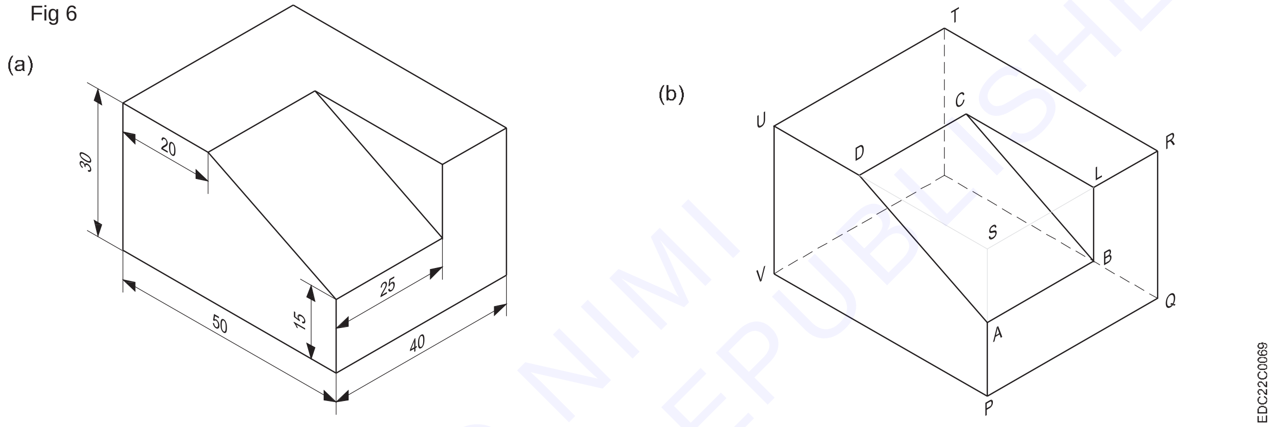


Exercise 5.6

Draw the isometric view of the machined block having non-isometric lines. (Fig 6a)

- Draw an isometric box. (Fig 6b)
- Mark point A on PS at a distance of 15 mm from P.
- Draw line AB = 25 mm parallel to PQ.
- From B, draw a vertical line cutting RS at L.
- Mark point D on US such that UD = 20 mm.
- Draw a line DC parallel to UT equal to AB.
- Join AD, BC and CL to complete the required isometric view of the block.
- Remove the extra lines and darken the required visible edges.
- Show hidden edges by dashed lines.

Fig 6

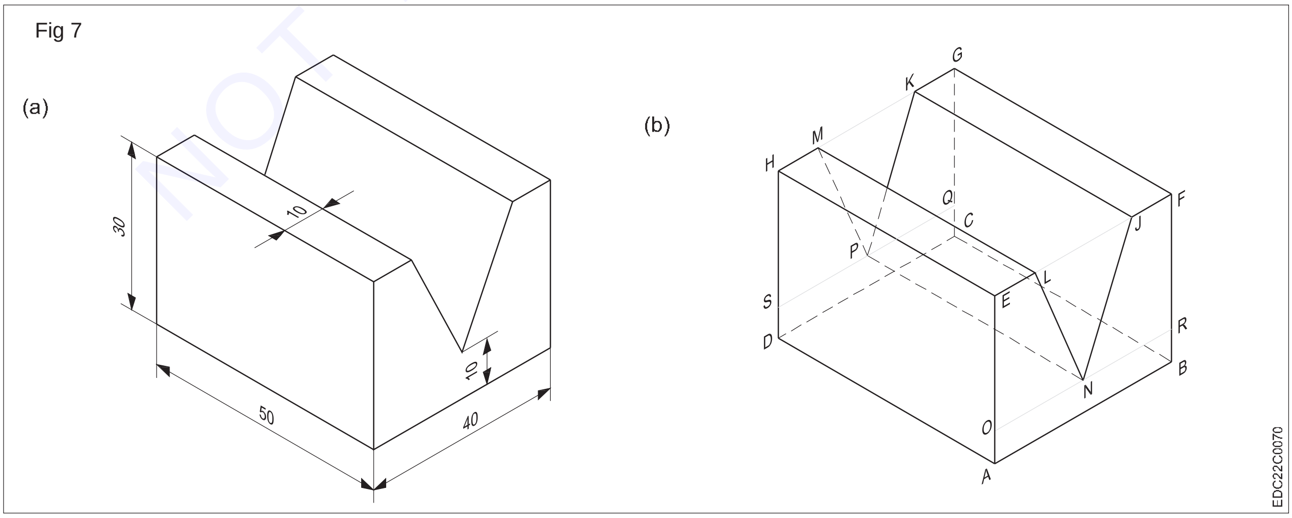


Exercise 5.7

Draw the isometric view of the `V' block. (Fig 7a)

- Draw the isometric view of a rectangular box of size 50 x 40 x 30.
- On the face ABFE, draw the lines JN & LN by offset method.
- Similarly draw lines KP & MP.
- Join ML, KJ and PN.
- Erase construction lines and make the remaining line thick and dashes according to the drawing.

Fig 7



Exercise 5.8 to 5.17

Draw the isometric drawing of the following slant cut blocks. (Fig 8 to 17)

- In each case draw the isometric view of a rectangular prism to the overall sizes of the each block.
- Follow the procedure adopted in the previous exercises and complete the each isometric view of the blocks.
- Remove the unwanted lines, draw the remaining lines thick and hidden lines as required. Complete the figures.
- Assume the missing dimensions if any proportionally.

Fig 8

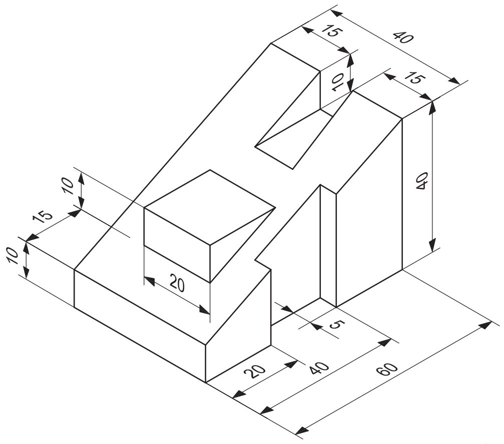


Fig 9

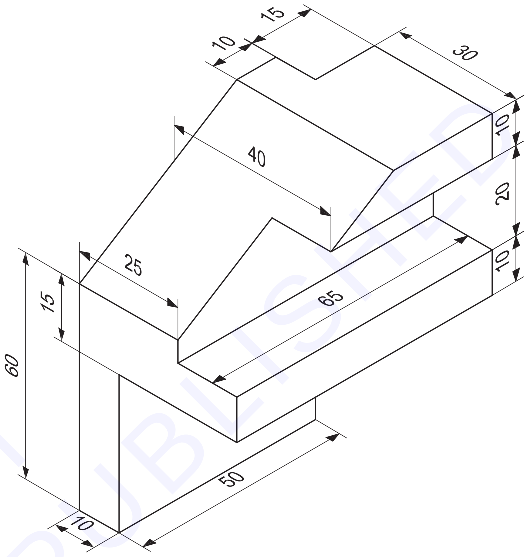


Fig 10

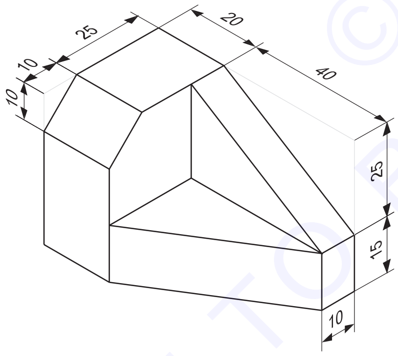


Fig 11

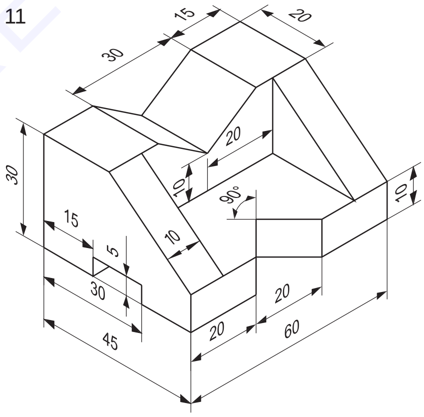


Fig 12

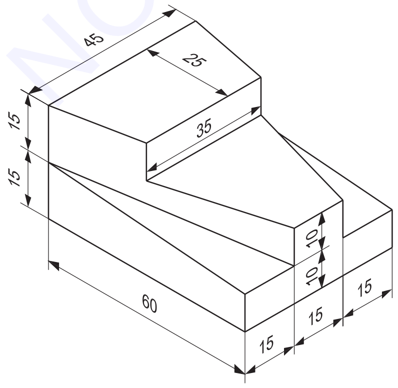


Fig 13

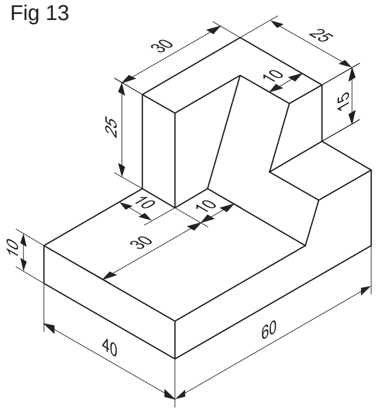


Fig 14

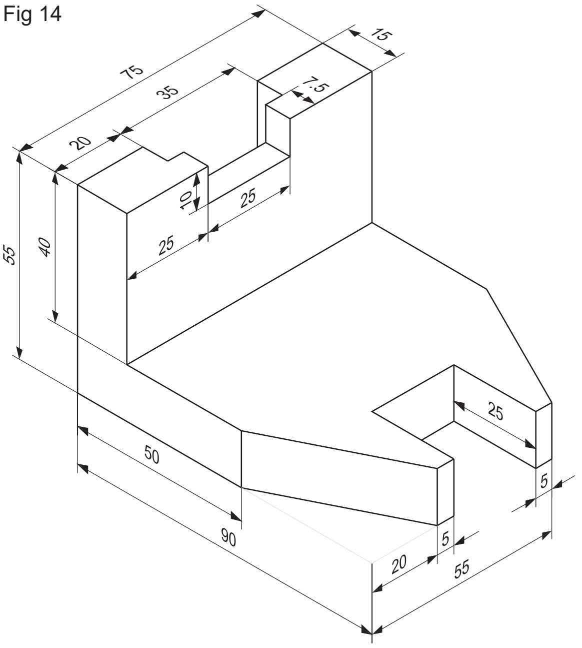


Fig 15

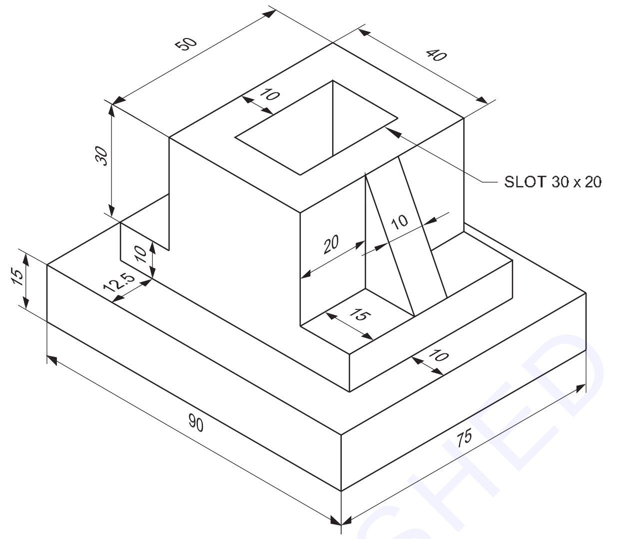


Fig 16

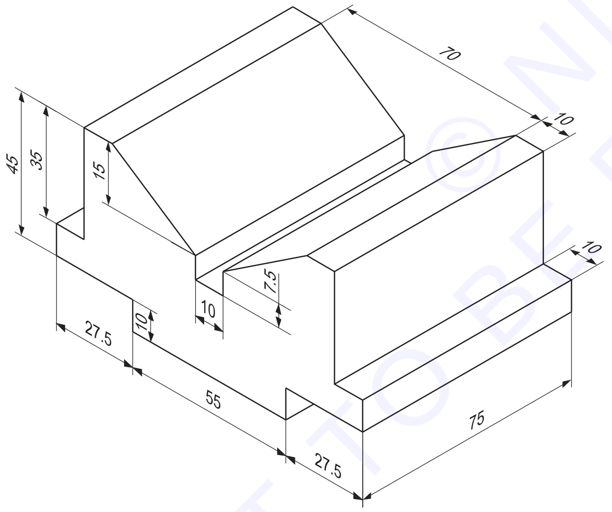
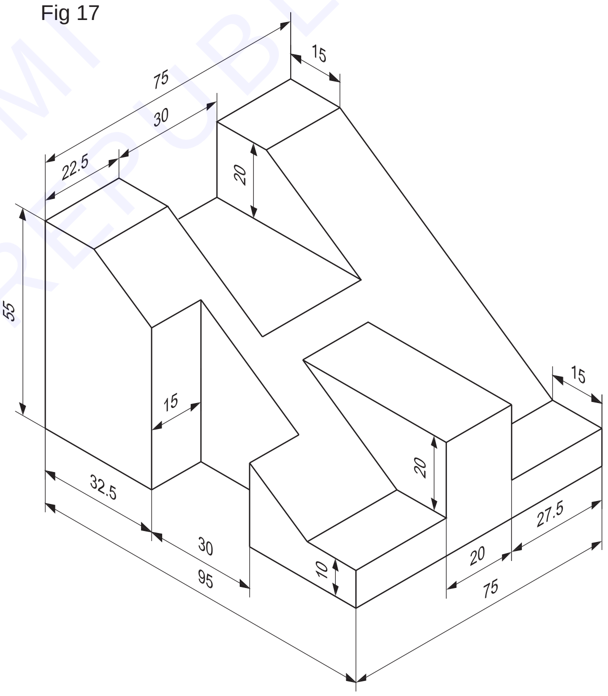


Fig 17



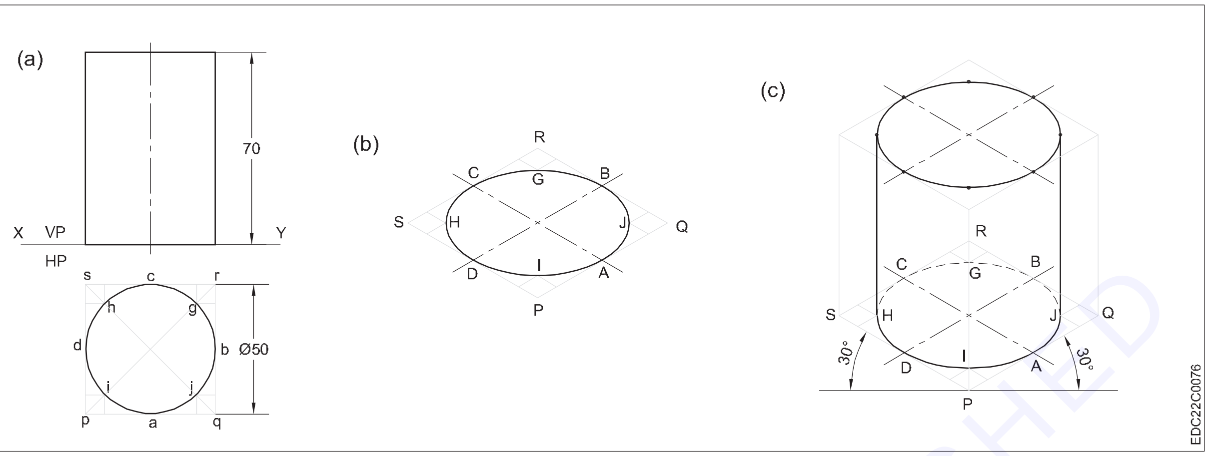
Draw the isometric view of a cylinder of base 50 mm and height/length of 70 mm with its base resting on horizontal surface by offset method (Fig 18)and four centre arc method. (Fig 19)

Off-set method

- Draw the shape of the cylinder looking from front and top.
- Draw the isometric view of a square of side equals to the dia of cylinder. (Fig 18b)
- Draw the isometric view of a square prism of height 70 mm on the square drawn.
- The mid-points of the sides of the square given four points ABCD and four more points HIJG by intersection of the diagonals with circles (located by off-set method) join the points to form isometric circle.

- Draw the isometric circles for the bottom and top face of the cylinder inside the square prism using offset method.
- Draw common tangents to top and bottom of isometric circles.
- Complete the view by drawing visible lines thick and invisible lines thin. (Fig 18c)

Fig 18



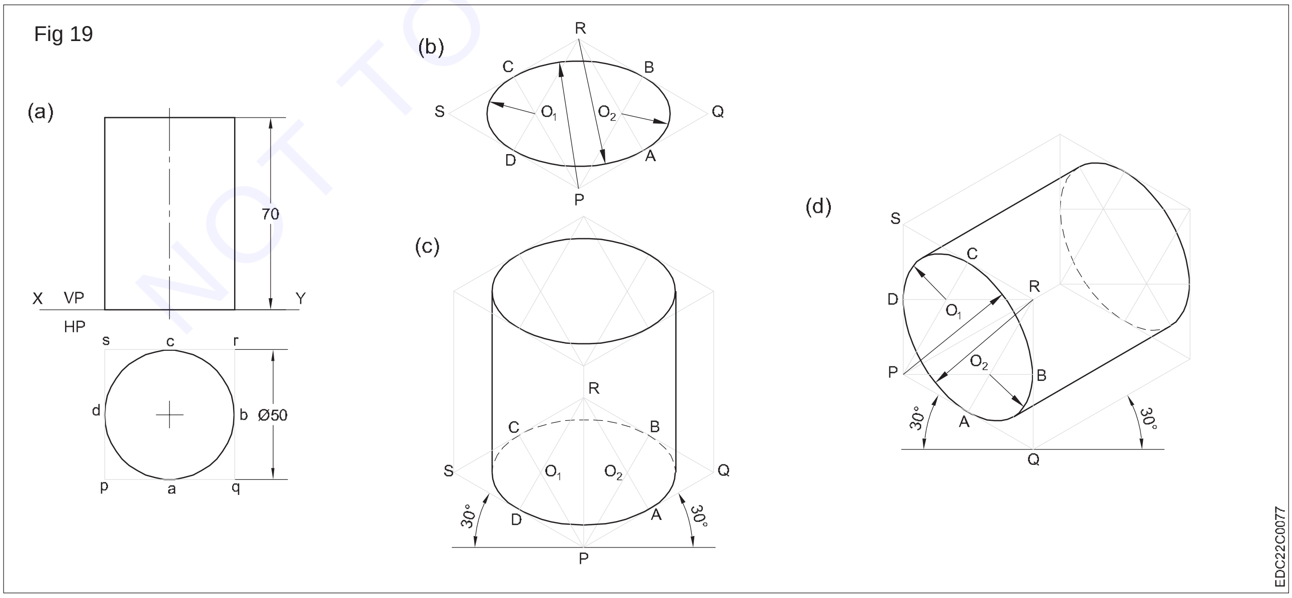
Exercise 5.19

Draw the isometric view of a cylinder of ϕ 50 and height 70 mm by four centre arc method. (Fig 19)

Four centre arc method

- Draw the front and top position of cylinder. (Fig 19a)
- Draw the isometric projection of a square of side equals to the dia of cylinder. (Fig 19b)
- Draw the isometric drawing of a square prism of height 70 mm on the square drawn. (Fig 19c)
- Draw the bisectors RD and RA from R and PC and PB from P.
- Draw arcs with O_1 and O_2 as centres and radius O_1D or O_2A.
- Draw arcs with P and R as centres and radius PC or RD.
- Draw vertical lines from the end of the ellipse.
- Complete the isometric view of the prism. (Fig 19c)
- Fig 19d shows the cylinder in horizontal position.
- Follow the procedure of the cylinder in vertical position and complete the prism.

Fig 19

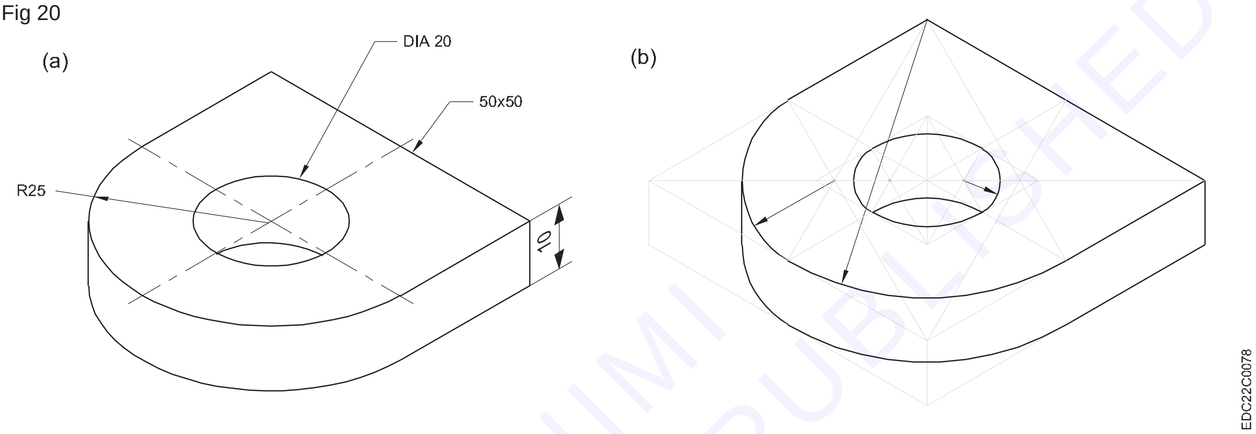


Exercise 5.20

Draw the isometric view of the solid block. (Fig 20a)

- Draw the isometric view of a square prism of size 50 mm, height 10 mm showing all six faces in thin lines. (Fig 20b)
- Draw isometric circles of diameter 20 mm both on top and bottom faces of the prism at the given location.
- With the same centres, draw two isometric semi-circles.
- Join the smaller circles and the semi-circles with common tangents as shown in Fig 20b.
- Erase unwanted lines.
- Make the remaining lines thick and hidden as required and complete the solid block.

Fig 20



Exercise 5.21 to 5.25

Draw the isometric projections of the given exercises. (Fig 21 to 25)

- Follow the procedures of previous exercises.

Fig 21

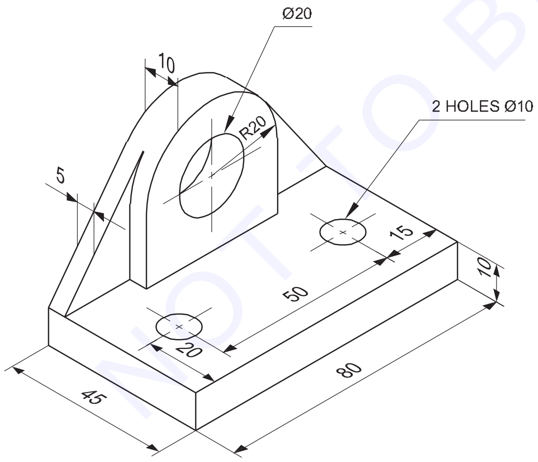


Fig 22

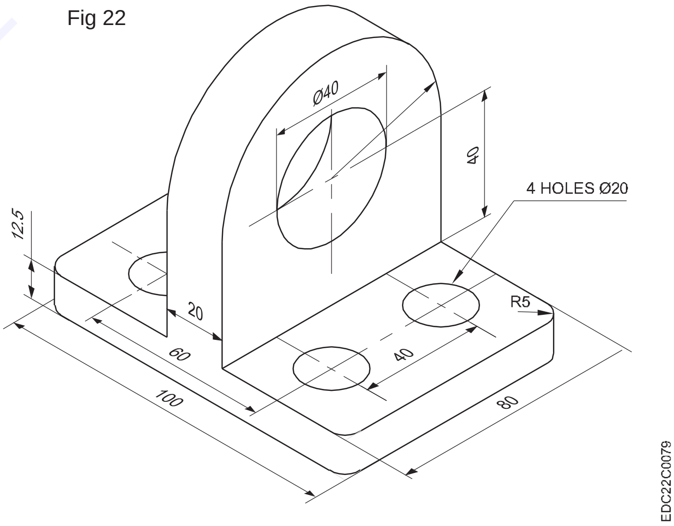


Fig 23

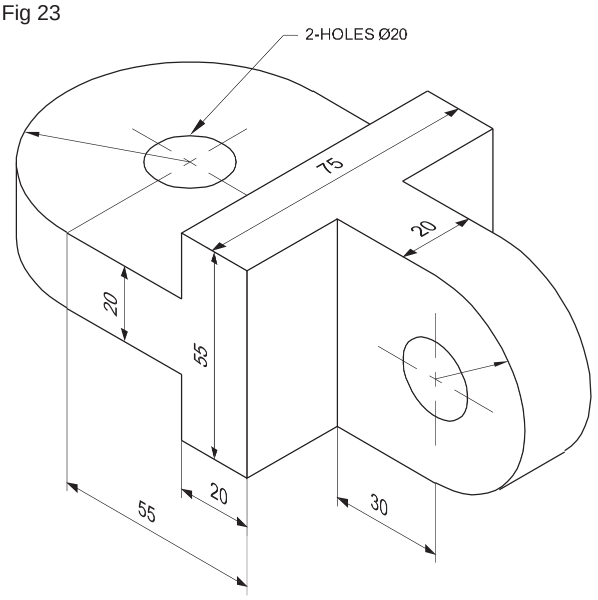


Fig 24

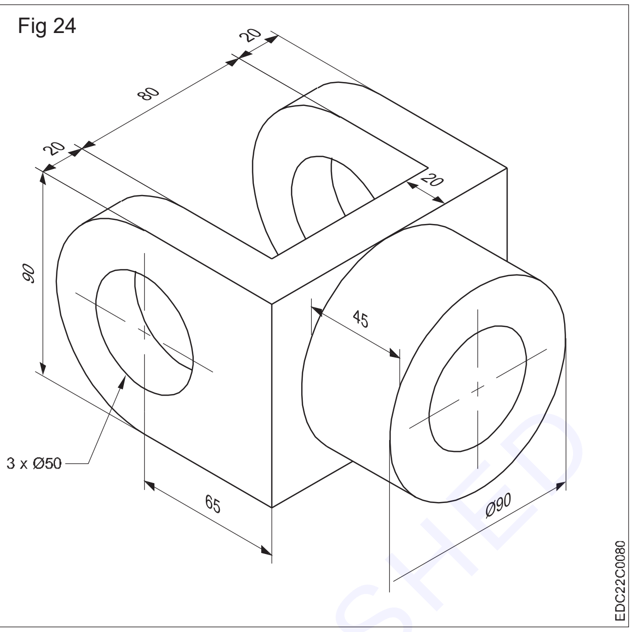
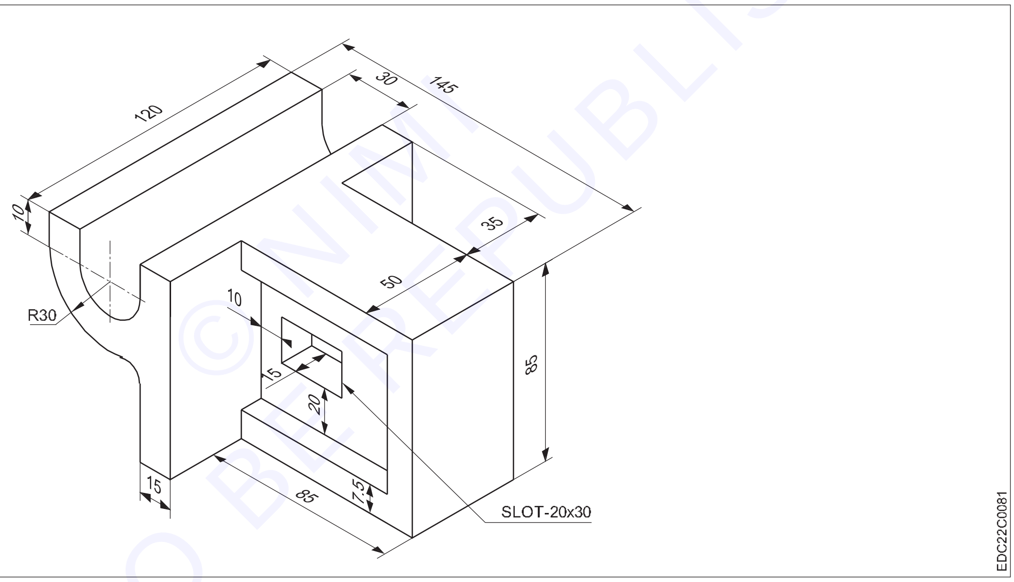


Fig 25



Exercise 5.26

Draw the isometric view of a cone whose base diameter 40 mm and height 60 mm when its base rest on horizontal plain surface. (Fig 26)

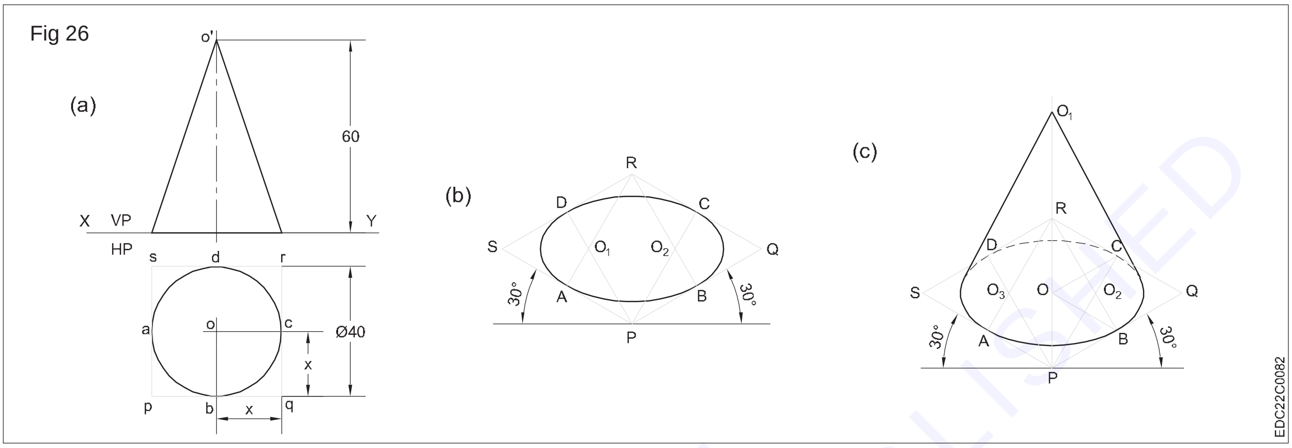
- Draw the front view and plan of the cone in the true scale as shown in the Fig 26a.
- Draw the isometric view of the base circle. (by four centre method) (Fig 26b)
- Mark the centre and draw a vertical line 0.0_1 such that 0.0_1 equals to 60 mm in isometric scale.
- From `0' draw tangents to the isometric circle of the base and complete the required isometric view of the cone. (Fig 26c)

Draw the isometric view of the hexagonal prism of 2.5 cm side of base and 60 mm height.

- Draw a hexagon of side 25 mm of its edge is horizontal.
- Draw a rectangular prism of side 25 mm and height 60 mm.

- Draw the isometric view of the hexagonal base $a_1b_1c_1d_1e_1f_1$ of the prism using offset method.
- Draw the top hexagonal face by drawing projection from the corners of the base.
- Make the visible edges by drawing thick lines and draw the invisible edges in hidden line.
- Rub off the unwanted lines and complete the isometric projection.
- Use isometric scale of all measurements.

Fig 26

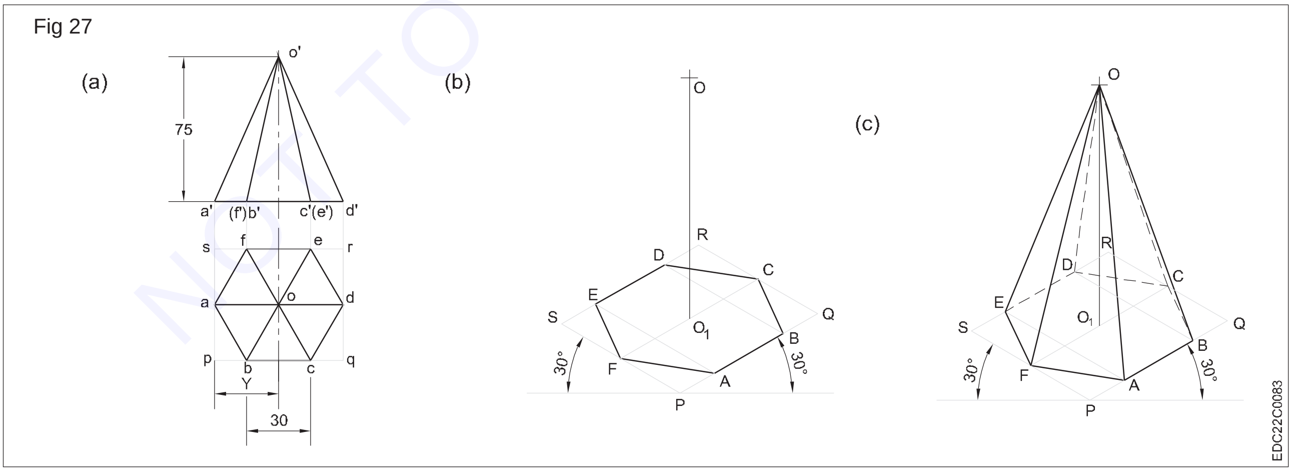


Exercise 5.27

Draw the isometric views of a hexagonal pyramid of side base 30 mm and height 75 mm given its position as under:

- its base resting on HP and the edge of the base parallel to VP. (Fig 27)
- Draw the plan and elevation of the pyramid (true scale) and enclose the plan in the rectangle PQRS. (Fig 27a)
- Draw the parallelogram with two of its adjacent edges at 30° to the horizontal. (Fig 27b) PQ = Isometric length of pq and PS = Isometric length of PS.
- Draw an isometric hexagon ABCDEF in the parallelogram PQRS. (Fig 27b)
- Mark the centre `O' and draw a vertical line from point `O' of height to 75 mm in isometric scale.
- Join `O', with ABCDE&F to complete the required isometric projection of the hexagonal pyramid. (Fig 27c)

Fig 27



Exercise 5.28

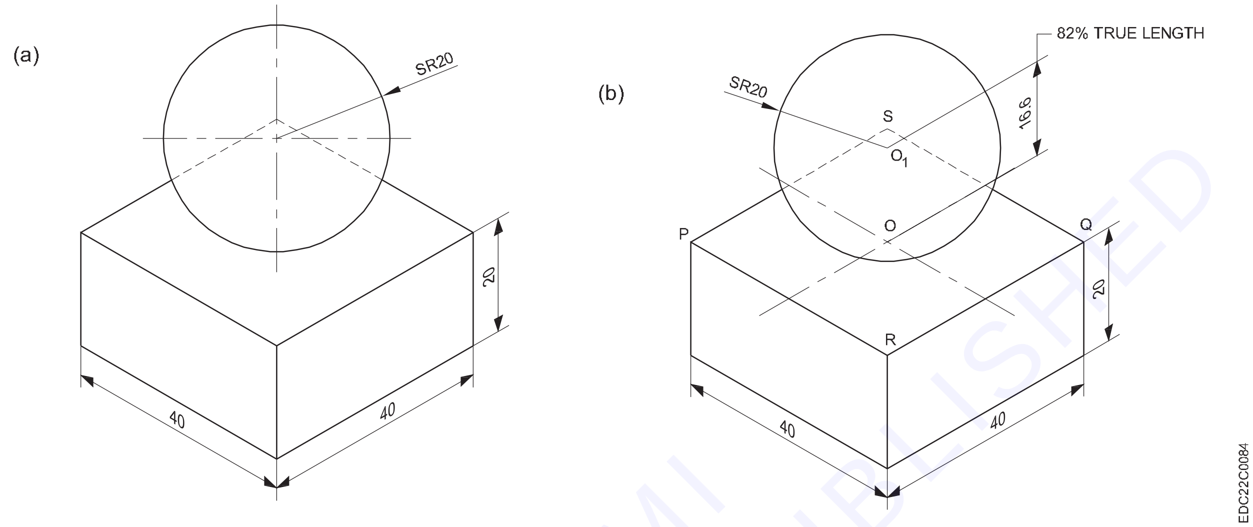
A sphere positioned at the centre of the top face of a square prism as shown in Fig 28. Draw the isometric projection.

- Draw the isometric projection of the square prism. (Fig 28a)
- Mark the mid points of the sides. Join the mid points and locate centre `O'.

- From `O' draw a vertical line and mark O_1, such that OO_1 is equal to isometric length (82%) of the radius 20 mm. (16.4 mm)
- With O_1 as centre and radius 20 mm, draw a circle which is the isometric drawing of the sphere.

Note: When a sphere is viewed in any direction its shape will be a circle of radius equal to the true radius of the sphere and hence the isometric drawing of the sphere will be a circle of radius equal to the true radius (1:1) of the sphere.

Fig 28



Exercise 5.29

Draw the oblique views in Ex.12.29 of work book according to the procedure given.

Reproduce the oblique view of the object shown in Fig 29.

- Draw the oblique axes (may be at any angle to the horizontal 30°, 45° or 60°) and extend. (Fig 29a)
- Mark off the overall length breadth and height of the object on the axis.
- Draw the enclosed box.
- Envelope the details of the object in the box by transferring dimension. (Fig 29b)
- Erase the unwanted lines and darken the required lines of the surfaces. (Fig 29c)
- Mark the dimensions and complete the drawing.

(In this oblique projection, the front face of the object is placed parallel to the picture plane and shown in its true size and shape)

Fig 29

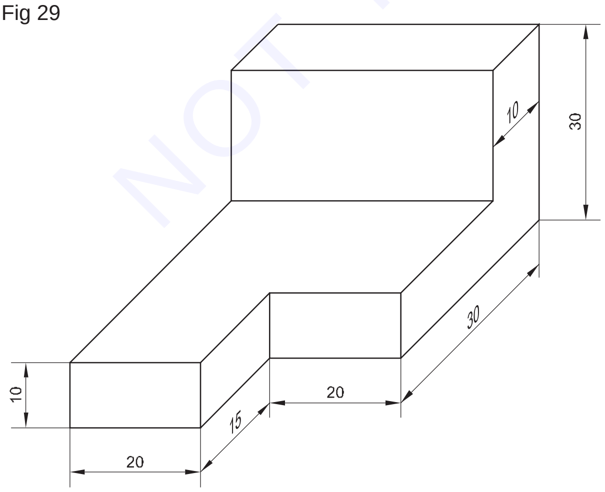


Fig 29 a

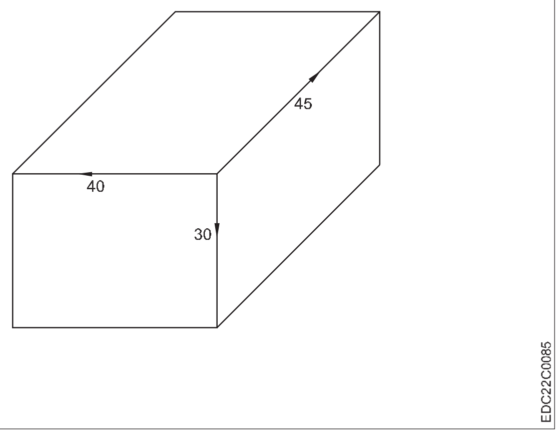


Fig 29 b

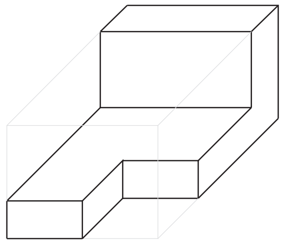
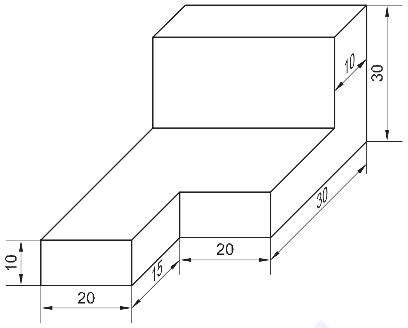


Fig 29 c



Exercise 5.30

Draw the oblique view in cavalier method of the component shown in isometric view. Consider each grid is of 5 mm. (Fig 30)

- Draw an enclosed box in oblique view to the overall size of the component i.e 50 x 25 x 40. (Fig 30a)
- Transfer shape of the view (isometric) to the box.
- Rub off the unwanted lines and complete the oblique view of the component.
- In cavalier method the projectors make an angle of 45° with the plane of projection and the inclined surfaces are drawn 1:1 scale. (Fig 30a)

Note: Construct grid of 5 mm in thin lines.

Fig 30

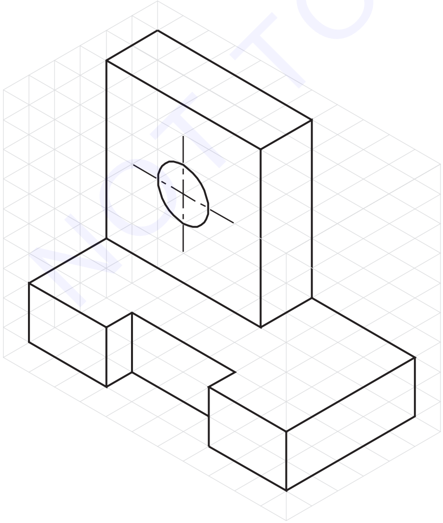
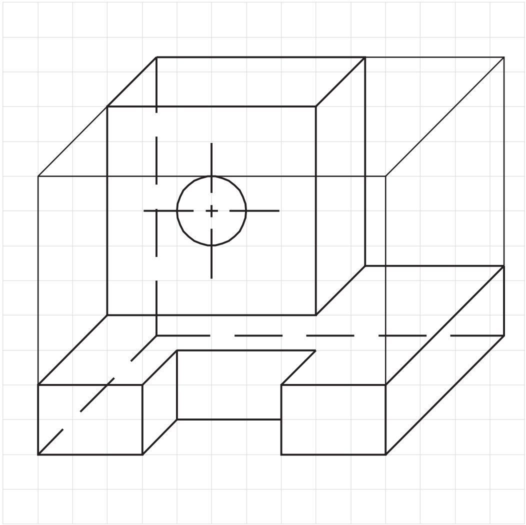


Fig 30 a



Exercise 5.31

Draw the oblique view of the bracket (Fig 31) by cavalier method.

- Construct boxes covering base and top.
- Envelope shape excluding the curved portion to the box drawn. (Fig 31 a)
- Locate and mark the centre of circle/curve on both surfaces. (Fig 31 b)
- Draw arcs and circles on both surfaces.
- Join the periphery of circles with a tangent.
- Rub off the unwanted lines and complete the oblique view. (Fig 31 c)
- Read and transfer the dimensions to the oblique view from the given view of the object.

Fig 31

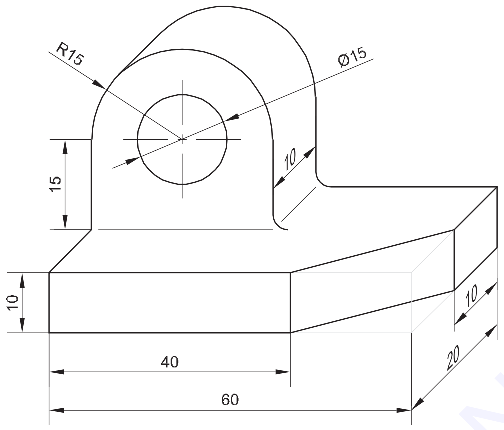


Fig 31 a

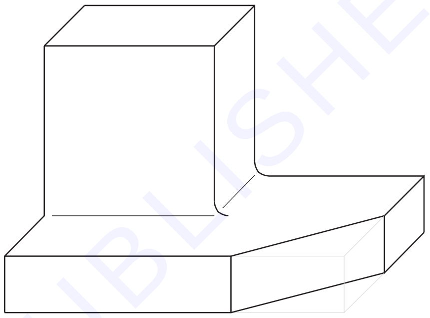


Fig 31 b

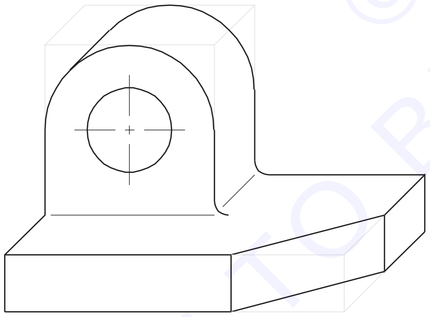
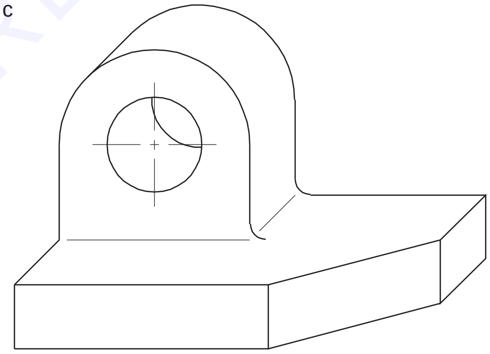


Fig 31 c



Exercise 5.32

Draw the oblique view of the component in cabinet method. (Fig 32)

- Draw the oblique projection of the component shown in cabinet method shown.
- Procedure of drawing this component is similar to that of the previous exercises except the inclined lines are to be drawn to half of the true length (inclined lines) as drawn to 12.5 mm instead of 25 mm.
- Figure 32 a shows the oblique view of the component in cabinet method.

Fig 32

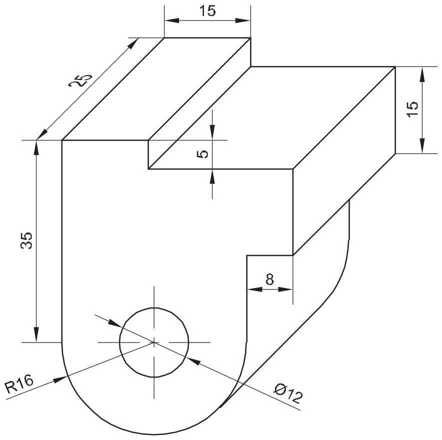
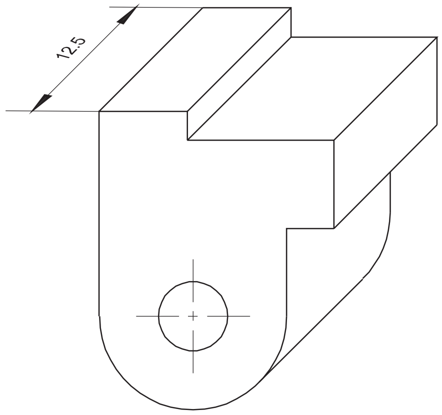


Fig 32 a



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Exercise 5.33

Draw the oblique view (cavalier method) of the hinge. (Fig 33)

- Draw a box in oblique view to the overall size of the hinge i.e 60 x 20 x 60.
- Envelope the details visualised from the multi-view. (Fig 33 a & b)

Fig 33

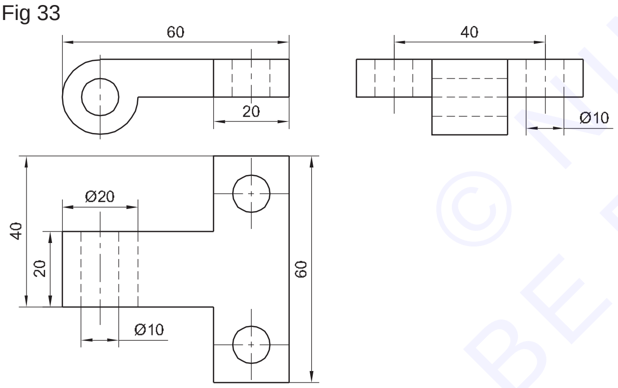

Fig 33 a

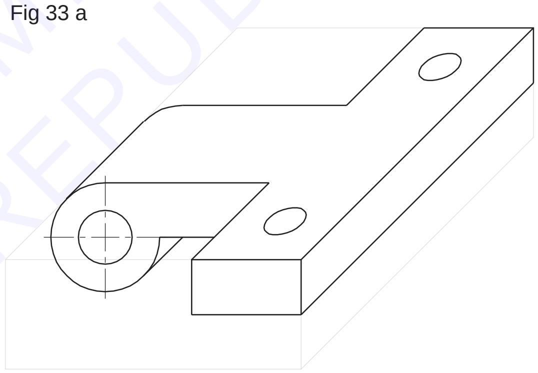

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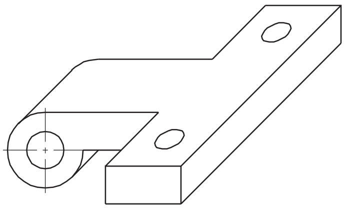
Fig 33 b



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EXERCISE 6 : Sectional views

Sectional views

In the normal Orthographic views (plan, elevation and side view), the internal details, their features and relative positions which cannot be seen are shown by dotted lines.

For example in the object shown in figure 1 the hole is invisible in the elevation and side view. Hence it is represented by dotted lines.

When there are too many dotted lines in a drawing (Fig 2) it is difficult to conceive the details of the object. In such a cases, details can be shown clearly and reading of drawing can be made easier by resorting to what are known as "Sectional views".

Fig 1

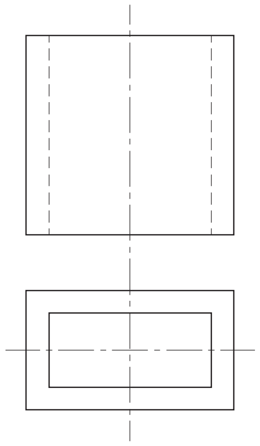
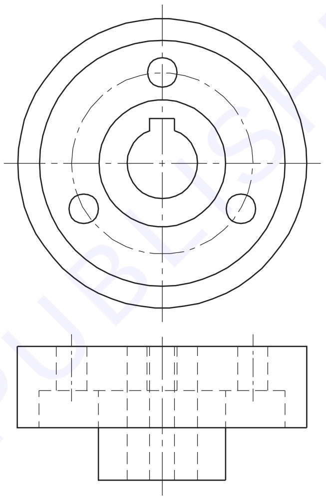


Fig 2



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Sectional views: For obtaining sectional views an object it is assumed to be cut by an imaginary plane called cutting plane. The part between the cutting plane and the observer is assumed as removed to reveal the internal details. Then the projection of part left out is projected/drawn as usual and the view thus made is the sectional view. (see Fig 3)

To distinguish a sectional view the surface formed when it is cut by the cutting plane is "hatched". (Fig 4)

Fig 3

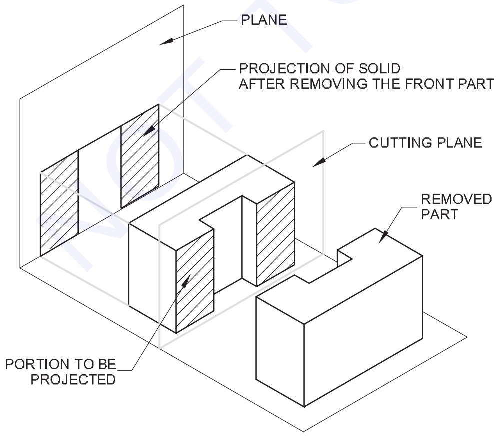
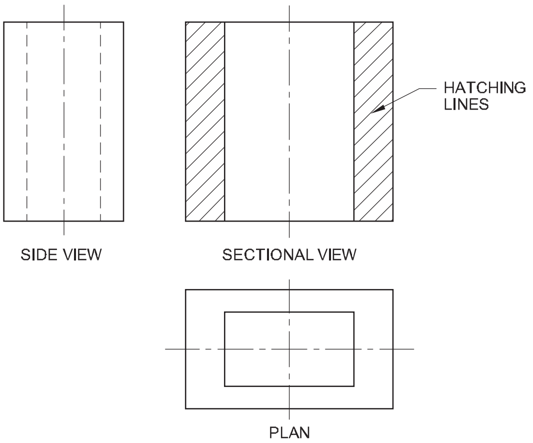


Fig 4



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Hatching means filling the surface with equi-distant parallel lines.

It may be recalled that the object whose internal details are to be drawn are assumed as cut by an imaginary cutting plane passing through them. The part between the cutting plane and the observer is assumed as removed and the surfaces cut are shown by the inclined parallel lines called section lines. The cutting planes are normally indicated by two capital letters i.e A-A, B-B etc. shown in Fig 5.

Fig 5 shows the general features of sectional views.

The general rules for the arrangement of the views apply equally when drawing sections.

Types of sections: Depending upon the details to be revealed the position of cutting plane can have various orientations. According to the orientation of cutting planes, sectional views are classified as:

– Full-section

– Half-section

– Section through two or more parallel planes

– Section on contiguous planes

– Section on two intersecting planes

– Removed section

– Revolved section

– Local or broken section.

Full-section: The component is cut by a straight cutting plane is divided into two parts. The part between the cutting plane and the observer is assumed as removed and the view of the cut surface will be a full-section. (Fig 6)

Fig 5

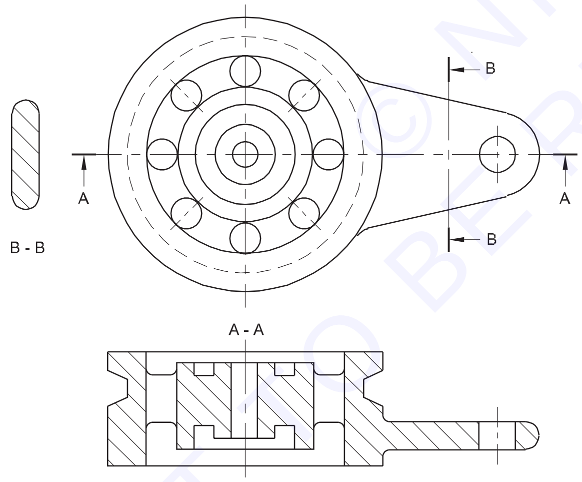
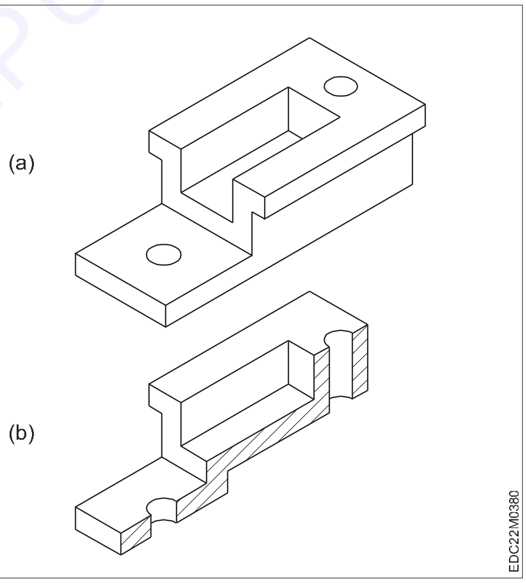


Fig 6



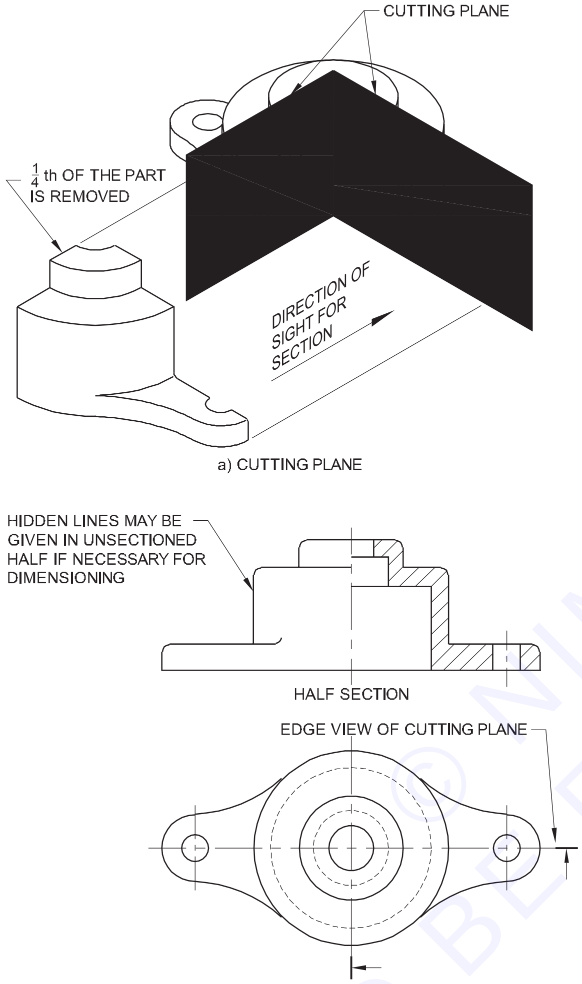
Where the location of a single cutting plane is obvious, no indication of its position is required. Where the locations is not obvious or where it is necessary to distinguish between several cutting planes, the cutting planes shall be indicated by means of thin chain line, thick at ends and change of direction.

Cutting planes are normally parallel to VP and sometime to HP or oblique also.

Half section: When a component is symmetrical it is not necessary to draw a full sectional view. In such case one half of the view is drawn in section and the other half is shown as normal view. Thus in one view we show both the external and internal details. For half section, the cutting plane removes 1/4th of the part. (Fig 7)

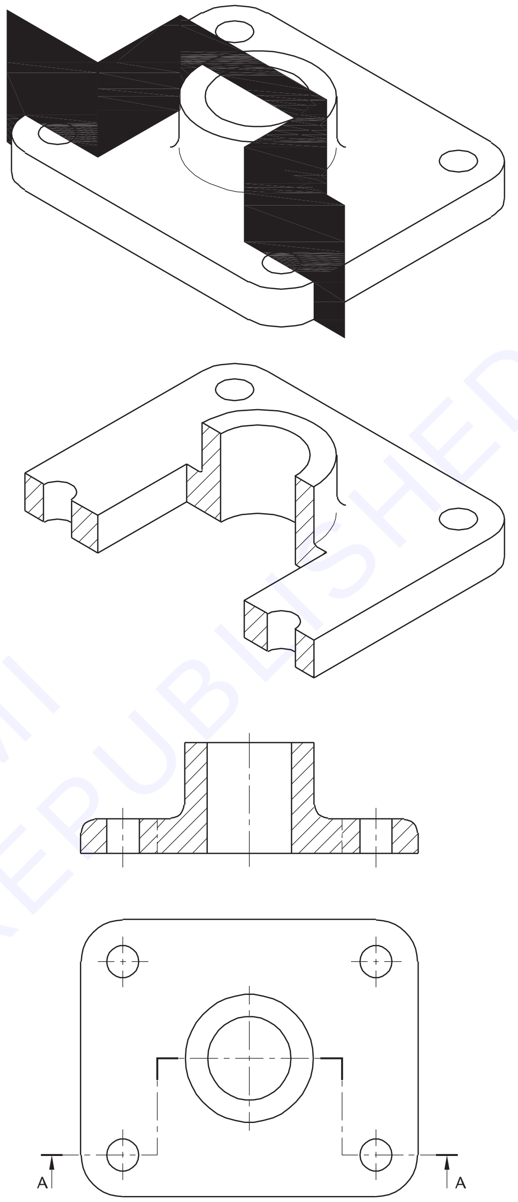
Off-set section: When the features of the component/object are not in one line a full-section or half-section does not reveal all the internal details. In such cases, the cutting plane is off-set as shown in Fig 8. The resulting sectional view is very much like a full-section expecting that the hatch area will not be in the same plane. In the sectional view in Fig 8 hatching line is staggered to indicate the change of plane.

Fig 7



HALF SECTION

Fig 8



OFFSET SECTION

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Section in contiguous planes: Certain components such as shown in figure 9 are bent offset at different position. In such cases the cutting plane follow the contour of the part. Here the sectional view is more like a full-section even though the cutting plane is not a single plane. (Fig 9)

Section in two intersecting planes: To reveal the details of the slots and holes in the part shown in figure 10, we will need sectional views along two planes meeting at an angle. Strictly speaking we have to draw different views for both the section planes. However in such cases the view on one plane is rotated to align with the other plane and view for both the cutting planes are shown in one view. In figure 10 the view on cutting planes OY is rotated to align with the view on OX.

Note: Certain parts like arms, webs are not sectioned. The cutting plane is assumed as passing just outside parallel to it.

Revolved section: The sectioning methods discussed so far may not be sufficient to reveal the certain features of a part.

Fig 9

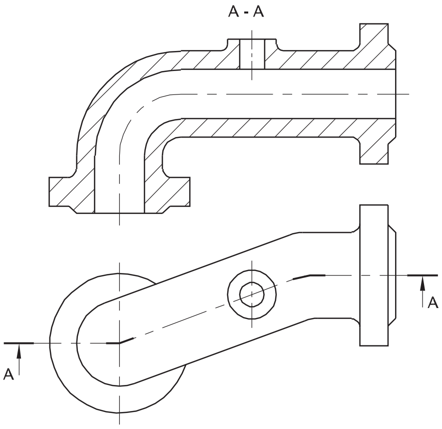
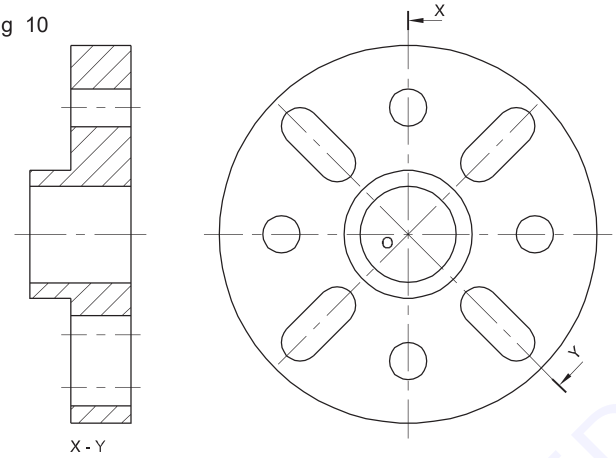


Fig 10



For example the crane hook shown in Fig 11 has varying cross-sections and this cannot be shown by any one of the sectional methods described earlier.

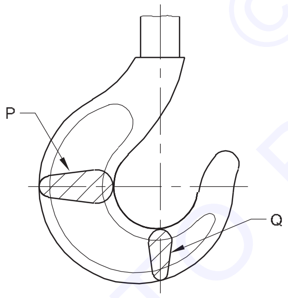
In such cases sections are taken at the desired position and the sectional view is drawn after as if the cut face is revolved as shown at P and Q in the figure.

Note: The outline of the revolved sections shall be drawn in continuous thin outline.

Removed sections: When the space does not permit to show the revolved section or it will be more clear if shown outside. The removed sections may be placed either near to and connected with the views by a chain, thin line or in a different position and identified in the conventional manner as shown in Fig 12.

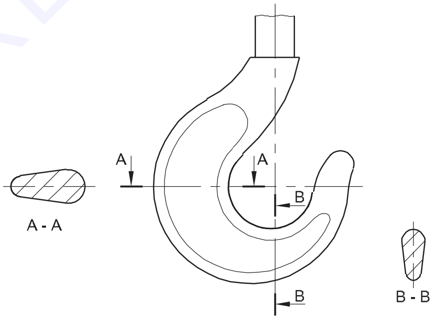
Local or Broken section: It often happens that only a partial section of a view is needed to expose the internal details. Such a section is limited by a break line as in Fig 13 is called a local or broken or part section.

Fig 11



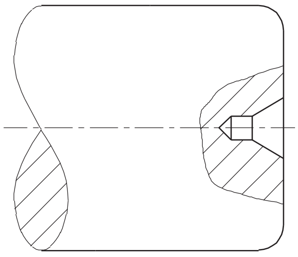
REVOLVED SECTION

Fig 12



REMOVED SECTION

Fig 13



BROKEN SECTION

Sectional views

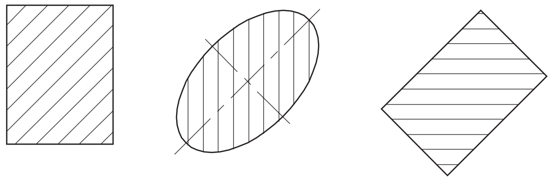
Hatching

Hatching angle: Hatching is used for making the sections evident. Hatching lines are thin lines and are usually drawn at an angle 45° to the horizontal and fill in the entire area undersection. But, depending upon the orientation of the area to be hatched the hatching lines may be horizontal, vertical or any convenient angle. (Fig 1)

Hatching assemblies: While hatching mating parts of an assembly hatching lines are drawn in different direction as shown in Figs 2a & 2b.

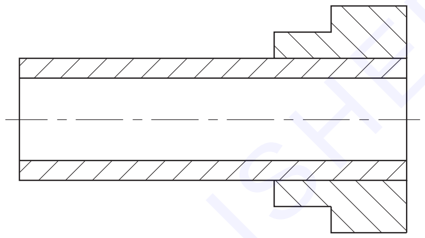
Hatching large areas: In the case of large areas, the hatching may be limited to a zone following the contour of the hatched area. (Fig 3)

Fig 1

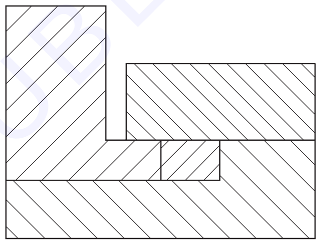


PREFERRED HATCHING ANGLE

Fig 2



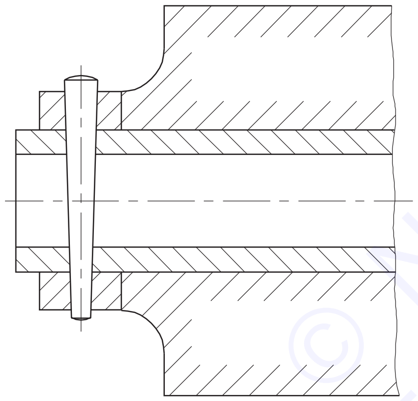
a) HATCHING OF ADJACENT COMPONENTS



PITCH OF HATCHING LINES IS VARIED
TO DISTINGUISH PARTS

b) HATCHING OF MORE THAN
TWO ADJACENT COMPONENTS

Fig 3



HATCHING OF LARGE AREAS

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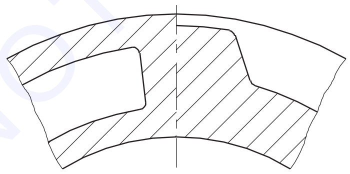
Hatching areas in different parallel planes: Where sections of the same part in parallel planes (offset) are shown side by side, the hatching lines should be similarly spaced, but offset along the dividing line between the sections. (Fig 4)

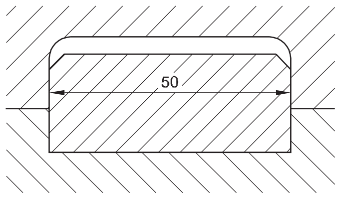
Dimensioning within the hatched area: Hatching may be interrupted for dimensioning, if it is not possible to place these outside the hatching. (Fig 5)

Fig 4



HATCHING OF A PART IN
DIFFERET PARRELLEL PLANES

Fig 5



HATCHING INTERUPTED FOR DIMENSION

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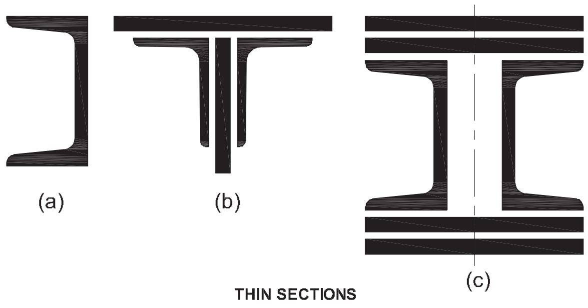
Thin sections: Thin sections may be shown entirely black. Thin space is left between adjacent sections of this type. (Fig 6a,b & c)

Omission of hatching lines: There are several cases where hatching lines are deliberately omitted even though they are cut by the section plane. For example ribs/webs are not hatched to avoid a false impression of thickness and solidarity. (Fig 7a & b)

Fig 6



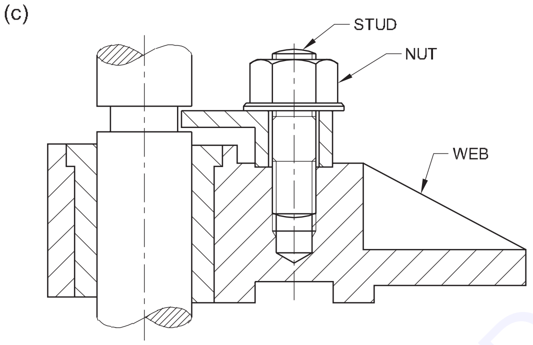
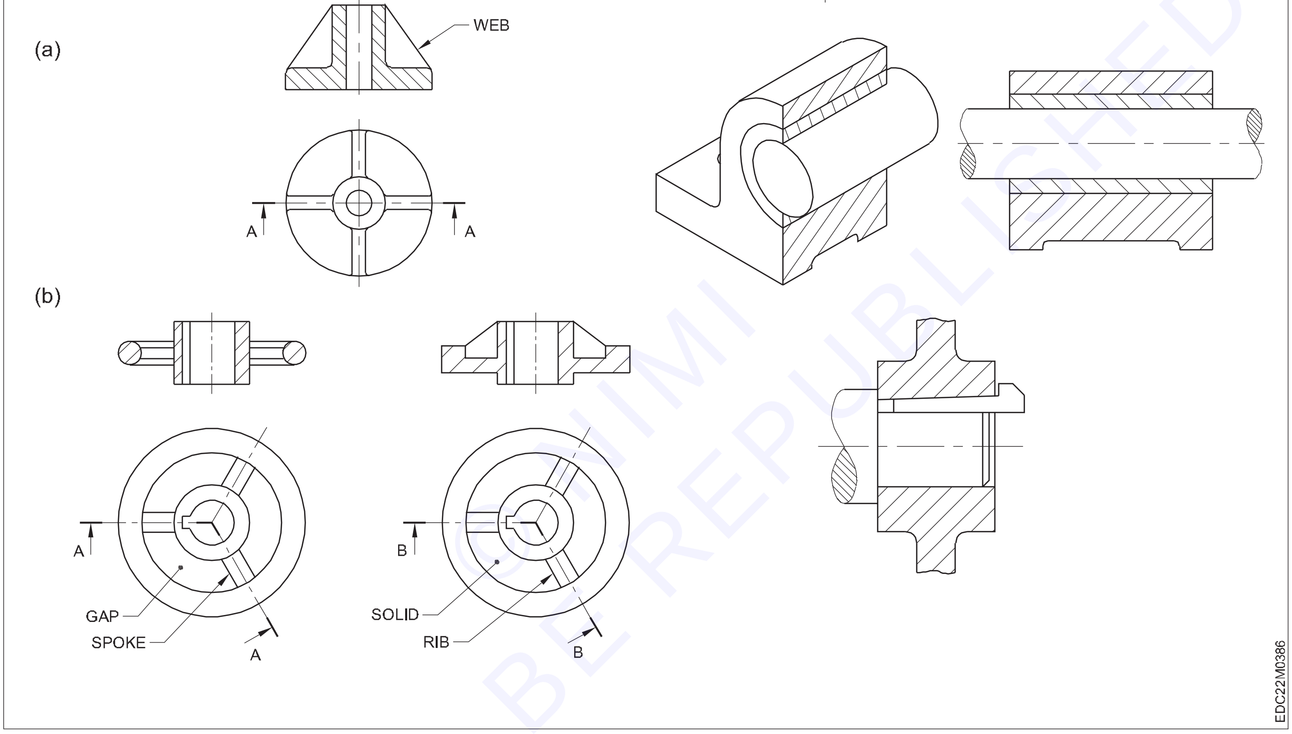


Fig 7



In some cases even though section planes passes through it is assumed as not cut. Hence, they are not hatched. Examples of these are rivets, bolts, nuts, shafts, balls, rollers, keys and pins. (Fig 7c)

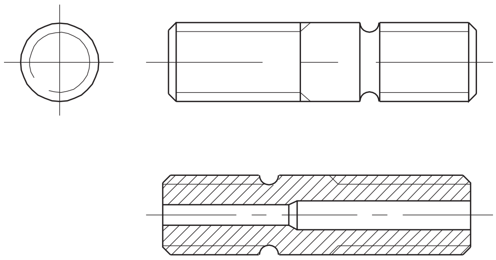
Fig 8 & 9 shows the conventional method of sectioning of external thread and internal thread.

Fig 10 to 16 shows different sections such as section on one plane, section on two parallel planes and etc. The methods shown in these features to be applied wherever applicable.

Hatching patterns can be used as a means to broadly idnicate the material of the part/object. To differentiate different materials in sectional view BIS laid out standards (IS:11663) on conventional representation of materials and the same is given in the table below:

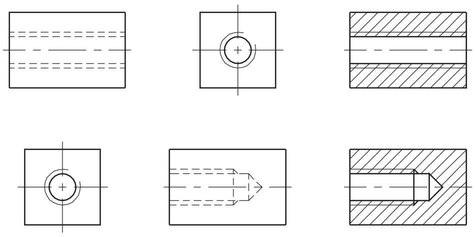
IS:11663 - 1986

Fig 8



CONVENTION OF EXTERNAL THREAD

Fig 9



INTERNAL THREAD

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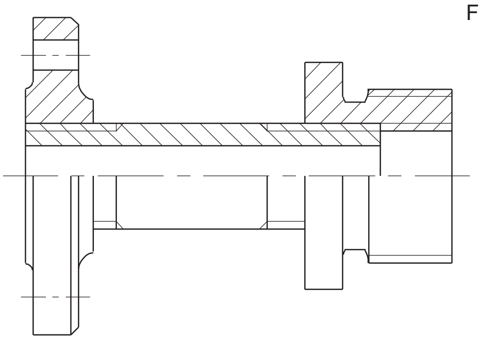
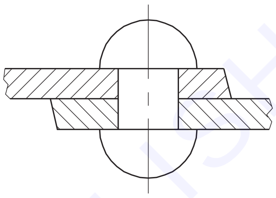
Fig 10



ASSEMBLY OF THREADED PARTS

Fig 11



SECTION OF RIVETED JOINT

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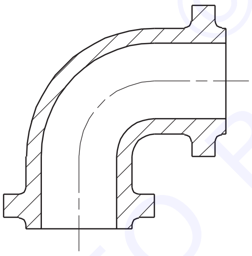
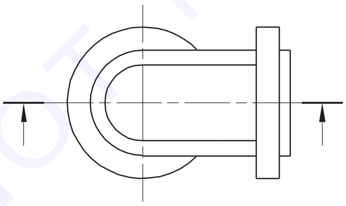
Fig 12

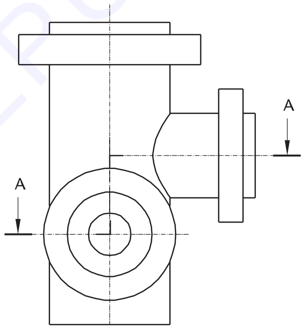


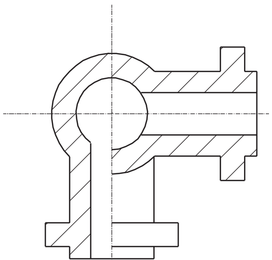
SECTION A-A



SECTION IN ONE PLANE

Fig 13





SECTION AA

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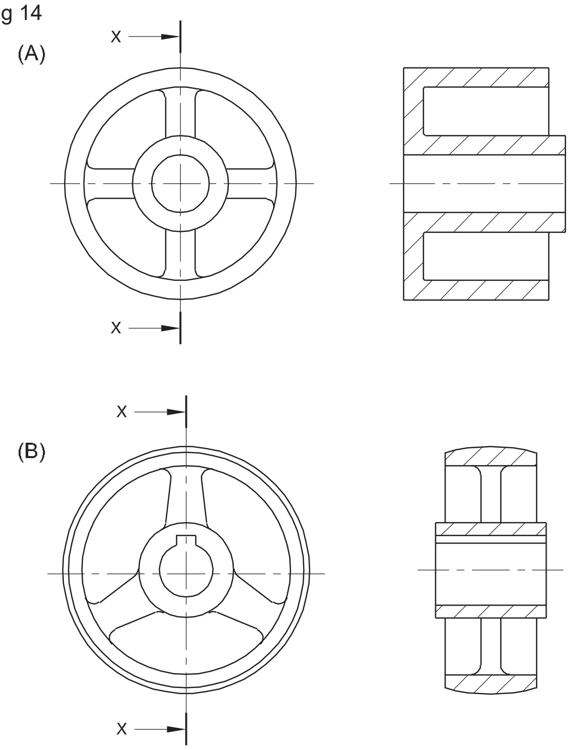
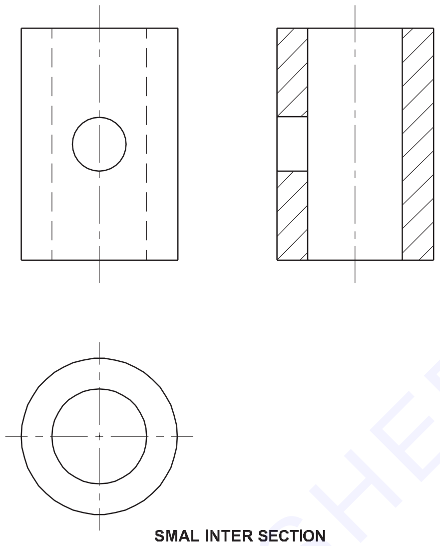
Fig 14



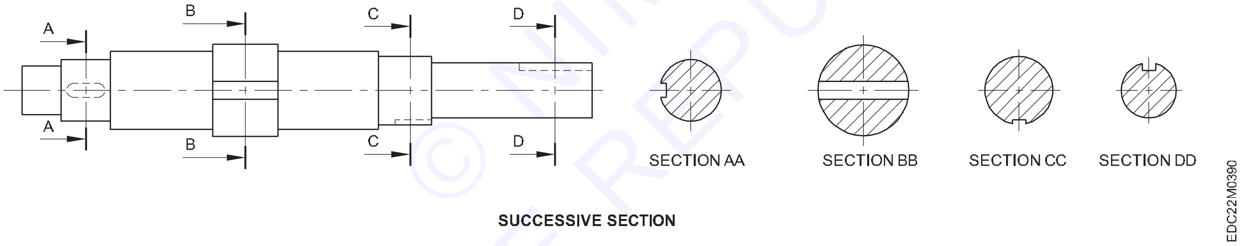
SECTION THROUGH CENTRE A RIBOR WEB

Fig 15



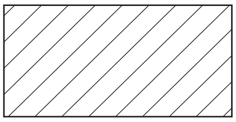
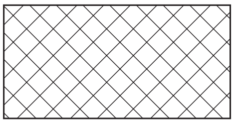
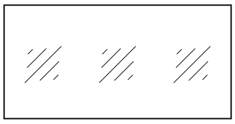
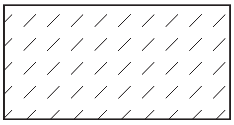
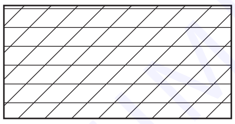
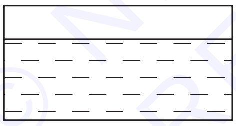
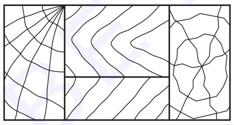
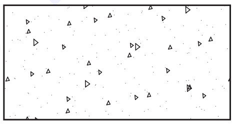
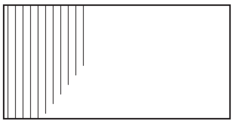
SMAL INTER SECTION

Fig 16



SUCCESSIVE SECTION

CONVENTIONAL REPRESENTATION OF MATERIALS USING HATCHING

Type	Convention	Material
Metals		Steel, cast iron, copper and its alloys, aluminium and its alloys etc.
		Lead, zinc, tin, white-metals etc.
Glass		Glass
Packing and insulating material		Porcelain, stoneware, marble etc.
		Asbestos, fibre, felt, synthetic resin products, paper, cork, linoleum, rubber feather, wax, insulating and filling materials
Liquids		Water, oil, petrol, kerosene etc.
Wood		Wood, plywood etc.
Concrete		-
Stacked lamination		-

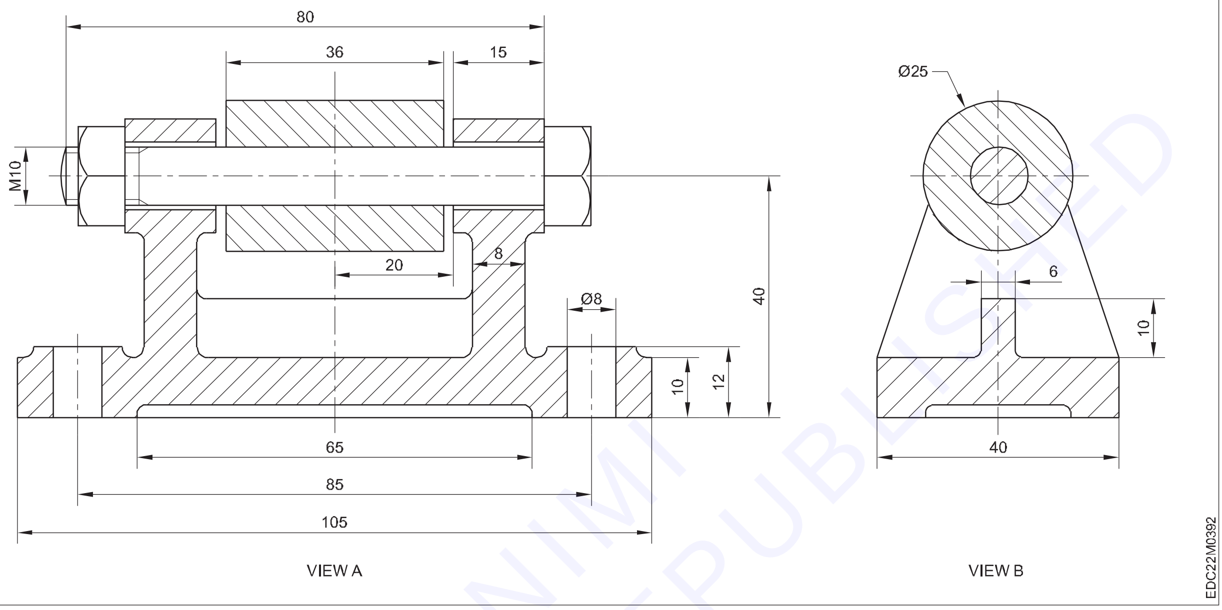
Procedure

Sectional views

Exercise 6 .1

The sectional views of an assembly are shown in Fig 1. Draw the views as it is and indicate the cutting plane for view `B'.

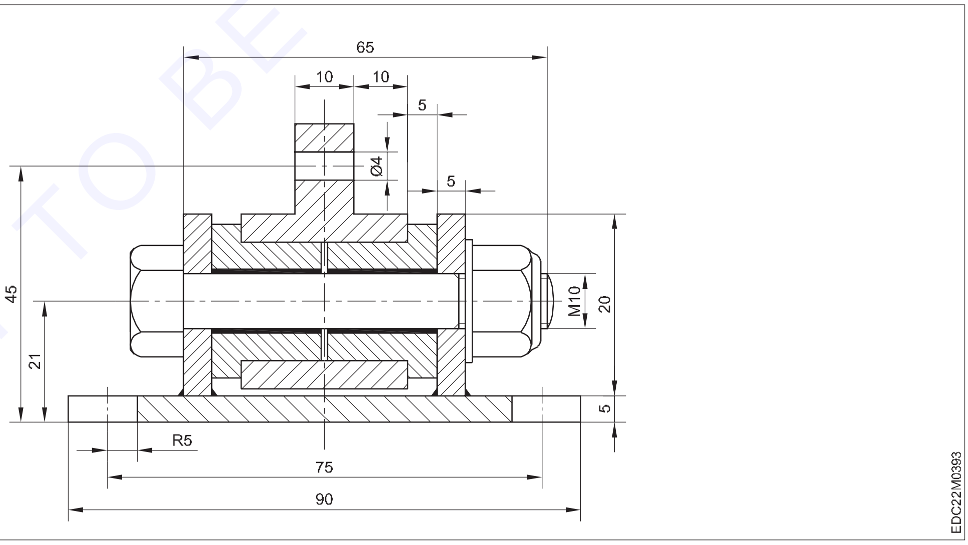
Fig 1



Exercise 6.2

Draw the sectional view of an assembly shown in figure and also indicate the following features. Bolt - Nut - Thin section welded joint - Slot - Hole.

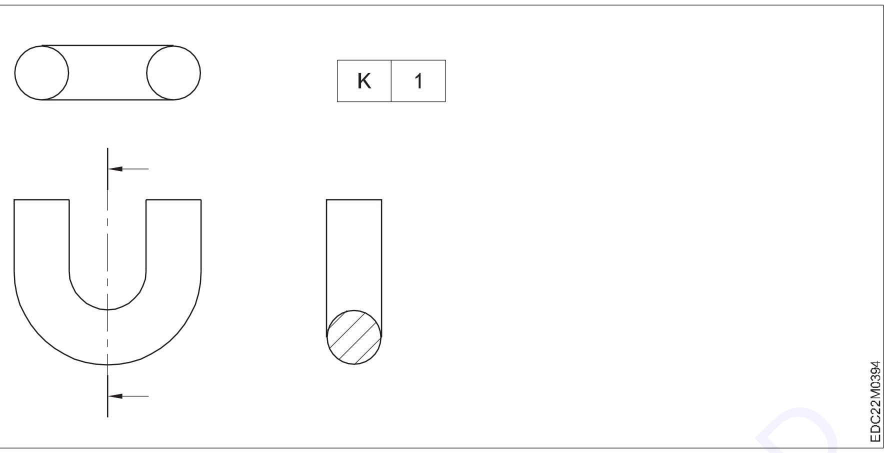
Fig 2



Exercise 6.3

Three views of ten objects are given in the work book (Ex.13.3) and numbered. According to the position of the cutting plane shown, select the appropriate sectional view. Place the alphabet in the space provided with the views in the work book.

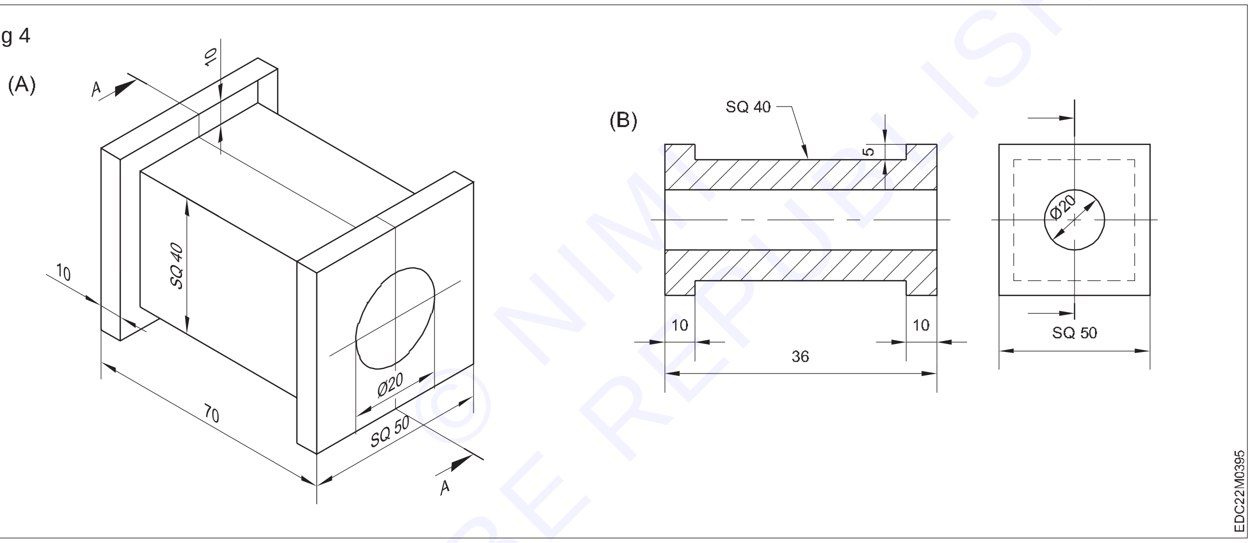
Fig 3



Exercise 6.4

Draw the full sectional view of the object shown in Fig 4 along the cutting plane AA.

Fig 4



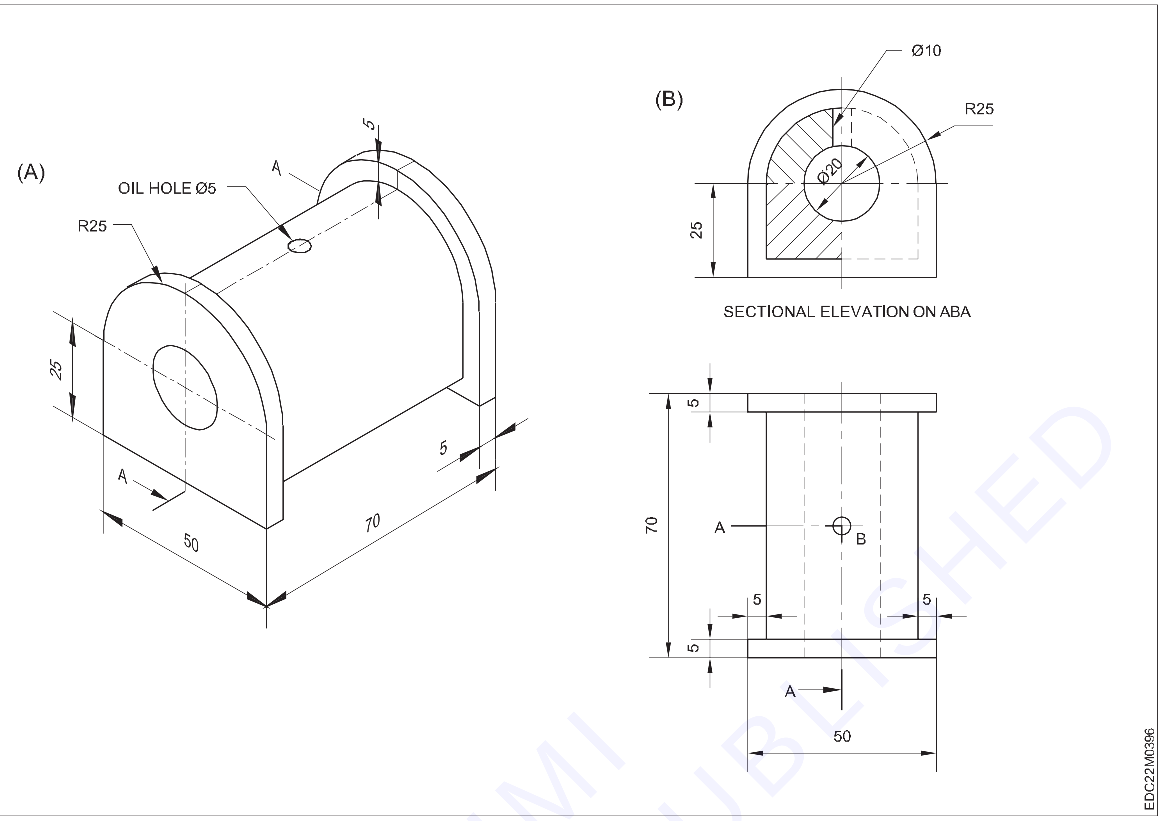
- Draw the front view as it appears externally in thin lines.
- The part between the cutting plane and the observer is assumed as removed and then viewed.
- The vertical lines for the edges of the holes and the horizontal lines for the top and bottom surfaces are drawn in thick lines.
- Draw the section lines on the surfaces cut by the cutting plane.
- Draw the side view assuming the object is to be uncut.
- Remove the unwanted lines, give dimensions and complete the drawing.
- Follow the technique of hatching as given in theory.

Exercise 6.5

Draw the half sectional front elevation along the cutting plane ABA and plan. (Fig 5)

- Draw the plan as shown in figure 5B.
- Draw the cutting plane line.
- Project from the plan to elevation.
- Draw the elevation as it appears externally.
- In this section, one fourth of the plan is assumed to be cut by the cutting plane.
- View the elevation after removal and draw the edges of the sectional part in thick lines.

Fig 5



- Complete the view and draw the sectional lines.
- As this object is off two halves, the section lines has to be drawn in opposite direction.
- Remove the unwanted lines and give dimensions.

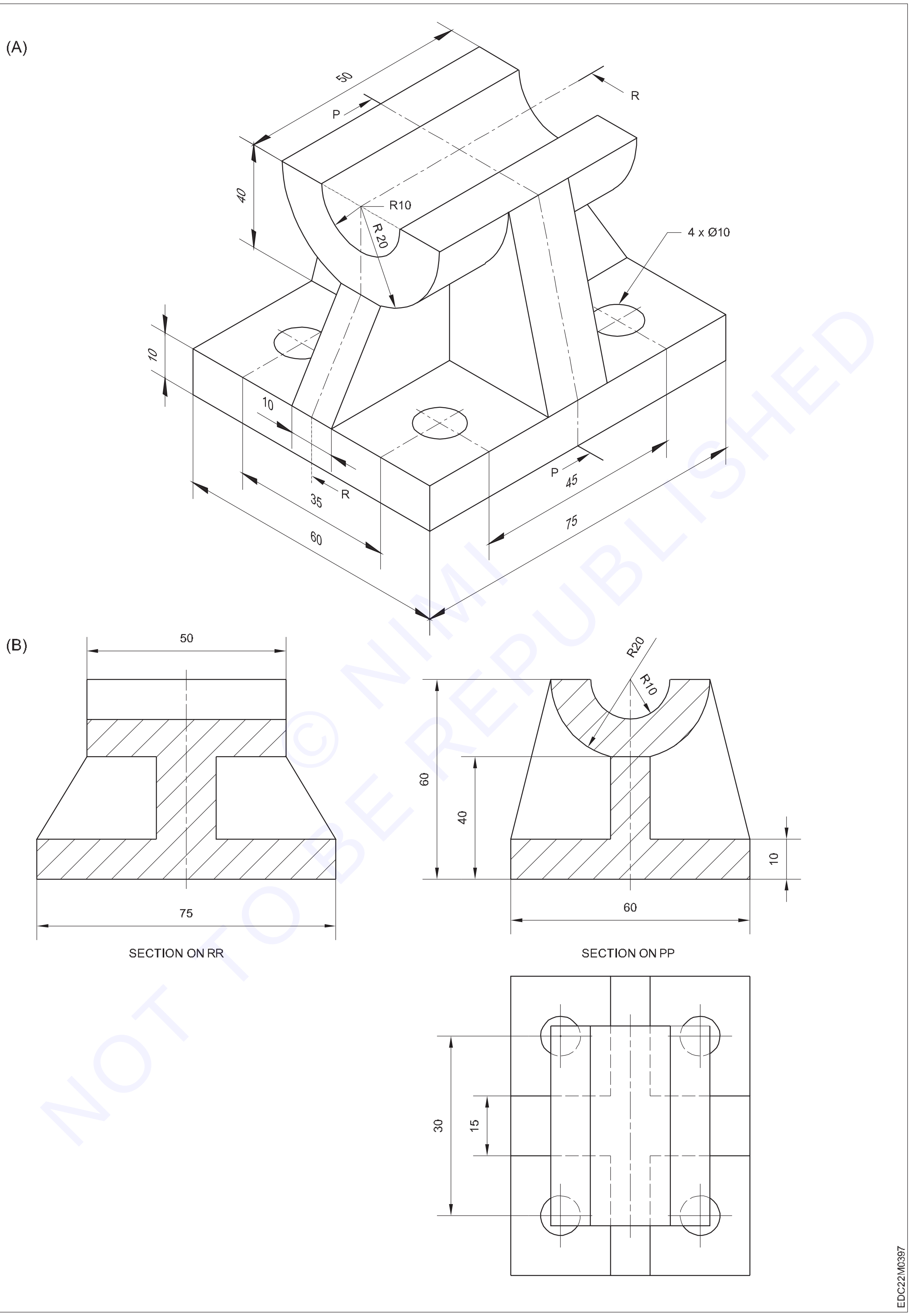
On sectional view the section lines has to be drawn after dimensioning.

Exercise 6.6

Draw the sectional front view, sectional side view and the plan of the casting shown. Give all required dimensions. (Fig 6A)

- Complete the sectional front view and side view as shown in Fig 6 and draw sectional lines.
- Project the front view and draw the plan assuming the object is in full.
- Remove the unwanted lines, give dimensions and complete the drawing. (Fig 6B)
- Similar to previous exercises, draw the front view and side view as viewed externally.
- For drawing sectional front view remove the part between the observer and the cutting plane PP and then viewed.
- For drawing sectional side view remove the part between the cutting plane RR and observer and then viewed.
- In both cases draw all visible edges in thick lines.
- Even though the cutting plane PP and RR passes through the rib, it is assumed that the cutting plane passes just outside parallel to the part.
- Hence the middle part of the rib is assumed to be cut and draw section lines.

Fig 6


(A)
50
P
R
40
R10
R 20
4 x Ø10
10
10
35
R
60
P
45
75
(B)
50
75
SECTION ON RR
R20
R10
60
40
10
60
SECTION ON PP
30
15
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Exercise 6.7 & 6.8

Draw sectional front elevation, plan and side view for the given objects. (Fig 7 & 8) Take direction of `X' as front view.

Fig 7

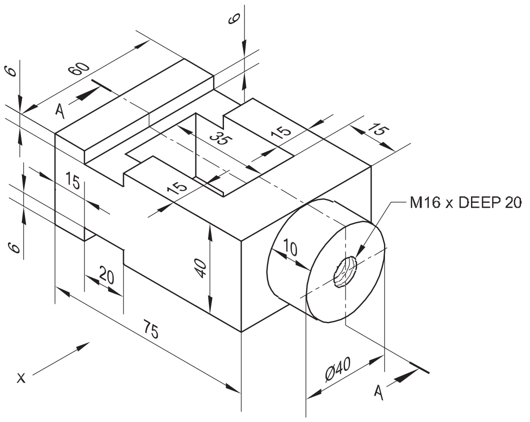
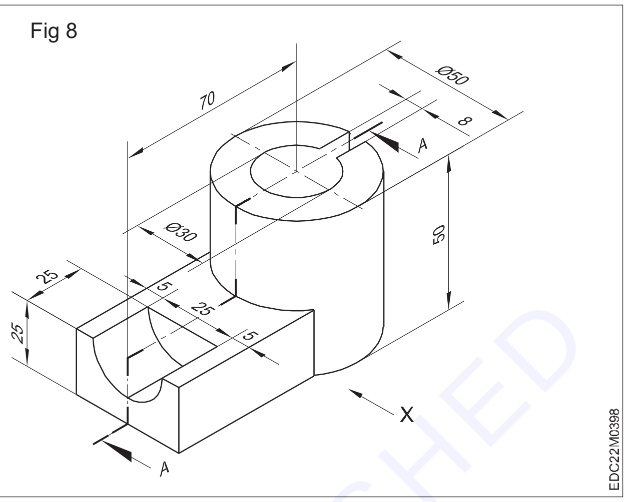


Fig 8



Exercise 6.9 & 6.10

Draw sectional elevations, side view and plan for the objects shown. (Fig 9 & 10) Draw in 1:2 scale. Assume the direction of `X' as front view.

Fig 9

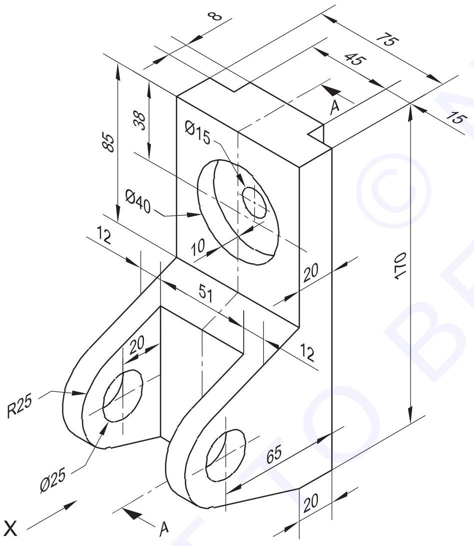
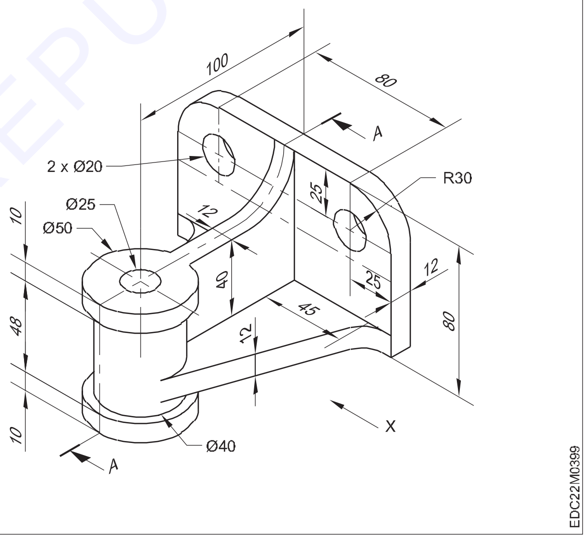


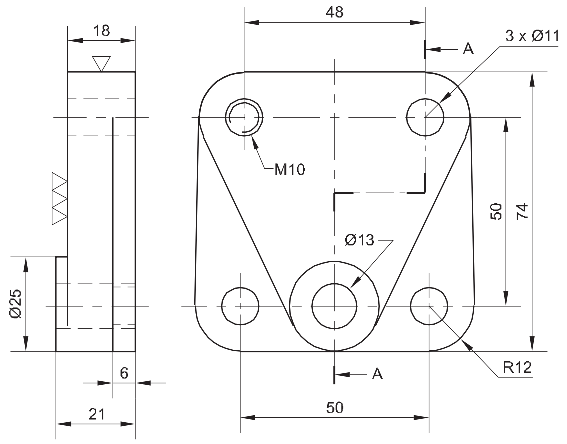
Fig 10



Exercise 6.11

Redraw the given front view and draw sectional side view along AA. (Fig 11)

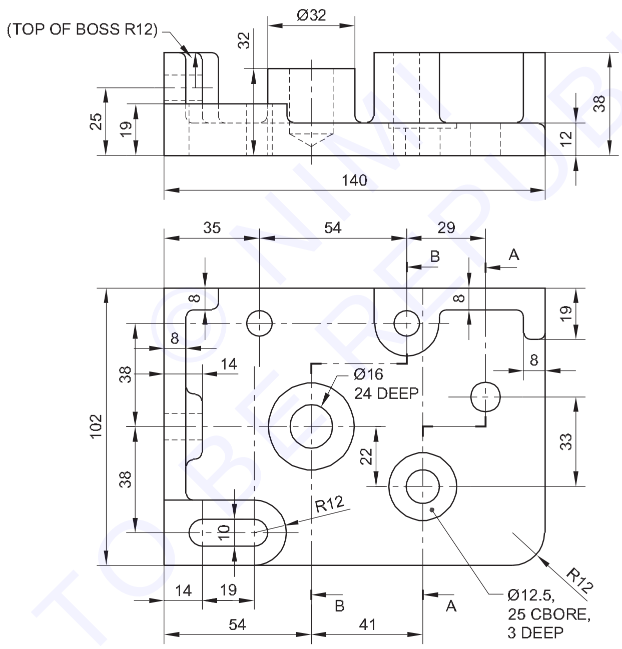
Fig 11



Exercise 6.12

Draw sectional views for the component shown in Fig 12 along the cutting planes, (1) A-A (2) B-B Scale 1:2.

Fig 12



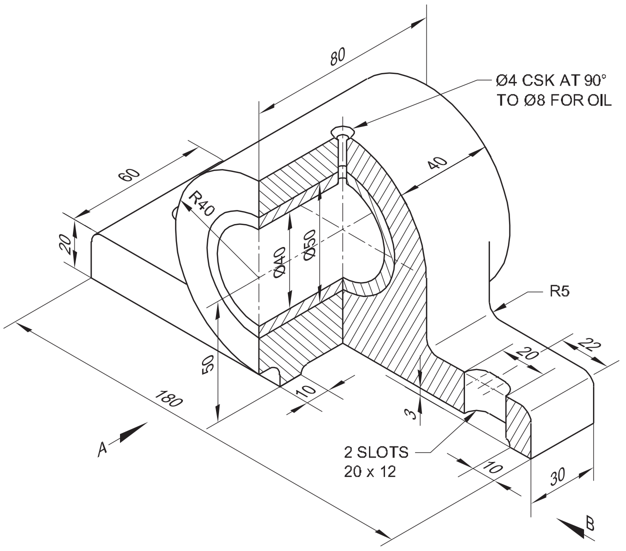
- Reproduce the views given in 3rd angle projection in 1:2 scale.
- Remove the part along with the cutting plane A-A and draw the sectional side view on right side of the front elevation.
- Remove the part alongwith the cutting plane B-B and draw the sectional side view on left of the front elevation.
- Follow the previous exercises procedures to complete the drawing.

Exercise 6.13

Draw half sectional elevation as seen from `A'. Draw half sectional side view from `B' and plan. (Fig 13)

- Draw the elevation as views from `A'. Left half in external view and right half in sectional view.
- Similarly draw in side elevation left half in sectional view and right half in external views.
- Draw the plan assuming the object is in full.

Fig 13

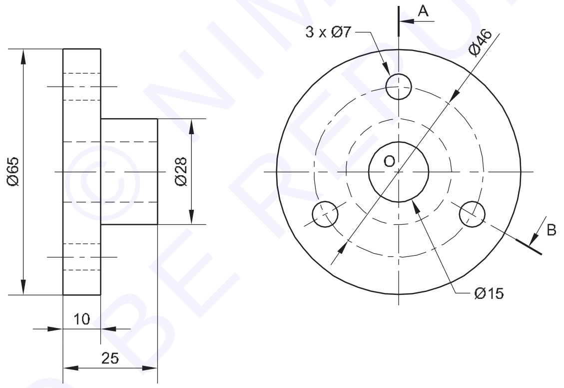


- Complete the drawing, following the previous exercises procedures.

Exercise 6.14

Draw sectional views along the cutting plane AB for the component shown in Fig 14.

Fig 14



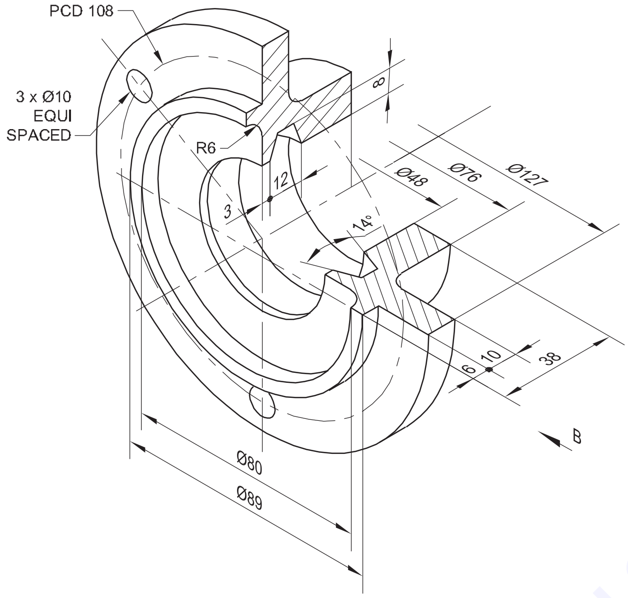
- The cutting plane is passing through two different planes.
- in such cases, as explained in theory, the view on cutting plane OB is rotated to align with the view OA and then viewed for sectional view.
- Draw the sectional elevation as said above and complete the drawing.

Exercise6.15

Draw half-sectional elevation as seen from `B' and add plan. (Fig 15)

- Select suitable scale according to size of the sheet in work book. (1:2)
- Follow the procedures of previous exercises.
- 3 x ϕ 10 holes equally spaced is given. Divide P.C.D into 3 equal parts i.e 120° C to C starting from vertical centre line as shown in Fig 15.

Fig 15



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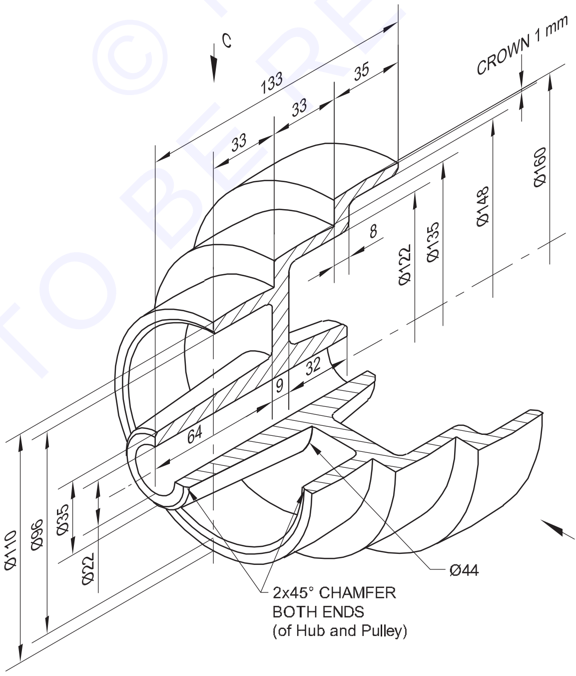
Exercise 6.16

Draw the half-sectional front elevation and half sectional plan for the given stepped cone pulley shown in Fig 16.

- Select convenient scale according to size of drawing sheet.
- Follow the previous exercises procedures and complete the drawing.

Fig 16



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EXERCISE 7 : Development and intersection

Development and intersection

Development of surfaces - Introduction

In our day-to-day life, we come across a number of objects. eg. Tins of round, square or rectangular shapes, funnels, hoppers, ducts, chimneys, trunk boxes, machine guards, medicine cartons etc are made of sheets metals and cardboards.

The process of manufacturing these objects involves cutting the flat sheet to the required size and then "folding it" to the final shape. The term development refers to the size and shape of the unfolded sheet or sometimes called as blanks. Development also implies the geometrical process of finding the size and shape of unfolded sheet. Hence, the development of surface plays very important role, such that enabling a machine to cut proper size of the sheet with reference to the development and to fold at proper places to obtain the required shape of the objects, say boilers, boxes, buckets, chimneys, hoppers, ducts etc.

The development of surfaces can be defined as the unrolling or unfolding of all the surfaces of the objects on a plane.

Types of surfaces

All objects are bounded by geometric surfaces, and so developments are made by the application of basic graphic and geometric principles in co-ordination with mathematics. As different shapes has to be joined together in many cases, the principle of intersections of solids are closely related to developments of surfaces.

A geometric surface is formed by the motion of a straight line or curved line, such surface formed by a straight line is identified (Fig 1) as ruled surface. It may be plane, single curved or warped surface. It is ruled surface generated by a straight line moving with one point along a straight line and another point is parallel with the straight line.

The surfaces may be classified as follows:

- Solids bounded by plane surfaces. Eg. cube, prism, pyramids etc. (Fig 2)

Fig 1

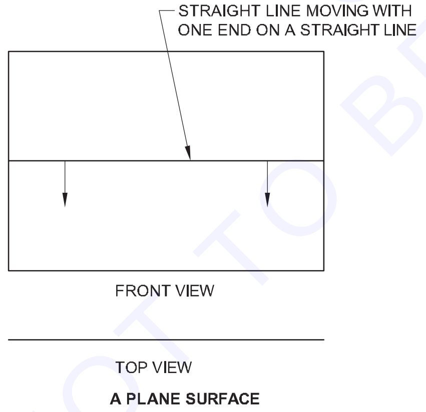
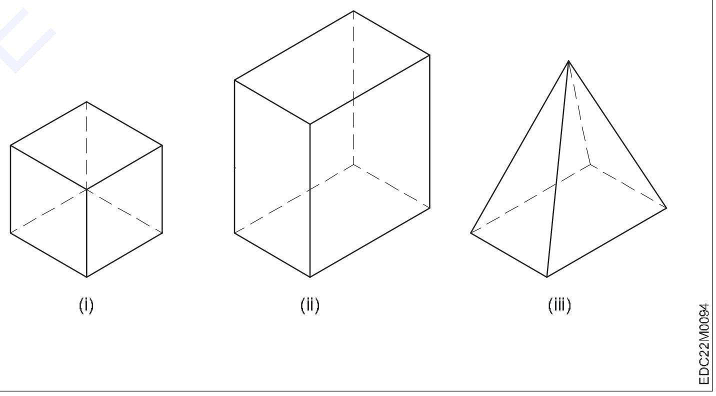


Fig 2



- Solids bounded by single curved surfaces. It can be unrolled to a plane. Eg. cylinder, cone etc. (Fig 3)
- Solids bounded by double curved surfaces. It is formed by revolving a curve about a straightline. Eg. sphere, paraboloid etc. (Fig 4)

Fig 3

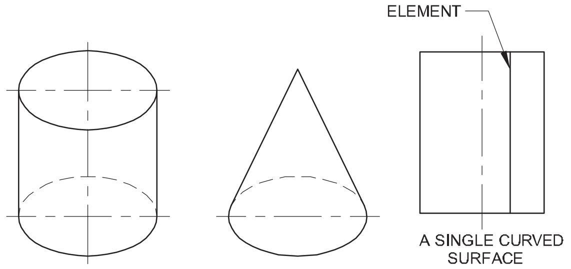
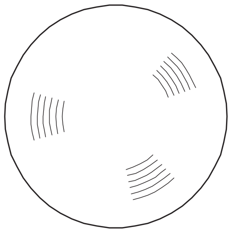


Fig 4



Methods of development

Parallel line method (Fig 6i)**:** This is method used for such of those object having parallel elements i.e developing prisms and single curved surfaces like cylinders, in which all the edges/generators of lateral surfaces are parallel to each other.

Radial line development (Fig 6ii): It is used for developing the surfaces of pyramids and also for single curved surfaces like cones, having its apex as centre and the slant edges or generators are taken as radius for drawing its development. Radial line development has the fold lines, coming from a point which is known as apex.

Intersection: In the context of engineering drawing, the term intersection refers to the common lines that are formed when two surfaces meet. In fact every edge of a solid is a line of intersection. We are more convinced with the lines of intersection formed when surfaces of two solid meet. Fig 5 shows the few examples of lines of intersection.

Intersection lines or just intersections are of great importance to both for making drawings for fabrication work.

Fig 5

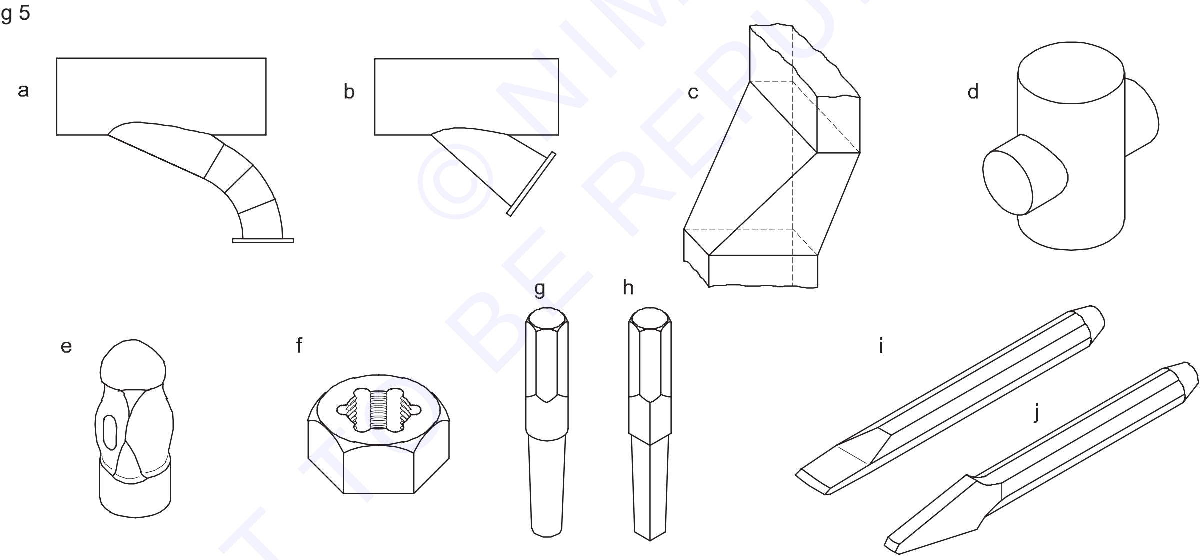
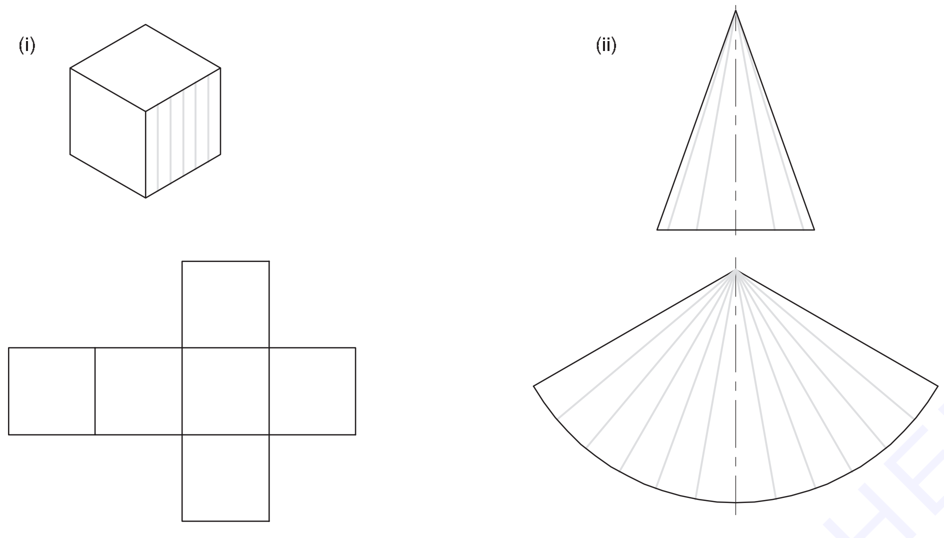


Fig 6



PARALLEL LINE DEVELOPMENT OF A CUBE

RADIAL LINE DEVELOPMENT OF A CONE

When drawing the views of intersecting solids, lines of intersection will have to be drawn in order to complete the views. For example Fig 7 shows a solid when two cylinders meet at right angles. Without the line of intersection cd, the view is incomplete. To draw the intersection lines in its correct form, principle of projections have to be followed and this will involve extensive geometrical construction.

In fabrication work, especially in sheet metal fabrication accurate intersection lines are a must for drawing development.

For example, the development of the part 2 of Fig 7 is shown in Fig 8. The accuracy of the fabricated cylinder is much dependent on the accuracy of the curve cd. To plot the line cd, intersection line cd (Fig 7) has to be drawn accurately.

Fig 7

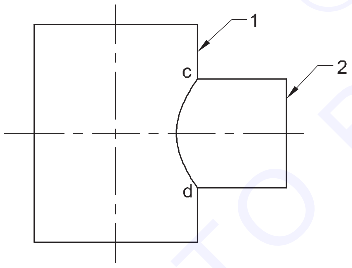
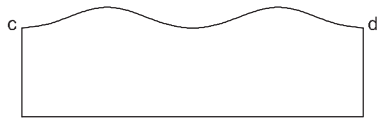


Fig 8



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Depending on the nature of the meeting surfaces, intersection lines may be straight lines or curves. When both the mating surfaces are flat (plain) intersection will be straight lines (Fig 5c). If one or both the mating surfaces are curved, the intersection lines are curved. (Fig 5a,b & d)

Methods of drawing intersection lines: In general intersection lines/curves are drawn by using the method of projection.

By going through the construction in figure, one can appreciate the principle adopted in drawing the intersection curves. (Fig 7 is a simple example). Our aim here is to draw the curve cd (Fig 8), since cd is a curve we have to plot it by finding intermediate points such as P_1,P_2,P_3. Actually P_1,P_2,P_3 etc are the projection points lie on the intersection. (Ref. Fig 9)

To plot the points required for drawing the intersection curve we can use either the line method or cutting plane method.

Line method (Fig 10)**:** Mark the points 1",2",3" on the circumference of the circle in the side view. (preferably symmetrical)

Fig 9

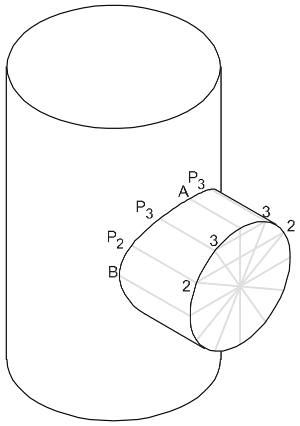
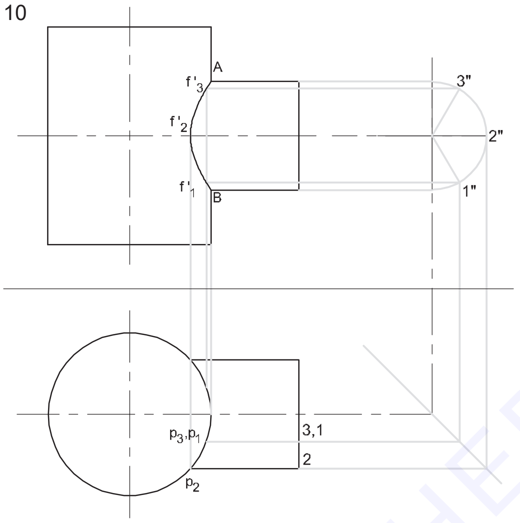


Fig 10



Project from side view and locate points P_2; P_1/P_3 in plan. Now these lines P_3 - 3, P_1 - 1 and P_2 - 2 are the true lengths of the generators or simply lines drawn from points 3, 1 and 2. These lines will also form the true lengths in the elevation.

Draw projectors of indefinite lengths from points 1",2" & 3" in the side view towards the elevation.

To fix the position of the above projectors, draw projectors vertically from points such as P_3/P_1 and P_2 to intersect with the corresponding lines in the elevation. Mark the intersecting points f_1',f_2',f_3' and draw the required curve.

Note: The above step was to simply transfer the true length of generators P_1 - 1, P_2 - 2 from plan to elevation.

Since we have drawn the projection of generators to get the points f_1',f_2',f_3'. This method is called generator method. This is also called as line method.

Cutting plane method (Fig 11)**:** In cutting plane method, we assume that the cylinders are cut by a series of parallel cutting planes passing through such points as 3-3, 2-2 etc.

In figure 9, the cylinders are shown after they are cut by a plane passing through 3 - 3. The line f3 - 3 is the same as the generating line f3' - 3" int he front view of figure and hence the rest of the procedure is same as that of line method.

Note: Though both line method and cutting plane method are appearing to be same, they are conceptually different. While any one method can be followed in many cases, but cutting plane method is more useful in solving cases in which none of the projections shows a line view of the surface of a solid.

Critical point or key point (Fig 12 & 13): There is no standard shape for intersection curves/lines. The curves/ lines may change direction. The point at the direction changes are referred as critical or key point. While drawing generator lines or cutting plane, one of the lines/cutting planes should invariably pass through critical/key point.

Figure 12 shows the point 4 is the critical point, whereas in Fig 13 points m and k are the critical points.

Fig 11

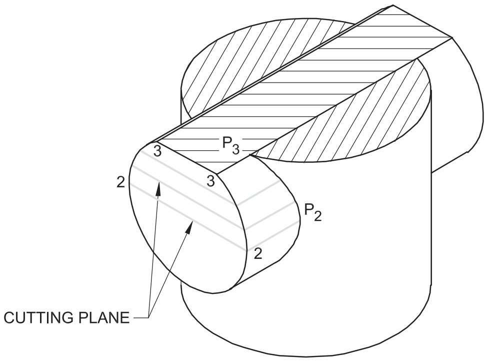


Fig 12

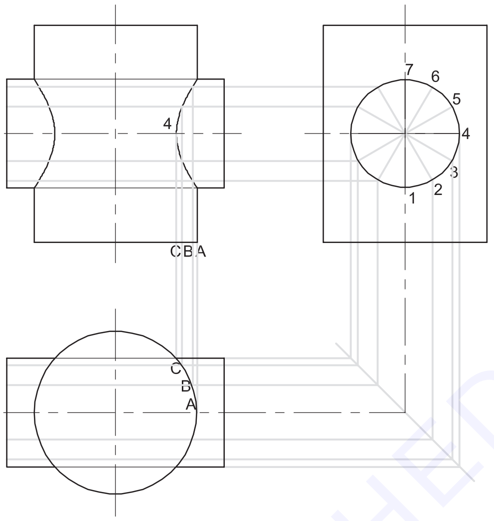
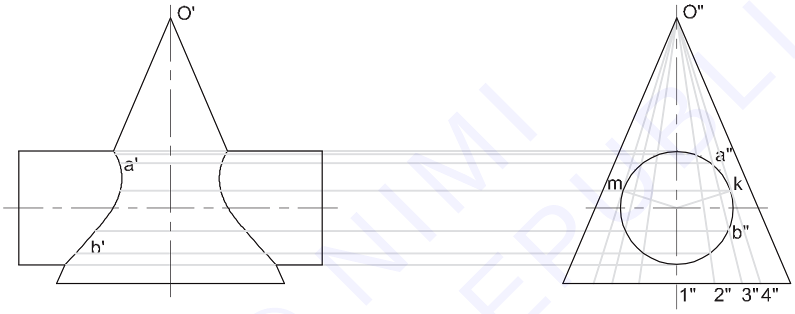


Fig 13



Procedure

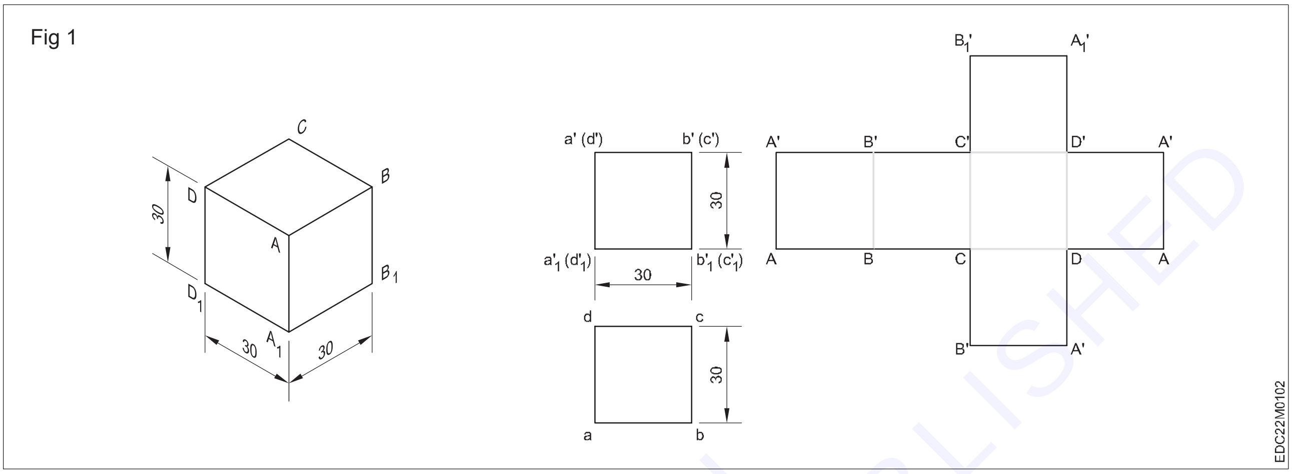
Development of surfaces of solids

Exercise 7.1

Draw the development of the surfaces of a cube of side 30 mm. (Fig 1)

- Draw the elevation and plan of the cube.

Fig 1



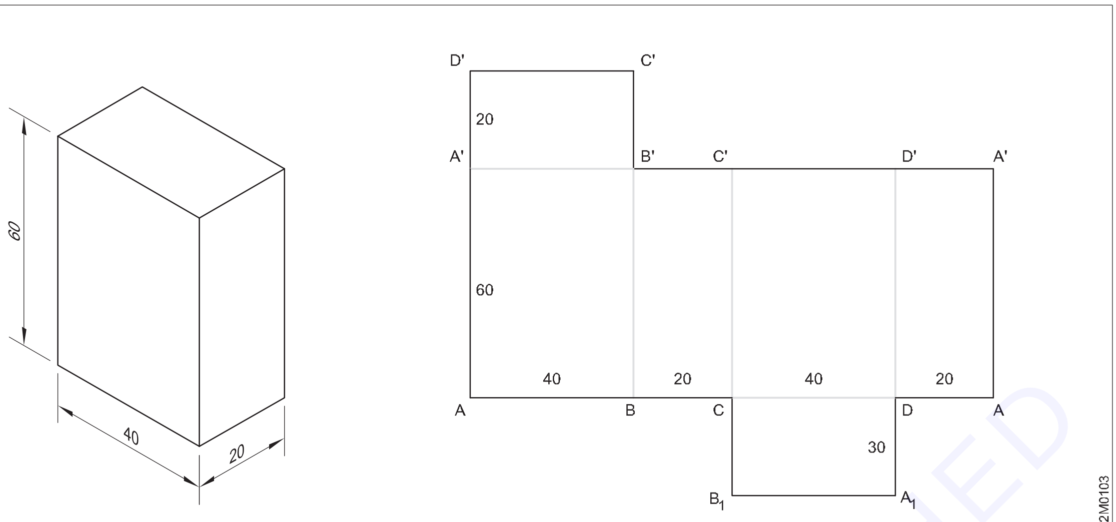
- From the front view stretch out (Project) the line AA & A'A' from points b'$_1$ b' horizontally.
- Set off AB, BC, CD & DA equal to 30 mm each. (side of square)
- From C & D draw perpendiculars and mark off/CB' & DA' equals to 30 mm.
- Similarly erect perpendiculars from points C' & D' such that C'B'$_1$ & D'A'$_1$ equals to 30 mm.
- Darken the lines as shown in Fig 1 to get the required development of the cube.

Exercise7.2

Draw the development of the surface/a rectangular prism of size 40 mm x 20 mm and height 60 mm by parallel line method. (Fig 2)

- Draw a rectangle of length equals to the perimeter of the prism. Length = 2 (l + b) = 120 mm and height equals to the height of prism (60 mm).
- Mark AB; BC; CD and DA equals to 40, 20, 40 & 20 mm respectively.
- Draw perpendiculars from the points A,B,C,D and mark as A', B', C', D'.
- Draw a rectangle A'B'C'D' on A'B' of size 40 x 20 mm.
- Draw another rectangle CD A$_1$B$_1$ on CD of size to 40 x 20 mm, the end faces of the prism. Figure obtained is the development of the total prism.

Fig 2



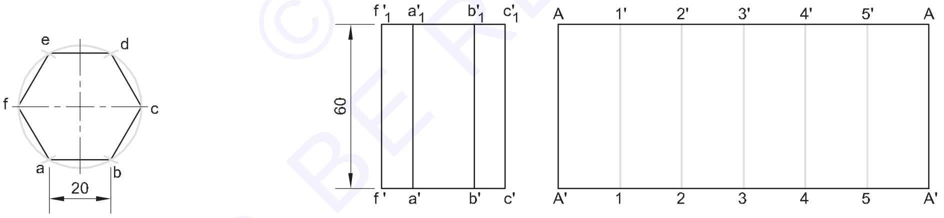
DEVELOPMENT OF A RECTANGULAR PRISM

Exercise 7.3

Draw the development of the lateral surface of a open hexagonal prism of side 20 mm and height 60 mm. (Fig 3)

- Draw the elevation and plan of the hexagonal prism.
- Project A'A' & AA from the elevation C'C' $_1$ shall be equal to 6 x 20 = 120 mm.
- On A'A' set of A'1, 1-2, 2-3 etc. equal to the side of the hexagon.
- Through A', 1,2,3 etc, draw vertical lines meet at AA, 1',2',3' etc to complete the required lateral surface development of the hexagonal prism.

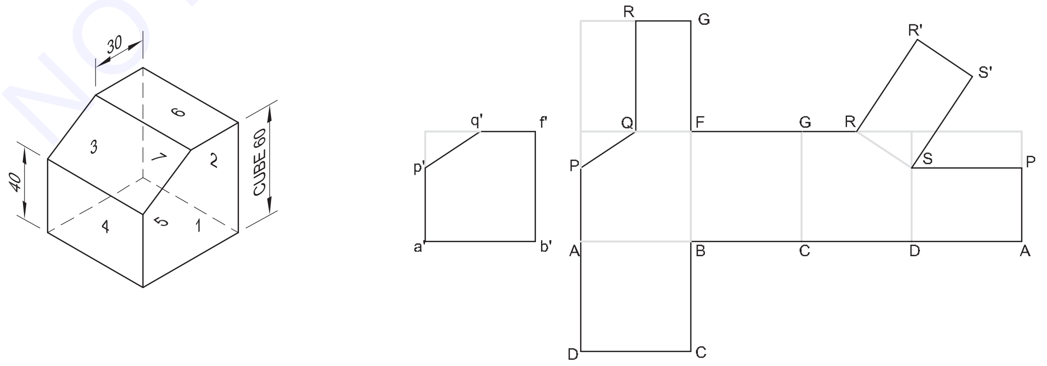
Fig 3



Exercise 7.4

Draw the development of surfaces of the box. (Fig 4)

Fig 4

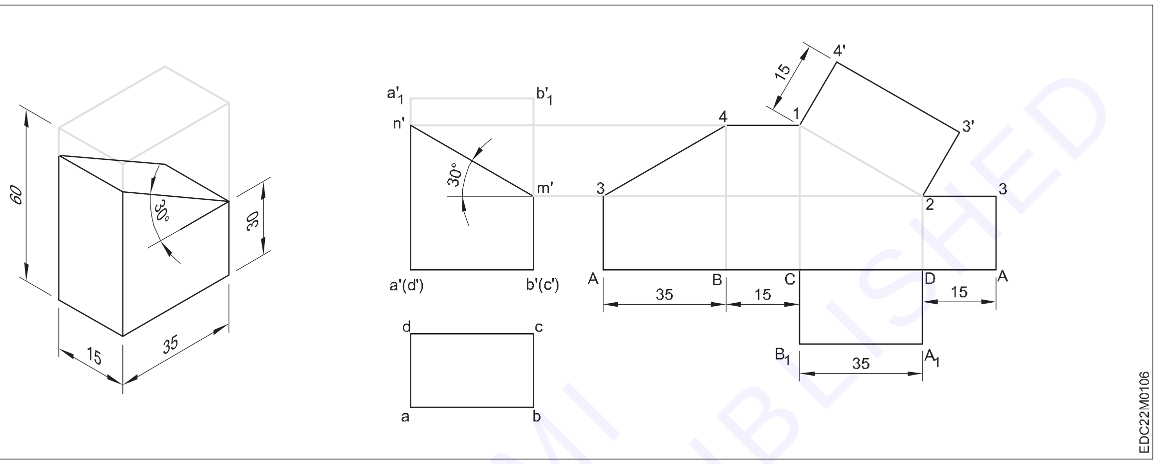


- Assume there is no slope surface and draw the development of cube of side 60 mm as given in Ex.No.14.1..
- Draw lines PQ and RS in the surfaces 1 and 3 in the basic development equivalent the shape shown in the isometric view of the box.
- Similarly join a rectangle of size equal to the surface 7 shown in isometric view with line RS or PQ and complete the development after erasing the construction lines.

Exercise 7.5

Draw the development of a rectangular prism as stated under and as shown in Figure. Prism is cut by a cutting plane at 30°, 30 mm from base on width side. (Fig 5)

Fig 5

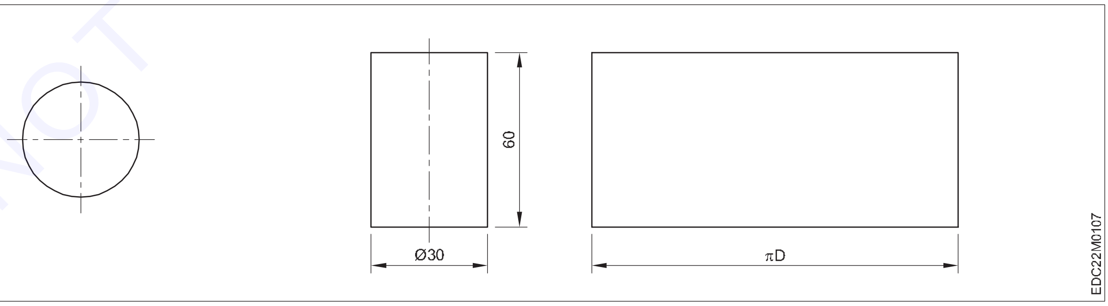


- Draw the top view and front view of the rectangular prism.
- Mark 30 mm on front view from b' (c') to meet the line b'b'$_1$ at m'
- At m' make an angle of 30° such that the cutting plane meet the line a'-a'$_1$. at n'.
- Project b', m', n' horizontally and mark of AB = 35, BC = 15, CD = 35 & DA = 15 mm.
- From A,B,C,D,A of the front view, draw vertical lines and mark points 3, 4,1,2 & 3.
- Draw 1-4' and 2-3' perpendiculars to 1-2 equals to 15 mm and draw CB$_1$ & DA$_1$ perpendicular to CD and equals to 15 mm.
- Darken the lines throughout the development of the given prism.

Exercise 7.6

Draw the development of the lateral surface of an open cylindrical drum of dia 30 mm and height is 60 mm. (Fig 6)

Fig 6



- Draw the plan and elevation of the drum as stated.
- Project the front view and draw the side view to a length of circumference of the base of the cylinder.

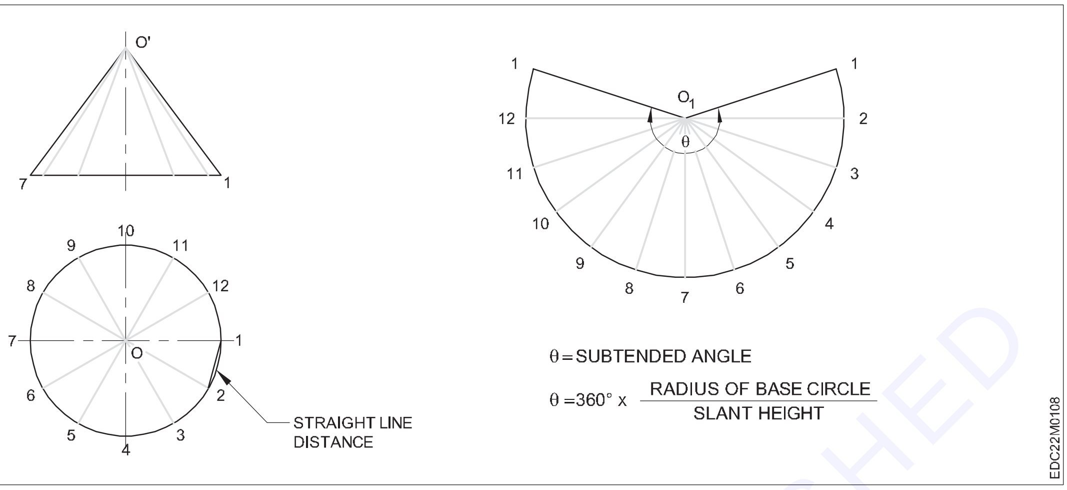
Circumference = π D = π x 30 mm = 94.26 or 94.3

Rectangle thus formed is the development of cylinder.

Exercise 7.7

Draw the development of a cone of base 60 mm and height 40 mm. (Fig 7)

Fig 7



- Draw the plan and elevation of the cone.
- Divide the plan circle into number of equal parts (say 12 parts) and mark.

Development is more accurate if the number of equal parts are more.

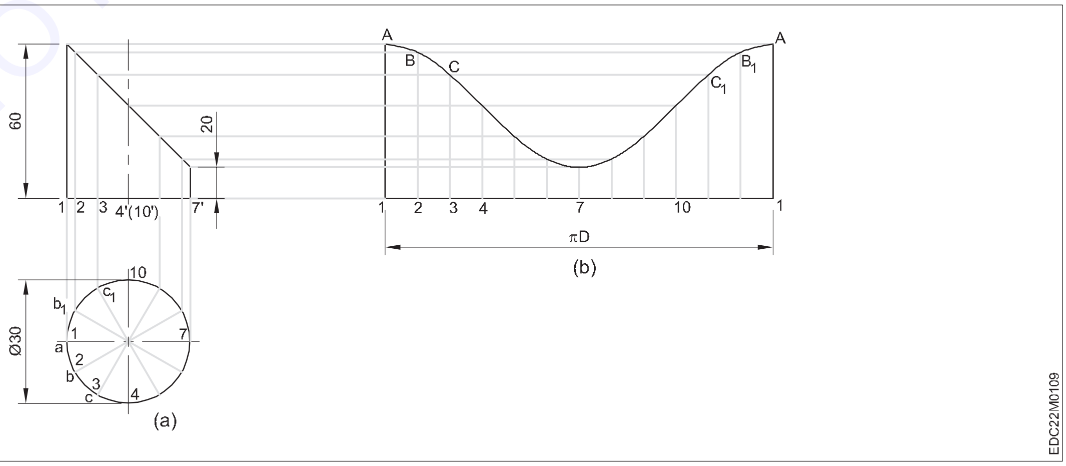
- Project these parts to the base of the cone.
- Join these points to the vertex of the cone.
- Locate O_1 vertex for the development of cone.
- Draw an arc from the vertex as centre and length of slope as radius.
- Set divider on the straight distance between two consecutive points on the plan.
- Transfer the distance along the arc as many times according to the divisions on the plan.
- Join points 1 O_1 1and complete the development.

This method is only an approximate method. The straight line distance taken is slightly smaller than the arc length. More accurate method is by calculating actual circumference and dividing it into number of segments.

Exercise 7.8

Draw the lateral surface of a truncated cylinder. (Fig 8)

Fig 8

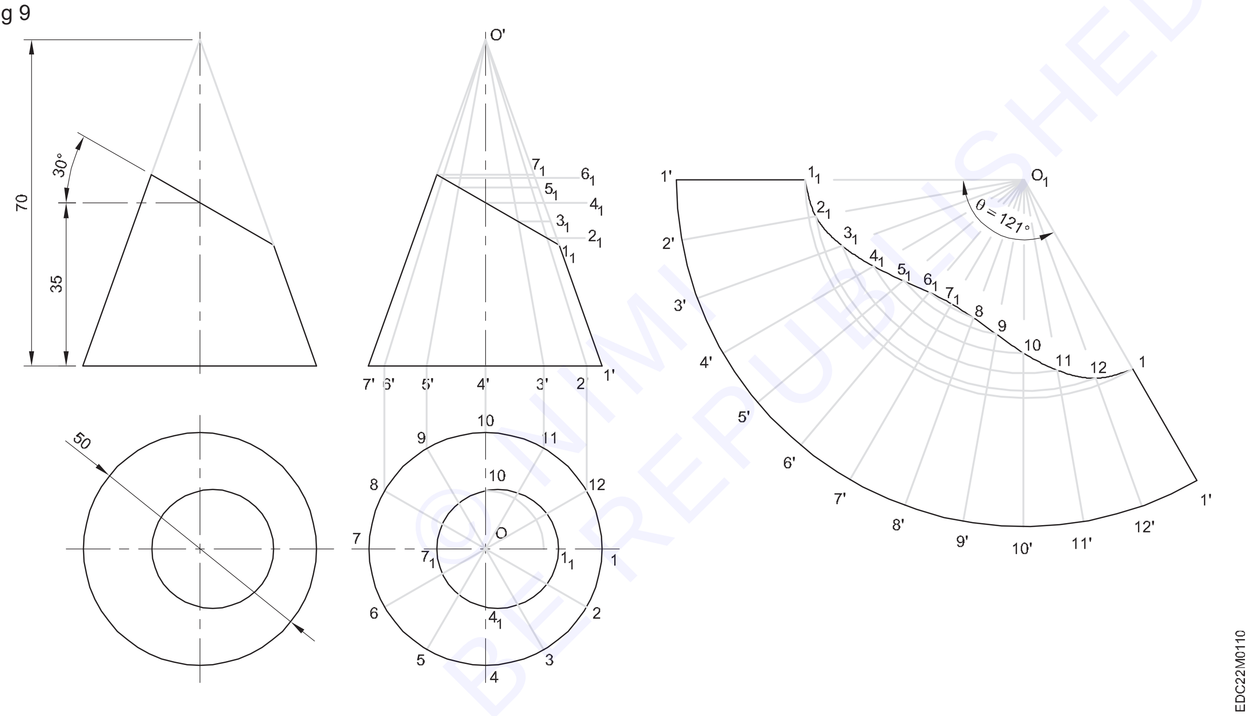


- Draw the plan and front view of the truncated cylinder.
- Divide the circle into any number of equal parts.
- Project these points to the front view to form the elements of surface of the cylinder.
- Assuming the cylinder as a whole draw the full development of the cylinder.
- Draw the length of the development also divide into 12 equal parts as in the plan.
- Project the true height of the each elements to cut the development of the cylinder.
- Join all the points by a smooth curve to obtain the required development.

Exercise 7.9

Draw the development of the lateral surface of the cone as positioned. (Fig 9)

Fig 9



- Draw the views of the cone and cutting plane on it.
- Divide the base into any number of equal parts, say 12 and mark them.
- Project the points on to FV and join to the apex.
- Mark the points on the cutting plane and draw lines parallel to base, touching the generator at $1_1, 2_1 .... 7_1$. These are the true lengths from 0'.
- Draw the sector with slope as radius.

$$\text{Sector angle } \theta = \frac{\text{Radius of cone}}{\text{Slope length (S)}} \text{x}360°$$

$$\frac{25}{74.33} \text{X}360° = 121°$$

- Mark O_1 and set angle θ.
- Transfer the true lengths on the cone on to the sectors and join by smooth curve.

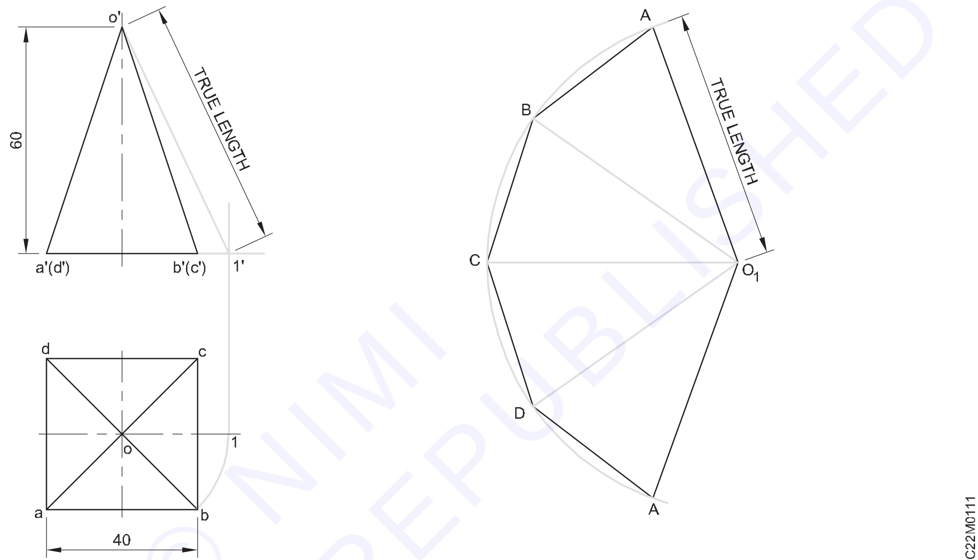
The area between this curve and the arc is the development of the lateral surface.

Exercise 7.10

Draw the development of the lateral surface of a square pyramid of base 40 mm and vertical height 60 mm. (Fig 10)

- Draw the plan and elevation of the pyramid.
- With O as centre in top view and ob as radius, draw an arc to meet the axis at 1.
- Project the point 1 to front view to meet the base line at 1'.
- Join O'1' which is the true slant length of the pyramid.
- With `O$_1$ 'as centre and true length as radius draw an arc and set off the sides of the pyramid on the arc at A,B,C,D & A.
- Join O$_1$A, B, C, D, AO$_1$ which is the required development.

Fig 10

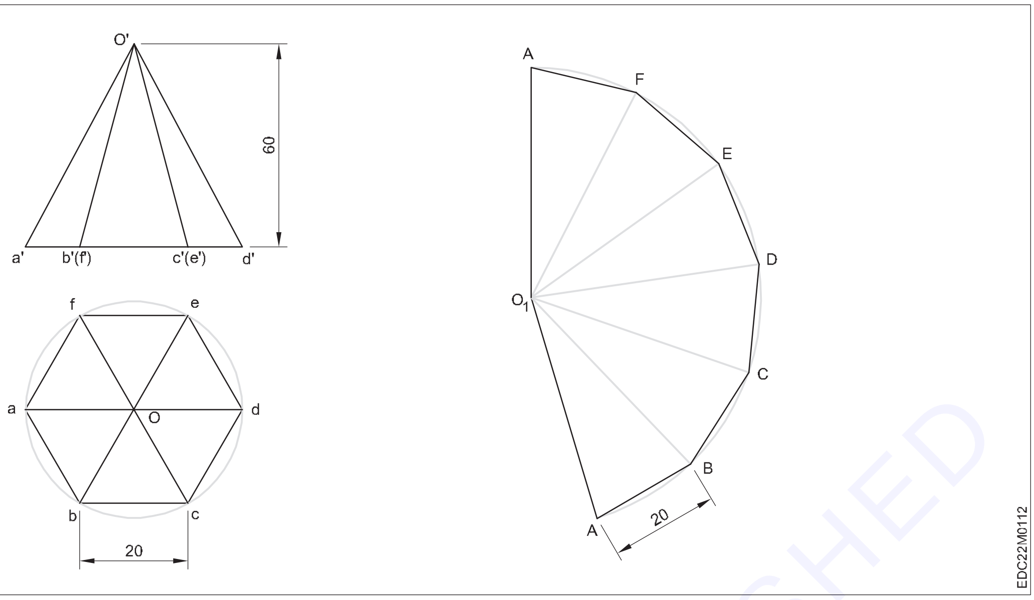


Exercise 7.11

Draw the development of the lateral surface of a hexagonal pyramid of side 20 mm and height 60 mm. (Fig 11)

- Draw the top view and front elevation of the hexagonal pyramid.
- With `O$_1$ ' as centre and slant length as radius draw an arc.
- Set off six equal sides (40 mm) on the arc at A,B,C,D,E,F & A.
- Join O$_1$, A,B,C,D,E,F and A O$_1$. Complete the lateral surface development of the pyramid.

Fig 11

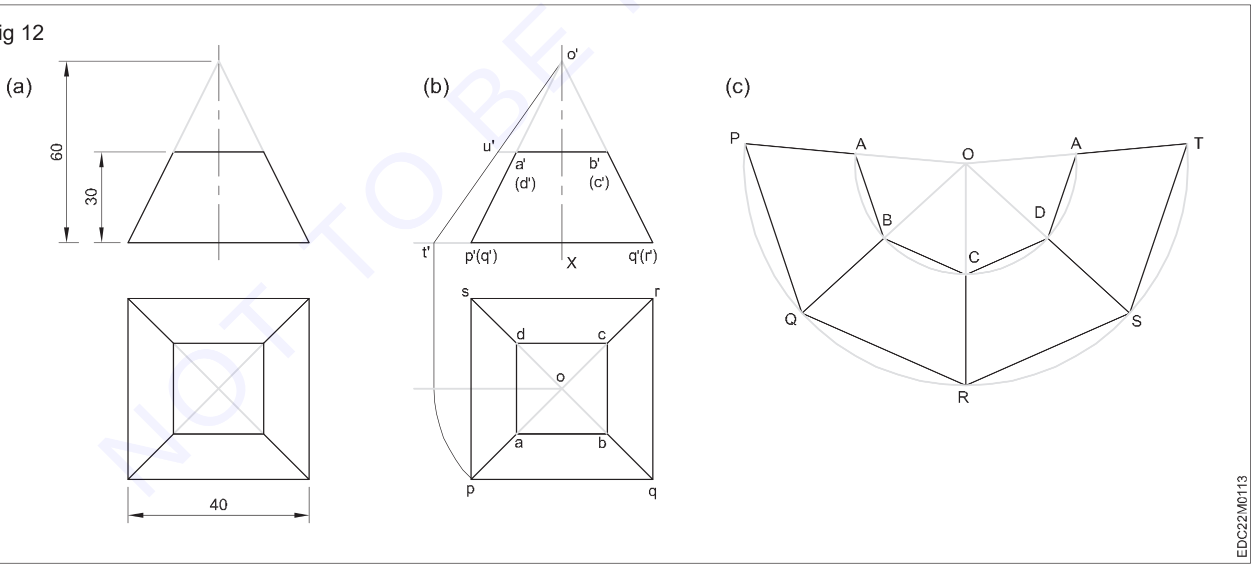


Exercise 7.12

Draw the development of the lateral surface of the frustum of a square pyramid as positioned. (Fig 12a)

- Draw the front view and plan of the pyramid. (Fig 12b) None of lines in these views show the true length of slant edge.
- Transfer OP in plan to xt in front view. Join o't' by straight line. o't' is the true length of slant edge.
- With `O' as centre draw the development of whole pyramid. (Follow the steps of the exercise 14.10)
- With `O' as centre set off OA, OB, OC, OD & OA equal to O'U' in front view. (Fig 12c)
- Join AB, BC, CD, DA. Surface bounded by ABCDATSRQP is the development.

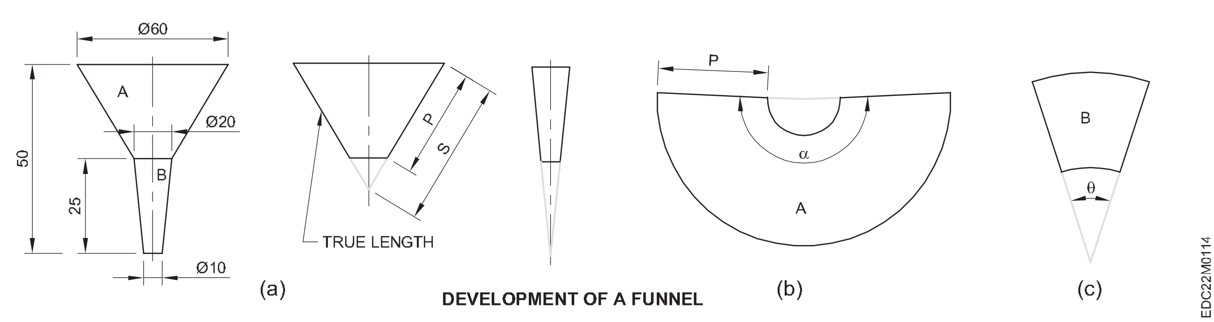
Fig 12



Exercise 7.13

Draw the development of the lateral surface of a funnel as positioned. (Fig 13)

- The parts (A & B)of the funnel are in the shapes of frustum of cones.
- Development of cone and frustum of cone are already dealt in earlier exercises. Follow the previous procedures and draw the developments of both parts A and B.



DEVELOPMENT OF A FUNNEL

Formula for finding angle

$$\frac{\text{Radius of base cone circle}}{\text{Slant height}} \text{x} 360^\circ$$

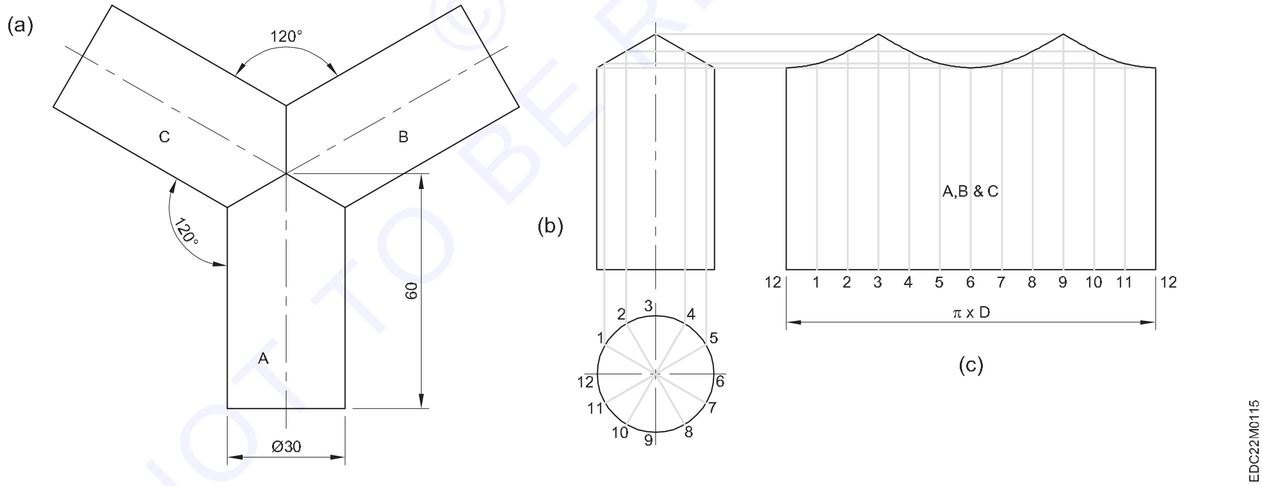
Exercise 7.14

Draw the development of intersecting cylinders of dia 30 mm at 120°. (Fig 14a)

All the cylindrical pipes are of same diameter and intersecting each at equal angles. Hence in this case the development of all the pipes are same and so the development of one pipe will represent other pipes.

- Draw the plan and elevation of the pipe `A' and mark the division on the plan. (Fig 14b)
- Draw the vertical projectors from the plan to front view to meet the line of intersection.
- Draw horizontal projectors from these points on to the development.
- Mark the intersecting points and join with a smooth curve to complete the required development. (Fig 14c)

Fig 14



Exercise 7.15

Two square prism of A & B positioned as stated below intersects each other. Draw the plan, elevations and show the line of intersection. Prism `A' is resting on its square base and its face are at 45° to XY line. (HP) Prism `B' intersects at 10 mm from base at 45°. The dimensions of prisms are:

- Side 40 mm height 60 mm (Prism A)
- Side 35 mm height 80 mm (Prism B)

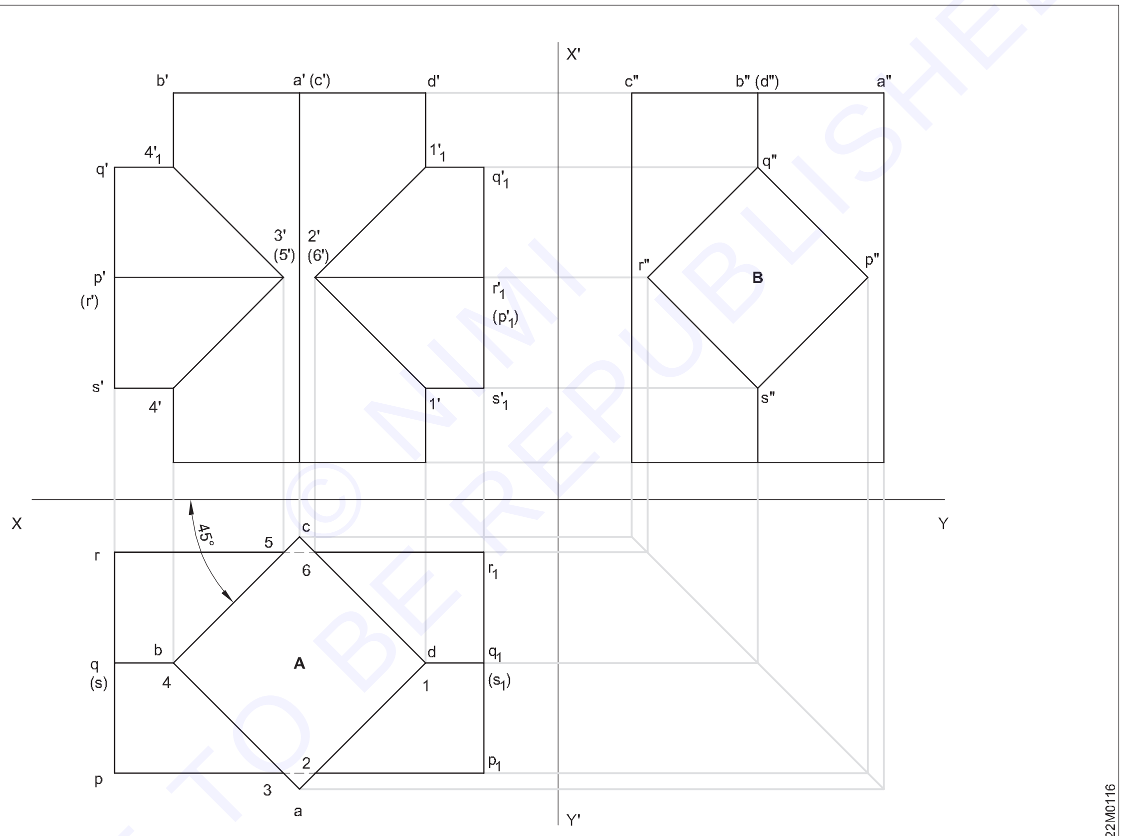
Draw the top view of 40 mm square prism, with its two sides at 45° to (VP) X'Y' line.

- Mark the corners as a,b,c & d.

- Draw the front view and the end view for the prisms A & B with space in between and mark the corners as shown.
- Draw a square of side 35 mm, faces at 45° to the (HP) XY on the end view.
- Mark the corners as p", q", r" & s" as shown.
- Draw its projection to the top view and mark the points where the lines meet the square a,b,c & d as 1,2,3,4,5 & 6.
- Draw the vertical projectors from points 1,2,3 & 4 on to the front view.
- Draw horizontal projectors from corners p",q",r" & s" of EV on to the FV intersecting projection line from TV.
- Mark the intersecting points 1', 2', 3', 4', $4_1'$ & $1_1'$.
- Join the points 1'2', 2'$1_1'$, 1'4', 3'$4_1'$.

The lines 1'-2', 2'-$1_1'$, 4'-3', 3'-$4_1'$ are the lines of intersection. (Fig 15)

Fig 15



PRISMS INTERSECTING AT RIGHT ANGLES

Exercise 7.16

The cylinders A & B are positioned as shown, (Fig 16) intersects each other at right angles. Draw the plan, elevation, side view and show the curve of intersection.

- Draw the three views of cylinder `A' as positioned shown. (Fig 17)
- Draw the side view of the cylinder `B', as circle of diameter equals to 40 mm, such that its centre is the midpoint of the axis of cylinder `A'.
- Divide the circle into 12 equal parts and number them.
- Draw the front view and top view of cylinder `B' projecting from the side view of the horizontal cylinder.

Note: Consider that a number of horizontal section planes passing through the generators of the horizontal cylinder to cut both cylinders. For all the horizontal sectional planes, the sectional top view will always be a circle of `n' diameter. Also the sectional top view of the vertical cylinder will a circle of `m' diameter.

- Draw series of cutting planes on the end view of cylinder `B'.
- Draw horizontal projectors on to the vertical cylinder (A).
- The line $1 - 1_1$ in top view intersects the circle at p_1 & q_1.
- Draw vertical projectors from these intersection points P_1 & q_1 to intersect the line 1'1', in front view at p'_1 and q_1' respectively.
- Considering the second horizontal plane $2\ 2_1$ in top view, obtain the corresponding point in the front view P'_2 (p'_{12}, q'_{12}) and q'_2.
- Repeat the same procedure for other series of section planes passing through generators, obtain intersection points in front view.

Join the obtained intersection points in the front view by a smooth curve, which is the curve of intersection of cylinders A & B as stated.

Fig 16

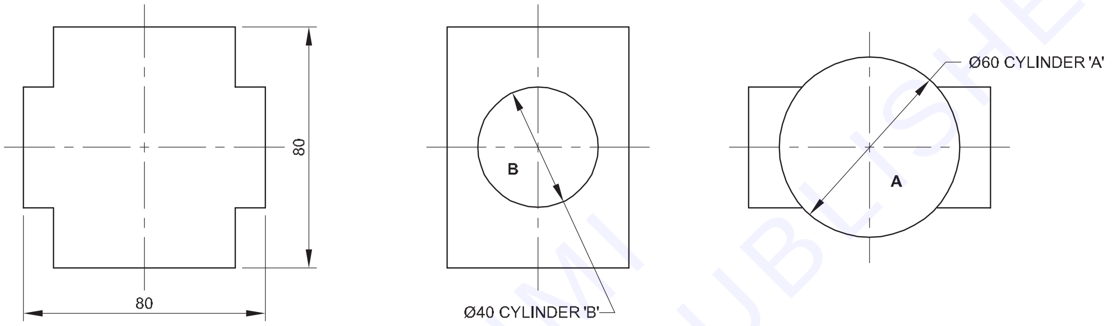
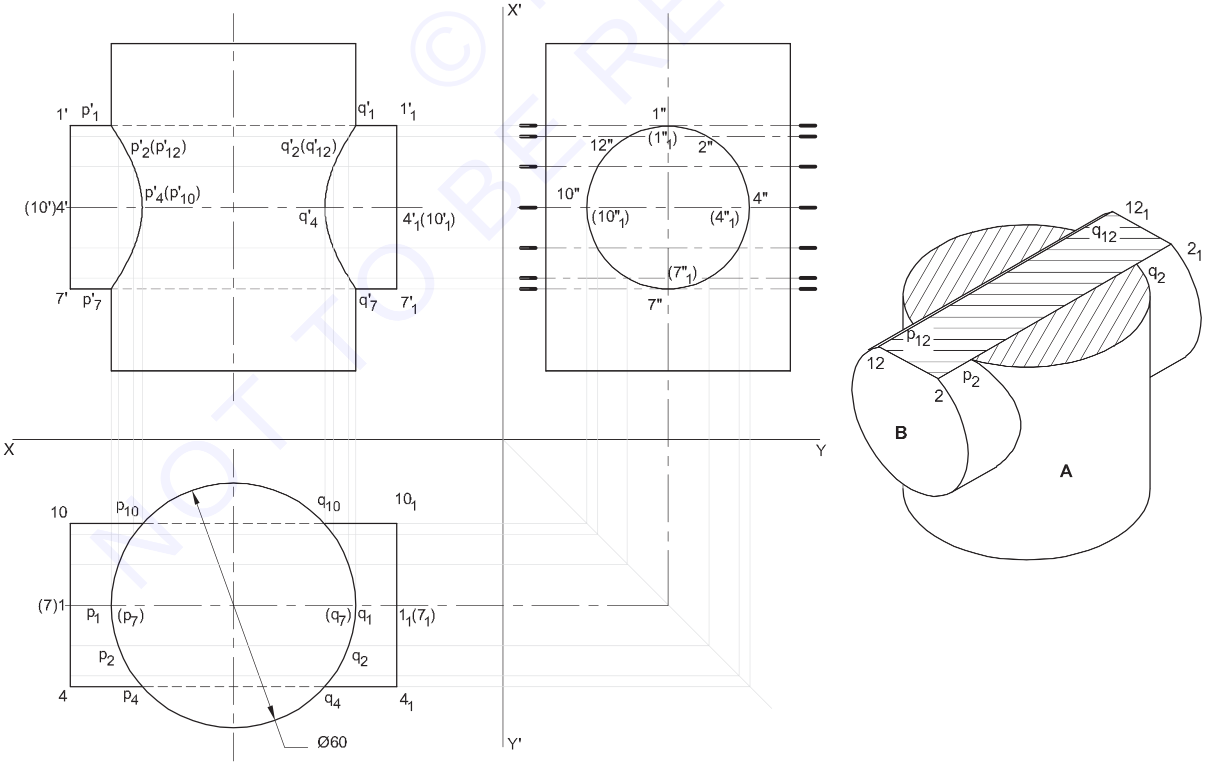


Fig 17

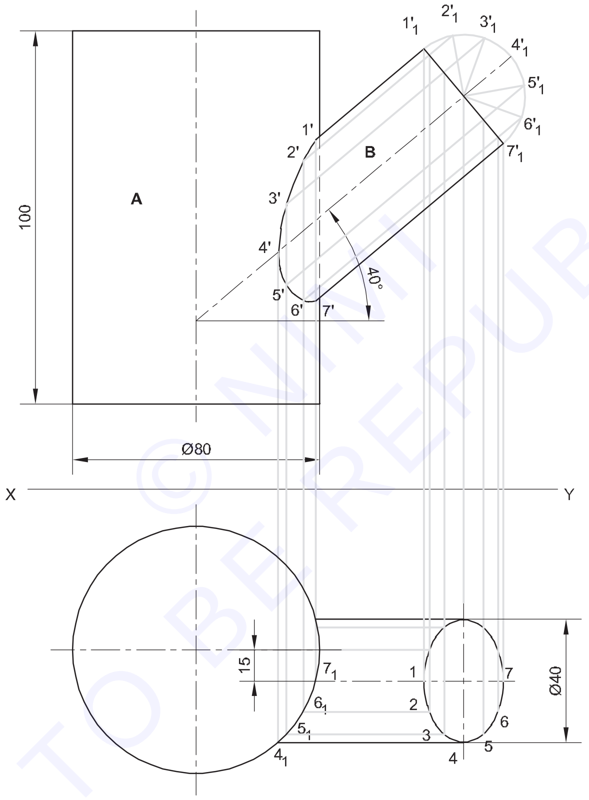


Exercise 7.17

A thin cylinder of dia 80 mm resting vertically is joined to a 40° branch pipe of dia 40 mm as shown in Fig 18. Draw the intersection curve of the cylinders. The axes of both the cylinders are 15 mm apart.

- Draw outlines of the pipes as shown in Fig 18.
- Divide branch pipe surface into a number of equal parts by drawing semi-circle and mark as 1'$_1$,2'$_1$,3'$_1$,....6'$_1$ as shown in Fig 18.
- Mark the corresponding points on the top view of the branch pipe projected from the front view as 1,2,3....7
- Draw horizontal projectors from these points to meet the dia 80 mm circle and number them in the top view.
- Draw vertical projectors from these points on to the front view intersecting the projection lines from the semi-circle.
- Mark the intersecting points and join with smooth curve.

Fig 18



CYLINDERS AXES OFFSET INTERSECTING AT ANGLE

Exercise 7.18

The axis of two cylinders A & B intersect on the centre line at 45° are positioned as shown in Fig 19. Draw the plan, elevation and the intersection curve.

- Draw the given front view of the cylinders as stated in position. (Fig 20)
- Draw the plan for the given elevation.
- Draw the semi-circle of the branch cylinder, divide it into 6 equal parts and number them.
- Draw the projectors from these points parallel to the inclined axis of cylinder `B'.
- Mark the corresponding points on the top view of the cylinder `A' by drawing horizontal projectors from the top view of cylinder `B'.
- Draw vertical projectors from these point to meet the inclined projectors drawn from the semi-circle of cylinder `B'.

- Join the intersecting points which is the required curve of intersection. (Fig 20)

Note: In this example the curve projected as straight lines.

Fig 19

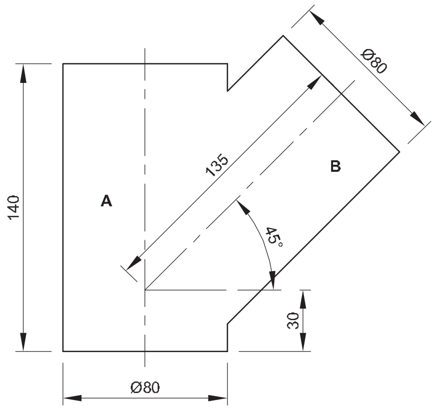
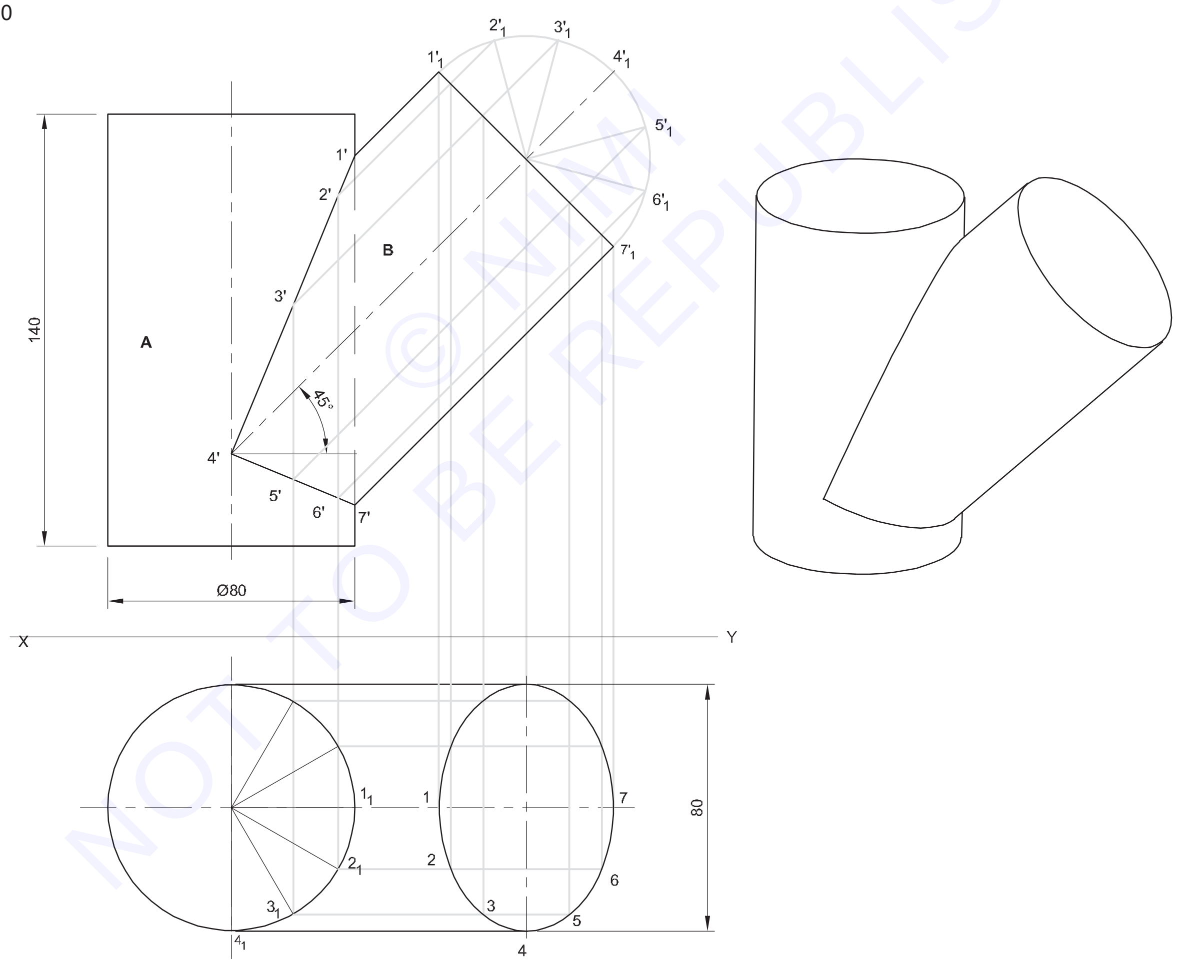


Fig 20



CYLINDERS AXES INTERSECTING AT ANGLE

EXERCISE 8 : Fasteners

Screw threads

Screw thread is a helical groove formed on cylindrical surfaces either external or internal. In other words **Screw thread** is a ridge of uniform section in the form of helix either on a cylinder or inside a cylindrical surface. It is most important single feature in engineering. The application of screw threads to bolts,nuts, screws, studs etc, enable us to join two or more components together easily and also can be dismantled without damaging any components. The first screw thread which was standardised is “Whitworth” Screw Thread (BSW) was designed by Sir Joseph Whit Worth in 1841 in England. In 1864, William Sellers designed Sellers Thread.

External thread: It is the thread formed on the outside of a shaft e.g., bolt, screw and stud.

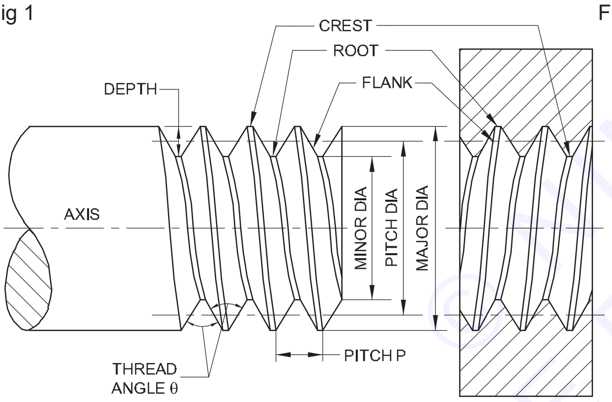
Internal thread: It is the thread formed inside a component say a hole or Nut.

Elements of screw thread: Fig 1 shows elements of a ‘V’ thread.

Right hand thread: is a thread that advances into the work when turned clockwise. It is the commonly used thread unless and otherwise stated. (Fig 2a)

Left-hand thread: It is the thread that advances into engagement when turned anti clockwise. (Fig 2b)

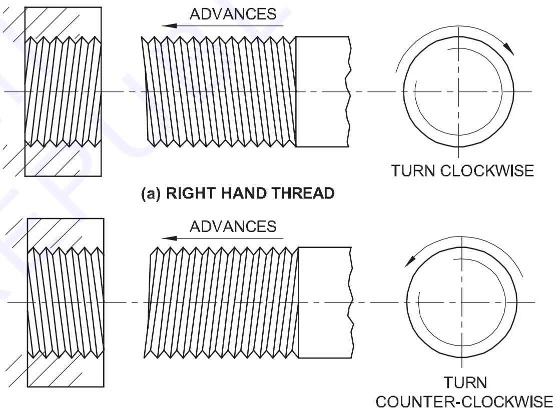
Fig 1



(a) EXTERNAL THREAD (b) INTERNAL THREAD

SCREW - THREAD NOMENCLATURE

Fig 2



(b) LEFT HAND THREAD

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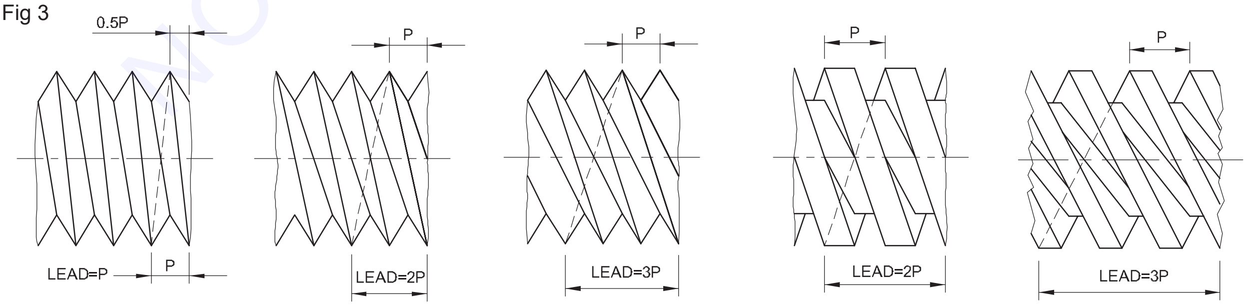
Single start thread: it is a thread form, cut on the cylinder. Unless otherwise stated, threads are single start thread, also called single start. (Fig 3a)

Multiple or multistart threads: a thread combination of same profile formed by two or more helices on the cylinder. For single revolution, the axial movement of nut or screw is double, triple or quadruple depending on the start of the thread. (Figs 3b, c,d & e)

Fig 3



a) SINGLE START b) DOUBLE START c) TRIPLE START d) SQUARE - DOUBLE START e) ACME TRIPLE START

MULTIPLE THREADS

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Vee threads

Types of `V' threads: The most commonly used thread before introducing metric thread is **British Standard Whit Worth thread (BSW),** it is adopted by U.K. Fig 1 shows the proportions of the thread profile of BSW. Its thread angle is 55°.

Pitch of a thread in inch system. P in inches = 1 divided by no.of threads in one inch.

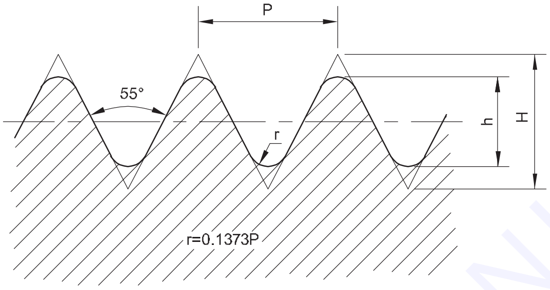
Example: Designation 1/2" BSW i.e 1/2" nominal diameter, BSW thread, 12 TPI.

British Standard Fine (BSF) threads: Profile of the BSF thread is same as of BSW (55^o) but the pitches are finer comparing to pitches of same dia BSW threads. BSF threads are used in Automobile, Air crafts and for fine mechanisms. Since the pitch is small, core dia or the effective dia is larger.

British Association thread (BA thread): Its angle is $47\frac{1^\circ}{2}$ .Designation example 1"BA - nominal dia 1" BA threads. This thread is used, generally for small instruments. All the British Standard threads are of Inch system. (Fig 2)

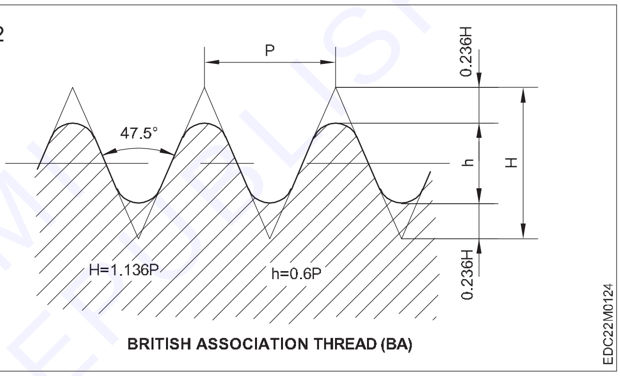
Metric threads: IS:4218 Bureau of Indian Standards (BIS) has recommended the use of ISO metric threads. Its thread angle is 60^o . Fig 3a & b shows the metric thread form with proportions.

Fig 1



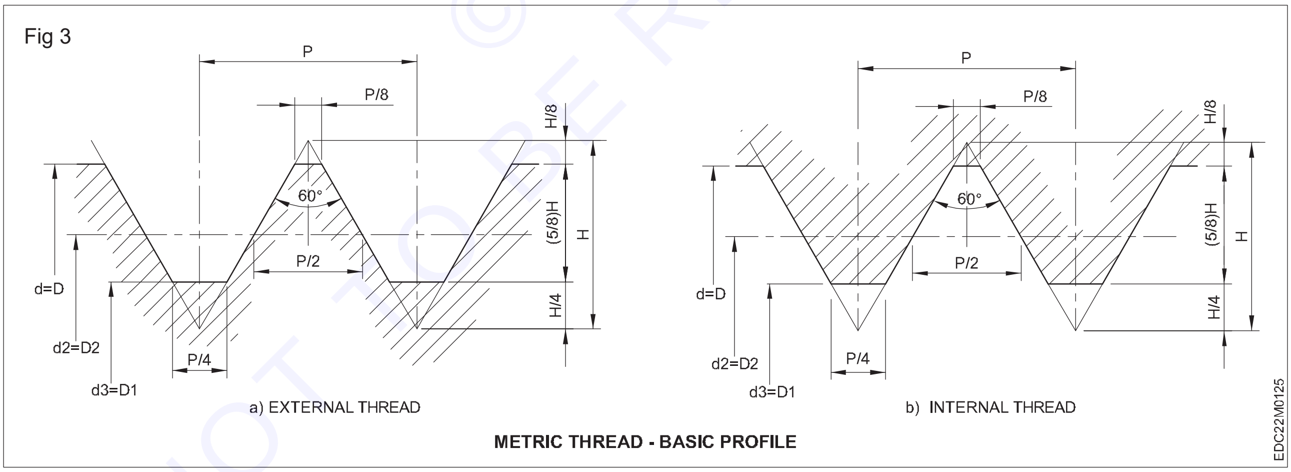
BRITISH STANDARD WHIT WORTH THREAD (BSW)

Fig 2



BRITISH ASSOCIATION THREAD (BA)

Fig 3



a) EXTERNAL THREAD

b) INTERNAL THREAD

METRIC THREAD - BASIC PROFILE

Different types of threads

Square threads (Fig 1)**:** In this thread the flanks are perpendicular to the axis of the thread.

Square threads are used for transmitting motion or power. Eg. Screw jack, vice handles, cross-slide and compound slide, activating screwed shafts.

Designation

A square thread of nominal dia. 60mm and pitch 9 mm shall be designated as Sq.60 x 9 IS: 4694 -1968.

Trapezoidal threads: These threads have a profile which is neither square nor 'V' thread form and have a form of trapezoid. They are used to transmit motion or power. The different forms of trapezoidal threads are:

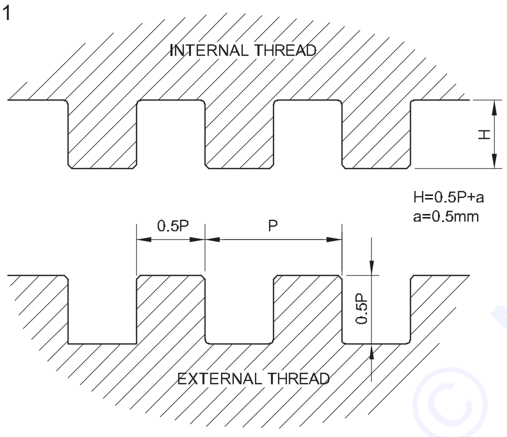
– acme thread

– metric acme thread

– buttress thread etc.

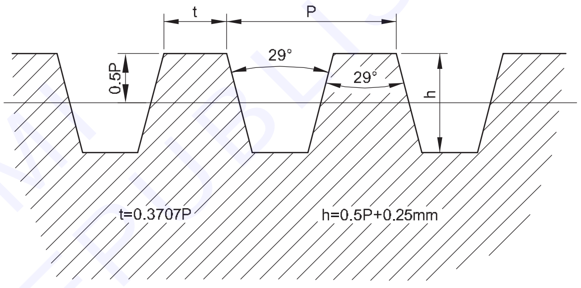
Acme thread (Fig 2)**:** This thread is a modification of the square thread. It has an included angle of 29°. It is preferred for many jobs because it is fairly easy to machine.Acme threads are used in lathe lead screws. This form of thread enables the easy engagement of the half nut. The metric acme thread has an included angle of 30°.

Fig 1



SQUARE THREAD

Fig 2



ACME THREAD

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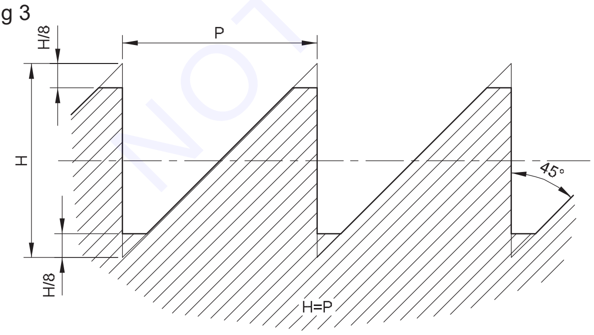
Buttress thread: In Buttress thread one flank is perpendicular to the axis of the thread and the other flank is at 45°. These threads are used on the parts where pressure acts at one flank of the thread during transmission. Fig 3 shows the various elements of a buttress thread. These threads are used in power press, carpentry vices, ratchets, etc.

Knuckle (round) thread (Fig 4)**:** it has a semi circular form, different from Vee and trapezoidal thread. It is not sensitive against damage as it is rounded. It is used for valve spindles, railway carriage couplings, hose connections, etc.

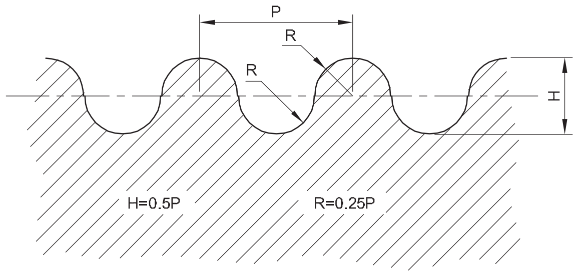
Fig 3



ONLY 45°

BUTTRESS THREAD

Fig 4



KNUCKLE THREAD

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Convention of threads: Since drawing the profile of threads is cumbersome and does not serve any special purpose, the thread forms are conventionally represented by thin line. Fig 5 shows the convention of threads on the screw and end view.

Designation of screw threads: eg Metric threads M20- meaning nominal diameter 20 mm coarse pitch (equal to 2.5 mm Ref: IS: 4218-part II) or M20 x 1.5 means nominal diameter 20 mm pitch 1.5 mm.

M 20 x 1.5 “Triple start” where pitch is 1.5 mm and lead is 4.5 mm.

Threads are designated by their nominal major diameter and type of thread. Apart from the nominal dia and type of thread, pitch; L.H (left hand) if it is not right hand thread and class of thread are indicated.

Eg. M24 - Metric thread nominal dia 24 m (Coarse pitch 3 mm)

M24 x 1.25 fine pitch 1.25 mm.

M24 x 2 5H Tolerance grade of internal thread.

M24 x 2 6g Tolerance grade of external thread.

M 24 x 2-7H8g fit of the nut and bolt.

For multi-start threads, Double Start, Triple start etc are mentioned.

Eg. M45 x Triple Start

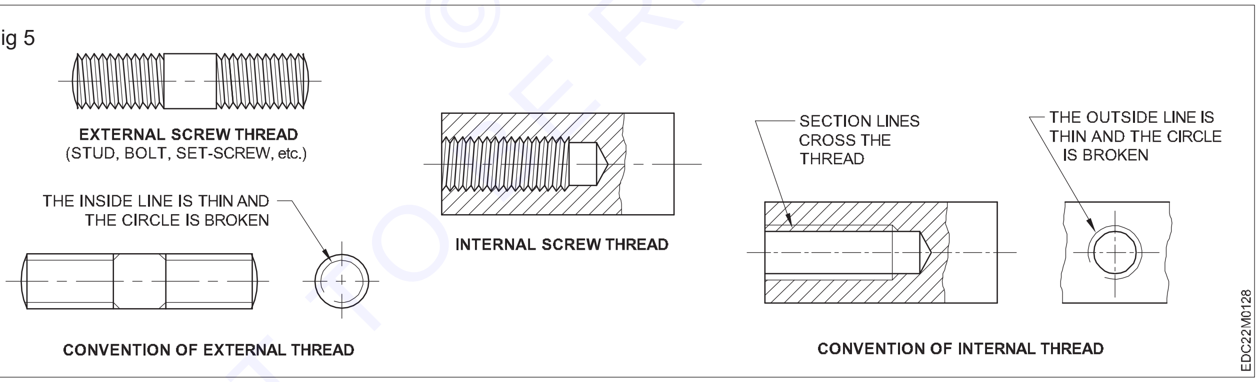
BSW Thread: $1\frac{1}{2}$ BSW - nominal diameter $1\frac{1}{2}$ TPI 6 (Threads per inch)

BSF Threads: $1\frac{1}{2}$ BSF

Example

- SQ 30 x 6 IS:4964 A square thread of nominal dia 30 mm and pitch 6 mm.
- Tr40 x 7 A single start ISO metric trapezoidal screw thread of nominal dia 40 mm and pitch 7 mm.
- Tr 40 x 14 (P7) LH
  A multiple start left hand (double start) ISO metric trapezoidal screw thread of nominal dia 40 mm, lead 14 mm and pitch 7 mm.

Fig 5



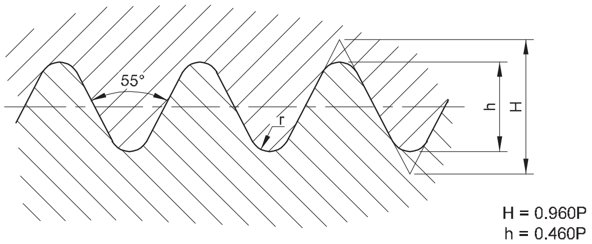
Pipe threads

Pipe threads are used for fastening purposes intended for mechanical assembly of components of pipe fittings, cocks, valves etc, where pressure type joints are not required on the threads. IS:2643 (Part 1 to 3) lays down the standard pipe sizes, pitch thread profile and tolerance etc.

The (nominal) sizes are expressed as 1/8, 1/4 .....1 ... $1\frac{1}{2}$ ...6 available in 24 sizes.

Fig 1 shows the basic profile of thread and proportions.

Fig 1



H = 0.960P
h = 0.460P
r = 0.137P

PIPE THREAD FOR FABRICATION PURPOSES

Thread angle = 55°

Theoratical depth H = 0.96049p

Actual depth h = 0.640327p = $\frac{2}{3} \times H$

Radius of crest and root r = 0.137329p

Crest and root are rounded to H/6

The pitch value varies from 0.907 to 2.309 mm.

According to the tolerances provided, **external threads** are classified as **Class A** and **Class B.**

Class A applies to screw threads requiring snug fit, where accuracy of thread form is essential.

Class B threads applies to threads of ordinary commericial quality. The choice of Class A or B depends upon the conditions of application. The tolerance values of Class B threads are double that of class A threads. Only one class of tolerance is specified for internal threads.

Designation of threads

Examples - External thread of size 3 with Class A tolerance shall be designated as

External pipe threads G3A- IS:2643.

Internal threads of size 3 shall be designated as Internal Pipe theads G3 - IS:2643.

Screwed fasteners

In joining number of parts together and dismandling without damaging any parts, devices called Bolts, nuts, screws etc are made use of. These are called **"Screwed fasteners".** Bolt is a metallic cylindrical rod having a specific shape on one end called **"Head**" and the other end called the **shank** with screw threads cut on it. All the fasteners are generally made of steel of good tensile strength.

Bolts are known by the shape of head viz., Hexagonal, Square, Cylindrical or cheese headed, cup or round, 'T' hook, eye bolt etc. and shank dia. The shape of head is selected depending upon the purpose for which it is used. While engaging or dismantling a nut on to bolt, to prevent the rotation of bolt, bolt head is held by another spanner.

All the fasteners size/specifications follow letter M, stands for Metric (size) e.g. Hex.bolt M 20 x 100 i.e hexagonal bolt shank dia 20 mm, 100 mm long.

Hexagonal head bolts: For drawing purpose, irrespective of shape of head, bolt head thickness is taken as **0.8d** where **d** is the diameter of the shank. The length of bolt varies according to dia and the requirements. Figs 1 & 2 show the bolt head and bolt. The top corners of the hexagon are chamfered to avoid sharp corners which get damaged while using spanner and also injurious while handling.

Fig 1

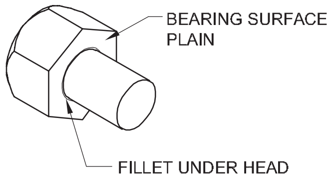
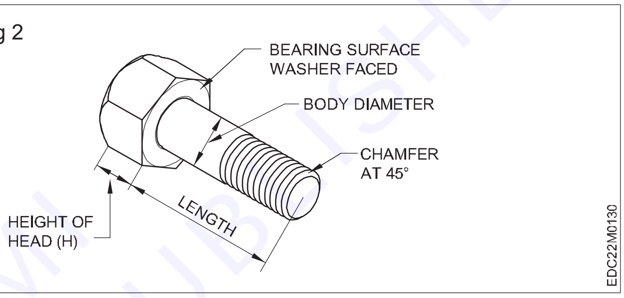


Fig 2



There are three grades of hex.head bolts viz (i) Precision, (ii) Semi precision and (iii) Black denoted by letters A, B & C respectively according to their dimensional accuracies.

Hexagonal bolts Grades A and B IS:1364 part-1 M3 to M36. (12 Sizes)

Grade C IS:1363-part-1 M5 to M36 (10 sizes)

Grade C Black IS:3138 M42 to M156 (23 Size)are available.

Hexagonal bolts Grades A and B IS:1364 part-1 M3 to M36. (12 Sizes)

Grade C IS:1363-part-1 M5 to M36 (10 sizes)

Grade C Black IS:3138 M42 to M156 (23 Size)are available.

Square head bolt: Square head bolt is the second commonly used bolt, when the bolt head is not accessible. The square head generally sits with in a square cavity, which prevents its rotation while the nut is handled. Fig 3 shows a square head bolt of plain shank and with square neck. They are of grade C (black) IS: 2585 Diameters available from 6 to 39 mm (15 sizes).

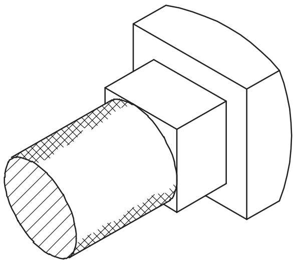
Cylindrical or Cheese head bolts: It is also called as round head bolts. As the name implies, its head is cylindrical. To prevent rotation while handling nut, the head is fitted with a small pin of 0.12 d projecting out by 0.2 d. The pin is fitted close to the head perpendicular to the axis or on to the head close to the shank parallel to the axis. The pin fits into corresponding recess in the adjacent part. There is no need to use spanner to hold the head. It is most commonly used on big ends of connecting rods, eccentrics etc. (Fig 4)

Cup head or round head bolt: It has hemispherical head with a snug or square neck. The snug is formed while forging. Its function is same as round head bolt. (Fig 5)

'T' head bolt: Its head is in the shape of `T' which is accommodated in 'T' slots of machine tool tables. The neck is usually of square cross section. Apart from their use in machine tools, they are used in stuffing box and gland in boiler mountings etc. (Fig 6a)

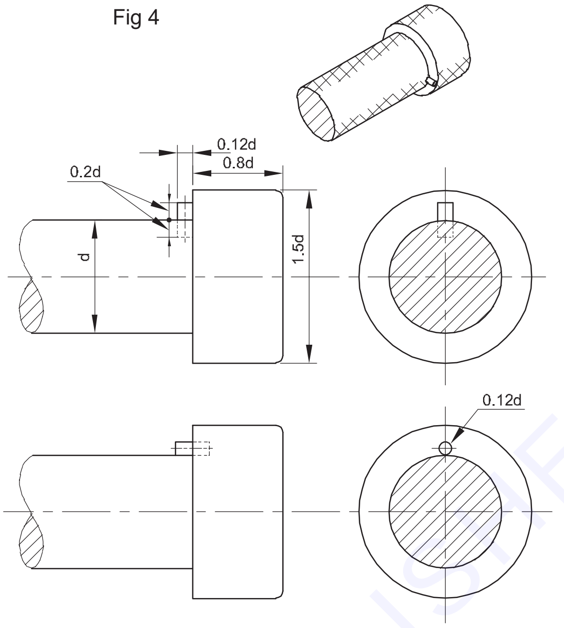
Standard 'T' bolts: There are T bolts as per IS;2014 to suit T slots according to IS:2013. (Fig 6b)

Fig 3



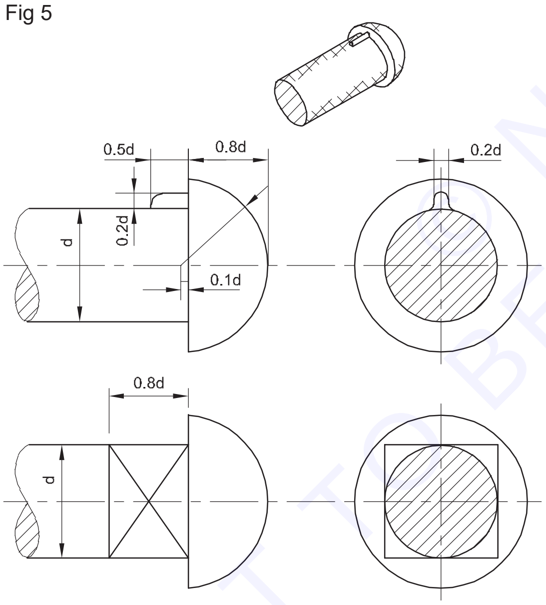
SQUARE HEADED BOLT (WITH SQUARE NECK)

Fig 4



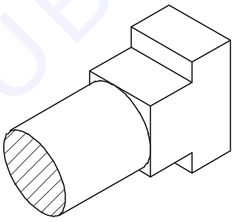
CYLINDRICAL OR CHEESE HEADED BOLT

Fig 5

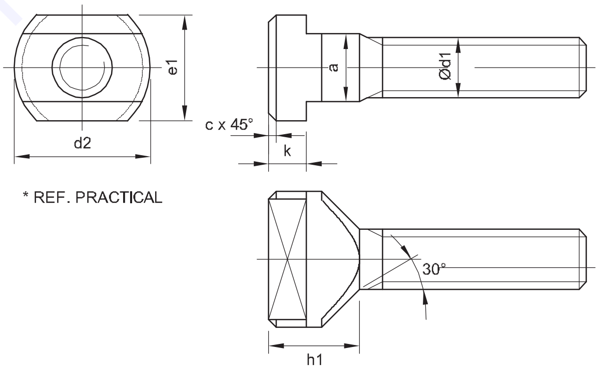


CUP HEADED OR ROUND HEADED BOLT

Fig 6



a) T - HEADED BOLT



(b) T - BOLT AS PER IS : 2014

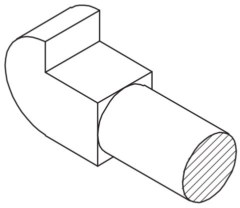
Hook bolt: It is slightly different from ‘T’ head bolt. It is used when it is not possible to drill a hole in the component in position. (Fig 7)

Eye bolt: It has a short hollow cylinder head attached to the shank perpendicular to its axis. Because of its flat circular faces, rotation of bolt is prevented while handling nut. (Fig 8)

Lifting eye bolt: Its head is in the form of Tarso, ring made of cylindrical rod. It is forged with a washer shaped base and shank. The shank is threaded and fits into the machine/motor body. It is used to lift the machines or motors for shifting and aligning during erection. (Fig 9)

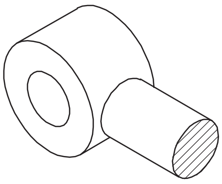
Counter sunk head bolt: It has a counter sunk head which does not project outside the hole. Its rotation in hole is arrested by means of a **snug** on the taper surface or square neck, by the suitable cavity on the component. (Fig 10)

Fig 7



HOOK BOLT

Fig 8



EYE BOLT

Fig 9

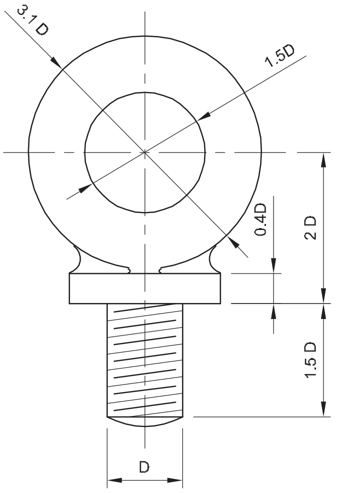
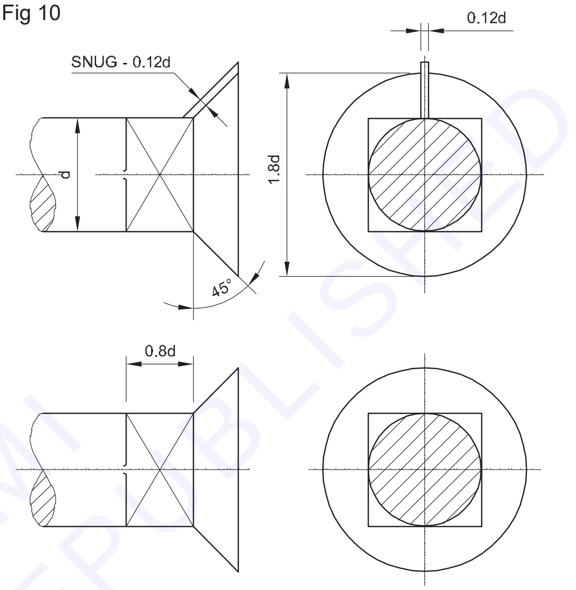


Fig 10



COUNTER SUNK - HEAD BOLT

Cap screw or tap bolt: Where there is no possibility of using a nut, tapped hole is provided in the major part of the assembly and a clearance hole is drilled on the part to be fitted. (Fig 11)

Designation: Threaded fasteners are designated in the sequence as an example shown Hex.bolt M20 x 75 -IS:1364-B.

1 Nomenclature: Hex.bolt (shape of head)

2 Size - M 20 x 75 x 1.5 dia 20; length 75, pitch 1.5

3 Type - A,B or C. A - Precision
B - Semi-Precision
C - Black

Type designation be avoided as far as possible in Product standard.

Holes for taking the bolts are of slightly larger diameter, ie; clearance holes of fine, medium and coarse series of tolerance grades H12,H13 & H14 vide IS:1821. (Dimensions for clearance holes for bolts and screws)

NB: Recommended dia and Nominal lengths of bolts, screws, studs and thread lengths are available in IS:4206

Dimensions for width across flats for hexagon, hexagonal flanged bolts, screws and nuts ref IS:9519.

IS:1363 Part-1, 2 & 3 for Grade C

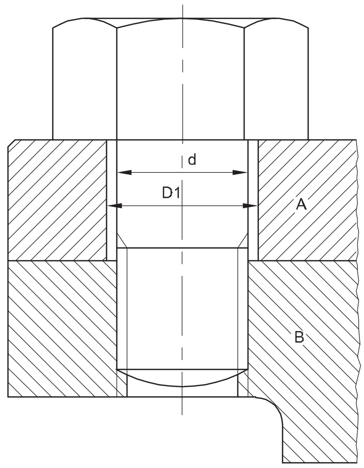
IS:1364 (Part 1 to 5) for Grade A and B of hexagonal bolts, screws and nuts.

Hexagonal socket head cap screw (IS:2269): These are available from M1.6 to M36 (16 sizes), length 2.3 mm to 200mm depending upon the dia. It has the advantage over the other set screws, its hexagonal socket head flushes in the metal, so the head does not project outside.

They are used on machine tables and on assemblies. (Fig 12) It is operated with an allen key.

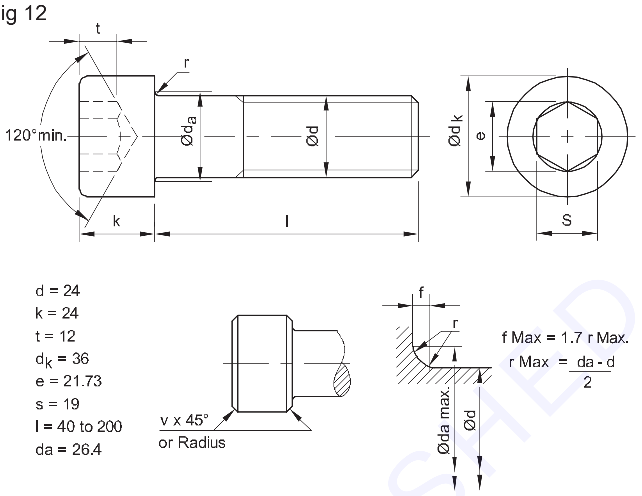
Designation: A hexagonal socket cap screw of size M12 length 60 mm property class 12.9 shall be designated as Hexagonal socket head cap screw M12 x 60-12.9-IS:2269.

Fig 11



TAP - BOLT OR CAP - SCREW

Fig 12



d = 24
k = 24
t = 12
d_k = 36
e = 21.73
s = 19
l = 40 to 200
da = 26.4

f Max = 1.7 r Max.

r Max = $\frac{da - d}{2}$

Nuts and washers

Nut is a metallic piece of definite shape with threaded (screwed) hole on the centre of the face. It is used on the end of the bolt/screw to hold the parts in position.

Nuts are known by their shape or their cross-section. The most commonly used forms are hexagonal and square. Nuts are specified by the shape of the nut and the nominal dia of bolt/screw on which they are used. (Fig 1)

Fig 1

Hexagonal nut (Fig 2)**:** it is made of hexagonal bar with a screwed/ threaded hole in the centre. To avoid the damaging of the corners on the face, they are chamfered at 30°, with reference to the base. Theoretically the thickness of the nut is equal to the diameter of the bolt and corner to corner is 2d i.e., twice the diameter of the bolt.

The actual sizes are specified in IS:1363, 1364, 3138.
Thin hex.nuts are available IS:1364 (Part-4).

Square nut (Fig 3)**:** It is made out of square bar. Side of the square is equal to 1.5d + 3 mm and radius of the chamfer is 2d. Sizes are from M6 to M39.

IS:2585 lays down the standard sizes available.

For drawing purposes, the following sizes of the nut are considered.

Bolt nom. dia = d

Thickness of nut T = d

Width across Flats W = 1.5d + 3mm

Angle of chamfer = 30°

Radius of chamfer arc = 2d (approx.)

Fig 2

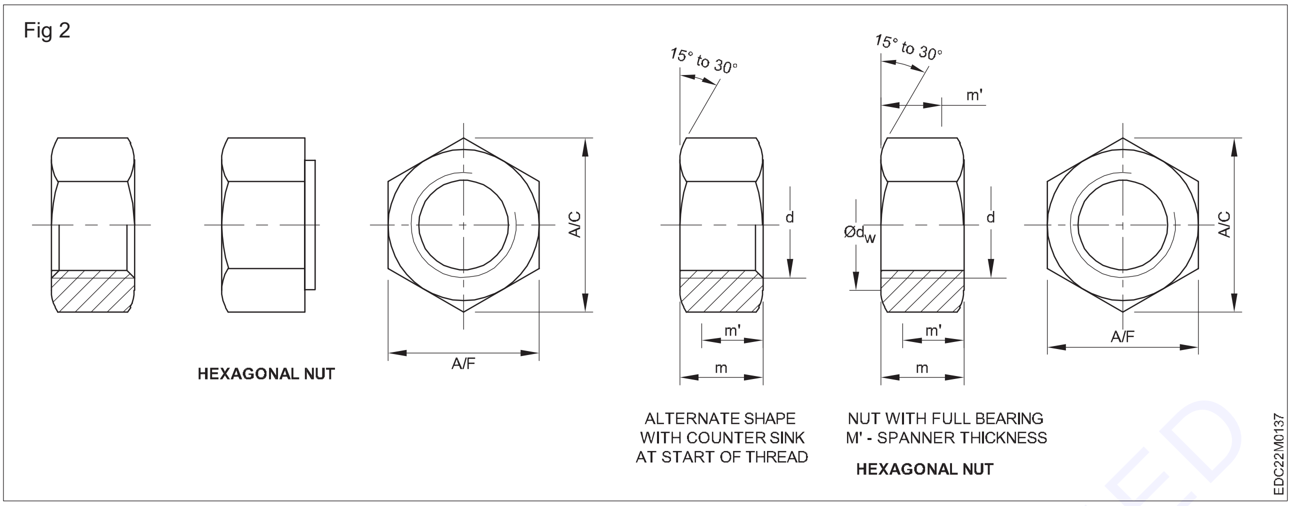
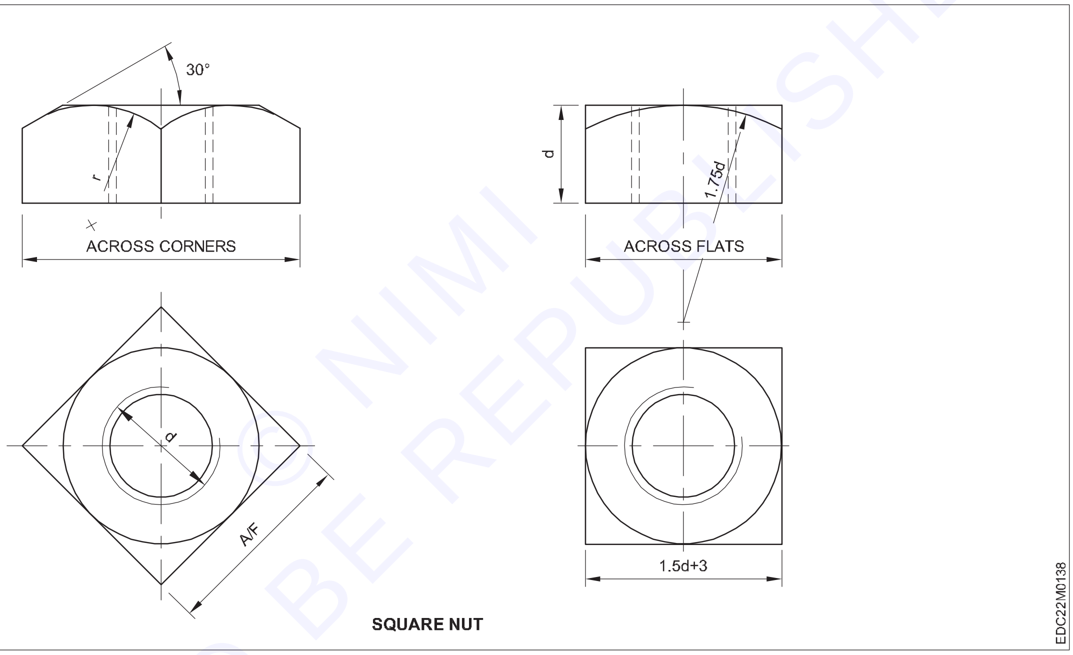


Fig 3



Special Nuts

Collared nut: It is a regular hexagon nut with a washer face. It avoids cutting groove on the metal while tightening and also can be used on comparatively larger holes. It is available from M8 to M36 IS:7795-1975. (Fig 4)

Cap nut: it is also a hexagonal nut, the hole covered by a cylindrical flat cap. It protects the end of the bolt from corrosion and leakages of oils through the threads. Figure shows the nut. It has a cavity of 0.25 d and end metal thickness 0.25d. (Fig 5a)

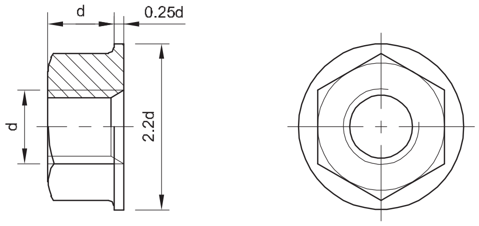
Domenut(as per BIS) Domenut: it is a hexagonal nut with blind hole having hemispherical top end. Its use is same as above. Dome nut is available from M6 to M34 IS;2687

It is made of steel, brass or aluminum. Steel nuts may be from rolled, forged or extruded. Nuts of sizes M24 and above. (Fig 5b)

Capstan nut or cylindrical nut: It has six blind hole equispaced on the circumference. The diameter and depth of the hole is equal to 0.2d. Diameter of the nut is 1.8d. A hook spanner is used to operate. (Fig 6)

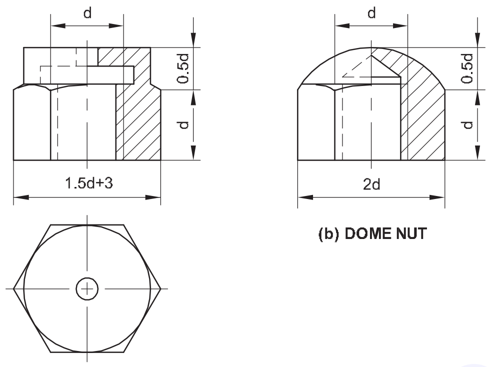
Ring nut: Its diameter is 1.8 d and thickness is 0.5 d. The thickness is reduced to 0.3d, by 0.1 d from both the faces forming 1.5d cylindrical faces. Six slots of 0.2d width are milled on the circumference to the depth of 0.15d. Special spanner is used to operate the nut. (Fig 7) These are used in pairs, one nut acting as a lock nut.

Fig 4



COLLARD OR WASHER FACED NUT

Fig 5



(a) CAP NUT

(b) DOME NUT

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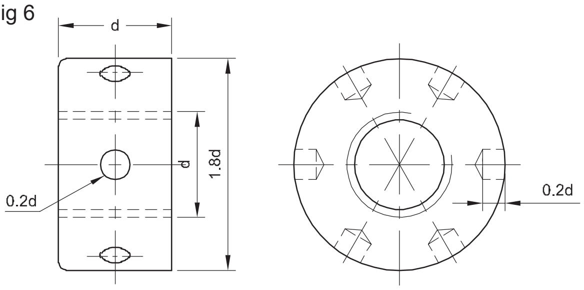
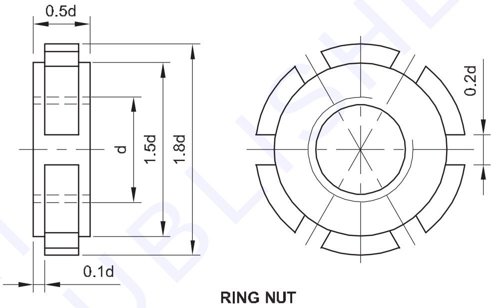
Fig 6



CAPSTAN NUT

Fig 7



RING NUT

EDC22M0140

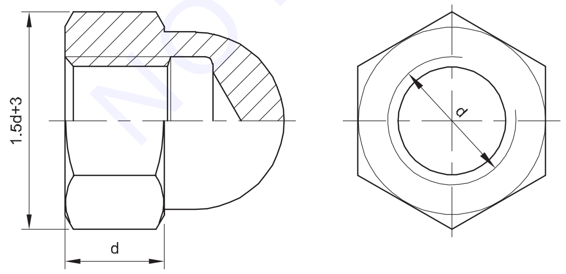
Domed Cap nut (IS:7790): Hexagon nut with a hemispherical top. Its function is similar to cap nut. It is available in 11 sizes from M6 to M24 diameter bolts/screw. They are made of steel, brass or aluminum alloy. It prevents corrosion and leakage of fluids from the threads. (Fig 8)

Washer: It is a cylindrical thin disc with a hole in the centre. Washer helps to provide smooth bearing surface between the nut and the part on which it is used. There by it prevents the nut corners from cutting into the metal. (Fig 9)

There are plain washers (IS:2016:5370) and spring washers (IS:6755,3063,6735).

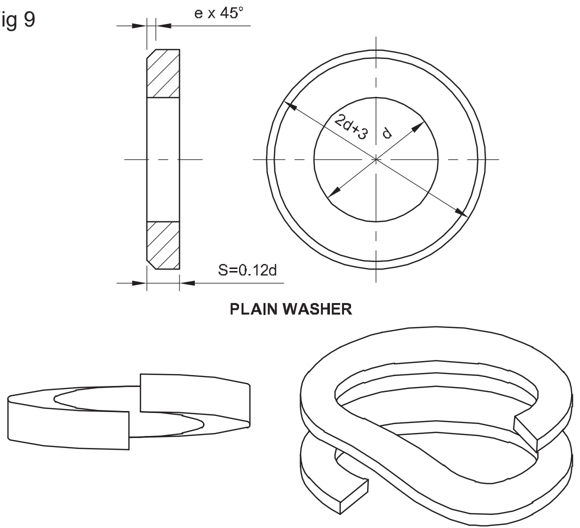
Plain washers are available from basic hole size 4 to 155 mm dia to suit bolt/screw size from M 1.6 to M 150 mm. One face is chamfered at 45$^{o.}$ While spring washer exerts pressure on the nut tightening and keeps the thread gripped on the thread.

Fig 8



DOMED CAP NUT

Fig 9



PLAIN WASHER

SPRING WASHER

EDC22M0141

Screws

Grub screws are comparatively smaller in dia and short in length. They are made of hardened steel,used for holding two parts in position eg., a collar pulley, or gear on a shaft.

On driving it into position, it produces a clamping force, resists relative motion between the assembled parts. The grub screws are fully cylindrical , on one end they are provided with hexagonal socket or screw driver slot. On the other end they are formed into different points. (Fig 1)

Hexagonal socket grub screws or screw driver slot are of following types.IS: 6094 (Fig 1)

They are namely as follows:

- Flat point (FP)
- Cone point (TP)
- Full dog point (FDP)
- Half dog point (HDP)
- Cup point (CP)
- Conical point

These are designated by the shape of their ends. These are available in sizes M3, M4, M5, M6, M8, M10, M12, M16, M20, M 24, Max. length 60 mm. These are operated by using hex. bent key (Allen key).

Slotted grub screws as per IS:2388 are designated by letters. (Fig 2)

Fig 1

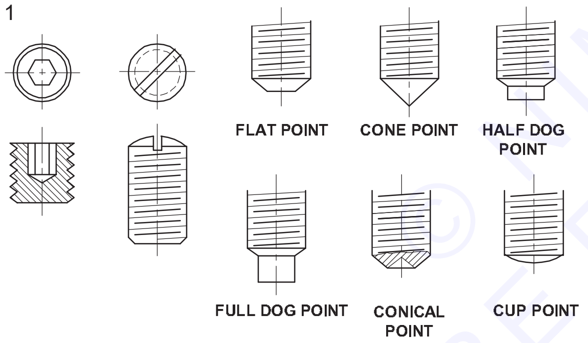
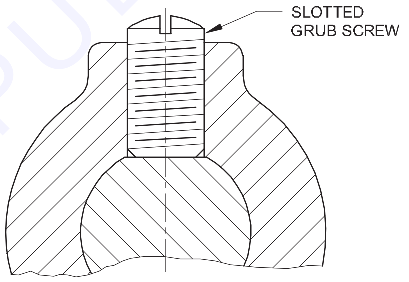


Fig 2



EDC22M0142

- Type A - Flat end
- Type C - Conical end
- Type E - Cylindrical dog point
- Type G - Tapered dog point
- Type J - Cup point
- Type K - Oval point

Set screws (Not as per BIS)**:** These screws are operated by screw drivers or a spanner. They are available in different sizes M1 to M24. The lengths vary according to dia.

Function of a set screw is same as that of grub screw. The set screw has different form of head (i) cup or round head (ii) square head (iii) cylindrical or cheese head (iv) filister head (v) counter sunk head (vi) rounded counter sunk head (vii) square head with collar. Fig 3 shows the above screws.

Screws are specified by designating the head, diameter; length and IS:No and property class.

Locking devices

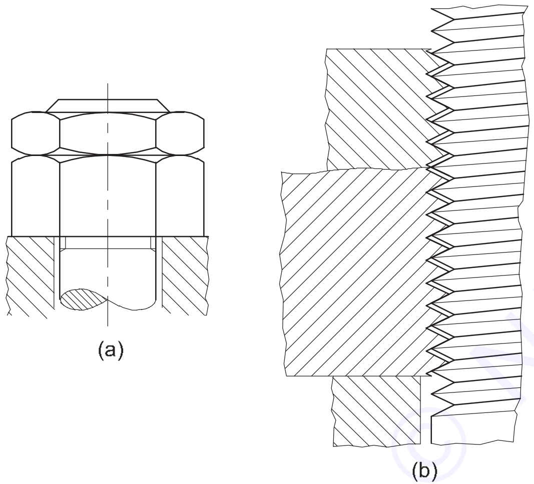
Locking devices: There are different types which are used to prevent nuts from getting loose on machines subjected to vibrations or impacts.

The most commonly used device is a lock nut. It is an additional nut called Lock nut engaged on the other. The thickness of lock nut may be 0.6 d to 0.8 d; d being the diameter of the bolt. The top nut when tightly engaged on the nut, it prevents the nut becoming loose. (Fig 1)

Split Pin: It is made of semi circular steel wires (IS:549) bent through 180°, forming a spherical head. It is passed through the split pin holes in the bolt/screw ends and the ends are opened back wards. There by it locks the nuts in their position. They are available in 16 sizes, 0.6 mm to 20 mm depending upon the bolt dia 2.5 to 170 mm dia. (Fig 2)

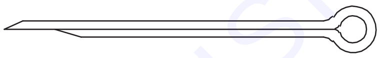
Slotted nut IS:2232: It is a hexagonal nut with slots of width 0.25 d cut on the top end of the nut through the opposite faces. A split pin is inserted through the slot in the hole drilled on the bolt end in line with the slot. The split end of the pin is opened securing the nut. The width of the slot is 0.25 d and 0.3 d deep, split pin dia is 0.2 d. (Fig 3)

Fig 1



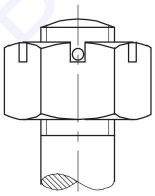
LOCK NUTS

Fig 2



SPLIT PIN

Fig 3



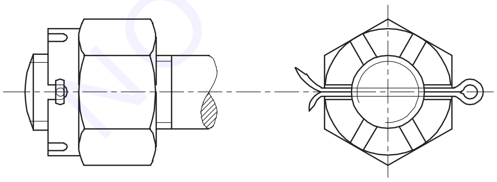
SLOTTED NUT

Castle nut IS: 2232: It is similar to slotted nut with semi circular end slots cut on the cylindrical part called “castle”, on the nut. A split pin is passed through and lock the nut. (Fig 4)

The above are available in grades A,B & C sizes M4 to M100.

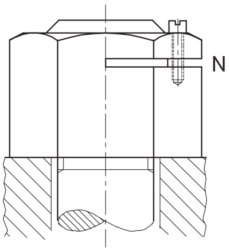
Sawn nut or Wiles nut: It is a normal hexagonal nut cut half way through one of its corners. The width of the slot is 0.15d located at 0.2 d from top of the nut. A tapped hole is cut on the nut as shown. A set screw when tightened at the top thin part of nut slightly bends, there by pressing on the threads. This prevents the slackening of nut. (Fig 5)

Fig 4



CASTLE NUT

Fig 5



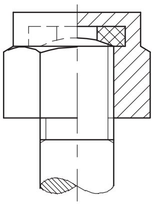
SAWN NUT

Simmond’s lock nut: The hexagonal nut has a closed cavity on one end and a fiber ring is fitted inside it. The internal dia meter of a fiber ring is slightly less than the core diameter. When the nut is screwed down, bolt end cuts thread on the fiber ring.

The fiber ring gives greater grip over the bolt threads and prevents slackening the nut. (Fig 6)

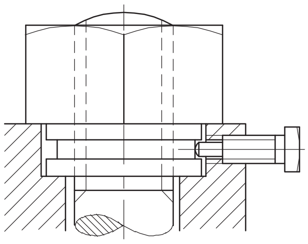
Penn, ring or grooved nut: A part of the nut is turned round with a round groove. When the bolt hole is very close to the edge, a counter bored hole is provided. The dog point of set screw, through the tapped hole, sits on to the groove. When screwed in, the nut is prevented from getting loosened. (Fig 7)

Fig 6



SIMMOND'S LOCK - NUT

Fig 7

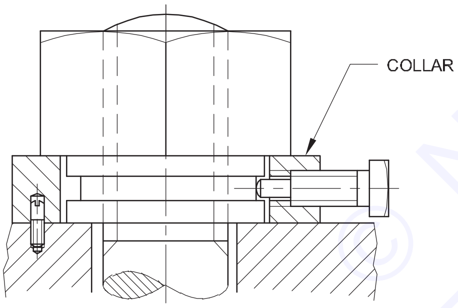


PENN, RING OR GROOVED NUT

The same nut can be locked as shown in Fig 8 with a collar when it is away from edge.

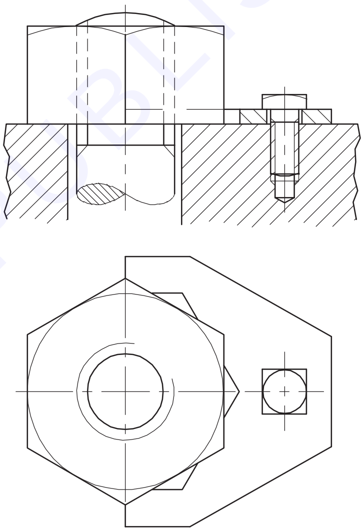
Locking by Locking plate or stop plate: a plate with grooves to suit the nut is placed on the nut and screwed down to the body with a screw. (Fig 9)

Fig 8



PENN, RING OR GROOVED NUT

Fig 9



LOCKING PLATE

Locking by screw: A screw fixed by the side of the face prevents movement of the nut. (Fig 10)

Apart from the methods mentioned above, spring washers, single coiled or double coiled washer are used to lock the nuts. By tightening the nut, the nut presses the spring and deform. This exerts pressure on the nut, prevents rotation of nut, when tightened in position. (Fig 11)

Locking by wire: screw ends have holes through which wire is passed after the nuts are screwed down. Wire prevents nuts coming off.

Stud or stud bolt (Fig 12)**:** It is only a cylindrical shank having threads on both ends, with the plain portion in between. One end is shorter, that is screwed into the machine body called “Metal end”. The other end longer one which takes nut is called Nut end. (Fig 12a) Some studs have a collar called “collar studs”. (Fig12c) Studs are screwed into the metal end, using double (lock nut) nut method. After removing the nuts, the other part with plain hole is placed in position and nuts are screwed on to the nut end. (Fig 12b)

There are three types of studs:

Type A - recommended for use in steel; metal end length = 1 d.

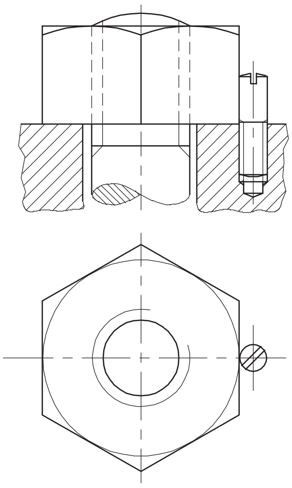
Type B - recommended for use in cast iron; metal end length = 1.5d.

Type C - recommended for use in aluminum, metal end length = 2 d

M4, M8, M12, M16 & M20 are available. Lengths 25 mm to 200 according to dia.

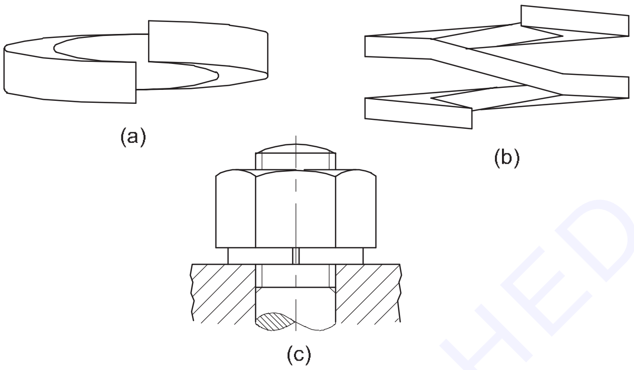
Designation stud B M24 x 200 IS:1862.

Fig 10



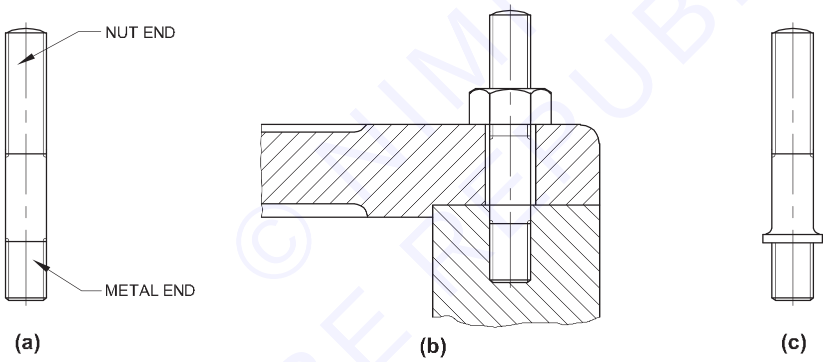
LOCKING BY SCREW

Fig 11



SPRING WASHER

Fig 12



Foundation bolts

Machines are generally subjected to vibration. Due to this the machines are likely to shift or move from their positions. To overcome this the machines are fixed to the ground with the help of devices called foundation bolts.

These bolts do not have a specific shape of head like Hexagonal or square bolts. The length of the shank is according to the thickness of nut and the thickness of machine base. This odd shaped part holds the machine firmly to the ground.

Types (Fig 1 & 2)

Fig 1

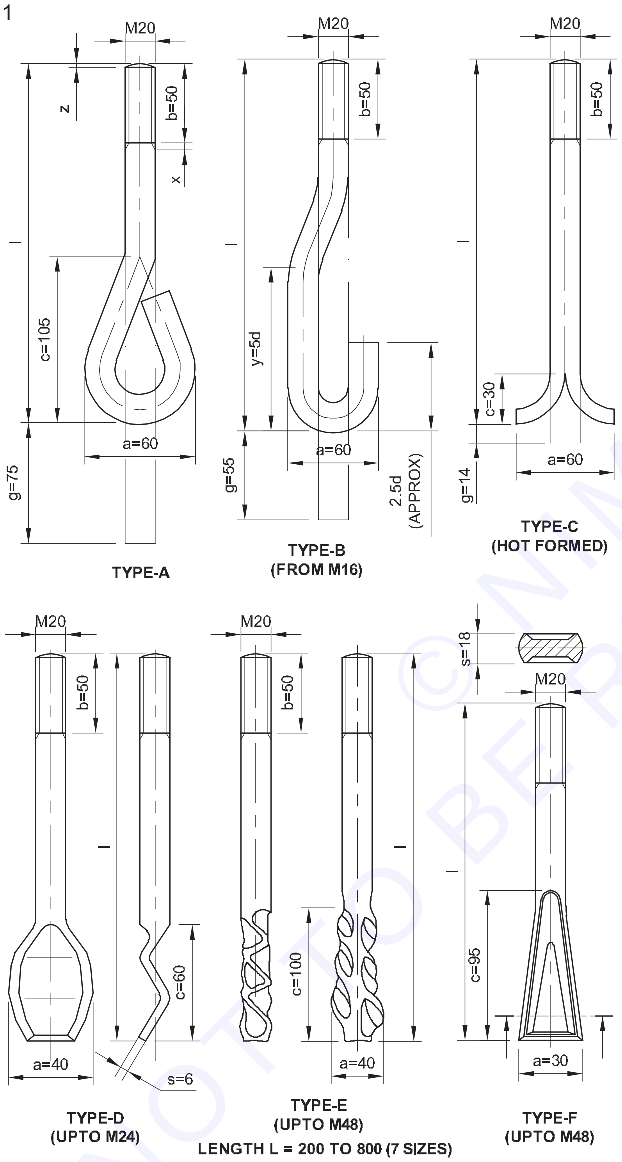
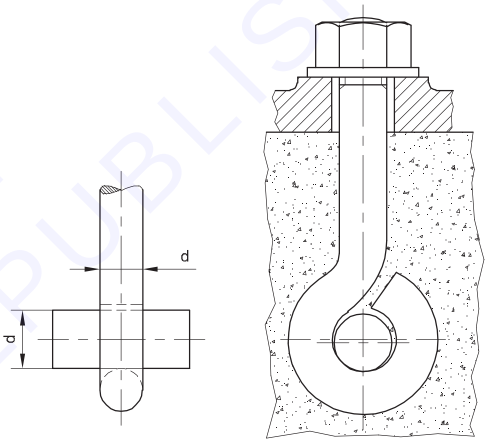


Fig 2



EYE FOUNDATION BOLT

EDC22M0149

As per IS:5624-1970, there are six types designated as Type A, B, C, D, E & F. Fig.1 shows the same. They are available in 13 dia sizes from M8 to M72, lengths 80mm to 3200 mm. These are designated by the shank dia and length, without nut. The ends are formed by forging.

There are other non standard forms which are generally used are:

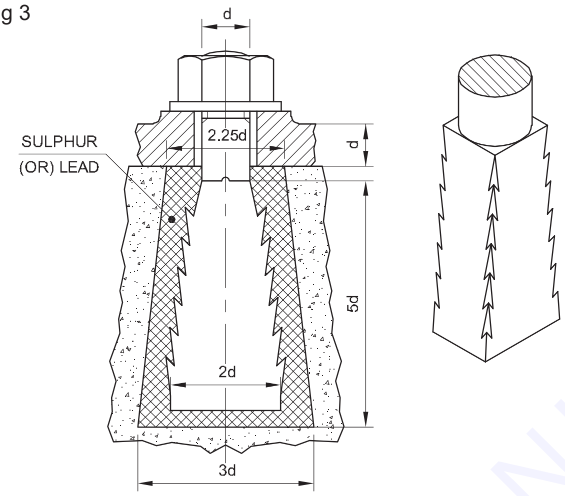
- Eye foundation bolt
- Rag foundation bolt
- Lewis foundation bolt and
- Cotter foundation bolt

Rag bolt (Fig 3)**:** It is in the shape of a rectangular pyramid with round shank on the tapered end. The corners are formed into rags or grooved, forming small projections. These are placed in the foundation cavity in position as above and molten lead or sulphur is poured around it. When the molten lead sulphur/lead solidified, the bolts are held firmly. The machines are placed in position and nuts are fixed. By melting the lead or burning sulphur the bolt can be removed.

Lewis Foundation bolt (Fig 4)**:** It is a rectangular shank bolt with one side taper. A gib head key is placed on the other side of taper as in earlier type and concrete is put around. It is aligned. The foundation bolt can be easily withdrawn by removing the gib head key first and then the bolt.

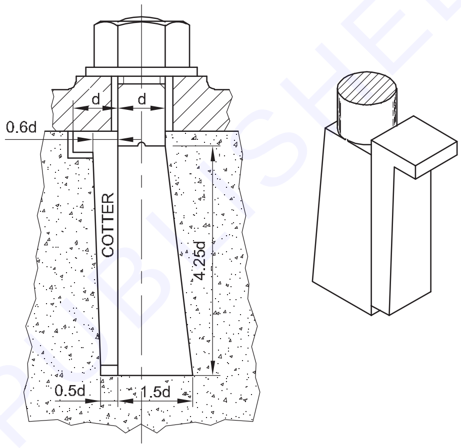
Cotter bolt (Fig 5)**:** It has a rectangular slot through which a double headed cotter is placed. A cast iron washer rests above the cotter. Through a hand hole, connecting the cavity in the concrete the bolt is pulled down and lifting the cotter, the cotter is placed in position.

Fig 3



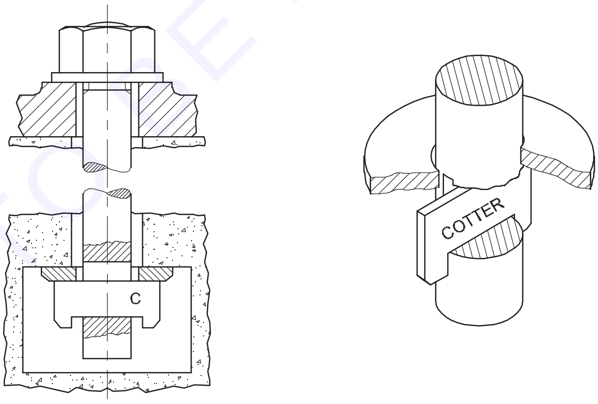
RAG FOUNDATION BOLT

Fig 4



LEWIS FOUNDATION BOLT

Fig 5



COTTER FOUNDATION BOLT

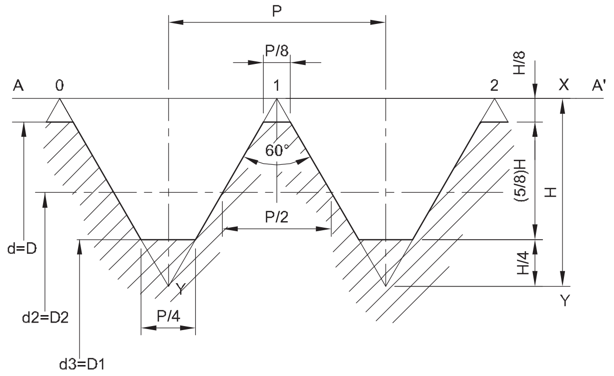
Procedure

Screw threads

Exercise8.1

Draw the basic profile of ISO metric external thread M30. (Fig 1)

Fig 1



EXTERNAL METRIC THREAD

Pitch M30 = 3.5 mm

P = 3.50 mm	P/8 = 0.4375 mm
P/4 = 0.8750 mm	H = 3.0311 mm
H/8 = 0.3789	H/4 = 0.758 mm

Since the above mensioned sizes are very small, we can take suitable enlarged scale.

We can take 10:1 scale and draw the external ISO metric thread as shown in Figure 1.

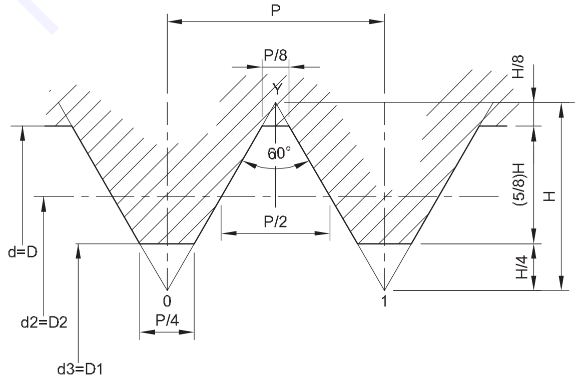
In 10:1 scale approximate values of different dimensions.

P = 35 mm	P/8 = 4.5 mm	P/4 = 9 mm
H = 30 mm	H/8 = 4 mm	H/4 = 8 mm

Exercise 8.2

Follow the procedure of Ex.15.1 and complete the drawing of Iso-metric internal thread. (Fig 2)

Fig 2

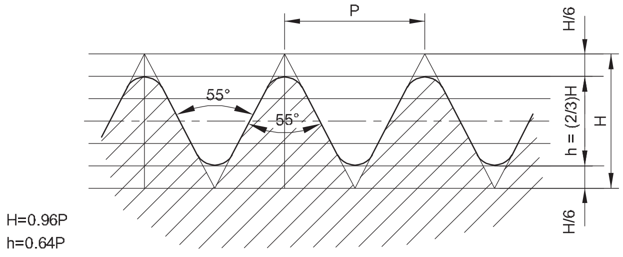


INTERNAL METRIC THREAD

Exercise 8.3

Draw the profile of 1" BSW external thread. (Fig 3)

Fig 3



BRITISH STANDARD WHITWORTH (BSW) THREAD

1 inch BSW has 8 threads per inch.

Therefore the pitch = 1/8 inch = 3 mm approximately.

H = 0.968P

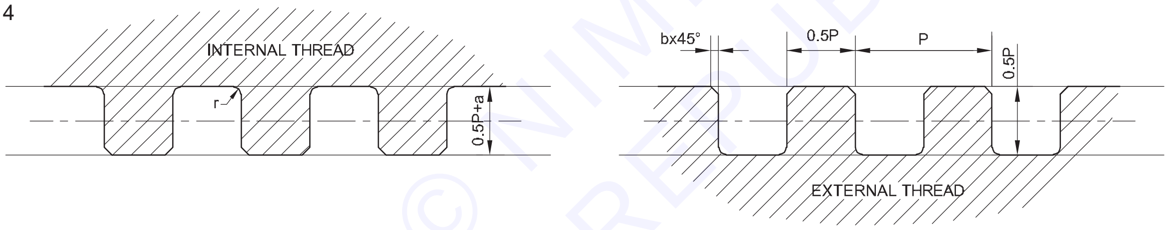
Radius of curve = 0.137P

Draw the thread profile to the enlarged scale 10:1. Assume the dimensions suitably and complete the drawing.

Exercise 8.4

Draw the profile of square thread (ϕ 30 mm & pitch 6 mm) (Fig 4)

Fig 4



a = 0.25mm, b = 0.5mm, r = 0.25mm

SQUARE THREAD

Depth of thread = 0.5P
Thread angle = 90°

Draw profile of square thread to a suitable magnified scale according the space available.

Exercise 8.5, 8.6 & 8.7

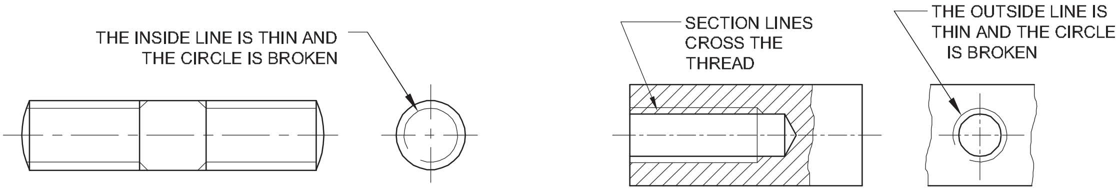
Draw the profiles of Acme, buttress and knuckle threads by referring figures 2,3 & 4 of Exercise different types of thread Ex.15. Use suitable enlarged scale according to the availability of space in the workbook sheet.

Exercise 8.8

Draw the conventions of external and internal thread with edge views. (Fig 5)

Draw the conventions of threads as shown above also indicate the type and size of thread as mentioned in the dimensioning technique exercise.

Fig 5



CONVENTION OF EXTERNAL THREAD

CONVENTION OF INTERNAL THREAD

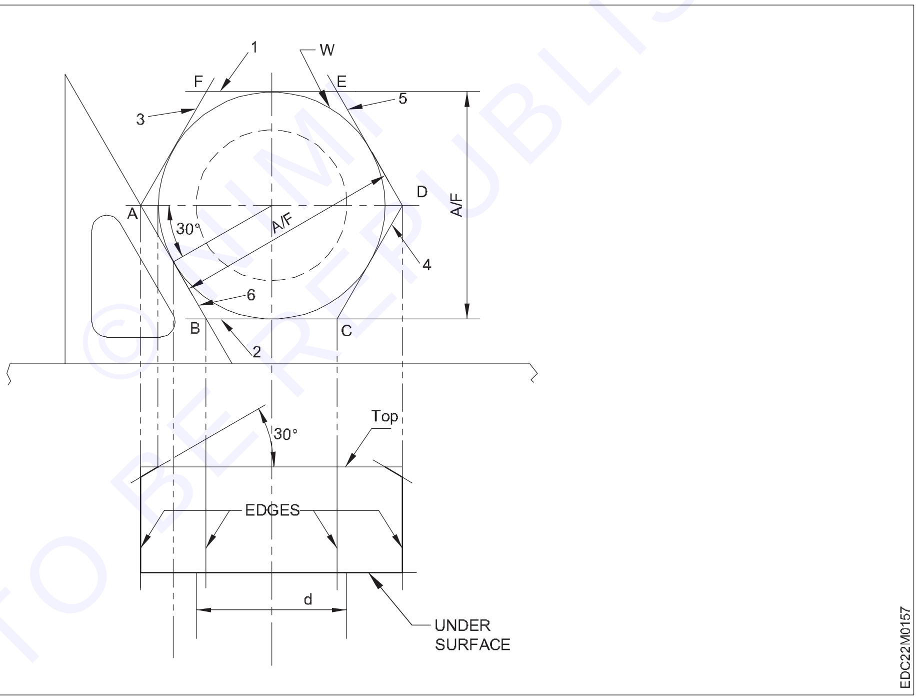
Exercise 8.9

Sketch the views of a hexagonal bolt M10 x 50.

Proportion of bolt head and other features.

- Given bolt diameter = 10 mm (M10).
- Bolt head across flat = 1.5d + 3 mm
  = 1.5 x 10 + 3 = 18 mm
- Bolt head thickness = 0.8d to 0.9d
  = 0.8 x 10 = 8 mm
- Radius of chamfer = 1.5d or 1.4d
- Length of bolt = 3d to 4d approx
  in this case = 50 mm.
- Sketch a regular hexagon of side 10 mm.
- Sketch a hidden circle touching the sides of the hexagon.
- Sketch a concentric circle of ϕ 10 mm. this is the plan of the bolt. (Fig 6)

Fig 6



- Sketch projectors from the corners of hexagon parallel to axis.
- Form a rectangle of the projectors equal to the thickness of the bolt head. (8 mm in this case)
- Sketch 30° lines as shown in Fig 6.
- Sketch arcs as shown in Figure 7.
- Sketch the shank to the specified length. Also sketch the threaded portion as per convention. (Fig 8)

Fig 7

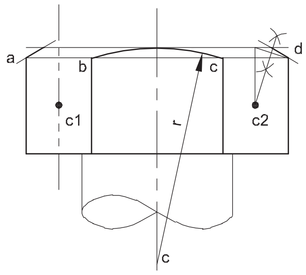
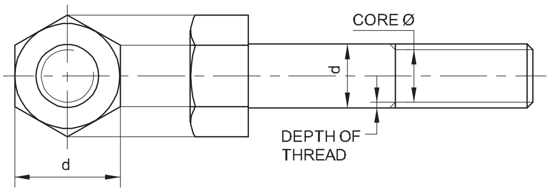


Fig 8

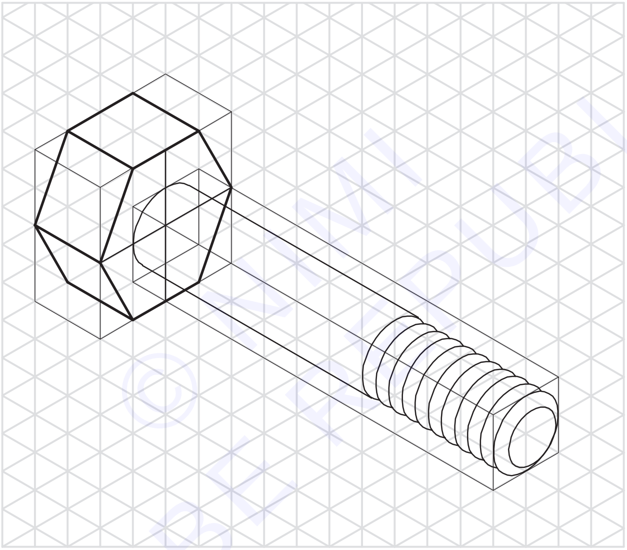


Exercise 8.10

Sketch the pictorial view of a hexagonal bolt.

Sketch the view as shown in Figure 9.

Fig 9



Exercise 8.11 to 8.18

Sketch or draw the exercises 15.11 to 15.18 by referring the following figures in the theory of Ex.15.

15.11 - Tee bolt

15.12 - Eye bolt

15.13 - Hexagonal nut (Views)

15.14 - hexagonal nut (Pictorial sketch)

15.15 - Hexagonal Socket set/Grub screw

15.16 - Flat point grub screw with screwdriver slot

15.17 - Flat washer

15.18 - Socket head cap screw.

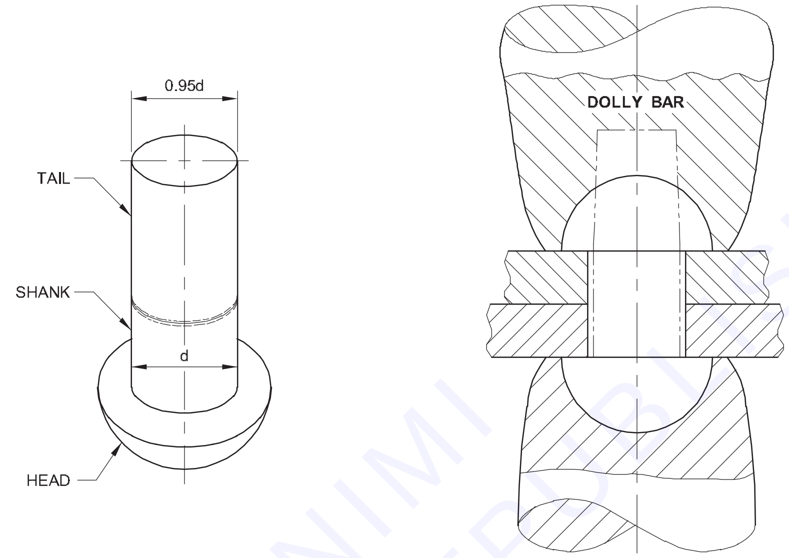
Exercise 8.19 to 8.24

Reproduce the figures of bolt assembly, nuts and foundation bolts shown in Ex. 15.19 to 15.24 of work book. For sketching/drawing use the space by the side of figures.

Rivets

Rivet is a cylindrical rod of either carbon steel or wrought iron or non-ferrous metal. It consists of a head and shank tapering at the end facilitating easy placement in the rivet holes. Rivets are used to hold the plates or steel sections firmly, just as in steam boilers, girders, steel structures, ship building. The rivet holes are made either by punching or by drilling. When the plates are aligned, the heated rivets are placed through the holes. By using rivet sets, the tail part of the shank is formed into the head closing the hole. The plates are held firmly between the heads on cooling. To prevent leakages of boiler shells, during which the plate ends and rivet head end are pressed by hammering using a tool similar to blunt cold chisel. Fig 1 shows how riveting is done.

Fig 1



RIVET AND RIVETING

Types of rivets:

- Boiler rivets
- Hot forged rivets for hot closing
- Cold forged solid steel rivets for hot closing
- Non-ferrous rivets
- Cold forged rivets for cold closing
- Bi-furcated rivets for general purposes
- Solid drilled tubular and semi-tubular rivets.

Rivets are also broadly classified as:

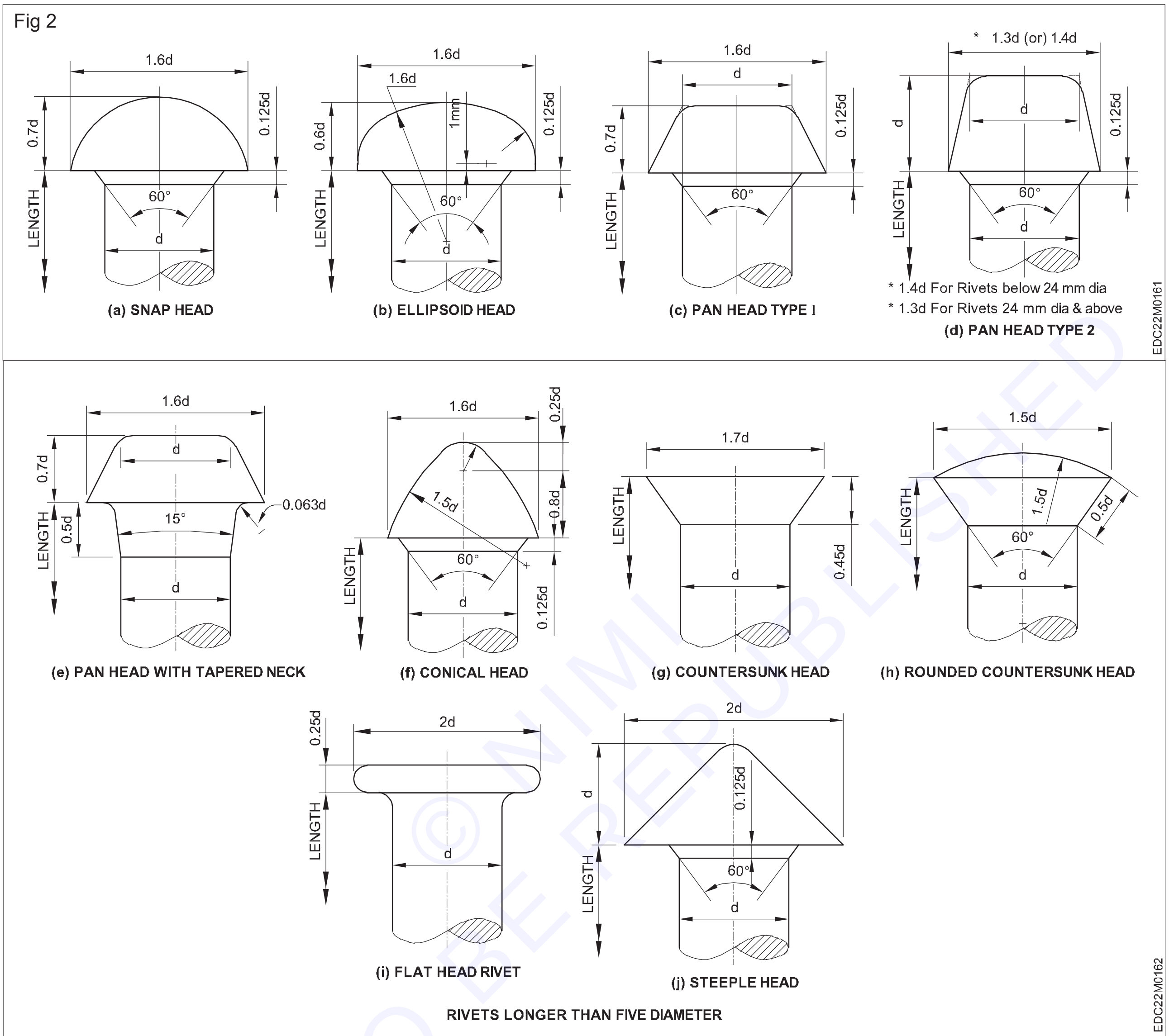
- Structural rivets
- Boiler rivets
- Small rivets for general work
- Rivets for ship building
- Aircraft rivets.

Rivets are manufactured either by cold heading or hot forging. Cold headed rivets are adequately heat-treated so that the stresses set up in the cold heading process are eliminated. If they are hot forged, the finished rivets are cooled gradually. The rivets should be concentric with the shank. They shall be free from imperfection and heavy scales. The ends of the rivets shall be clearly sheared and free from rags or burrs.

They are available in different lengths and selected according to the requirements (IS:1928 for boiler rivets, dia 12 to 48 mm).

Boiler rivets of different types of heads as per IS:1928-1961 are shown in Fig 2 . They are

Fig 2



(a) SNAP HEAD

(b) ELLIPSOID HEAD

(c) PAN HEAD TYPE I

* 1.4d For Rivets below 24 mm dia

* 1.3d For Rivets 24 mm dia & above

(d) PAN HEAD TYPE 2

(e) PAN HEAD WITH TAPERED NECK

(f) CONICAL HEAD

(g) COUNTERSUNK HEAD

(h) ROUNDED COUNTERSUNK HEAD

(i) FLAT HEAD RIVET

(j) STEEPLE HEAD

RIVETS LONGER THAN FIVE DIAMETER

- Snap head
- Ellipsoidal head
- Pan head type I & II
- Pan head with taper neck
- Conical head
- Countersunk head
- Rounded countersunk head
- Flat head
- Steeple head

Boiler rivets are used in boiler shells, mountings gussets stays etc. They have generally tapered necks at 60° chamfer for preventing the leakages of steam. Table 1 shows the preferred length and diameter combination for boiler rivets.

Structural/General purpose rivets IS:1929, IS:2155: Structural rivets are used in structures like girders of bridges, trusses of roof etc.

Their shank dia (IS:1929) varies from 12 mm to 36 mm. The proportions shown are for rivets of dia from 12 mm to 36 mm. The above rivets are used even for general work.

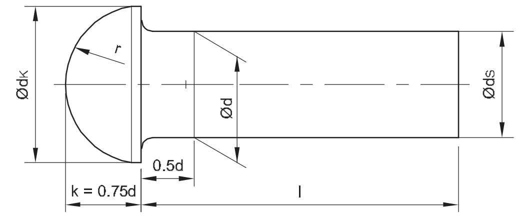
The dimensions are from 12 mm to 50 mm. Structural rivet and boiler rivets are made of low carbon steel and wrought iron. (Fig 3)

Small rivets are used for general work, may be made of "non-ferrous metals" viz copper, brass or aluminium. Diameters are 1.6, 2, 2.5, 3, 4, 5, 6, 8 and 10 mm of varying lengths in steps of 2 mm upto 24 mm and then in steps of 3 mm. IS:2907.

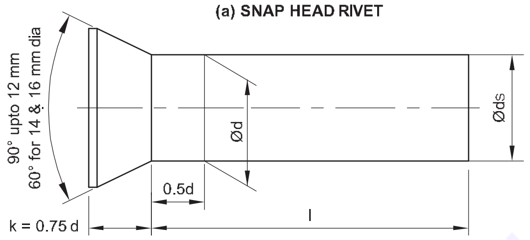
Rivets are designated by its shape, dia, length, IS. No and materials.

Ship building rivets (Fig 4)**:** (IS:4732)

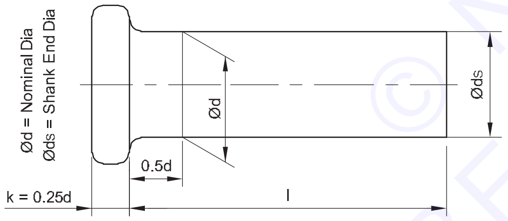
Fig 3



(a) SNAP HEAD RIVET



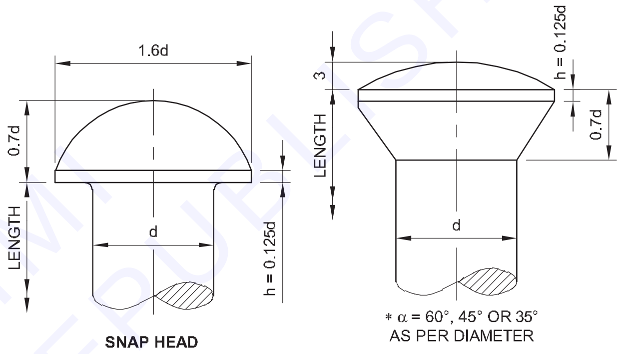
(b) FLAT COUNTERSUNK HEAD RIVET



(c) FLAT HEAD RIVET

HOT CLOSING RIVETS

Fig 4



SNAP HEAD

COUNTERSUNK HEAD

SHIP BUILDING RIVETS

- Snap head
- Countersunk head.

The dia ranges from 16 to 33 mm in steps of 2 mm upto 24 mm and rest in steps of 3 mm. The length of rivets vary from 27 mm to 180 mm in steps of 3 mm. They are made of mild steel bars.

The snap head or cup head rivet is the most commonly used rivet. Countersunk head rivets are used where the top surface of the plate is required to be free from projecting rivet heads. Rivet proportions given in the figure are with reference to shank diameter of the rivets.

Selection of rivet dia: The diameter of the rivet is based on the thickness of the plates to be joined. A general formula to find the dia `d' for plate thickness `t' is

d = $6\sqrt{t}$, where `t' is in millimeters.

d = 1.2 $\sqrt{t}$ to 1.4 $\sqrt{t}$, where the `t' is in inches.

Recommended value

Thickness of plate t in mm

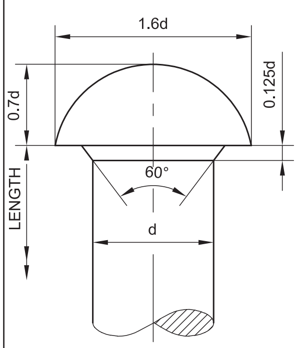
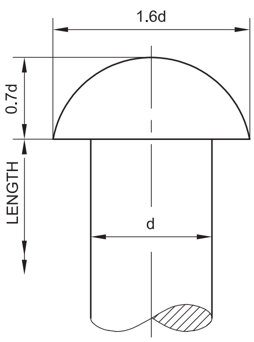
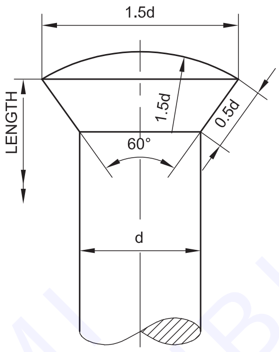
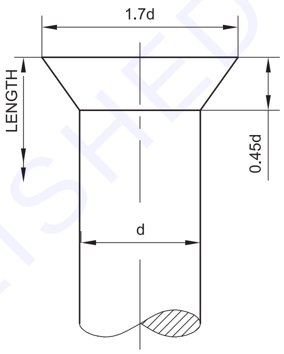
8, 9, 10, 11, 12, 14, 16, 18, 20, 22, 25

Dia of rivet d in mm

17, 18, 19, 20, 21, 22, 24, 25, 27, 28, 30

Length of rivet is selected according to the thickness of plates and just long enough to fill the hole completely and make the head. Rivet joints are formed by drilling holes in plates together (grade of hole H 12). The rivet is placed through the holes and the shank is either hot forged or cold forged.

Table 1

<table>
<tr><td colspan="4">Rivets for general purposes and boiler rivets (12 to 48 mm diameter) IS:1929-1961
IS:1928-1961</td></tr>
<tr><td>Snap head rivet for boilers</td><td>Snap head rivet for general purpose</td><td>Rounded countersunk head for general purpose and boilers</td><td>Flat countersunk head for general purpose and boilers</td></tr>
<tr><td> </td><td> </td><td> </td><td> </td></tr>
<tr><td colspan="4">Basic size of rivets, D, mm: 12 14 16 18 20 22 24 27
30 33 36 39 42 48</td></tr>
<tr><td colspan="4">Preferred length and diameter combination for rivets for general purposes and boiler rivets</td></tr>
</table>

Length mm	Diameter, mm													
	12	14	16	18	20	22	24	27	30	33	36	39	42	48
28	x	–	–	–	–	–	–	–	–	–	–	–	–	–
35.5	x	x	x	–	–	–	–	–	–	–	–	–	–	–
40	x	x	x	x	–	–	–	–	–	–	–	–	–	–
50	x	x	x	x	x	x	–	–	–	–	–	–	–	–
63	x	x	x	x	x	x	x	x	–	–	–	–	–	–
80	x	x	x	x	x	x	x	x	x	–	–	–	–	–
100	–	–	x	x	x	x	x	x	x	x	x	x	x	x
150	–	–	–	–	–	x	x	x	x	x	x	x	x	x
200	–	–	–	–	–	–	–	–	x	x	x	x	x	x
250	–	–	–	–	–	–	–	–	–	–	–	–	x	x

Note: Preferred combinations are indicated by "x"

Types of riveted joints

Types of riveted joints

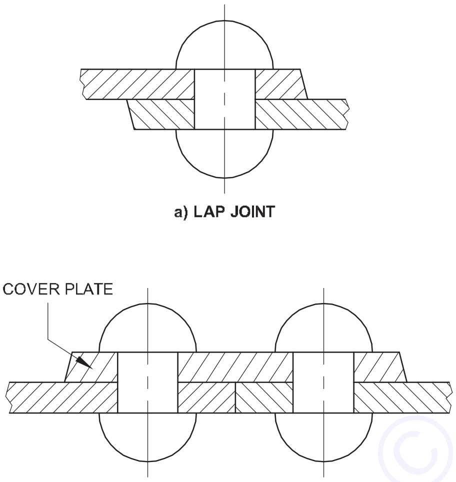
- Lap joint
- Butt joint

When the plates are placed one above the other, the joint is called Lap joint. The distance between the centres of adjoining rivets in the same row is called the pitch. (Fig 1)

In the lap joints only two plates are used placing one over the other and riveted. In butt joints, the plate ends are placed face to face, single or double cover plates are used to rivet them together.

When the number of rows are two or more, rivets are arranged in chain or zig-zag formation. (Fig 2)

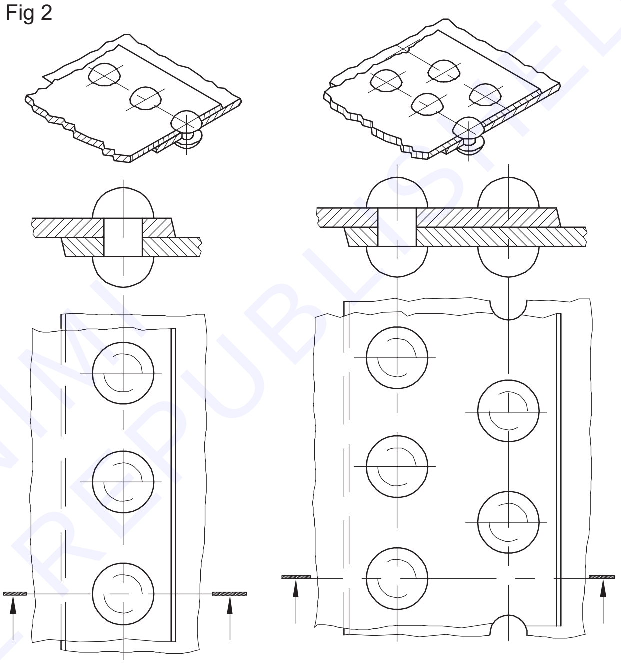
Fig 1



a) LAP JOINT

b) BUTT JOINT

Fig 2



SINGLE RIVETED LAP JOINT

DOUBLE RIVETED LAP JOINT (ZIG-ZAG)

LAP JOINTS

Types of lap joints

Single riveted lap joint: There is only one row of rivets. The plates to be joined are placed one above the other to a lap length of 3d, d being dia of the rivet. In any riveted joint, the distance between the end of the plate to the rivet hole is called `Margin' (m).

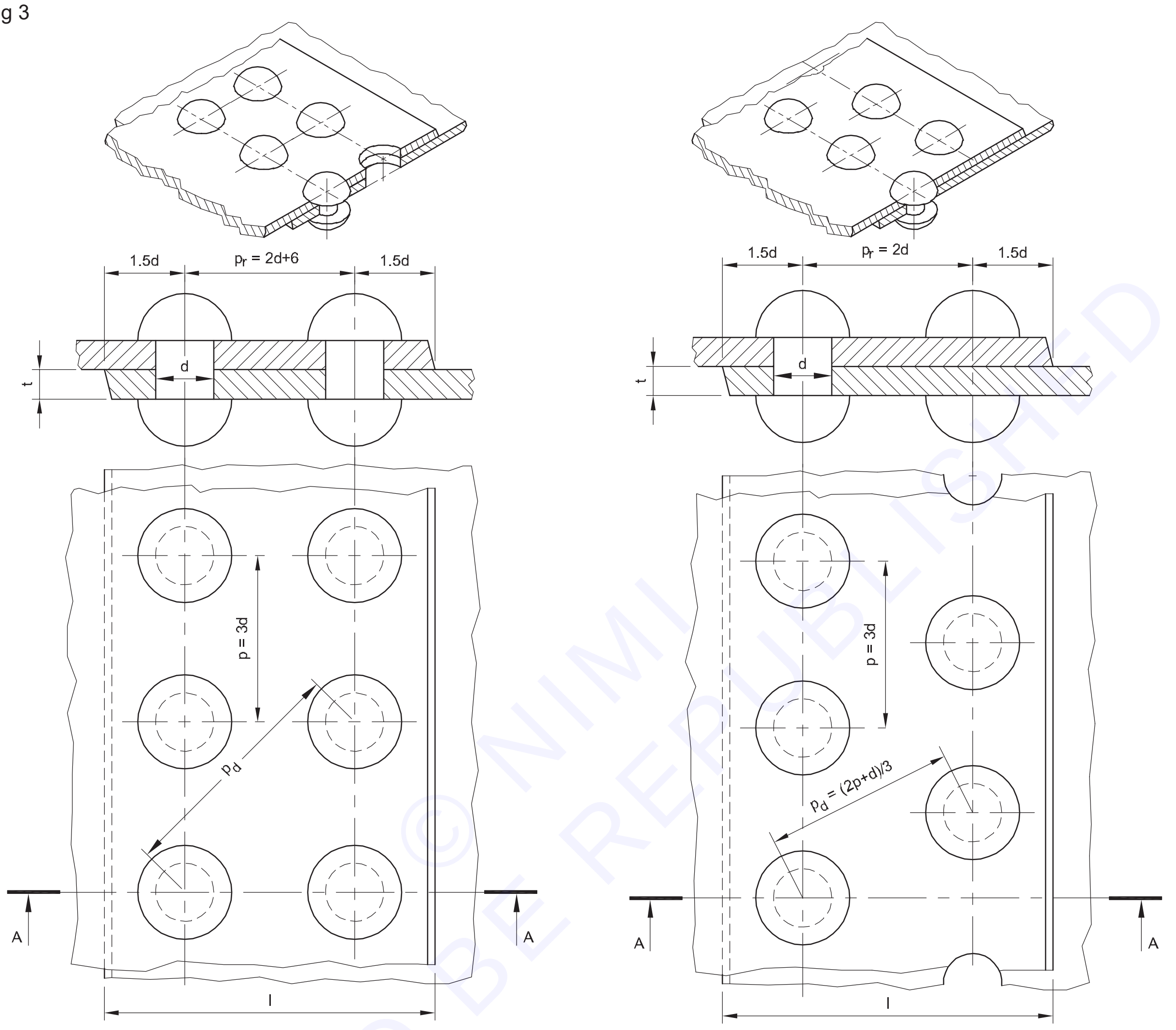
Riveted joints are named according to the number of rows of rivets, a single riveted lap joint has one row of rivets, double riveted lap joint has two rows of rivets - triple riveted joint has three rows of rivets. The rows may be chain or zig-zag. (Fig 3a & b)

Proportions of riveted joints: The diameter of rivet, pitch and margins are calculated to prevent failure of joint, forming a safe and efficient joint. For all sizes, thickness of the plates is considered first.

- Diameter of rivet d = $6\sqrt{t}$ called Unwim's rule
- Pitch (Max.) P = 3d
- Margin M = 1.5d
- Min. Pitch = d + 30 mm
- Row pitch for chain riveting (Pr) = 0.8P
- Row pitch of zig-zag (Pr) = 0.6P

- Diagonal pitch (Pd) = (2P + d) /3
- Length of rivet = thickness of plates (plate grip) + 1.25d or 1.7d

Fig 3



a) DOUBLE RIVETED (CHAIN) LAP JOINT

b) DOUBLE RIVETED (ZIG-ZAG) LAP JOINT

LAP JOINTS

Butt joints: When the plates butt against each other the joint is called butt joint. The joint may have either single cover plate or double cover plate and single riveted, double riveted and triple riveted etc. on each plate. These are generally used on longitudinal seams of cylindrical boiler shells etc. (Fig 4)

Fig 4

SINGLE-RIVETED (SINGLE STRAP) BUTT JOINT

SINGLE-RIVETED (DOUBLE STRAP) BUTT JOINT

In butt joint, the thickness of cover plates or strap with respect to plates being joined is

– single strap = 1.125t

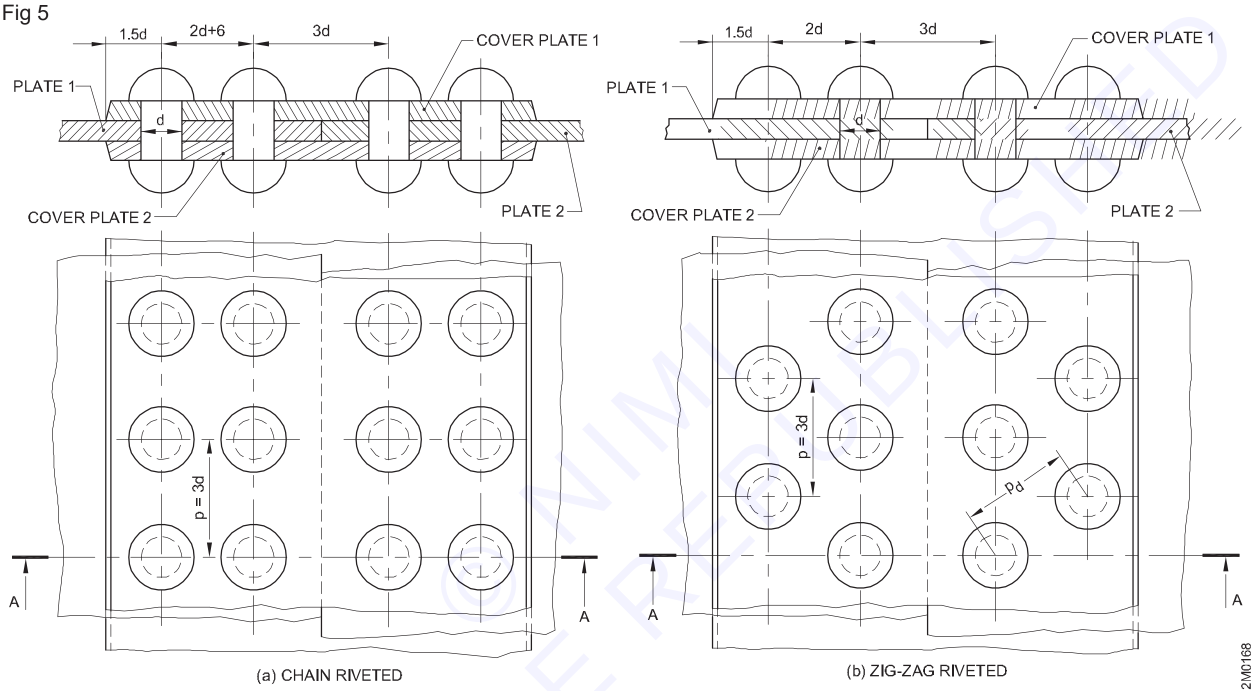
– double strap = 0.6t to t

"Chain riveting" and zig-zag riveting: When the rivets in the rows are placed directly opposite to each other it is called chain riveted. (Fig 5a) If rivets are staggered it is called zig-zag riveted. (Fig 5b)

The distance between the rows of rivets is called "Row pitch" Pr. The value of row pitch (Pr) in the case of chain riveting is 0.8 P and 0.6 P for zig-zag riveting.

The distance between the centre of one rivet in one row to the centre of the nearest rivet in the adjoining row is called diagonal pitch - Pd (Fig 5b)

Fig 5



(a) CHAIN RIVETED

(b) ZIG-ZAG RIVETED

DOUBLE RIVETED BUTT JOINTS

The figures show the double riveted butt joint (zig-zag) double strap.

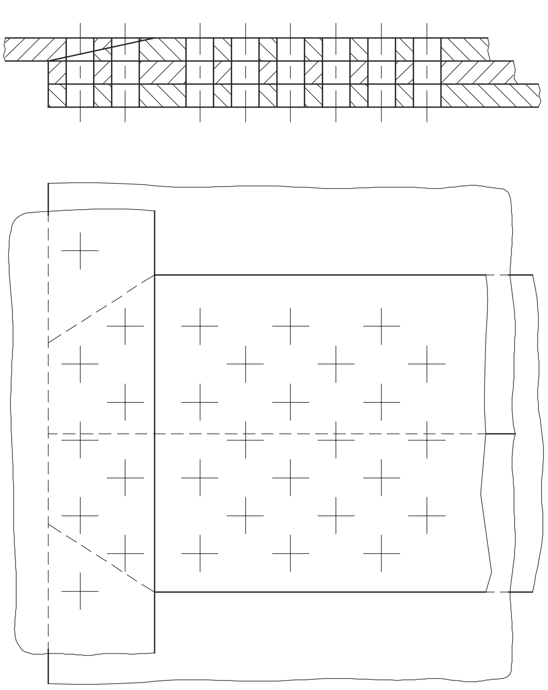
Boiler shells plates are joined by riveted joints as in Fig 6.

Chaulking and Fullering: Due to high internal pressure of steam or water in the boiler, there is the possibility of the leakage of steam or water through rivet holes or joint i.e., between plates. To prevent such leakages the rivet heads and plate ends are firmly forced together. It is called chaulking and fullering. For caulking a blunt type chisel is used by hammering the rivet head/plates. (Fig 7a)

In fullering a tool with thickness equal to that of plate is used to hammer the edges of the plate. It is used to press the plate ends. (Fig 7b)

Figures 8 show how the plates at right angles can be joined using rivets.

Fig 6



BOILER-SHELL PLATE JOINT

Fig 7

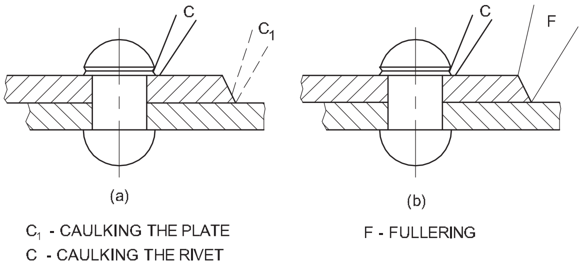
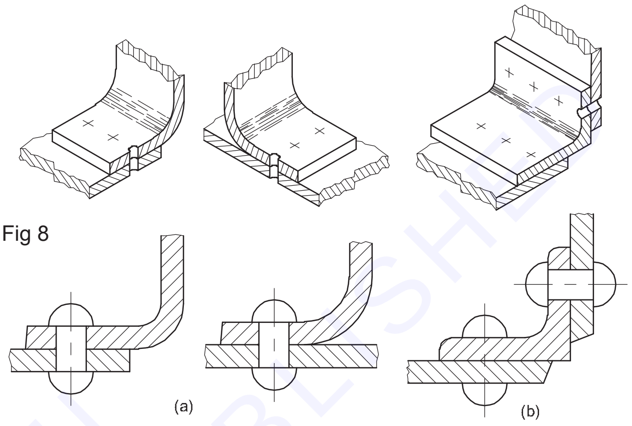


Fig 8



CONNECTION OF PLATES AT RIGHT ANGLES

Method of indicating and dimensioning of rivets

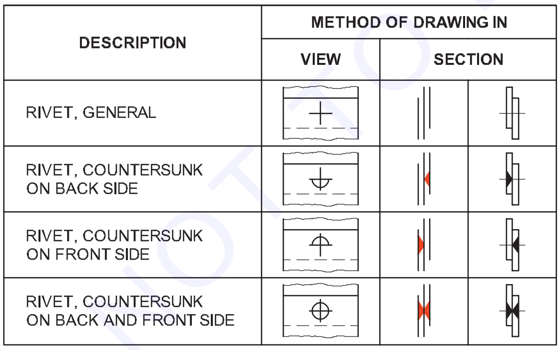
Generally rivets are indicated by their centre lines thickened in the direction of and at right angles to the seam. (Fig 1)

The Table No.1 shows the symbols for rivets.

Table 1

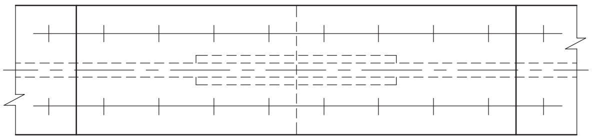
TABLE OF SYMBOLS FOR RIVETS AND BOLTS

(CLAUSES 6.3.1.1, AND 6.5.3.1)



DESCRIPTION	METHOD OF DRAWING IN		
	VIEW	SECTION	
RIVET, GENERAL			
RIVET, COUNTERSUNK ON BACK SIDE			
RIVET, COUNTERSUNK ON FRONT SIDE			
RIVET, COUNTERSUNK ON BACK AND FRONT SIDE			

Fig 1



To distinguish between bolts and rivets in a side view or section, it may be useful to add an arrow (angle about 90°) at each end of the centre line of the bolts/rivets.

In case to nuts for bolts if any, this is marked by double arrow as in Fig 2.

The centre lines for rivets/bolts shall be represented by continuous thin lines. (Fig 3)

The centre line or the neutral fibre is indicated when necessary by a long chain thin line is shown in Fig 4.

In case of special methods of dimensioning a note should be added to explain to which feature the dimension, refers For example dimension referred to centre line or dimensions referred to graduation. (Fig 5)

Fig 2

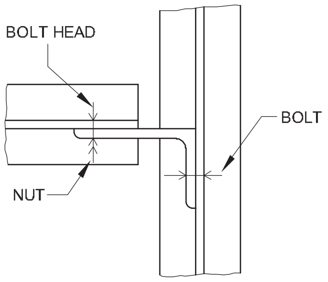


Fig 3

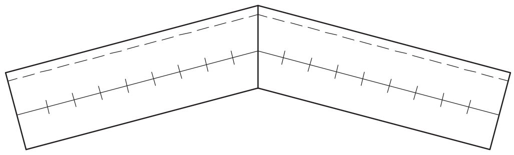


Fig 4

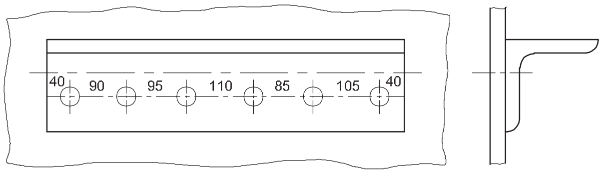
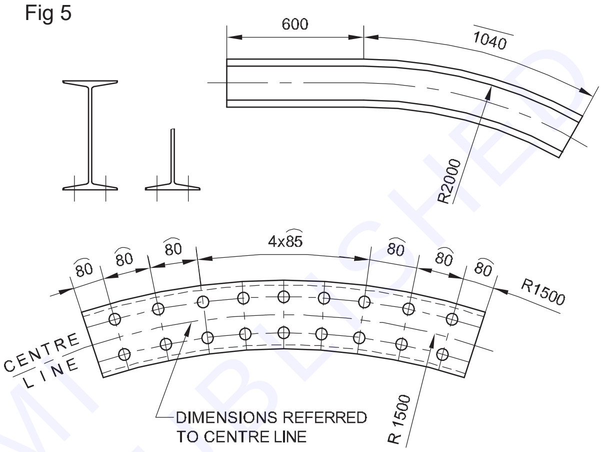


Fig 5



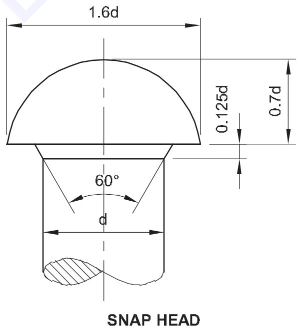
Procedure

Rivets and Riveted joints

Exercise 8.25

Snap head rivet (Fig 1)

Fig 1



Draw a snap head rivet suitable for joining plates of 25 mm thick.

Diameter of rivet = 6 x $\sqrt{25}$ = 6 x 5 = 30 mm

- Draw a thin horizontal line.
- Draw vertical centre lines.
- Mark diameter of rivet shank and head.
- Form the rivet head as per proportions.

- Draw the shank with short break.
- Draw firm lines of the edges of the rivet.
- Show the dimensions in terms of dia of shank. (Fig 1)

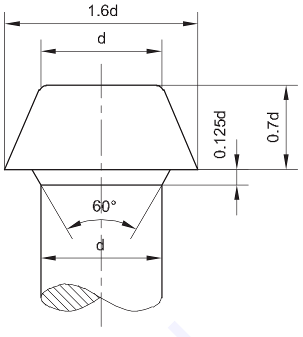
Exercise 8.26

Pan head rivet

Draw a pan head rivet of shank diameter 30 mm. (Fig 2)

- Follow the procedure of Ex.15.25

Fig 2



PAN HEAD

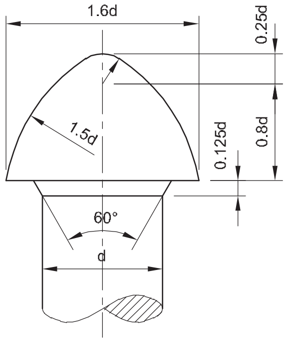
Exercise 8.27

Conical head rivet

Draw a conical head rivet of shows diameter 25 mm. (Fig 3)

- Draw a horizontal thin line.
- Draw a vertical centre line.
- Mark the diameter of rivet head and shank as per proportion.
- Set off 0.8d on the centre line as shown in Fig 3 from the horizontal line.
- 0.25d as radius, draw an arc as shown in Fig 3.
- 1.5d as radius, draw another arc, tangential to the previous arc of radius 0.25d.
- Draw the firm lines of the edges of the rivet.
- Show the dimensions in terms of diameter of shank.

Fig 3



CONICAL HEAD

Exercise 8.28

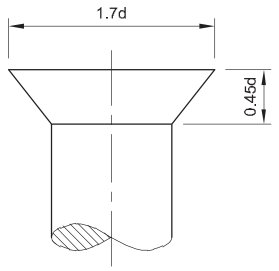
Countersunk head rivet

Draw a countersunk head rivet of shank diameter 25 mm.

- Draw a horizontal line.

- Draw a centre line.
- Follow the procedure of earlier exercisesand complete the rivet head. (Fig 4)

Fig 4



COUNTERSUNK HEAD

Exercise8.29

Single riveted lap joint

Draw a single riveted lap joint, joining 25 mm thick plates. Show minimum 3 rivets.

Calculations:

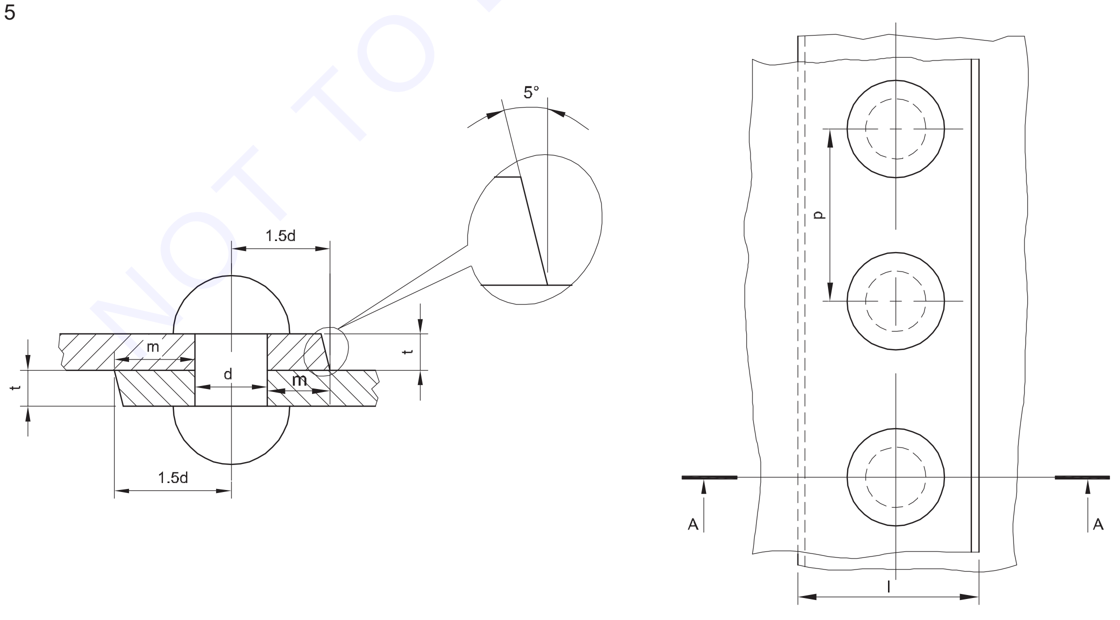
Dia of rivet = $6\sqrt{t}$ = $6\sqrt{25}$ = 6 x 5 = 30 mm

Pitch of rivet = 3d = 3 x 30 = 90 mm

Margin from rivet hole to end of plate = 1.5d = 1.5 x 30 = 45 mm.

- Mark vertical centre line.
- Draw three horizontal lines 25 mm apart in front elevation.
- Form the rivet heads (on 30 mm shank) on the centre line.
- Draw the plates and section them as shown
- Draw the rivet head circle and hidden circles for shank dia.
- Draw the edge of the plates in front view at 80° - 85°.
- Project the edges on to top view.
- Mark two pitch distances of 90 mm each.
- Draw visible edges of plates and rivets in firm lines. (Fig 5)

Fig 5



SINGLE RIVETED LAP JOINT

Exercise 8.30

Double riveted (chain) lap joint (Fig 6)

Draw a double riveted lap joint/chain riveting with 35 mm thick plates. Show a minimum of 3 rivets.

Data:

Dia of rivet = $6\sqrt{t}$ = $6\sqrt{5}$ round to = $6\sqrt{6}$ = 36 mm (approx.)

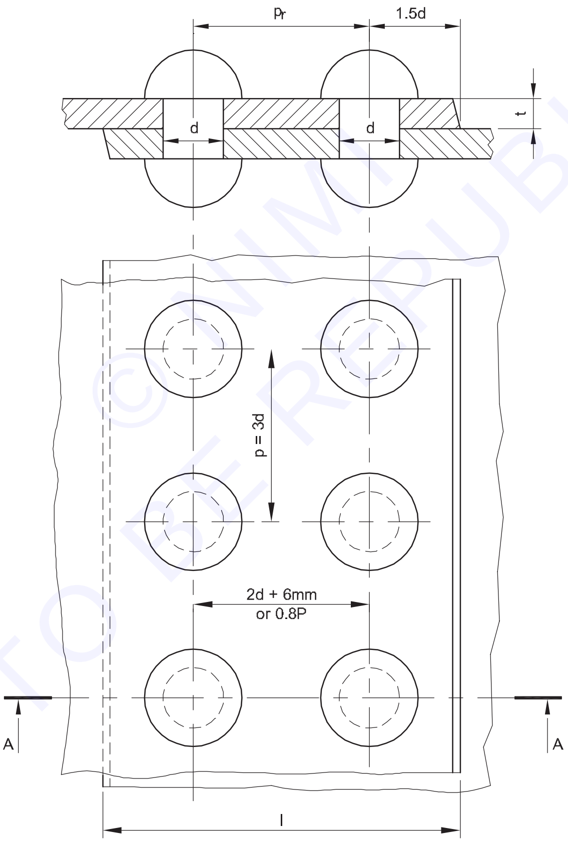
Pitch p = 3d = 3 x 36 mm = 108 mm

Row pitch (Pr) = 0.8 P or 2d + 6 mm = 86.4 mm or 78 mm

Margin = 1.5d = 1.5 x 36 mm = 54 mm

- Draw the plates one over the other, mark pitch (pr) and lap (l) 54 mm on either side.
- Draw the rivet heads, ends of the plates at lap and beyond short break.
- Hatch the plates in opposite directions.
- Project and draw the top view as in exercise 15.29.
- Show the cutting plane. Show the proportions instead of dimensions in terms of dia of rivet. (Fig 6)

Fig 6



DOUBLE RIVETED (CHAIN) LAP JOINT

Exercise 8.31

Double riveted (zig-zag) lap joint (Fig 7)

Draw a double riveted zig-zag lap joint, joining 35 mm thick plates.

Data:

Dia of rivet = $6\sqrt{t}$ = $6\sqrt{5}$ = 36 mm (approx.)

Pitch P = 3d = 108 mm

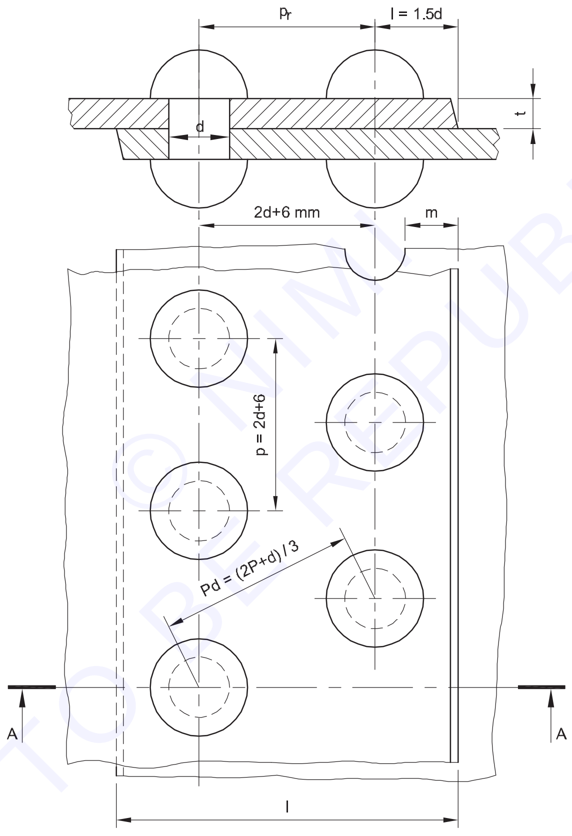
Row pitch (Pr) = 0.8 P or 2d + 6 mm = 86.4 d or 78.0 mm

Diagonal pitch (pd) = $\frac{(2p-d)}{3}$ = $\frac{252}{3}$ = 84 mm

Margin = d = 36 mm or lap (l) = 1.5d = 54 mm

- Draw the plates as in previous exercise and mark centre lines at Pr = 78 or 86 mm, draw rivet heads.
- Mark the ends of the plates.
- Project on to top view, mark three rivet centres at distance. (108 mm)
- Draw intersecting arcs with Pd as radius from centre.
- Draw rivet head circles and shank as hidden circles.
- Draw the edges of the plates as shown
- Show proportions of plate thickness. (Fig 7)

Fig 7



DOUBLE RIVETED (ZIG-ZAG) LAP JOINT

Exercise 8.32

Single riveted (Single strap) butt joint (Fig 8)

Draw a single riveted butt joint with single cover plate. Two plates of 36 mm thick butting each other are to be joined by riveting.

Calculations:

Given plate thickness t = 36 mm

Dia of rivet = $6\sqrt{t}$ = 36 mm

Pitch of rivets P = 3d = 108 mm

Lap l = 1.5d = 54 mm

for butt joints

Single cover plate thickness

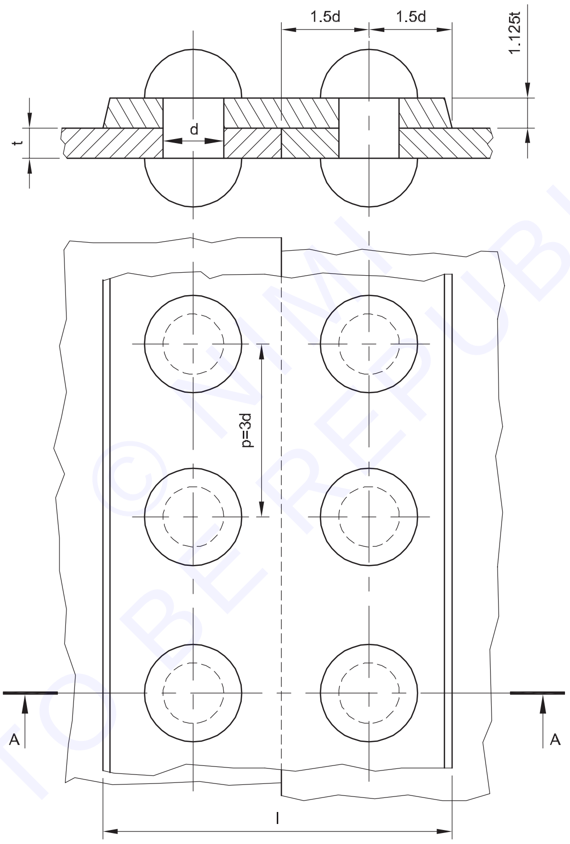
1.125 t = say 40 mm

Pitch of rivets = 3d = 108 mm

Row pitch for chain riveting = 2d + 6 = 78

- Draw the plates in position with single cover plate.
- Mark the centre of rivets and draw rivets.
- Hatch the plates.
- Project and draw the top view showing minimum 3 rivets in a row.
- Show the broken end of the plates.
- Dimension the views. (Fig 8)

Fig 8



SINGLE-RIVETED (SINGLE STRAP) BUTT JOINT

Exercise 8.33

Single riveted (Double strap) butt joint (Fig 9)

Draw single riveted double strap butt joint (chain). Plate thickness 25 mm.

Calculate the rivet dia, pitch, distance etc.

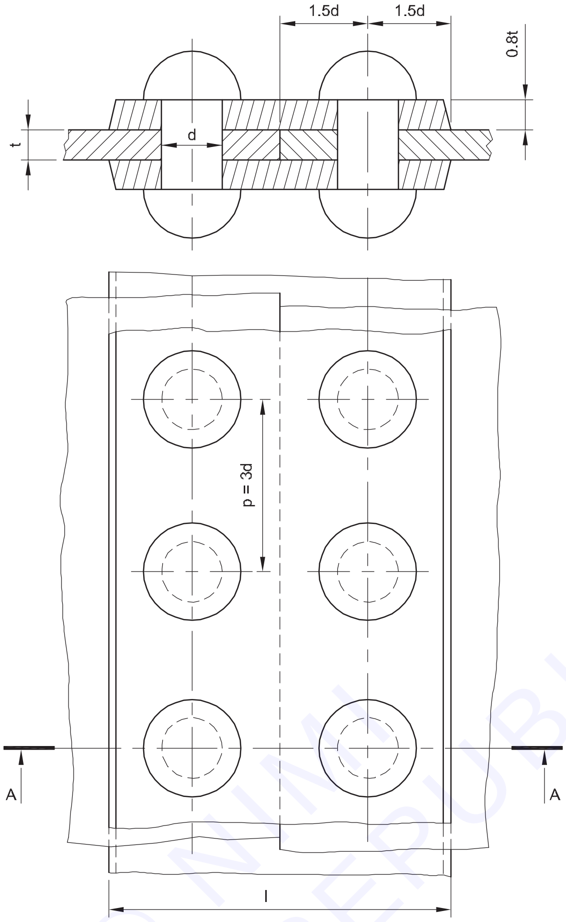
Double cover plate thickness (0.6 t to t)

Average = 0.8t

0.8 t = 20 mm

- Draw the main plates and cover plates in position.
- Mark two rows of rivets with distance as shown in Fig 9.

Fig 9



SINGLE-RIVETED (DOUBLE STRAP) BUTT JOINT

- Draw the rivets and hatch the plates.
- Project and draw the top view with minimum 2 pitches.
- Show the end cover plates and main plates.
- Show dimensions in terms of dia (d). (Fig 9)

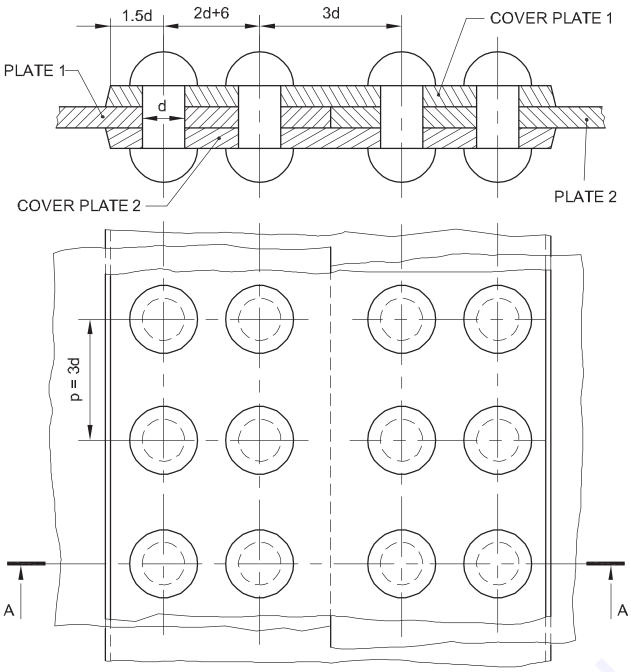
Exercise 8.34

Double riveted butt joint (chain) (Fig 10)

Draw a double riveted butt joint to join 25 mm thick plates, double cover plates chain riveting.

- Calculate the rivet diameter, row pitch, pitch, margin etc as in previous exercise.
- Draw the plates with double cover plates (10d + 12 mm) long.
- Mark the centres of rivets as shown in the Fig 10.
- Draw the rivet head and the plate ends.
- Project and draw the plates showing thickness and rivets.
- Hatch the plates
- Show dimensions/proportions. (Fig 10)

Fig 10



CHAIN RIVETED / DOUBLE RIVETED BUTT JOINTS

EDC22M0182

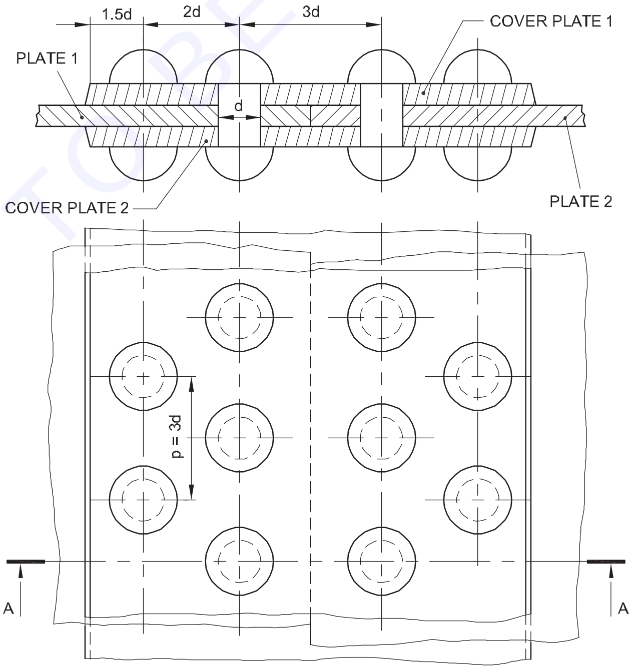
Exercise 8.35

Double riveted butt joint (zig-zag)

Draw a double riveted butt joint to join 25 mm thick plates, double cover plates by zig-zag riveting. (Fig 11)

- Calculate the rivet diameter, row pitch, pitch, diagonal pitch margin etc as done in previous exercises.
- Follow the same procedure of earlier joints and complete the drawing.

Fig 11



ZIG-ZAG RIVETED / DOUBLE RIVETED BUTT JOINTS

EDC22M0183

Exercise 8.36

Riveted joint with connection plates at right angles

Draw the riveted joints connecting plates at right angles with a & b bent plate c angle iron, plate thickness 25.

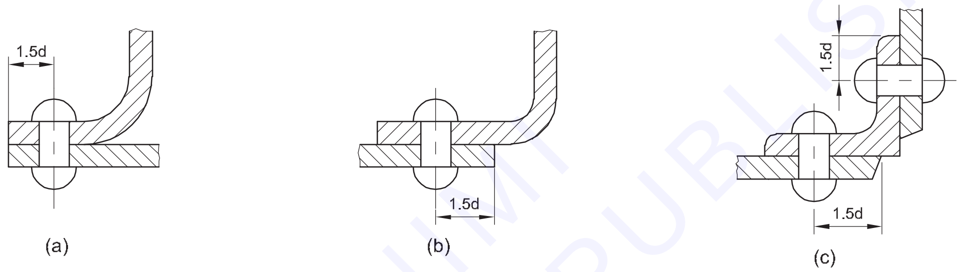
Using bent plate

- Draw the plate bent at radius not less than 2t.
- Mark the centre of the rivet 1.5d from centre of arc.
- Draw the other plate to be riveted and rivet (two methods are shown)
- Show proportions of Fig 12a and b.

Using angle plate

- Mark the centre of rivet holes at 1.5d + t from the corner of the angle plate.
- Draw the plates, draw the rivets and hatch.
- Show proportions. (Fig 12c)

Fig 12



CONNECTION OF PLATES AT RIGHT ANGLES

Keys

Key is a metallic piece of wedge inserted between a shaft and hub, parallel to the axis of shaft. It is proportionate to the shaft dia.

The key prevents relative motion between a shaft and hub of the mating parts. The key sits in a groove in the shaft called **"key seat"** and **"key way"** in the hub both parallel to the shaft axis. Key seat on the shaft is normally produced by milling a key way in the hub by slotting. Fig 1 a & b shows key, key seat, key way, hub and assembly.

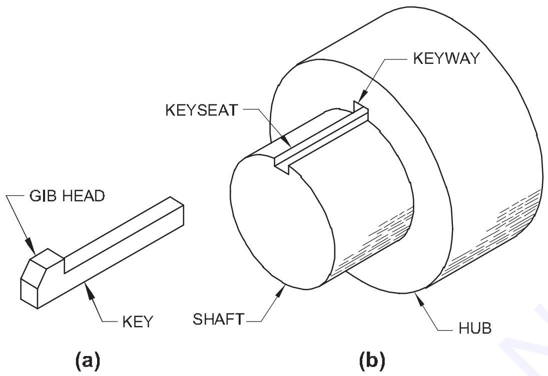
Keys are classified according to their relative position of the shaft and the cross-section.

According to their position with reference to shaft, there are two types:

- sunk keys
- saddle keys

Sunk keys (Fig 2)**:** They partly fit in the shaft (sunk) and hub.

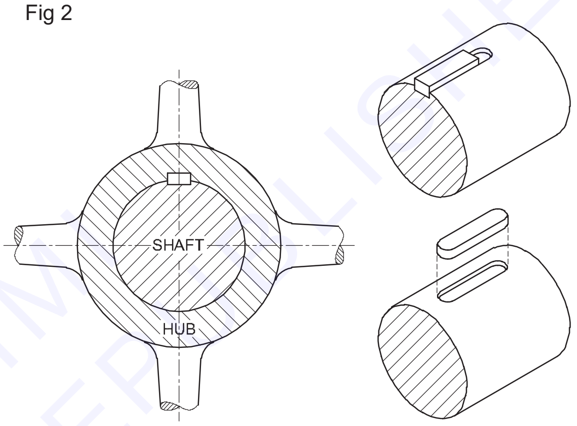
Fig 1



(a) (b)

SUNK KEYS - HEAVY DUTY KEYS

Fig 2



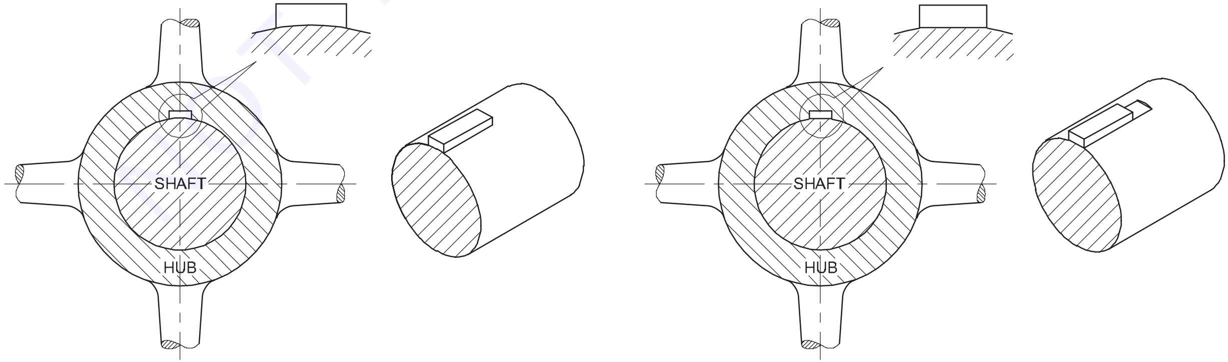
SUNK KEYS - LIGHT DUTY KEYS

Saddle keys (Fig 3)**:** Like saddle on the horse back, they rest on the shaft and get fitted in the hub. Only due to friction the drive takes place, so they are meant for light duties only. Saddle keys are two types: hollow and flat.

If the dia of the shaft is d width of key b = 0.25d + 2 mm nominal thickness (max) h = 1/3b Taper = 1:100 (facing the hub).

Parallel keys: these keys have uniform thickness, also called as "Feather keys". While preventing realtive rotation they permit sliding or axial movement of the hub (matching part) on the shaft.

Fig 3

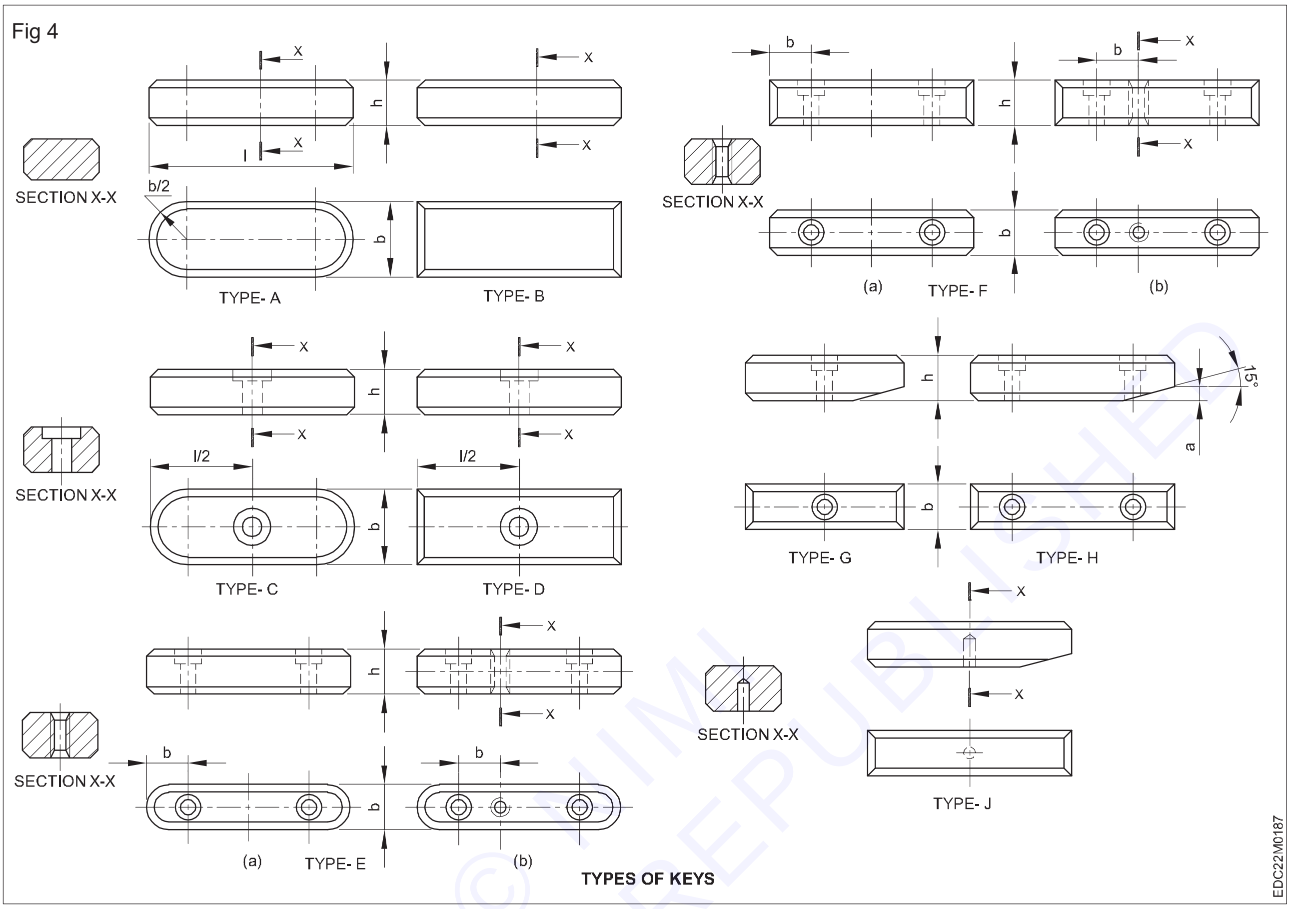


SADDLE KEYS

IS:2048 lays down standards for Parallel keys or feather keys and key ways (Fig 4)

There are nine types of parallel key as follows:

Fig 4



TYPES OF KEYS

Type A - keys with both ends round.

Type B - keys with both ends square.

Type C - keys with both ends round with one hole for retaining screw for sizes 8 x 7 and upwards .

Type D - keys with both ends square with hole for one retaining screw for sizes 8 x 7 and upwards.

Type E - (i) keys with both ends round with holes for two retaining screws for sizes 8x 7 and 10 x 8.

(ii) additionally provided with tapped hole for one or two jacking screws for sizes 12 x 8 and upwards.

Type F - (i) keys with both ends square with holes for two retaining screws.

(ii) additionally provided with tapped hole for one or two jacking screws for sizes 12 x 8 and upwards.

Type G - keys with both ends square with chamfer and hole for one retaining screws.

Type H - keys with both ends square with holes for two retaining screws.

Type J - keys with both ends square and hole for one spring dowel sleeves.

The keys except type A and type B are screwed to the shaft with the help of retaining screws viz. slotted cheese head screw (IS:1366) or hexagon socket head cap screw (IS: 2269). Sometimes spring dowel sleeves (IS:5988) are made use of. The cross section of key varies from 2 x 2 mm to 100 x 50 mm length 6 to 400mm depending upon the dia of the shaft. Jack screw helps in removing key from key seat.

For design purposes, the key sizes are based on shaft dia on which they are fitted.

Let the dia of the shaft = d, then the width of key b = 0.25d + 2 mm thickness of key h = 0.66 b.

Tolerance on keys and key ways

Key width (b) h9 for square key, h11 for rectangular key

Key way (b) D10

Thickness (h) h9 for square section, h11 for rectangular section.

Key has chamfer S x 45° where S varies from 0.16 to 3.00mm and key way has radius varying from 0.08 to 2.5mm according to the size of the shaft. Depth of key slot 0.6h (approximately on shaft) 0.4h (approximately on hole)

Material for keys: Steel of tensile strength of 600 MPa (60MN/m^2: 60 kgf/mm^2 approx.)

Designation of keys: Keys are designated by their Type, width, height and length followed by IS numbers.

Parallel key type A width 70 mm height 36 mm, length 250 mm is designated as parallel key A 70x36x250 IS:2048.

Fixing of key (Fig 5)**:** As the feather keys are parallel, they are likely to get loosened from their position. Except Type A and B, other keys are fastened to the shaft by means of retaining screws and spring dowel sleeves. The retaining screws are either slotted cheese head screw (IS:1366) or hexagon socket head cap screws (IS:2269). Spring dowel sleeves are as per IS:5938. Fig 5 shows how the retaining screws and spring dowels are used. Width of keys vary from 2 mm to 100 mm and height or thickness varies from 2 mm to 50 mm according to shaft diameter from 6 mm to 500 mm.

Taper keys: they are rectangular in cross section. They have a taper of 1:100 on the thickness. Taper on the key helps to hold the hub and shaft together firmly.

The key seat on the shaft is parallel to the axis i.e. uniform depth whereas key way in the hub is tapered to 1:100 to suit the key.

There are three types of taper keys:

Type A - Keys with both ends round.

Type B - Keys with both ends square.

Type C - keys with one end round and other end square.

Fig.6 shows the above keys.

All the edges are chamfered at S x 45° where S varies from 0.16 mm to 1.20 mm according to the width of the key (b).

Keys for shafts upto 22 mm dia. are of square section, keys for shafts above 22 mm dia are of rectangular section. Approximate proportion of sunk keys of rectangular cross section for a shaft of diameter 'd'.

Width of the key (b) = 0.25d + 2 mm (¼ d + 2 mm)

Thickness of the key (h) = 0.66 b or (2/3b)

Taper 1 in 100 (1:100)

Length = d to 3d (approx.) maximum upto 140 mm.

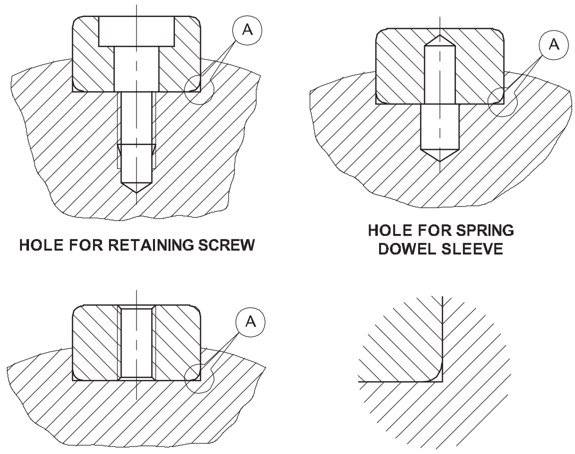
IS: 2292 lays down the specifications for taper keys and key ways. Width and thickness vary from 2 x 2 to 100 x 50 mm according to the shaft dia ranging from 6mm to 500 mm. Length varies from 6 mm min. to 400 mm max.

A taper key of type B, width 32 mm, height 18 mm, length 100 mm is designated as Taper key B32 x 18 x 100 IS: 2292.

Tolerance on key width 'b' is h9 and for thickness h, h9 for square section and h11 for rectangle section. Tolerance on key way width is D10.

Gib-Head Keys: It is a taper key with a square head chamfered 30° at the thick end of the key. When the key is not accessible to remove it, a head is provided called gib. A wedge is placed between the gib and hub, by giving a jerk to the wedge, the key is removed from assembly.

Fig 5



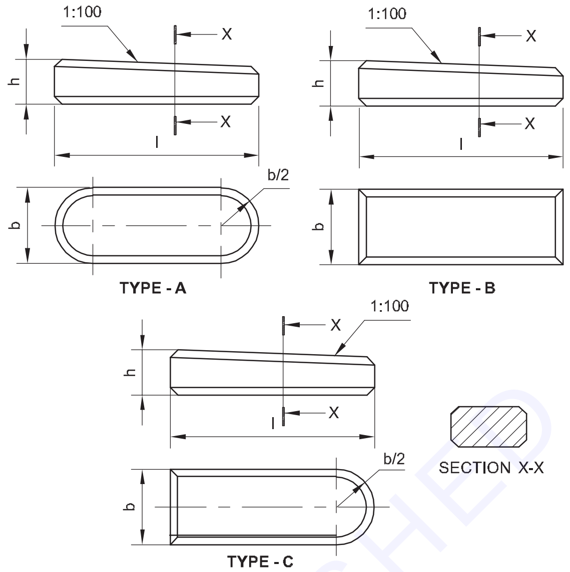
HOLE FOR RETAINING SCREW

HOLE FOR SPRING DOWEL SLEEVE

HOLE FOR JACKING SCREW

DETAIL - A

Fig 6



TYPE - A

TYPE - B

TYPE - C

SUNK KEYS - TAPER KEYS

Fig 7 shows the approximate proportions of the key. All the edges along the length are chamfered.

Width of key b = 0.25 d + 2 mm

Thickness h = 0.66 d

Taper = 1:100

Thickness of head h_1 = 1.75 h

Length of head = 1.5 h

Chamfer of head = 30° chamfer along the length 45°

Gib head keys are of square section upto shaft dia 22 and for dia 22 and above are of rectangular cross section. IS:2293 lays down the specification of gib head keys and keyways for shaft dia 6 mm to 500 mm.

Thin taper keys: these are similar to taper keys discussed except these keys are comparatively longer IS:6166 lays down the specification for thin keys and key ways. These are used on dia 22 to dia 230 mm. Key sizes vary from 8 x 5 to 50 x 18 mm according to diameter of shaft

The fit between the key way and key is "D10/h9" providing clearance fit.

Note: Tolerance, limits and fits will be dealt later.

Woodruff keys (IS:2294): It is in the shape of a segment of a circular disc. It gets aligned in the key seat of the shaft. Figs 8 & 9. The flat projected part of the key fits into the key way formed on the hub. This key is intended for light duty drives only. There are two series of wood ruft keys series 1 for torque application (transmission of power) series 2 for position application.

For a shaft of dia 'd' (24)

Thickness of key b = d/6
Radius of key R = 2b or d/3
projection of key above shaft = T/2
height of the key h = 1.75b
Tolerance on width = h9
Tolerance on height h1 = h11
Tolerance on D = h12

Max. 40 mm dia of shafts on which the woodruff key is used; max. width key is 10 mm.

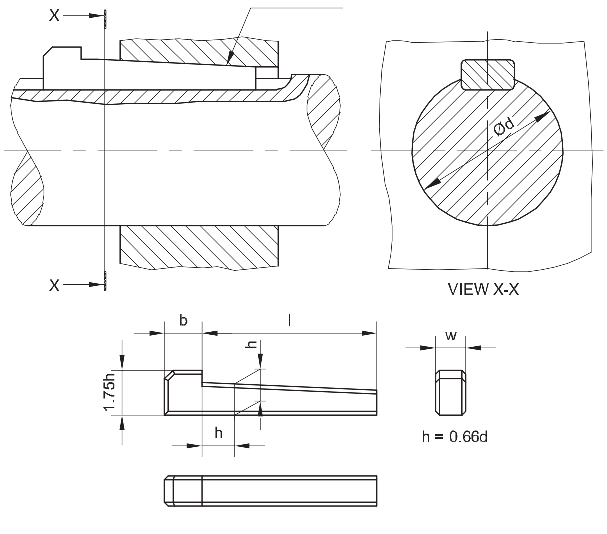
Wood ruff keys are designated by its width and its height followed by BIS no.IS:2294

Example

Woodruff key of width 8 mm

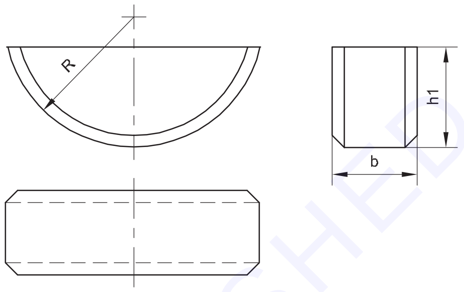
Woodruff key 8.0 x 11.0 IS.2294; dia. shall be 28 mm.

Fig 7



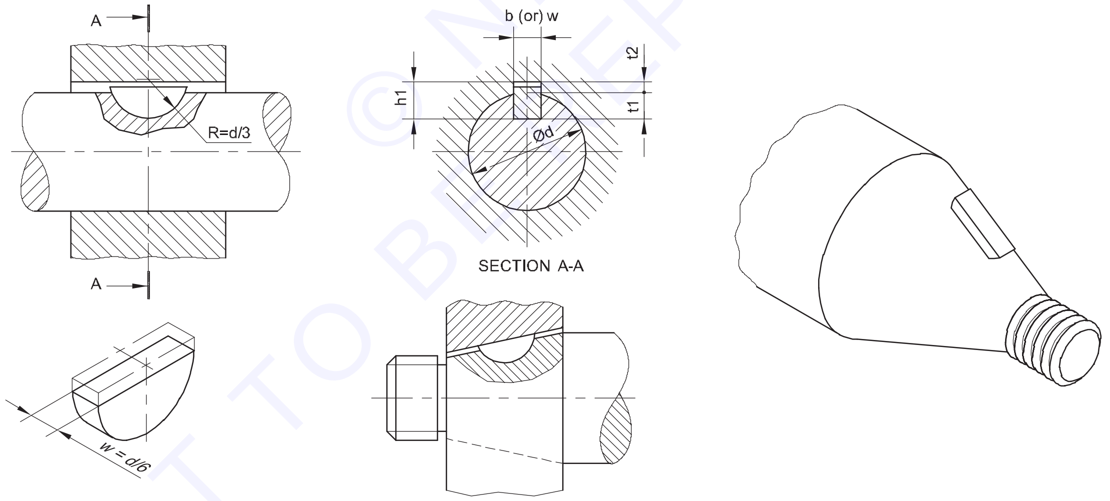
GIB HEAD KEY

Fig 8



WOOD RUFF KEY

Fig 9



WOOD RUFF KEY FOR TAPERED SHAFT

Special keys and keys not as per BIS

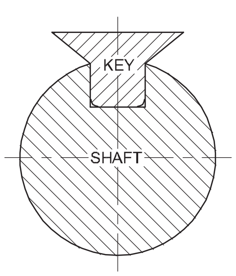
There are some keys in use though not as per BIS viz

- Dove tail key
- Peg key
- Single head key with screw
- Double head key.

Dove tail key: The rectangular part of the key is fitted into the shaft and dove tail part into the hub. They are driven into their position by using soft hammers. (Fig 1)

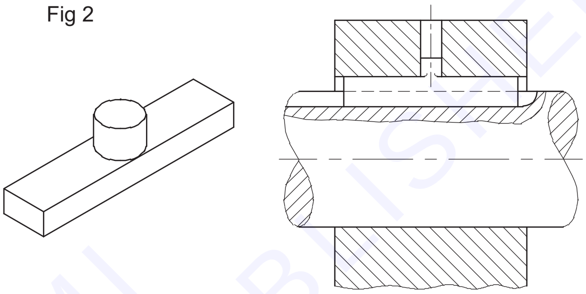
Peg key: Hub which takes the peg key has a suitable hole in the slot. The key is first mounted on to the hub hole. The shaft is then aligned to the hub. Key is comparatively longer than those used with taper keys. (Fig 2)

Fig 1



DOVE TAIL KEY

Fig 2



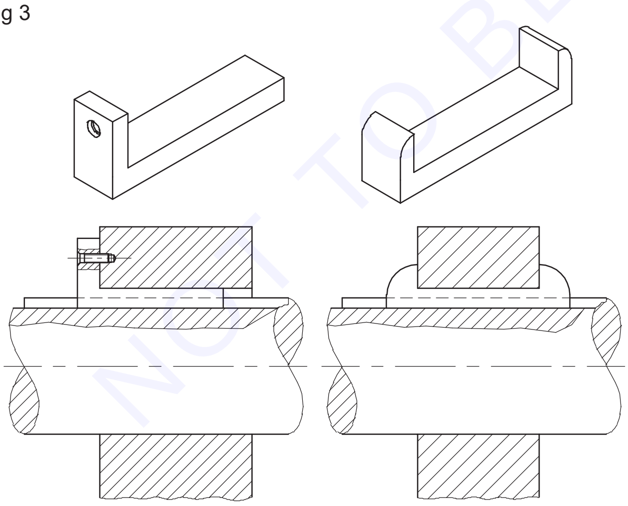
PEG KEY

Single headed key: After driving into the key way, it is fastened to the hub with a counter sunk screw. (Fig 3a)

Key seat on the shaft to take double headed key is from one end of the shaft. The key is first fitted to the hub and then the key seat of the shaft is aligned and fixed. (Fig 3b) The key, in all these cases cannot get loosened from its position of the hub. It allows the hub to slide if required along the axis to some extent.

Cone keys: These are segments of a truncated cone with cylindrical hole, on the axis, when they are mounted on a shaft, they can take up tapered hollow hub of different dia meters or shafts of varying diameter. (Fig 4)

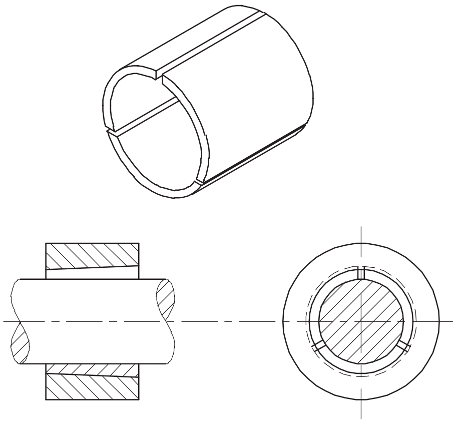
Fig 3



(a) SINGLE HEADED KEY (b) DOUBLE HEADED KEY

FEATHER KEYS

Fig 4



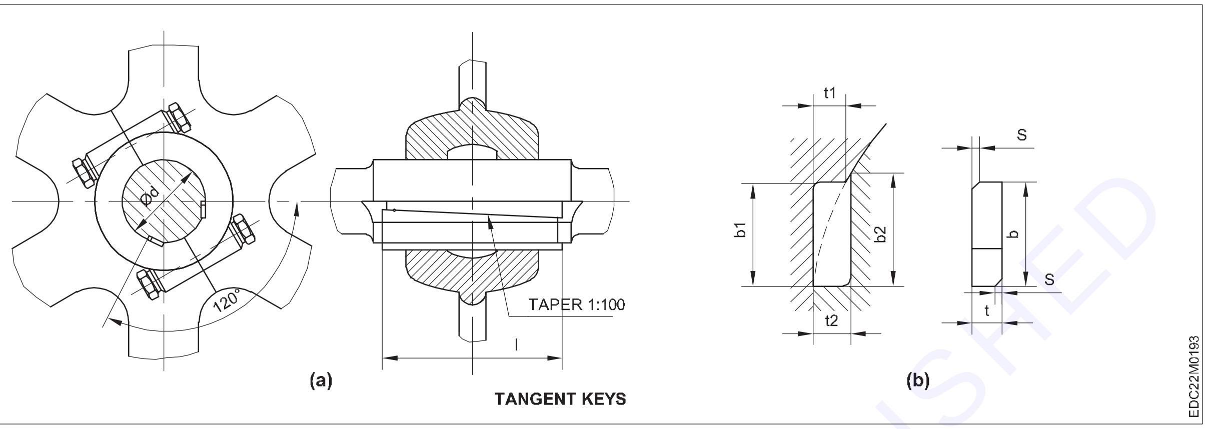
CONE KEYS

Tangential keys: two sets of pair of taper keys are used tangential to the shaft. They are used on shaft for dia 60 mm above upto 500 mm (IS:2291)

The keys are placed with their taper face to face as in Fig 5a. They are driven into, the width across the parallel faces increase. Fig 5b shows how the tangential keys are used on reverse direction. They are used on shafts where loads on reverse direction also are to be taken up.

Splined shafts and splined hubs: These are like integral multi keys on their periphery. The faces of each spline are parallel and align with each other and can slide over the shaft. (Figs 6 & 7) They got greater strength with regard to key and keyways IS:2327.

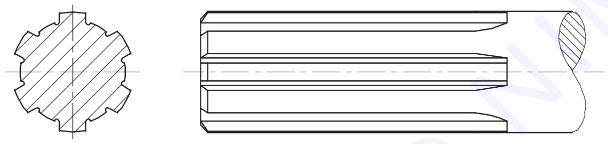
Fig 5



(a) (b)

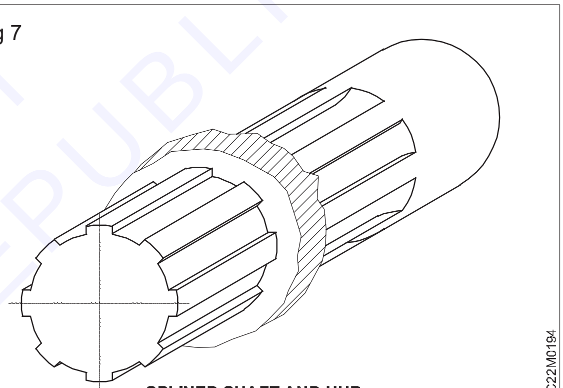
TANGENT KEYS

Fig 6



SPLINED SHAFT WITH SIX SPLINES

Fig 7



SPLINED SHAFT AND HUB

Splined shafts and hubs are used extensively in the automobile, machine tool and other industries.

Splined shafts are generally used:

- for coupling shafts when relatively heavy torques are to be transmitted without slip;
- for transmitting power by sliding or fixed gears, pulleys and other rotating members and
- for attaching parts that may require removal for indexing or change in angular position

The straight sided splines have been divided into two series, light duty and medium duty.

The inside diameter has been kept the same for both the series. It will be larger than the bearing diameter by 1 to 2 mm, so that a suitable fitting dimension may be obtained.

Spline: It is the machine element consisting of integral keys (spline teeth) or keyways (spaces) equi-spaced around a circle or portion thereof.

Internal spline is a spline formed on the inner surface of a cylinder.

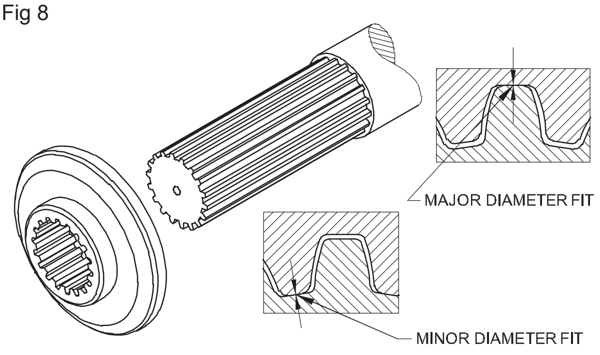
External spline is the spine formed on the outer surface of a cylinder.

Straight sided spline or splineway: A spline or splineway whose sides (or flanks) are normal to a tangent touching the centre of the arc as its major diameter. The sides are parallel to each other throughout their depth and length and are known as straight to distinguish them from those having flanks of involute or other curved form. (Fig 8 & 9)

The dimensions of the straight sided splined shafts and given in the Table 1.

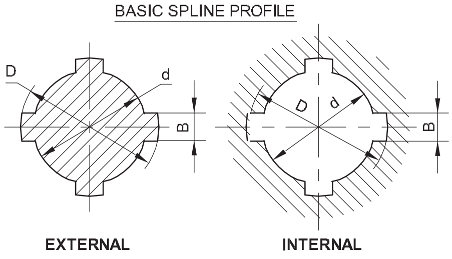
The designation of straight sided splines for general use shall consist of the nominal size (as given in the first column of table) and the number of this Indian standard. For example, a 6-splined shaft or hub with minor diameter of 28 mm and major diameter of 32 mm shall be designated as;

Fig 8



INVOLUTE FORM

Fig 9



STRAIGHT SIDED SPLINES FOR MACHINE TOOLS-4 SPLINES

TABLE - 1

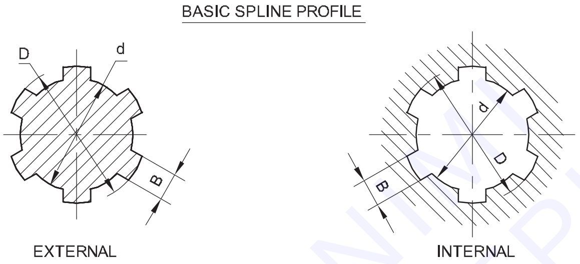
DESIGNATION N x d x D	B
4 x 16 x 20	6
4 x 18 x 22	
4 x 21 x 25	8
4 x 24 x 28	
4 x 28 x 32	10
4 x 32 x 38	
4 x 36 x 42	12
4 x 42 x 48	
4 x 46 x 52	14
4 x 52 x 60	
4 x 56 x 65	16
4 x 62 x 70	
4 x 68 x 78	

EDC22M0195

Spline 6 x 28 x 32 IS:2327 (Fig 10)

The splines may be straight sides or involuted sided (IS:2327 and IS:3665). The dimensions for straight sided splines for machine tools are given in IS:2610. Designated no. of splines x min. dia x max.dia 1.5 (IS:2327).

Fig 10



STRAIGHT SIDED SPLINES FOR MACHINE TOOLS-6 SPLINES

DESIGNATION N x d x D	B
6 x 21 x 25	5
6 x 23 x 28	6
6 x 26 x 32	
6 x 28 x 34	7
6 x 32 x 38	8
6 x 36 x 42	
6 x 42 x 48	10
6 x 46 x 52	12
6 x 52 x 60	14
6 x 58 x 65	
6 x 62 x 70	16
6 x 68 x 78	
6 x 72 x 82	
6 x 90 x 90	
6 x 82 x 95	
6 x 88 x 100	

EDC22M0196

Circlips

Circlips are fastening devices used to provide shoulders for positioning or limiting the movement of parts in an assembly. (Fig 1) Circlips are also called “retaining rings” IS: 3075 part 1, 2, and 3)

The rings are generally made of materials having good spring properties so that the fastener may be deformed elastically to a considerable degree and still spring back to its original shape. This permits the circlips to spring back into a groove or other recess in a part or they may be seated on a part in a deformed condition, so that they grip the part by functional means. Circlips are manufactured from spring steel with high tensile and yield strength.

Types: There are two types.

Internal circlips (Fig 2)**:** This type of rings are assembled in holes, bores or housing.

Circlips for bores (IS:3075 - Part II): These are available for bores of nominal diameter range 8 to 300 mm in normal type and 20 to 100 mm in heavy type.

These are made from spring steel of grade 70 C6 or 75 C6.

External circlip (Fig 3)**:** This type of circlips are installed on shafts, pins,studs and similar parts.

Circlips for shafts Normal type (N) and heavy type (H).

These are available for shafts dia 3 mm to 300 mm.

Heavy type: These are suitable for shafts dia 15 mm to 100 mm.

Fig 1

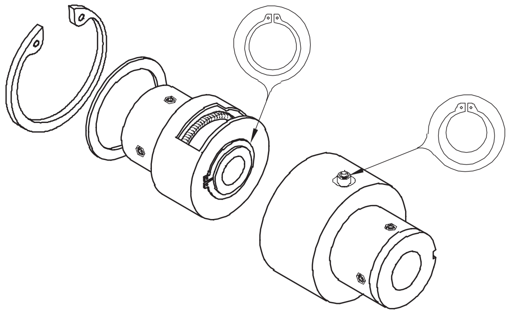
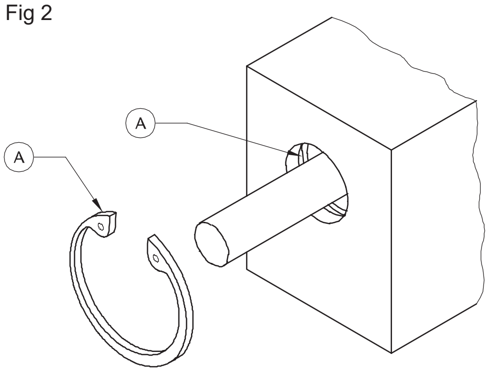
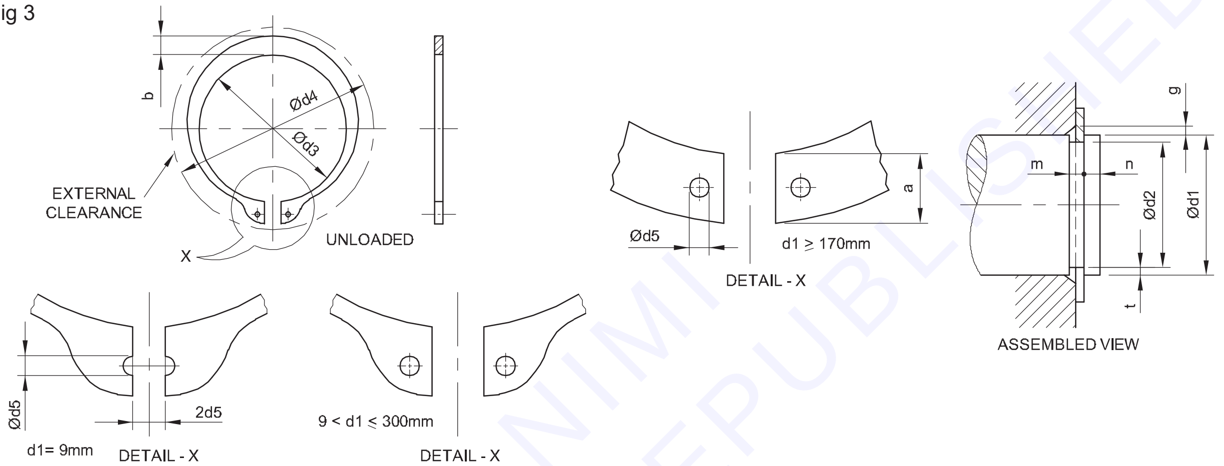


Fig 2



INTERNAL CIRCLIP

Fig 3



CIRCLIP FOR SHAFTS

Both types offer a number of advantages over other fastening methods.

- Their cost is relatively low when compared with other types of fasteners.
- Their use often results in savings in raw materials and simplified machining operations for other parts in the assembly.
- One circlip often can replace two or more parts.
- Assembly toolings developed for circlips usually permit very rapid assembly of the fasteners, even by unskilled workers.

Material: Circlips are manufactured from spring steel with high tensile and yield strength.

Designation : A circle for shaft dia (normal)d_1 = 50 mm and thickness (s) = 2 mm Normal type shall be designated as

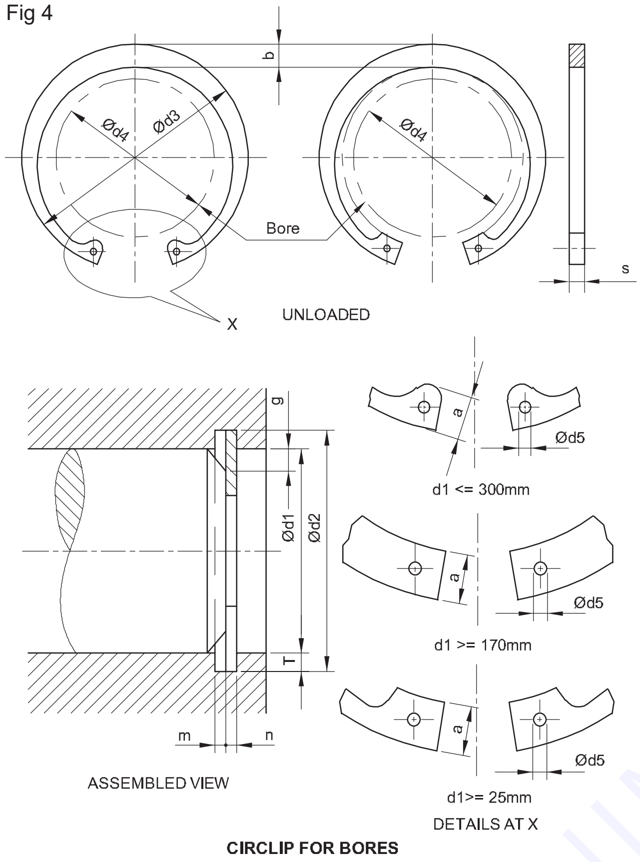
Circlip 50 x 2 N IS:3075 (Part I)

For heavy duty circlip 50 x 3 H (IS:3075 - Part I) (Fig 4)

Circlip for bore nominal dia d_1 normal type circlip 50 x 2 N IS:3075 (Part II)

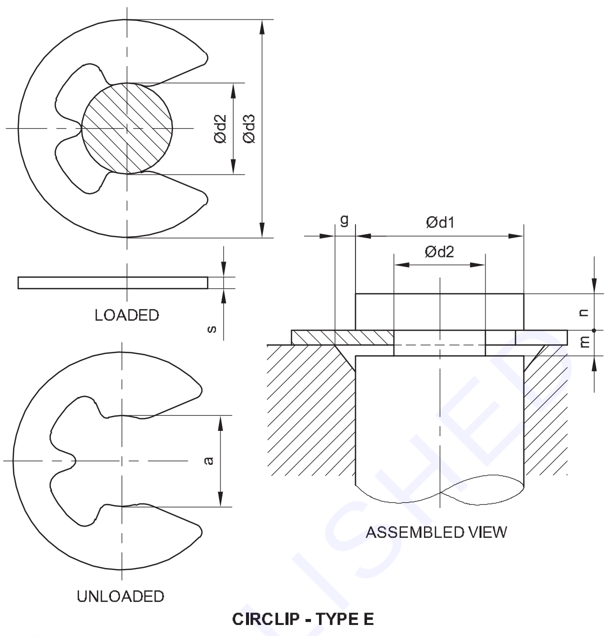
For Heavy duty circlip 50 x 2.5 H (IS:3075 - Part II) (Fig 5)

Fig 4



CIRCLIP FOR BORES

Fig 5



CIRCLIP - TYPE E

Collars, taper pins and cotters

Set collars are used in general engineering for locking and for locating bearing and other running parts in the axial direction to obtain the inter- changeability of the collars with shafts. Set collars are specified in IS:2995 - 1965 suitable for shafts dia 2 mm to 200 mm.

Set collars light series are of two types.

Type A used on shafts with slotted set screw IS:2388 for fixing on to shafts. Single set screw is used for shafts dia 2 mm to 70 mm bore collar. Two slotted grub screws at 135° are used on shafts and collar bores 72 to 200 mm dia for fixing on the shaft. The grub screws M2 x 3 to M20 x 2 are used according to shaft diameter. There are 28 preferred sizes and 35 non-preferred sizes.

Type B set collars use taper pins (IS:2393 - 1963), for bores of collars dia 4.5 to 150 mm (53 sizes). A taper pin (IS:6688-1981) dia 1.5 x 12 mm to dia 60 x 220 mm long are used according to the collar board. (Figs 1 & 2) Set collars are made of Fe 410.

Taper pins (Fig 3)**:** Taper pins like round keys are used for locking collars on shafts and also between shaft and hub for transmission of motion. Taper is 1:50, small end as ref. nominal dia. Its ends are spherical and radius equal to dia of the pin.

Taper pins are three types:

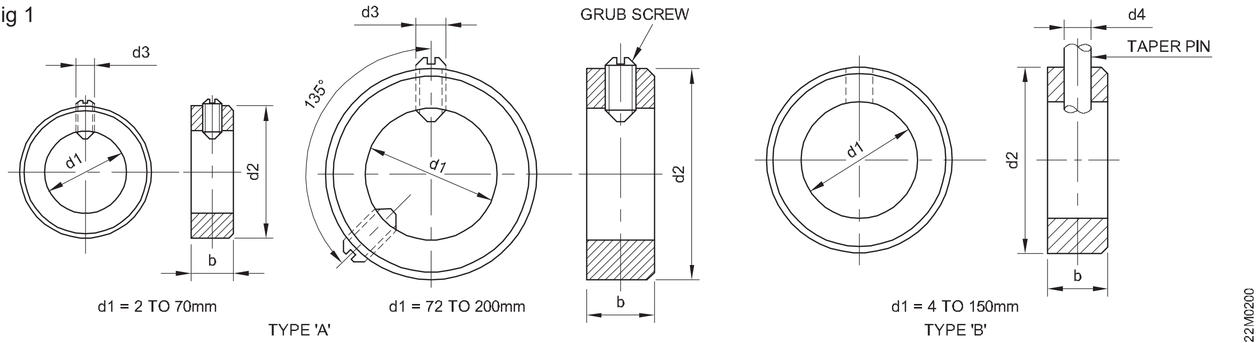
Type A - pins ground with a surface finish N6

Type B - pins turned with a surface finish N7

Type C - split pins with a surface finish N7

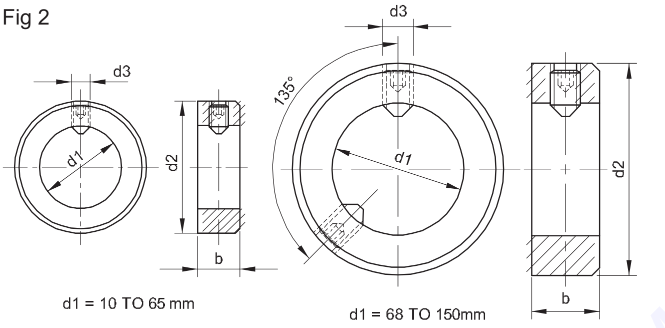
The nominal dia range from 0.6 to 50 mm and of varying lengths 4 to 200 mm according to dia of pin.

Fig 1



LIGHT SERIES SET COLLARS

Fig 2



HEAVY SERIES SET COLLARS

Fig 3

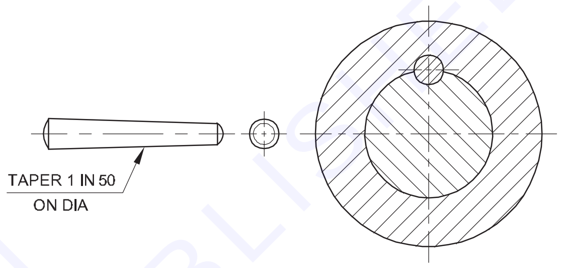
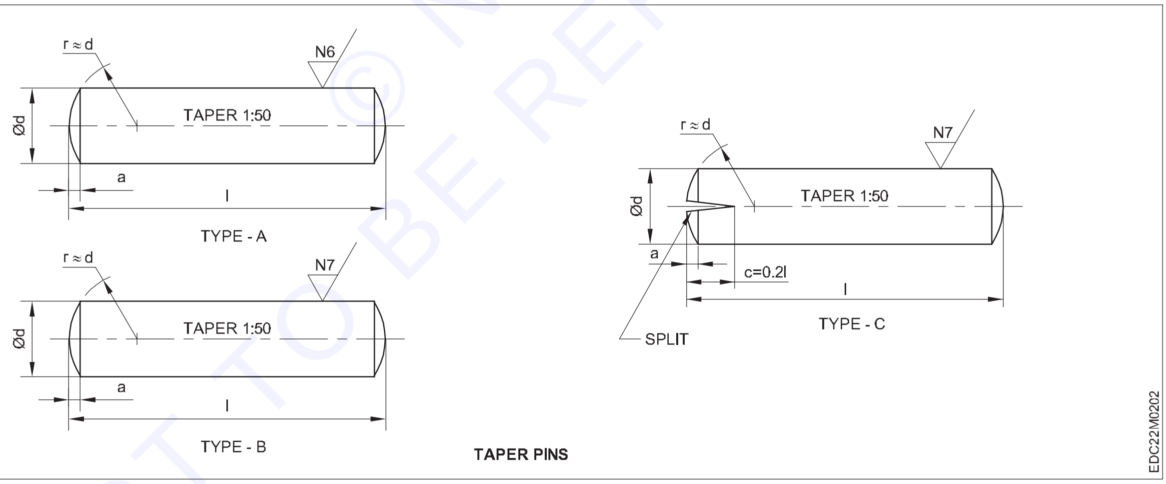


Fig 4 shows three types of taper pins.

Fig 4



TAPER PINS

Designation: Taper pin shall be designated by name, type A.B or C, nominal dia, nominal length and BIS number.

Taper pin A 16 x 90 IS:6688

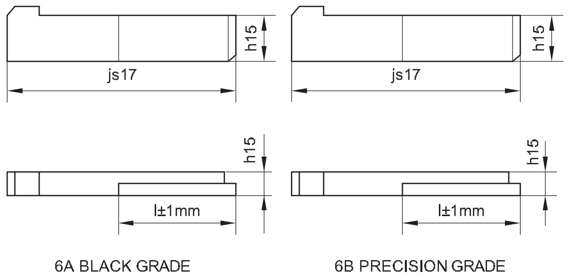
Taper pin B 20 x 60 IS:6688

Split taper pin C 5 x 40 IS:6688

General proportion: normal dia of pin = 1/6 (dia of shaft).

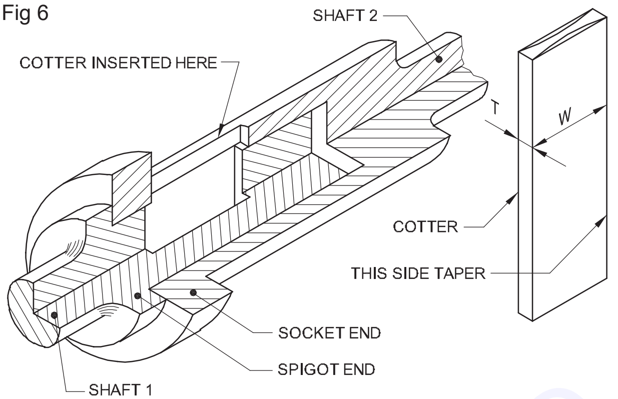
Cotter/cotter joint: Cotter is a rectangular wedge with taper on one side of the width, thickness being same. It is used to connect shafts, with reciprocating motion only. The ends of the shafts to be joined are formed into socket and spigot. A rectangular slot at right angle to the axis is made with taper on one side to suit the cotter. The socket and spigot are aligned and the cotter is driven in locking them together. (Figs 5 & 6)

Fig 5



FLAT SPLIT COTTERS

Fig 6



COTTER & COTTER JOINT

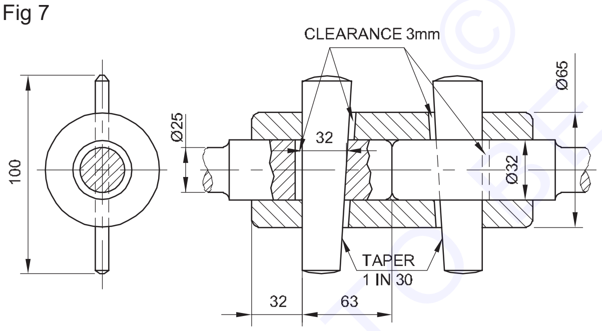
EDC22M0203

Two cotters are used to join shafts with a sleeve. The enlarged shaft ends with slots are placed facing each other in a sleeve with slots. (Fig 7) On driving the cotters, with a bearing surface on the sleeve, the tapered or slope surface of the cotters pull the shafts closer. The clearance on the sleeve and shafts allow the variation of cotters width to certain extent.

Cotter joint: Is also used to connect square or rectangular members. Fig 8 shows a strap joint with a gib and cotter. One end of the member is made as fork end which takes the end of the other member to prevent the fork end getting bend while driving the cotter a gib is placed. Fig 9 shows the bending effect on the fork end and how the gibs are made use of. Single gib is used for cotter with slope on one side. Two gibs are used if the cotter has slope on both sides.

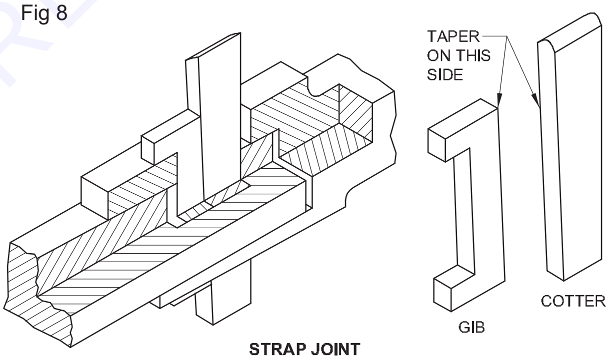
Use of pin in connecting shafts: Similar to the cotter, cylindrical pin is also used in connecting shafts. One end of the shaft is made as Fork (fork end) with holes and the end of the other shaft is formed as eye end. (Fig 10) The eye end fits into the fork end, holes being in one line. A collared cylindrical pin with a small hole is inserted into the eye and fork. The pin is held in position using a coller and a small taper pin or split pin.

Fig 7



COTTER JOINT WITH SLEEVE

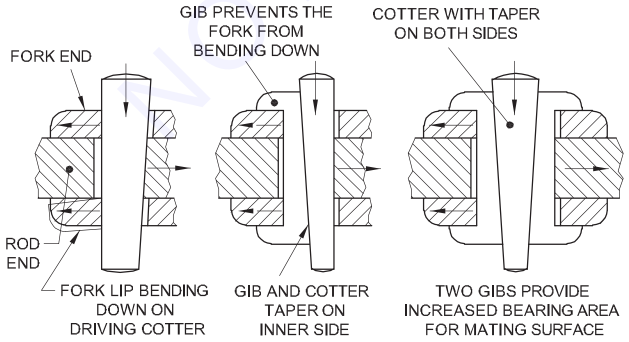
Fig 8



STRAP JOINT

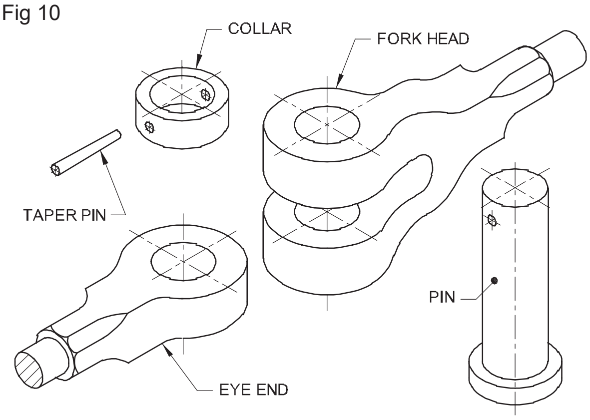
EDC22M0204

Fig 9



COTTER JOINT WITH SLEEVE

Fig 10



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Fasteners

Exercise 8.37

Parallel keys

Draw the parallel keys as per IS:2048 to shaft of diameter 180 mm using general formula and as per standard. (Fig 1A, C, D, F & G)

- Type A - Both ends round R = 22.5 mm.
- Type C - Similar to type A but with counterbore dia 14 mm, counterbore 208.3 deep.
- Type D - Both ends square with counterbore similar to type C.
- Type F- Both ends square with hole for two retaining screws M12.
- Type G - Similar to type D but with a chamfer at 15^0.

General formula

Diameter of shaft (d) 180 mm.

Width of key 0.25 d + 2 mm

Thickness of key = 0.33 W (1/3W)

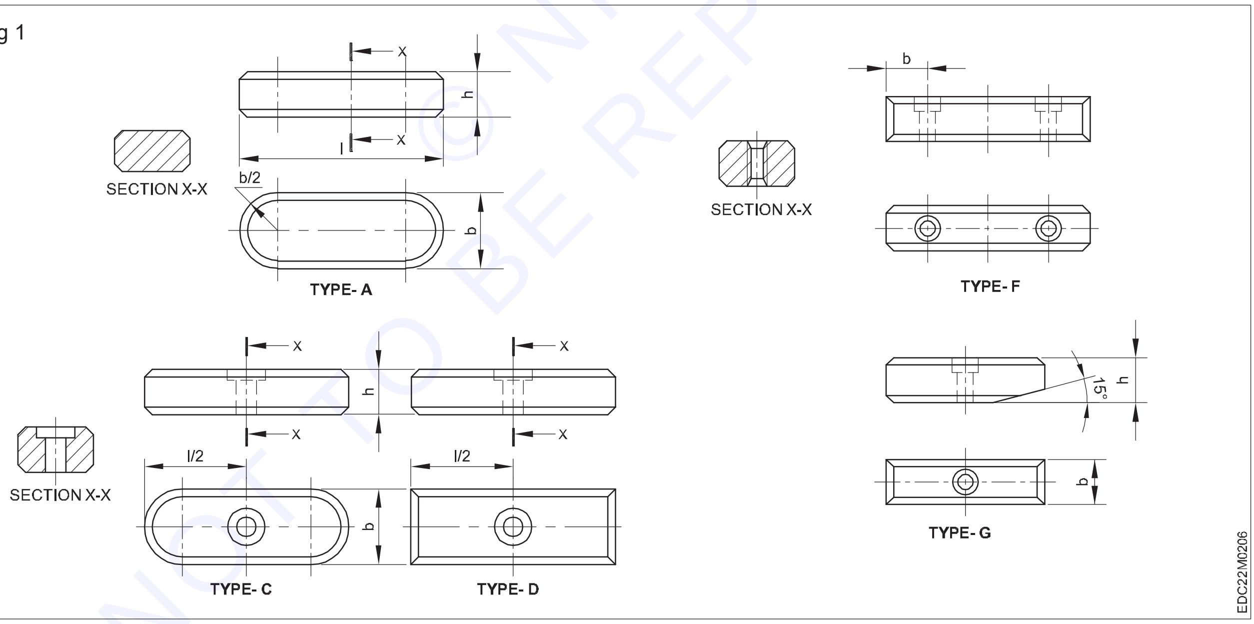
Length of key = 2d

Width = 0.25 x 180 + 2 47 mm

Thickness = 1/3 W = 47/3 = 16 mm

Length 2d = 360 mm

Fig 1



Exercise 8.38

Taper keys

Draw the taper keys type A to C as per IS: 2292 to suit the shaft diameter 100 mm. (Fig 2A,B & C)

General formula/sizes

Taper on taper key = 1:100

Length of key approx. 2d

Width (W) = 0.25d + 2 mm

Thickness = 2/3 x W

Length range = 80 to 360 mm

Length = 2 x 100 mm = 200 mm

Width = 28 mm

Thicknes = 16 mm

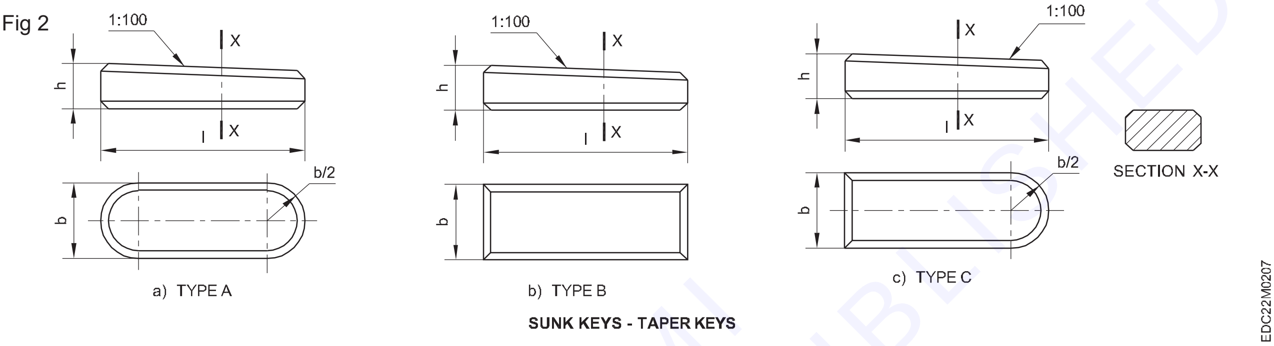
Chamfer S = 0.6 to 0.8 mm

Type A - both ends round

Type B - both ends square

Type C - one end square and one end round

Fig 2



a) TYPE A

b) TYPE B

c) TYPE C

SUNK KEYS - TAPER KEYS

Exercise 8.39

Draw the assembly of parallel keys on shaft of diameter 100 mm using general formula. (Fig 3)

Parallel key Type A to suit shaft dia 180 mm

General formula for all keys = dia of shaft d

width of key = 0.25 d + 2 mm

Thickness of key = 0.33 W (1/3 W)

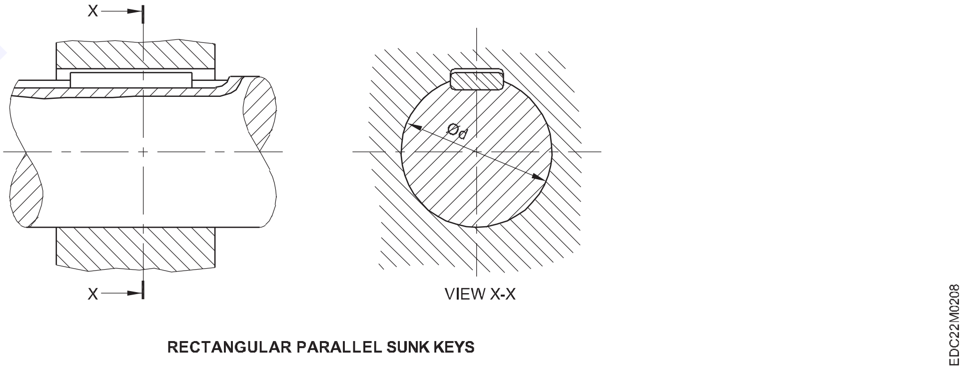
Length of key = 2d

Width = 0.25 x 100 + 2 = 27 mm

Thickness = 1/3 W = 27/3 = 9 mm

Length 2d = 200 mm.

Fig 3



RECTANGULAR PARALLEL SUNK KEYS

Exercise 8.40

Draw the saddle keys - hollow and flat in isometric view and also the assembly. (Fig 4)

Diameter of shaft = 50 mm

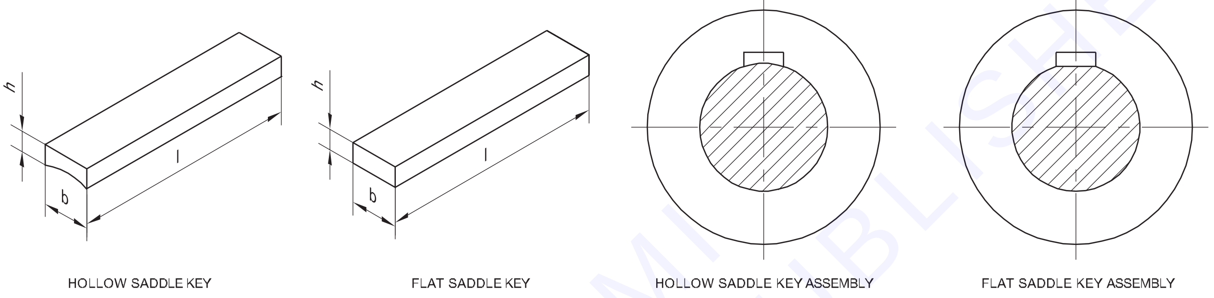
Width (b) = 0.25 d = 12.5 mm

Thickness (h) = 0.08d = 4 mm

Length: approx. 2d = 100 mm

- Hollow saddle key
- Flat saddle key
- Flat saddle key assembly
- Flat saddle key assembly

Fig 4



SADDLE KEYS

Exercise 8.41

Woodruff keys

Draw the woodruff key and also its assembly for shaft diameter 50 mm.(Fig 5 & 6)

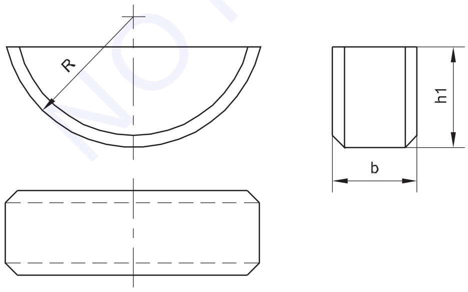
Thickness (b) = d/6 = 8 mm

Radius (R) = 2b = 16 mm

Max. height of key (h) = 2b - 0.25b = 1.75b = 14 mm

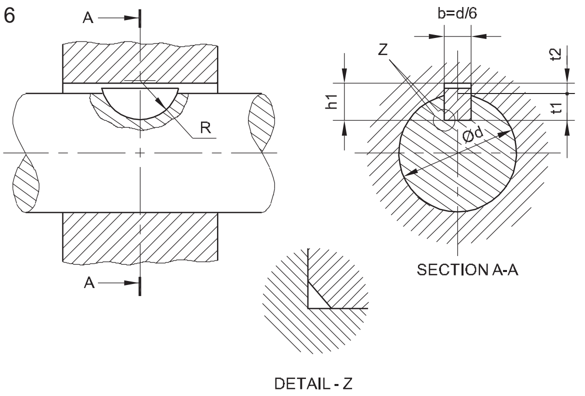
- Wood ruff key
- Assembly

Fig 5



WOOD RUFF KEY

Fig 6



Exercise 8.42

Draw the assembly of taper keys as per standard (IS:2292) to suit shaft dia 100 mm. (Fig 7)

Shaft dia d = 100 mm

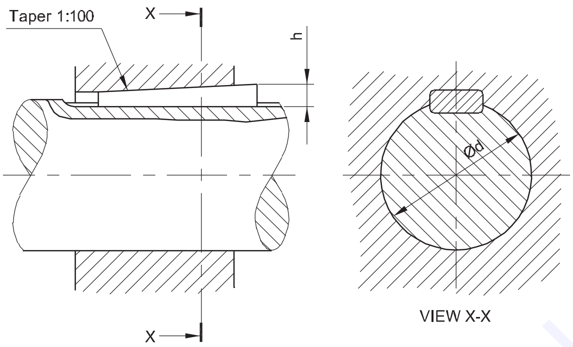
Breadth (width) = 25 mm h9

Limitation width (limits) = 0;-0.062

Height (thickness) h = 16 mm

Limits on thickness 030.130

Fig 7



Exercise 8.43

Draw an assembly of shaft and hub connected with a gib head key. Assume shaft dia d = 100 mm. (Fig 8)

d = 100 mm; as per formula

Width of key b = 0.25d + 2 mm

100 x 0.25 + 2 = 25 + 2 = 27 mm

Thickness (nominal) h = 0.66b or 2/3 b = 27 x 0.66 = 18 mm

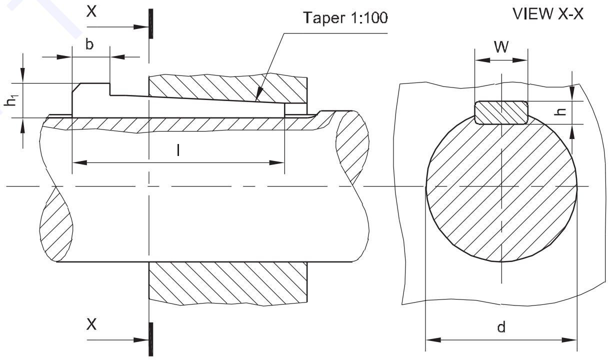
Length of key = 150 or 120 mm min. (say 1.5d)

Difference in thickness = 150 x 1/100 = 1.5 mm

Gib height = 1.75 h = 18 x 7/4 = 31.5 say 32 mm

Width of gib = 1.5h = 27 mm

Fig 8



Exercise 8.44

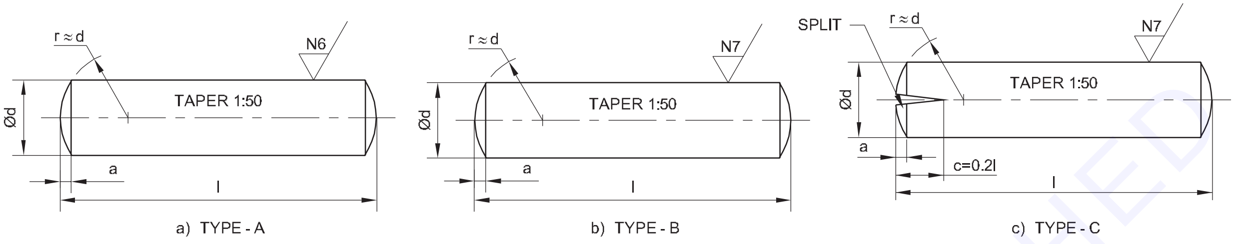
Taper pins

Draw taper pin 12 x 60 - Type A & B. Split taper pin C 20 x 80 IS:6688.

Type A & B

- Draw the cylinder of dia 12 mm, 60 long.
- Draw taper of 1:50 over the length 60 mm for type A mark surface finish N6 and N7 for type B.
- Draw the pin as above.
- Draw the split along the centre line.
- Mark surface finish as N7. (Fig 9)

Fig 9



a) TYPE - A b) TYPE - B c) TYPE - C

TAPER PINS

EDC2230213

Exercise 8.45

Details of cotter and sleeve joint is given in work book sheet. Draw the details as per instructions given in the sheet. For assembly follow the procedure given below.

- Draw the enlarged shafts butting each other.
- Draw the sleeve over it.
- Mark the cotter holes from the end of the sleeve.
- Draw the cotter with mean width on the centre line of the shaft.
- Mark the taper end and the length.
- Mark the ends.
- Draw the visible lines after section.
- Give the required dimensions.
- Project for plan and side and complete the drawing.
- Prepare bill of material list.

Exercise 8.46

Details of gib and cotter joint for square bar is given in the work book. Draw the details as per the note given in the work book sheet and follow the steps given for assembly.

- Draw the fork as per the dimension.
- Locate the strip in the fork.
- Insert the cotter and align the gib on taper sides of the cotter.
- Note that both the tapers of Cotter & Gib are merged with each other.
- Draw the section and complete the drawing.
- Draw plan and complete.

Exercise 8.47

Details of Socket and Spigot joint is given. Prepare the drawing as per instructions given in work book sheet. Follow the steps given below for assembly drawing.

- Draw a horizontal centre line.

- Draw the socket as per dimensions.
- Insert/Draw the spigot and locate with the hole of the socket.
- Insert the cotter keeping the straight edge of the cotter with the straight end of the socket.
- Follow the procedures of section and complete the view.
- Draw plan and side view.
- Prepare bill of material list.

Exercise 8.48

The details of the knuckle joint is given in the work book. Draw the details as given with complete dimensions. For assembly follow the procedure given below.

- Draw the fork end and eye end in line aligning with the centre line.
- Insert the cylindrical pin through the hole in them.
- Draw the collar in position and insert the taper pin.
- Follow the other procedures for section and complete the sectional view.
- Project the plan.
- Prepare the bill of material list.

Pipes and pipe fitting

Pipes are hollow cylinders of uniform wall thickness. Pipes are used with their fittings to carry water, oil, chemical fluids, steam gas etc.

Selection of pipe material is based on the following factors.

- Cost of production/connection
- Type of joint required
- Pressure and temperature of fluid
- Velocity of fluid
- Physical and chemical properties etc.

Following metals and materials are used for making pipes.

- Galvanised iron
- Cast iron
- Cement
- Copper
- Poly Vinyle chloride (PVC)
- Poly Proplene (PP) etc.

Pipe connections are made to different lengths with branches and bends using pipe fitting accessories according to the requirements. (Fig 10)

There are different pipe fitting accessories available for different purpose. They are

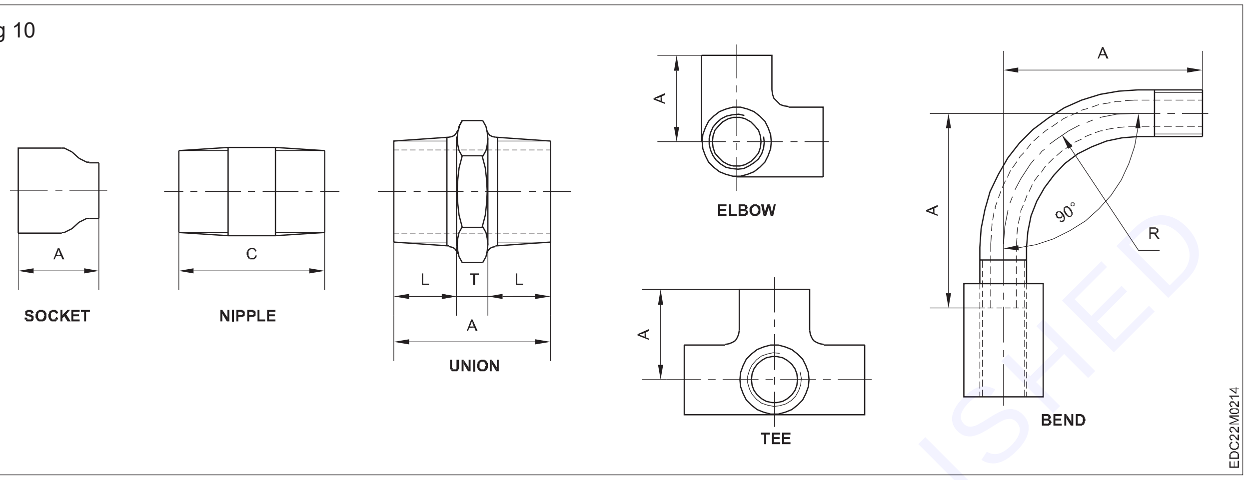
- Sockets
- Nipples
- Elbows
- Tees
- Bends
- Unions etc.

These pipe fitting accessories are made of cast iron, steel, brass, copper, plastic etc.

Pipe connections are shown in drawing in different ways like

- Scale layout
- Diagrammatic layout or Line drawing.

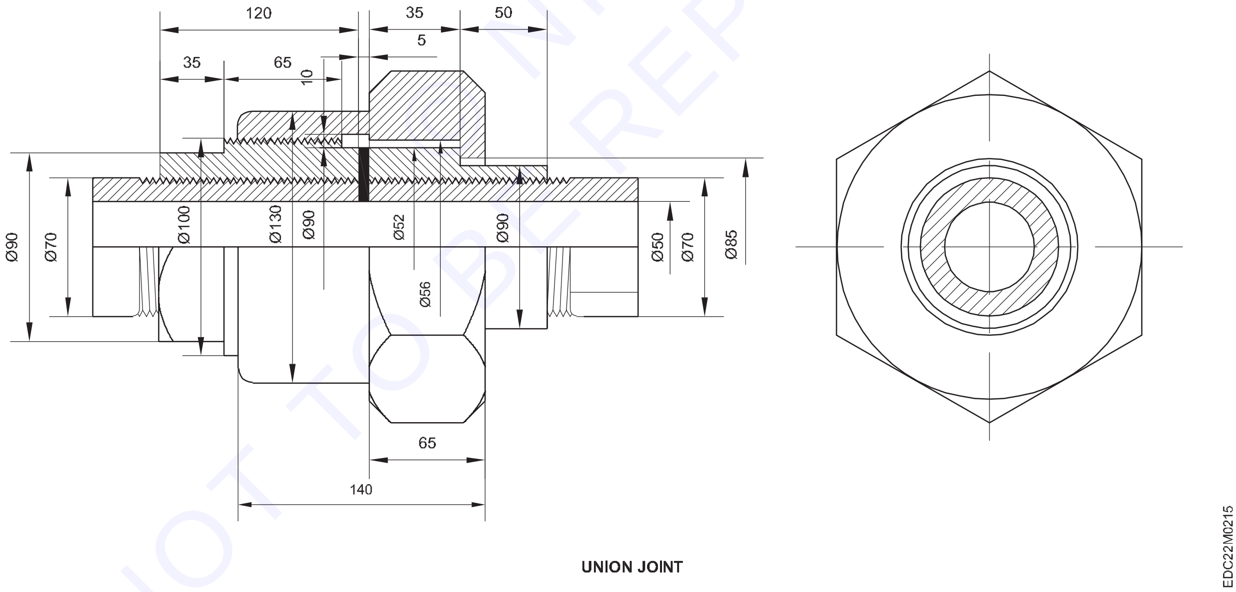
Fig 10



Exercise 8.49

Redraw the given union joint in 1:2 scale in work book sheet.

Fig 11

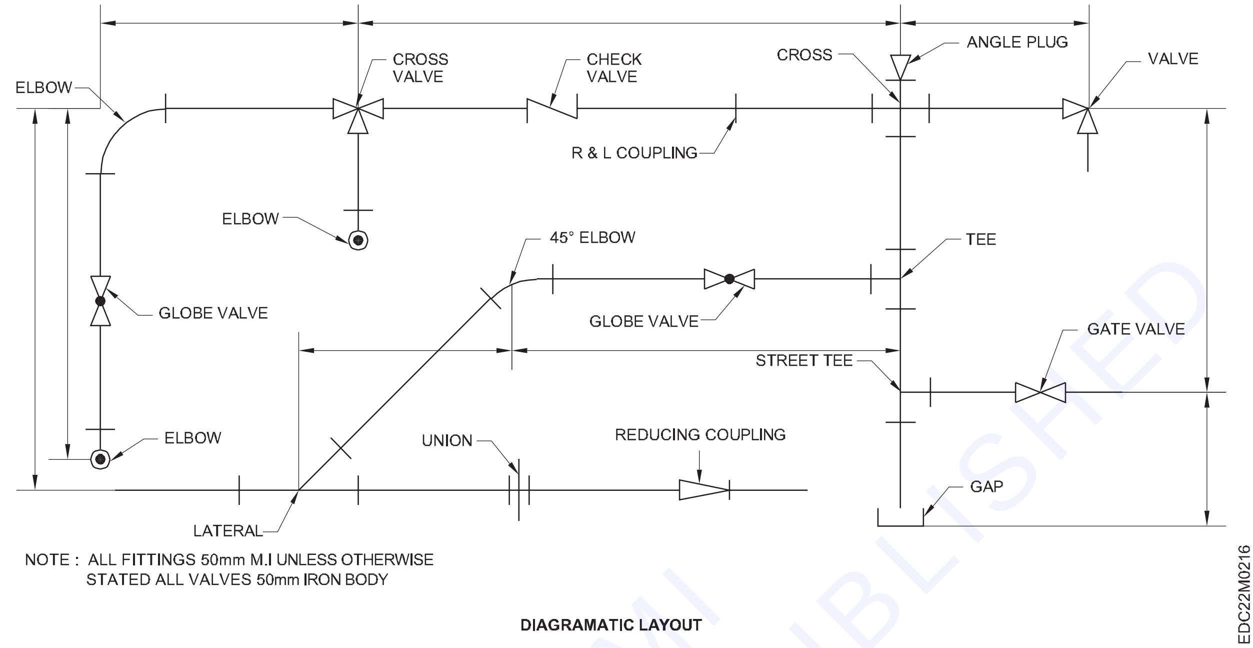


UNION JOINT

Exercise 8.50

Reproduce the given diagrammatic pipe layout in 2:1 size in work book sheet.

Fig 12



NOTE : ALL FITTINGS 50mm M.I UNLESS OTHERWISE
STATED ALL VALVES 50mm IRON BODY

DIAGRAMATIC LAYOUT

Assembly view of split muff coupling (Four bolts type)

Assembly view of split muff coupling (Four bolts type)

Draw the part details of a split muff coupling having four bolts are given in Fig 1. Assemble them and draw the front view, top view and assembly (Fig 2).

Fig 1

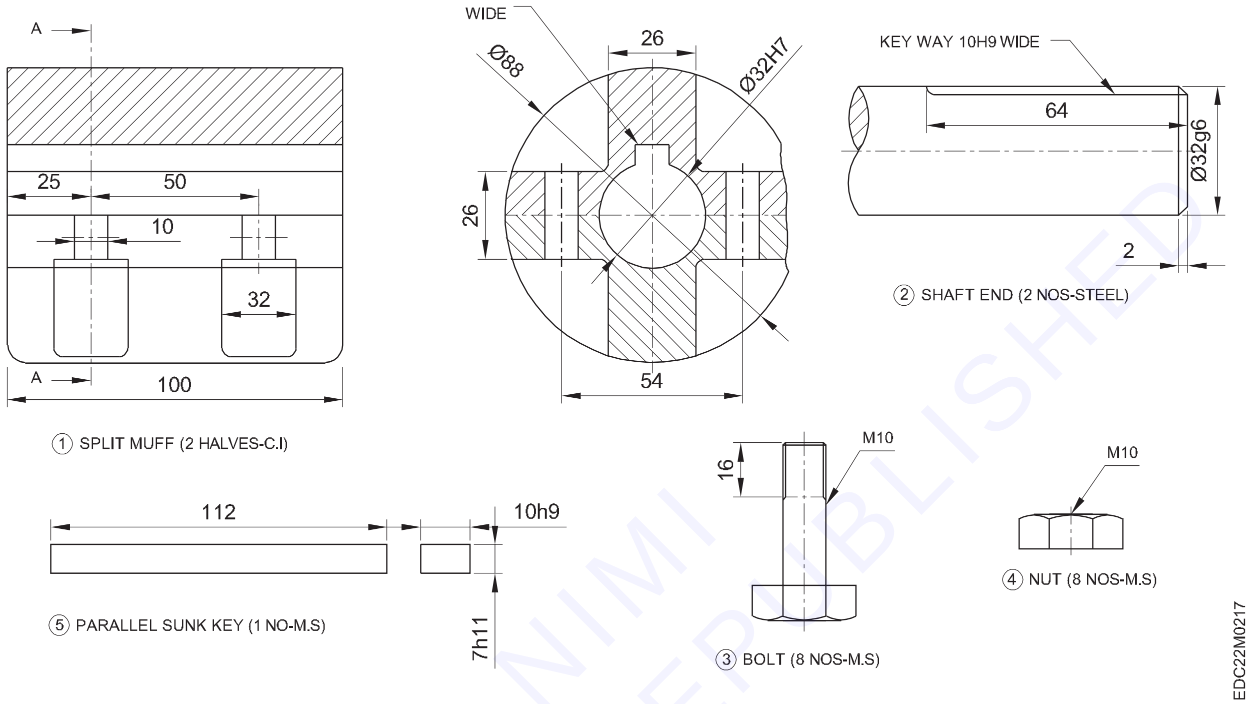
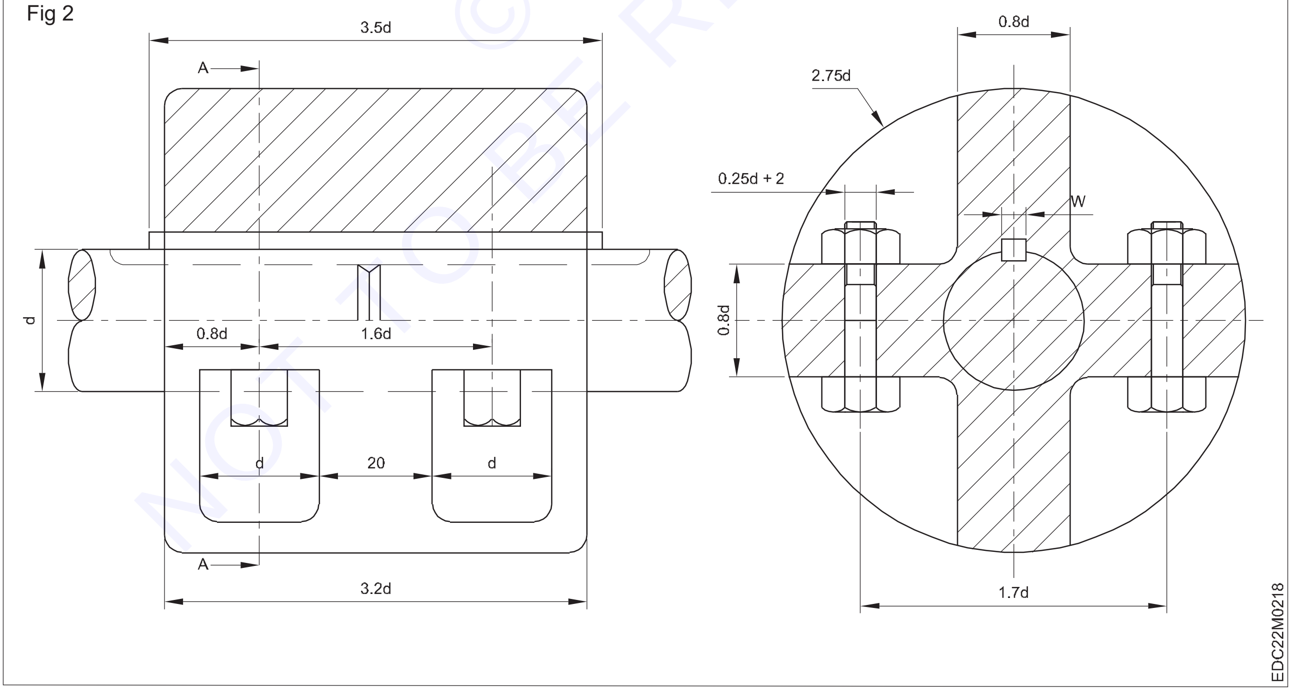


Fig 2



Assembly view of split muff coupling (Eight bolts type)

Draw the part details of a split muff coupling of eight bolt type shown in (Fig 3). Draw the front view, top view and assembly (Fig 4).

Fig 3

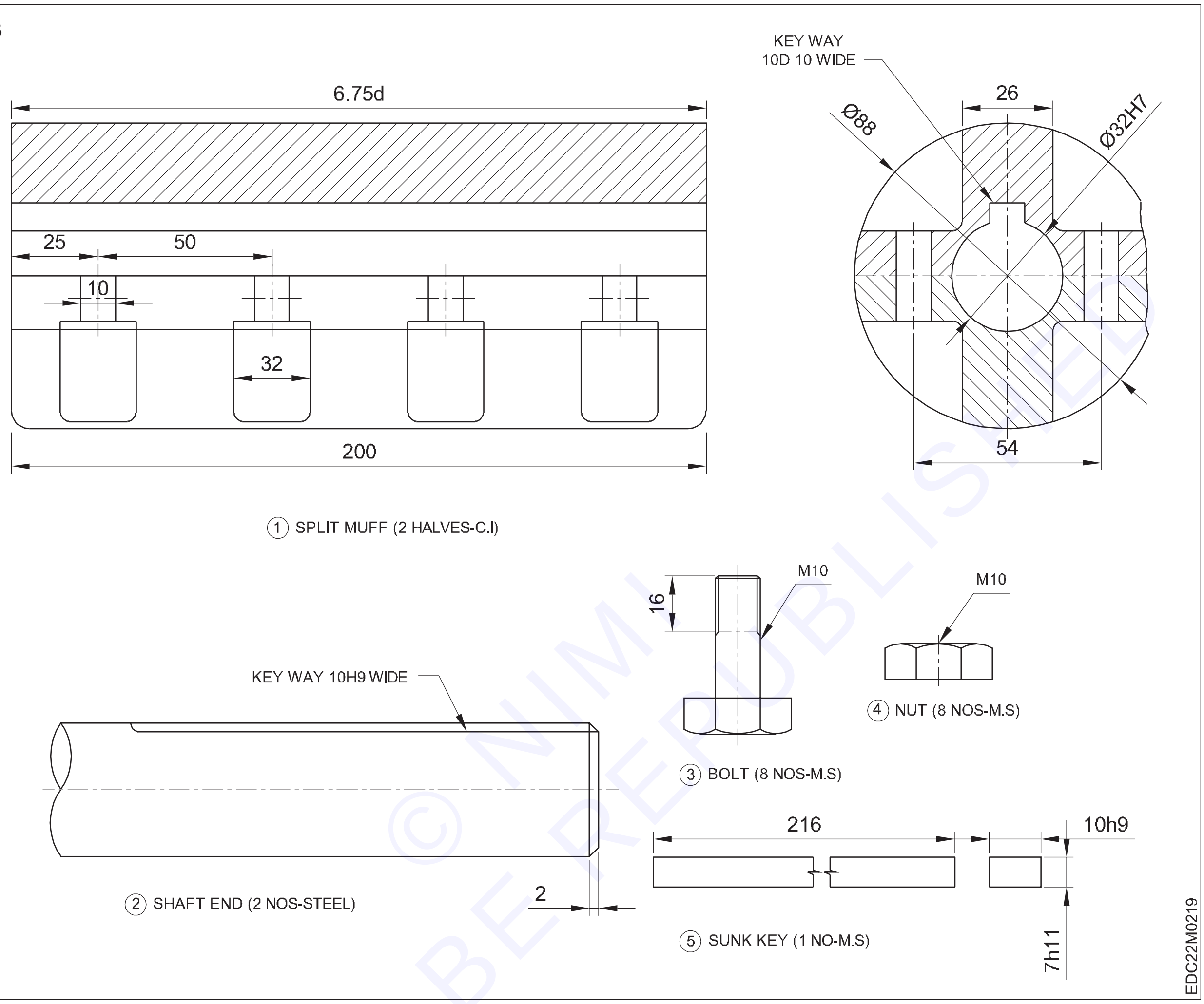
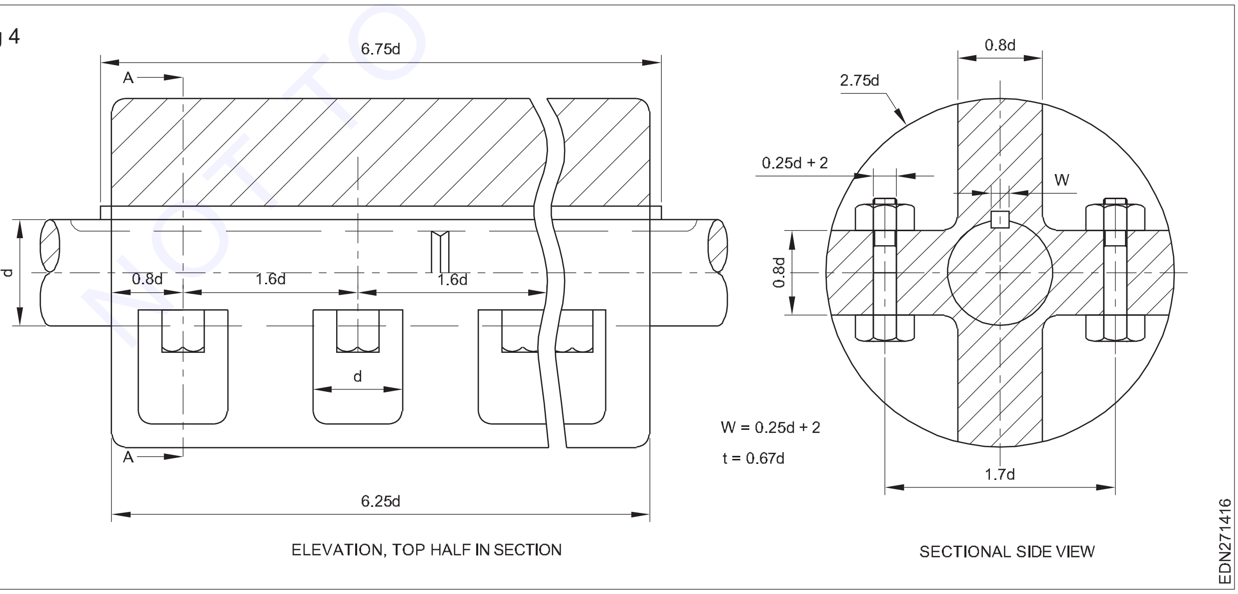


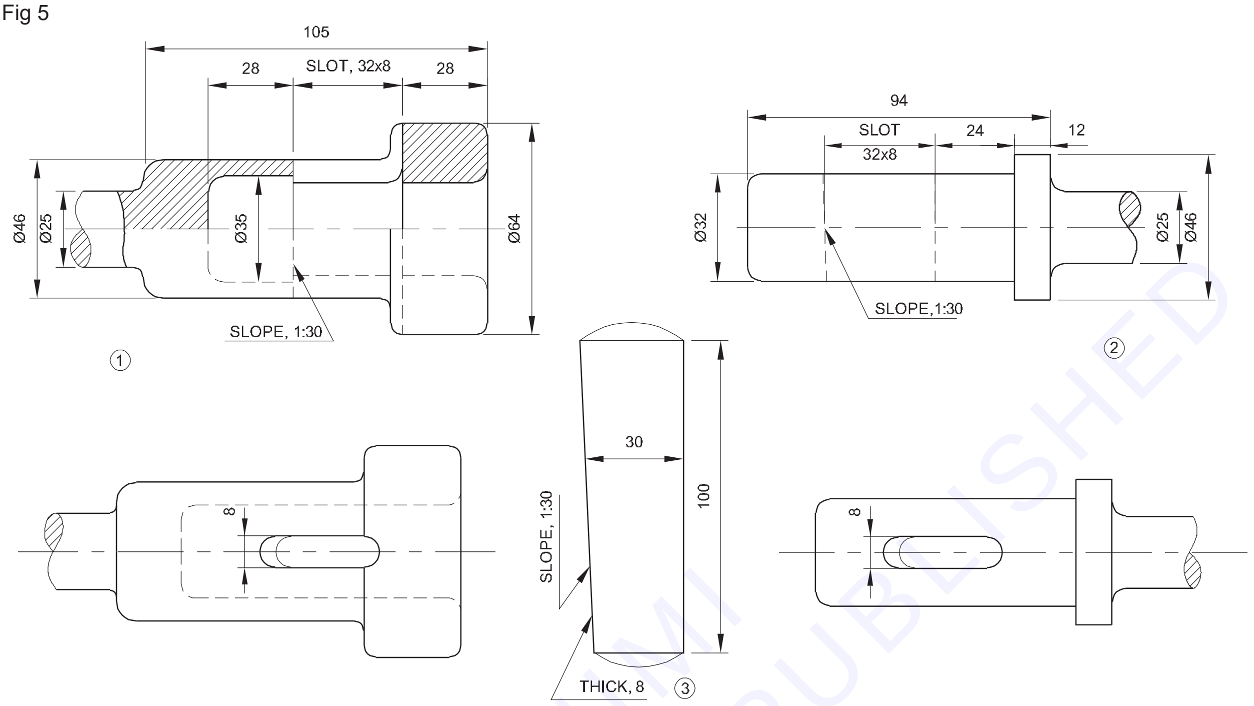
Fig 4



Assembly view of half lap coupling

Draw the part details of a half lap coupling given in (Fig 5). Assemble them and draw the front view, top view and assembly (Fig 6).

Fig 5

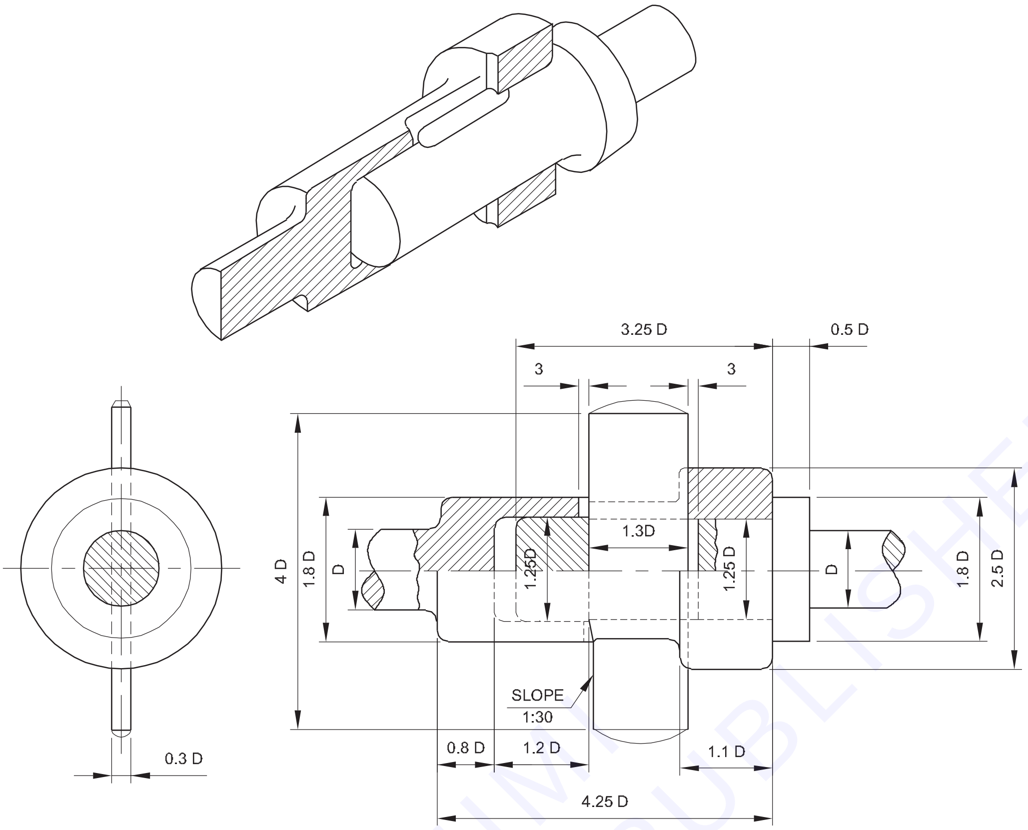


SL. NO.	NAME	MATERIAL	QTY
1	SOCKET END	MS	1
2	SPIGOT END	MS	1
3	COTTER	HCS	1

PARTS AND ASSEMBLY OF SOCKET AND SPIGOT JOINT

EDC22M0221

Fig 6

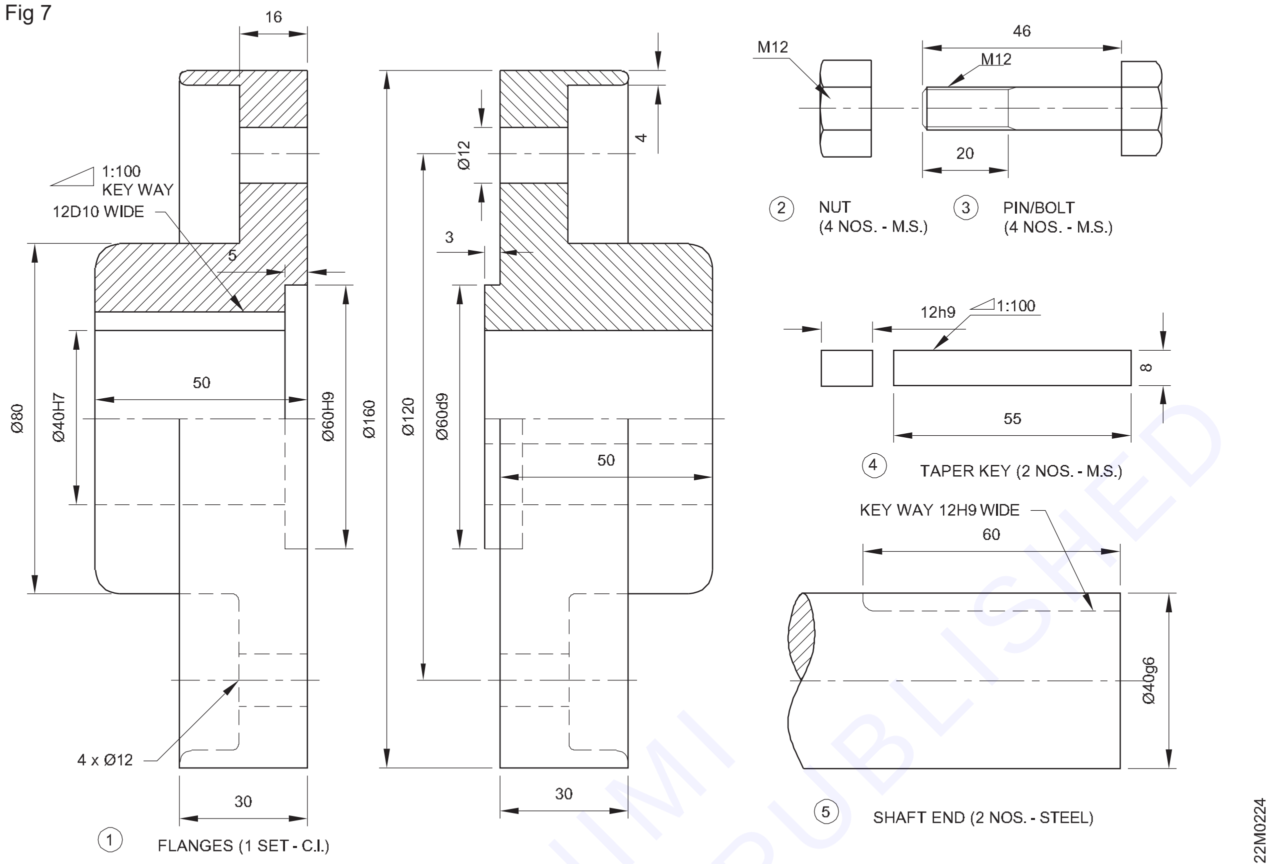


ASSEMBLY OF SOCKET AND SPIGOT

Assembly view of flange coupling

Draw the part details of a flanged coupling having four bolts are given in (Fig 7). Assemble them and draw the top side sectioned front view, top view and assembly (Fig 8).

Fig 7



FLANGED COUPLING (DETAILS) (PROTECTED TYPE)

Fig 8

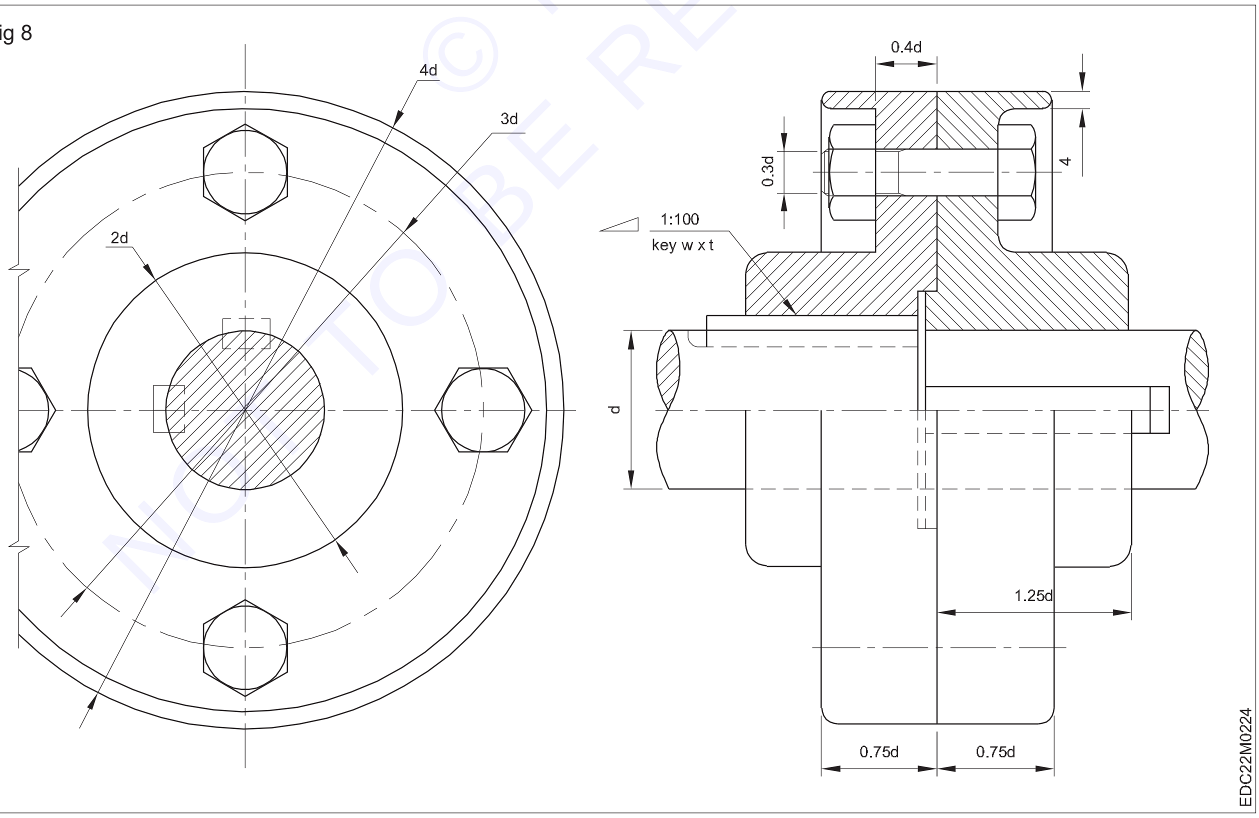
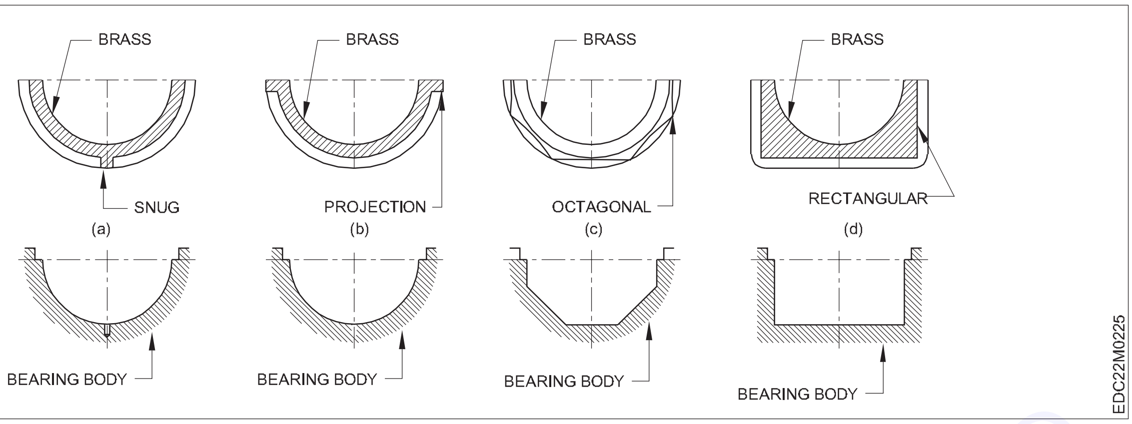


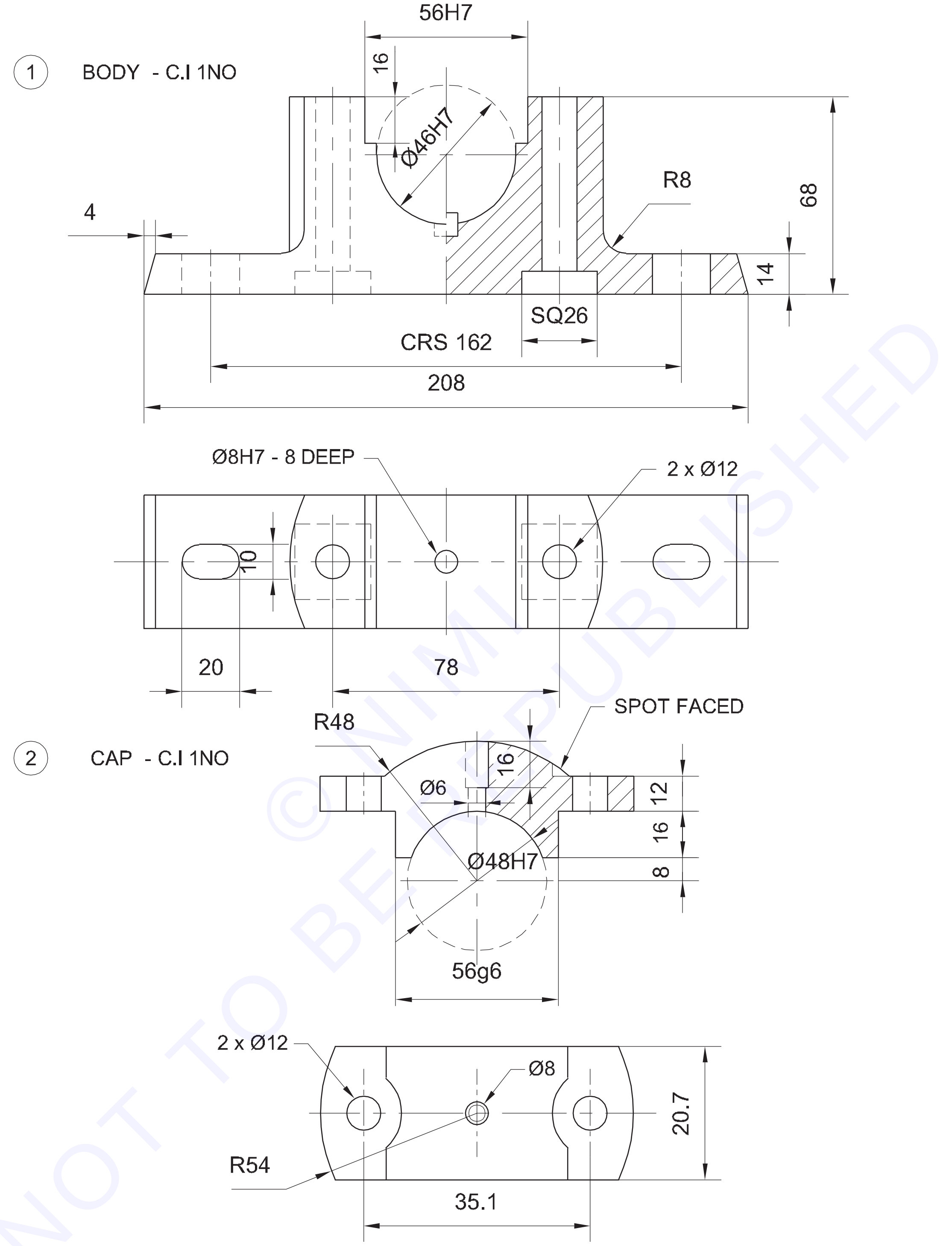
Fig 9



Draw the given details of the pedestal bearing and draw the following views of the assembly

1 The front view 2 Top view 3 Slide view

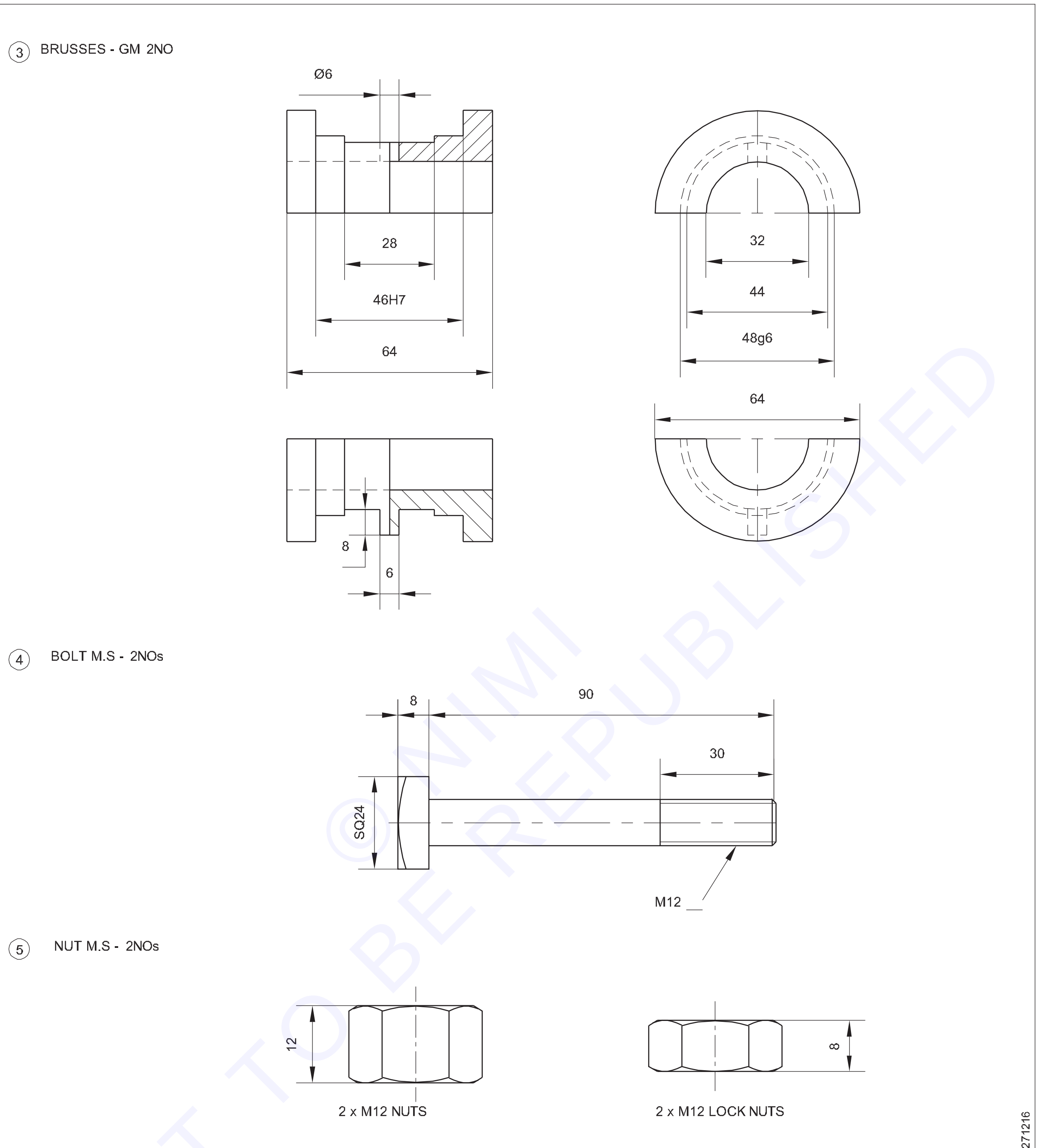
Fig 10



PEDESTAL BEARING DETAILS

EDC22M0226

Fig 11



PEDESTAL BEARING DETAILS

EXERCISE 9 : Detail Drawing and Assembly Drawing

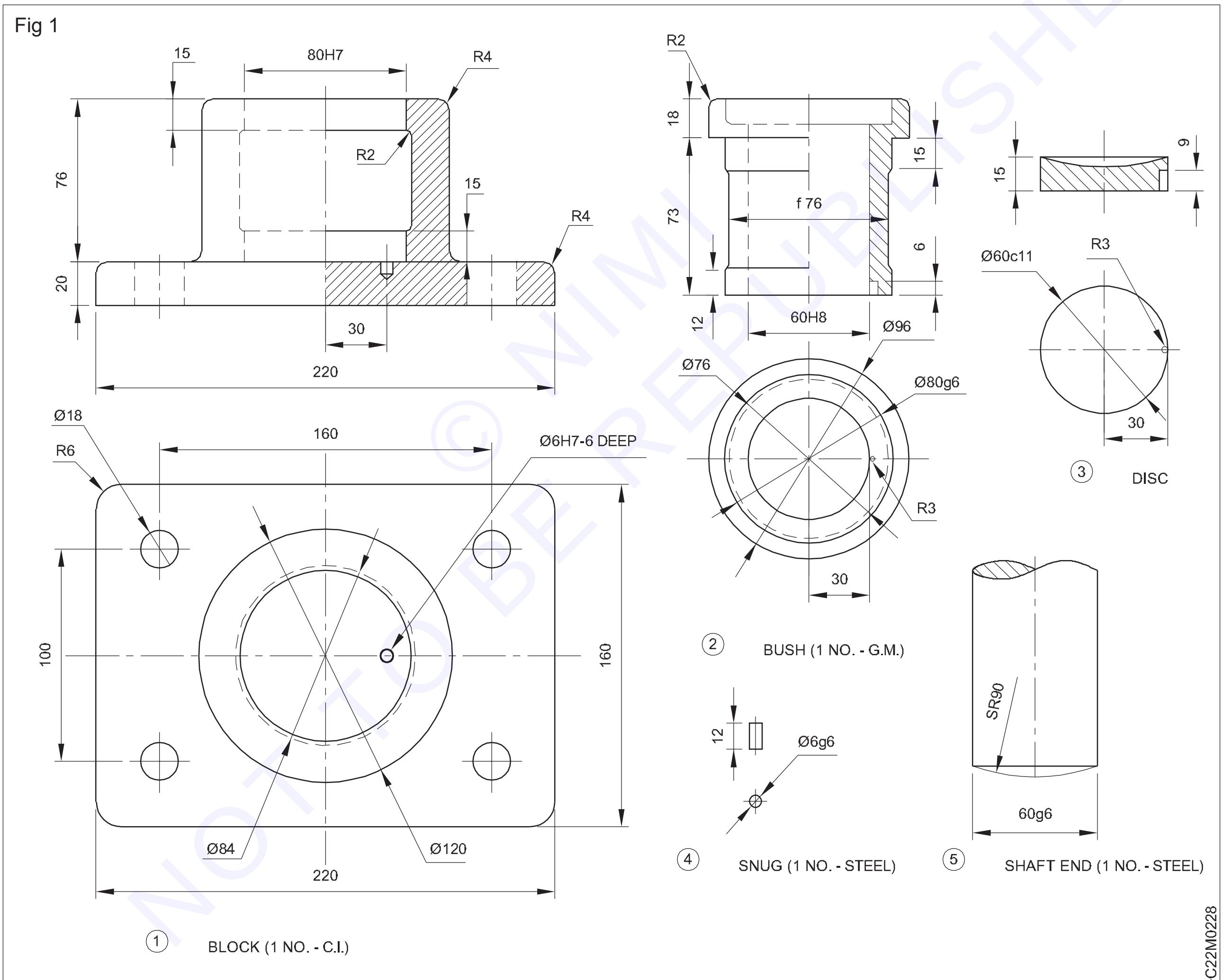
Assembly view of bearings

Foot step Bearing

Footstep or pivot bearing is used to support vertical shaft ends for axial loads. A gunmetal bush having collar on top is placed inside a cast iron body called sole. Inside the bush, a gunmetal disc is placed to support the end of the shaft. Fig 1 shows the part details. A snug is provided on the sole between the bush and the disc in order to prevent the rotation of them. Note that the shaft end has a convex surface and the disc has a matching concave surface to support. The bush and the block are recessed to reduce the machining length of the bush and to result better fitting. The hollow collar portion of the bush provides a slot around the shaft to admit lubricating oil.

The body has to be drawn first and then the bush and disc are placed in position. Locate the snug and place the shaft to complete the thin line drawing. After removing the unnecessary lines, convert the thin lines to visible and hidden lines. Write the dimensions and hatch the sectioned areas to complete the views.

Fig 1



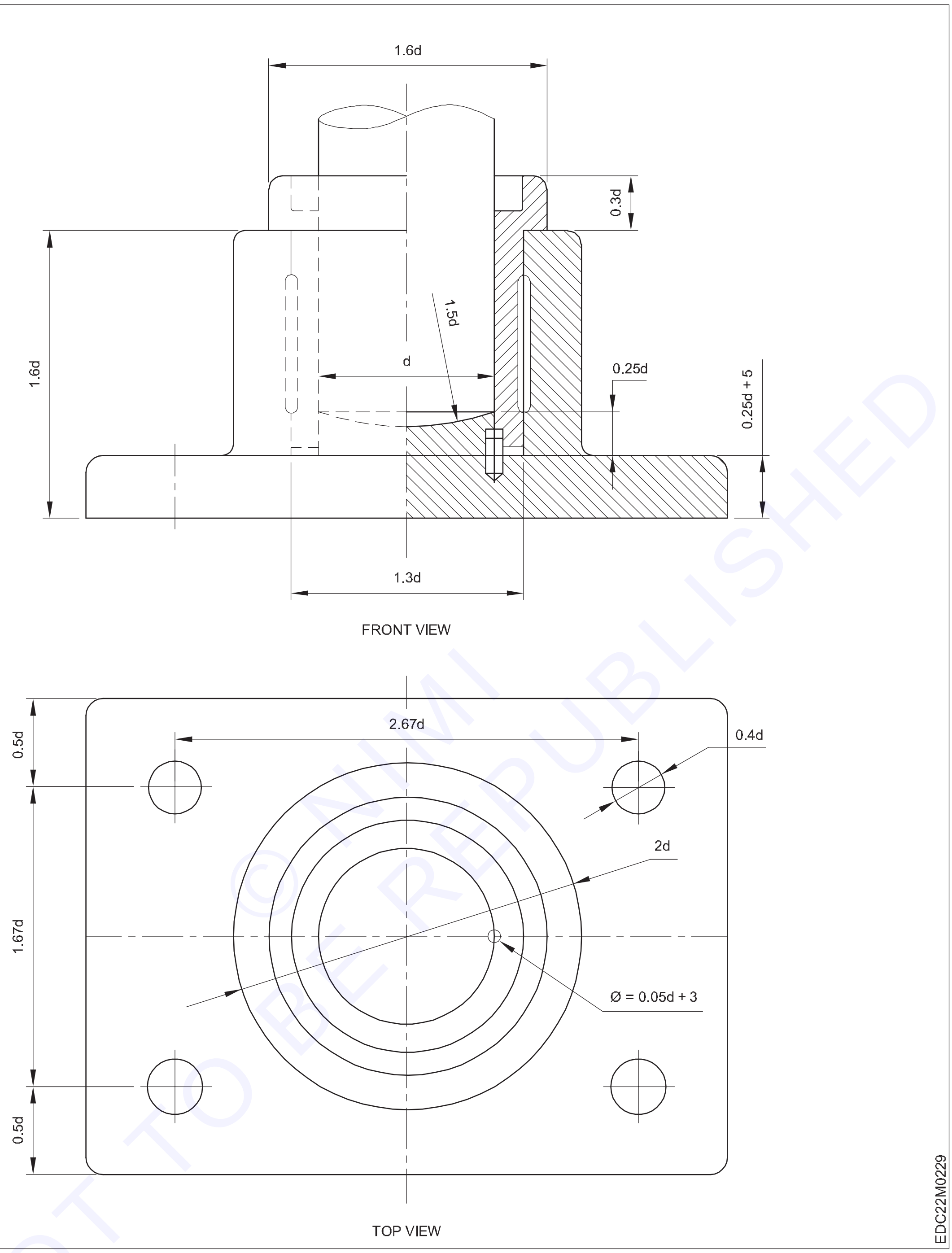
FOOTSTEP BEARING (PARTS)

Practice : Assemble and draw the part drawing of a foot step bearing shown in Fig 2.

1 Front view with right half in section

2 Top view

Fig 2


1.6d
0.3d
1.5d
d
1.6d
0.25d
0.25d + 5
1.3d
FRONT VIEW
2.67d
0.4d
0.5d
2d
1.67d
Ø = 0.05d + 3
0.5d
TOP VIEW
EDC22M0229

Shaft bearings

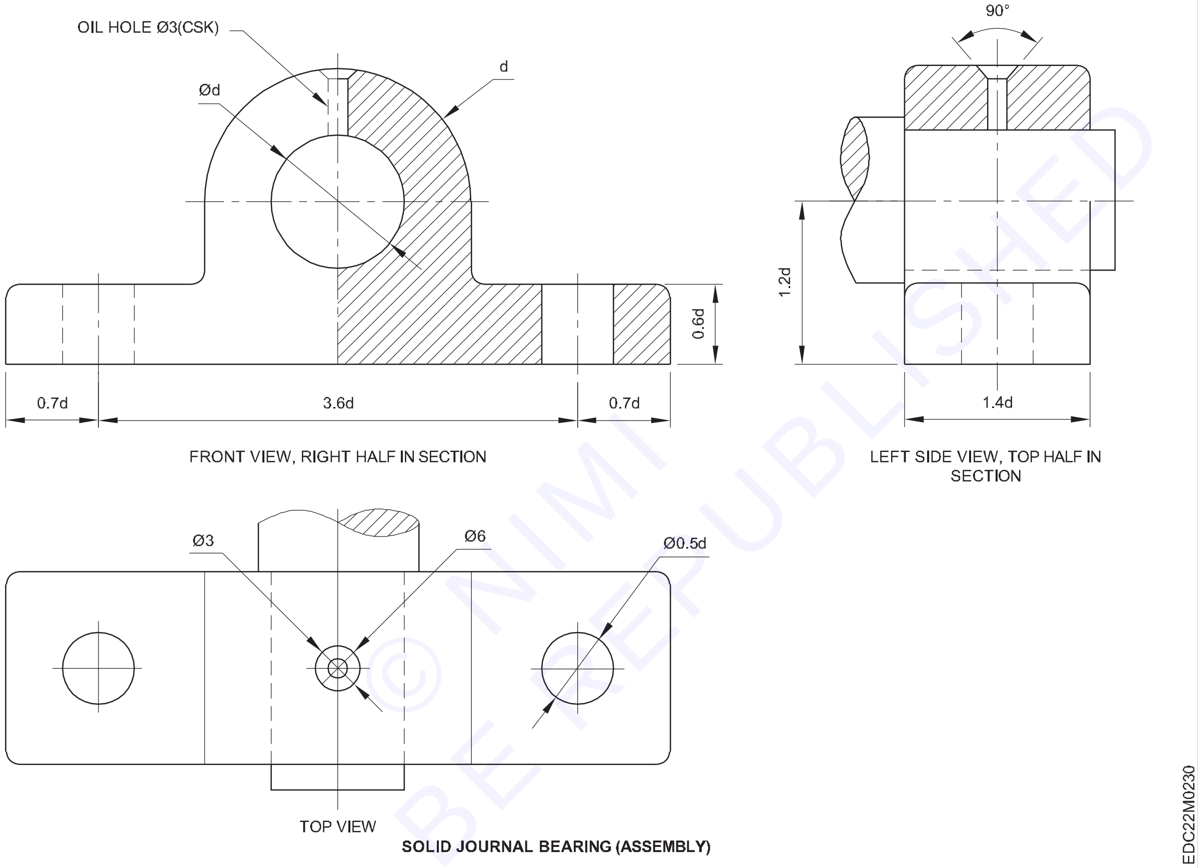
Practice : Sketch the following view of a solid journal bearing assembly and mark the dimensions in terms of diameter (d) (Fig 1).

1 Front view with right half in section

2 Top view

3 Side view with top half in section

Fig 1



SOLID JOURNAL BEARING (ASSEMBLY)

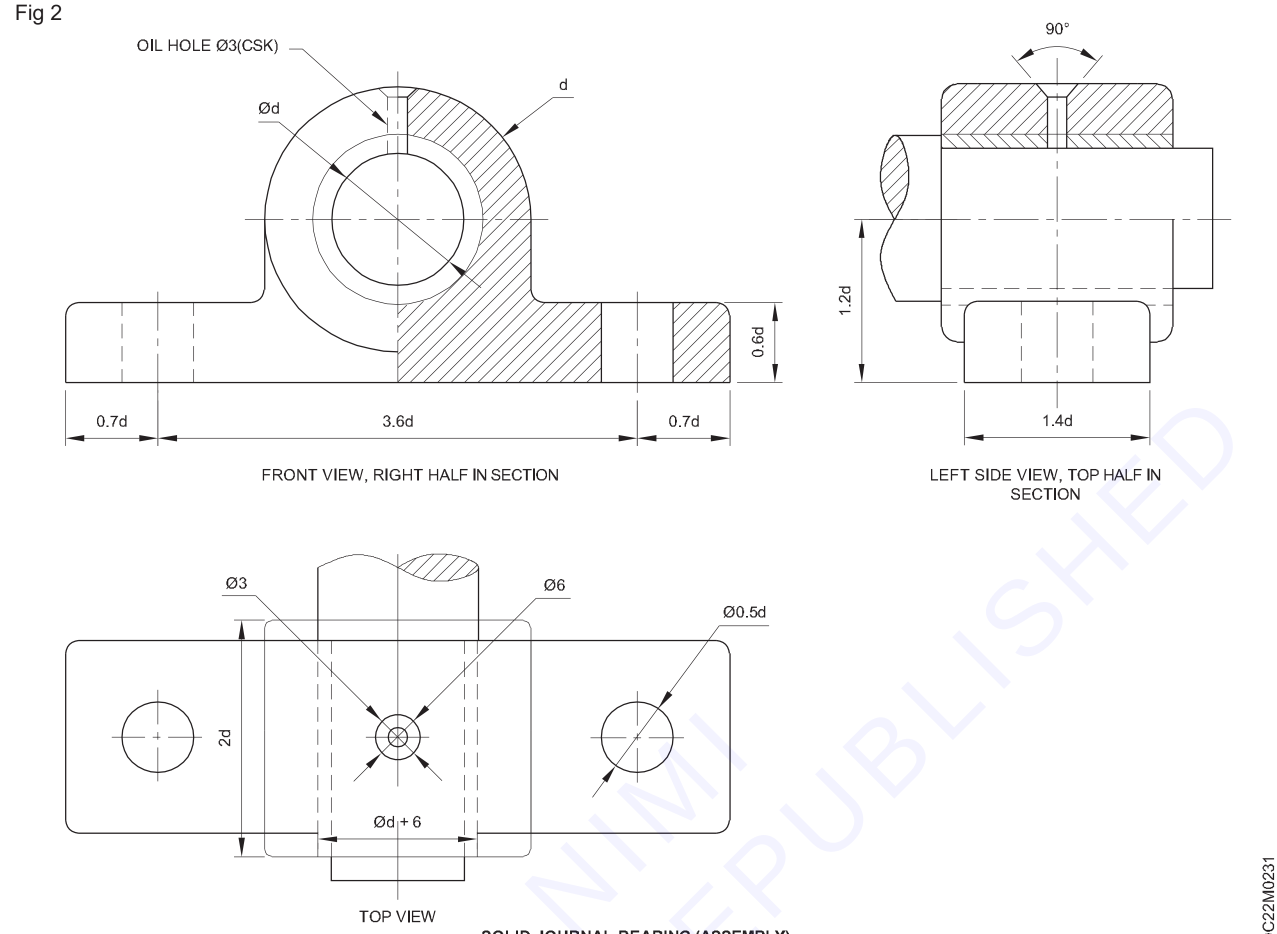
Practice : Draw the following views of a bushed journal bearing assembly and mark the proportions in terms of diameter d (Fig 2).

1 Front view with right half in section

2 Top view

3 Side view with top half in section

Fig 2



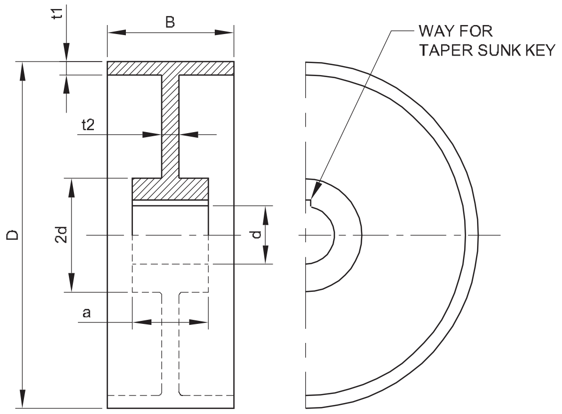
SOLID JOURNAL BEARING (ASSEMBLY)

Draw a solid pulley

Excercise 1: Solid flat pulley (disc type) using by CAD

Sketch two views of solid flat pulley of disc type made of C.I the outer diameter is 200 mm width 72 mm and shaft diameter 32 mm. (Fig 1)

Fig 1



APPROX.PROPORTIONS:

D = OD (160 TO 450)
B = WIDTH
d = SHAFT DIA

a = 0.7 to 1B
t1 = 6 to 10 mm
t2 = 8 to 12 mm

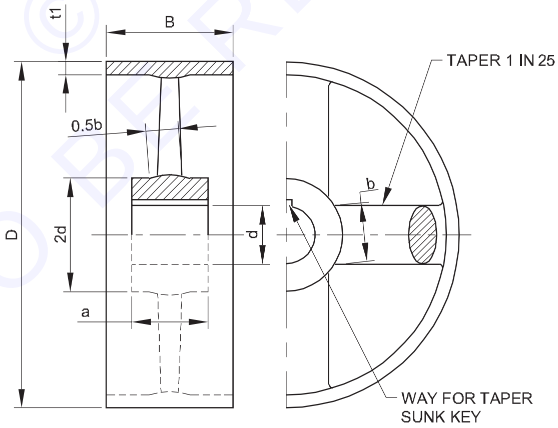
SOLID FLAT PULLEY (DISC TYPE)

EDC22M0232

Excercise 2: Solid flat pulley (for arms) by using CAD

Draw the cast iron flat pulley has outer dia 240 mm and width 80 mm If the pulley has four arms and shaft size is 32 mm draw half section elevation and side view of pulley. (Fig 2)

Fig 2



APPROX.PROPORTIONS:

D = OD (UP TO 450)
t1 = 6 to 10 mm
d = SHAFT DIA.

a = 0.7 to 1B
B = WIDTH
b = [B*D/1000] + 10mm

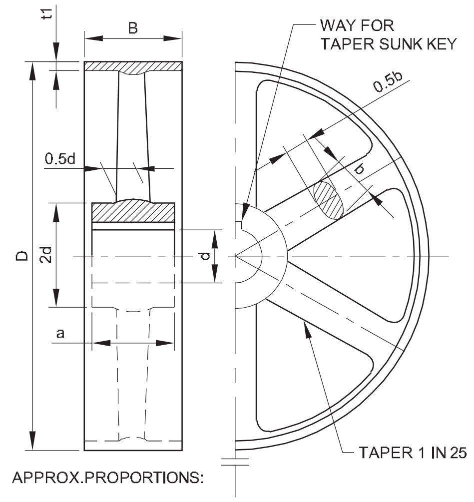
SOLID FLAT PULLEY (FOUR ARM TYPE)

EDC22M0233

Excercise 3: Solid flat pulley (six arm types)

Draw the below mentioned solid that pulley (six arm type) in proportional dimensions. (Fig 3)

Fig 3



APPROX.PROPORTIONS:

D = OD (450 TO 1000)

t1 = 6 TO 10 mm

b = [B*D/1500] + 20 mm

B = WIDTH

a = 0.7 to 1B

d = SHAFT DIA

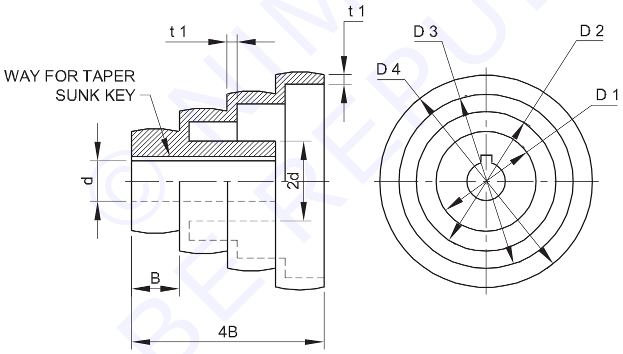
SOLID FLAT PULLEY (SIX ARM TYPE)

EDC22M0234

Excercise 4: Stepped cone pulley (for arms) by using CAD

Draw full size sectional front view and side view of the stepped pulley by using CAD. (Fig 4)

Fig 4



APPROX.PROPORTIONS:

D1 TO D4 = OD in STEPS

B = WIDTH

d = SHAFT DIA

t1 = 6 to 10 mm

STEPPED CONE PULLEY - C.I. (SPEED CONE)

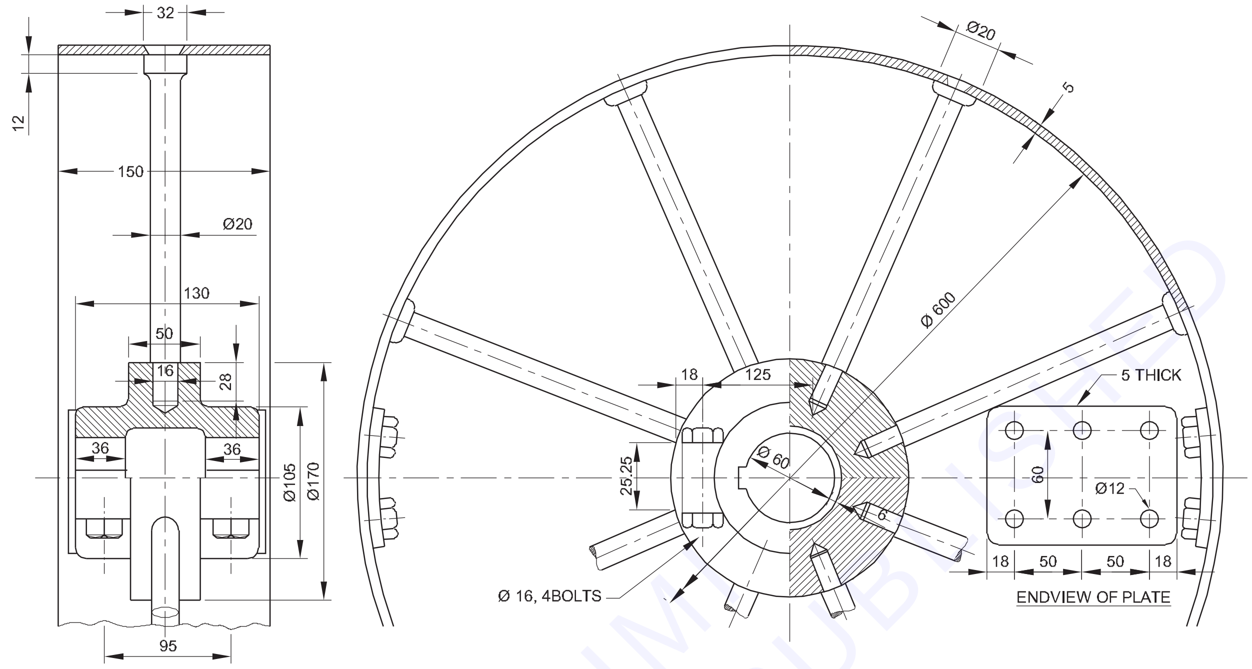
EDC22M0235

Draw a Built up pulleys

Excercise 1

Draw neat giving main dimensions of the built up pulley 60 mm mounted on a shaft 600 mm dia. (Fig 1)

Fig 1



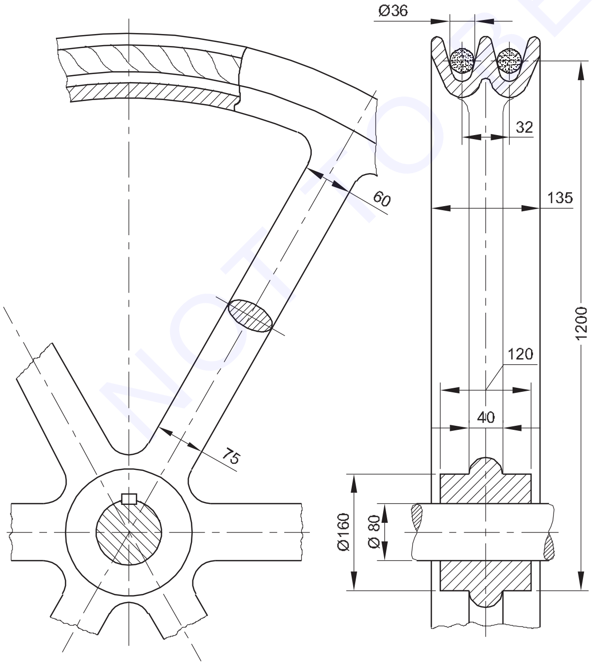
BUILT-UP PULLEY

Excercise 2

Demonstrate pulley with dimensions by using CAD (Fig 2)

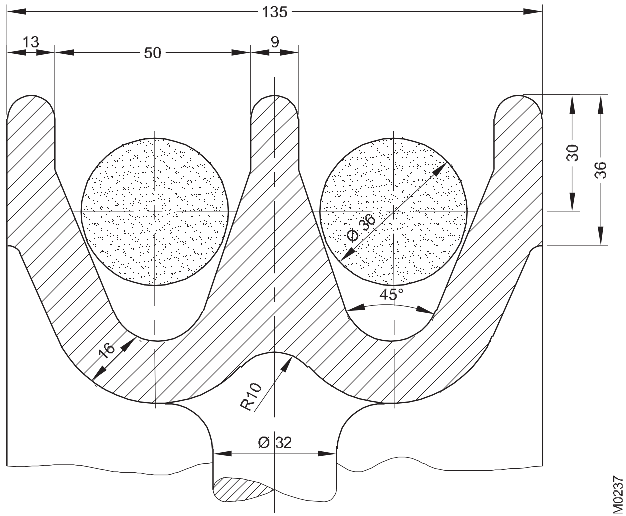
Fig 3 gives the detail of rim for Fig 2

Fig 2



ROPE PULLEY

Fig 3



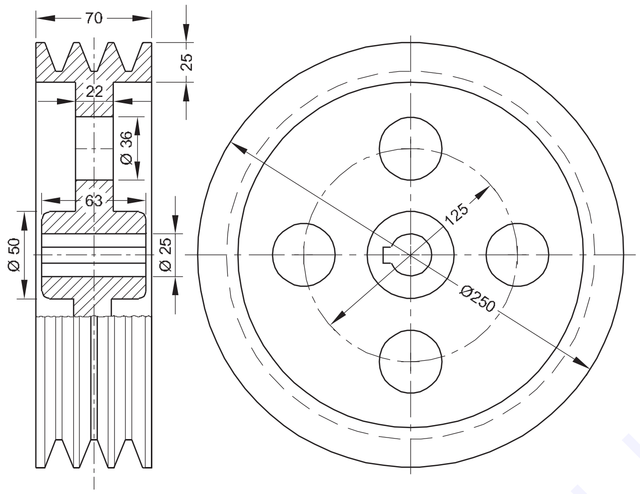
SECTION OF RIM

Excercise 3

Reproduce the ‘V’ belt pulley sectional front elevation side vice by using CAD. (Fig 4)

Note: The details of ‘V’ groove refer the ‘V’ groove details given under Excercise 4.

Fig 4



V-BELT PULLEY

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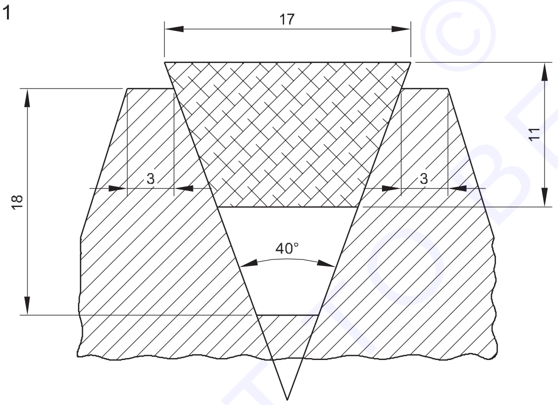
Exercise 4

Draw the details GV groove as given in figure 5.

Note: Table 1 and Table 2 are given for the feature reference for drawing ‘V’ belt pulleys of different dimensions.

Fig 1



DETAILS OF V-GROOVE

DMN34100X1

Table 1

Cross - Section symbol	Nominal top width mm	Nominal thickness mm
A	13	8
B	17	11
C	22	14
D	32	19
E	38	23

Table 2

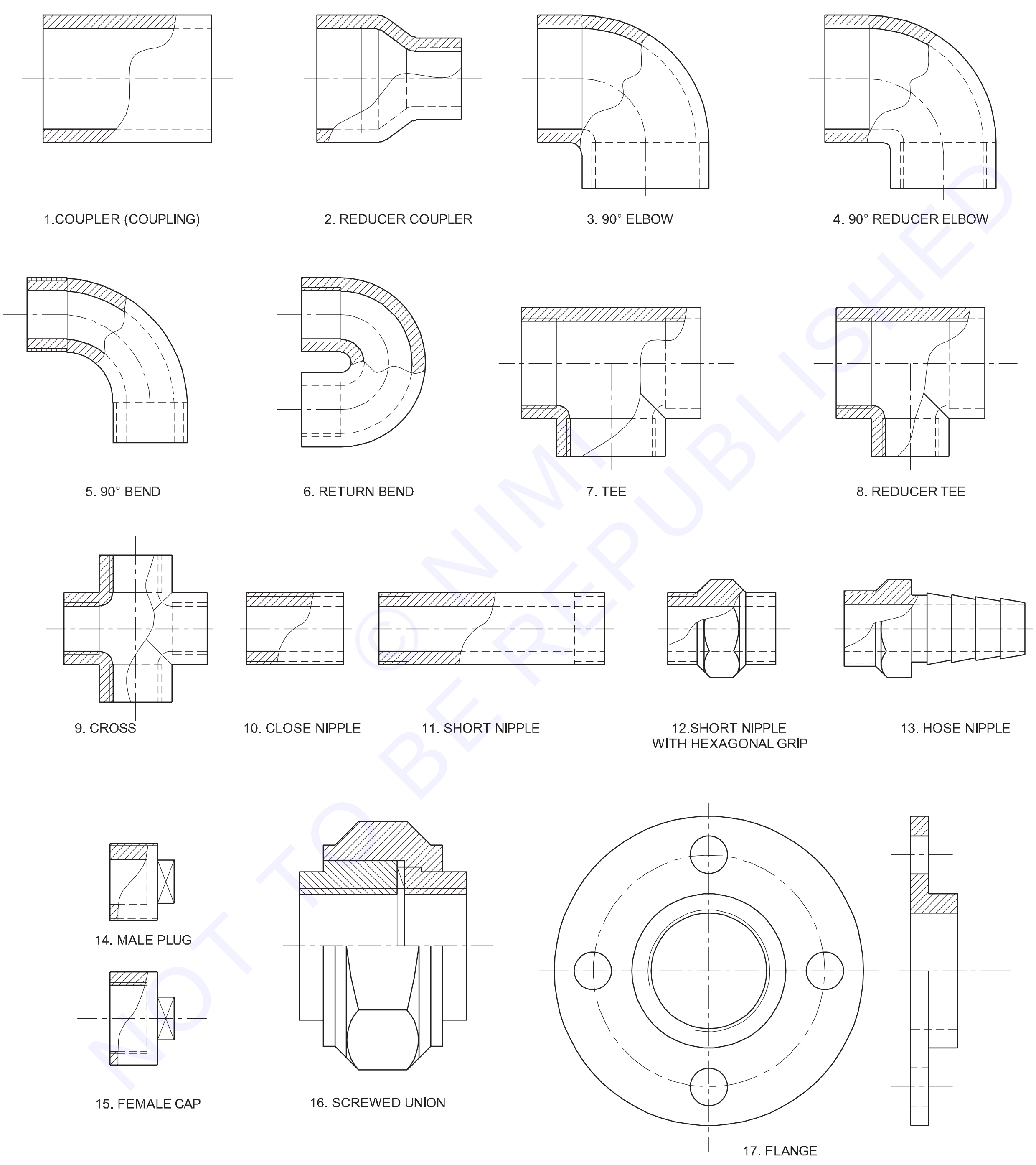
Width of belt (mm	5	6	8	10	13	17	20	22	32	38
Height of belt (mm)	3	4	5	6	8	11	12.5	14	19	23
Length of belt (mm) from to	150	212	296	420	585	832	1100	1650	2303	3230
	860	1262	1916	2820	4275	6332	9540	14050	18063	18080

Draw the pipe fittings

Draw the pipe fittings

TASK 1: **Draw the pipe fittings with proportional dimensions (Fig 1)**

Fig 1



1.COUPLER (COUPLING)

2. REDUCER COUPLER

3. 90° ELBOW

4. 90° REDUCER ELBOW

5. 90° BEND

6. RETURN BEND

7. TEE

8. REDUCER TEE

9. CROSS

10. CLOSE NIPPLE

11. SHORT NIPPLE

12.SHORT NIPPLE WITH HEXAGONAL GRIP

13. HOSE NIPPLE

14. MALE PLUG

15. FEMALE CAP

16. SCREWED UNION

17. FLANGE

PIPE FITTINGS

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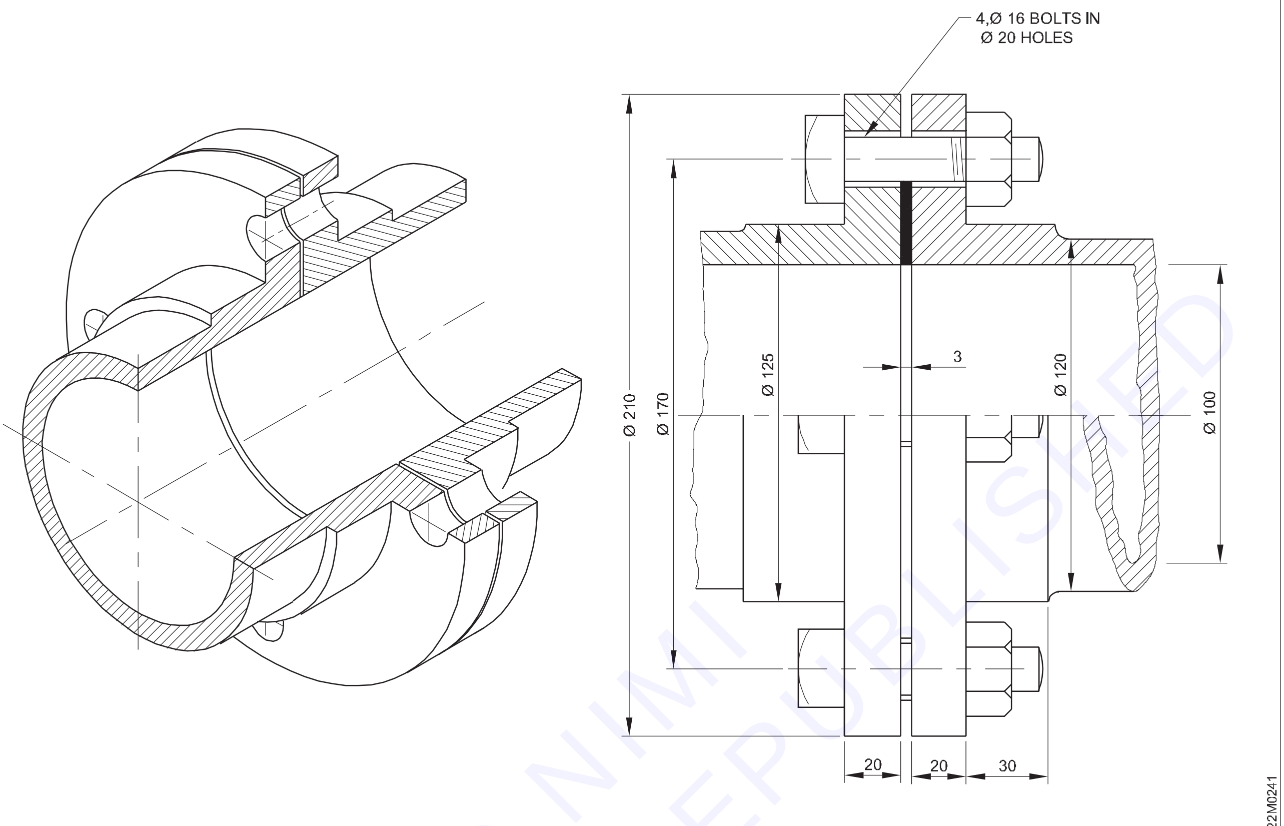
TASK 2: **Draw the cast iron flanged joint for two 100 mm diameter. (Fig 2)**

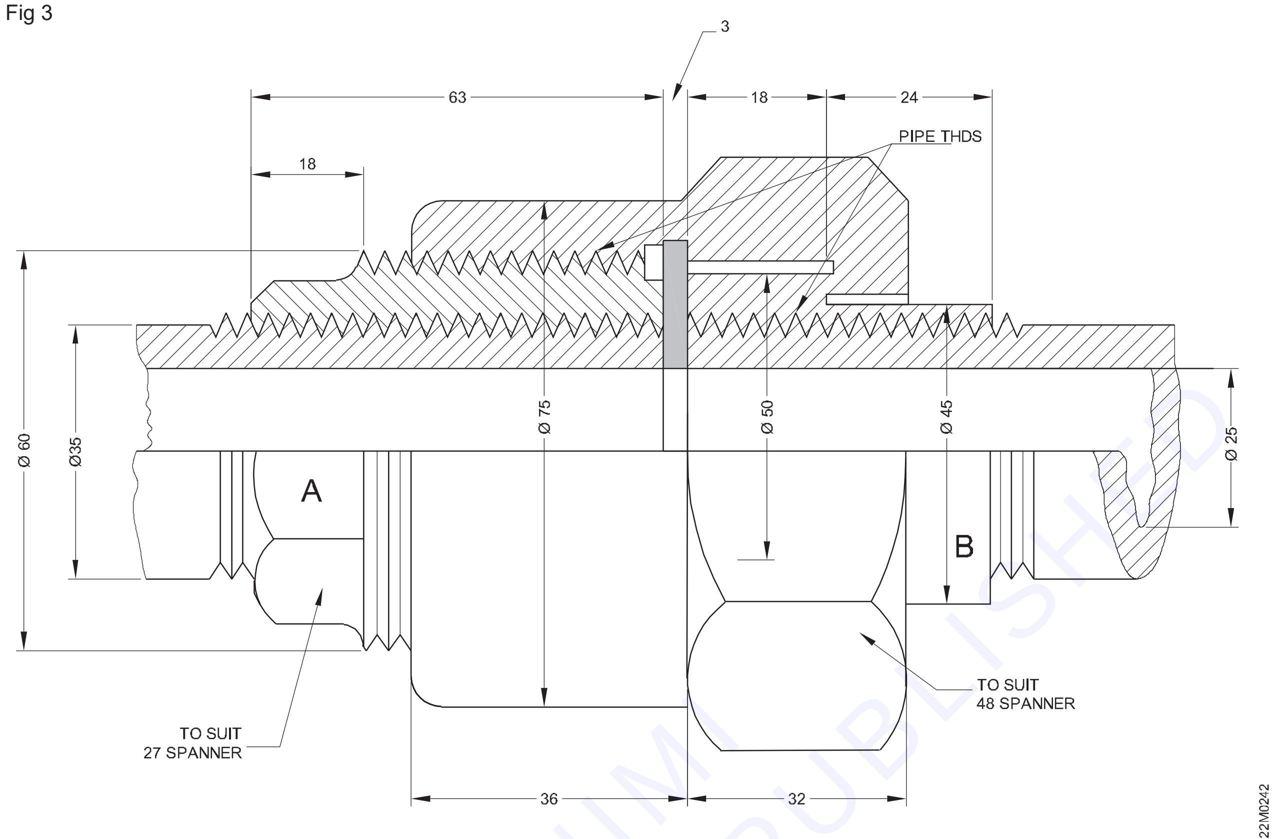
Fig 2



CAST-IRON FLANGED JOINT

TASK 3: **Draw full size two views of a union joint for 25 mm diameter pipes. (Fig 3)**

Fig 3



UNION JOINT

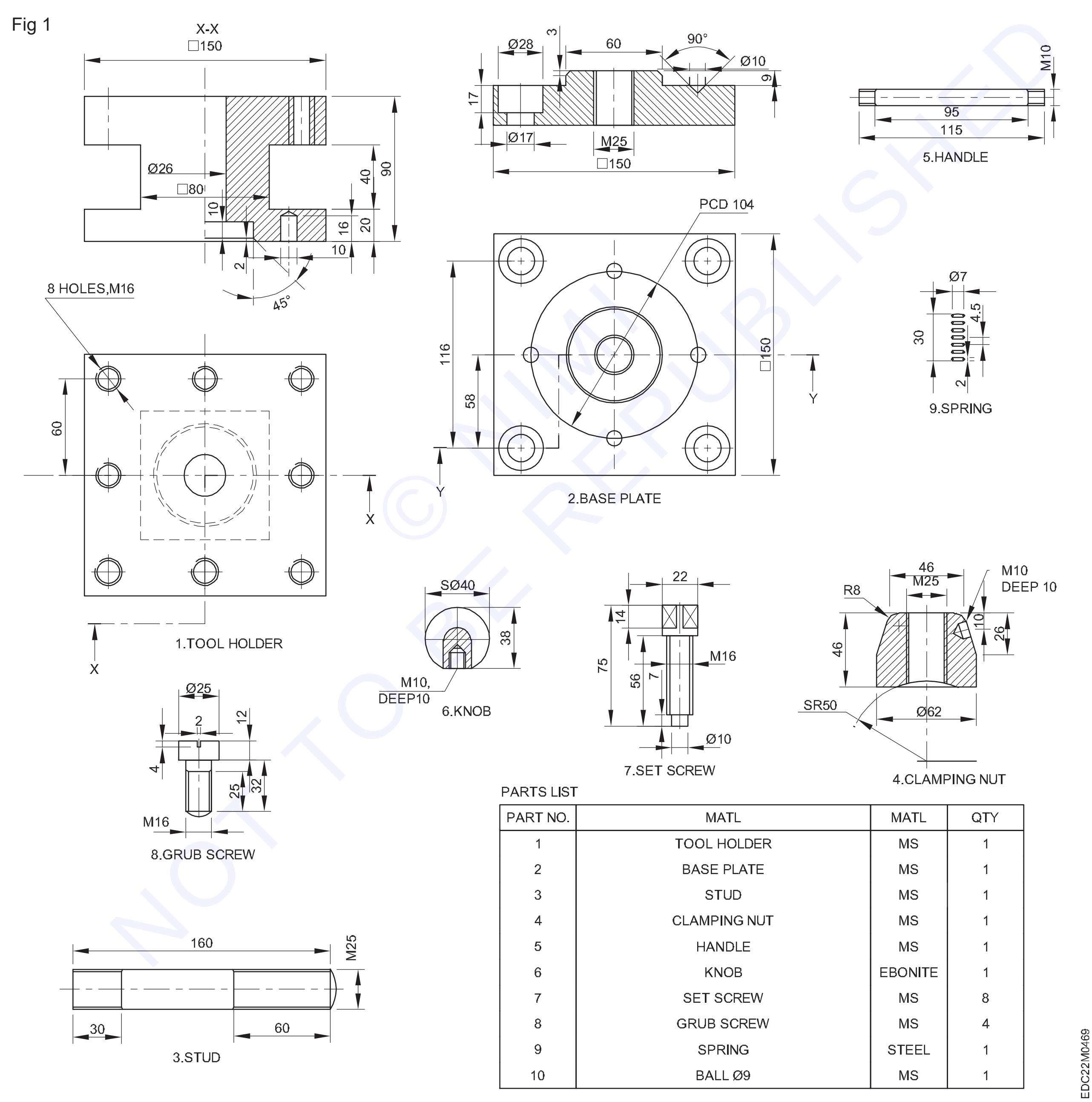
Draw the detailed drawing of a lathe part - tool post using autocad

TASK 1: **Draw the detailed drawing of lathe tool post using autocad**

- Incorporate all the required dimensions for each details.
- Select suitable scale.
- Prepare the part list.

Note : Ball is not shown in detail.

Fig 1



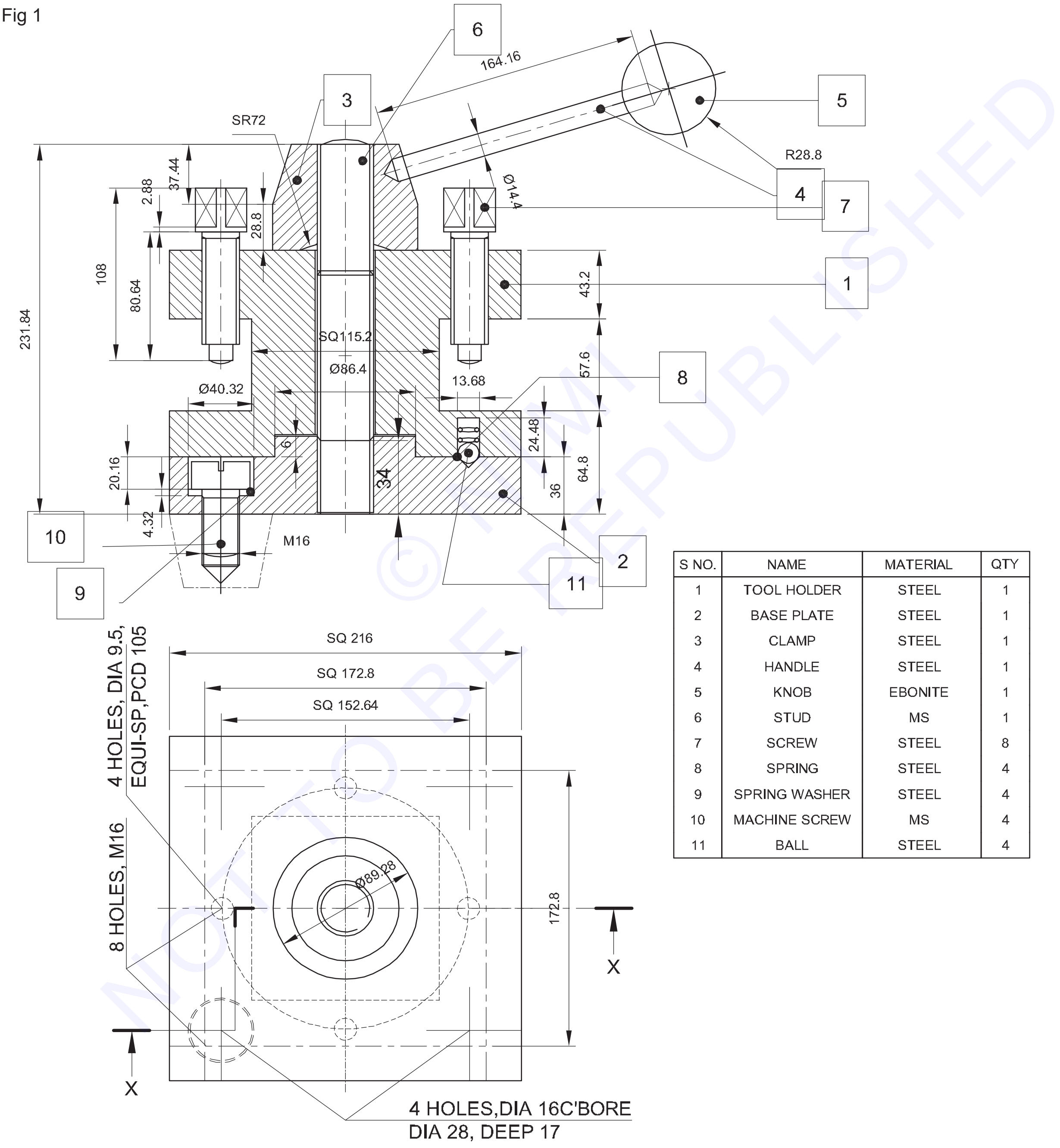
PARTS LIST

PART NO.	MATL	MATL	QTY
1	TOOL HOLDER	MS	1
2	BASE PLATE	MS	1
3	STUD	MS	1
4	CLAMPING NUT	MS	1
5	HANDLE	MS	1
6	KNOB	EBONITE	1
7	SET SCREW	MS	8
8	GRUB SCREW	MS	4
9	SPRING	STEEL	1
10	BALL Ø9	MS	1

TASK 2: **Draw the assemble drawing of lathe tool post using autocad**

- i Draw the full sectional elevation of the lathe tool post.
- ii Draw the full plan of the elevation of the lathe tool post.
- iii Give overall dimensions.
  - – Indicate the part No as per detailed drawing.
  - – Prepare the part list.

Fig 1

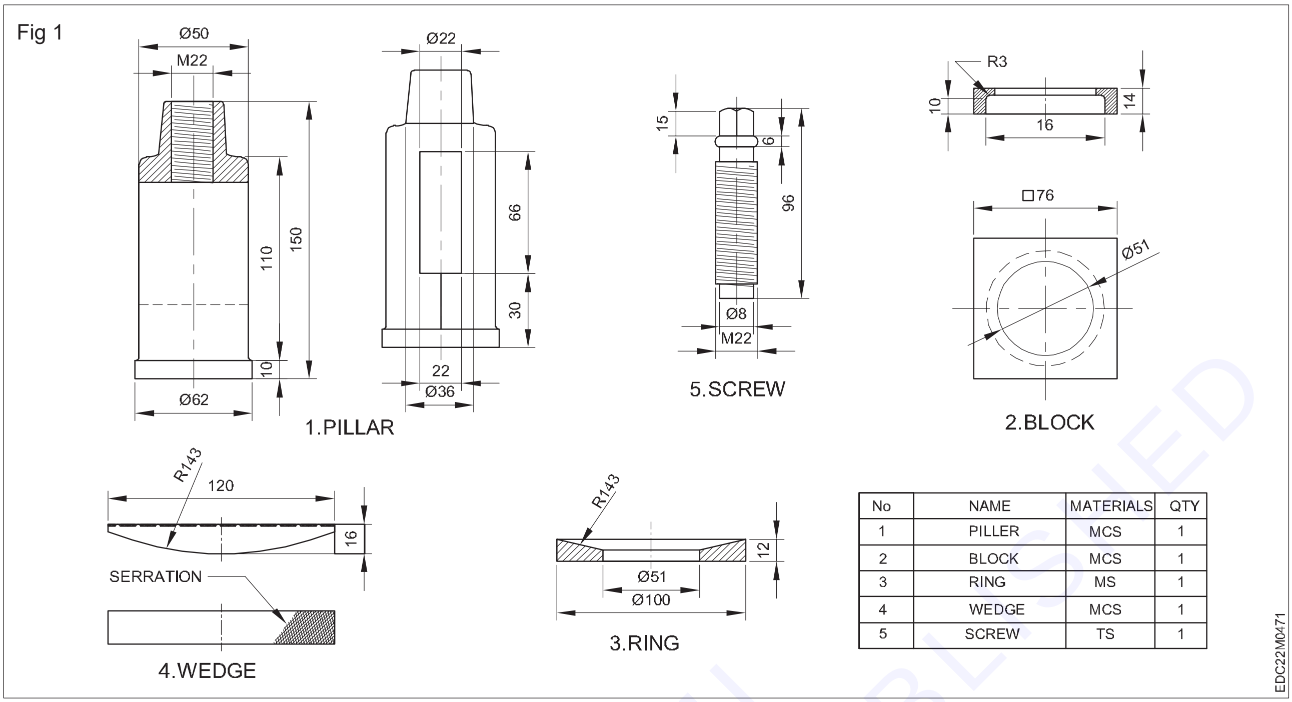


S NO.	NAME	MATERIAL	QTY
1	TOOL HOLDER	STEEL	1
2	BASE PLATE	STEEL	1
3	CLAMP	STEEL	1
4	HANDLE	STEEL	1
5	KNOB	EBONITE	1
6	STUD	MS	1
7	SCREW	STEEL	8
8	SPRING	STEEL	4
9	SPRING WASHER	STEEL	4
10	MACHINE SCREW	MS	4
11	BALL	STEEL	4

SQUARE TOOL POST ASSEMBLY

TASK 3**: Draw the detailed drawing of single tool post**

Fig 1



No	NAME	MATERIALS	QTY
1	PILLER	MCS	1
2	BLOCK	MCS	1
3	RING	MS	1
4	WEDGE	MCS	1
5	SCREW	TS	1

TASK 4: **Draw the follwing single tool post views for the assembly drawing by using autocad**

- Full sectional front view
- Full plan view
- Side elevation looking from left of the front elevation
- Prepare bill of material.

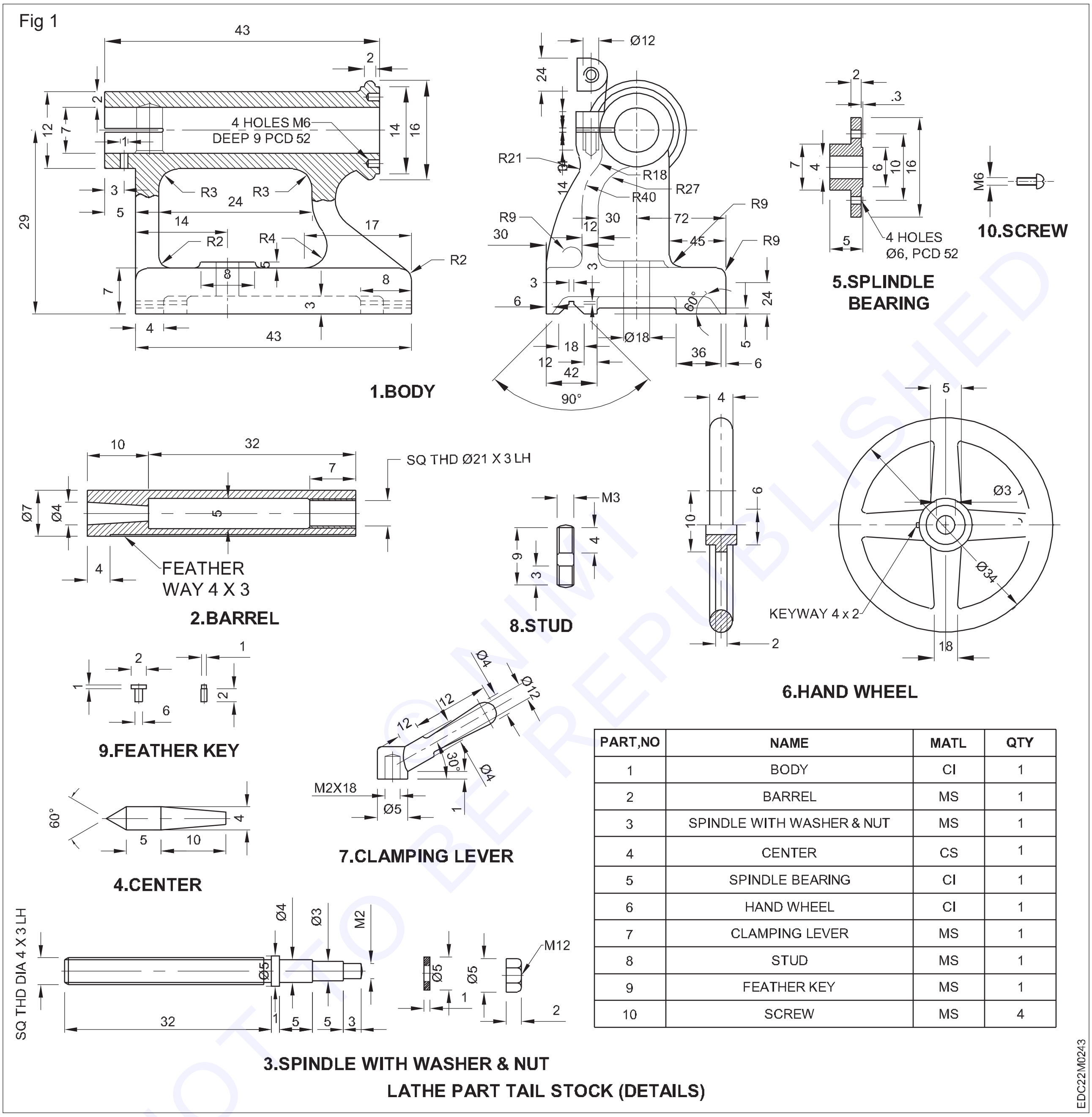
Note: Provide over all dimensions in the assembled views refer detailed drawings.

— — — — — — — —

Construct detailed drawing of a lathe part tail stock

TASK 1: **Draw the detailed parts of a lathe tail stock (Fig 1)**

Fig 1



PART,NO	NAME	MATL	QTY
1	BODY	CI	1
2	BARREL	MS	1
3	SPINDLE WITH WASHER & NUT	MS	1
4	CENTER	CS	1
5	SPINDLE BEARING	CI	1
6	HAND WHEEL	CI	1
7	CLAMPING LEVER	MS	1
8	STUD	MS	1
9	FEATHER KEY	MS	1
10	SCREW	MS	4

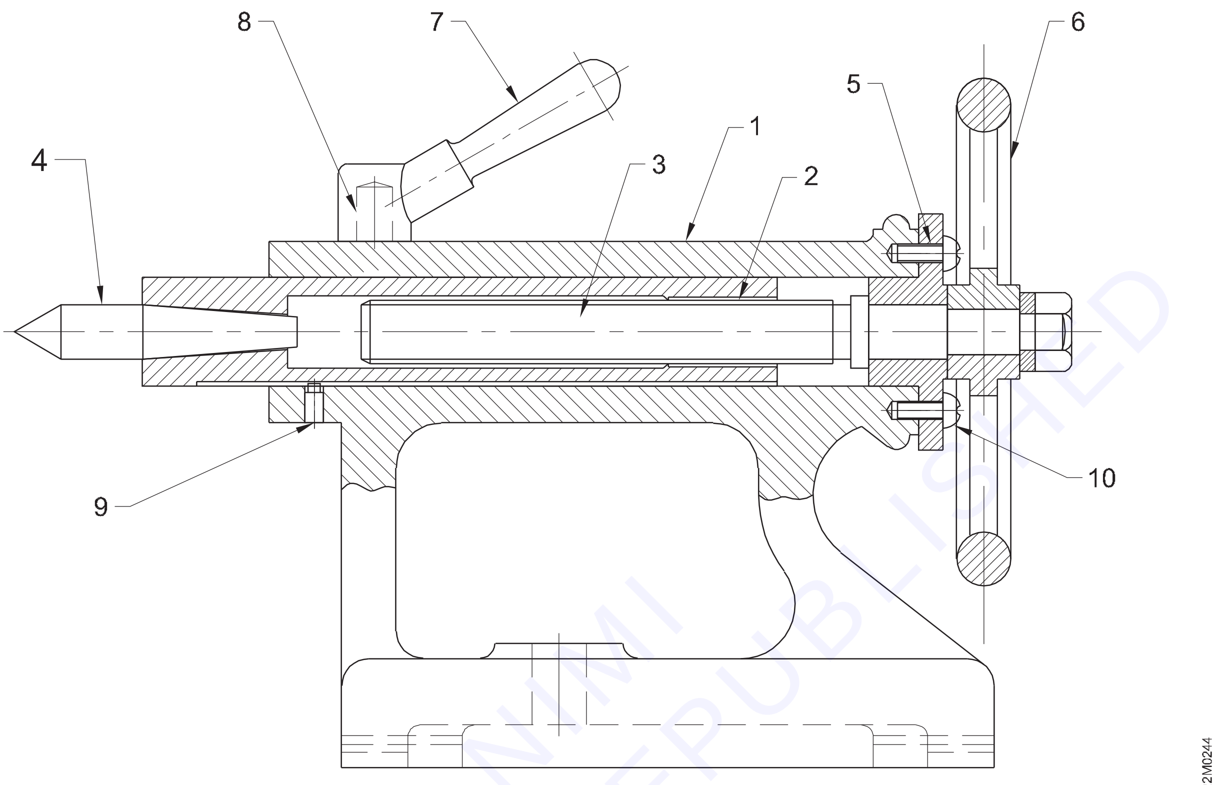
LATHE PART TAIL STOCK (DETAILS)

- Draw the details of lathe tail stock.
- Add left side elevation for the detail No.5 (Spindle bearing).
- Select suitable scale.
- Incorporate all the required dimensions.
- Prepare the part list.

Note: Screw detail not shown. Assume as per dimensions

TASK 2: **Draw the assembly of the tail stock drawing using by Auto CAD**

Fig 1



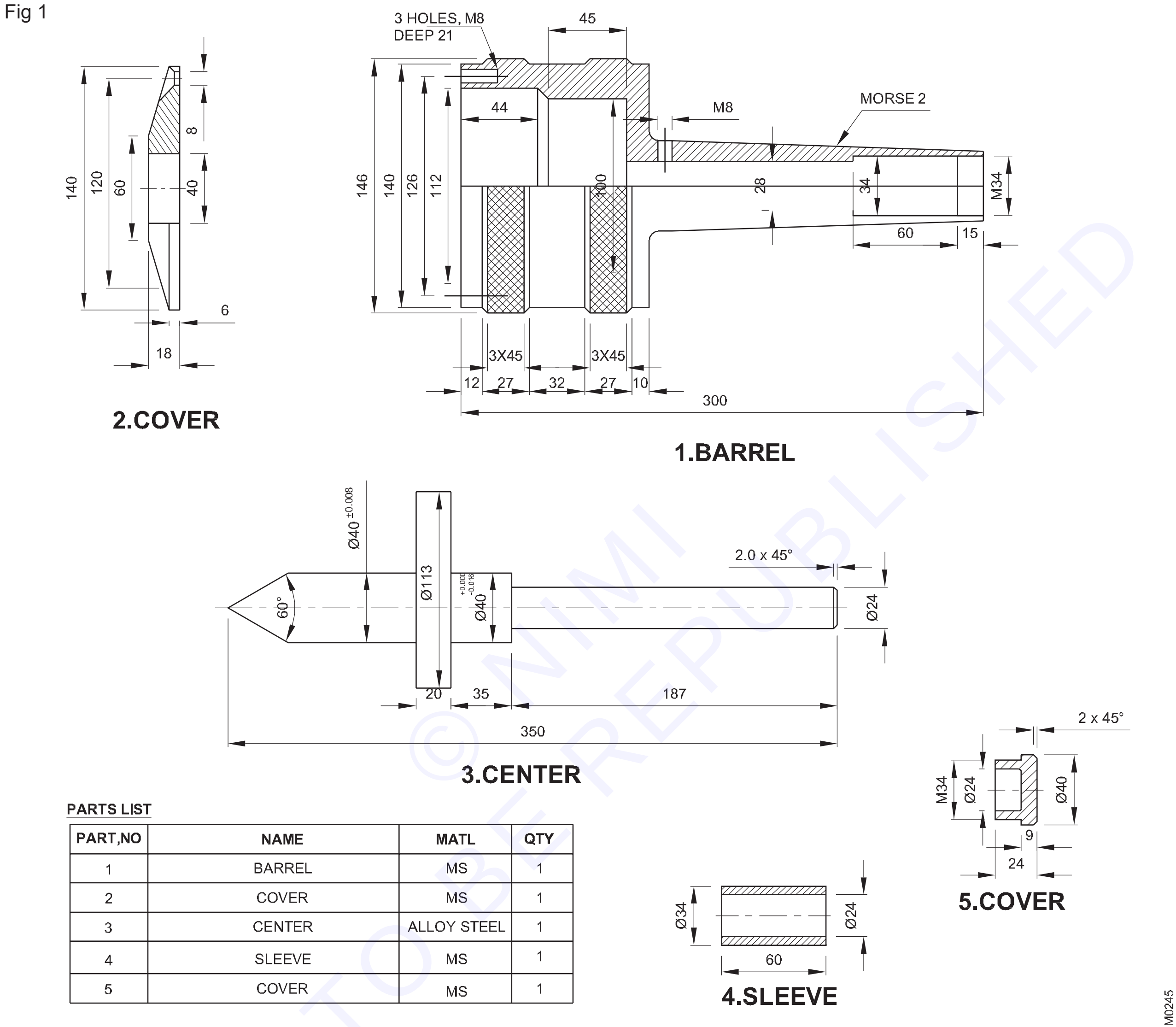
LATHE PART TAIL - STOCK ASSEMBLY

Drawing of a lathe part - revolving centre

TASK 1 : **Draw the details of the revolving center**

- Draw the details of the revolving centre. (Fig 1)

Fig 1



PARTS LIST

PART,NO	NAME	MATL	QTY
1	BARREL	MS	1
2	COVER	MS	1
3	CENTER	ALLOY STEEL	1
4	SLEEVE	MS	1
5	COVER	MS	1

TAIL STOCK- REVOLVING CENTRE (DETAILS)

- Select suitable scale.
- Incorporate all the required dimensions.
- Draw the radial bearing refering standard manufacturing manual.
- Draw the screw M8x21 as per IS standard.
- Prepare the part list.

— — — — — — — —

TASK 2: **Assemble the parts of revolving centre as given below**

- Draw the assembled full sectional elevation of the revolving centre and side view from left.
- Give all dimensions.
- Indicate part no as per part list.

Hint to Assembly

- Draw the full sectional elevation of the barrel (1).
- Draw the radial ball bearing (6) and thrust ball bearing (7) in space provided in the barrel.
- Draw the cover (2) and fix it with M8x21 screw (8).
- Insert the centre (3) through the hole 40 of cover plate.
- On the right side of the barrel draw the sleeve (4) inside the hole.
- On the right side and draw the cover (5).
- Complete the drawing and draw the sectional lines.
- Give overall dimensions.
- Indicate part No as per part list.

Detail drawing and Assembly drawing

classes of machine drawing: Preparation of any type of engineering drawing is to communicate the idea of the designer/engineer to others in the industry or institution. The preparation of pictorial and multi-view sketches/drawings are discussed in the earlier exercises. Now we can discuss the two main classes of machine drawing. They are

– detail drawing

– assembly drawing.

Detail drawing

A detail drawing provides the following information concerning the manufacture/fabricate the part/parts of an assembly of component/machine/structure.

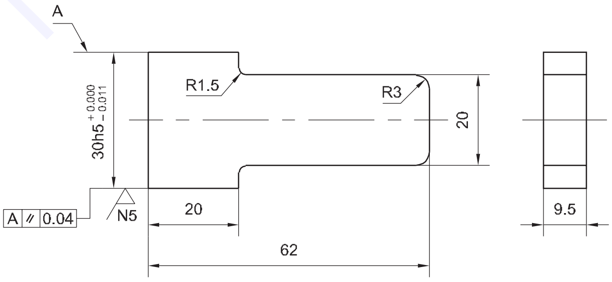
Shape description: The shape of a part of an assembly is expressed by drawing the part in orthographic views.

Size description: The size of the part is expressed by one of the methods of dimensioning technique on the orthographic views.

Manufacturing process: Manufacturing process may be placed directly on the surfaces of the part drawing or indirectly expressed by placing the roughness value or by deviation of the size. (Fig 1)

Heat treatment: Hardening, tempering etc required to be given to the part also indicated in the drawing.

Fig 1



NOTE: USE THIS JOB FOR CHECKING SNAP GAUGE (CNC-2-05)

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Bill of material: Number of part, name of part, material etc are to be placed in the drawing paper. (Fig 2)

Fig 2

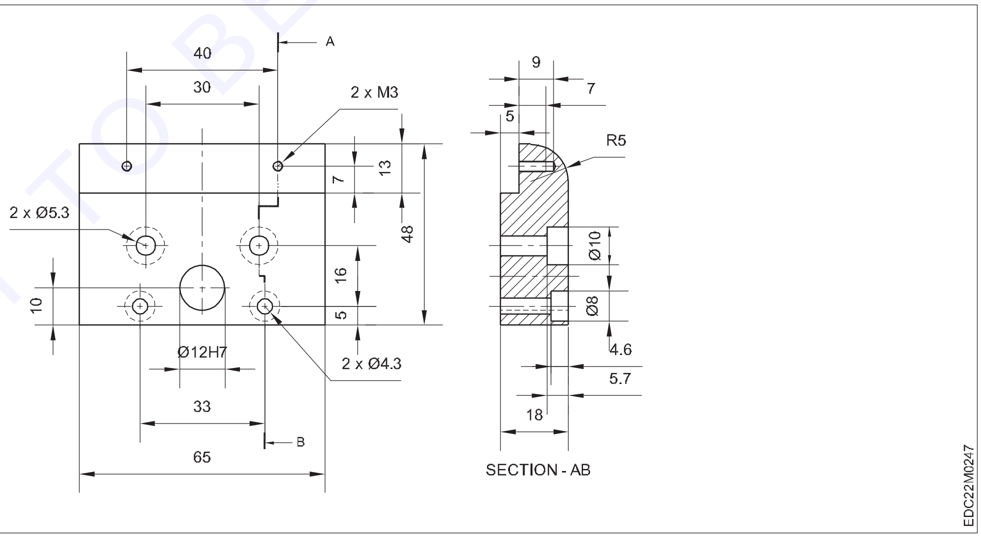
19	2	Washer	OD12xID6.5x2thk		Std
18	1	Cylindrical pin	4h8 x 16	IS:2393	Std
17	4	Snap head rivet	6 x 15	IS:2155	Std
16	4	Slotted CSK head screw	BM4 x 14	IS:6761	Std
15	4	Slotted cheese headscrew	BM3 x 25	IS:1366	Std
14	2	Hex. Socket head screw	M4 x 25	IS:2269	Std
13	2	Hex. Socket head screw	M5 x 25	IS:2269	Std
12	1	Locking coltar	O18.12	Fe310	BF Ex.37
11	1	Guide nut	O25.30	Fe310	BF Ex.37
10	1	Vice screw rod	O20.152	Fe310	BF Ex.37
9	1	Vice handle	O8.115	Fe310	BF Ex.37
8	1	Clamping pad	ISST35 x 2.35	Fe310	BF Ex.37
7	1	Clamping screw	O8.175	Fe310	BF Ex.37
6	2	Jaws	20 ISF 10.67	40C8	BF Ex.36
5	1	Sliding pieces	40 ISF 25.115	Fe310	BF Ex.34
4	1	Clamping jaw	<6545 x 8.70	Fe310	BF Ex.35
3	1	Clamping jaw	<7070 x 8.70	Fe310	BF Ex.35
2	1	Movable jaw	50 ISF 20.70	Fe310	BF Ex.33
1	1	Fixed jaw	50 ISF 30.70	Fe310	BF Ex.32
Part No.	No off	Description	Stock size	Material	Remarks

Explanatory work: General/specific note may also be given separately along with the drawing if required.

Detail drawings are drawn on paper by the following two methods according to the industry.

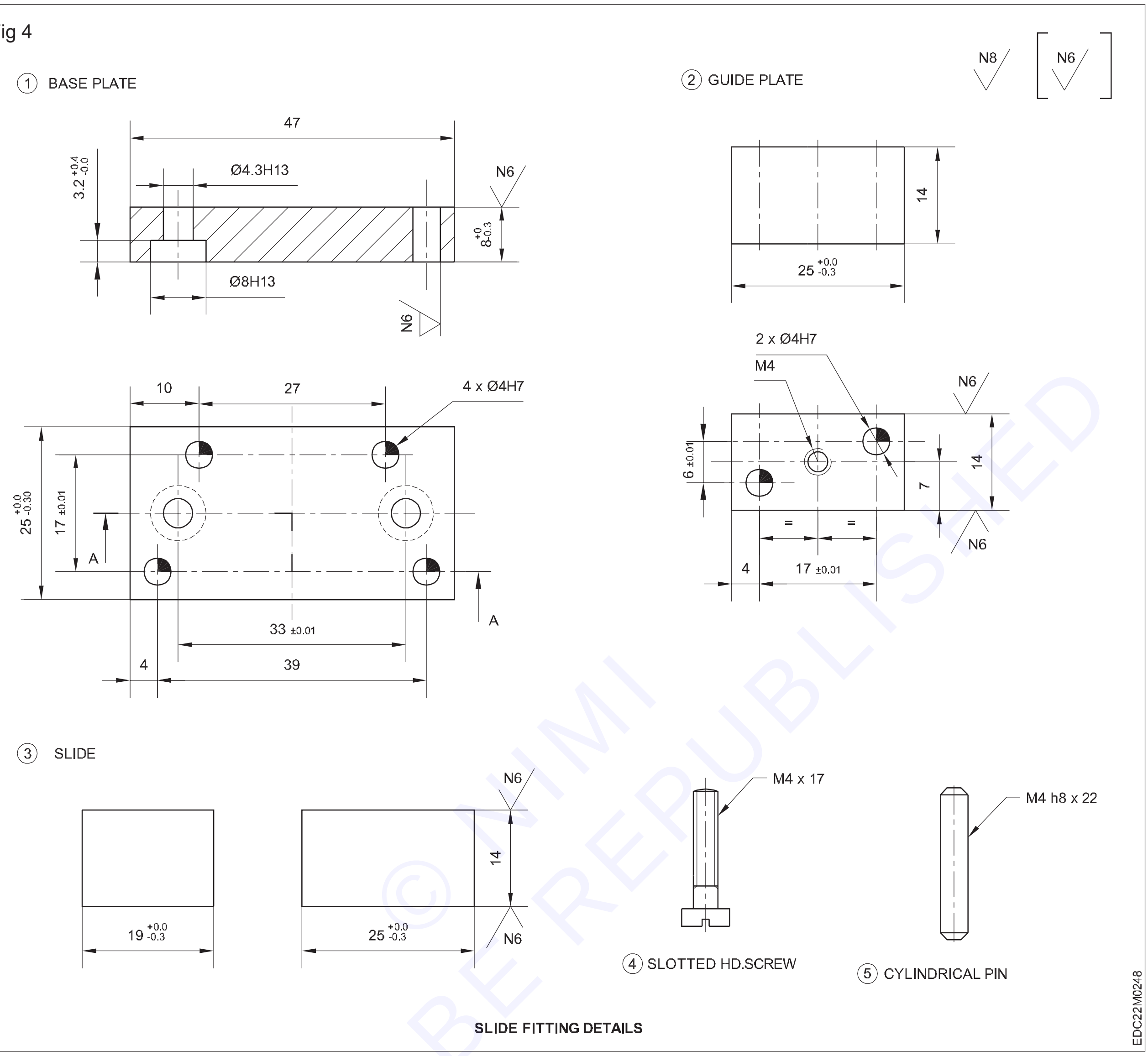
- Detail drawing of a part of an assembly drawn on a single paper separately with all information related to the part. A number of detail drawing sheets are prepared according to the number of parts of the assembly. (Fig 3)

Fig 3



- Detail drawings of all the parts of the assembly are drawn on a single sheet with detailed bill of material list etc. (Fig 4)

Fig 4



SLIDE FITTING DETAILS

Generally while making detail drawing of an assembly, parts like screw, bolt, nut, dowel, cotter, key, washer etc are given in the material list with specification of the particular part Fig2 Part No.13 to 19. In some cases above said parts are shown by a view with dimension as per Indian Standard. (Fig 4 part 4 & 5)

Assembly drawing

A drawing which shows the parts of a job or machine or structure assembled in their respective relative working position is called assembly drawing.

Different types of assembly drawings are available according to their nature and use. They are

- Design assembly drawing
- General assembly drawing
- Unit/Sub-assembly drawing
- Installation assembly drawing

Among the above said types of assembly drawings, we will be going to consider mainly general assembly drawing and unit/sub-assembly drawing.

Following points are to be considered while making assembly drawing.

- Draw only minimum number of views required to understand the assembly.
- Section the view/views according to the requirement.
- Provide minimum necessary dimensions which will help us to arrive at the overall size of the assembly.
- Label all the parts of the assembly.

Draw the detail and assembly drawings on separate A3 drawing paper.

Abbreviations: Most of the blue prints have some abbreviations. The use of abbreviations which help to reduce drafting time, save the valuable drawing space etc. A selection of the more commonly used ones are given below:

Term	Abbreviations/Symbol
Across Corners	A/C
Across Flats	A/F
Alteration	ALT
Ampere	A
Angle (Structural section)	L
Approved	APPD
Approximately equal to	≈
Assembly	Assy
Auxiliary	AUX
Beam	I
Brake horse power	bhp
Brinell hardness	HB
Cast Iron	CI
Centimetre	Cm
Centres	CRS
Checked	CHD
Chamfered	CHAM
Cheese head	CH. HD
Centre line	CL, ∉
Centre to centre	C to C
Channel	]
Circular pitch	CP
Circumference	O^{ce}
Counterbore	C' BORE
Countersunk	CSK
Cylindrical	CYL
Degree	°
Degree celsius	°C
Degree Kelvin	K
Diameter	DIA
Diameter Preceding Dimension	ϕ
Dimension	DIM
Drawing	DRG
Figure	FIG
Gram	g
Ground Level	GL

Term	Abbreviations/Symbol
Hexagon	HEX
Horizontal	HORZ
Horse power	hp
Hour	h
Indian Standard	IS
Inspection	INSP
Inside diameter	ID
Internal	INT
Indicated horse power	ihp
Joule	J
Kilogram	kg
Kilolitre	kl
Kilometre	km
Kilometre per hour	km/h
Kilogram force	kgf
Left hand	LH
Litre	l
Litres per minute	l/min
Long	LG
Material	MATL
Machine	M/C
Manufacturing	MFG
Maximum	max
Metre	m
Mechanical	MECH
Micrometer (0.001 mm)	μm
Millimetre	mm
Minimum	min
Miscellaneous	MISC
Not to scale	NTS
Number	No.
Opposite	Opp.
Outside diameter	OD
Parallel	//
Perpendicular	⊥
Pitch circle diameter	PCD
Pitch circle	PC

Term	Abbreviations/Symbol
Pressure	p
Pattern number	PATT NO.
Quantity	QTY
Radius	R
Reference	REF
Required	REQD
Revolution per minute	rpm
Rockwell hardness `A' scale	HRA
Rockwell hardness `B' scale	HRB
Rockwell hardness `C' scale	HRC
Right hand	RH
Round head	RD.HD
Second of angle	"
Serial number	SL.No.
Second of time	S
Spot face	SF
Screwed	SCR

Term	Abbreviations/Symbol
Square	SQ,
Standard	STD
Square centimetre	Cm^2
Square metre	m^2
Square millimetre	mm^2
Symmetrical	SYM
Standard wire gauge	SWG
Tee (Structural section)	T
Threads per inch	TPI
Temperature	TEMP
Time	t
Tanne	t
Undercut	U' CUT
Volt	V
Volume	VOL
Vickers hardness	HV
Weight	W

Surface quality

Various components are manufactured by different machining processes. The surfaces of the components differ in their appearance as well as `feel' when we move our hand over the surface. (Fig 1)

The surface will have ups and downs. These ups and downs are due to the tool marks. The pattern of these tool marks depends on the machining processes. The irregular patterns of tool marks depend on the feed, speed, tool angles, depth of cut etc. So all the machined surfaces are rough due to the inherent tool marks left in the machining processes. The surface appearance of components is shown in Figs 2 to 4.

Fig 1

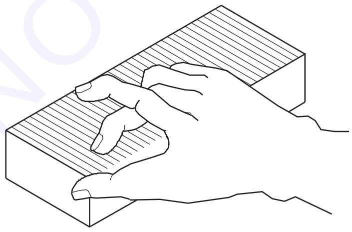
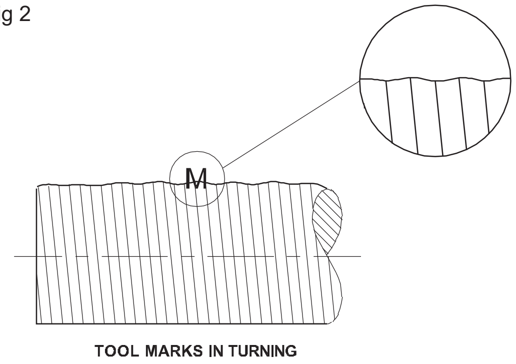


Fig 2



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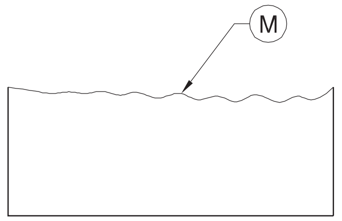
In other words, the selection process and setting of machining parameters are dictated by the type of surfaces quality demanded in the drawing of the part.

Surface roughness measurement

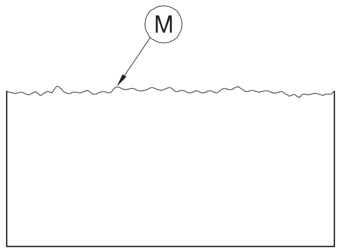
To control the roughness of a surface precisely, we need to define and establish a measuring system for it.

Fig 3



TOOL MARKS IN MILLING

Fig 4



TOOL MARKS IN SHAPING. PLANING, SLOTTING

Roughness is defined as the average height or depth from the hill to the valley of a surface pattern (Fig 5) and it is possible to measure this by instruments specially designed for this purpose.

This instrument has a very sharp stylus. (Fig 6) This stylus is moved across the surface to be measured mechanically over a short distance and during this time the instrument calculates the average depth and displays the value as a roughness number.

Fig 5

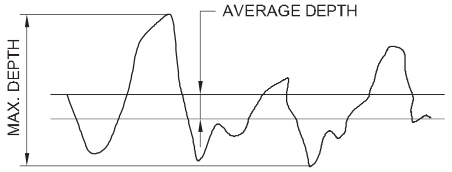
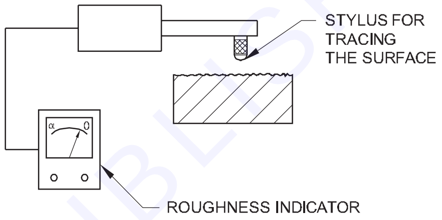


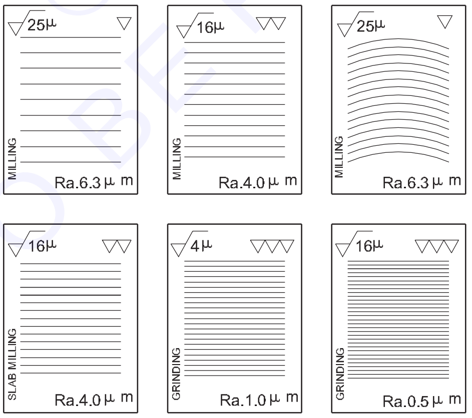
Fig 6



Surface finish standard

One method of determining the surface roughness is by using a surface finish standard. (Fig 7) This is a box which consists of 20 blocks of a specific surface finish obtained by a specific machining operation.

Fig 7



The type of machining operation is marked on each block together with the surface roughness number for height and width. Using the surface finish standard, we can make comparisons between the machined surface and the standard surface using our sense of touch. (Fig 1)

However, this method is sometimes not accurate enough and the individual must be very sensitive to the different surface roughness.

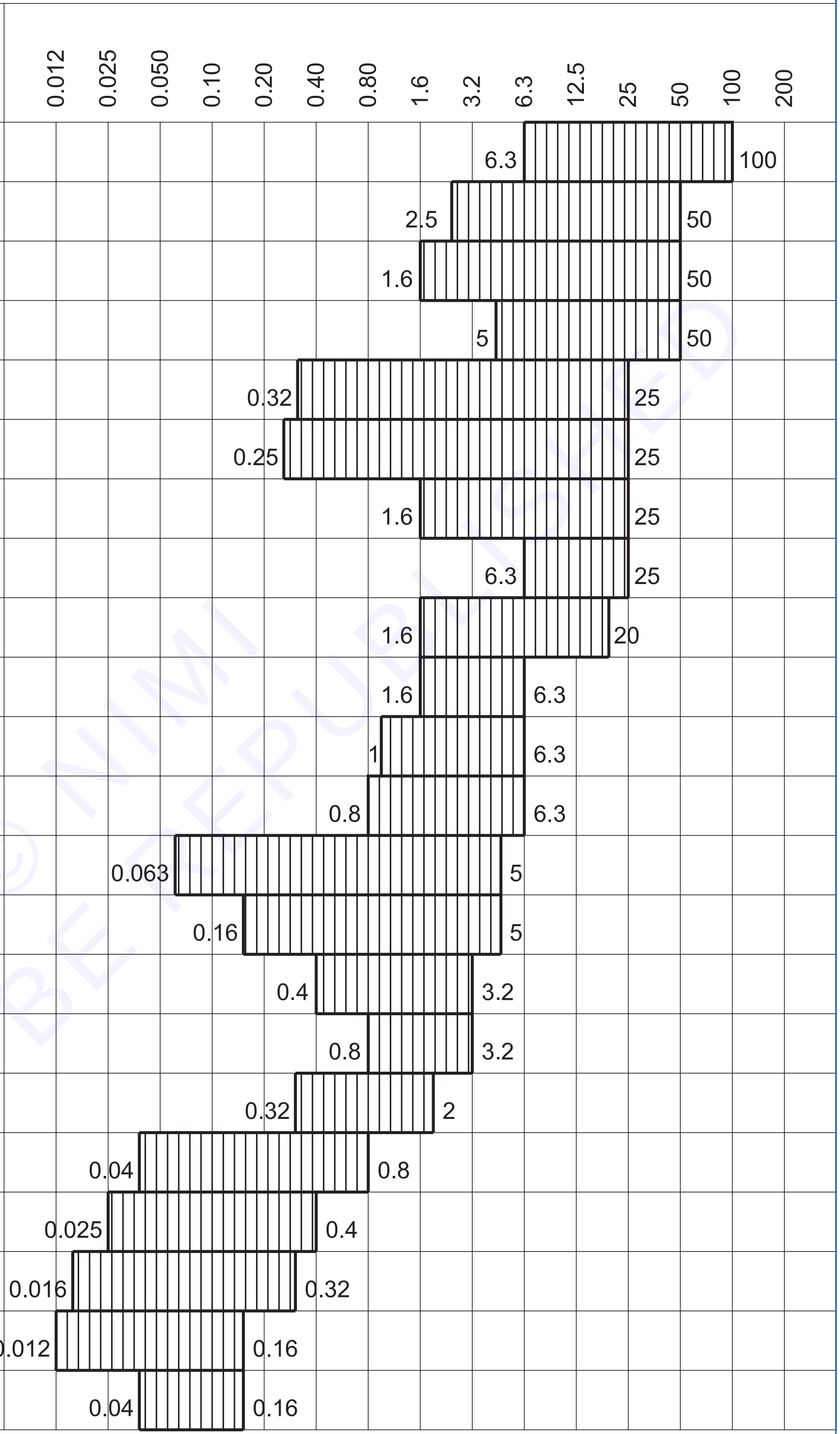
If the degree of accuracy of checking is high, then the application of a sensitive instrument is inevitable.

In order to obtain the required surface quality, it is necessary to choose the appropriate manufacturing process. Table 1 appended here gives an idea about the different processes and range of surface quality attainable.

For more detailed information on surface texture, symbols and their representations refer to IS:10719.

TABLE 1

Surface roughness expected from manufacturing processes IS : 3073 - 1967



Manufacturing process		
Flame cutting, sawing and chipping	6.3	100
Hot rolling	2.5	50
Planing	1.6	50
Sand casting	5	50
Turning and milling	0.32	25
Filing	0.25	25
Disc grinding	1.6	25
Hand grinding	6.3	25
Drilling	1.6	20
Boring	1.6	6.3
Radial cut-off sawing	1	6.3
Permanent mould casting	0.8	6.3
Surface and cylindrical grinding	0.063	5
Extrusion	0.16	5
Reaming,broaching and jobbing	0.4	3.2
Die casting	0.8	3.2
High pressure casting	0.32	2
Burnishing	0.04	0.8
Honing	0.025	0.4
Super finishing	0.016	0.32
Lapping	0.012	0.16
polishing	0.04	0.16

Method of indicating surface roughness for general engineering drawing

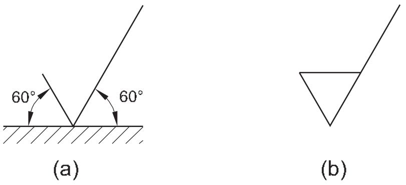
Scope: This section of the code specifies the symbols and additional indications of surface roughness.

Symbols used for indication of surface roughness

The basic symbol consists of two legs of unequal length inclined at approximately 60° to the line representing the surface under consideration as shown in Fig 1a.

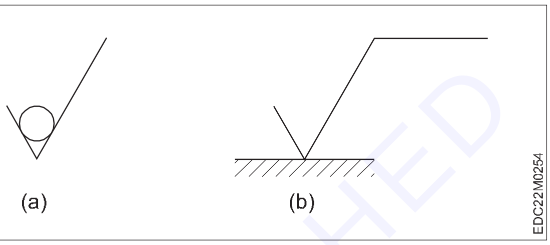
If the removal of material by machining is required, a bar is added to the basic symbol, as shown in Fig 1b.

Fig 1



(a) (b)

Fig 2



(a) (b)

If the removal of material is not permitted, a circle is added to the basic symbol, as shown in Fig 2a.

The symbol in Fig 2a may also be used in a drawing relating to a production process to indicate that a surface is to be left in the state resulting from a preceding manufacturing process, whether this state was achieved by removal of material or otherwise.

In this case none of the indications are added to the symbol.

When special surface characteristics have to be indicated a line is added to the longer leg of any of the above symbols, as shown in Fig 2b.

Indication added to the symbols

Indication of surface roughness

The value or values defining the principal criterion of roughness are added to the symbols given in Fig 3a,b,c.

Surface roughness specified

– As in Fig 3a, may be obtained by any production method.

– As in Fig 3b, shall be obtained by removal of material by machining.

– As in Fig 3c, shall be obtained without removal of material.

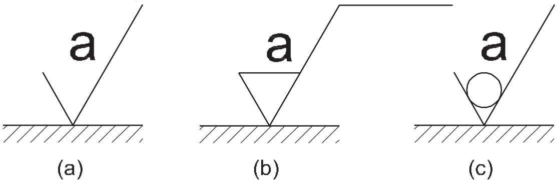
When only one value is specified it represents the maximum permissible value of surface roughness.

If it is necessary to impose maximum and minimum limits of the principal criterion of surface roughness, both values should be shown as in Fig 4, with the maximum limit a_1 above the minimum limit a_2.

For general application, roughness values are graded and we have 12 grades designated as N1, N2...N12.

The principal criterion of roughness R2 may be indicated by the corresponding roughness grade symbol as shown below:

Fig 3



(a) (b) (c)

Fig 4

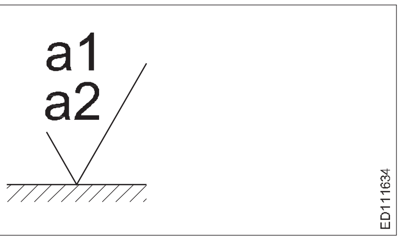


TABLE 1

Roughness Values R_2 µm	Roughness Grade symbols
50	N12
25	N11
12.5	N10
6.3	N9
3.2	N8
1.6	N7
0.8	N6
0.4	N5
0.2	N4
0.1	N3
0.05	N2
0.025	N1

The unit of roughness is a micron (0.001) and greater this number is higher the roughness. The roughness number described here is called the Ra value of the surface.

The surface roughness for a given surface is indicated by a symbol. (Fig 5) Here 1.3 means that the average depth of the tool marks is not to exceed 0.0013 mm. Any value less than this is acceptable. In case we need to control the maximum and minimum permissible value of roughness it is to be indicated as shown in Fig 6 & 7.

Fig 5

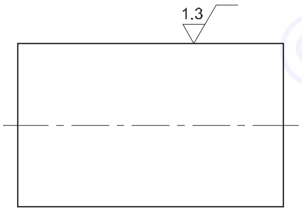
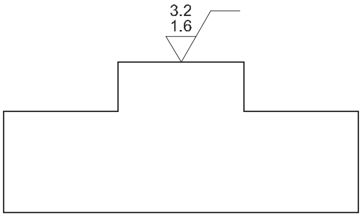
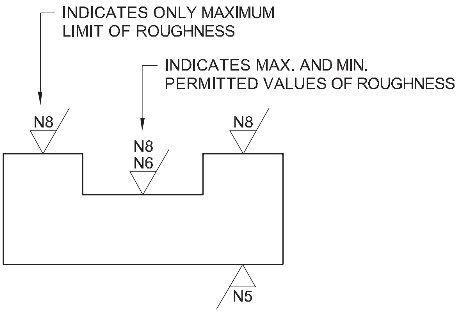


Fig 6



EDC22M0256

Fig 7



INDICATION OF ROUGHNESS USING ROUGHNESS GRADES

EDC22M0257

Roughness symbols

These symbols indicate the practice followed in the industry and are given here for guidance only. B.I.S 3073-1967 (Table 2) states this as assessment of surface roughness.

TABLE 2

ROUGHNESS GRADE	ROUGHNESS SYMBOL
N12	∼
N11 N10	▽
N9 N8 N7	▽▽
N6 N5 N4	▽▽▽
N3 N2 N1	▽▽▽▽

Indication of special surface roughness characteristics.

In certain circumstances, for functional reasons, it may be necessary to specify additional special requirements concerning surface roughness.

If it is required that the final surface roughness be produced by one particular production method, this method should be indicated in plain language on an extension of the longer leg of the symbol shown in Fig 8.

Also on this extension line should be given any indications relating to treatment or coating.

Unless otherwise stated, the numerical value of the roughness applies to the surface roughness after treatment of coating.

If it is necessary to define surface roughness both before and after treatment, this should be explained in a suitable note or in accordance with Fig 9.

If it is necessary to indicate the sampling length, it should be selected from the series given in IS:3073-1967 `Assessment of surface roughness', and be stated adjacent to the symbol, as shown in Fig 10a.

If it is necessary to control the direction of lay, it is specified bya symbol (see Table 3) added to the surface roughness symbol as shown in Fig 10b.

Note: The direction of lay is the direction of the predominant surface pattern, ordinarily determined by the production method employed.

Symbols for the direction of lay: The series of symbols for the common directions of lay are specified in Table 3.

Fig 8

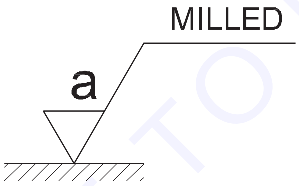
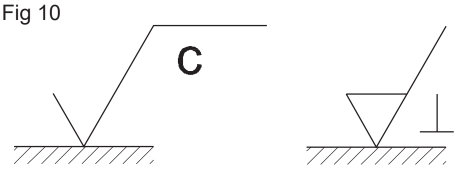


Fig 10



(a)

(b)

EDC22M0259

Fig 9

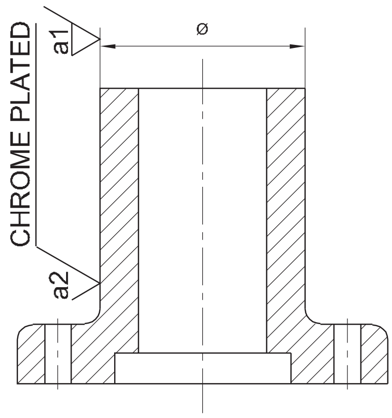


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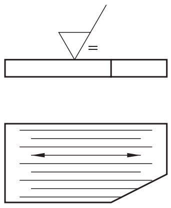
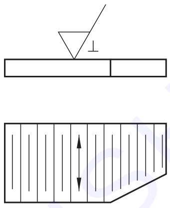
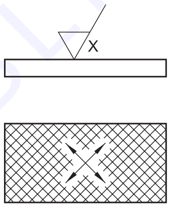
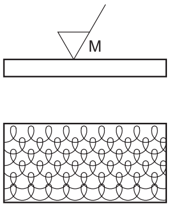
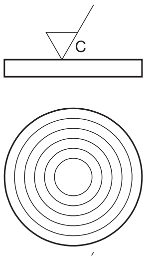
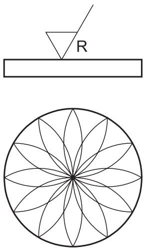
TABLE 3

Symbols for Direction of lay

SYMBOL	INTERPRETATION	
=	Parallel to the plane of projection of the view in which the symbol is used.	
⊥	Perpendicular to the plane of projection of the view in which the symbol is used.	
X	Crossed in two slant directions relative to the plane of projection of the view in which the symbol is used.	
M	Multi-directional	
C	Approximately circular relative to the centre of the surface to which the symbol is applied.	
R	Approximately radial relative to the centre of the surface to which the symbol is applied.	

Note: It should be necessary to specify a direction of lay not clearly defined by these symbols then this shallbe achieved by a suitable note on the drawing.

Indication of machining allowance: Where it is necessary to specify the value of the machining allowance, this should be indicated on the left of the symbol as shown in Fig 11. This value should be expressed in millimetres, according to the general system used for dimensioning the drawing.

Position of the specification of surface roughness in the symbol: The specifications of surface roughness should be placed relative to the symbol as shown in Fig 12.

Fig 11

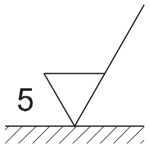
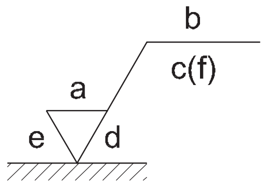


Fig 12



a = Roughness valuve R_a in micrometres
or
Roughness grade symbol N1 to N12

b = Production method, tretment or coating

c = Sampling length

d = Direction of lay

e = Machining allowance

f = Other roughness values (in brackets)

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Indication on drawings: The symbol as well as the inscriptions, should be orientated such that they may be read from the bottom or the right-hand side of the drawing. (Fig 13)

If it is not practicable to adopt this general rule, the symbol may be drawn in any position, but only provided that it does not carry any indications of special surface roughness characteristics or of machining allowances. Nevertheless, in such cases the inscription defining the value of the principal criterion of roughness (if present) shall always be written in conformity with the general rule. (Fig 14)

Fig 13

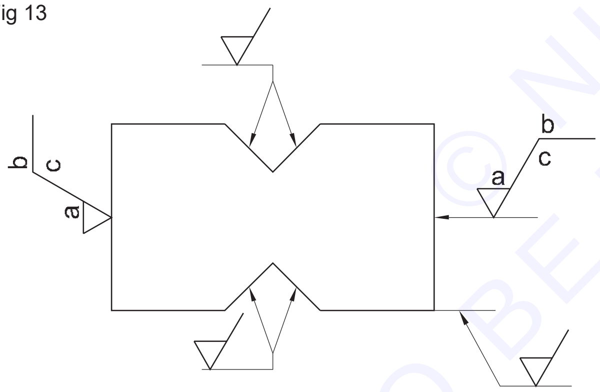
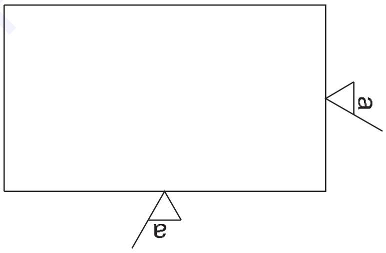


Fig 14



EDC22M0263

If necessary, the symbol may be connected to the surface by a leader line terminating in an arrow.

The symbol or the arrow should point from outside the material of the part, either to the line representing the surface or to an extension of it.

In accordance with the general principles of dimensioning, the symbol is only used once for a given surface, if possible, on the view which carries the dimension defining the size or position of the surface. (Fig 15)

If the same roughness is required on all the surfaces of a part, it is specified:

- Either by a note near a view of the part (Fig 16a) near the title block or in the space devotated to general notes; or
- Following the part number on the drawing. (Fig 16b)

If the same surface roughness is required on the majority of the surfaces of a part, it is specified as in Fig 16 with the addition of

Fig 15

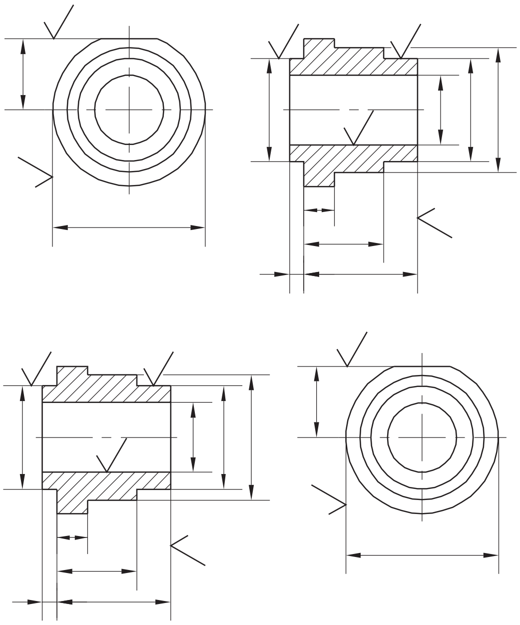
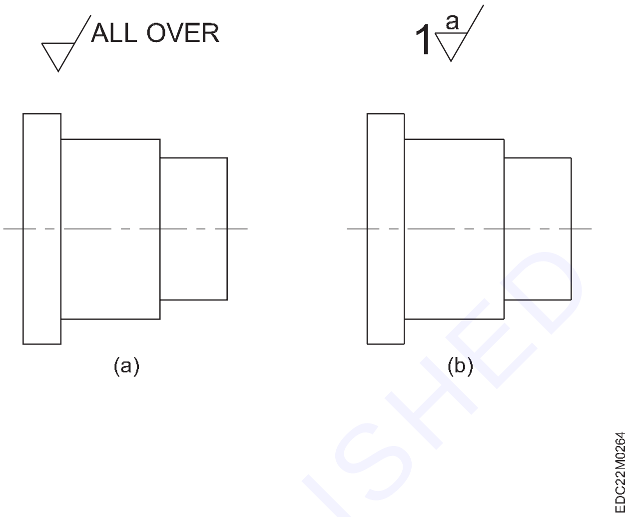


Fig 16



- the notation except where otherwise stated in Fig 17.
- a basic symbol (in brackets) without any other indication Fig 18.

Fig 17

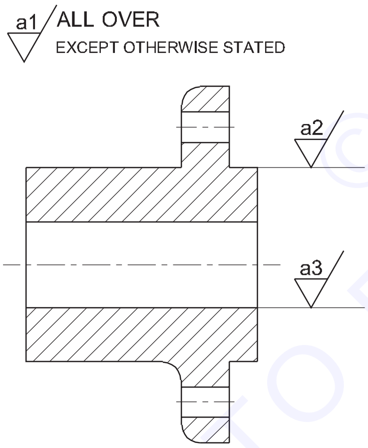
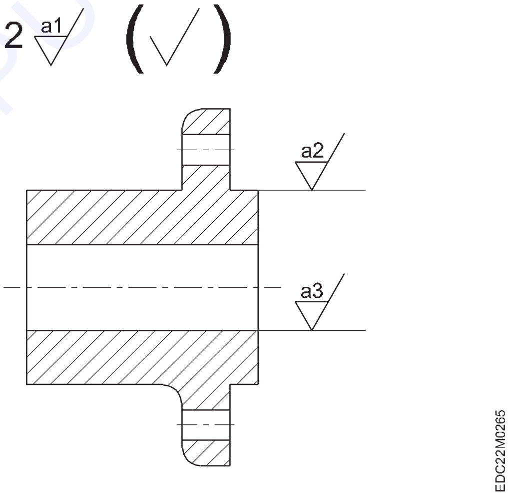


Fig 18



- the symbol or symbols (in brackets) of the special surface roughness or roughnesses. (Fig 19)

The symbols for the surface roughness which are exceptions to the general symbol are indicated on the corresponding surfaces.

To avoid the necessity of repeating a complicated specification a number of times, or where space is limited, a simplified specification may be used on the surface, provided that its meaning is explained near the drawing of the part, near the title block or in the space devoted to general notes. (Fig 20)

If the same surface roughness is required on a large number of surfaces of the part, one of the symbols shown in Fig 1,2 & 3a may be used on the appropriate surfaces and its meaning given on the drawing, for example, as shown in Fig 21a,b & c.

Fig 19

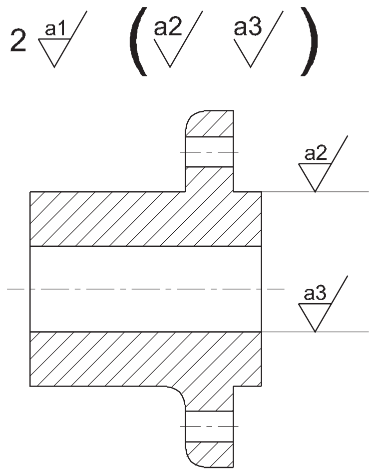
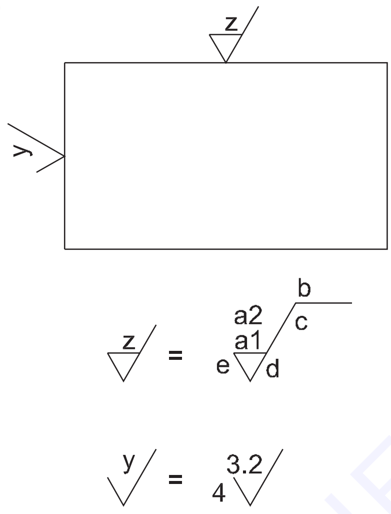
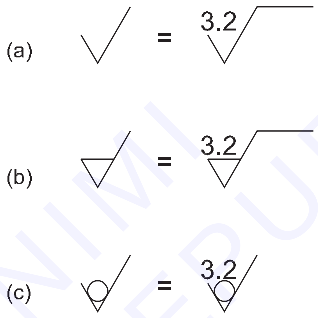


Fig 20



EDC22M0266



EDC22M0267

Important notes

The specification of surface roughness is unnecessary whenever the ordinary manufacturing processes by themselves ensure an acceptable surface finish.

Illustrative examples

Examples incorporating the surface roughness values, production method, sampling length, direction of lay and machining allowance are given below as illustration. The values quoted are purely typical and illustrative.

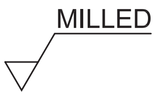
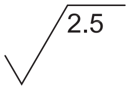
Symbols with inscription

TABLE 4

Symbol	Meaning
	Basic symbol may only be used alone when it meaning is explained by a note.
	A machined surface with no indication of any detail.
	A surface from which the removal of material is prohibited. This symbol may also be used in the drawing relating to production process to indicate that a surface is to be left in the state resulting from a preceeding manufactu-ring process, whether this state was achieved by removal of material or otherwise.

Symbols with Additional indications: These may be used singly, in combination or combined with an appropriate symbol given in Table 6.

TABLE 6

Symbol	**Meaning**
	Production Method: Milled
	Sampling length 2.5 mm
	Direction of lay: Perpendicular to the plane of projection of the view.
	Machining allowance 2 mm

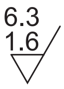
Simplified symbols

TABLE 7

Symbol	**Meaning**
	A note indicates the meaning of the symbol
 	A note indicates the meaning of the symbols.

Symbols with indication of the Principal criterion of roughness

TABLE 5

SYMBOL			MEANING
Removal of material by machining is			
Optional	Obligatory	Prohibited	
 or 	 or 	 or 	A surface with a maximum surface roughness value of 3.2 µm
 or 	 or 	 or 	A surface with a maximum surface roughness value of 6.3 µm and a minimum of 1.6 µm

Mass production and interchangeable manufacture

Mass Production

Mass production means production of a unit, component or part in large numbers.

Advantages of mass production

Time for the manufacture of components is reduced.

The cost of a component is reduced.

Spare parts can be made available quickly.

Gauges are used to check the components.

Even unskilled workers can be employed for checking.

Measuring time is saved.

Disadvantages of mass production

Special purpose machines are necessary.

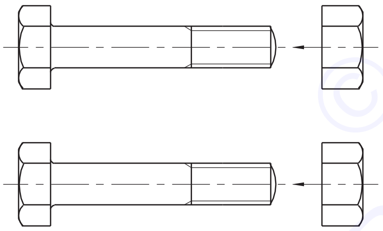
Jigs and fixtures are needed.

Gauges are to be used, hence the initial expenditure will be high.

Selective assembly

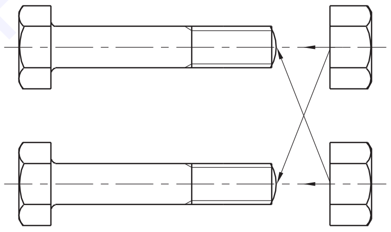
Figures 1 & 2 illustrate the difference between a selective assembly and a non-selective assembly. It will be seen in Fig 1 that each nut fits only one bolt. such an assembly is slow and costly and maintenance is difficult because spares must be individually manufactured.

Fig 1



SELECTIVE ASSEMBLY

Fig 2



NON-SELECTIVE ASSEMBLY INTERCHANGABILITY

Non-selectvie assembly

Any nut fits any bolt of the same size and thread type. Such an assembly is rapid and costs are reduced.

Maintenance is simpler because spares are easily available. (Fig 2)

Non-selective assembly provides interchangeability between the components.

In modern engineering production, i.e mass production there is no room for selective assembly. However in some special circumstances, selective assembly is still justified.

Interchangeability: When components are mass produced, unless they are interchangeable, the purpose of mass production is not fulfilled. By interchangeability, we mean that identical components, manufactured by different personnel under different environments, can be assembled and replaced without any rectification during the assembly stage are without affecting the functioning of the component when assembled.

Necessity of the limit system: If components are to be interchangeable, the need to be manufactured to the same identical size which is not possible, when they are mass-produced. Hence, it becomes necessary to permit the operator to deviate by a small margin from the exact size which he is not able to maintain for all the components. At the same time the deviated size should not affect the quality of the assembly. This sort of dimensioning is known as limit dimensioning.

A system of limits is to be followed as a standard for the limit dimensioning of components.

System of limits and fits are followed by countries based on the ISO (International Standard Organisation) specifications.

System of limits and fits followed in our country is that stipulated by the BIS (Bureau of Indian Standard)

Other systems of limits and fits

- British Standard System (BSS)
- German Standard (DIN)

The Indian standard system of limits and fits - Terminology

Size

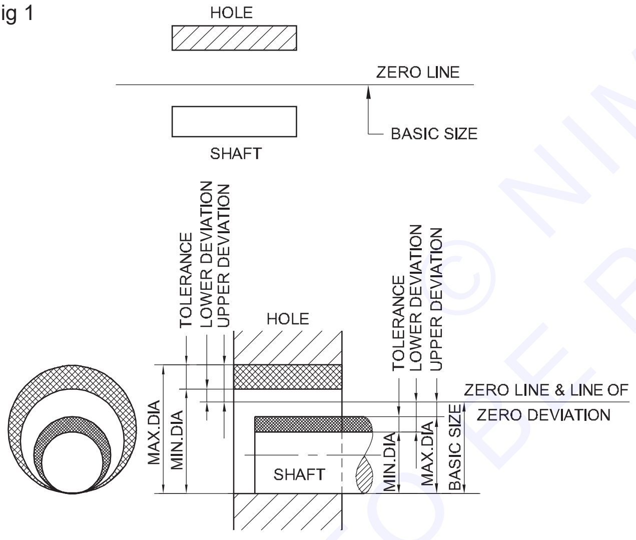
It is a number expressed in a particular unit in the measurement of length.

Basic size: It is the size based on which the dimensional deviations are given. (Fig 1)

Actual size: It is the size of the component by actual measurement after it is manufactured. It should lie between the two limits of size if the component is to be accepted.

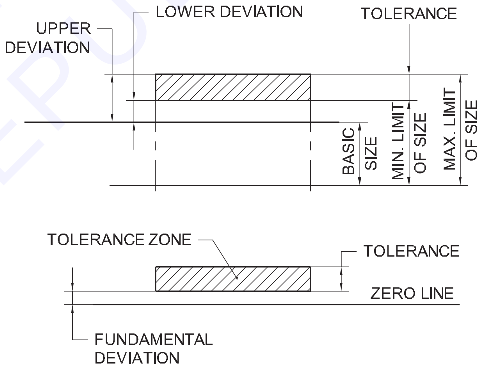
Limits of size: These are the extreme permissible sizes within which the operator is expected to make the component. (Maximum and minimum limits) (Fig 2)

Fig 1



SIMPLIFIED SCHEMATIC DIAGRAM OF CLEARANCE FIT

Fig 2



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Maximum limit of size: It is the greater of the two limits of sizes. (Fig 2) & (Table 1)

Minimum limit of size: It is the smaller of the two limits of sizes. (Fig 2) & (Table 1)

TABLE 1

(Examples)

Sl. No.	Size of component	Upper deviation	Lower deviation	Max. limit of size	Min. limit of size
1	+.008 20 -.005	+0.008	-0.005	20.008	19.996
2	+.028 20 +.007	+0.028	+-0.007	20.028	20.007
3	-.012 20 -.021	+0.012	-0.021	19.988	19.979

Hole: In the BIS system of limits and fits, all internal features of a component including those which are not cylindrical as designated as hole. (Fig 3)

Shaft: In the BIS system of limits and fits, all external features of a component including those which are not cylindrical as designated as shaft. (Fig 3)

Deviation: It is the algebraic difference between a size and its corresponding basic size. It may be positive, negative zero. (Fig 2)

Upper deviation: It is the algebraic difference between the maximum limit of size and its corresponding basic size. (Fig 2) (Table 1)

Lower deviation: It is the algebraic difference between the minimum limit of size and its corresponding basic size. (Fig 2) (Table 1)

The upper deviation is the deviation which gives the maximum limit of size. Lower deviation is the deviation which gives the minimum limit of size.

Actual deviation: It is the algebraic difference between the actual size and its corresponding basic size. (Fig 2)

Tolerance: It is the difference between the maximum limit of size and the minimum limit of size. It is always positive and is expressed only as a number without a sign. (Fig 2)

Zero line: In the graphical representation of the above terms, the zero line represents the basic size. This line is also called the line of zero deviation. (Figs 1 & 2)

Fundamental deviation: There are 25 fundamental deviations in the BIS system represented by letter symbols (capital letters for holes and small letters for shafts), i.e for holes - ABCD.....Z excluding I,L,O,Q and W. (Fig 4)

Fig 3

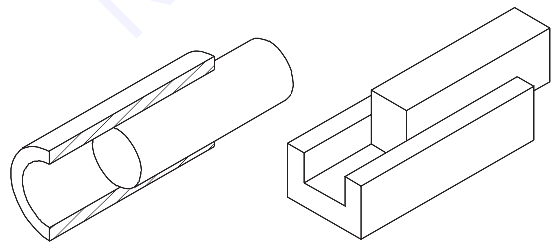
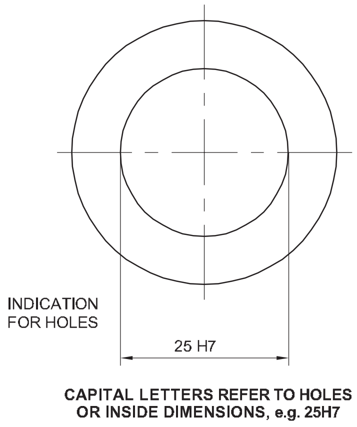


Fig 4



In addition to the above, four sets of letters JS, ZA, ZB and ZC are included.

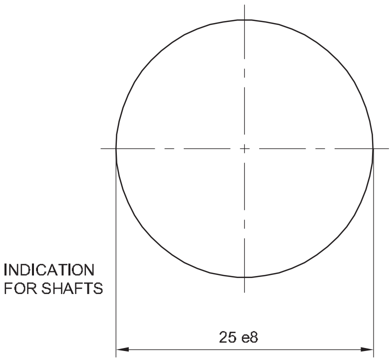
For shafts, the same 25 letter symbols but in small letters are used. (Fig 5)

The position of tolerance zone with respect to the zero line as shown in Figs 6 & 7.

The fundamental deviations are for achieving the different classes of fits. (Figs 8 & 9)

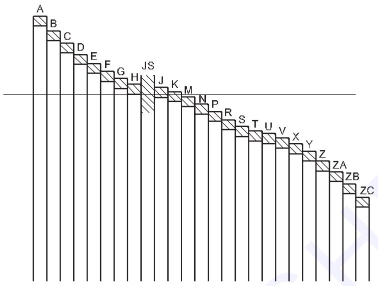
Fundamental tolerance: This is also called `grade of tolerance'. In the BIS system, there are 18 grades of tolerances represented by number symbols both for hole and shaft, denoted as IT01, IT0, IT1, IT2....IT16. (Fig 10)

Fig 5



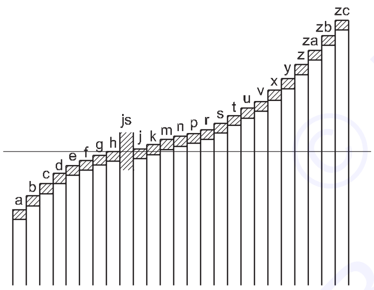
SMALL LETTERS REFER TO SHAFTS
OR OUTSIDE DIMENSIONS, e.g. 25 e8

Fig 6



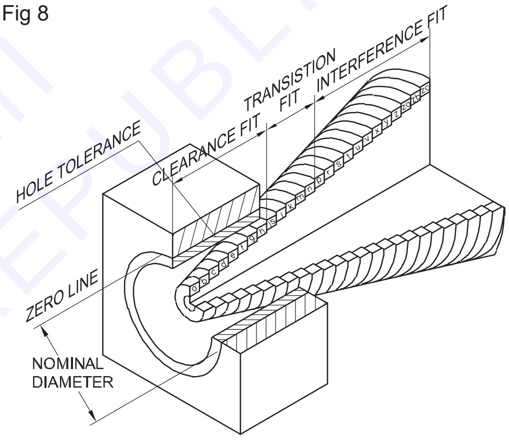
CAPITAL LETTERS INDICATE FUNDAMENTAL DEVIATION FOR HOLES

Fig 7



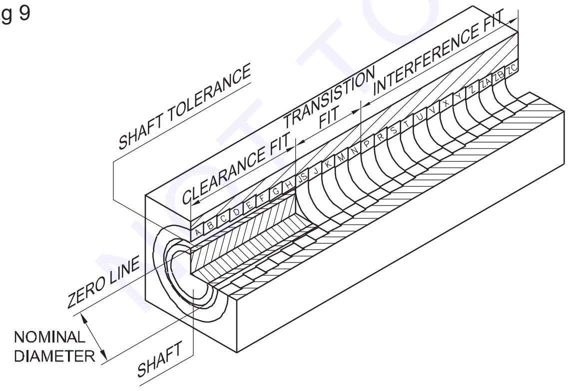
SMALL LETTERS INDICATE FUNDAMENTAL DEVIATION FOR SHAFTS

Fig 8



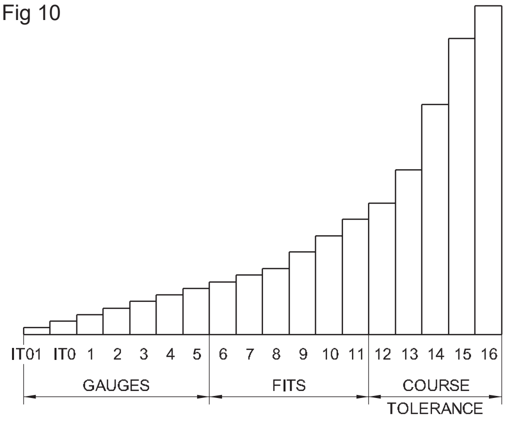
HOLE BASIS

Fig 9



SHAFT BASIS

Fig 10



A higher number gives a larger tolerance.

Grade of tolerance refers to the accuracy of manufacture of component.

In a standard chart, the upper and lower deviations for each combination of fundamental deviation and fundamental tolerance are indicated for sizes ranging upto 500 mm. (Refer to IS:919).

Tolerance size: This includes the basic size, the fundamental deviation and grade of tolerance.

Examples

25 H7 - is the tolerance size of a hole whose basic size is 25. The fundamental deviation is represented by the letter symbol H and the grade of tolerance is representated by the number symbol 7. (Fig 11)

25 e8 - is the tolerance size of a shaft whose basic size 25. The fundamental deviation is represented by the letter symbol e and the grade of tolerance is represented by the number 8. (Fig 12)

Fig 11

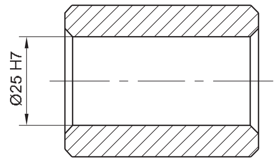
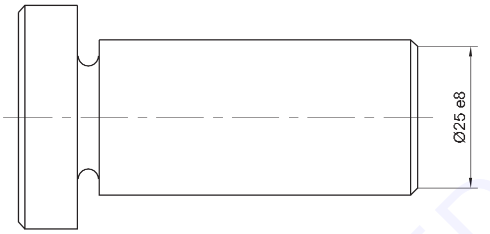


Fig 12



A very wide range of selection can be made by the combination of the 25 fundamental deviations and the 18 grades of tolerances.

Example

In figure 13, a hole is shown as 25 ± 0.2 which means that 25 mm is the basic dimension and ± 0.2 is the deviation.

As pointed out earlier, the permissible variation from the basic dimension is called `DEVIATION'.

The deviation is mostly given on the drawing with dimensions.

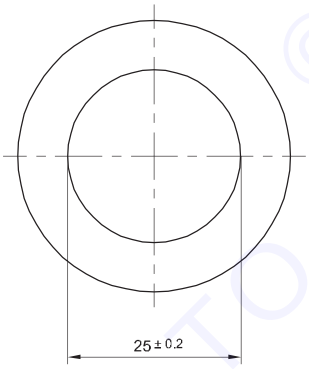
In the example, 25 ± 0.2, ± 0.2 is the deviation of the hole of 25 mm diameter. (Fig 13) This means that the hole is of acceptable size, if its dimension is between

25 + 0.2 = 25.2 mm

or 25 - 0.2 = 24.8 mm.

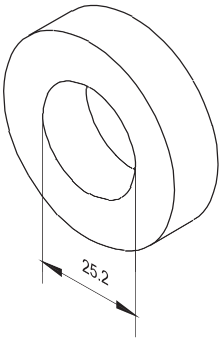
25.2 mm is the maximum limit. (Fig 14)

Fig 13



TOLERANCED SIZES OF HOLES

Fig 14



24.8 mm is the minimum limit. (Fig 15)

The difference between the maximum and minimum limit of size is the TOLERANCE. Tolerance here is 0.4 mm. (Fig 16)

Fig 15

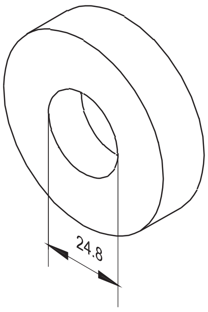
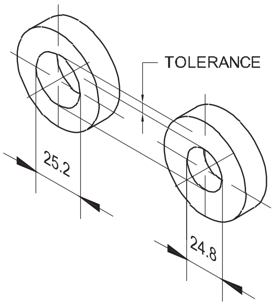


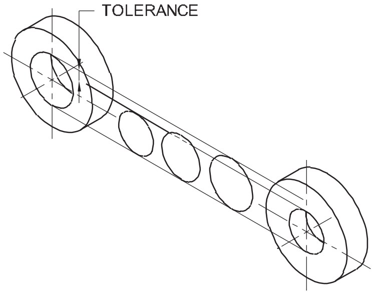
Fig 16



All dimensions of the hole within the tolerance zone are of an acceptable size as shown in Fig 17.

As per IS:696, while dimensioning the components as a drawing convention, the deviations are expressed as tolerances.

Fig 17



The Indian standard system of limits and fits - Terminology

Fit: It is the relationship that exists between two mating parts, a hole and a shaft, with respect to their dimensional differences before assembly.

Expression of a fit: A fit is expressed by writing the basic size of the fit first, (the basic size which is common to both the hole and the shaft) followed by the symbol for the hole and the symbol for the shaft.

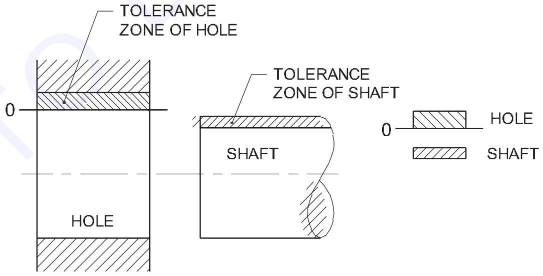
Example

30 H7/g6 or 30 H7 - g6 or $30\,\frac{H7}{g6}$

Clearance: In a fit the clearance is the difference between the size of the hole and the size of the shaft, when the hole is bigger than the shaft. (Fig 1)

Clearance fit: It is a fit which always provides clearance. Here the tolerance zone of the hole will be above the tolerance hole of the shaft. (Fig 1)

Fig 1



Example

$20\,\frac{H7}{g6}$

With the fit given, we can find the deviations from the chart.

For a hole 20 H7 we find from Table +21.

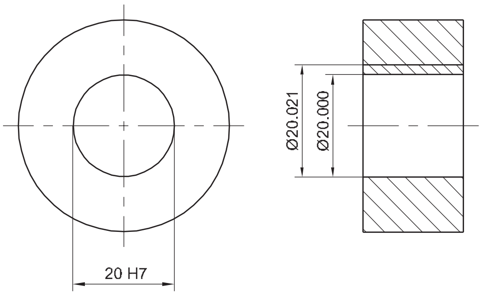
These numbers indicate the deviations in microns. (1 micrometre = 0.001 mm)

The limits of the hole are 20 + 0.021 = 20.021 mm and 20 + 0 = 20.000 mm. (Fig 2)

For a shaft 20 g6 we find from the Table - 7

- 20

Fig 2

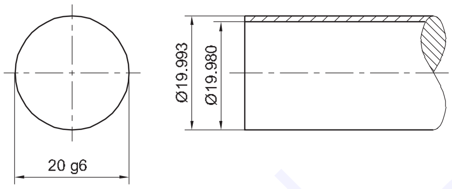


So the limits of the shaft are

20 - 0.007 = 19.993 mm

and 20 - 0.020 = 19.980 mm. (Fig 3)

Fig 3



Maximum clearance: In a clearance fit or transition fit, the maximum clearance is the difference between the maximum size hole and the minimum size shaft. (Fig 4)

Minimum clearance: It is the clearance fit, the minimum clearance is the difference between the minimum hole and the maximum shaft. (Fig 4)

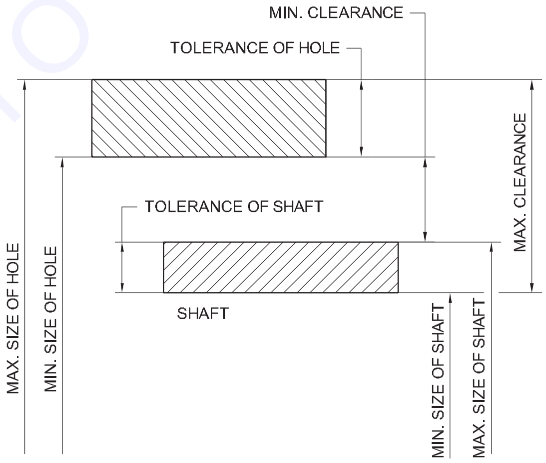
The minimum clearance is 20.000 - 19.993 = 0.007 mm.

The maximum clearance is 20.021 - 19.980 = 0.041 mm. (Fig 4)

There is always a clearance between the hole and the shaft. This is the clearance fit.

Interference: It is the difference between the size of the hole and the shaft before assembly and this is negative. In this case, the shaft is always larger than the hole size.

Fig 4



Interference fit: It is a fit which always provides interference. Here the tolerance zone of the hole will be below the tolerance zone of the shaft. (Fig 5)

Example

Fit 25 H7/p6 (Fig 6)

Fig 5

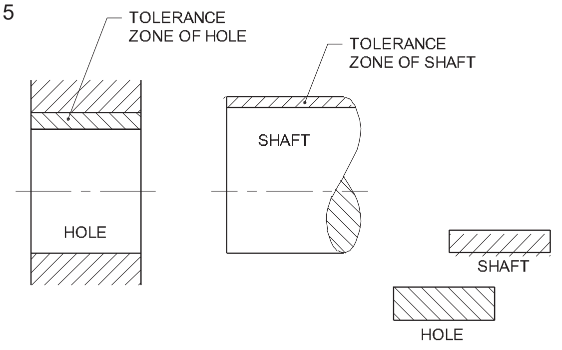
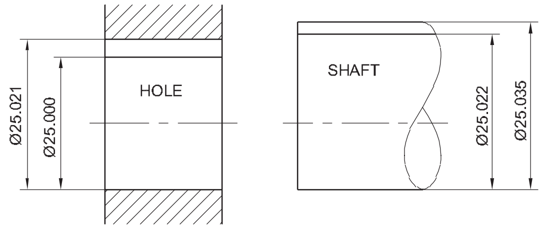


Fig 6



The limits of the hole are 25.000 and 25.021 mm and the limits of th eshaft are 25.022 and 25.035. The shaft is always bigger than the hole. This is an interference fit.

Maximum interference: In an interference fit, it is the algebraic difference between the minimum hole and the maximum shaft. (Fig 7)

Minimum interference: In an interference fit, it is the algebraic difference between the maximum hole and minimum shaft. (Figs 7 & 8)

In the example shown in figure 6,

the maximum interference is = 25.035 - 25.000

= 0.035.

the minimum interference is = 25.022 - 25.021

= 0.001.

Transition fit: It is a fit which may sometimes provide clearance and sometimes interference. When this class of fit is represented graphically, the tolerance zones of the hole and shaft will overlap each other. (Fig 8)

Example

Fit 75 H8/j7 (Fig 9)

Fig 7

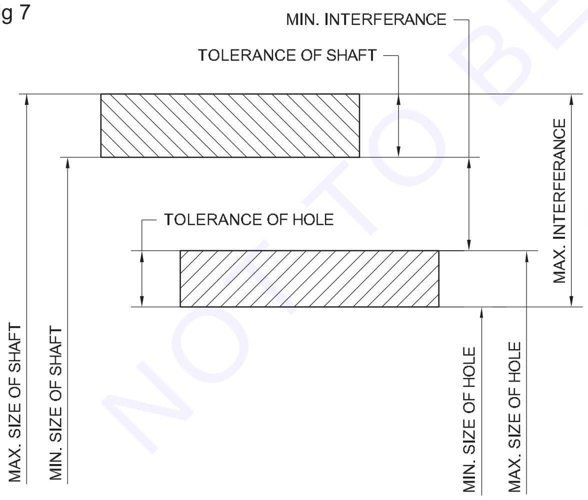
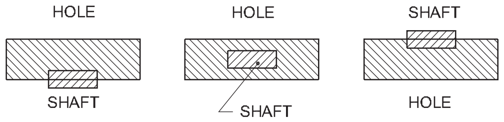
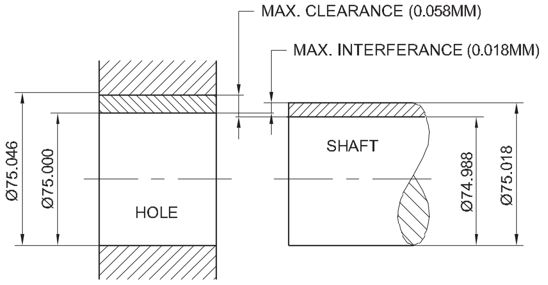


Fig 8



TRANSITION FIT

Fig 9



The limits of the hole are 75.000 and 75.046 mm and those of the shaft are 75.018 and 74.988 mm.

Maximum clearance = 75.046 - 74.988

= 0.058 mm.

If the hole is 75.000 and the shaft 75.018 mm, the shaft is 0.018 mm bigger than the hole. This results in interference. This is a transition fit because it can result in a clearance fit or an interference fit.

Hole basis system: In a standard system of limits and fits, where the size of the hole is kept constant and the size of the shaft is varied to get the different classes of fits, it is known as the hole basis system.

The fundamental deviation symbol `H' is chosen for the holes, when the hole basis system is followed. This is because the lower deviation of the `H' hole is zero. It is known as the `basic hole'. (Fig 10)

Shaft basis system: In a standard system of limits and fits, where the size of the shaft is kept constant and the variations are given to the hole for obtaining different classes of fits, then it is known as shaft basis system. The fundamental deviation symbol `h' is chosen for the shaft when the shaft basis is followed. This is because the upper deviation of the `h' shaft is zero. It is known as the `basic shaft'. (Fig 11)

Fig 10

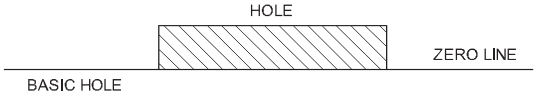
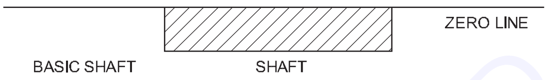


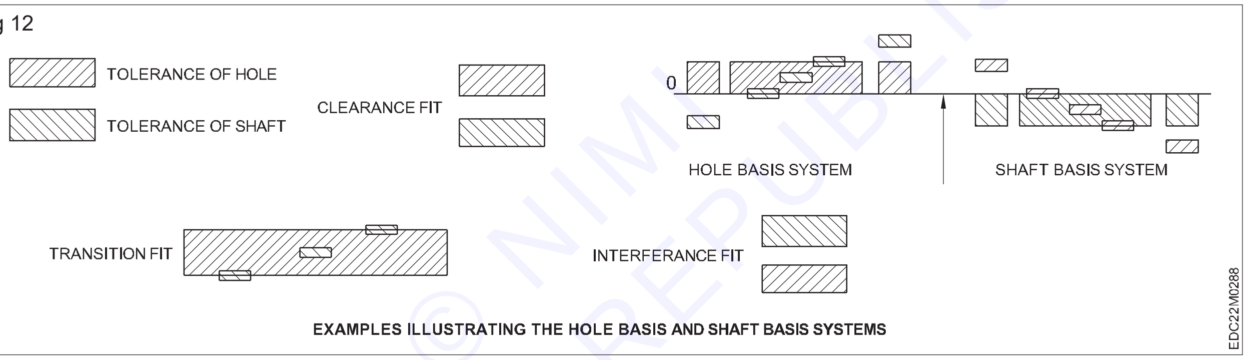
Fig 11



The hole basis system is followed mostly. This is because, depending upon the class of fit, it will be always easier to alter the size of the shaft as it is external, but it is difficult to do minor alternations to a hole. Moreover the hole can be produced by using standard toolings.

The three classes of fits, both under the hole basis and the shaft basis are illustrated in figure 12.

Fig 12



EXAMPLES ILLUSTRATING THE HOLE BASIS AND SHAFT BASIS SYSTEMS

The B.I.S system of limits and fits - Reading the standard chart

The standard chart covers sizes upto 500 mm (IS:919 of 1963) for both holes and shafts. It specifies the upper and lower deviations for a certain range of sizes for all combinations of the 25 fundamental deviations and 18 fundamental tolerances.

The upper deviation of the hole is denoted as ES and the lower deviation of the hole is denoted as EI. The upper deviation of the shaft is denoted as `es' and the lower deviation of shaft is denoted as `ei'.

NOTE`es' is expanded ECART SUPERIOR and `ei' as ECART INFERIOR.

Determining the limits from the chart

Note whether it is an internal measurement or an external measurement.

Note the basic size.

Note the combination of the fundamental deviation and the grade of tolerance.

Then refer to the chart and note the upper and lower deviations which are given in microns with the sign. Accordingly add or subtract from the basic size and determine the limits of size of the components.

Example

30H7 (Fig 1)

It is an internal measurement. So we must refer to the chart for `holes'.

The basic size is 30 mm. so see the range 30 to 40.

Look for es, and ei values in microns for H7 combination for 30 mm basic size.

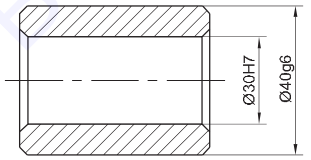
It is given as + 25
+ 0

Therefore, the maximum limit of the hole is 30 + 0.025 = 30.025 mm.

The minimum limit of the hole is 30 + 0.000 = 30.000 mm.

Refer to the chart and note the values of 40 g6.

Fig 1



EDC22M0289

Geometrical tolerance

Definition of geometrical tolerance

Geometrical tolerance is the maximum permissible overall variation of form or position of a feature.

Reason for using geometrical tolerance

This will help the operator to produce the components, particularly those parts which must fit together precisely.

The idea is to have an international system which will overcome the usual language barrier. This is achieved by the use of symbols which represent geometrical characteristics.

Form: Straightness, flatness, roundness, cylindricity and profile of a line and a surface.

Attitude: Parallelism, squareness and angularity.

Location: Position, concentricity and symmetry.

General principles of geometrical tolerances

The geometrical tolerance consists of a frame which contains a symbol, representing the geometrical tolerance zone; in this instance 0.05 for the characteristic of parallelism. The symbol for flatness is shown accompanied by the tolerance zone figure of 0.02 in the lower frame. (Fig 1)

You will notice that from each of the frames a leader is drawn so that it is normal, i.e at 90° to the relevant face and ending with an arrow head against the face.

Notice also that from the `parallelism' frame, another leader is drawn terminating in a blacked-in equilateral triangle on a projection drawn out from the base line. The blacked-in triangle (about 4.5 mm high from base to apex) is the symbol used to represent a datum face or line.

An alternate method of arranging the frames and symbols is shown in Fig 2 where the datum is given a letter and a frame of its own with an independent leader line ending in the blacked-in triangle, inverted and drawn against the actual component base line. The datum letter `A' is then added as an extra component in the geometrical tolerance frame.

Fig 1

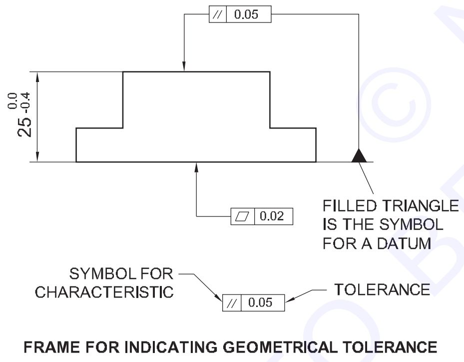
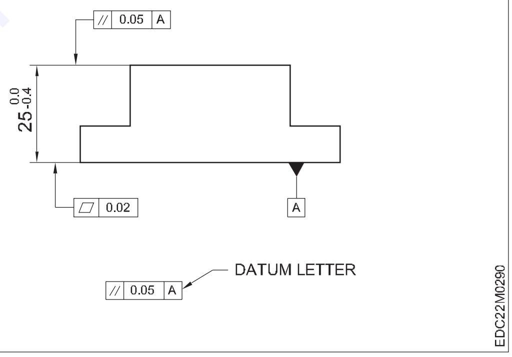


Fig 2



Recommended symbols for geometrical tolerancing

Geometrical tolerances are arranged into three groups. They are tolerances of form, attitude and location.

Tolerances of FORM are identified by the use of symbols for the following characteristics.

Characteristics	Symbol
Straightness	⏤
Flatness	⏥
Roundness	○
Cylindricity	⌭
Profile of a line	⌒
Profile of a surface	⌓

The application of symbols is indicated in Fig 3, where (3a), (3b), (3c) & (3d) show the use of geometrical tolerances controlling the straightness of a circular section part. In (3a) & (3b) the leader lines from the tolerance frame end in an arrow head against the axis of the part. This means that the geometrical tolerance applies to the full length of the part. The interpretation at (3a) shows that for functional acceptance, the entire main axis must lie between two parallel straight lines 0,1 apart in that plane. At (3b) the symbol for the diameter ϕ precedes the tolerance. This means that the entire main axis must lie within a cylindrical tolerance zone 0.1 mm diameter.

Figures (3c) & (3d) show the same geometrical tolerance, applied this time to the diameter dimension of the smaller diameter of the part.

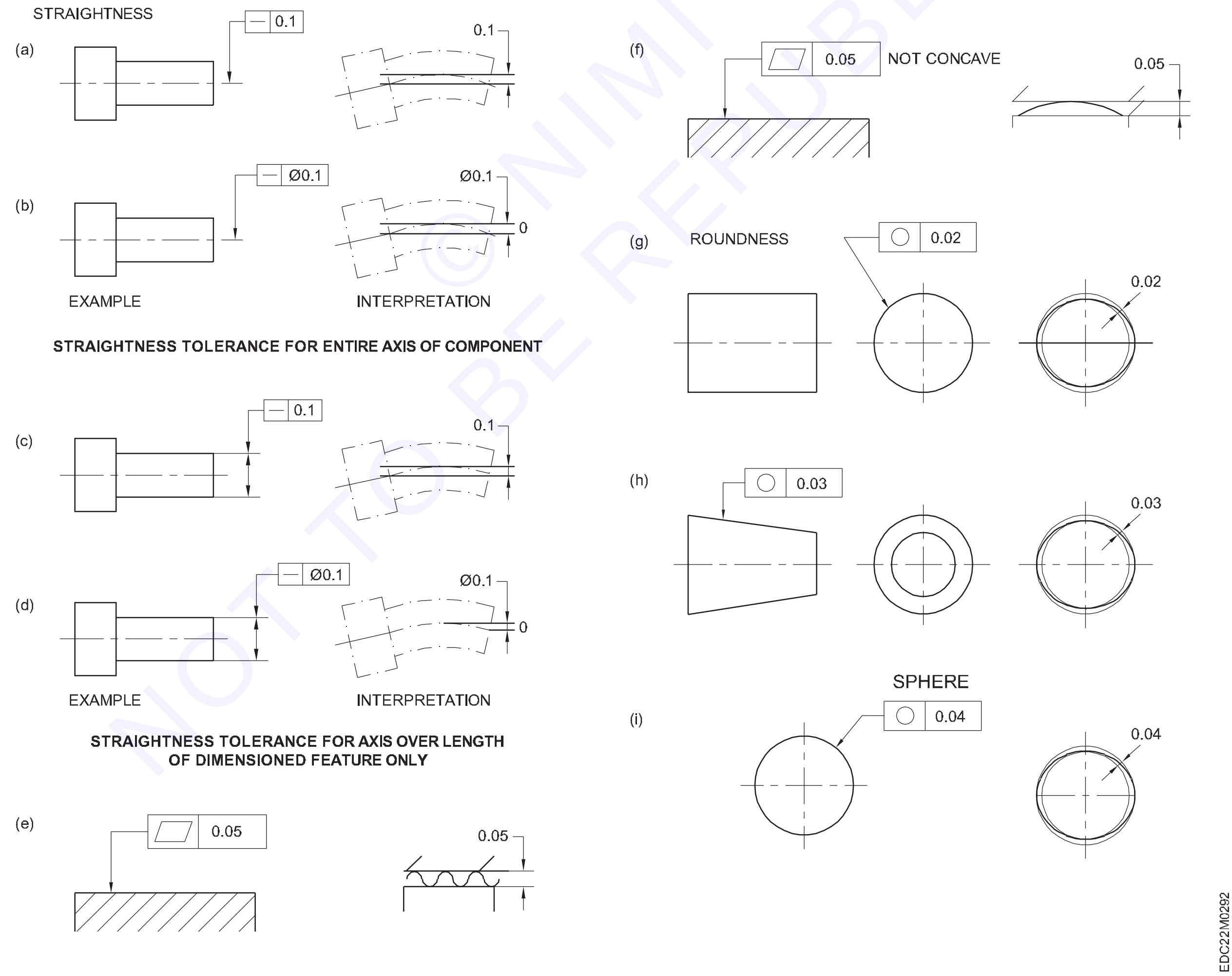
This means that the geometrical tolerance applies over the length of the dimensional feature only.

Figure (3e) and figure (3f) deal with the geometrical tolerance for flatness of a surface where the symbol for flatness is followed by the tolerance figure of 0.05. This figure indicates that the actual surface (as previously shown in Figure 1) must be between two parallel planes 0.05 apart. If a particular form of direction is prohibited, then this is stated in a note form against the tolerance frame. Eg. `Not concave'.

The geometrical tolerance controlling the roundness of a part is shown in figures (3g), (3h) & (3i). The interpretation for (3g) & (3h) is that the true form of the periphery of the part at any cross-section perpendicular to the axis must lie between two concentric circles whose radial distance apart is 0.02 for (3g) and 0.03 for (3h).

For the sphere shown in (3i) the geometrical tolerance applies to concentric circles with teh radial distance 0.04 apart at the periphery at any section of maximum diameter.

Fig 3



The sphere controlling cylindricity is shown in Fig 4. Here the interpretation shows, that for acceptance, the surface of the part must be within two co-axial cylinders, whose radial distance apart is 0.05.

Fig 5 shows the method of applying a geometrical tolerance to a curved surface. The symbol is followed by the tolerance 0.05 which means that the actual surface must lie between two surfaces enveloping a succession of sphere 0.05 diameter whose centre lies on the theoretical surface.

Fig 4

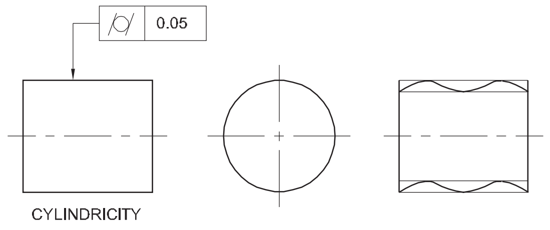
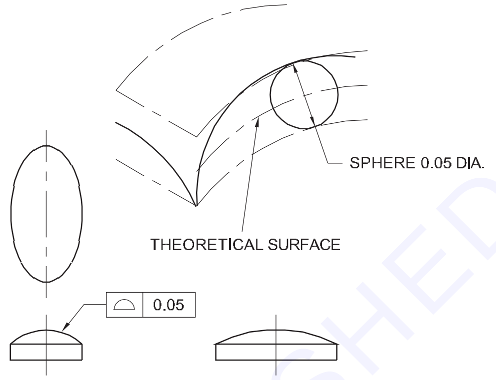


Fig 5

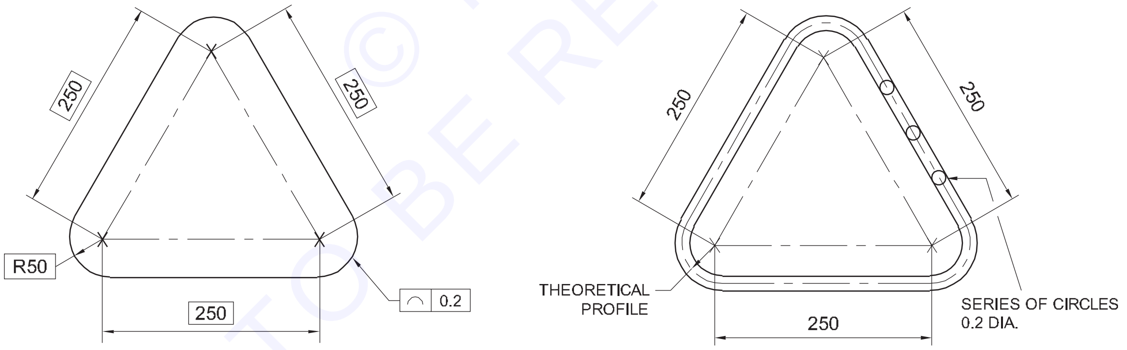


In Fig 6 the geometrical tolerance is applied to linear dimensions controlling the profile. The rectangular `boxes' around the 250 centre dimension and the 50 radius is the method used to indicate theoretical dimensions i.e the dimensions relevant to perfect form.

The interpretation of the geometrical tolerance is that the actual profile must be between two lines which touch a succession of circles 0.2 dia whose centre lies on the theoretical profile.

Tolerances of attitude are identified and indicated by the use of symbols for the following.

Fig 6



Characteristics	Symbol
Parallelism	//
Squareness	⊥
Angularity	∠

A typical application of tolerances for these three characteristics is shown in Figures 7,8 and 9. Figures (7a), (7b) & (7c) show the application of tolerancing to control `parallelism'. (7a) shows that the axis of the upper hole must lie between the two lines 0.08 apart the lower hole, as indicated by the leader ending in the blacked-in triangle. In (7b) the method uses a separate datum letter `A' which is added to the frame after the tolerance of 0.05 diameter. The requirement is that the upper hole axis must lie within a cylindrical zone 0.05 diameter with its axis parallel with the axis of the datum hole `A'. (Fig 7c) shows a component whose upper surface must be between two parallel planes 0.05 apart, parallel with the bottom datum surface. While the overall tolerance zone is 0.05 as shown in the upper section of the frame, the figures in the lower section of the frame stipulate that over any length of 100 the parallelism tolerance is reduced to 0.02.

Examples of the application of the geometrical tolerance for `squareness' are shown in (8a), (8b), *8c) and (8d) with (8a), (8b) and (8c) using the separate box method for indicating the datum.

The interpretation is as follows:

The axis of the vertical hole must be between two parallel lines, 0.05 apart, which are perpendicular to the common datum axis `A' of the two horizontal holes. (8a)

The axis of the upper cylindrical portion must lie within a cylindrical tolerance zone of 0.1 diameter, the axis of which is perpendicular to the datum axis `A'. (8b)

This shows that the right hand end face must lie between two parallel planes 0.05 apart, which are perpendicular to the datum axis. (8c)

Here the datum surface is indicated by a leader from the frame. The requirement is that the right hand face must lie within the two parallel planes, 0.05 apart, which are perpendicular to the datum surface. (8d)

Geometrical tolerances for the control of ANGULARITY are shown in figures (9a), (9b) and (9c).

The figure (9a) shows tha tthe requirement is the axis of the hole must lie within the cylindrical tolerance zone 0.1 diameter, the axis of which must be included at the theoretical angle of 60° to the datum surface `A'.

Fig 7

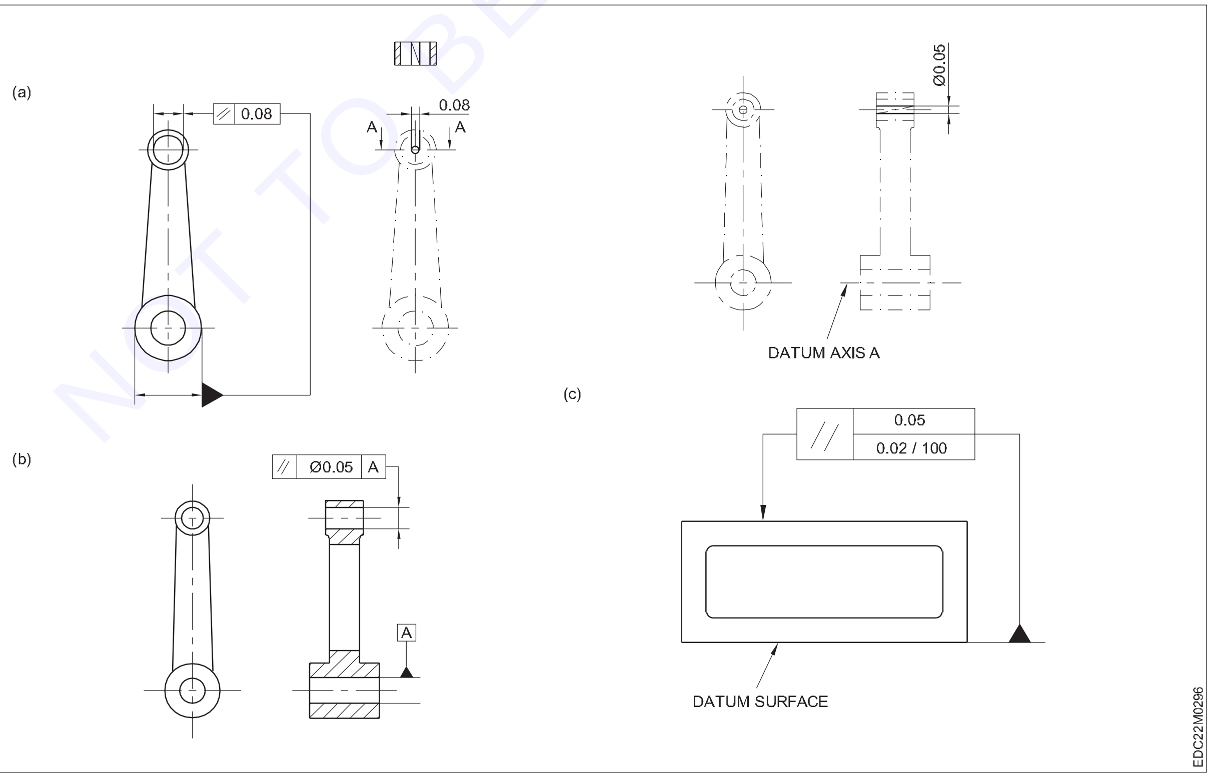


Fig 8

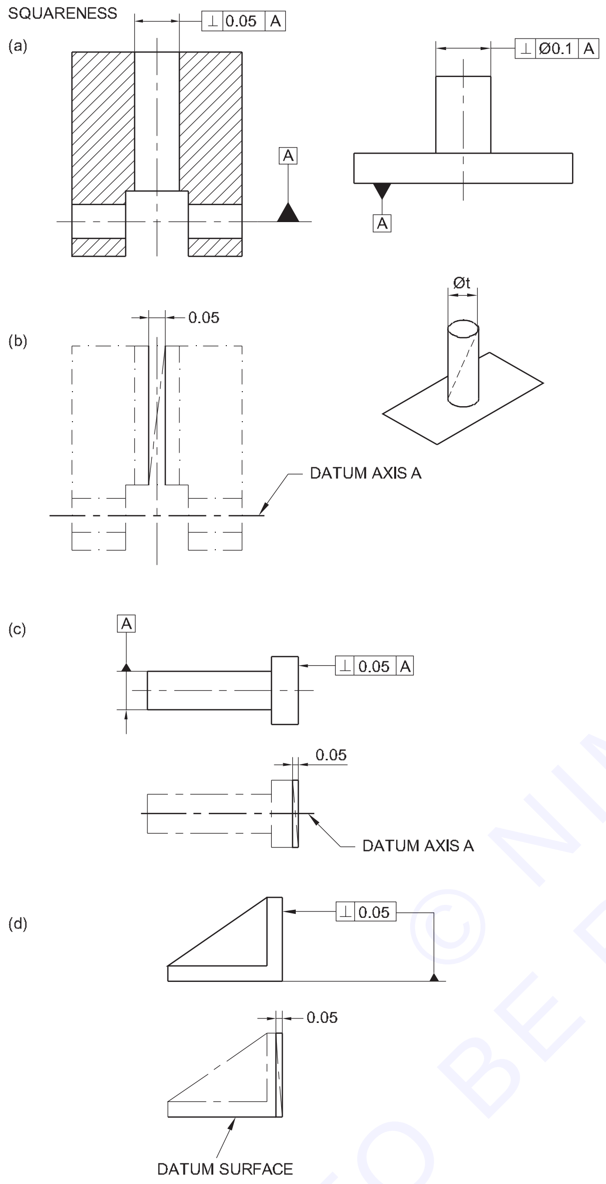
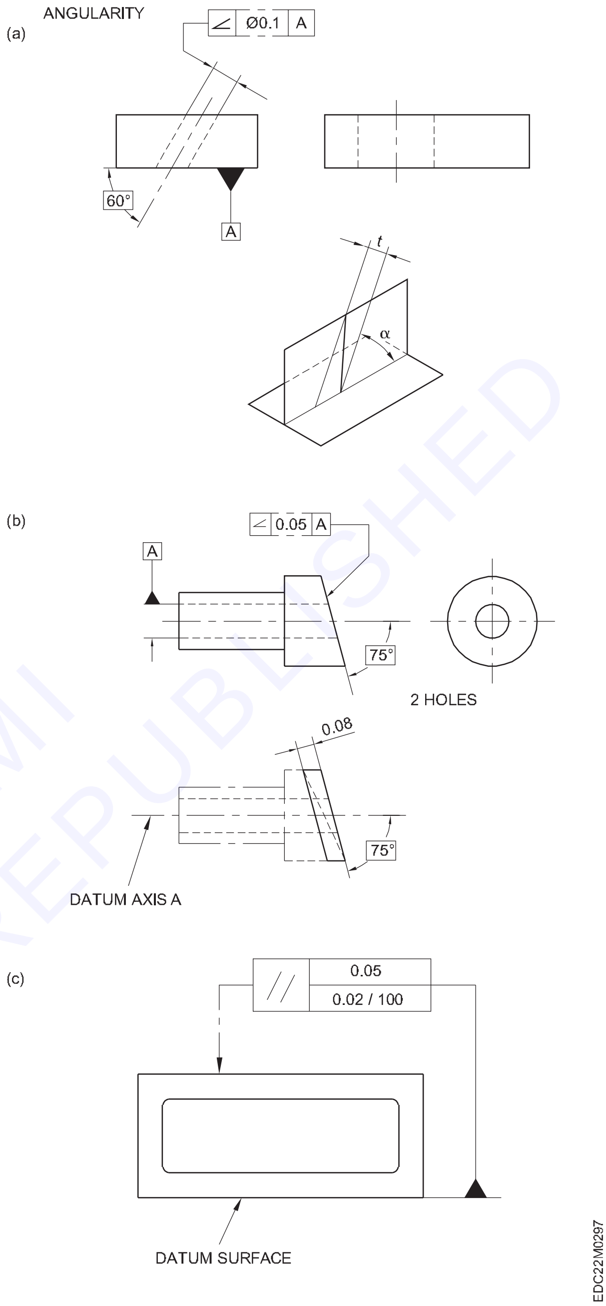


Fig 9



In (9b) the requirement is that the right hand end face must lie within the two parallel planes 0.08 apart which are inclined at the theoretical angle of 75° to the datum axis `A' of the through hole.

Figure (9c) shows a component whose upper angle face, must lie between the two parallel planes 0.05 apart which are inclined at the theoretical of 35° to the base, the datum surface. Notice that the theoretical angle in each example is boxed.

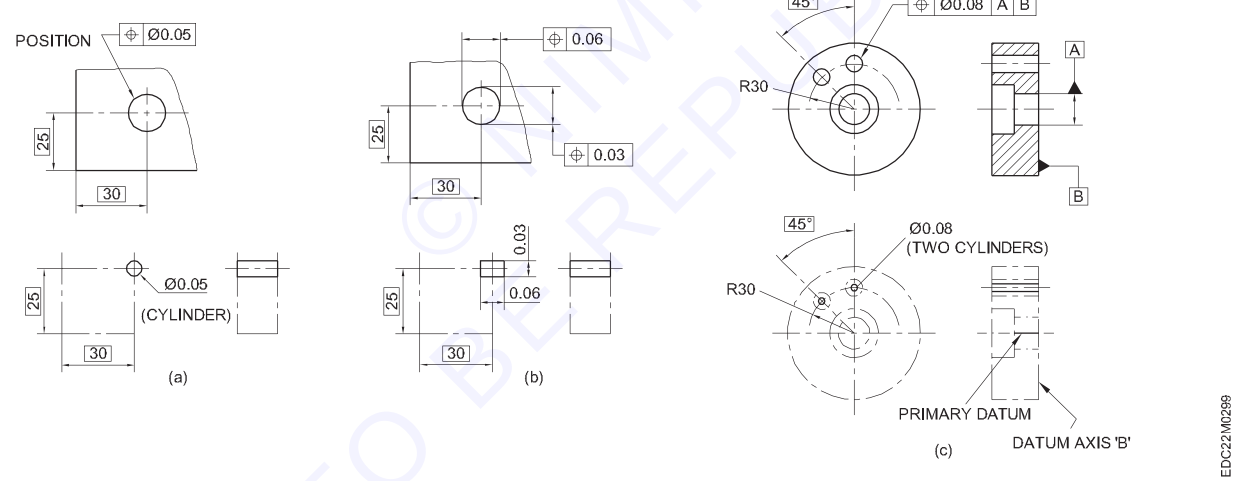
Tolerances of location are identified and indicated by the symbols for the following characteristics.

Characteristics	Symbol
Position	⌖
Concentricity	◎
Symmetry	⌯

Figures (10), (11), (12) show typical examples of these characteristics and symbols. In figure (10a) the hole of the dimensions of 25 and 30 are boxed to show that these are the theoretical dimensions. The geometrical tolerance requires that the hole centre must lie within a cylindrical zone 0.05 diameter. The use of theoretical positions also known as `true positions', implies that the axis of the cylinder is square with the plane of the drawing. Figure (10b) shows the hole with the same true positions, but with the geometrical tolerances arranged to give greater tolerance along the horizontal axis. The resulting requirement is that the axis of the hole must lie within a triangular box whose sides are 0.03 and 0.06, and length equal to the width of the component.

Figure (10c) the two holes are shown with their true position placed at 45° on a 30 mm pitch circle radius. The geometrical tolerance shows that each actual hole centre that lie within a cylindrical zone 0.08 diameter whose axis lies at the theoretically exact true centre position. The tolerance cylinders are disposed relative to the two datum features, namely the axis of the smaller bore and the right right end face. The datum letters are included in the clearance frame.

Fig 10



Examples of geometrical tolerance for `CONCENTRICITY' are given in figures (11a), (11b) and (11c). The interpretations are as follows:

In figure (11a) the axis of the smaller diamter must lie within the cylindrical zone 0.08 diameter which must be co-axial with the datum axis i.e the axis of datum diameter `A'.

In figure (11b) the axis of two end portions must lie within a cylindrical tolerance zone 0.08 dia.

Figure (11c) shows that teh axis of the large central portion must lie within a cylindrical zone 0.1 diameter which is co-axial with the mean axis of the datum diameters `A' and `B'. (Notice that to indicate the requirement of the mean axis the datum letters are separated by a hyphen and enclosed in the same compartment of the tolerance frame).

The geometrical tolerance of `SYMMETRY' is indicated in figures (12a), (12b) & (12c) where the interpretations are:

- (12a) the axis of the hole must lie between two parallel planes, 0.08 apart which are symmetrically disposed about the mean axial plane of datum width `A' and `B'.
- (12b) the mean plane of the slot must be between two parallel planes 0.05 apart symmetrically disposed about the mean plane of the datum width `W'.
- (12c) the median planes of the two end slots must be between two parallel planes 0.06 apart.

Fig 11

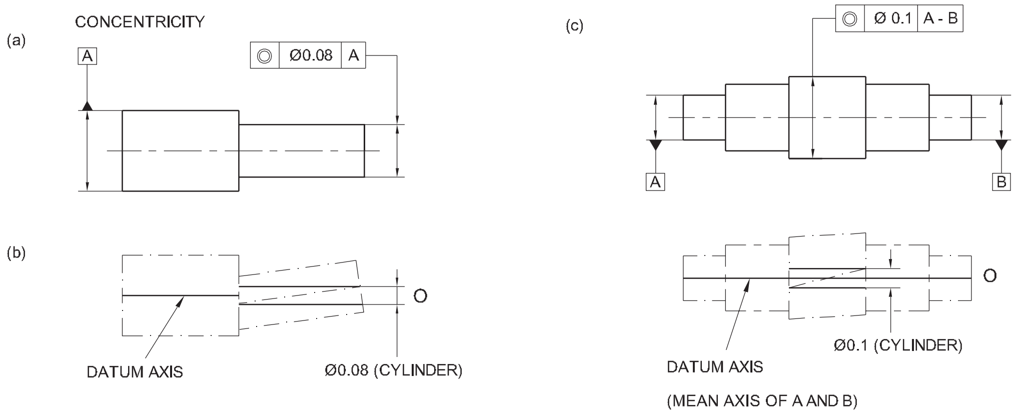
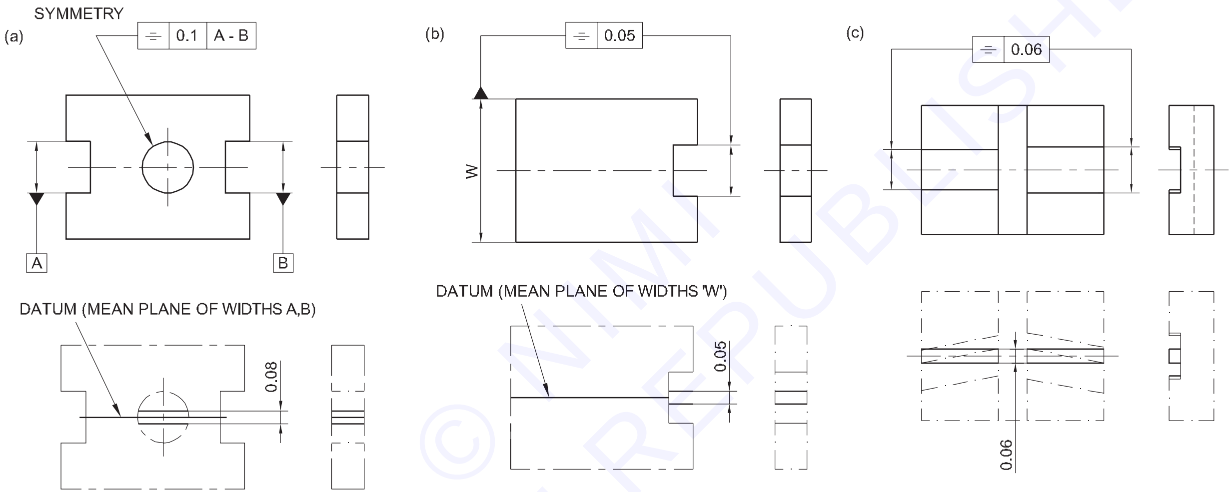


Fig 12

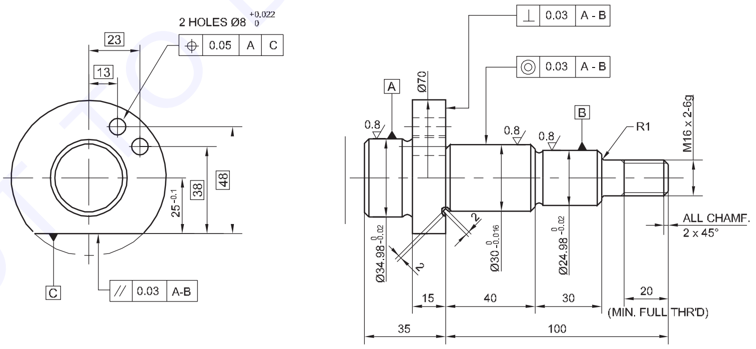


TWO PLANES 0.08mm APART SYMMETRICALLY DISPOSED ABOUT THE DATUM

TWO PLANES 0.05mm APART SYMMETRICALLY DISPOSED ABOUT THE DATUM

TWO PLANES 0.06mm APART

Fig 13



EXERCISE 10 : Reading of Engineering Drawing

Reading of Engineering

The heading "Blue pring reading" given in the syllabus is nothing but reading of engineering drawing.

In order to manufacture an engineering component, the shape of the component must be understood in unmistakable terms and those who are associated with the manufacture must be able to visualise the component from its multi-view drawing.

Multi-view drawing has no 3D effect and do not convey the shape of a component in the first sight itself.

There is no one set of rules to read and visualise the shape of an object from the given multi-view drawing. Visualisation is essentially the reverse of the mental process that is gone through for obtaining the shape of the object.

In this reading of engineering drawing, different styles of exercises are given to improve the visualisation ability.

Sufficient "Reading of engineering drawing"

Finding/Sketching the missing

– lines
– views
– symbols
– sections etc.

When it is difficult to form a mental image of the object from the available views, one can resort to modelling by means of isometric sketching. This can be done in many ways.

Example

Sketch/Draw the missing view of the block shown in Fig 1.

First sketch the isometric view of a rectangular block keeping in mind the overall length, breadth and height from the views. (Fig 2a)

Then draw its isometric view by cutting away the portions of the prism drawn as required till we satisfy with the outcome. (Fig 2b & c)

Instead of cutting away, we can also form the object by adding or joining portions together Fig 3a,b & c.

After sketching the final isometric view, missing view or line or anything can be found out easily.

Fig 1

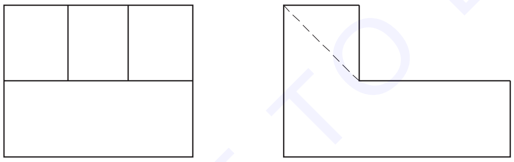
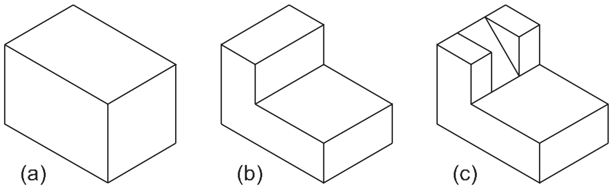


Fig 2



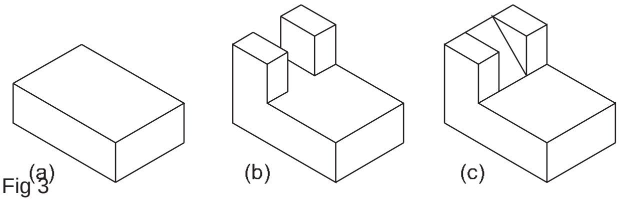


Fig 3

Reading of engineering drawing - Part A

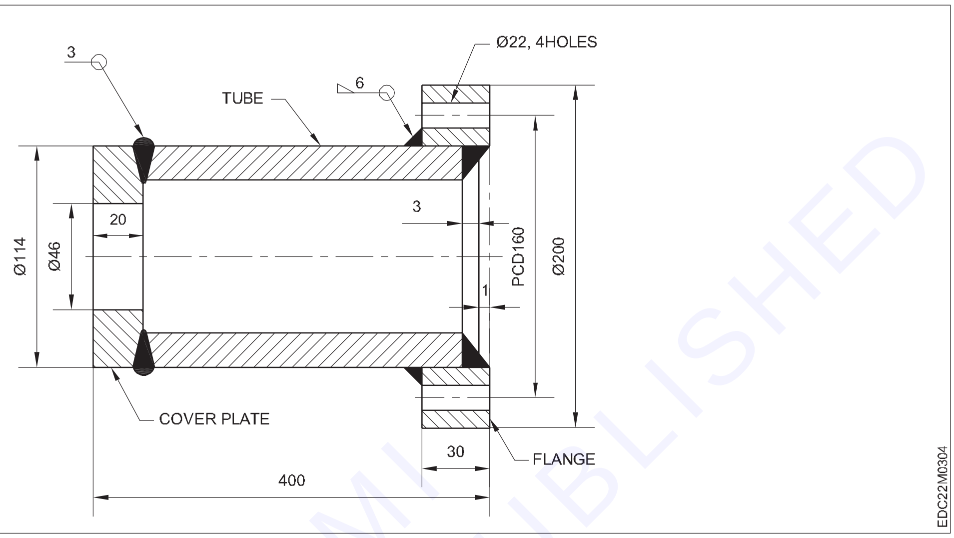
Exercise 10.01 to 10.20

Study the drawings (10.01 to 10.20) carefully and answer the questions related to the drawing in the space allotted for answer in the workbook - 10.01 to 10.20)

Exercise 10.1

Stand assembly (Fig 1)

Fig 1



1 Name components of the stand assembly.

2 What is the P.C.D of the holes in th flange?

3 How many holes are there in the flange?

4 What is the diameter of the flange?

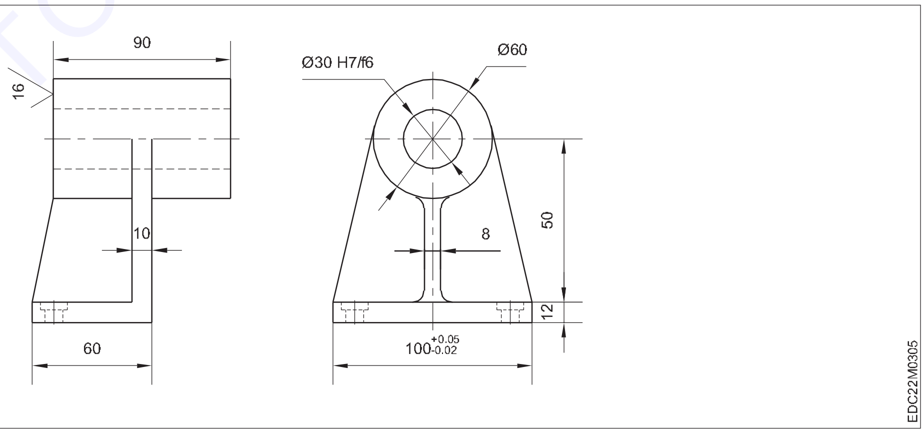
5 What type of welding is used in connecting the cover plate and the tube?

6 What is the length of the tube?

Exercise 10.2

Simple bearing (Fig 2)

Fig 2



1 What is the maximum value of the dimension for width?

2 How much the thickness of the web?

3 Explain the meaning of the dimension ϕ 30 H7/f6?

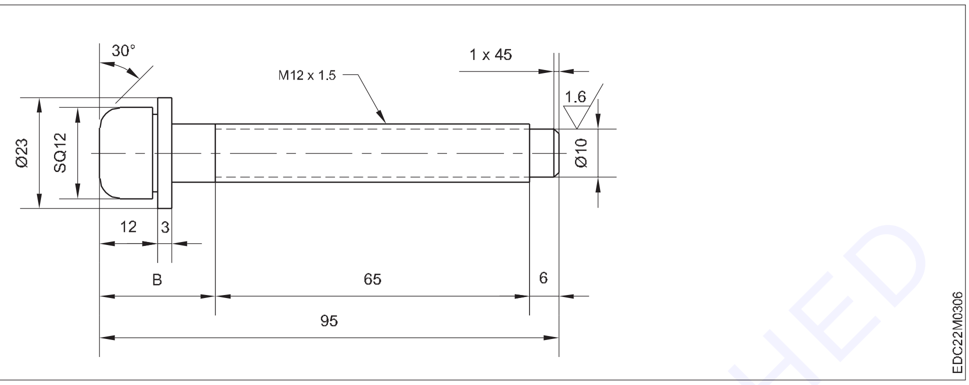
4 What is the type of fit obtained from the dimension ϕ 30 H7/f6?

5 Explain the meaning of the symbol 1.6

Exercise 10.3

Neck bolt (Fig 3)

Fig 3



1 Calculate the distance `B'.

2 How many necks are shown?

3 What does the symbol 1.6 indicate?

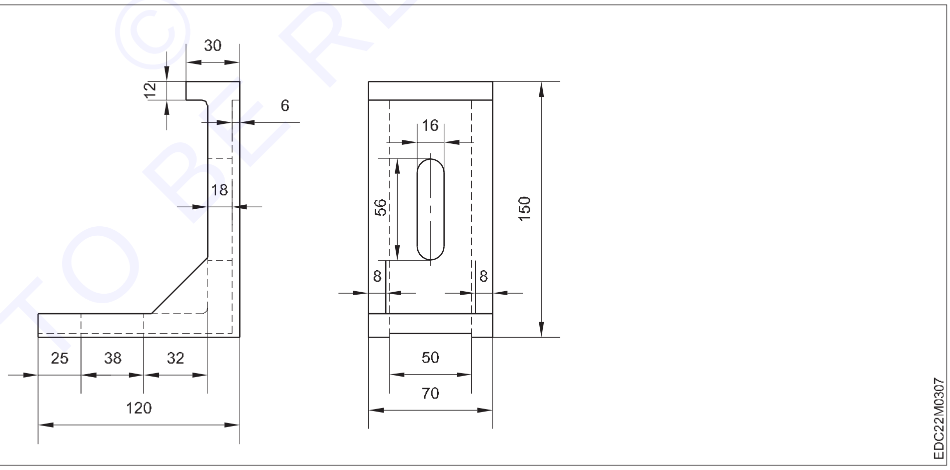
4 Is the thread pitch fine or coarse?

5 How many threads are there in 12 mm length?

Exercise 10.4

C.I.Angle bracket (Fig 4)

Fig 4



1 For what purpose the angle bracket is used.

2 How many holes are there in the bracket?

3 How many webs are there in the bracket?

4 What is the overall size of the bracket?

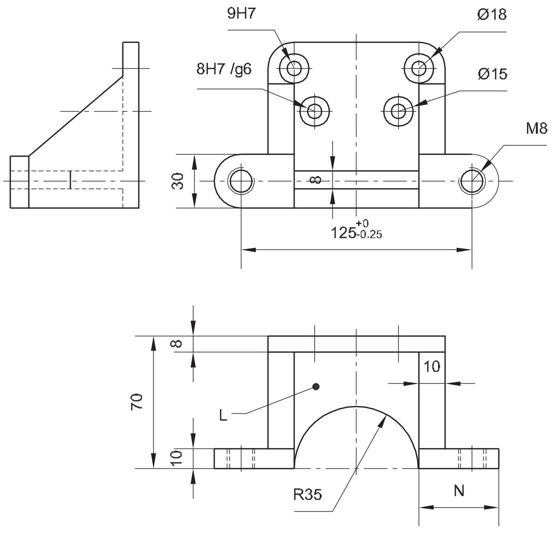
5 What is the material of this bracket?

Exercise 10.5

Bracket (Fig 5)

1 What is the value of dimension `N'?

Fig 5



2 What is the thickness of the surface marked as `L'?

3 How many threaded holes are there in the bracket?

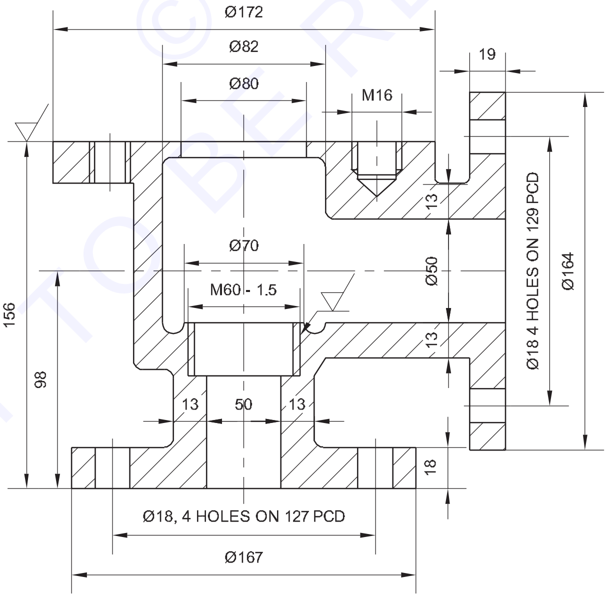
4 What is the meaning of dimension 8H7/g6?

5 Mention the type of tolerance used in the dimension 125.

Exercise 10.6

C.I.Valve body (Fig 6)

Fig 6



1 What is the height of the valve body?

2 What do you mean by M 16 tapped hole?

3 What do you understand by M60 1.5 hole?

4 State the material of the component.

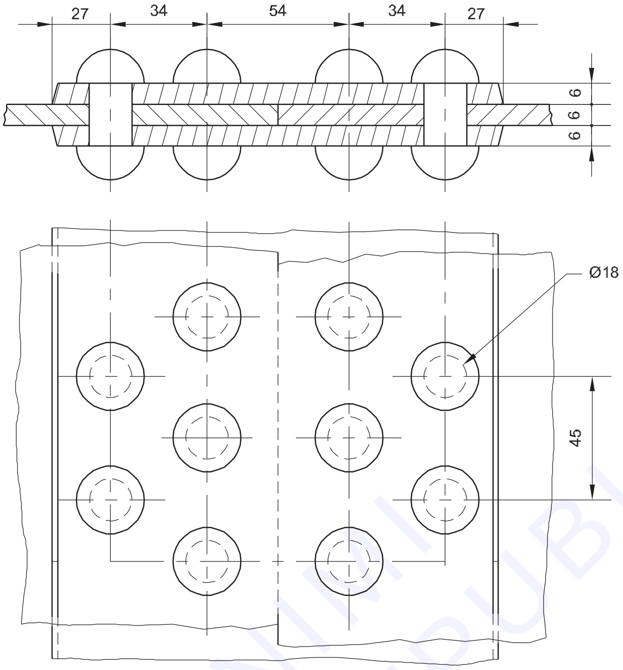
5 State the type of dimensioning adopted in the drawing.

6 What does the symbol ▽ indicate?

Exercise 10.7

Rivetted joint (Fig 7)

Fig 7



EDC22M0310

1 What is the name of the joint?

2 What is the diameter of the rivet?

3 What is the thickness of cover plate?

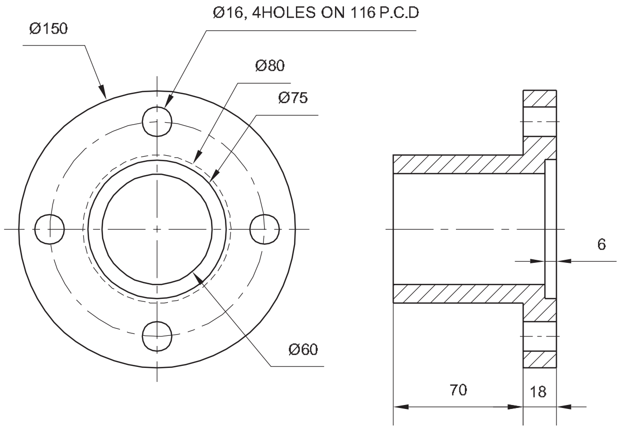
4 What type of rivets are used in the joint?

5 What is the pitch of the rivets?

Exercise 10.8

Flange (Fig 8)

Fig 8



EDC22M0311

1 What is the thickness of the pipe?

2 What is the thickness and diameter of the flange?

3 What is the length of the pipe?

4 What is the purpose of ϕ 75 x 6 mm circular gap in the flange?

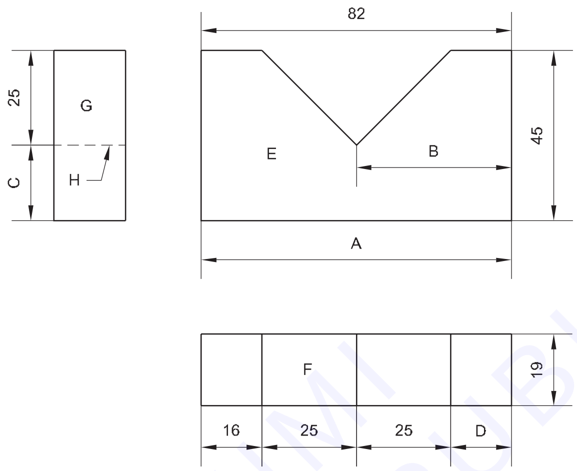
5 What is the distance between the ϕ 16 mm opposite holes?

6 What does the dotted circle in the side view represent?

Exercise 10.9

`V' Block (Fig 9)

Fig 9



1 What is the name of view `E'?

2 What is the name of view `F'?

3 What is the name of view `G'?

4 How wide is the `V' at the top?

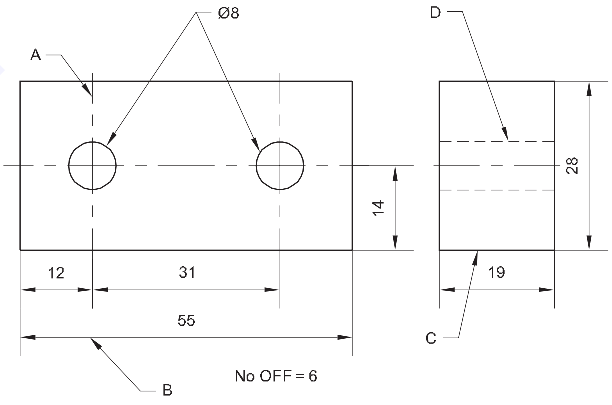
5 What is the purpose of broken line `H'?

6 Give the dimensions of the following alphabets A,B,C & D.

Exercise 10.10

Top plate (Fig 10)

Fig 10



1 What is the name of the part?

2 How many such parts are required?

3 What is the overall length of the part?

4 What is the overall height?

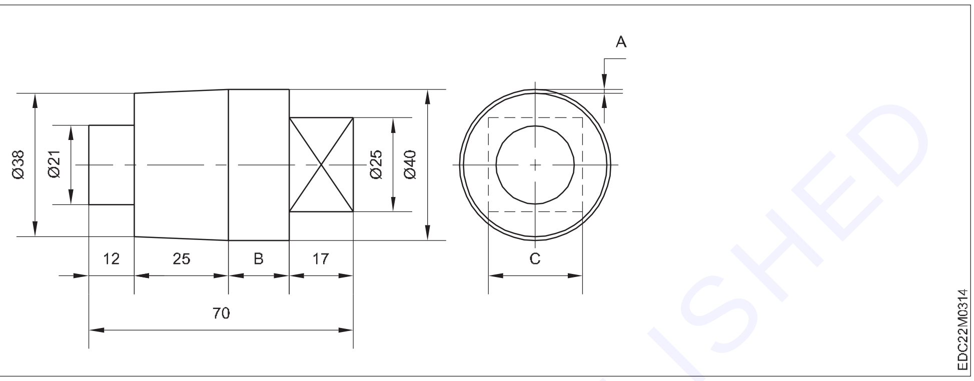
5 Give the thickness of the part?

6 What type of lines are those marked A,B,C,D ?

Exercise 10.11

Turned component (Fig 11)

Fig 11



1 Give dimensions A,B and C.

2 Give the overall size of this object.

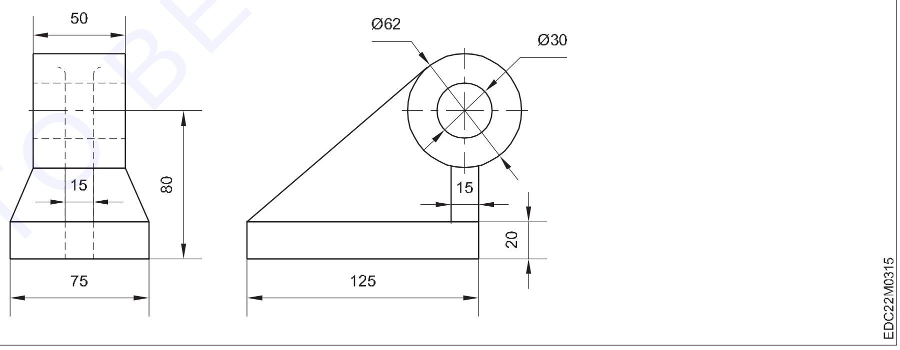
3 Looking at the front view which end is square.

4 Is the left end of the object with the ϕ 21 dimension round or square?

Exercise 10.12

Bearing (Fig 12)

Fig 12



1 How many webs are there in the bearing?

2 What is the size of the bearing?

3 What is the length of ϕ 30 hole?

4 Which surface of the bearing required toleranced dimension?

Exercise 10.13

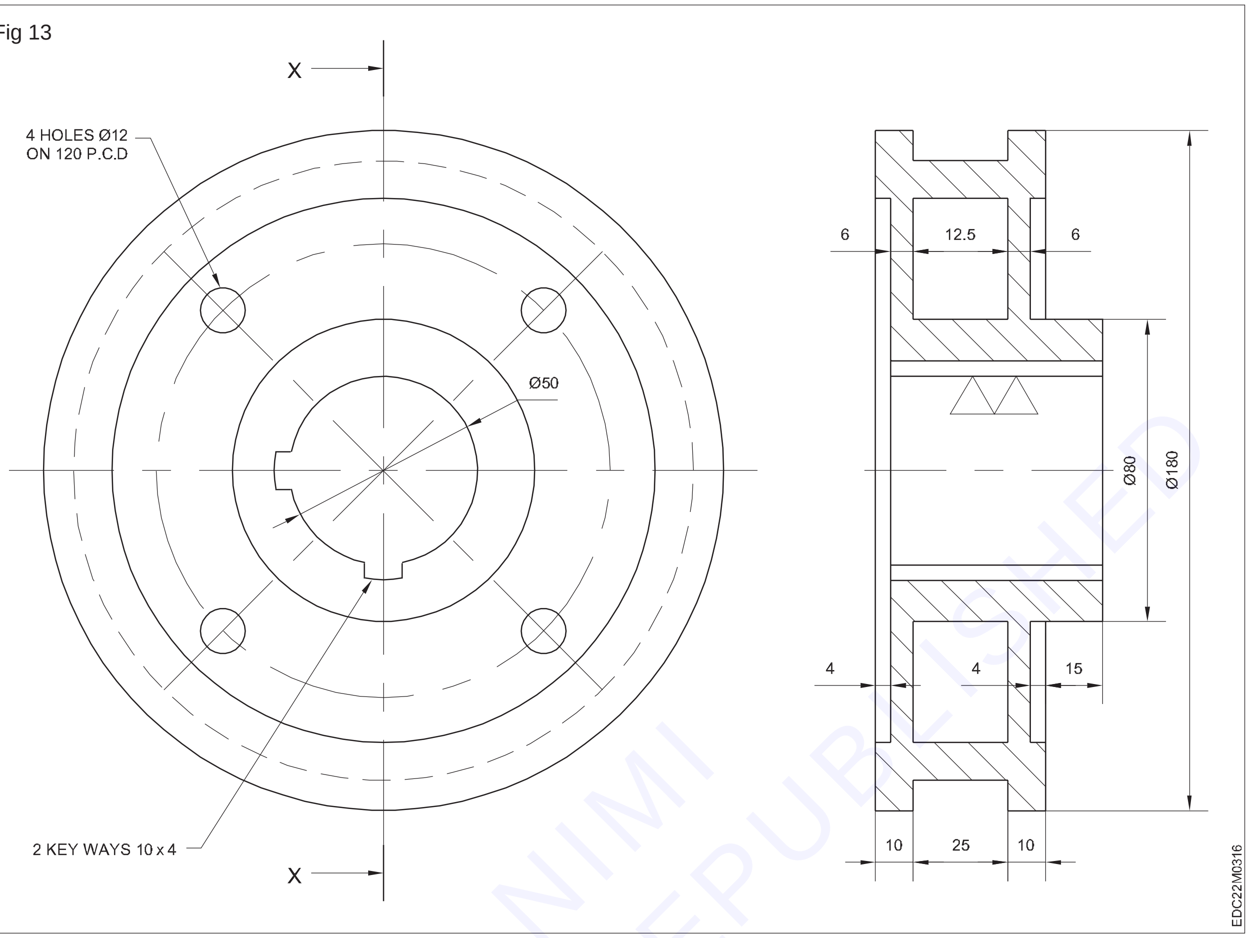
Wheel (Fig 13)

1 What is the outside diameter of the wheel?

2 What is the diameter of the hub?

3 What is the size of the keyway?

Fig 13



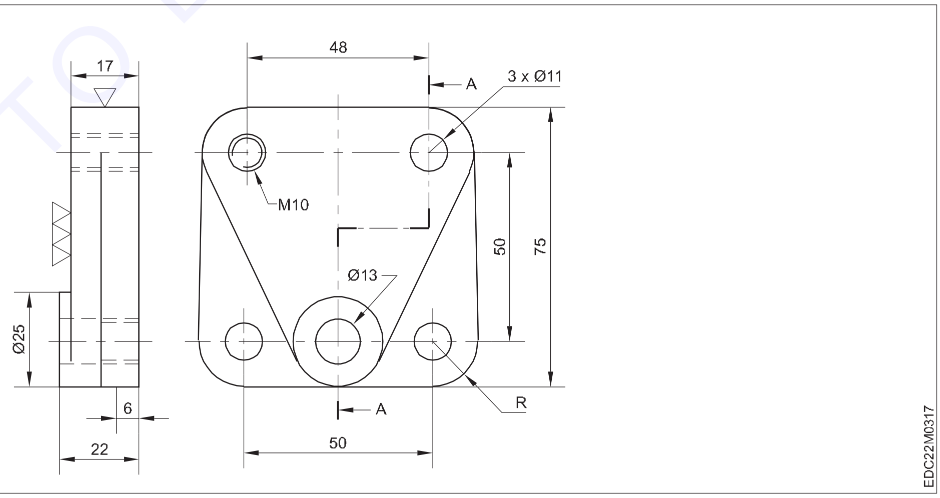
4 What does the sign ▽ mean?

5 How many drilled holes are there in the wheel?

Exercise 10.14

Housing cover (Fig 14)

Fig 14



Study carefully the details of the housing cover in figure and answer the following:

1 How many holes are to be drilled and what are their diameters?

2 What does ϕ means?

3 What do you understand by ▽▽▽ ?

4 Give the overall height of housing cover?

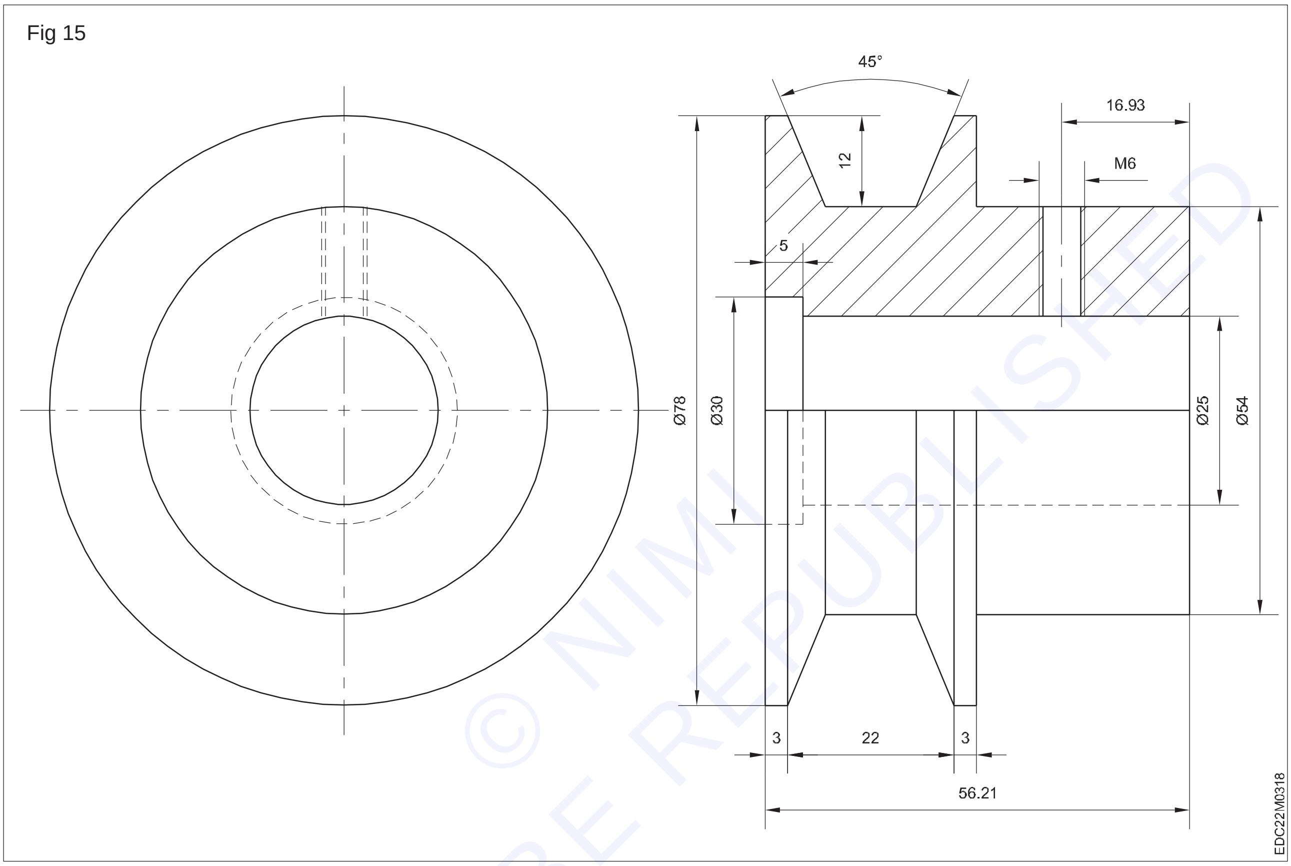
5 How many holes are to be tapped?

6 How many surfaces are to be machined?

Exercise 10.15

Idler pulley (Fig 15)

Fig 15



1 What is the name of the part?

2 What is the largest diameter?

3 What is the overall length?

4 What type of section is indicated?

5 Name four other types of section?

6 Give the diameter of the body of the pulley?

7 State the angle and depth of the `V' groove.

8 What is the size of the threaded hole?

9 Where this hole is located?

10 Give the dia of the hidden circle in the end view?

11 What is the width of the `V' groove in the crest?

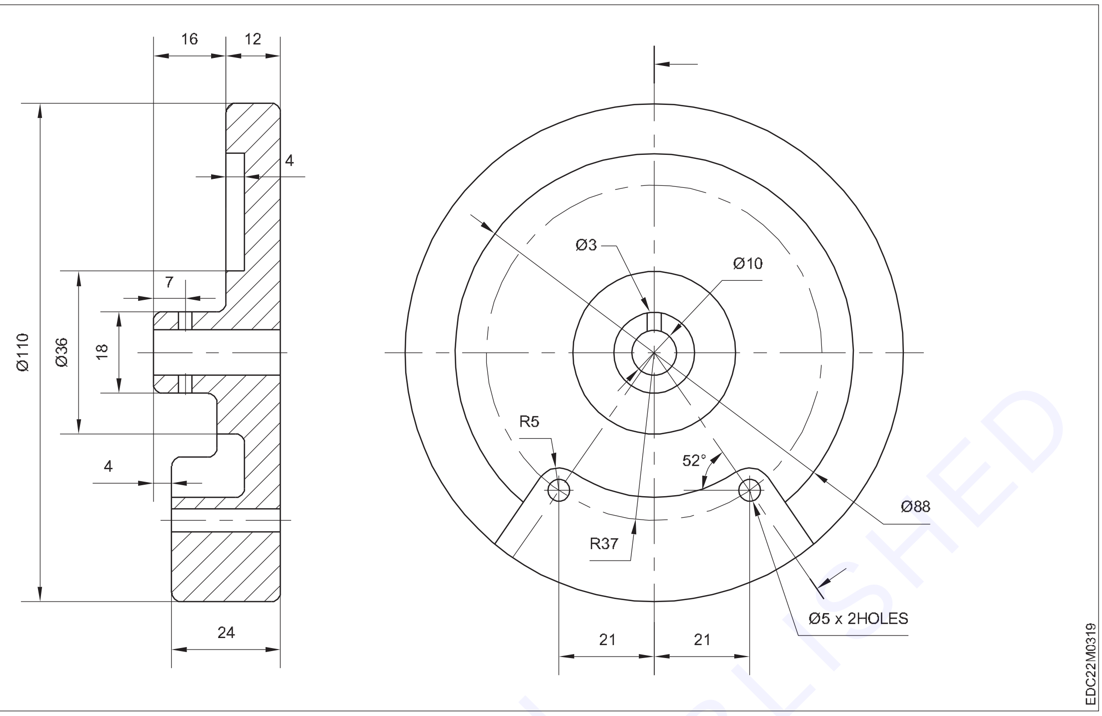
12 State the purpose of tapped hole.

Exercise 10.16

Adapter (Fig 16)

1 What is the name of the part?

Fig 16



2 What type of section is indicated?

3 Is the section is true projected or is it rotated?

4 What is the largest diameter?

5 What is the overall width of the part?

6 How far apart are the centres of the two smaller drilled holes?

7 What is the dia of the centre hole?

8 What is the dia of the body into which the centre hole is drilled?

9 What is the dia of the third largest circle?

10 What angle do the two drilled holes make with the centre hole of the adapter?

11 What is the least thickness of the part?

Exercise 10.17

Piston rod end (Fig 17)

1 What is the outside diameter of the piston?

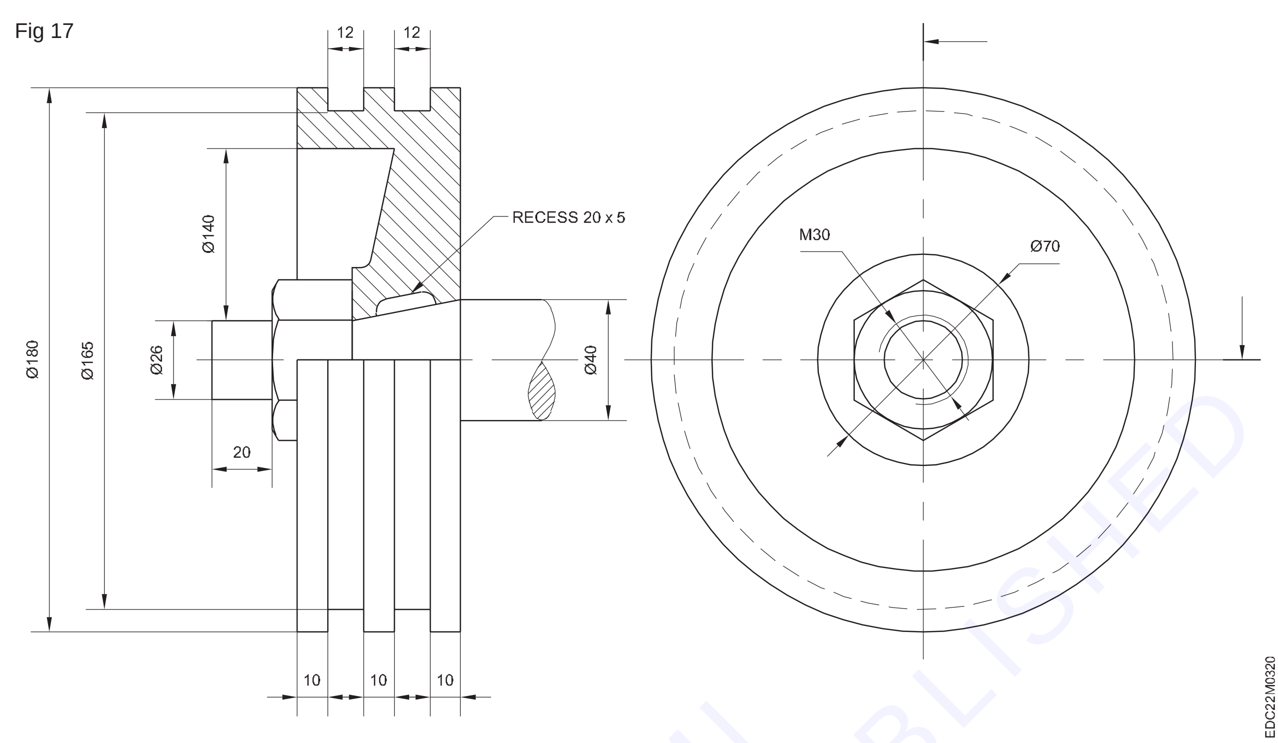
2 What type of nut is used in the assembly?

3 What is the size of the recess in the piston?

4 How many slots are there and what is the size of the slot?

5 What is the size of the nut?

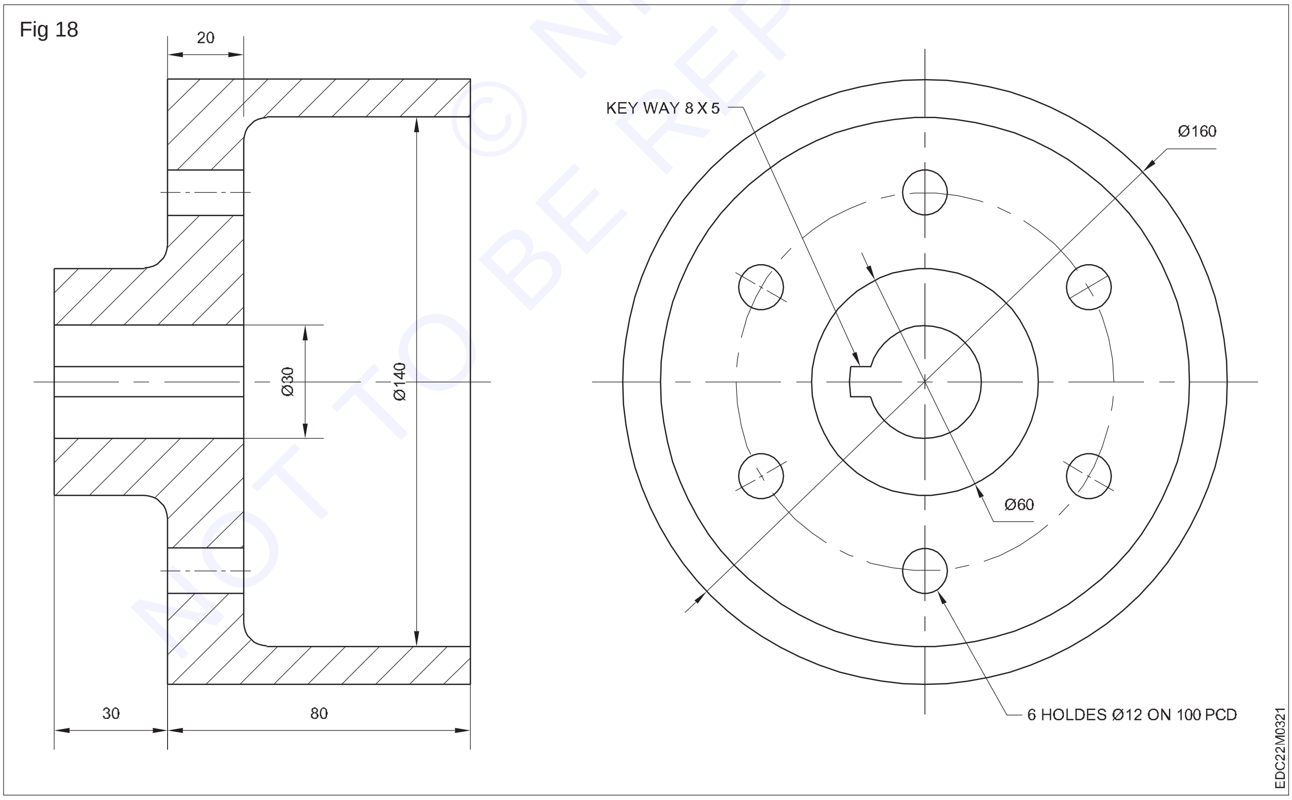
Fig 17



Exercise 10.18

Flange (Fig 18)

Fig 18



1 What is the view shown on the left side of the drawing?

2 What is the width of the keyway?

3 What is the diameter of the hub?

4 What is the depth of the ϕ 140 bore?

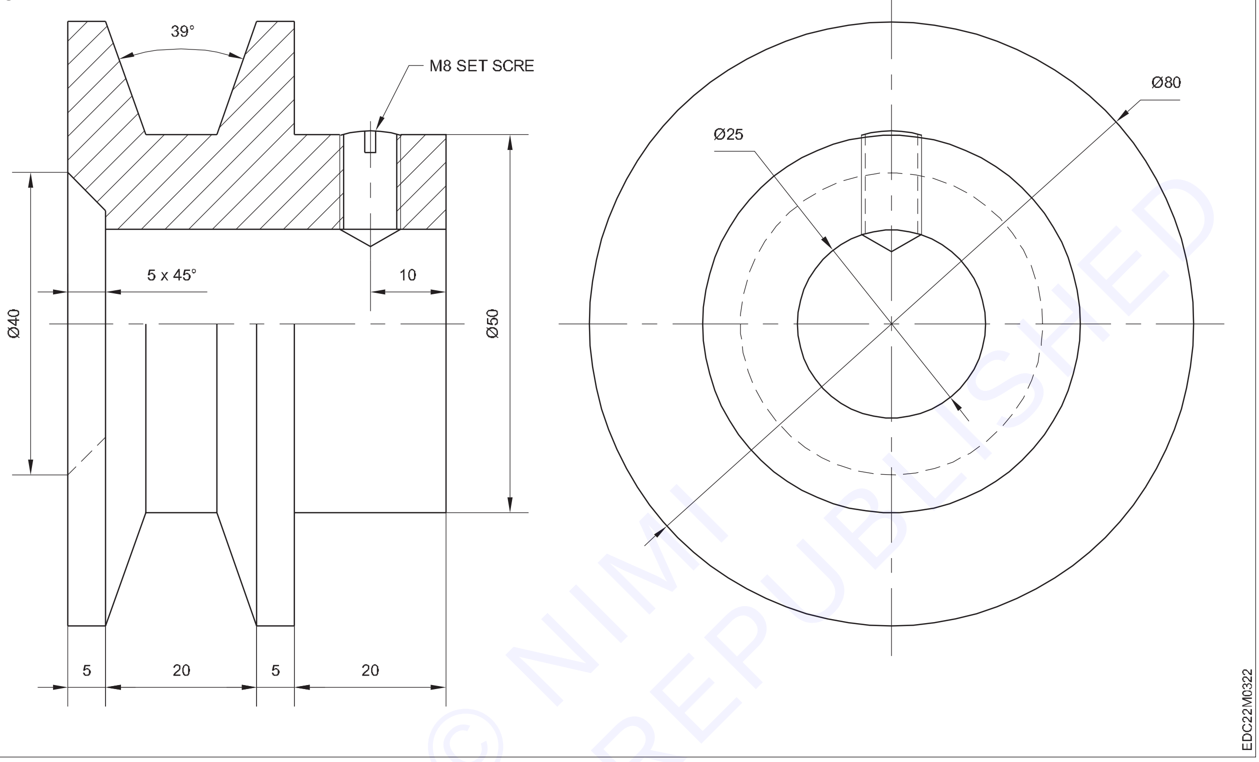
5 What is the length of the hub?

6 What is the size of the drilled hole?

Exercise 10.19

Vee pulley (Fig 19)

Fig 19



1 What is the name of the part?

2 Where is it used?

3 What is the thickness of the pulley?

4 What is the size of the setscrew?

5 What is the angle of the groove?

6 What is the length of the pulley including the hub?

7 What is the depth of the V-groove?

Exercise 10.20

Flat pulley (Fig 20)

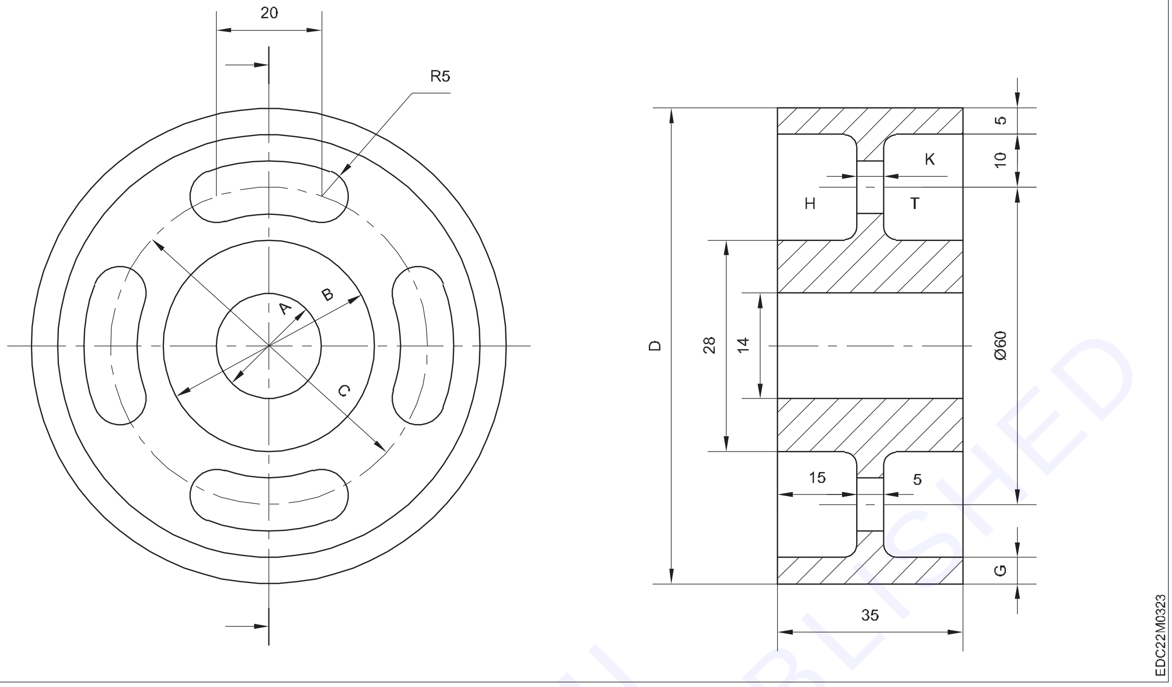
1 Indicate the diameter of each of the following circles A,B,C and D.

2 Why are H and T not sectioned?

3 How thick is the metal at `G'?

4 How thick is the metal at `K'?

Fig 20



Exercise 10.21.01 to 10.21.48: Choose the correct view 48 questions are given below. Give your answer (alphabet) in the response sheet attached in the work book - 10.21.01 to 10.21.48.

Fig 1

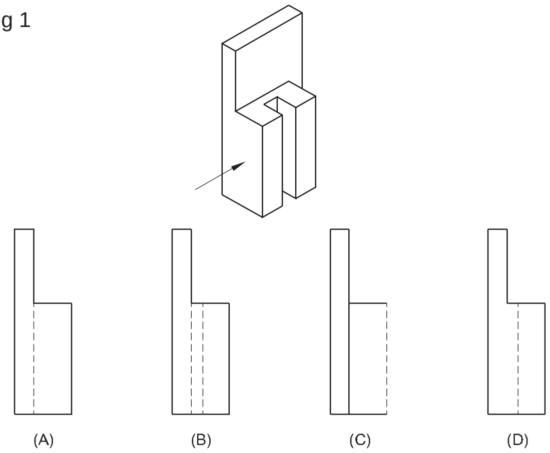


Fig 2

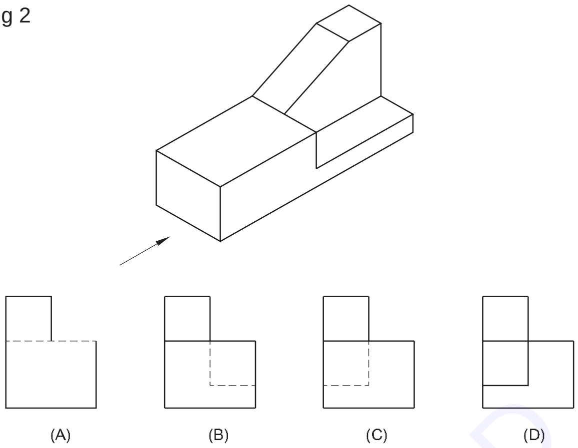


Fig 3

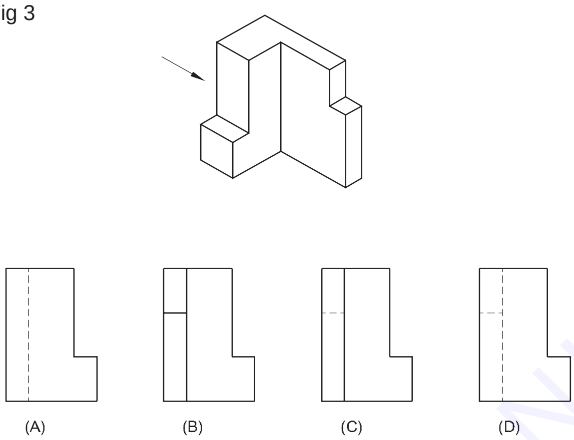


Fig 4

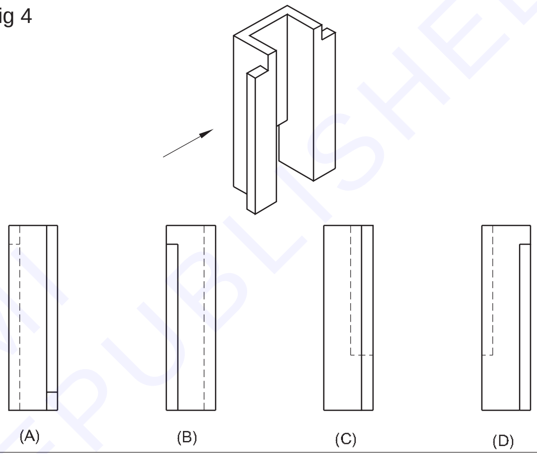


Fig 5

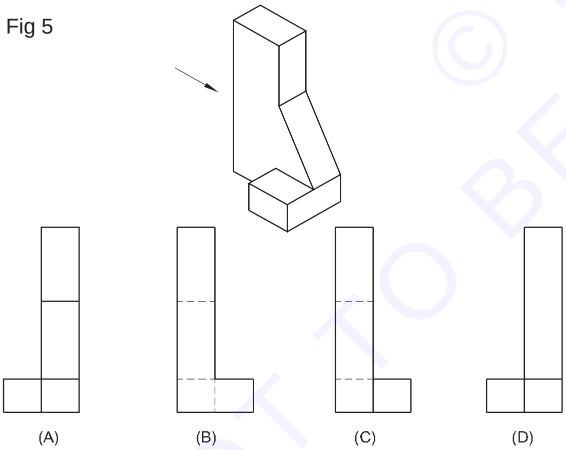


Fig 6

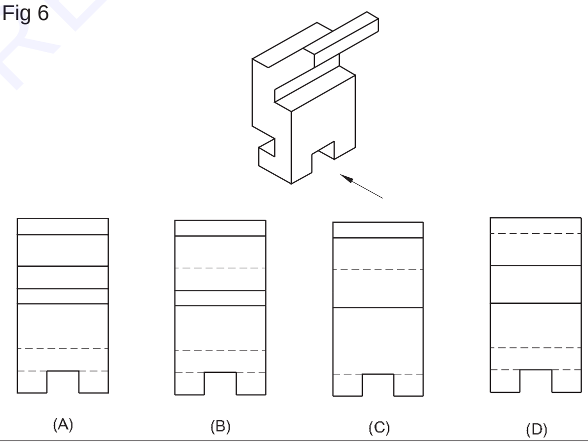


Fig 7

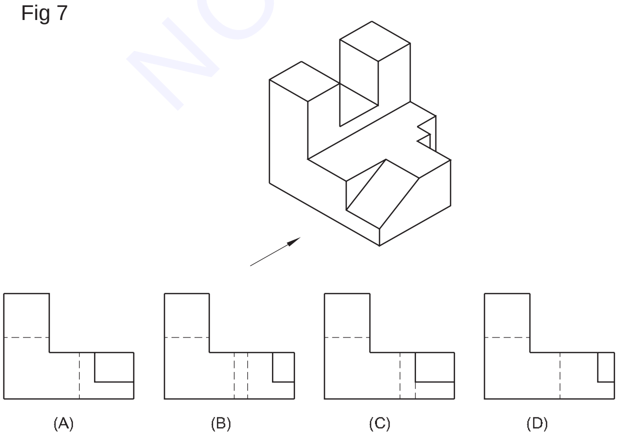


Fig 8

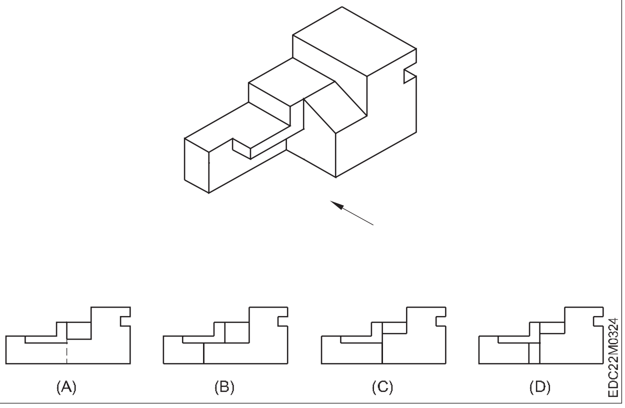
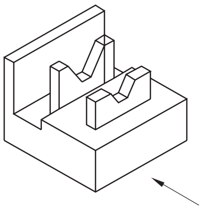
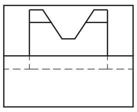
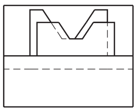


Fig 9

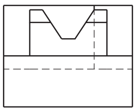




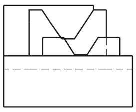
(A)



(B)

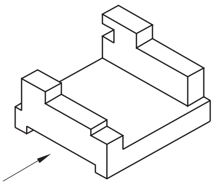


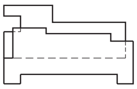
(C)



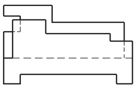
(D)

Fig 10

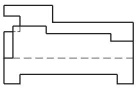




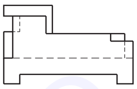
(A)



(B)

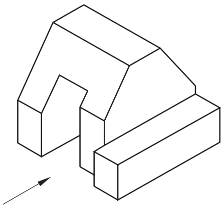


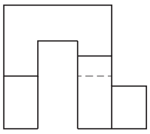
(C)



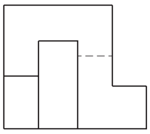
(D)

Fig 11

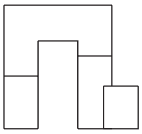




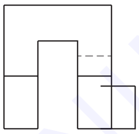
(A)



(B)

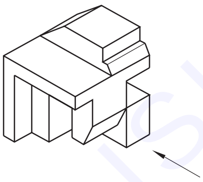


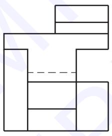
(C)



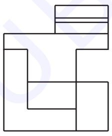
(D)

Fig 12

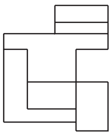




(A)



(B)



(C)

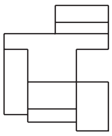


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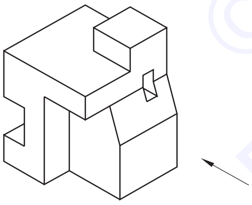
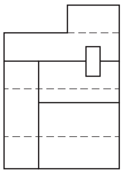
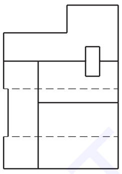
Fig 13

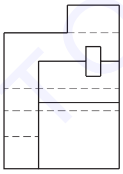




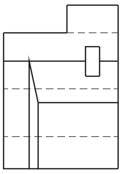
(A)



(B)

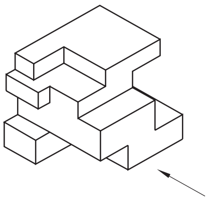


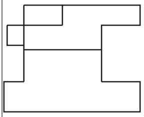
(C)



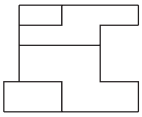
(D)

Fig 14

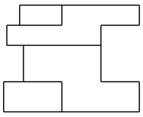




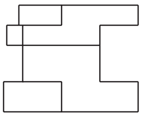
(A)



(B)

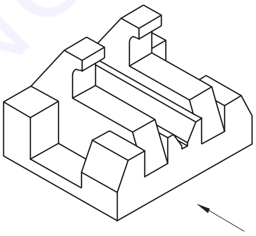


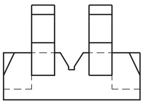
(C)



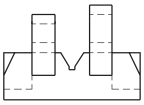
(D)

Fig 15

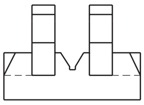




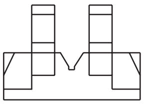
(A)



(B)

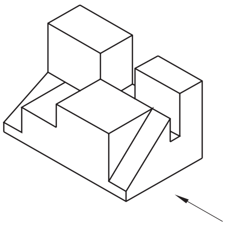


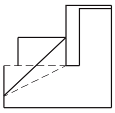
(C)



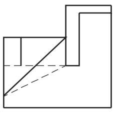
(D)

Fig 16

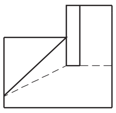




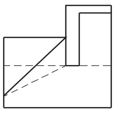
(A)



(B)



(C)



(D)

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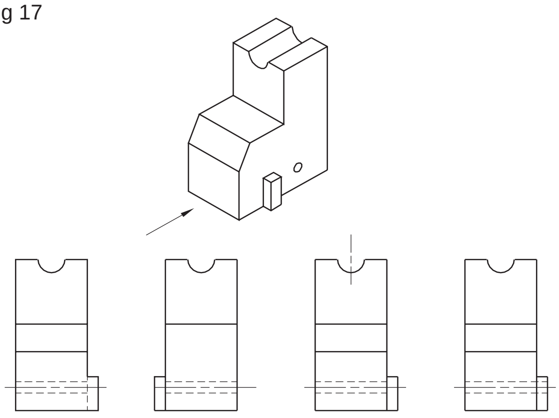
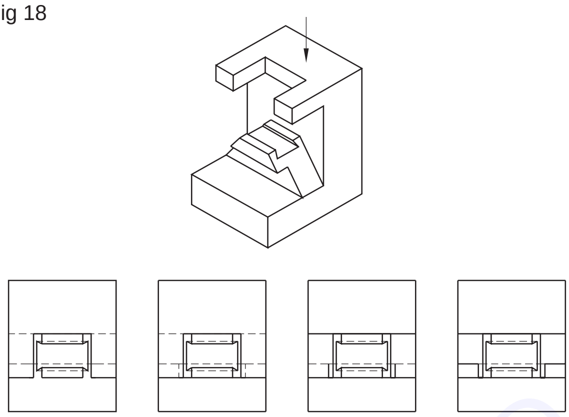
Fig 17



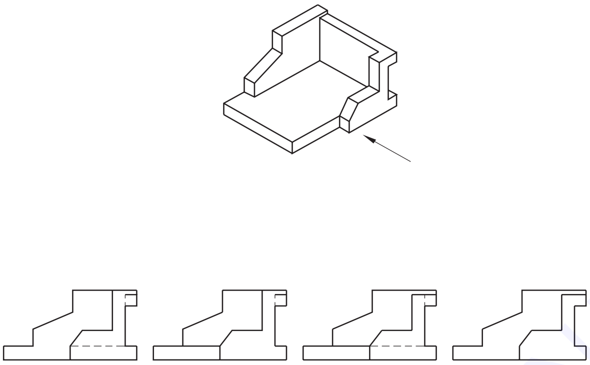
(A) (B) (C) (D)

Fig 18



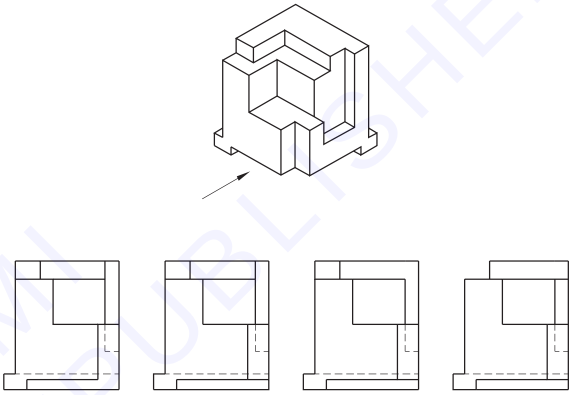
(A) (B) (C) (D)

Fig 19



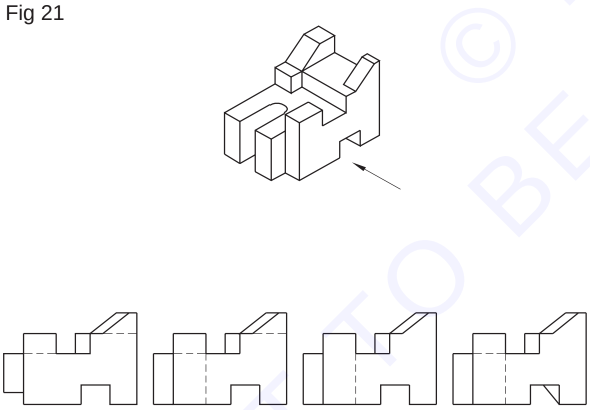
(A) (B) (C) (D)

Fig 20



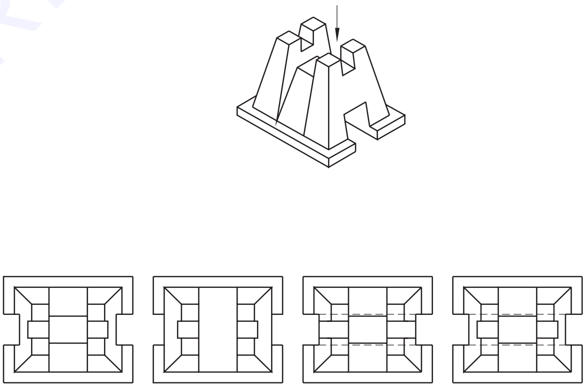
(A) (B) (C) (D)

Fig 21



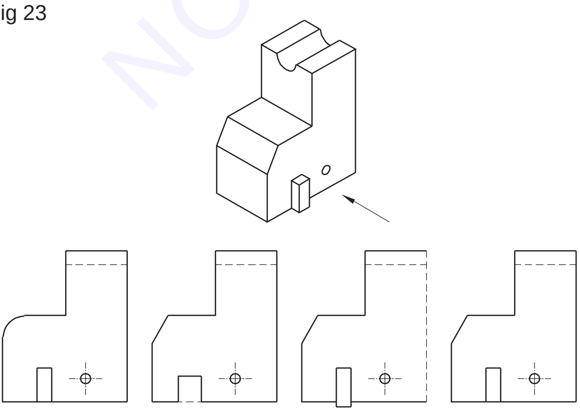
(A) (B) (C) (D)

Fig 22



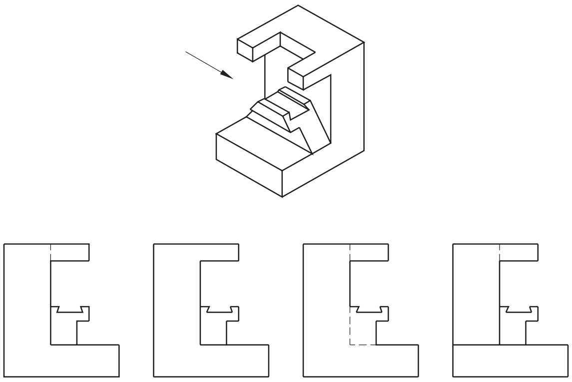
(A) (B) (C) (D)

Fig 23



(A) (B) (C) (D)

Fig 24



(A) (B) (C) (D)

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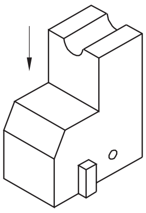
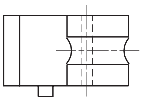
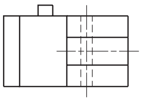
Fig 25

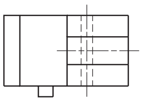




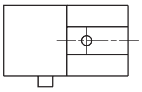
(A)



(B)

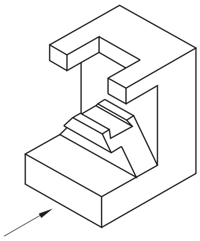


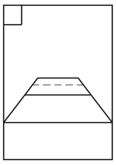
(C)



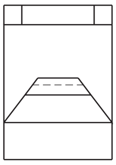
(D)

Fig 26

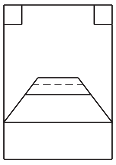




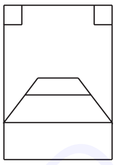
(A)



(B)

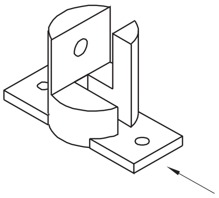


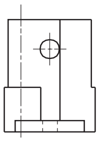
(C)



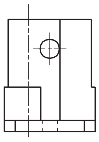
(D)

Fig 27

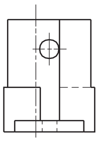




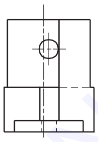
(A)



(B)

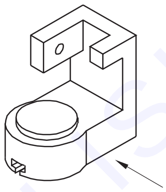


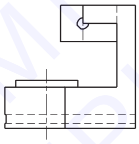
(C)



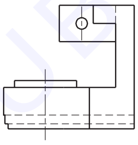
(D)

Fig 28

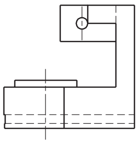




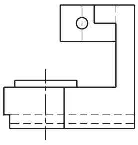
(A)



(B)

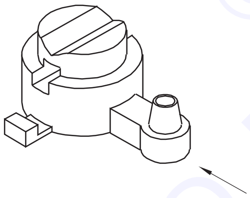


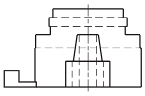
(C)



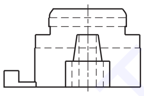
(D)

Fig 29

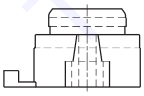




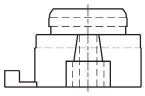
(A)



(B)

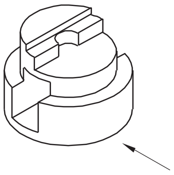


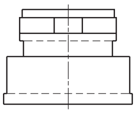
(C)



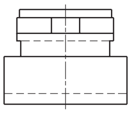
(D)

Fig 30

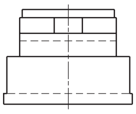




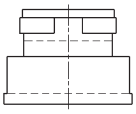
(A)



(B)

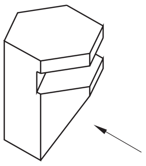


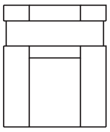
(C)



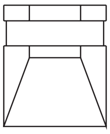
(D)

Fig 31

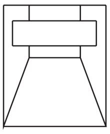




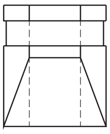
(A)



(B)

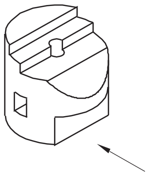


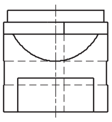
(C)



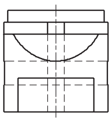
(D)

Fig 32

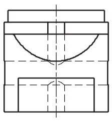




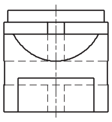
(A)



(B)



(C)



(D)

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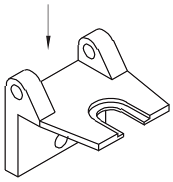
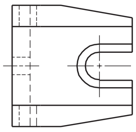
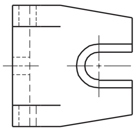
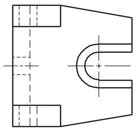
Fig 33




(A)


(B)


(C)

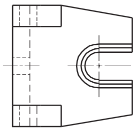
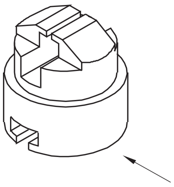
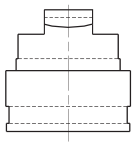
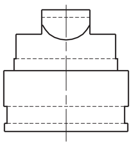
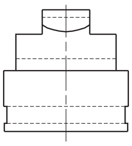

(D)

Fig 34




(A)


(B)


(C)

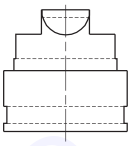
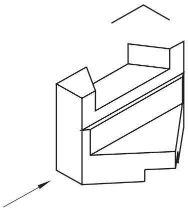
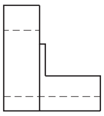
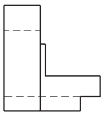
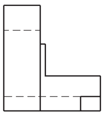

(D)

Fig 35




(A)


(B)


(C)

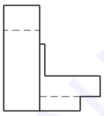
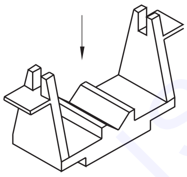
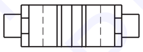
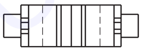
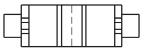

(D)

Fig 36




(A)


(B)


(C)

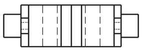
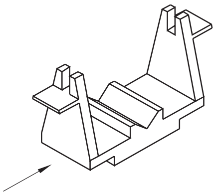
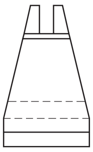
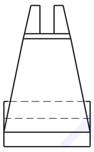
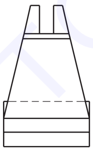

(D)

Fig 37




(A)


(B)


(C)

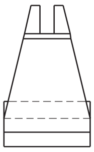
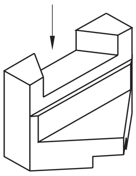
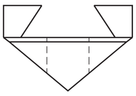
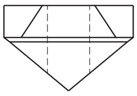
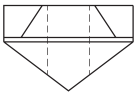

(D)

Fig 38




(A)


(B)


(C)

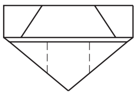
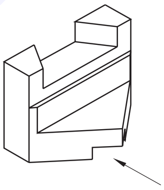
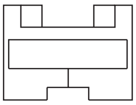
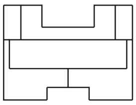
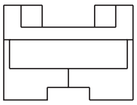

(D)

Fig 39




(A)


(B)


(C)

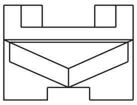
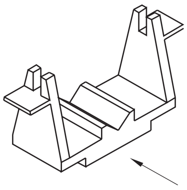
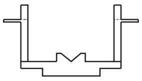
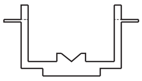
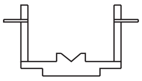

(D)

Fig 40




(A)


(B)


(C)

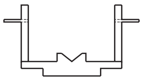

(D)

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Fig 41

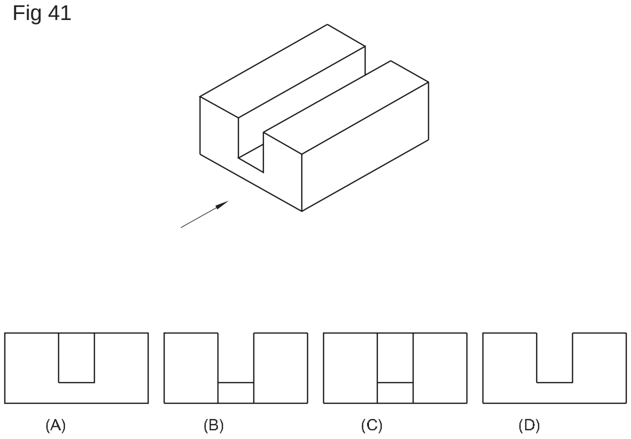


Fig 42

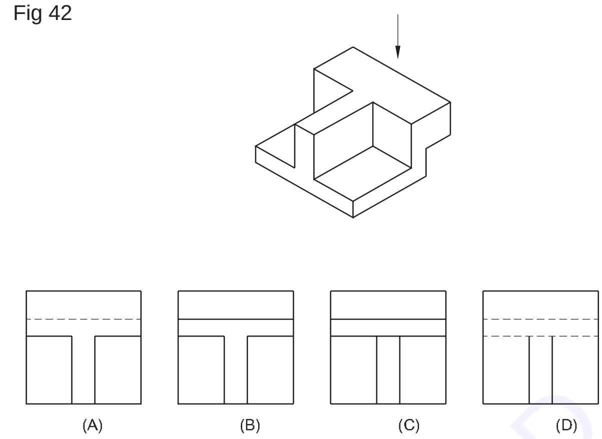


Fig 43

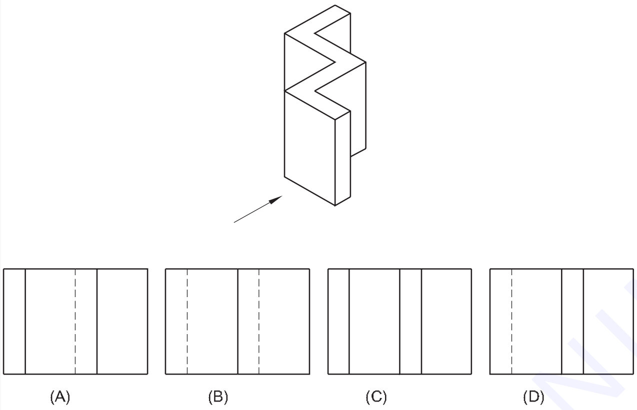


Fig 44

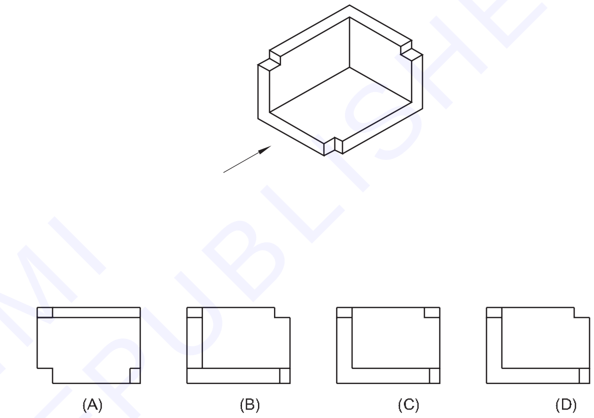


Fig 45

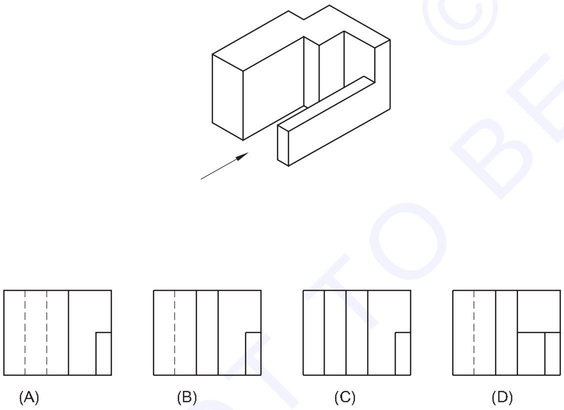


Fig 46

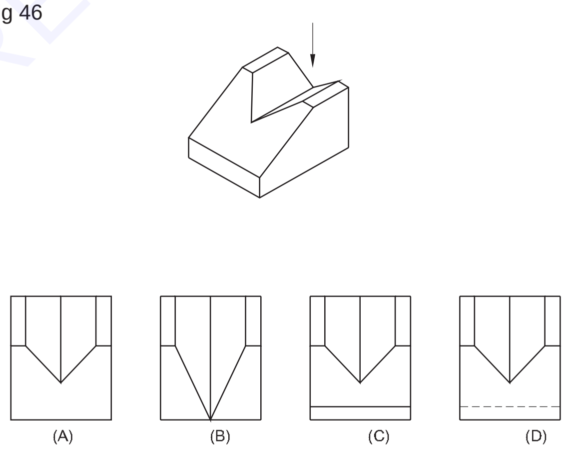


Fig 47

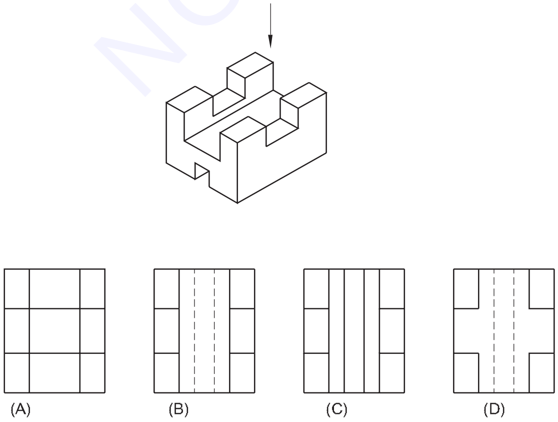
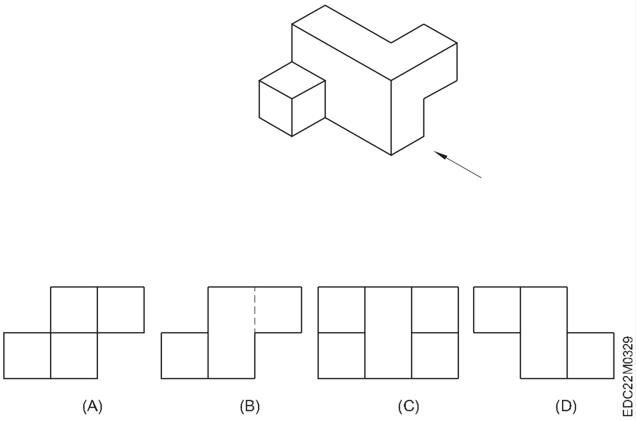


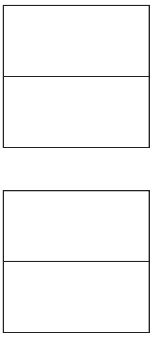
Fig 48



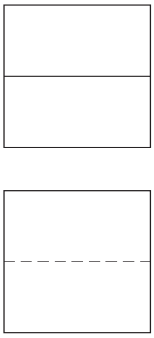
Exercise 10.22.01 to 10.22.12: Front and top view of twelve blocks are given. Select and enter the alphabet of side view concerning the front and top view in the work book - Ex.No.10.22.01 to 10.22.12.

FRONT & TOP VIEWS

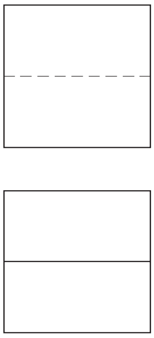
01



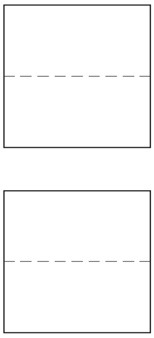
02



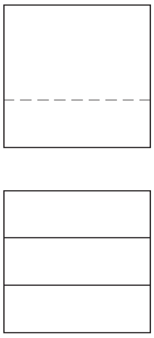
03



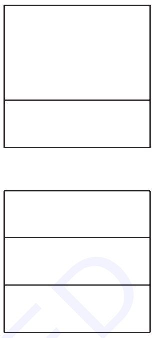
04



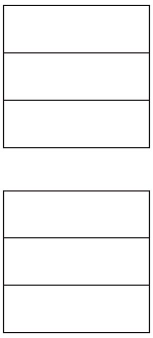
05



06

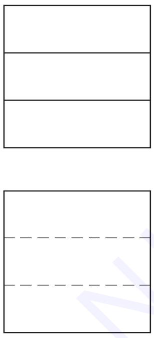


07

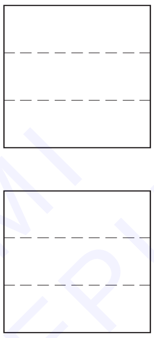


08

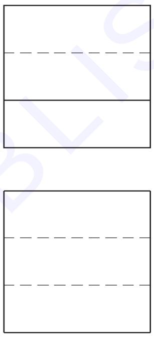
09



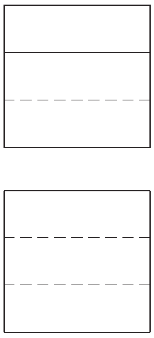
10



11

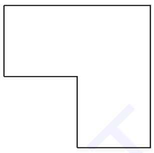


12

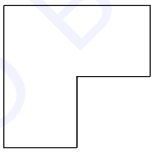


SIDE VIEWS

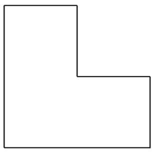
A



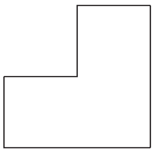
B



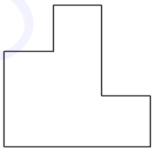
C



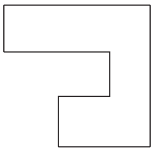
D



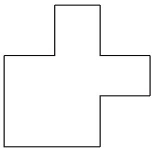
E



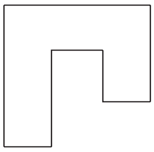
F



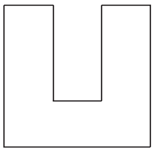
G



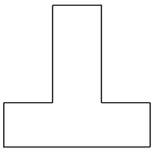
H



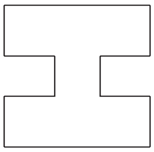
I



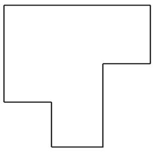
J



K



L



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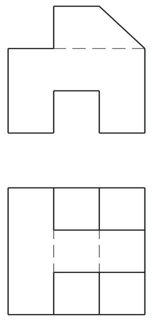
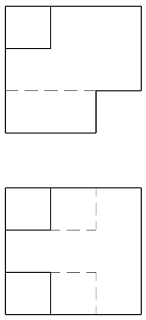
Exercise 10.22.13 to 10.22.20: Front and top view of eight blocks are given. Select and enter the alphabet of side view related to the front and top view in the work book - 10.22.13 to 10.22.20.

FRONT & TOP VIEWS

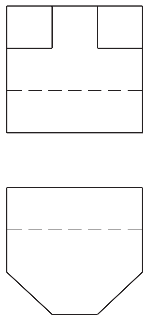
13



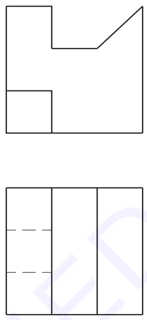
14



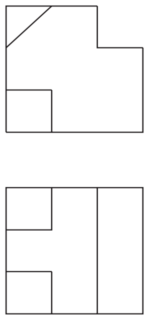
15



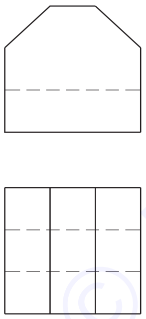
16



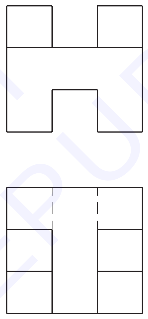
17



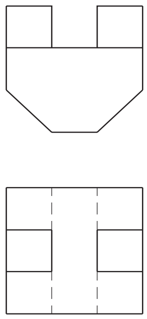
18



19

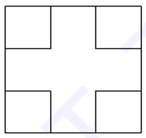


20

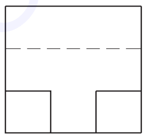


SIDE VIEWS

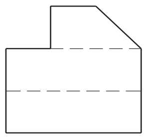
A



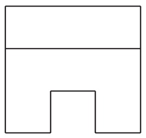
B



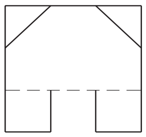
C



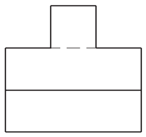
D



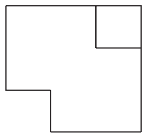
E



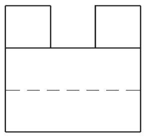
F



G



H

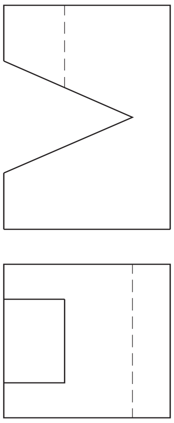


EDC22M0331

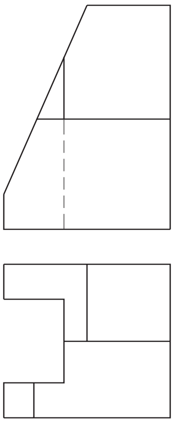
Exercise 10.22.21 to 10.22.28: Front and top view of eight blocks are given. Select and enter the alphabet of side view related to the front and top view in the work book - 10.22.21 to 10.22.28.

FRONT & TOP VIEWS

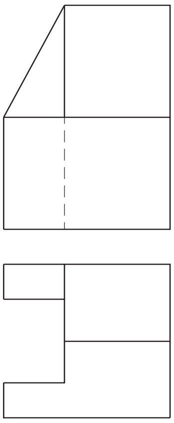
21



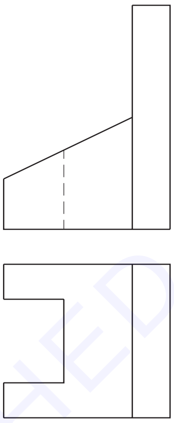
22



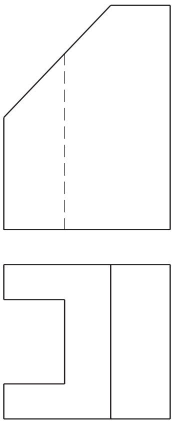
23



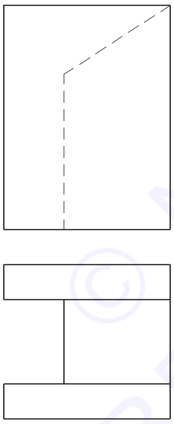
24



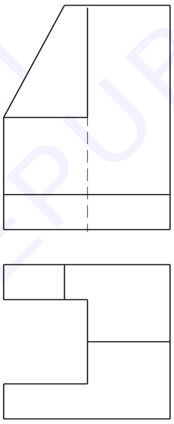
25



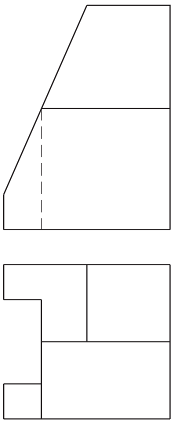
26



27

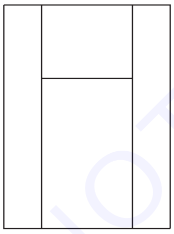


28

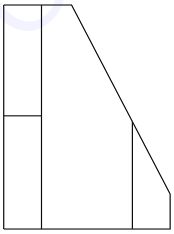


SIDE VIEWS

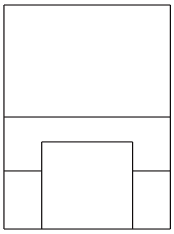
A



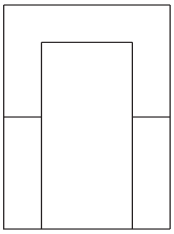
B



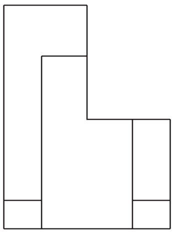
C



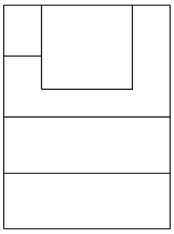
D



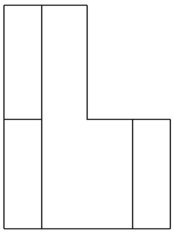
E



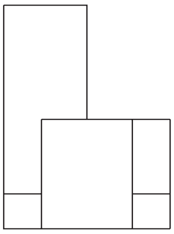
F



G



H

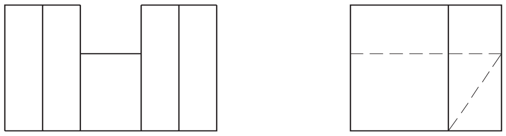


EDC22M0332

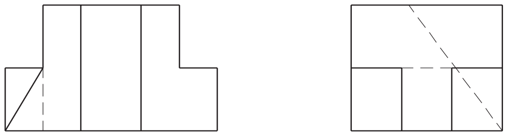
Exercise 10.23.01 to 10.23.06: Front and side view of cut blocks are given. Select and enter the alphabet of suitable top view in the work book - 10.28.01 to 10.23.06.

FRONT & SIDE VIEWS

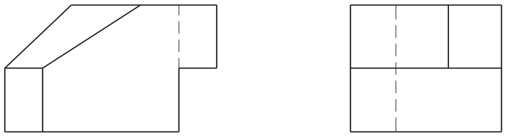
01



04

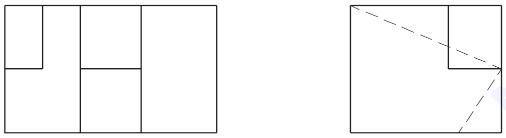


02

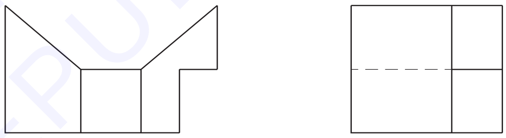


05

03

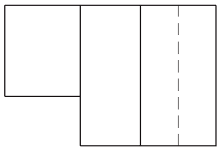


06

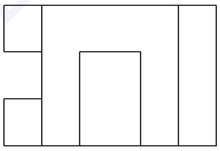


TOP VIEWS

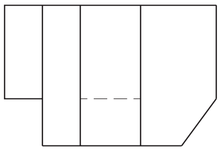
A



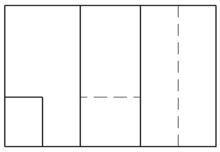
B



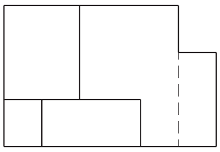
C



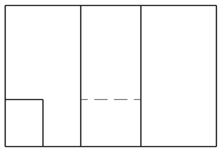
D



E



F

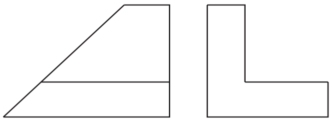


EDC22M0333

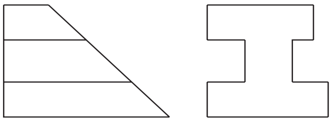
Exercise 10.23.07 to 10.23.18: Front and side view of blocks are given. Select and enter the alphabet of suitable top view in the work book - 10.23.07 to 10.23.18.

FRONT & SIDE VIEWS

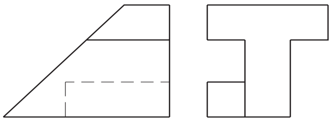
07



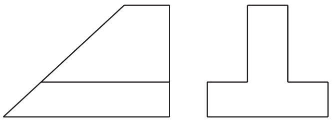
08



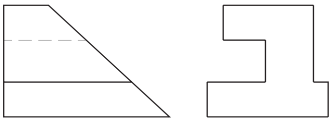
09



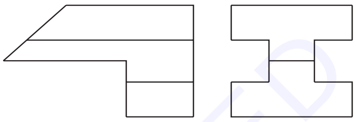
10



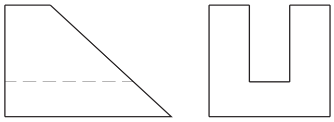
11



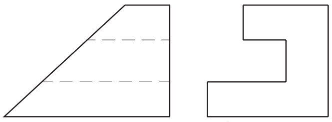
12



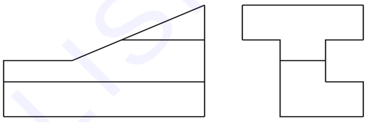
13



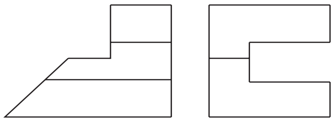
14



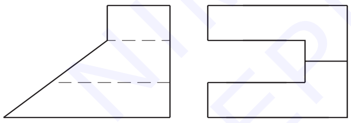
15



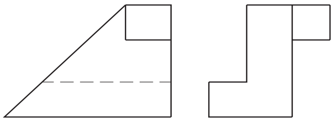
16



17

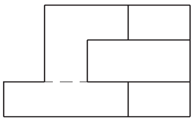


18

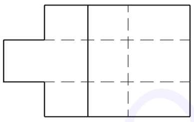


TOP VIEWS

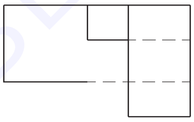
A



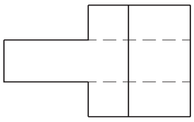
B



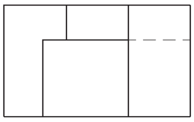
C



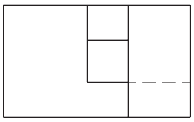
D



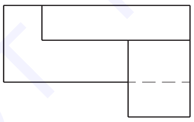
E



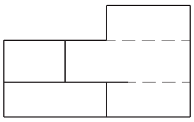
F



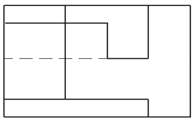
G



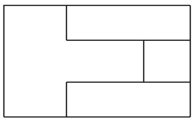
H



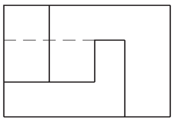
I



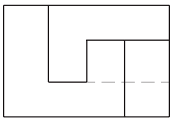
J



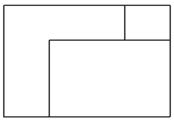
K



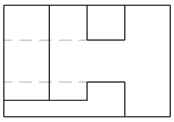
L



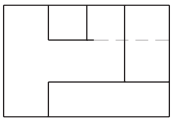
M



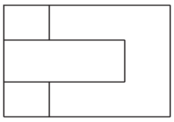
N



O

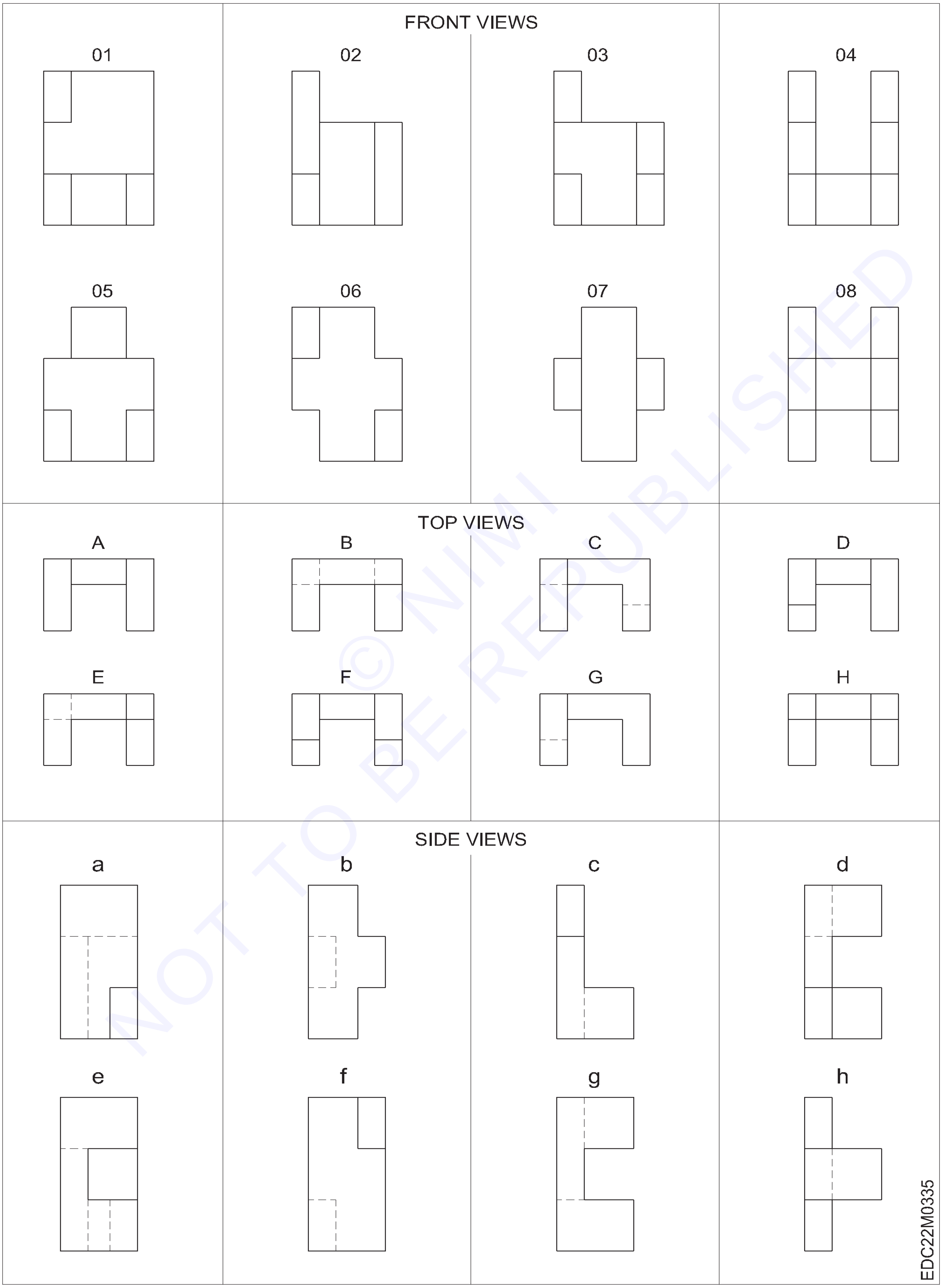


P



EDC22M0334

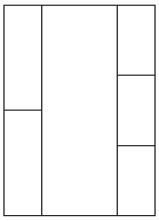
Exercise 10.24.01 to 10.24.08: Top views and side views at the given for eight front elevation. Match the top views and side views with the given front views and label them in work book - 10.24.01 to 10.24.08.



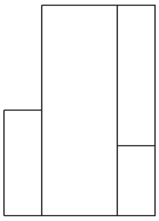
Exercise 10.24.09 to 10.24.16: Top views and side views are given for eight front elevation. Match the top views and side views with the given front views and lebel them in the work book - 10.24.09 to 10.24.16.

FRONT VIEWS

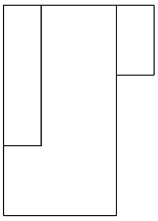
09



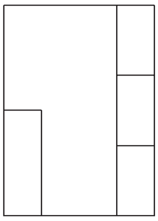
10



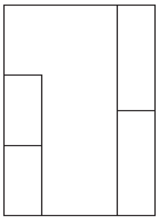
11



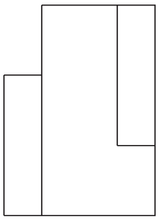
12



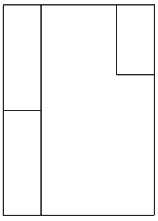
13



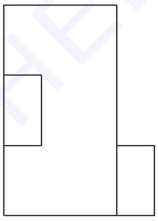
14



15

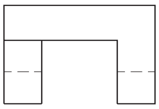


16

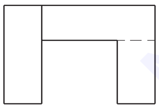


TOP VIEWS

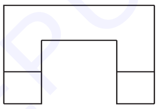
A



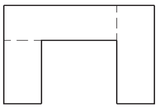
B



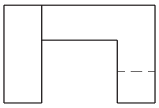
C



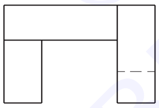
D



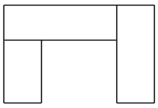
E



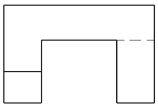
F



G

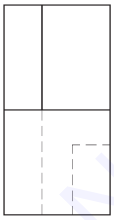


H

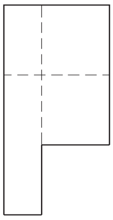


SIDE VIEWS

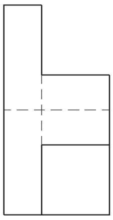
a



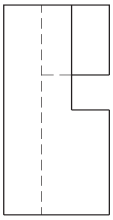
b



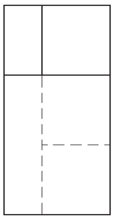
c



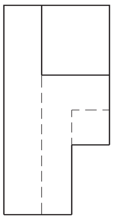
d



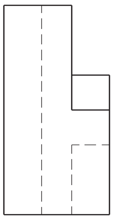
e



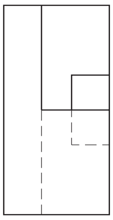
f



g



h



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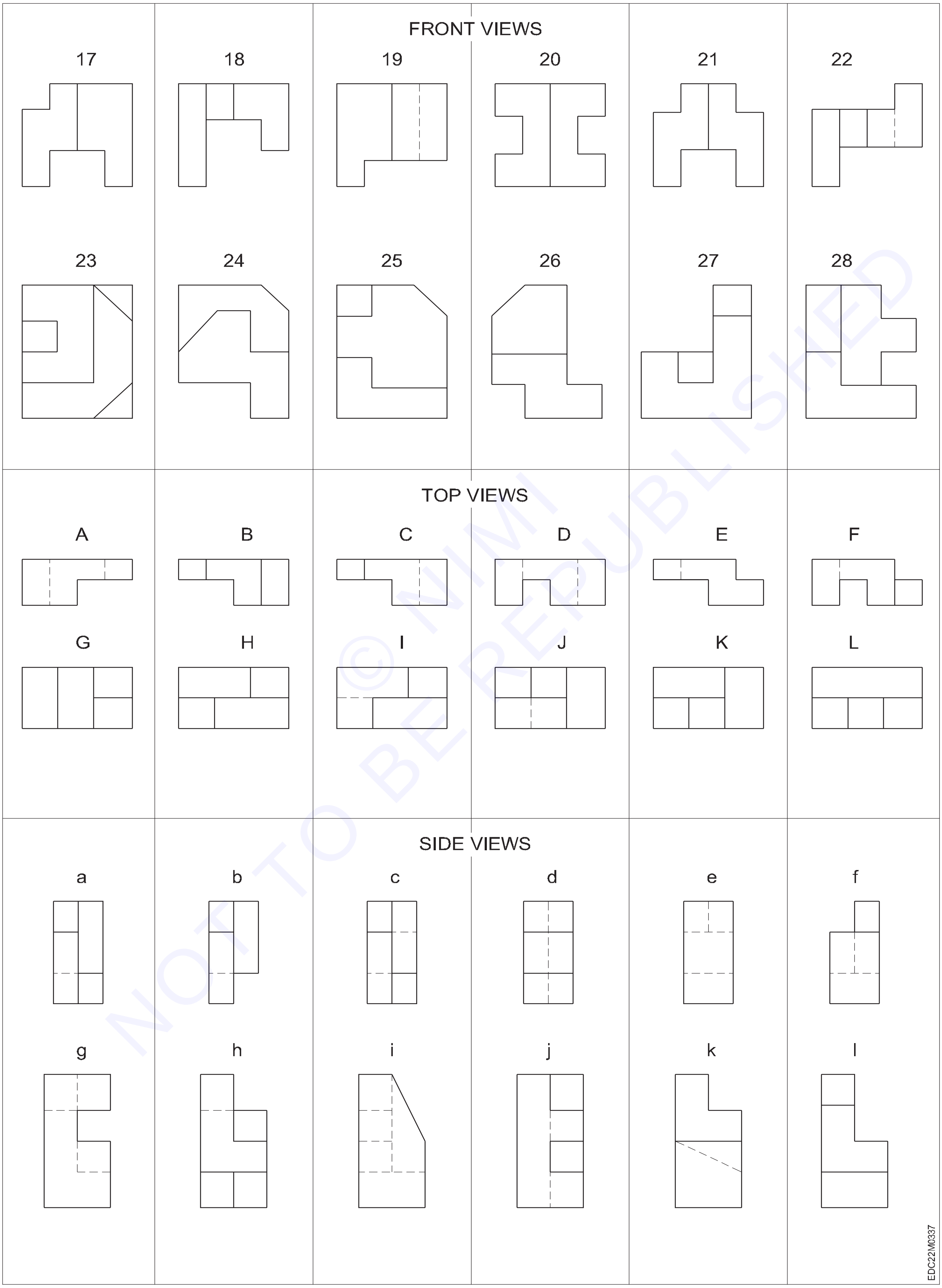
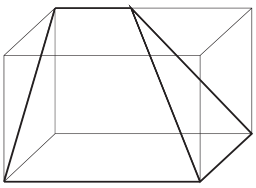
Exercise 10.24.17 to 10.24.28: Top views and side views are given for twelve front views. Match the top views and side views with the given front views and answer them in work book - 10.24.17 to 10.24.28.



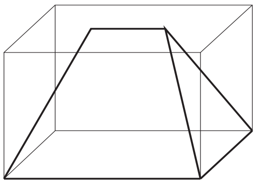
Exercise 10.25.01 to 10.25.08: Match front and top view and the side view with their oblique drawing. Give your answer (Capital & small letters) in the response sheet of this exercise in the work book.

CUT BLOCKS IN OBLIQUE VIEW

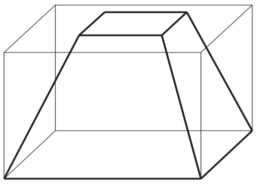
01



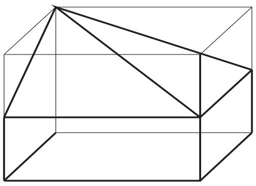
03



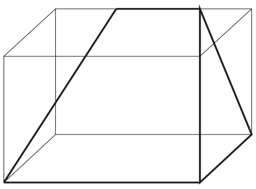
05



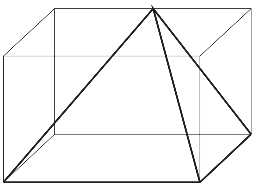
07



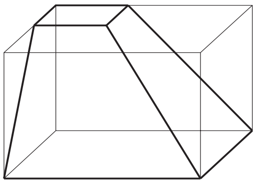
02



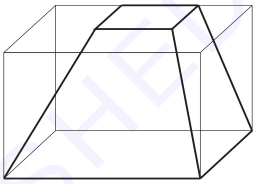
04



06

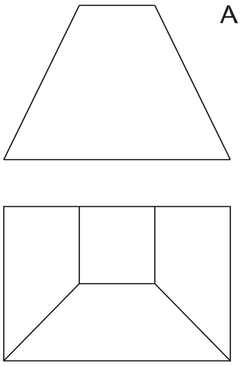


08

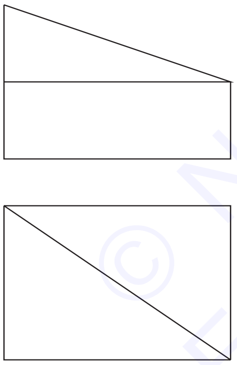


FRONT & TOP VIEW

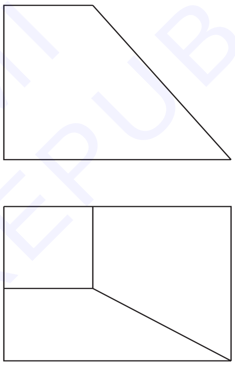
A



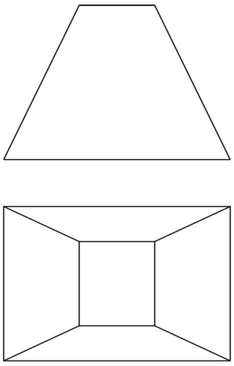
B



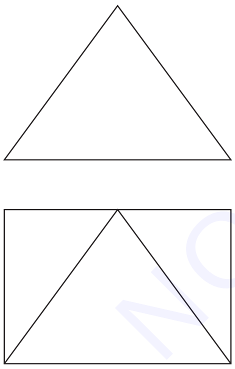
C



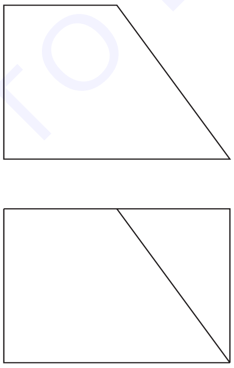
D



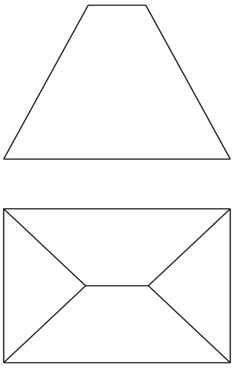
E



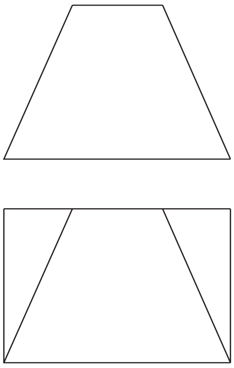
F



G

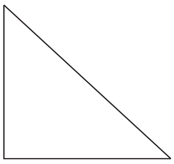


H

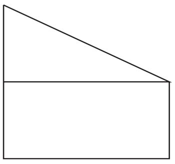


SIDE VIEW

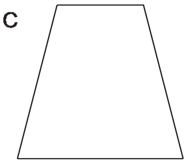
a



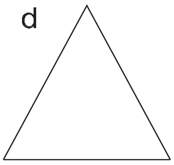
b



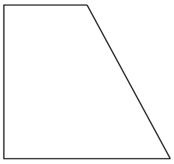
c



d



e



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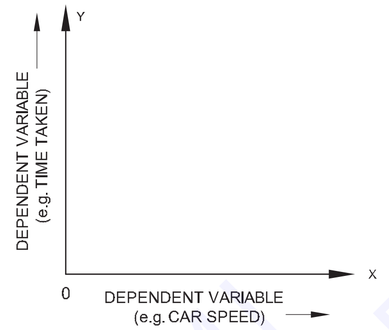
EXERCISE 11 : Graph & Charts

Graph & Charts

Concept

Representation of the relationship between two variable quantities on a squared paper known as graph paper is referred as plotting of graphs. A dependent variable is the one which depends upon another variable called the independent variable. For example in a graph showing the relationship between the speed and time of a car that is moving to cover a certain distance, the speed is the independent variable and the time is the dependent variable.(Fig 1)

Fig 1



Axis and quadrants

'x' axis is the horizontal line drawn on the graph sheet, to represent the independent variable by means of a conveniently chosen scale. The 'y' axis is the vertical line drawn on the graph sheet, to represent the dependent variable by means of a conveniently chosen scale. The point of intersection of the two axes is called as the origin.(Fig 2&3)

Fig 2

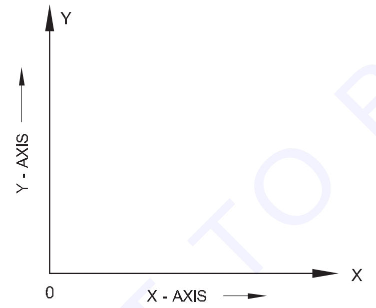
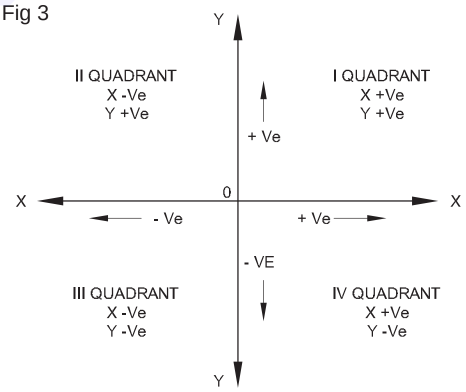


Fig 3



To plot the graph to represent the two variables, which may have both positive both negative, positive and negative values, the graph sheet is conveniently divided into four quadrants and the origin is also conveniently fixed. To plot positive and negative values of 'x' and 'y', the two variables the rule is

+ve values of 'x': to the right of the origin along the horizontal axis.

-ve values of 'x': to the left of the origin along the horizontal axis.

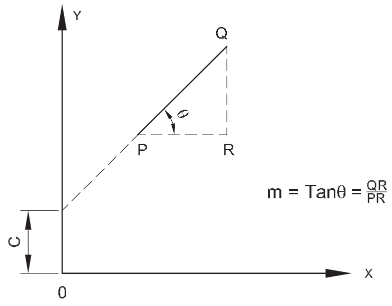
+ve values of 'y': above the origin along the vertical axis.

-ve values of 'y': below the origin along the vertical axis.

The straight line graph

The direct relationship between the two variables is represented by a straight line graph. The formula in the form of y = mx + c denotes a straight line graph of the relationship between y and x. 'm' is the slope of the straight line with the horizontal and 'c' is a constant. If the straight line passes through the origin 'c' will be equal to zero. (Fig 4)

Fig 4



Example

To draw a graph illustrating the relationship between inches and millimeters.

X	inch	1	2	3	4	5	6
Y	millimeter	25.4	50.8	76.2	101.6	127	152.4

The scale chosen for the representation of the points by plotting and joining of the points to get the straight line graph is shown in the graphics.

In some graphs the points located are to be joined by a smooth curve.

Example

The indicated horsepower of a marine diesel engine for different speeds of the vessel is given in the table below. To draw a graph to show the relationship and to determine the horsepower when the speed is 9 knots. (Fig 5)

X	Speed in knots	7.5	8.8	9.5	10.2	11.3
Y	i.h.p	750	1100	1350	2000	2400

In this example the 'x' axis scale can be commenced from 7.5 itself and the 'y' axis from 700, so that the origin will be 7.5, 700 instead of 0, 0. After plotting the graph to find the horsepower when the speed is 9 knots, draw a vertical line from the point on the 'x' axis at 9 knots to intersect the plotted curve. Draw a horizontal line from this intersecting point to meet the 'y' axis. Read the value and that is the answer. It is 1160.

$3x - y = 7$

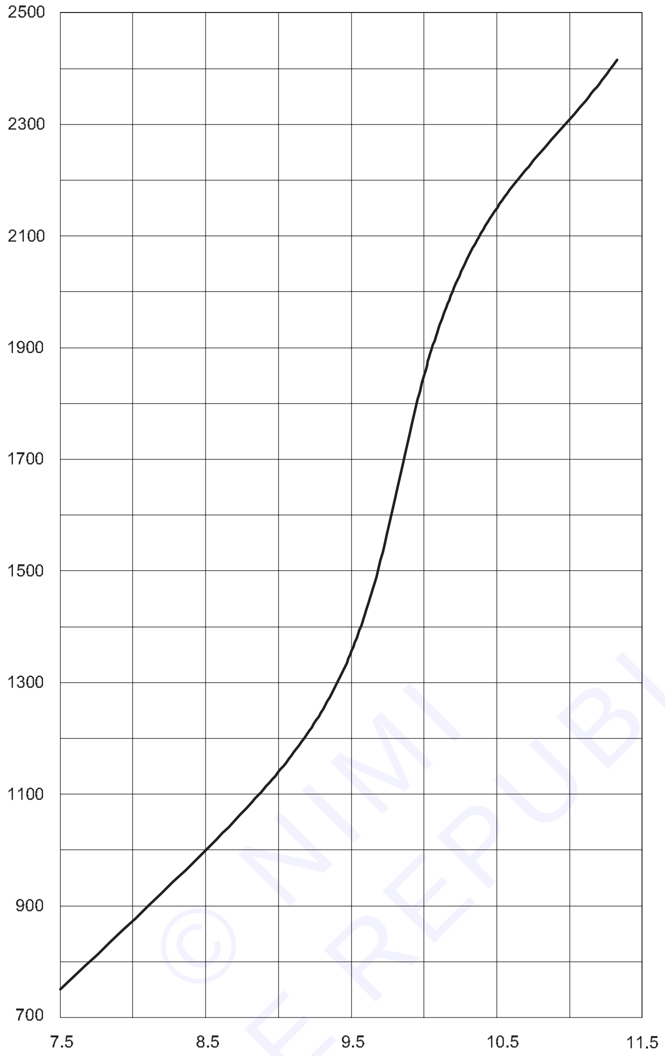
$4x - y = 10$

To find the values of 'x' and 'y' graphically.

Equation 1

X	3	4	5	6	7	8
Y	2	5	8	11	14	17

Fig 5



Equation 2

X	3	4	5	6	7	8
Y	2	6	10	14	18	22

Solving a simultaneous equation

The intersecting point of the two lines gives the values of
x and y.x = 3 and y = 2.
Points to be remembered when plotting

- Indicate the name or symbol of the quantity plotted along each axis, with its unit.
- Indicate clearly the scale division on each axis.
- Choose a convenient scale.
- Mark clearly the points plotted.
- Draw the axes lines thicker than the plotted and joined line/curve for distinguishing.

Types of Graphs and Their Uses

Every graph is a visual representation of data. This article describes five common types of statistical graphs widely used in any science.

1 Line Graph

A Line Graph displays data that change. Every Line Graph consists of data points that are connected. The purpose of connecting their lines is to help illustrate a trend, for example, a change or other pattern.

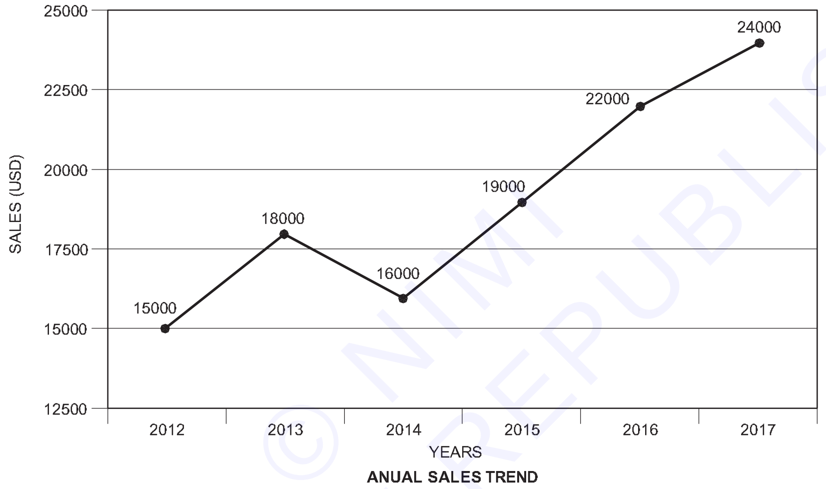
Uses of Line Graphs (Fig 1)

When you want to show trends over time, for example, how house prices have increased over time

When you want to show cumulative growth or increase

The following Line Graph shows annual sales of a particular business company for the period of six consecutive years:

Fig 1



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The above Line Graph contains only one line. However, Line Graphs can illustrate more than one set of data, and therefore can contain more than one line.

2 Bar Graph

A Bar Graph represents discrete data with rectangular columns (or bars). Bar Graphs are among the most popular types of graphs in economics,

statistics, and marketing. They are commonly used to illustrate categories of data.

Each rectangular bar in a Bar Graph has a height corresponding to the values that they represent. The x-axis of a Bar Graph presents the discrete categories, and the y-axis shows a measured value.

Uses of Bar Graphs

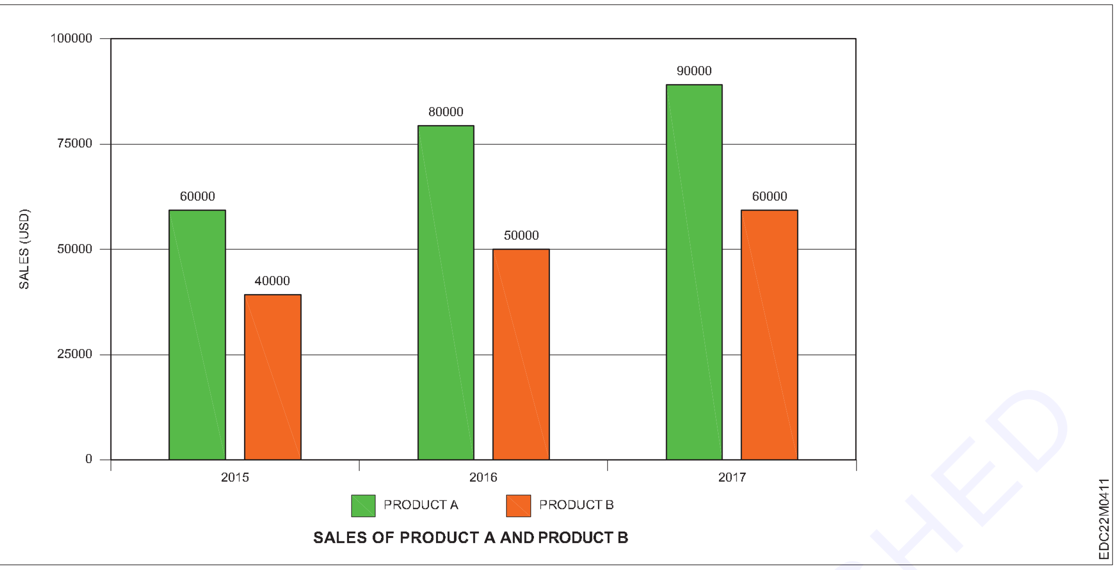
When you want to display data that are grouped into discrete categories

When you want to compare differences among categories

Example: The Bar Graph below illustrates the total sum of Sales of Product A and Product B for each of three years (three categories).

The bars in Bar Graphs can be vertical or horizontal. The above Bar Graph has vertical bars.(Fig 2)

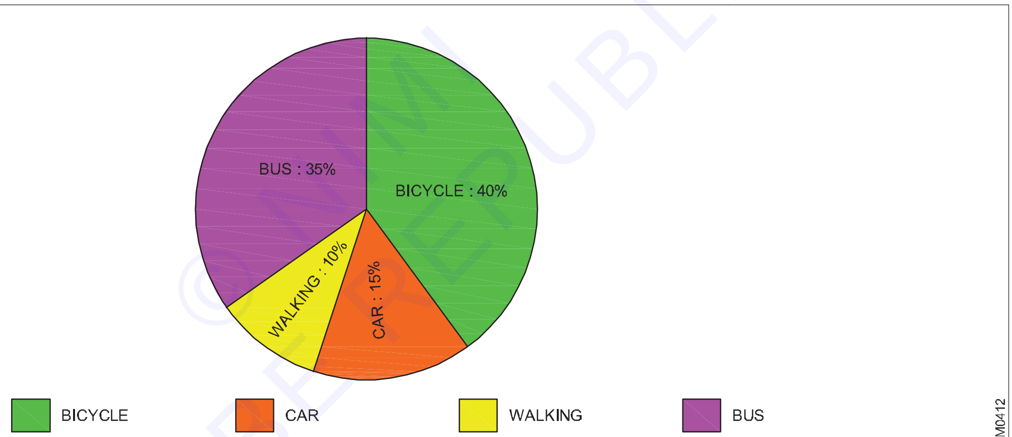
Fig 2



SALES OF PRODUCT A AND PRODUCT B

3 Pie Chart (Fig 3)

Fig 3



TYPES OF TRANSPORTATION TO SCHOOL

A Pie Chart displays data in a ‘pie-slice’ format and illustrate proportion.Each pie slice represents the size of one category relative to the size of other categories -- and in proportion to all the categories together (the whole pie).

Therefore, a Pie Chart illustrates part-whole relationships, and -- for every Pie Chart -- the whole pie should always add up to 100%.

When you want to create and represent the composition of something When you want to show percentages or proportional data

A Pie Chart works best for displaying data from four to seven categories.

Example: The pie chart below represents the proportion of types of transportation used by 1000 students to go to their school.

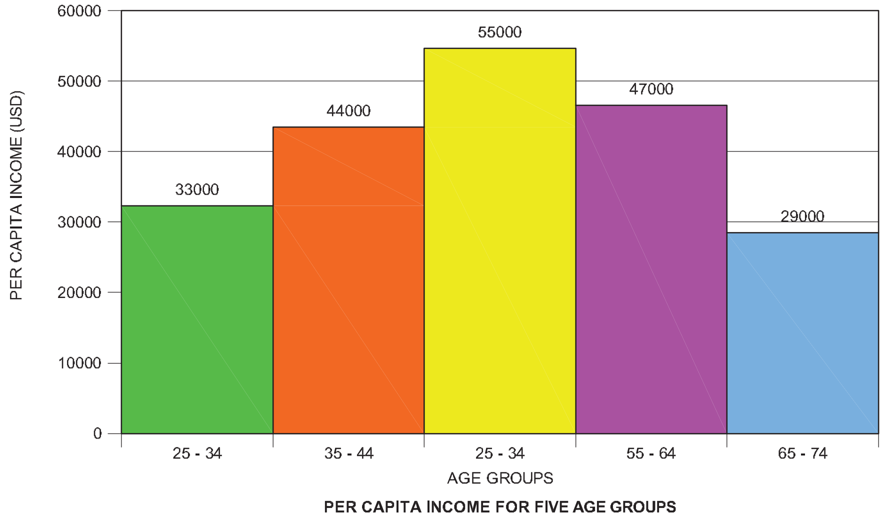
4 Histogram

A Histogram shows continuous data in ordered rectangular columns.

Usually, there are no gaps between the columns in a Histogram.

A Histogram displays a frequency distribution (shape) of a data set. At first glance, histograms look like bar graphs. However, there is a key difference between them. Bar Graphs represents categorical data and Histograms represent continuous data.(Fig 4)

Fig 4



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Uses of Histograms

When the data are continuous.

When you want to represent the shape of the data's distribution.

Example: The Histogram below illustrates per capita income for each of five age groups (bins or ranges).

Histograms are widely used in statistics, psychology, business, and economics.

5 Scatter Plot

A Scatter Plot is an x-y diagram that shows a relationship between two variables. It is used to plot data points on both the horizontal x-axis and the vertical y-axis.

The purpose of a Scatter Plot is to show the relationship between two variables.

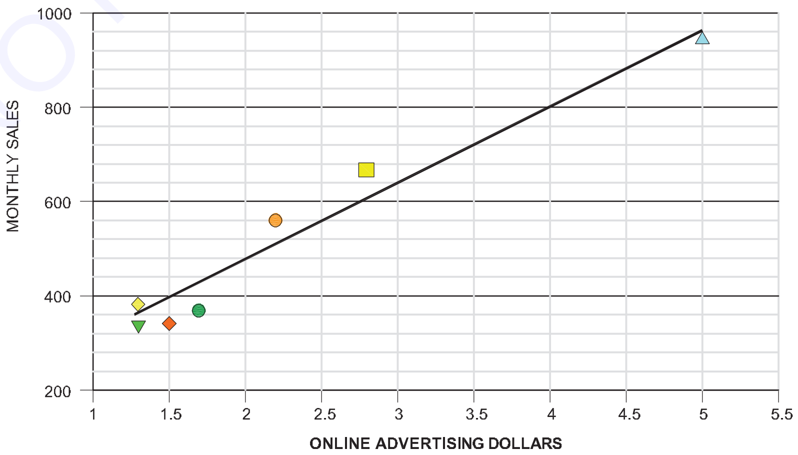
Sometimes, but not always, when there is a relationship between two variables, the first variable is called the independent variable, and the second variable is called the dependent because its values depend on the first variable. In these cases, a Scatter Plot allows you to visualize how one variable (the dependent variable) depends on the other variable (the independent).

Example: The Scatter Plot below represents the relationship between the monthly sales achieved by each of seven stores and the online advertising dollars spent by each of the same seven stores.

The orange line in the above Scatter Plot is called a "line of best fit" or a "trend line."

Scatter Plots are used widely in data science, statistics, and psychology.(Fig 5)

Fig 5



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EXERCISE 12 : Introduction of CAD

Introduction of CAD

Introduction

Computer are increase singly of a becoming a part of everyday life computers calculates our electricity and telephone bill find its useful applications in the field up medicine and medication assists various business organization system to keep their accounts and other basic jobs up to date its also provides the facility of playing games and surfing over the internet so as to gain to information on different filed into days time.

AutoCAD: AutoCAD is the leading computer-aided design and drafting (CAD) program in the world. Since its original introduction in November, 1982, AutoCAD has grown in sales and functionality to become the standard PC-based CAD program against which all other similar programs complete and against which they are judged. Over the years, AutoCAD has kept pace with developments in the computer industry. The program has grown from its original command line driven DOS-based roots to become a fully compatible windows application.

CAD hardware

The following are the main hardware components of CAD

- System unit
    - Central processing unit.
    - Memory.
    - Hard disk, CD-ROM pen drive.
- External storage devices.
- Monitor.
- Printers and plotters.
- Keyboard.
- Digitizer, puck and mouse.

System unit

The system unit is the computer that is used for all data processing. The main components of the system units are the central processing unit (CPU) and memory. In mainframe and minicomputers CPU and memory are usefully separate compartments that house thousands of devices. In today's PCs, however, they all fit in a small box commonly known as a desktop computer. Most desktop computers today come equipped with a hard disk, and CD-ROM. Let us have a look at the components of a system unit:

- Central processing unit.
- Memory.
- Hard disk, CD-ROM.

External storage devices: There are a number of external storage devices available such as magnetic tapes, zip drives and removable hard disks. They are commonly used to keep backup copies of electronic files for safekeeping.

Magnetic tapes are quite common for storing large volumes of data. A magnetic tape that looks like a small videocassette can store thousands of megabytes of data. However, they are quite slow and require a lot of time to store or retrieve data.

The new option for data storage is the removable hard disk. You can remove the entire hard disk from your computer and use it on another computer. This approach is commonly used when you need to work on different computers and you want the same information to be available instantly.

Computer Aided Design (CAD): Is simply, design and drafting with the aid of a computer. Design is creating a real product from an idea. Drafting is the production of the drawings that are used to document a design. CAD can be used to create 2D or 3D computer models. A CAD drawing is a file that consists of numeric data in binary form that will be saved onto a disk.

Why should you use CAD?

Traditional drafting is repetitious and can be inaccurate. It may be faster to create a simple "rough" sketch by hand but larger more complex drawings with repetitive operations are drawn more efficiently using CAD.

Why use AutoCAD?

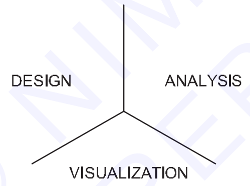
AutoCAD is a computer aided design software developed by Autodesk Inc. AutoCAD was first introduced in 1982. By the year 2000,it is estimated that there were over 4 million AutoCAD users worldwide.

What this means to you is that many employers are in need of AutoCAD operators. In addition, learning AutoCAD will give you the basics for learning other CAD packages because many commands, terms and concepts are used universally.

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

In general, a Computer Aided Design (CAD) package has three components: a) Design, b) Analysis, and c) Visualization, as shown in the sketch. A brief description of these components are follows as Fig 1.

Fig 1



COMPONENTS OF COMPUTER AIDED DESIGN

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Hardware and Software overview: There are two parts of a computer system, hardware and software, and a CADD system in no exception. Computer hardware is the physical components of the computer such as system unit, monitor and plotter. Computer software is the program that determines the application of a system.

There are three main categories of computers with respect to hardware:

- Mainframe.
- Minicomputer.
- Microcomputers, for example personal computers (PCs).

CADD software: A CADD program contains hundreds of functions that enable you to accomplish specific drawing tasks. A task may involve drawing an object, editing and existing drawing, displaying a view of the drawing, printing or saving it, or controlling any other operation of the computer. The functions contain a number of commands that enable you to specify exactly what you want to do and how you want to do it.

The functions are organized into modules that provide easy access to all the commands. The program is divided into modules such as draw, edit, data output, function control, data storage and management. A program may also have a number of specialized functions such as layers, database and 3D. Let s have a look at the CADD modules.

- Draw.
- Edit.
- Data output.
- System control.
- Data storage and management.

- Special features.

Draw: The draw module provides access to all the drawing functions of CADD. Whenever you need to draw something this group of functions is used. The draw module enables you to draw lines, arcs, circles, ellipses, text, dimensions, symbols, borders and many other drawing components.

Draw is CADD s most frequently used module because all drawing work is accomplished using it.

Edit: The edit module lets you change existing drawing elements and manipulate them in a number of ways. You can move, copy or erase drawing components. You can enlarge or reduce the sizes of diagrams or change the colour and line type of drawing components. You can also change the size and style of text and dimensions, as well as edit a dimension to show different units of measurement. A good CADD program is designed to change the appearance of all drawing elements created with CADD.

The edit functions also act as convenient drawing-aid tools. They enable you to join missing corners of lines, trim drawing components along a line, stretch them to fit a new shape, etc. The list of editing capabilities goes on and on. The edit functions make CADD a dynamic drawing tool.

Data output: The data output module enables you to display drawings on the screen and then print them on paper. There are two separate sets of functions that help accomplish this:

- **View-display functions.**
- **Print/plot functions.**

The view-display functions allow you to display different views of a drawing on the screen. These functions are used quite often, because every time you need to draw something or edit something, you need to focus on that portion of the drawing. With the help of view-display functions, you can zoom in on a specific portion of the drawing.

The print and plot functions allow you to print drawings using a printer or a plotter. You can control many aspects of printing and plotting. You can print the same drawing in different sizes by applying the appropriate scale factor. You can plot the drawings with specific colours, pen thickness, and line types.

Data storage and management: The data storage and management module allows you to store and manage drawing data. Through the use of the functions in this module, you can store drawings as files on the hard disk. You can manage the files in directories and sub-directories, and move, copy or delete them as needed.

CADD data management functions also let you translate drawings created by other CADD programs.These functions convert drawing data to a generic format that can be read by any CADD program. Data exchange format (DXF) is one of the common data translation formats used by CADD program. There are a number of data exchange formats available.

System control: The system control module (also known as system defaults) allows you to control how CADD works. CADD programs are designed for a broad range of professionals, including architects,designers, engineers and surveyors. With the help of system control functions, you can set the working environment of CADD to suit your needs.

Example: You can set the type of units that you will be using, the accuracy of the units, a style for dimensions and text, colours, layers, line type in a drawing, etc. Additionally, you can customize screen menus, the display of colours on the screen, resolution of the screen, size, the speed of the cursor, etc.

You can also specify whether the selected defaults should apply to a single drawing, to a specific project or to all the projects in a specific category. The defaults can be set on a temporary or permanent basis.

Special features: CADD programs usually offer a number of special features that make working with CADD easier and allow you to automate many drawing tasks. For example, you can create layers in a drawing that allow you to segregate drawing components. You can develop spreadsheets and databases that can be used to create many types of project reports. You can create three-dimensional (3D) drawings, such as isometrics and perspectives, with the help of 3D functions. You can also accomplish many other automated tasks with the help of macros.

The number of special features a CADD program has or how elaborate they are various from one program to another. Some vendors sell specialized features as separate packages, while others include them in a single package. It all depends how a program is written, how big or small it is, and how it is sold.

CADD user interface: CADD user interface provides the environment and the tools that allow you and the computer to communicate. Each CADD program establishes an environment that best suits its purpose. The goal is to make working with CADD efficient. Most programs use a Graphic User Interface (GUI) to communicate with the user. The

GUI provides visual aids for quick data entry. You are given tools to select functions, enter textual or mathematical data, locate points in the drawing window, select objects in the drawing window, etc.

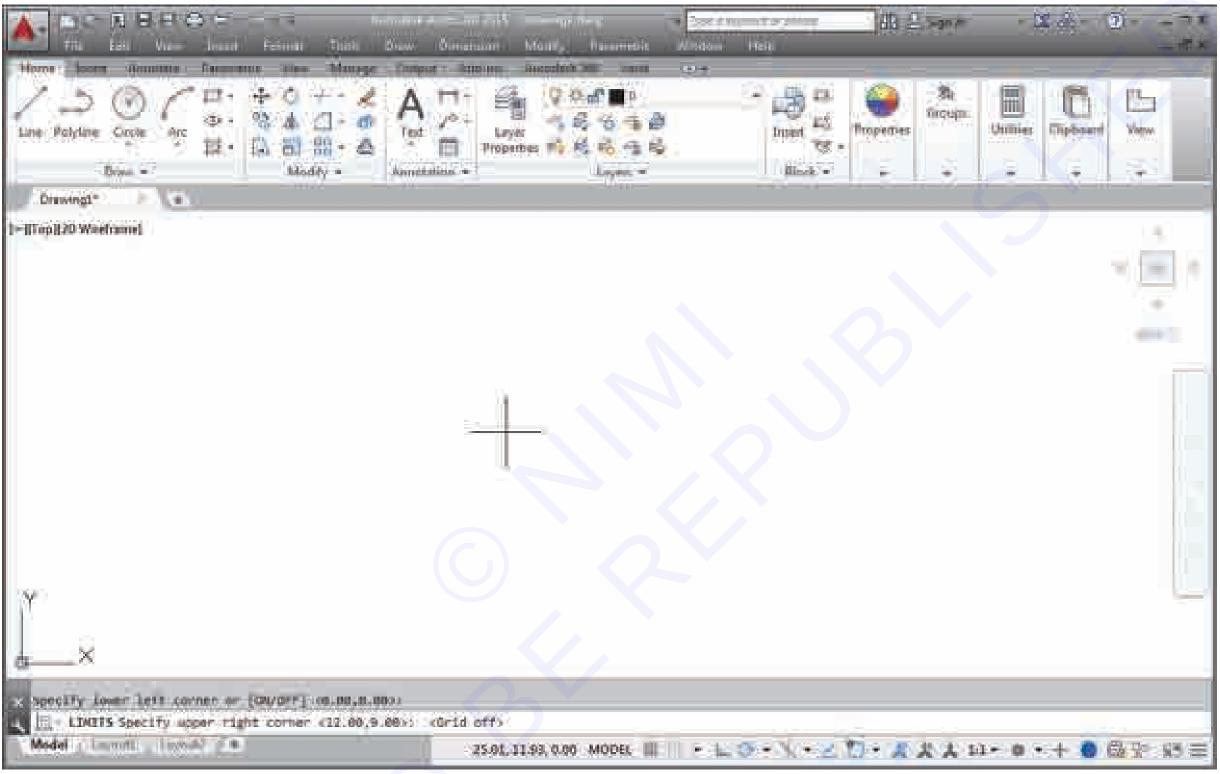
Introduction of CAD

Introduction

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

Graphical user interface (GUI) OF Auto CAD (Figs 1, 2 & 3)

Fig 1



```
Specify lower left corner or [ON/OFF] <0.00,0.00>:
LIMITS Specify upper right corner <12.00,9.00>: <Grid off>
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Fig 2

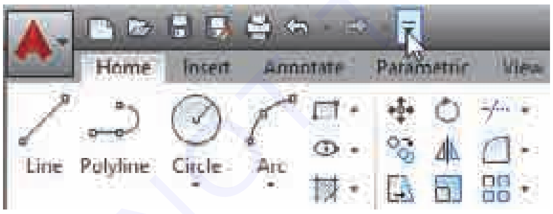


Fig 3

Quick access toolbar (Fig 4)

1 Click on one of the following icons for quick access to commands QNEW, OPEN, SAVE, PLOT, and UNDO/REDO.

Fig 4

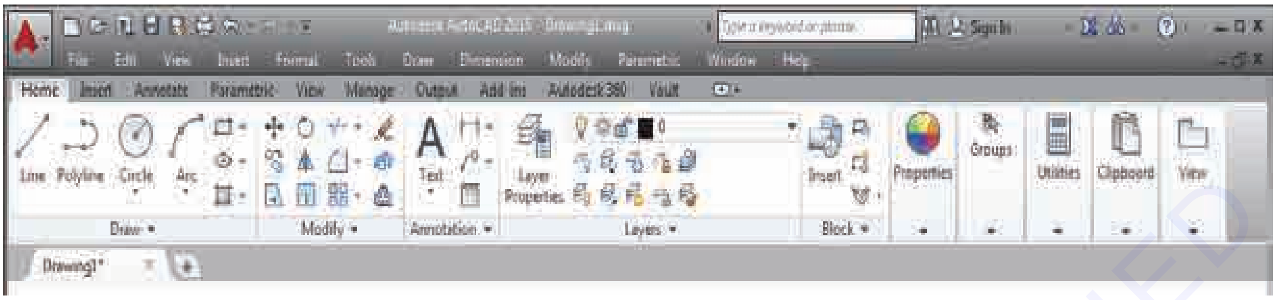
Right-click the quick toolbar and click customize quick access toolbar. The customize user interface dialog opens and displays the list of commands available.

Drag commands you want to add from the command list pane in the customize user interface dialog box to the quick access toolbar.

Info center: Quickly search for a variety of information sources, access product updates and announcements, and save topics with info center.

Ribbon: The ribbon provides a single, compact placement for operations that are relevant to the current workspace. It eliminates the need to display multiple toolbars, reducing clutter in the application window. The ribbon maximizes the area available for work using a single compact interface. (Fig 5)

Fig 5



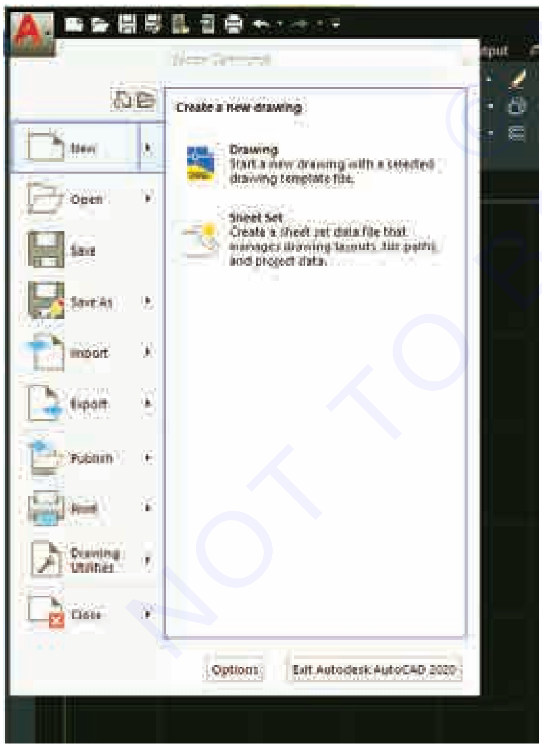
The ribbon can be displayed horizontally, vertically, or as a floating palette. The horizontal ribbon is displayed at the top of the drawing window by default when you create or open a drawing.

You can create your own panels to display on the ribbon; you can also modify the commands and controls on existing ribbon panels.

1.5 Menus and colours.

Menu browser (Fig 6)

Fig 6



Create a new drawing

Drawing
Start a new drawing with a selected drawing template file.

Sheet Set
Create a sheet set data file that manages drawing layouts, file paths and project data.

1 Click on the a icon in the upper left corner of the drawing area.

2 Click the desired pull down menu.

3 Click on the command to be executed from the pull down.

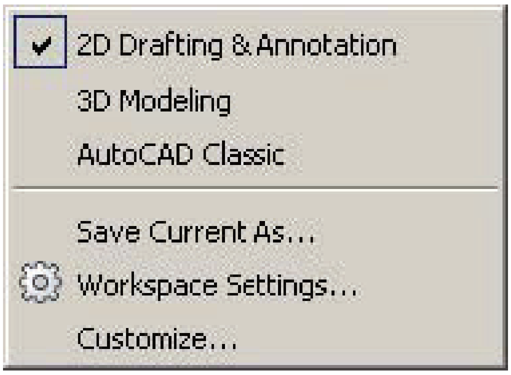
Workspaces: You can switch between the workspaces from the menu browser.

1 Click the workspace switching icon in the lower left corner of the screen. (Fig 7)

Fig 7

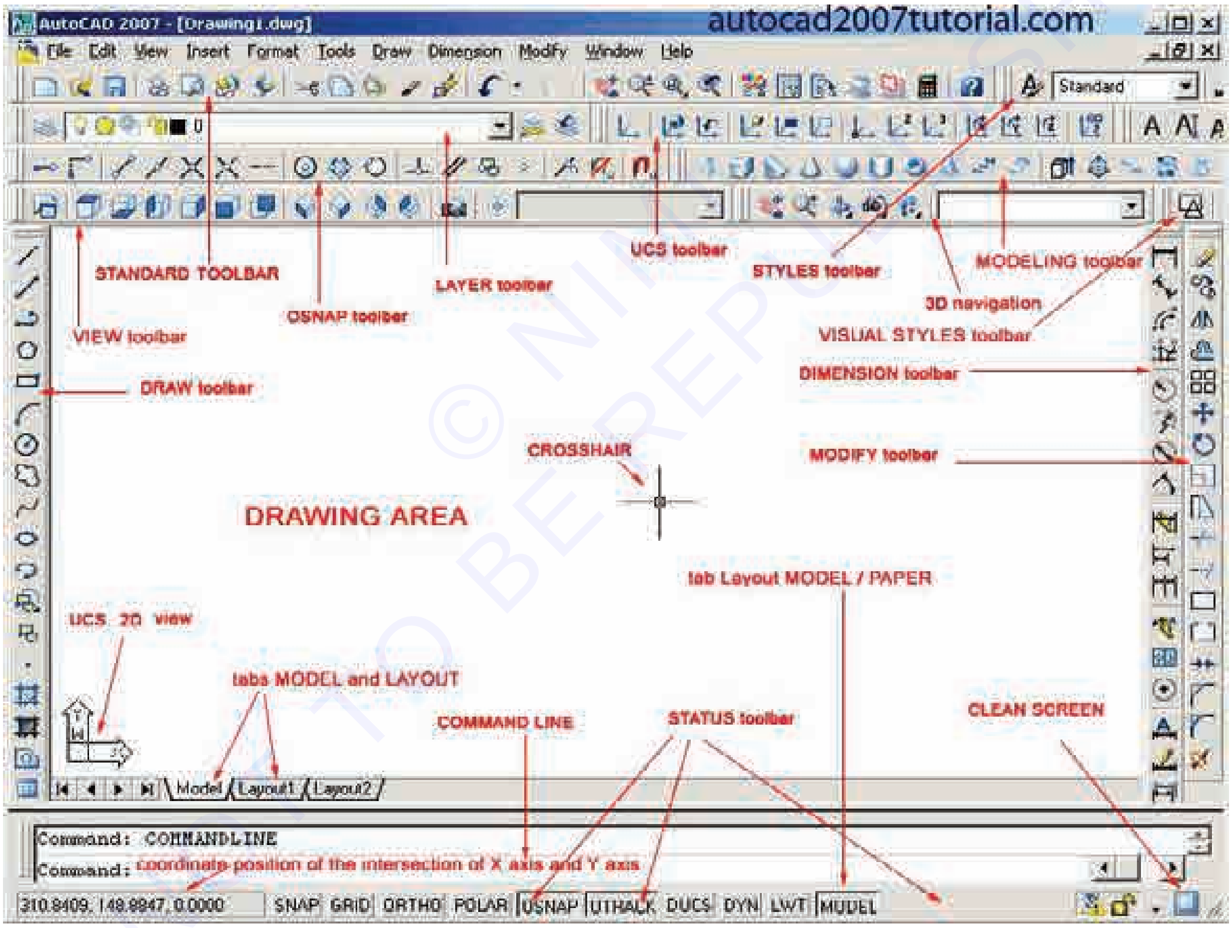
2 Click on one of the following workspace options. (Fig 8)

Fig 8



Auto CAD classic workspace (Fig 9)

Fig 9



Title bar: This shows the name of drawing which is currently used.

Menu bar: This menu bar help us quicker way to access the general controls and setting for AutoCAD. The main commands and functions are available in this menu bar it has the following facilities.

1 It gives a command that requires key board or drawing input.

2 It displays additional menus choice with > symbol, in this menu called cascading menus.

3 It displays a dialogue box that contains settings which have changing options.

Standard tool bar: This tool bar contains the standard functions of commands which is used for getting information's and modifications.

Properties tool bar: This tool bar have the properties of the entity such as thickness of line, colour, layer type of line etc. We can change the properties of the entity by using this tool bar.

Draw tool bar: This tool bar contains the group of drawing commands such as line, arc, circle etc.

Modify tool bar: This tool bars are used to do the modifications in the entities such as erase, trim etc.

Draw area: This is a black space to draw the drawings. This area has formed as grids, we can increase or decrease the area by using boundary limit command.

UCS: UCS (user coordinate system is an indication to the use of for which plane the drawing is drawn. We can change any plane according to our wish to draw the drawing in views.

Command prompt window: This window is used to give commands by typing in key board.

Cross hair: This is the pointer used to draw, select and to locate.

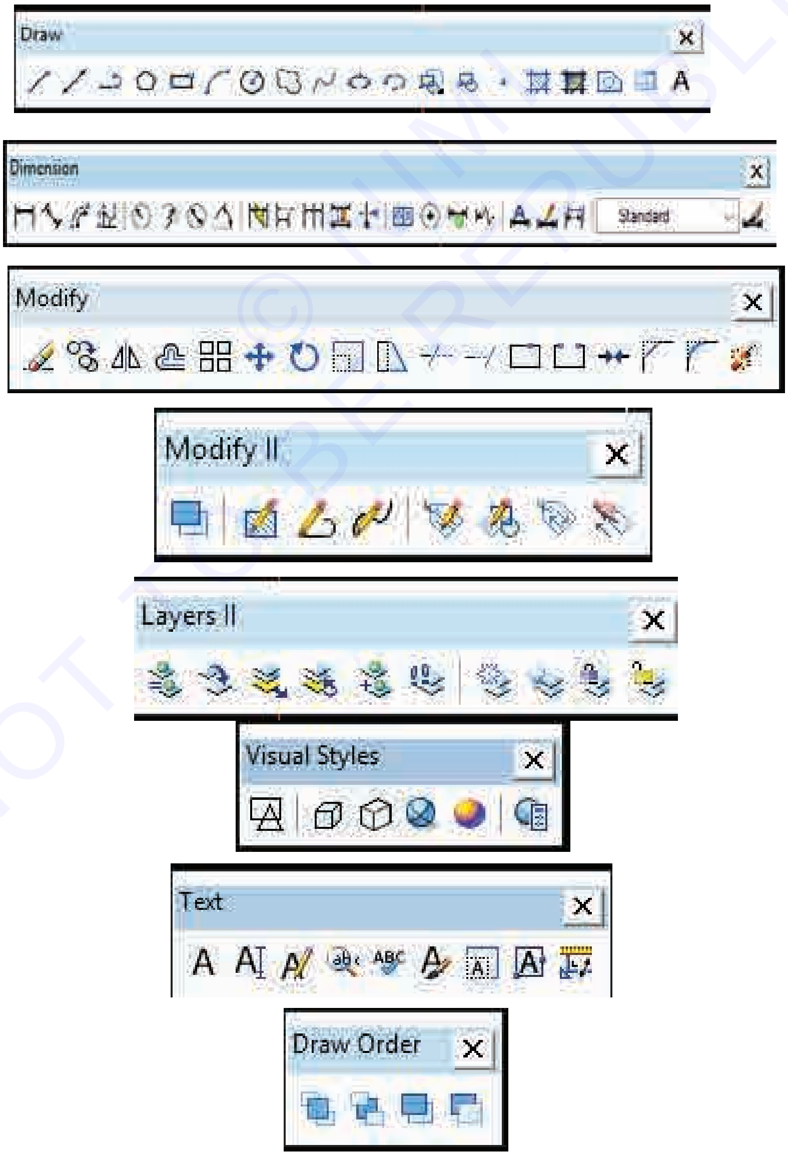
Layout tabs: These tabs are used to select the particular lay out of the drawing.

Function tabs: Below the command prompt window drawing function tabs are available. These tabs show us the position of grid, other, o snap etc. The functional keys are used for effective function of the drawing.

Key board function keys

There are some function keys in the keyboard for quick access to certain commands. These keys are pressed for the following purposes.

Fig 10



Key	Function defined
F1	Online help.
F2	Toggles between command window on and off.
F3	Toggles between OSNAP on and off.
F4	Toggles between Tablet on and off.
F5	Switches among isoplanes top, right and left.
F6	Toggles between coordinates on and off.
F7	Toggles between grid on and off.
F8	Toggles between ortho mode on and off.
F9	Toggles between snap mode on and off.
F10	Toggles between polar tracking on and off.
F11	Toggles between objects snap tracking on and off.
F12	Save as.

You can disable the group selection quickly by pressing FUNCTION KEYS (Ctrl + Key) combination to quickly toggle some of the modes and invoke some of the commands.

Function keys used in Auto CAD

Key strokes	Function defined
Ctrl+Z	Undo
Ctrl+C	Copy clip
Ctrl+E	Osoplane top/right /left
Ctrl+G	Grid on/off
Ctrl+L	Ortho on/off
Ctrl+O	OPEN command
Ctrl+S	QSAVE command
Ctrl+U	Polar tracking on/off
Ctrl+W	Object snap tracking on/off
Ctrl+Y	Redo
Ctrl+2	ADCENTER command
Ctrl+F6	Switch between open drawings
Ctrl+B	Snap on/off
Ctrl+D	Coordinate display on/off
Ctrl+F	Osnap setting dialog box
Ctrl+K	HYPERLINK command
Ctrl+N	NEW command
Ctrl+P	PRINT command

Ctrl+T	Tablet on/off
Ctrl+V	Paste
Ctrl+X	Delete
Ctrl+1	Object properties window on/off
Ctrl+6	DBCONNECT command
Ctrl + Tab	Switch between open Drawings.

The functionality of these Ctrl + Key combination depends on the settings done on the User Preferences tab on the option dialog box.

Example:

That is to say Ctrl+ C works for COPYCLIP command, if the check box is cleared, Ctrl + C works for the CANCEL command.

Text edit keys

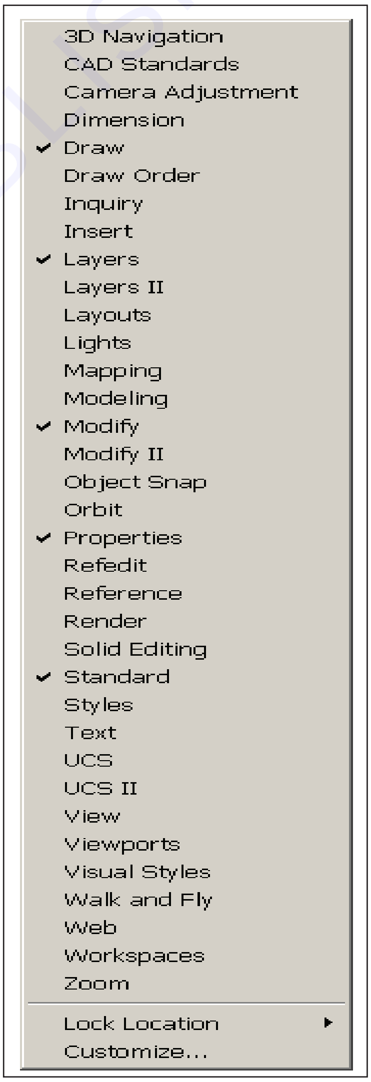
The following accelerator keys, which are effective within the Multilane Text Editor, dialog box.

Key strokes	Function defined
Ctrl+A	Select all text in the multiline text editor.
Ctrl+B	Applies or removes bold format for selected text.
Ctrl+C	Copies selected text to the clip board.
Ctrl+1	Applies or removes italic format for selected text.
Ctrl+Shift+L	Converts selected text to lower case.
Ctrl+Shift+U	Converts selected text to upper case.
Ctrl+U	Applies or removes underline format for selected text
Ctrl+V	Pastes Clipboard contents to cursor location
Ctrl+X	Cuts selected text to the clipboard
Ctrl+SPACE	Removes character formatting in slected text.

Loading toolbars.

Right- clicking on an icon in any toolbar.

This will show a list of all available toolbars.



Introduction of CAD

Basic commands - I

Introduction

AutoCAD allows you to have across to a large number of commands, a general rule is that you use 20% of the commands 80% of the time. I will start by introducing you to the most common drawing commands. When you combine these with the basic modify commands, you will be able to make elaborate drawings quite quickly. In other words, most of the commands you will use while using Auto cad are taught in level1.

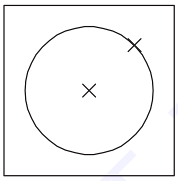
The important thing to remember is that AutoCAD will expect you give it information in a very particular order. The most frustrating thing when you begin using this program is that you will try to do something, but auto cad will not work. In most cases, it means that you are trying to input information at the wrong time. This is why it is very important to be in the habit of looking at the command line.

Circles

Circle command (Fig 1,2, & 3)

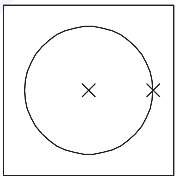
1	Choose	Draw, circle.
		(or)
2	Click	The Circle icon.
		(or)
3	Type	Circle at the command prompt.
4	Type	One of the following options: 3P/2P/TTR/<<center point>>:
		(or)
5	Pick	A center point
6	Type	A radius or diameter.
		(or)
7	Pick	A radius or diameter Diameter/<<radius>>:

Fig 1



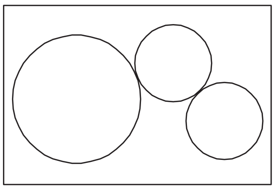
CIRCLE COMMAND

Fig 2



CIRCLE COMMAND

Fig 3



CIRCLE COMMAND

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TIPS

- To create circles that are the same size, press ENTER when asked for the circle radius.
- When selecting a circle with a pick box, be sure to select the **circumference** of the circle.

The command line tells you what information AutoCAD requires to continue.

Your first drawing alignment will be to use the drawing commands in conjunction with the co-ordinate system it is very important to understand how to give the program accurate information. You will use the following commands.

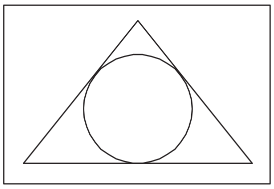
Drawing arcs and circles

CADD provides many ways to draw arcs and circles. There are a number of advanced techniques available for drawing arcs and circles, which can simplify many geometrical drawing problems. You can draw an arc by specifying circumference and radius, radius and rotation angle, chord length and radius, etc.

Arc command (Fig 4)

1	Choose	Draw, arc. (or)
2	Click	The Arc icon. (or)
3	Type	Arc at the command prompt command: ARC
4	Draw	One of the arcs.

Fig 4

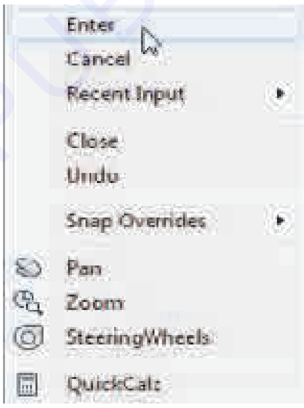


CIRCLE COMMAND

TIPS

- Except for 3 point arcs, arcs are drawn in a **counter clock wise direction.**
- While in the arc command, press the right mouse button to select the following options for arcs: (Fig 5)

Fig 5



Arc examples (Fig 6)

3 point arc

Start, centre, chord length

Start, centre, end

Start, end, radius

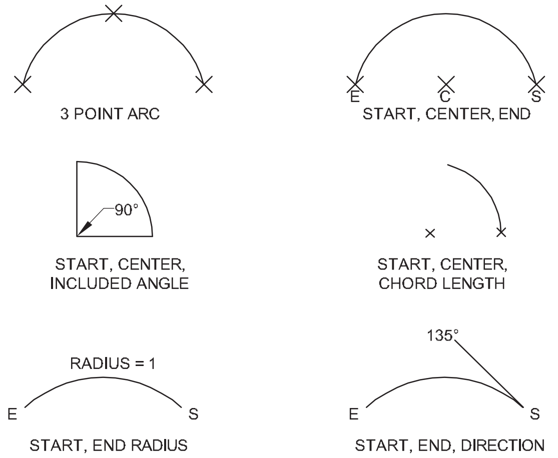
Start, centre, included angle

Start, end direction

Drawing ellipses and elliptical arcs

Ellipses are much easier to draw with CADD than on a drawing board. On a drawing board, you need to find the right size template or draw a series of arcs individually to draw an ellipse. With CADD, all you need to do is specify the size of the ellipse.

Fig 6



ARC COMMAND

The following are two basic methods for drawing ellipses:

- Length and width.
- Axis and rotation angle.

Ellipse.

Creates an ellipse or an elliptical arc:

1 **Choose** Draw, Ellipse.

(or)

2 **Choose** The ellipse or partial ellipse icon

(or)

3 **Type** Ellipse at the command prompt
Command: ellipse

4 **Type** One of the following options:

Arc/Center/Isocircle/<Axis endpoint1>:

Ellipse options

Axis endpoint 1: Defines the first axis by two specified endpoints. The angle of the first axis determines the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

Ellipses options

Axis end point 2: <Other axis distance>/Rotation: Specify a point or enter a distance

Arc: Creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis of the elliptical arc.

Center : Creates the ellipse by a specified center point.

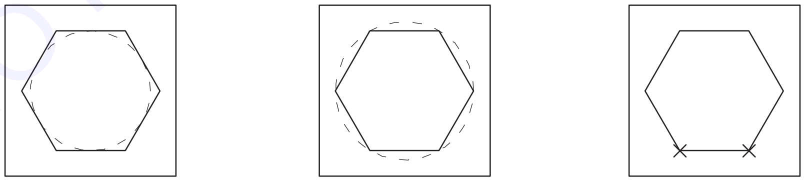
Isocircle : Creates an isometric circle in the current isometric drawing plane.

Rotation : The major axis is now treated as the diameter of a circle that will be rotated a specified amount around the axis. You enter an angle between 0 and 89.4 degrees.

Polygon (Fig 7)

1	Choose	Draw, ploygon.
		(or)
2	Click	The polygon icon.
		(or)
3	Type	Polygon at the command prompt.
4	Type	The number of sides for the polygon (3-1024).
5	Pick	The center of the polygon.
		Edge/<Center of polygon>: pick.
		(or)
6	Type	E to define the polygon by two edges.
		(or)
7	Type	I or C to place the polygon in side or outside of an imaginary (It should not displayed on the screen) circle. Inscribed in circle/ circumscribed about circle (I/C)

Fig 7



POLYGON INSCRIBED IN AN IMAGINARY CIRCLE

POLYGON CIRCUMCRIBED AROUND AN IMAGINARY CIRCLE

POLYGON DRAWN WITH EDGE

POLYGON COMMAND

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Basic commands - II

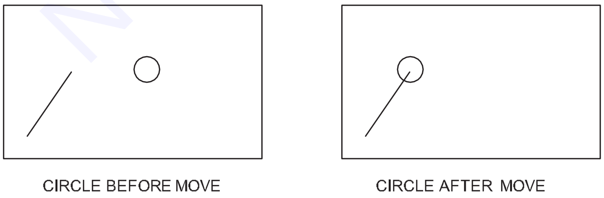
The previous lesson dealt with drawing commands. This lesson will introduce some common **modifying commands.** In AutoCAD, you may actually use modifying commands more often than drawing commands. Now that you know the basics, here's some more commands to add to your collection. Three commands, Trim, Extend and Offset are used often in 2D AutoCAD work.

Command	Keystroke/Short cut	Location	Result
Rectangle	RECTANGLE/REC	Home>Draw>Rectangle	Draws a rectangle after you enter one corner and then the second.
Trim	TRIM/TR	Home>Modify>Trim	Trims objects to a selected cutting edge.
Extend	EXTEND/EX	Home>Modify>Extend	Extends objects to a selected boundary edge.
Offset	OFFSET/O	Home>Modify>Offset	Offsets an object (parallel) by a set distance.
Object snaps	OSNAP/OS/F3	Tools>Object Snap Settings	Brings up the OSNAP dialog box.
Move	Move/M	Home>Modify>Move	Moves an object or objects.
Copy	Copy/CP	Home>Modify>Copy	Copies object(s) once or multiple times.
Stretch	Stretch/S	Home>Modify>Stretch	Stretches an object after you have selected a portion of it.
Mirror	Mirror/MI	Home>Modify>Mirror	Creates a mirror image of an object or selection set.
Rotate	Rotate/RO	Home>Modify>Rotate	Rotates objects to a certain angle.
Fillet	Fillet/F	Home>Modify>Fillet	Creates a round corner between two lines.
Chamfer	Chamfer/CHA	Home>Modify>Chamfer	Creates an angled corner between two lines

Move command (Figs 1 & 2)

1	Choose	Modify, Move. Or ✥
2	Click	The Move icon or
3	Type	Move at the command prompt command: MOVE or M
4	Pick	Objects to move select objects (select)
5	Pick	A point to move from base point of displacement: (Pick point)
6	Pick	A point to move to second point of displacement: (pick point)

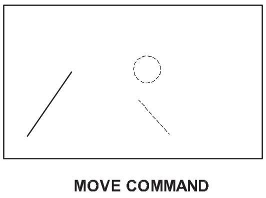
Fig 1


CIRCLE BEFORE MOVE

CIRCLE AFTER MOVE

MOVE COMMAND

Fig 2


MOVE COMMAND

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TIP

To move an object a specified distance, type a distance at the second point of displacement prompt:@1<0

Moving drawing objects

CADD allows you to move drawing objects within a drawing in a convenient manner. Unlike on a drawing board, you don't need to first erase and then redraw in a new place. You can simply rearrange the existing drawing objects, as you like. This is a very useful tool for analyzing design alternatives and making quick adjustments to drawings.

Previous selection

Places selected objects in the previous selection set

1 Choose Modify, move.

(or)

2 Click The move icon.

(or

3 Type Move at the command prompt.

Command: Move or M

4 Pick Objects to move.

Select objects : (P)

Previous selection set highlighted

TIP

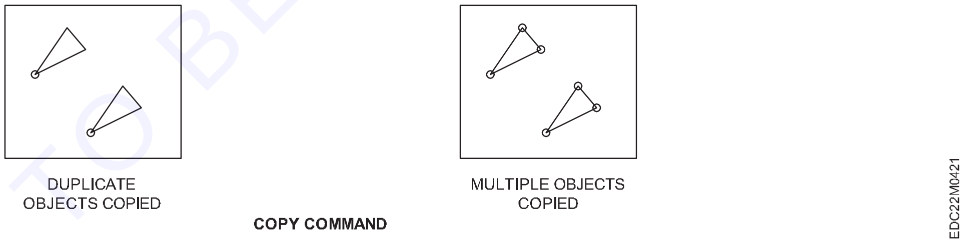
AutoCAD requires that objects be selected in order to be processed. The Select Objects prompt occurs after many commands, including the SELECT command itself.

Copying drawing objects (Fig 3)

CADD allows you to make quick and easy copies of existing drawing objects. You can copy individual drawing objects or the entire drawing all at once. You can even make multiple copies of drawing objects within seconds.

Using the copy function is quite similar to the way the move function is used. First, you need to select objects using any of the methods described earlier. Then you need to indicate a base point and a relocation (or destination) point. The copied objects are placed according to the relocation point.

Fig 3



COPY COMMAND

Making multiple copies in a rectangular fashion

There are separate functions available in CADD that allow you to make multiple copies in a linear or rectangular fashion (commonly known as a rectangular array). You can make hundreds of copies within seconds. You don't need to enter a base point and a destination point. You just need to select the objects, specify how many rows and columns you need and the distance between them.

Copy command

1 Choose Modify, copy.

(or)

2 Click The copy icon

(or)

3 Type Copy at the command prompt.

4 Pick Objects to copy.

Select objects: (select)

5 Pick A point to move from.

Base point or displacement /multiple: (pick point).

6 Pick A point to copy to.

Second point of displacement: (Pick point)

(or)

7 Type A point to copy to.

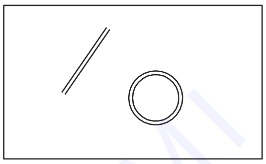
Second point of displacement: @1<0

TIP

To copy many objects in the same copy command, type M for Multiple at the “Base point or displacement/Multiple” option.

Offset command (Fig 4)

Fig 4



OFFSET COMMAND

EDC22M0422

Offset distance

To offset a specified distance:

1 Choose Modify, Offset.

(or)

2 Choose The offset icon.

(or)

3 Type OFFSET at the command prompt.Command: OFFSET or O

4 Type The distance to offset. Offset distance or <Through point>: (number)

5 Pick The object to offset. Select object to offset: (select object)

6 Pick A side to offset object to. Side to off set: (pick side)

7 Pick Another object to offset

Select object to offset: (Pick side)

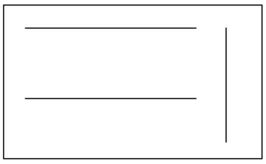
(or)

8 Press Enter to end the command.

Offsetting objects by specifying a distance

Offset through point (Fig 5)

Fig 5



OFFSET COMMAND

EDC22M0423

To offset through point

1 Type Offset at the command prompt

Command: Offset

2 Type T to specify a through point

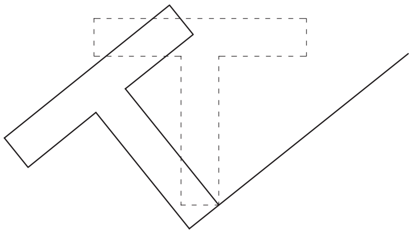
Offset distance or <Through point>: (T)

3 Pick A point to offset through (HINT: useobject snaps) Select object to offset: (pick) through point: (select object)

Offset through a point

Rotate (Fig 6)

Fig 6



ROTATE COMMAND

EDC22M0424

1 Choose Modify, rotate

(or)

2 Click The modify icon.

(or) ↻

3 Type Rotate at the command prompt

Command: Rotate

4 Pick Objects to rotate:

Select objects: (select)

5 Pick A pivot point to rotate around

Base point: (point)

6 Type A rotation angle<Rotation angle>/ Reference: (number)

(or)

7 Pick A rotation angle<Rotation angle>/ Reference: (point)

Rotating the drawings

CADD drawing allows you to rotate selected drawing objects to a specified angle. To rotate, you need to select the drawing objects, enter a reference point **(or base point)** and the **rotation angle.** The base point acts as **a pivot** point around which the objects are rotated. The rotation angle determines by how much the objects will be rotated and in which direction.

Reference angle rotation

A positive angle causes **counterclockwise rotation**, and a negative angle produces **clockwise rotation**. If you respond to the last prompt with r, you can specify the current rotation and the new rotation you want. AutoCAD prompts

1 Type R for a rotation angle<Rotation angle>/Reference: (R)

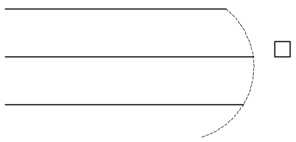
2 Choose An existing rotation angle Rotation angle: (number or points)

3 Choose A new rotation angle New angle:

(number or points)

TIP (Fig 7)

Fig 7



TRIM COMMAND

You can show AutoCAD the reference angle (by pointing to the two endpoints of a line to be rotated), and then specify the new angle. You can specify the new angle by pointing or by dragging the object.

Trim

The trim command allows you to trim objects in a drawing so they end **precisely at a cutting edge** defined by one or more other objects in the drawing.

1 Choose Modify, trim

(or)

2 Click The trim icon.

(or)

3 Type Trim at the command prompt

Command: trim

Select cutting edge(s)

4 Pick The cutting edge to extend to

Select objects: (select)

5 Press Enter to accept the cutting edge

Select objects: (press enter)

6 Pick Objects to trim

<Select object to trim>/Project/Edge/Undo:

Select an object, enter an option, press enter

7 Press ENTER when you are done choosing objects

Select object to trim/Undo: (press enter)

TIP: Hold the shift key to interactively extend instead of trim.

Cutting drawing objects along an edge

CADD allows you to erase drawing objects along a selected edge (this technique is often called trimming).When you use this function, you are prompted to select the drawing object that is to be used as the cutting edge and then select the objects that are to be erased along that edge.

Making sharp and rounded corners

CADD allows you to make fine corners of any two lines or arcs. This technique, often called filleting, is the quickest way to join the missing corners of lines and arcs. With this function active, to make a corner all you need to do is select the lines or arcs that have missing corners. CAD automatically extends or shortens the selected objects to form a corner. You can also specify whether you want a sharp corner or a rounded corner.

Fillet (Fig 8)

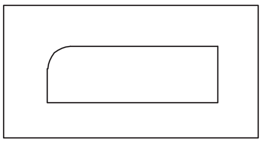
1 Choose Modify, fillet.

(or)

2 Click The fillet icon.

(or)

Fig 8



FILLET COMMAND

3	Type	FILLET at the command prompt. Command: FILLET
4	Pick	First object to fillet. Polyline/radius/trim<Select two objects>: Select first object.
5	Pick	Second object to fillet. Select second object: select second object. (or)
6	Type	One of the following options:
	P	Fillets an entire Polyline
	R	Sets the fillet radius.
	T	Sets the trim mode (trim cuts the fillet corner and no trim keeps the fillet corner).

TIP

You can also fillet PARALLEL lines as well as PLINES with LINES

Type a radius of Zero (0) to create a clean 90 degree corner.

Chamfer

1	Choose	Modify, chamfer. (or)
2	Click	The chamfer icon. (or)
3	Type	CHAMFER at the command prompt. Command: Chamfer
4	Pick	First object to chamfer. Polyline/ distance/angle/trim/ method<Select first line>: select first object.
5	Pick	Second object to chamfer. Select second object: select second object. (or)
6	Type	One of the following options:
	P	Chamfers entire Polyline.
	D	Sets chamfer distances.
	A	Uses a distance and angle method instead of two distances.
	T	Sets the trim mode
	M	Sets the method to distance or angle.

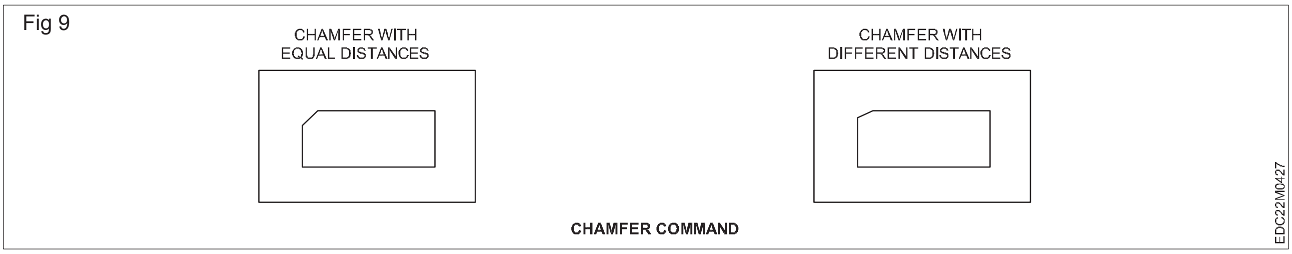
Chamfer with equal distances (Fig 9)

Chamfer with different distances

Making chamfered corners

CADD allows you to make a chamfered corner between two lines. It works quite like the fillet command. When you enter the chamfer command, you are prompted to select the lines that are to be chamfered and enter a chamfer distance. The chamfer distance determines the size of the chamfer.

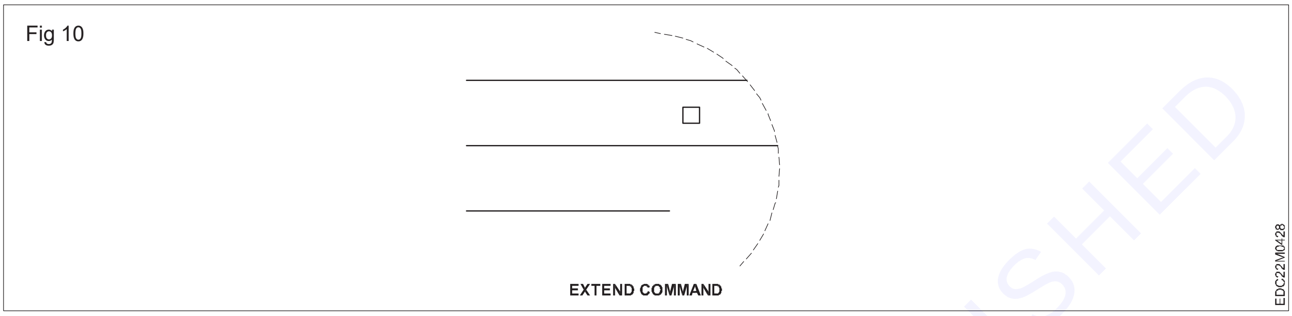
Fig 9



CHAMFER COMMAND

Extend (Fig 10)

Fig 10



EXTEND COMMAND

1 Choose Modify, extend,

(or)

2 Click The extend icon.

(or)

3 Type EXTEND at the command prompt command: EXTEND

Select boundary edge (s)...

4 Pick The BOUNDARY edge to extend to select objects: (select)

5 Press ENTR to accept the boundary edge select objects: (press enter)

6 Pick The object to extend

<Select object to extend> / Project/ edge/ undo: Select an object, enter an option, or press enter: (select)

7 Press ENTER when you are done choosing objects.

Lines extended to an arc (Arc is boundary edge)

TIP

- Use the object selection option FENCE to choose multiple objects.

Extending drawing objects to an edge

CADD allows you to extend lines to a selected drawing object. Often you need to extend lines to construct a drawing and to fix any graphical errors. **To extend lines, you need to select an edge to which the lines should extend and then select the lines to be extended.**

Dividing an object into equal parts

CADD allows you place dividing marks on a drawing object such as a line, arc, ellipse or spline. To use this command, you need to select an object and specify how many divisions are required. This function places markers at equal distances on the drawing object.

Break (Figs 11 & 12)

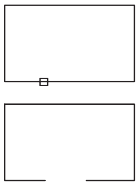
1 Choose Modify, break.

(or)

2 Click The break icon.

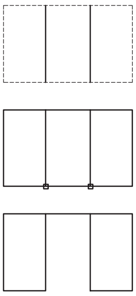
(or)

Fig 11



BREAK COMMAND

Fig 12



BREAK COMMAND

3 Type BREAK at the command prompt. Command: BREAK

4 Pick Object to break.

Select object: (select one object)

5 Pick A second break point. Enter second point: (point)

6 Type F to choose a different break point Enter second point (or F for first point): (F)

7 Pick The first break point on the object

Enter first point: (point)

8 Pick A second break point

TIP

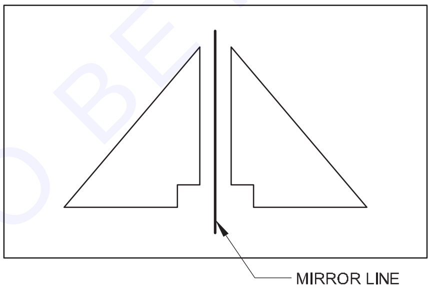
You can also type coordinates instead of picking a break point. Enter second point (or F for first point): @3’<0

If you break a circle, it changes to an arc by deleting the portion from the first point to the second, going counterclockwise.

Breaking a polyline with nonzero width will cause the ends to be cut square.

Mirror command (Fig 13)

Fig 13



MIRROR COMMAND

1 Choose Modify, mirror.

(or)

2 Click The mirror icon

(or)

3 Type MIRROR at the command prompt.

Command: MIRROR

4 Pick Objects to mirror. Select objects: (Select)

5 Pick First point of mirror line: (point)

6 Pick Second point of mirror line: (Point)

7 Type Yes to delete the original objects and No to keep them.

Delete old objects? Y or N

Mirroring drawings

CADD allows you to create mirror images of drawings. This capability is very useful when you want to draw something that is **symmetrical on both sides**. You need to draw only one half of the drawing; the rest of the drawing can be completed using the mirror function. To make a mirror image, you need to select the objects to be mirrored and indicate a mirror axis. The mirror axis is an imaginary line along which the diagram is mirrored.

Array

Rectangular array

To draw rectangular array

1 Choose Modify, array.

(or)

2 Click The array icon.

Or

3 Type ARRAY at the command prompt. Command: ARRAY objects to array. Select

4 Pick Objects to array. Select Objects:(select)

5 Type The number of rows top to bottom. Number of rows (----) <1>: (number)

6 Type The number of columns left to right. Number of columns (III) <1>: (number)

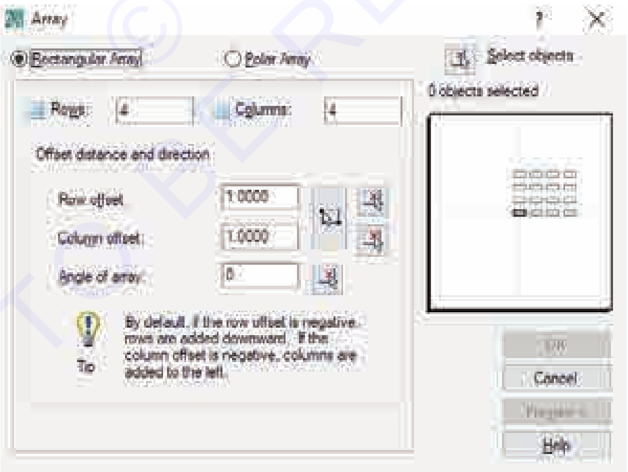
7 Type The unit cell distance between items in each row. Distance between rows: (+number=up, number = down)

8 Type The unit cell distance between items in each column.

Distance between columns: (+ number = right, - number = left)

Creating an array of objects (Fig 14)

Fig 14



Array

● Rectangular Array ○ Polar Array

Select objects

0 objects selected

Rows: 4 Columns: 4

Offset distance and direction

Row offset: 1.0000

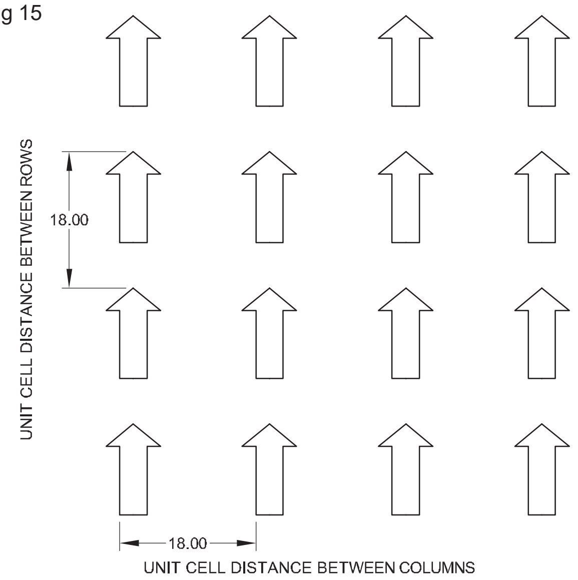
Column offset: 1.0000

Angle of array: 0

Tip: By default, if the row offset is negative, rows are added downward. If the column offset is negative, columns are added to the left.

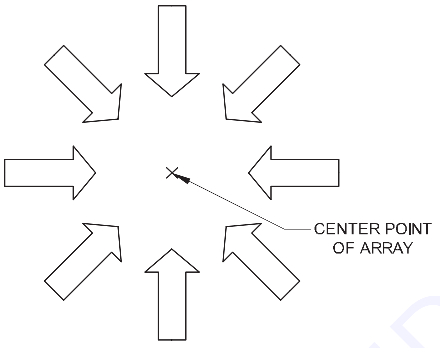
The array command in AutoCAD is used to make multiple copies of objects. Although you can use the copy command to duplicate objects, the array command is more flexible and precise. One advantage of using the array command is that it allows you to copy objects in a defined angle and exact number of copies. Therefore, you can create array in various pattern. For example, you can show multiple objects in a row, column, or irregular pattern such as a spiral. Let’s look at a few examples below: (Fig 15 & 16)

Fig 15



ARRAY COMMAND

Fig 16



ARRAY COMMAND

EDC22M0431

Polar array (Fig 16)

To draw a polar array: (Fig 17)

1 Choose Modify, array.

(or)

2 Click The array icon.

(or)

3 Type Array at the command prompt. Command: Array

4 Pick Objects to array. Select

Objects: (select)

5 Type P to draw a polar array. Rectangular or Polar array (R/P):P

6 Pick A center point for the array. Center point of array. Pick point

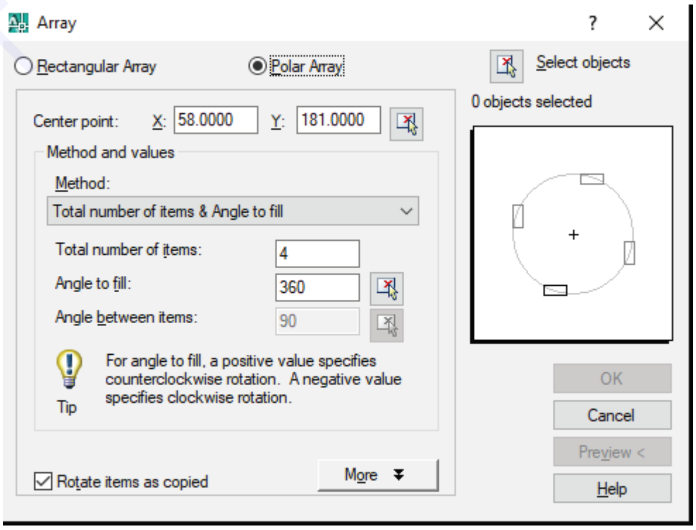
7 Type The total number of items in the array.Number of items: number

8 Type The number of degrees to rotate the objects. Degrees to fill (+=CCW, -+CW) <360>:

Number

9 Type Yes No to rotate objects. Rotate objects as they are copied? <y> Y or N

Fig 17



Array

○ Rectangular Array ● Polar Array

Select objects

0 objects selected

Center point: X: 58.0000 Y: 181.0000

Method and values

Method: Total number of items & Angle to fill

Total number of items: 4

Angle to fill: 360

Angle between items: 90

Tip: For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.

☑ Rotate items as copied

Stretch

1 Choose Modify, stretch

(or)

2 Click The stretch icon.

3 Type STRETCH at the command prompt.Command: STRETCH select objects to stretch by window.

4 Type C to choose CROSSING window Select objects: C

5 Pick A first corner to stretch. First corner: (point)

6 Pick The opposite corner to window the objects to stretch.Other corner: (point)

7 Press ENTER to accept objects to stretch

8 Pick A base point to stretch from Base point: (point)

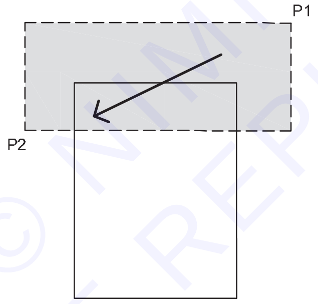
Stretching diagrams (Fig 18)

CADD allows you to **quickly change** the size of diagrams by stretching lines, arcs, splines, etc. This function is very helpful to make **quick alterations** to drawings. To use the stretch function, you need to select the drawing objects to be stretched and specify the distance and direction of stretching.

9 Pick A point to stretch to New point: (point)

10 Type A distance to stretch. New point: @ 1<0

Fig 18



STRETCH COMMAND

EDC22M0432

TIP

The Stretch command must use a CROSSING window or a CROSSING POLYGON window.

Lengthen (Fig 19)

1 Choose Modify, lengthen.

(or)

2 Type Lengthen at the command prompt.Command:_ lengthen (Fig 19)

Select an object or [DElta/Percent/Total/Enter delta length or [Angle]<0.0000>:2

Select an object to change or [Undo]: pick object

Object before lengthen

Object after lengthen

Fig 19

OBJECT BEFORE LENGTHEN

OBJECT AFTER LENGTHEN

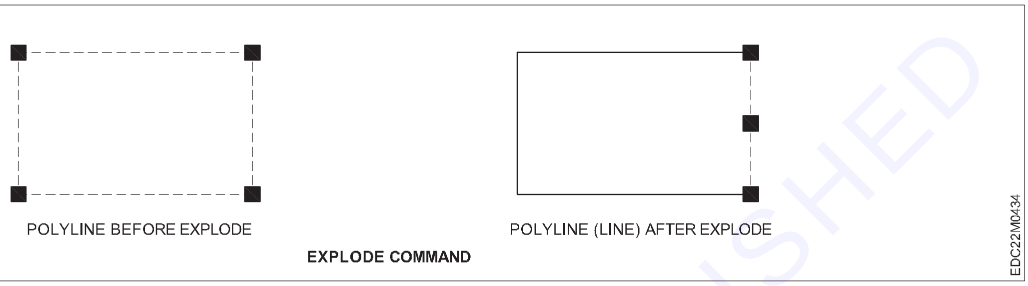
LENGTHEN COMMAND

EDC22M0433

Explode command (Fig 20)

1 Choose Modify, explode.
(or)
2 Pick The explode icon.
3 Type EXPLODE at the command prompt.
Command: EXPLODE (Fig 20)
(or)
4 Pick The object to explode. Select objects: (pick)

Fig 20



OOPS commands

Reinserts the last erased set of objects or block even if it was not the last command issued. Otherwise oops acts like UNDO.

1 Type OOPS at the command prompt to reinsert erased objects
Command: OOPS

Other CAD commands

Introduction

Drawing multiple parallel lines

CADD allows you draw parallel lines simultaneously just by indicating a starting point and an end point. These lines can be used to draw something with heavy lines or double lines. For example, they can be used to draw the walls of a building plan, roads of a site map, or for any other presentation that requires parallel lines.

Most programs allow you to define a style for multiple parallel lines. You can specify how many parallel lines you need, at what distance and if they are to be filled with a pattern or solid fill.

A number of add-on programs use multiple lines to represent specific drawing features. For example, an architectural program has a special function called “wall”. When you use this option, it automatically draws parallel lines representing walls or specified style and thickness.

Drawing flexible curves

CADD allows you to draw flexible curves (often called splines) that can be used to draw almost any shape. They can be used to create the smooth curves of a sculpture, contours of a landscape plan or roads and boundaries of a map.

To draw a flexible curve, you need to indicate the points through which the curve will pass. A uniform curve is drawn passing through the indicated points. The sharpness of the curves, the roughness of the lines and the thickness can be controlled through the use of related commands.

Adding hatch patterns to drawings

The look of CAD drawings can be enhanced with the hatch patterns available in CAD. The patterns can be used to emphasize portions of the drawing and to represent various materials, finishes, and spaces. Several ready-made patterns are available in CAD that can be instantly added to drawings.Hatch patterns are quite easy to draw. You don't need to draw each element of a pattern one by one. You just need to specify an area where the pattern is to be drawn by selecting all the drawing objects that surround the area. The selected objects must enclose the area completely, like a **closed polygon**. When the area is enclosed, a list of available patterns is displayed. Select a pattern, and the specified area is filled.

Point command (Fig 1)

1 Choose Draw, point, single or multiple point

(or)

2 Click The point icon

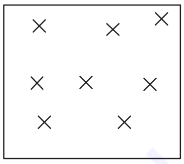
(or)

3 Type Point at the command prompt

Command: Point

4 Pick A point on the drawing

Fig 1



POINT COMMAND

Point (point)

Point styles (Fig 2)

Changes the appearance of points and point sizes.

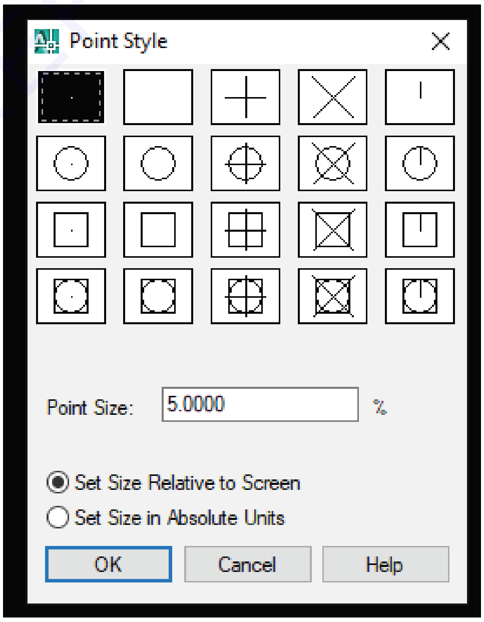
1 Choose Format, Point Style...

(or)

2 Type DDP type at the command prompt.

Command: DDP type

Fig 2



Point Size: 5.0000 %

● Set Size Relative to Screen

○ Set Size in Absolute Units

Rectangle (Fig 3)

1 Choose Draw, rectangle.

(or)

2 Click The rectangle icon

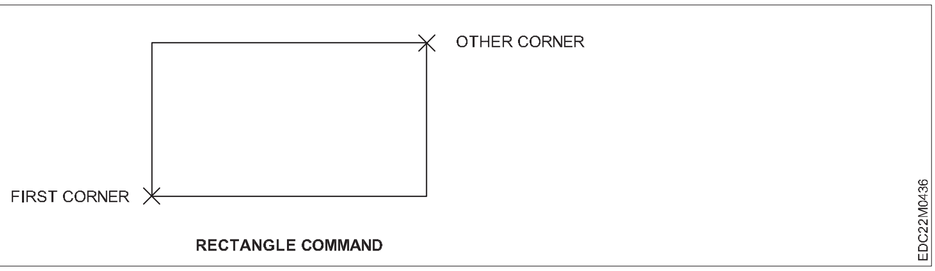
(or)

3 Type Rectangle at the command prompt Command: RECTANG chamfer /Elevation/Fillet/Thickness/Width/<First corner>

4 Pick first corner

5 Pick other corner or type coordinates (i.e. @4,2)

Fig 3



RECTANGLE COMMAND

Pline command (Fig 4)

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.

1 Choose Draw, polyline.

(or)

2 Pick The pline icon.

3 Type PLINE at the command prompt Command: PLINE or PL

4 Pick A point on the drawing to start the polyline

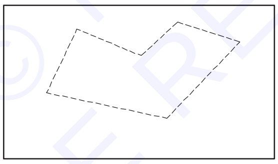
Form point: (select)

5 Type One of the following options Arc/Close/ Half width/Length/Undo/Width/<endpoint of line>:

(or)

6 Pick A point to continue drawing Arc/Close/ Half width/Length/Undo/Width/<endpoint of line>: (pick point)

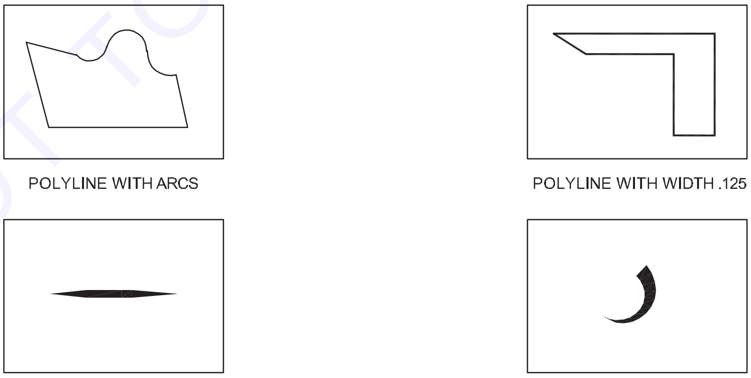
Fig 4



PLINE

PLINE options (Fig 5)

Fig 5



POLYLINE WITH ARCS

POLYLINE WITH WIDTH .125

TAPERED WIDTH POLYLINE

TAPERED WIDTH ARC POLYLINE

PLINE

Arc : Toggles to arc mode and you receive the following: Angle/CEnter/CLose/Direction/Half width/Line/Radius/ Second Pt/Undo/Width/<enter of arc>:

Close : Closes a polyline as it does in the line command.

Halfwidth : Specifies the half width of the next polyline segments. Can be tapered.

Length : Specifies the length to be added to the polyline in the current direction.

Undo : Undoes the previous spline segment as with the line command.

Width : Specifies the width of the next polyline segments. Can be tapered.

Polyline with arcs

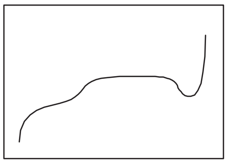
Polyline with width 125

Tapered width polyline

Tapered width arc polyline

Convert pline to spline (Fig 6)

Fig 6



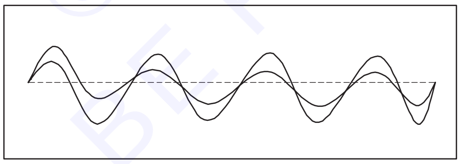
PLINE TO SPLINE

1 Draw A Pline
2 Type PEDIT to edit the polyline as a spline.
3 Choose Draw, Spline
4 Type Object at the command prompt.
5 Click Once on the polyline to turn it into a spline.

Spline (Fig 7)

The spline command creates a particular type of spline known as a non uniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points.

Fig 7



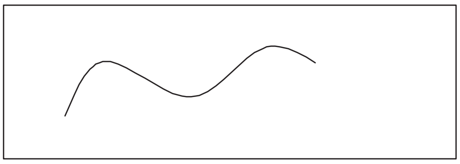
SPLINE

1 Choose Draws, spline.
(or)
2 Click The spline icon
(or)
3 Type Spline at the command prompt Command: Spline
4 Pick A start point for the spline Object /<Enter first point> (pick point)
5 Pick Points until you are done drawing splines Enter point: (pick points)
6 Press Enter or close to complete the spline
7 Pick Starting tangent point for the spline Enter start tangent (pick point)
8 Pick Ending tangent point for the spline
Enter end tangent (pick point)

Spline options (Fig 8)

Object: Converse 2D or 3D spline-fit polylines to equivalent splines

Fig 8



SPLINE

Points: Points that defines the spline

Close: Closes a spline

Fit Tolerance: Allows you to set a tolerance value that creates a smooth spline.

TIP: Refer to AutoCAD online help topic for more information on spline options.

Editing splines

1. Choose Modify, Object, Spline.

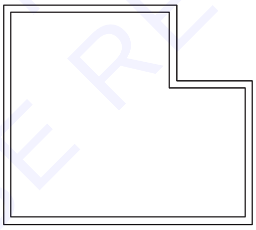
TIP

Drawings containing splines use less memory and disk space than those containing spline-fit polylines of similar shape.

Multilines (Fig 9)

MLINE Command

Fig 9



MULTILINES

1	Choose	Draw, multiline. (or)
2	Type	MLINE at the command prompt Command: MLINE
3	Pick	A point to start the multiline. Justification/Scale/Style/<From point>:pick point
4	Pick	A second point to continue the multiline. <To point>: Pick point
5	Pick	The next point to continue drawing Multilines. Undo/<To point>: pick point
6	Press	ENTER to end the multiline Close/Undo/<To point>: press enter (or)
7	Type	C to close the multiline back to the first point. Close/Undo/<To point:C

Multiline styles (Figs 10 & 11)

1 Choose Format, multiline style.

2 Type Mlstyle at the command prompt.

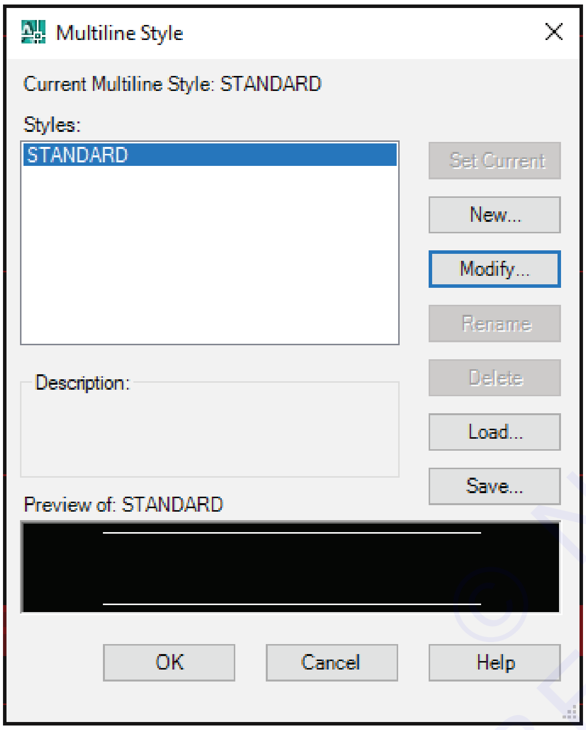
Command: Mlstyle

3 Rename The existing style called STANDARD to your new style.

4 Choose Element properties to change the appearance of the Multilines.

5 Choose ADD to create the new multiline.

Fig 10



Multiline Style

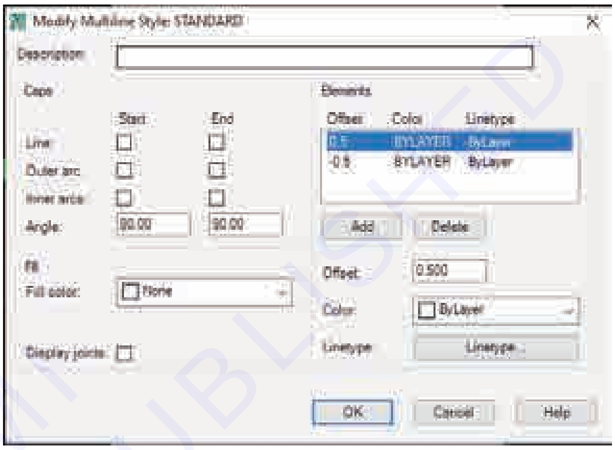
Current Multiline Style: STANDARD

Styles: STANDARD

Description:

Preview of: STANDARD

Fig 11



Modify Multiline Style: STANDARD

Description:

Caps

	Start	End
Line	☐	☐
Outer arc	☐	☐
Inner arcs	☐	☐
Angle:	90.00	90.00

Fill

Fill color: None

Display joints: ☐

Elements

Offset	Color	Linetype
0.5	BYLAYER	ByLayer
-0.5	BYLAYER	ByLayer

Offset: 0.500

Color: ByLayer

Linetype

Editing multilines (Figs 12 & 13)

1 Choose Modify, Multiline

(or)

2 Type MLEDIT at the command prompt

Command: MLEDIT

3 Choose from one of the mledit options

Construction line

Creates an infinite line.

1 Choose Draw, Construction Line

(or)

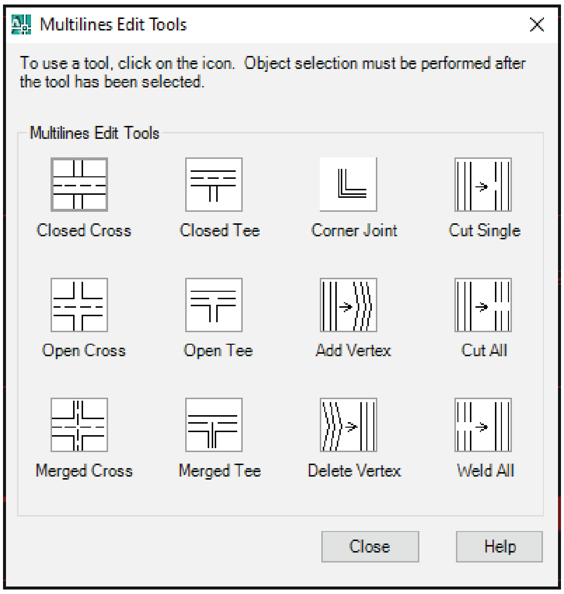
2 Choose the XLINE icon.

(or)

3 Type XLINE at the command prompt. Command: XLINE

Specify a point or [Hor/Ver/Ang/Bisect/Offset]

Fig 12



To use a tool, click on the icon. Object selection must be performed after the tool has been selected.

XLINE options (Figs 14 & 15)

HOR: Creates a horizontal xline passing through a specified point

VER: Creates a vertical xline passing through a specified point

ANG: Creates an xline at a specified angle.

BISECT: Creates an xline that passes through the selected angle vertex and bisects the angle between the first and second line

OFFSET: Creates an xline parallel to another object.

Ray command

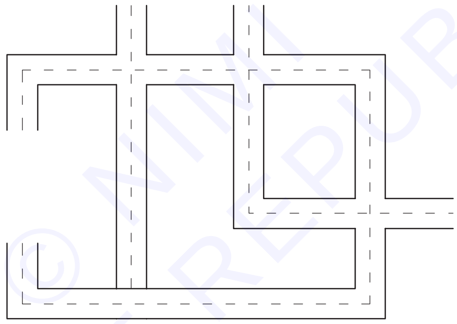
Creates an infinite line in one direction

1 Choose Draw, ray

(or)

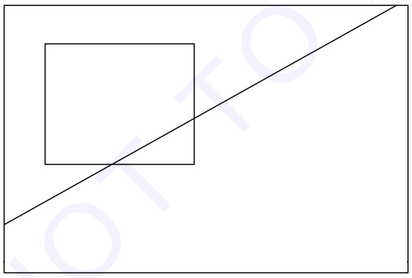
2 Type Ray at the command prompt. Command: Ray specify a point: (pick through point)

Fig 13



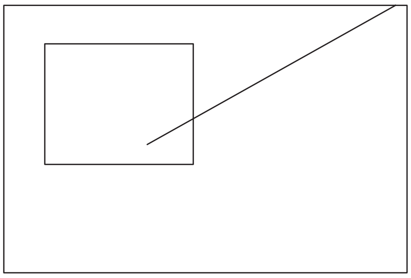
MULTILINES

Fig 14



XLINE

Fig 15



XLINE

Hatch command (Fig 16)

1 Choose Draw, hatch...

(or)

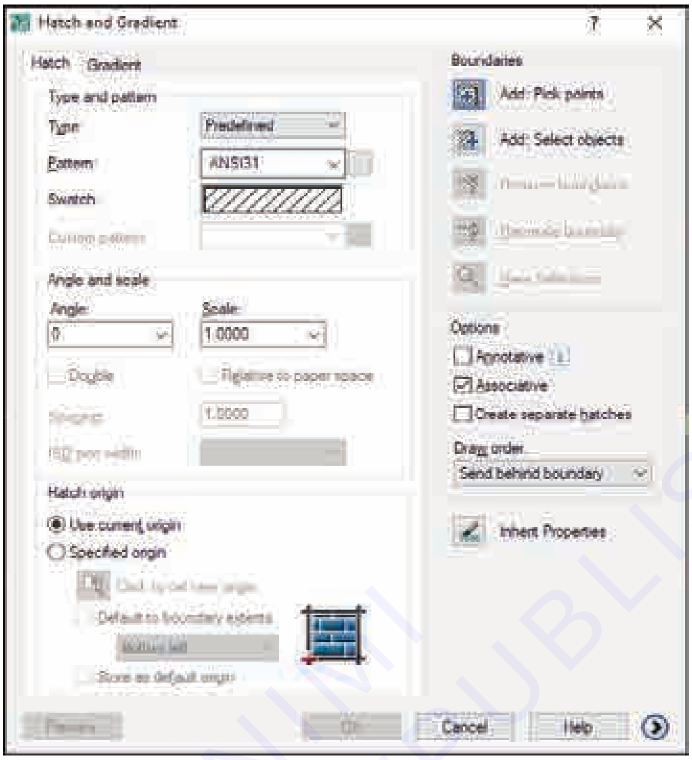
2 Click The hatch icon.

(or)

3 Type BHATCH at the command prompt

Command: BHATCH

Fig 16



Hatch and Gradient

Type: Predefined

Pattern: ANSI31

Angle: 0

Scale: 1.0000

☐ Annotative

☑ Associative

☐ Create separate hatches

Draw order: Send behind boundary

● Use current origin

○ Specified origin

Batch options

Pattern type: Sets the current pattern type by using AutoCAD's Predefined patterns or user defined patterns.

Pattern properties: Sets the current pattern, scale, angle, and spacing, Controls if hatch is double spaced or exploded.

Pick points: Constructs a boundary from existing objects that form an **enclosed area.**

Select objects: Selects specific objects for hatching. The boundary hatch dialog box disappears and AutoCAD prompts for object selection.

Inherit properties: Applies the properties of an existing associative hatch to the current pattern type and pattern properties options.

Preview hatch: Displays the hatching before applying it. AutoCAD removes the dialog box and hatches the selected areas.

Associative: Controls associative hatching.

Apply: Crates the crosshatching in the boundary.

Annotative hatch

Hatching from the design center

1 Choose: A cross hatch pattern from the following AutoCAD directly\AutoCADxxxx\Support\acad.pat or \AutoCADxxxx\Backup

2 Drag: and drop a pattern into a drawing.

TIP

Be sure the HPSCALE is set before dropping a hatch pattern into a drawing.

CAD basics

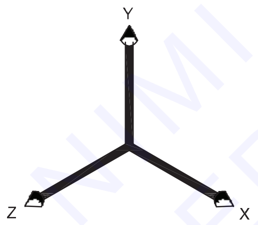
The CAD Database and the user coordinate system

Designs and drawings created in a CAD system are usually defined and stored using sets of points in what is called world space. In most CAD systems, the world space is defined using a three-dimensional Cartesian coordinate system. Three mutually perpendicular axes, usually referred to as the X-, Y-, and Z-axes, define this system. The intersection of the three coordinate axes forms a point called the origin. Any point in world space can then be defined as the distance from the origin in the X-, Y- and Z- directions. In most CAD systems, the directions of the arrows shown on the axes identify the positive sides of the coordinates.(Fig 1)

A CAD file, which is the electric version of the design, contains data that describes the entities created in the CAD system. Information such as the coordinate values in world space for all endpoints, center points, etc., along with the descriptions of the types of entities are all stored in the file. Knowing that AutoCAD stores designs by keeping coordinate data helps us understand the inputs required to create entities.

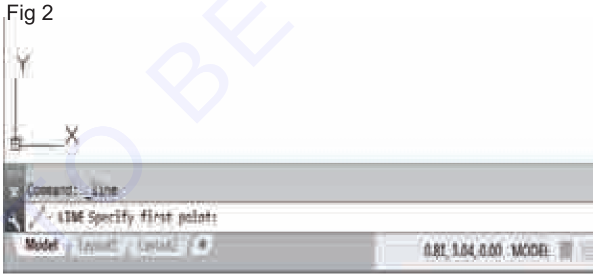
The icon near the bottom left corner of the default AutoCAD graphics window shows the positive X-direction and positive Y-direction of the coordinate system that is active. In AutoCAD, the coordinate system that is used to create entities is called the user coordinate system (UCS). By default, the user coordinate system is aligned to the world coordinate system (WCS). The world coordinate system is a coordinate system used by AutoCAD as the basis for defining all objects and other coordinate systems defined by the users. We can think of the origin of the world coordinate system as a fixed point being used as a reference for all measurements. The default orientation of the Z-axis can be considered as positive values in front of the monitor and negative values inside the monitor. (Fig 2)

Fig 1



USER COORDINATE SYSTEM

Fig 2



Command: _line

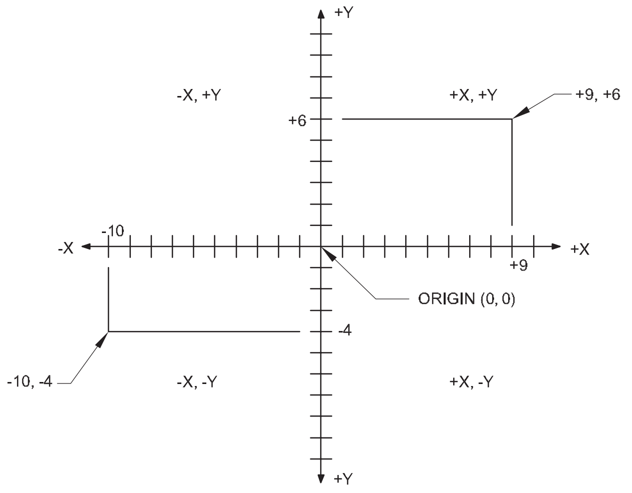
LINE Specify first point:

AutoCAD uses points to determine where an object is located. There is an origin where it begins counting from. This point is (0,0). Every object is located in relation to the origin. If you were to draw a line straight out to the right from the origin, this would be considered the positive X-axis. If you were to draw a line straight up, this would be the positive Y-axis. The picture above shows a point located at (9,6). This means that the point is 9 units over in the X-axis and 6 units up in the Y-axis. When you are working with points, X always comes first. The other point shown is (-10-4). This means that the point is 10 units in the negative X-axis (left) and 4 units in the negative Yaxis(down) (Fig 3)

A line has two points, a start point and an end point. AutoCAD works with the points to display the line on the screen. Move your cursor over the picture above and you will see line drawn from the absolute points of (-10-4) to (9,6).

Most of the time you will not have an indication of where the origin is. You may need to draw a line from the endpoint of an existing line. To do this you use relative points. These work the same way, but you have to add the @ symbol (shift+2) to tell AutoCAD that this next point is relative from the last point entered.

Fig 3



USER COORDINATE SYSTEM

To review

Absolute points are exact points on the drawing space.

Relative points are relative to an object on the drawing space.

its simple system, but mastering it is the key to working with AutoCAD and is explained in more detail further below. In order to work effectively with AutoCAD, you have to work with this system. Until you are comfortable and familiar with it, learning AutoCAD will be more of a chore. My experience in teaching is that the better a student is with coordinates, the faster they will learn.

Entering points in AutoCAD

You can enter points directly on the command line using three different systems. The one you use will depend on which is more applicable for the situation. The first assignment will get you used to this. The three systems are as follows.

Absolute co-ordinates: Using this method, you enter the points as they relate to the origin of the WCS. To enter a point just enter in the exact point as X,Y.

Relative co-ordinates : This allows you to enter points in relation to the first point you have entered. After you've entered one point, the next would be entered as @ X,Y. This means that AutoCAD will draw a line from the first point to another point X units over and Y units up relative to the previous point.

Polar co-ordinates: You would use this system if you know that you want to draw a line a certain distance at a particular angle. You would enter this as @ D<A. In this case, D is the distance and A is the angle. Example: @10<90 will draw a line 10 units straight up from the first point.

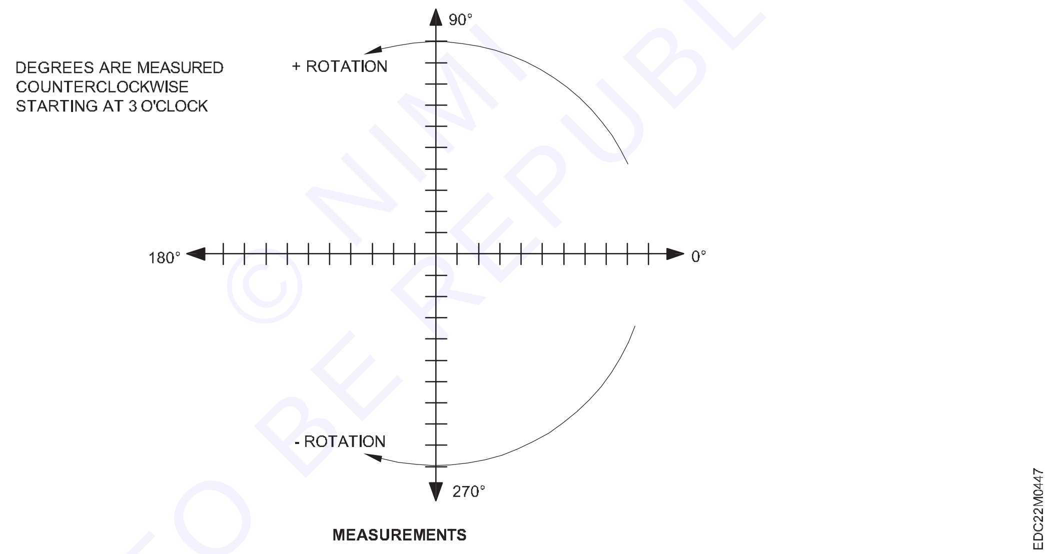
The three ways of entering coordinates shown above are the ONlY way AutoCAD accepts keyboard input. First decide which style you need to use, and then enter as shown. Remember that X is always before Y (alphabetical). Don't forget the '@' symbol when you are entering relative points. Any typing error or omission will give you results you don't want. If you make a mistake and need to see what you typed, Press F2 to bring up the text screen and check your typing. (press F2 to get back to your drawing.)

Key terms

Term	Description
Absolute coordinates	Distance measured from a fixed reference point.
Aperture	Effective diameter of the cursor on the screen.
Cartesian coordinates	A rectangular system of measurement to locate points in the drawing area.
Object snaps	A method for indicating point locations using existing drawing objects as a reference.
Origin point	The 0,0 location of the coordinate system.
Polar coordinates	A system to locate of the coordinate system.
Prototype drawing	A template drawing that has a last location of the cursor.
Relative coordinates	Distance measured from the last location of the cursor
User-defined co-ordinates	A mode of measurement that allows the user to set up a customized coordi nate System system.

Angular measurement (Fig 4)

Fig 4



AutoCAD measures angles in a particular way also. Look at the diagram below and then place your mouse on it to see how this is

Degrees are measured counterclockwise starting at 3 O'CLOCK

When drawing lines at an angle, you have to begin measuring the angle from 0 degrees, Which is at the 3 O'clock position. If you drew a line at 90 degrees, it would go straight up. The example above (When you move your mouse over it) shows a line drawn at+300 degrees (270+30), or-60 degrees.

Your might not always have an obvious reference point for 0 degrees. Look at the example below and place your mouse on the image to find out the angle in question.

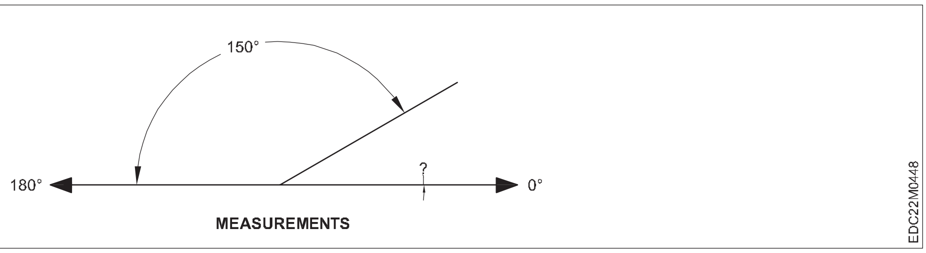
In this example, you are given information about the lines, but not the angle AutoCAD needs to draw the line from the start point. What you are given though, is (a) the knowledge that 0°is at the 3 o'clock position (b) the knowledge that180° is at the 9 o'clock position and (c) the angle between 180° and the line you want to draw is 150°. With this information, you can figure out what angle you need. Here is a fool-proof way of getting the angle you need (Fig 5)

1 Start at the 0° position and measure counter-closkwise (+) to 180°

2 From 180°, measure clockwise 150°(-)

3 Consider that you just went+180-150 and use that as an equation:+180-150=30 (Fig 5)

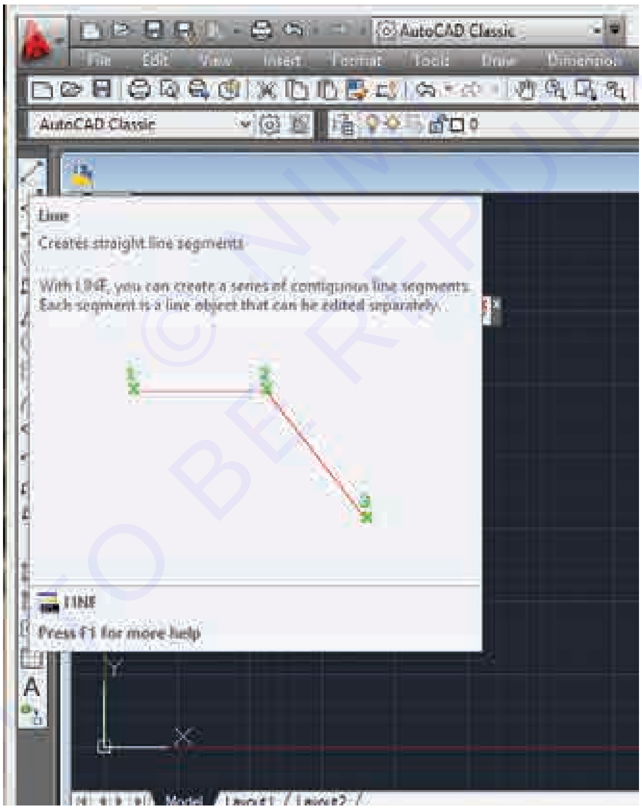
Fig 5



4 Now you can draw your line using polar coordinates (discussed below)

There are many ways to do things in most windows programs. AutoCAD is no exception. Everyone will develop a way that works best for him or her. In this course, we will primarily be working with the keystroke commands. The reason for this is because they will work in most AutoCAD versions (including DOS versions), and in some other CAD programs. The icons work well, but as you will see, icons can be placed anywhere on the screen and can be difficult to find quickly. You may be working on another employee's computer that is set up differently than what you're used to. The pull down menus will access almost all commands, but are a slower way of doing things. Icons in AutoCAD 2010 are found on the ribbon, divided into panels-just click on the appropriate tab to open the panel you need. (Fig 6)

Fig 6



Line

Creates straight line segments

With LINE, you can create a series of contiguous line segments. Each segment is a line object that can be edited separately.

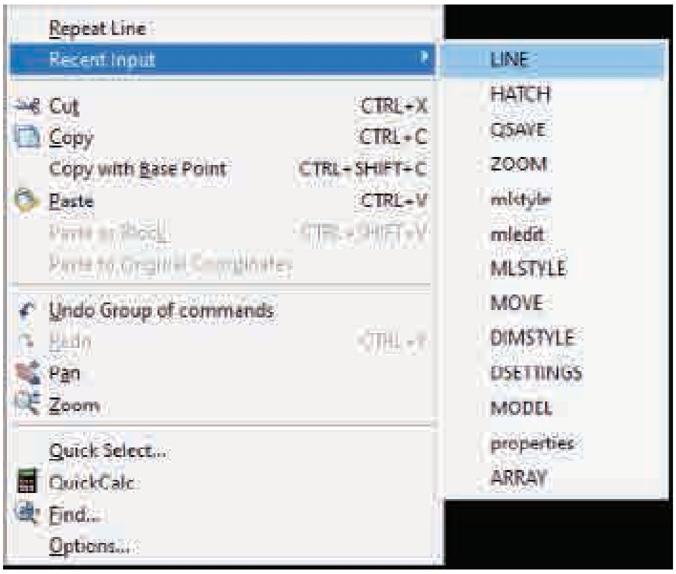
LINE

Press F1 for more help

Example: If you want to draw a line, you can do it a few ways:

At the command line type: LINE (or) L and press the ENTER key. (Fig 7)

Fig 7



Select the line icon from the DRAW Panel.

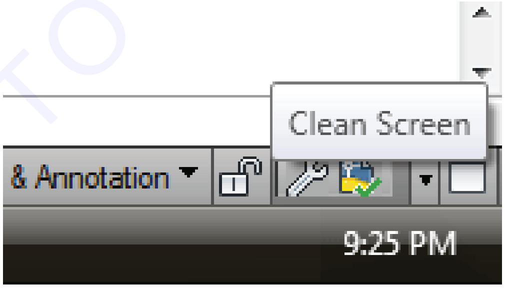
Another way is to Right-Click on the drawing space and choose “Recent Input” from the menu. This will give a list of the most recent command that you have used.

All three approaches will do the same thing: prepare AutoCAD to draw a line where you tell it.

AutoCAD is a popular program because it can be customized to suit an individual’s needs. The toolbars are a good example of this. You can have the toolbars you use most often on the screen all the time. You can easily make them go away so that you have more drawing space. You can also customize them so you have the most common commands on one toolbar. For example, the dimensioning toolbar is one that you will not want taking up space on your screen while drawing, but is very handy when you’re dimensioning your drawing.

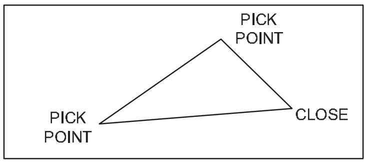
To remove the ribbon and have the most drawing space available, click on the “Clean Screen” icon in the bottom right corner of the screen (or press CTRL+O[letter O]. To go back the to the standard display, click again on the same icon. (Fig 8)

Fig 8



Line command (Fig 9)

Fig 9



LINE COMMAND

Create single straight line segments

1 Choose draw, line

(or)

2 Click the line icon

(or)

3 Type line from the command prompt command: line or L

4 Press enter

5 Pick from point: (point)

6 Pick specify next point or [Close/Undo]: (point)

7 Pick specify next point or [Close/Undo]: (point)

8 Press ENTER to end line sequence

(or)

9 TypeU to undo the last segment to point: U (undo)

(or)

10 Type C to create a closed polygon to point: C (close)

Erase and selection sets

Erasing Objects (Fig 10)

1 Choose Modify, erase

(or)

2 Click The Erase icon

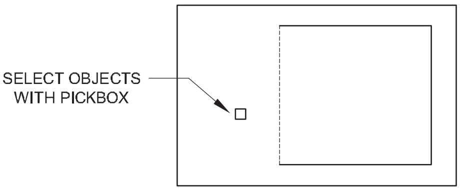
(or)

3 Type Erase at the command prompt. Command: Erase or E

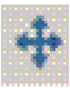
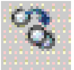
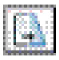
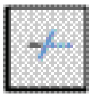
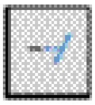
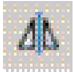
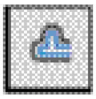
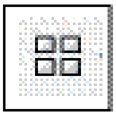
4 Pick Object at the select object prompt.

5 Press ENTER when you are done choosing objects. Select objects: ENTER

Fig 10



ERASE COMMAND

Symbol	Command	Purpose
	Erase	Delete object
	Move	Move object one place to other place
	Copy	Create one or more copies of object
	Stretch	Stretch, shorten, or move object
	Trim	Shorten object using other object
	Extend	Lengthen object using object
	Mirror	Creates a mirror image of objects.
	Rotate	Rotate objects around a specified point.
	Offset	Create a new object at a specified distance from an existing object or through a specified point.
	Array	Each object in an array can be manipulated independently.

Auto CAD Drawing Commands

Symbol	Command	Major option	Toolbar button	Draw men
╱	Line	Start, End Point	Line	Line
▫	Mline	Justification, Scale Style	None	Multiline
⤻	Pline	Vertices	Polyline	Polyline
⬠	Polygon	Number of sides, Inscribed / Circumscribed	Polygon	Polygon
▭	Rectangle	Two Corner	Rectangle	Rectangle
◜	Arc	Various methods of definition	Arc	Arc, submenu for definition methods
◯	Circle	Three point, two point, Tangent	Circle	Circlesubmenufor definition methods
■	Donut	Inside, Outside Diameters	None	Donut
∼	Spline	Convert polyline or Create new	Spline	Spline
⬭	Ellipse	Arc, center, axis	Ellipse	Ellipse, submenu for definition methods
☁	Revcloud	Arc Length	Revcloud	Revision cloud

Break, offset, trim, chamfer, copy, explode commands (Modify commands)

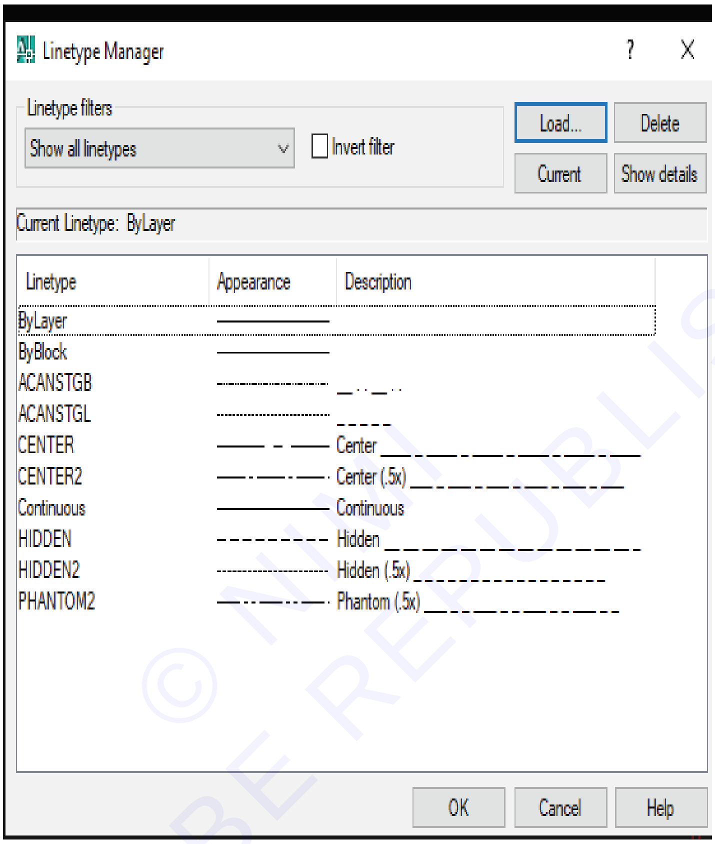
Rectangle (REC) (Fig 1)

To create a rectangle you must specify two diagonal corners or Length and Width.

The rectangle can be any size and the sides are always drawn horizontal and vertical.

A Rectangle is one object (Poly line) , not four separate lines.

Fig 1



Current Linetype: ByLayer

Linetype	Appearance	Description
ByLayer	————————	
ByBlock	————————	
ACANSTGB	··························	_ . . _ . .
ACANSTGL	··························	- - - - -
CENTER	—— - ——	Center ____ _ ____ _ ____ _ ____
CENTER2	—·—·—	Center (.5x) ___ _ ___ _ ___ _ ___ _ ___
Continuous	————————	Continuous
HIDDEN	- - - - - - - - -	Hidden __ __ __ __ __ __ __ __ __
HIDDEN2	--------------------	Hidden (.5x) _ _ _ _ _ _ _ _ _ _ _ _ _
PHANTOM2	—··—··—	Phantom (.5x) ___ _ _ ___ _ _ ___ _ _

Explode (D)

Separates a Polygon, block, dimension or hatch pattern into its constituent entities or makes a polyline into a series of straight lines. In the case of a block that is exploded, if it was originally drawn on the 0 layer, it returns to that layer, regardless of the layer it was inserted on, and it loses its referential connection to the original block. In the case of a dimension or hatch pattern that has been exploded, their parts go back to the 0 layer, and are assigned the logical color (BYBLOCK) regardless of the layer they were drawn on. In the case of an exploded polyline, it loses any width it had.

Circle (C)

Draws a circle of any size. The default method is to pick a center point and pick a point on the radius or type the radius dimension, but other methods can be selected.

2P Specifies circle by picking 2 points on the diameter 3P Specifies circle by picking 3 points through which the circle will pass D Allows entering the diameter

dimension instead of radius dimension TTR Specifies circle by picking two lines, arcs or circles for the circle to be tangent to, and entering the dimension of the radius <RET> Enters radius of circle (the default)

Polygon

Draws regular polygons with a specified number of sides. Polygons are Polyline entities.

E Specifies size and rotation of polygon by picking endpoints of one edge C Circumscribes polygon around a circle I Inscribes polygon within a circle

Copy (CP,CO)

Allowing the copying of a selected objects in to number of identical objects. M Allows multiple copies to be made of an object.

Osnap(F3)

Enables points to be precisely located on reference points of existing objects.

CEN CENter of arc or circle

END closest ENDpoint of arc or line

INS INSertion point of Text or Block

INT INTersection of line, arc, or circle

MID MIDpoint of line, arc, rectangle side, or polygon side

NEA NEArest point selected by aperture on line, polyline, arc, or circle

NOD NODe (another name for a Point)

PER PERpendicular

QUA QUAdrant point of arc or circle (top, bottom, right or left side)

TAN TANgent point to arc

Are some of the Osnap Options

Break (BR)

Erases part of a line, arc or circle, or splits it into two lines or arcs.

ERASE(E)

Erases selected entities from the drawing.

Trim(TR)

Erases a portion of selected entities that cross a specified “cutting edge”

Extend(EX)

Lengthens a line, arc, or polyline to meet a specified “boundary edge”

You can extend the lines with trim command by the combination of Shift key and vice versa

Linetype (P)

A linetype is a series of lines and spaces ...... dots and dashes ---- Sets the current line type to be used for all subsequent entities to be drawn, allows creation of new linetype definitions, and allows loading of previously created linetypes (Fig 1). Lists a line type library , or named linetypes which are Loaded. C Creates a line type definition L Loads a previously created line type definition S Set current line type - note that the current line type should always be set to the logical line type “BYLAYER”

Chamfer (CHA)

Creates a chamfer (a angled line connection) at the intersection of two lines D Sets chamfer distances P Chamfers an entire polyline

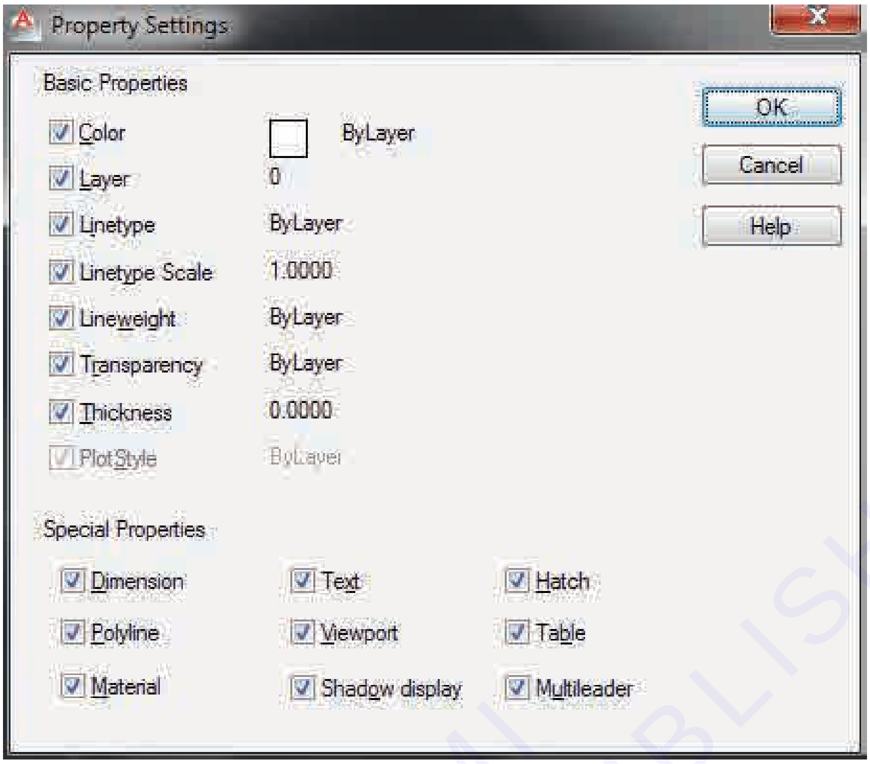
Matchprop (MA)

It is a useful tool used for copying the whole properties of an object to several objects.

Click the source hatch pattern you want to copy. Click the target hatch pattern you want to change. The target pattern changes to match the source pattern.

After selecting the source object enter S for getting property setting dialog box where you can set the properties to be transferred (Fig 2).

Fig 2



Basic Properties

☑ Color ☐ ByLayer
☑ Layer 0
☑ Linetype ByLayer
☑ Linetype Scale 1.0000
☑ Lineweight ByLayer
☑ Transparency ByLayer
☑ Thickness 0.0000
☑ PlotStyle ByLayer

Special Properties

☑ Dimension
☑ Polyline
☑ Material
☑ Text
☑ Viewport
☑ Shadow display
☑ Hatch
☑ Table
☑ Multileader

Ellipse, hatch and move commands

Ellipse (EL)

ELLIPSE Draws ellipses C Allows specification of Center point of ellipse rather than first axis endpoint.

R Allows specification of eccentricity rotation rather than length of second axis.

I draws Isometric circle in current isoplane

Move (M)

Moves selected entities to another location.

The following prompts are displayed.

Select objects

Specify base point or [Displacement]

Specify second point or <use first point as displacement>:

The two points you specify define a vector that indicates how far the selected objects are to be moved and in what direction.

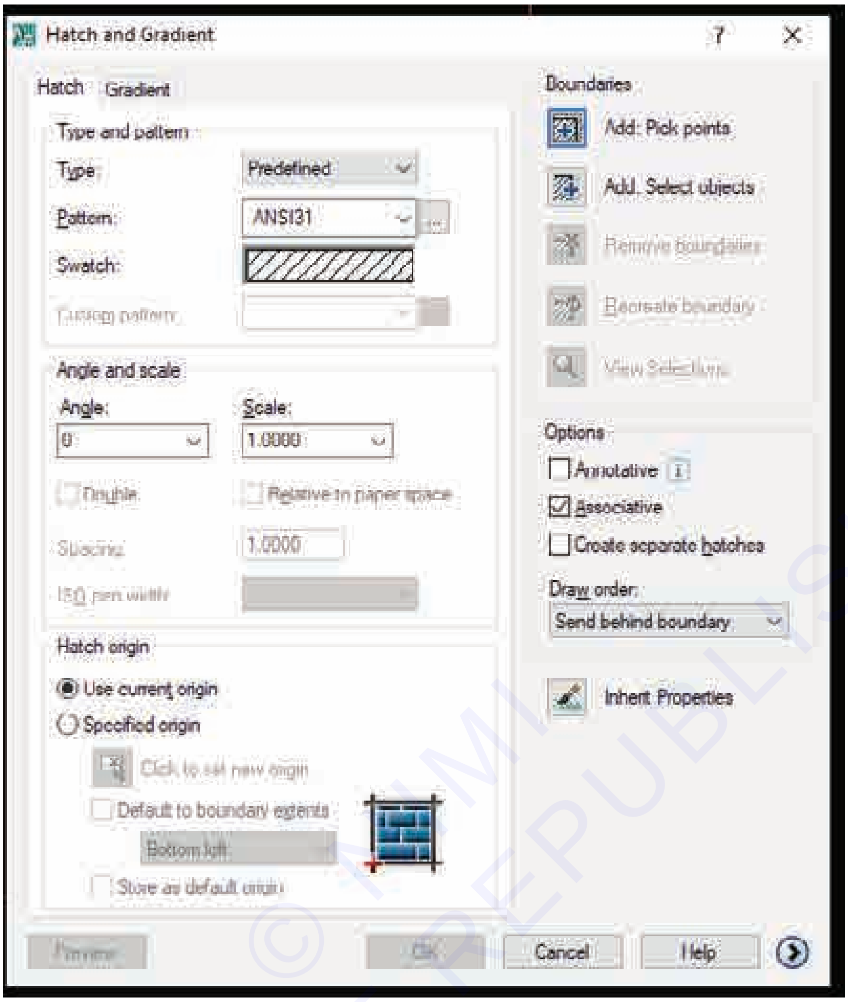
If you press Enter at the Specify Second Point prompt, the first point is interpreted as a relative X,Y,Z displacement. For example, if you specify 2,3 for the base point and press Enter at the next prompt, the objects move 2 units in the X direction and 3 units in the Y direction from their current position. Displacement Enter coordinates to represent a vector

The coordinate values that you enter specify a relative distance and direction.

Hatch (H)

Allows drawing of cross-hatching and pattern filling <pattern-name> uses hatch pattern name from library file to fill designated area with a pseudo-block hatch entity - can be reduced to individual lines and points by EXPLODEing it. <*pattern-name> uses hatch pattern name from library file to fill designated area with individual lines and points, not a block U “User-defined” hatch pattern drawn of parallel lines with a specified distance between, at a specified angle, and either single-hatching or double (cross) - hatching (Fig 1).

Fig 1



Type: Predefined

Pattern: ANSI31

Swatch: ▨

Angle: 0

Scale: 1.0000

☐ Double

☐ Relative to paper space

Spacing: 1.0000

● Use current origin

○ Specified origin

☐ Default to boundary extents

Bottom left

☐ Store as default origin

☐ Annotative

☑ Associative

☐ Create separate hatches

Draw order: Send behind boundary

Pick the internal point of the boundary to be hatched after choosing the **add: pick point button** in the hatch dialog box or select objects of the boundary after choosing add : select object button

Select required hatch pattern from swatch/pattern option Alter Angle and scale of pattern if necessary.

Watch **preview** before applying hatch.

User defined hatch pattern can also be made.

Fillet and polar array commands

Array (AR)

Makes multiple copies of selected objects in a rectangular (parallel with the snap rotation) or circular pattern R rectangular array type P "polar" array type in circular pattern (Fig 1).

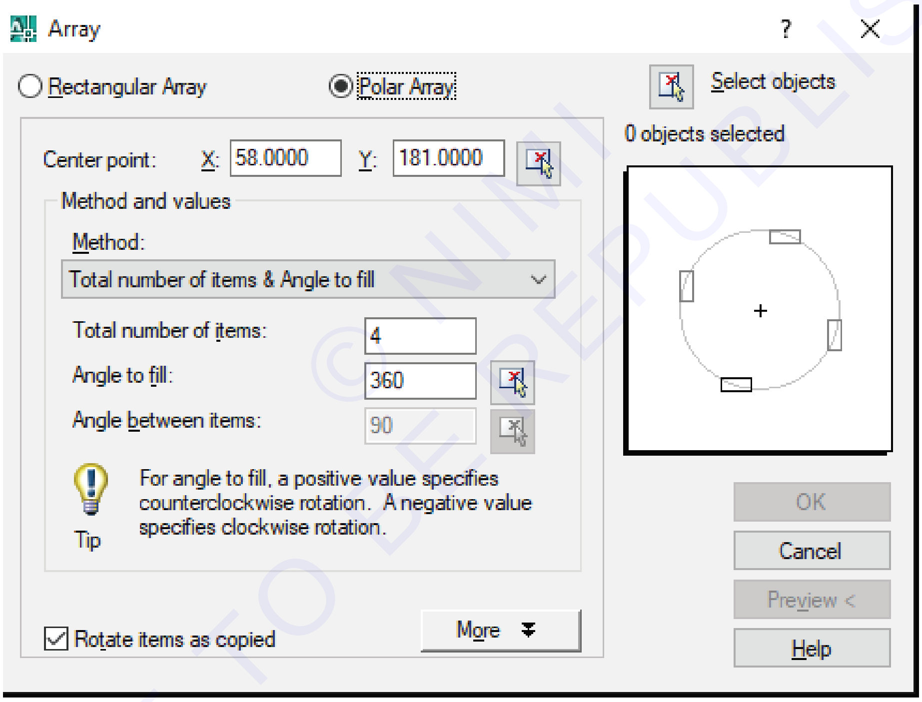
Polar array

To make multiple copies of an object in circular pattern. Check the option button Polar in the array dialog box. Select object to array after choosing select object button in the dialog box. Enter number of items required and pick for the center of array after **choosing the center point** button in the dialog box. Supply the angle of array. Finally check the check box for rotating the object with respect to array center.

Fillet (F)

Constructs an arc of specified radius between two lines, arcs, circles, or will create arcs of the specified radius at the vertices of a polyline. Radius of the arc to be constructed may be set to 0, which will make a sharp corner P Fillets an entire Polyline at the vertices R Allows **setting of the fillet radius**. Default value is 0. Radius remains set until changed again.

Fig 1



Array

○ Rectangular Array ● Polar Array

Select objects

0 objects selected

Center point: X: 58.0000 Y: 181.0000

Method and values

Method: Total number of items & Angle to fill

Total number of items: 4

Angle to fill: 360

Angle between items: 90

Tip: For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.

☑ Rotate items as copied

Rectangular array and area commands

Polyline (PL)

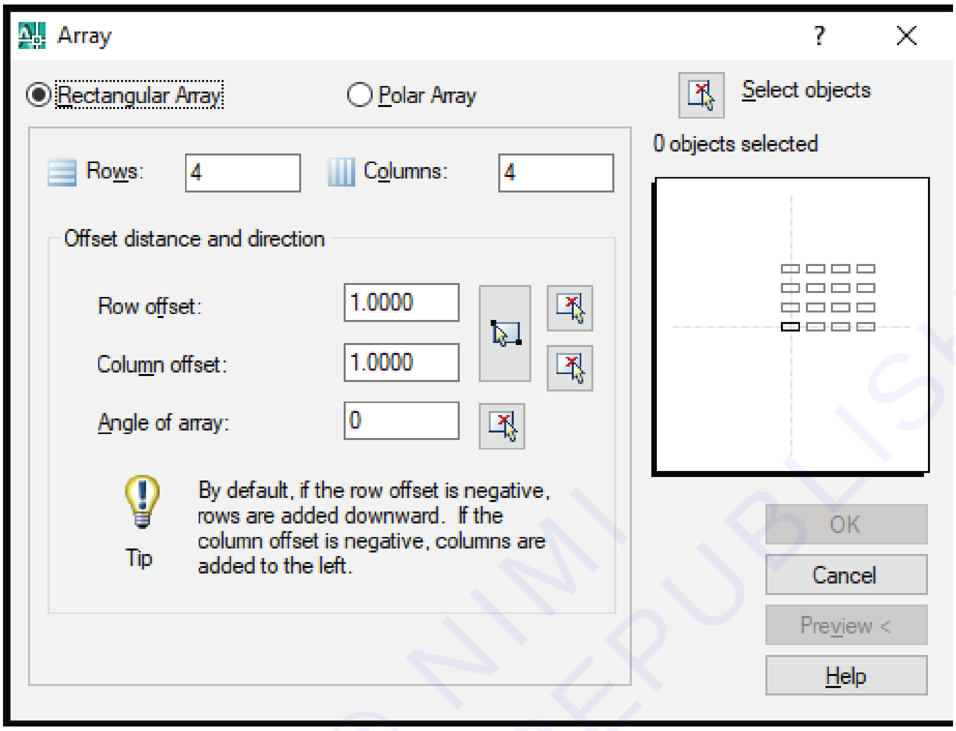
PLINE Draws a 2D polyline, which are connected line or arc segments with user-determined width and taper H sets the half-width of the polyline segments U Undoes previous segment W sets the width of polyline segments <RET> Exits the Pline command The following options are available only as long as you are drawing in Line mode: A Switches to the Arc mode to allow integration of arcs into the polyline C Closes the polyline with straight segment L Segment length (continues previous segment)

The following options are available only as long as you are in the Arc mode: A Included angle CE Center Point CL Closes with arc segment D Starting direction.

Rectangular Array (Fig 1)

To make multiple copies of an object in Rectangular attend ie. In rows and columns. Check the option button rectangular in the array dialog box. Select object to array after choosing select object button in the dialog box. Enter number of rows and columns required and row offset (distance) and column offset (distance) in the dialog box. Supply the angle of array if any. Watch preview before apply the array to the drawing.

Fig 1



● Rectangular Array ○ Polar Array

Select objects

0 objects selected

Rows: 4

Columns: 4

Offset distance and direction

Row offset: 1.0000

Column offset: 1.0000

Angle of array: 0

Tip

By default, if the row offset is negative, rows are added downward. If the column offset is negative, columns are added to the left.

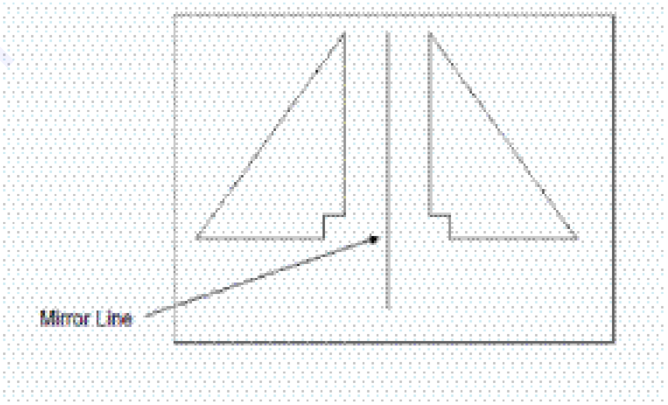
Arc and mirror commands

Mirror (MI)

Reflects designated entities about a user-specified axis

Click the Mirror tool on the Home tab’s Modify panel. You can also enter MI at the Command prompt (Fig 1).

Fig 1



Select objects: prompt, pick the long arc on the left side of the tub to highlight the arc. Press m to indicate that you've finished your selection.

Specify first point of mirror line: prompt, pick the midpoint of the top horizontal line. By now, you should know how to use the automatic osnap modes you set up earlier.

Specify second point of mirror line: prompt, use Polar Tracking mode to pick a point directly below the last point selected.

Erase source objects? [Yes/No] <N>: prompt, press m to accept the Mirror command's. Default Erase source objects option (No) and exit the Mirror command.

Arc (A) (Fig 2)

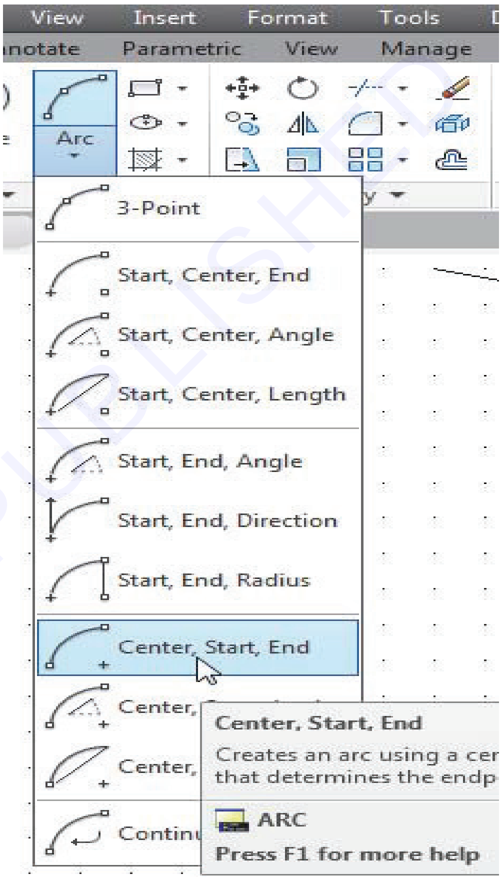
Draws an arc : The default method of drawing arcs is selecting three points (so-called "3 point arc"), which are the two endpoints of the arc and some other point along its locus. Other methods of drawing an arc can be specified by three letters, such as SEA, which means "Start Point, End Point, and Included Angle.

The following letters used to select the described option.

- Included angle
- Center point of arc
- Direction angle of a line tangent to the arc
- End point of arc
- Length of chord passing through both end points of the arc
- Radius
- Start point of arc

  < RET> uses the end of the last line or arc as the start point for the arc.

Fig 2



Center, Start, End

Creates an arc using a cer
that determines the endp

Press F1 for more help

Dimensioning on AutoCAD

Dimensioning

Dimensioning can be crucial to how well a design works and how quickly it develops.

The dimensions answer questions about tolerances, fit, and interference if you're involved in mechanical applications.

After you and your design team reach a design on a schematic level, communicating even tentative dimensions to others on the team can accelerate design development. Dimensions represent a point from which you can develop your ideas further.

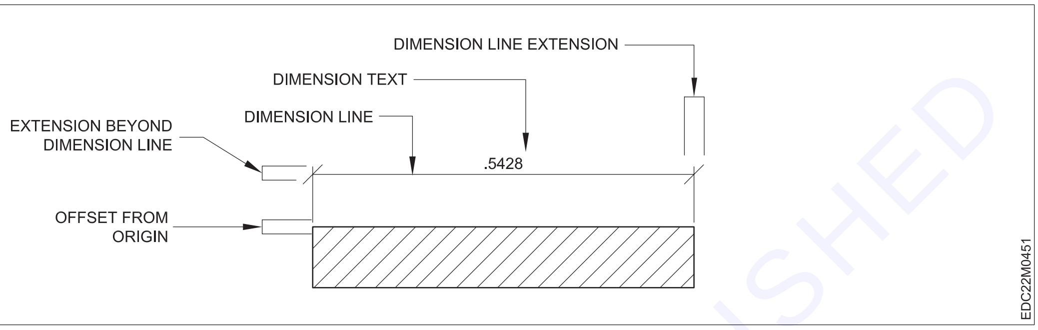
With the AutoCAD software, you can easily add tentative or final dimensions to any drawing. AutoCAD gives you an accurate dimension without having to take measurements. You pick the two points to be dimensioned and the dimension line location, and AutoCAD does the rest. The associative dimensioning capability of AutoCAD automatically updates dimensions whenever the size or shape of the dimensioned object changes. These dimensioning features can save you valuable time and reduce the number of dimensional errors in your drawings.

Components of a Dimension (Fig 1)

Before you start the exercises in this chapter, it will help to know the names of the parts of a dimension. Figure shows a sample of a dimension with the parts labeled. The dimension line is the line that represents the distance being dimensioned. It's the horizontal line with the diagonal tick marks on either end. The extension lines are the lines that originate from the object being dimensioned. They show you the exact location from which the dimension is taken. The dimension text is the dimension value, usually shown inside or above the dimension line. They show you the exact location from which the dimension is taken. The dimension text is the dimension value, usually shown inside or above the dimension line.

Here Describe few Dimensioning commands

Fig 1



Dimlinear

Draws a linear dimension with the dimension either horizontal or vertical.

Dimaligned

Draws a linear dimension with the dimension line parallel to the selected dimension origin points. This lets you align a dimension with an angled line that is not either exactly horizontal or vertical.

Dimradius

To draw dimensions of radius circles or arcs

Dimdiameter

To draw dimensions of diameter circles

Dimcontinue

Allows continued or series dimensioning.

Dimbaseline

Allows baseline or series dimensioning.

Qleader

Draws a line or series of lines with an arrow head (commonly called a "leader") to point to an object to notate it. This command will also prompt you for the note at the end of the leader line, but it will allow only one line of text. The most useful method of using this command is to simply draw leaders between the object to be notated and text that you create with the DTEXT command.

Dimedit

Editing the dimension object. Adjusts the obliquing angle of a linear associative dimensions and extension lines. Useful in dimensioning isometric drawings.

Dimangular

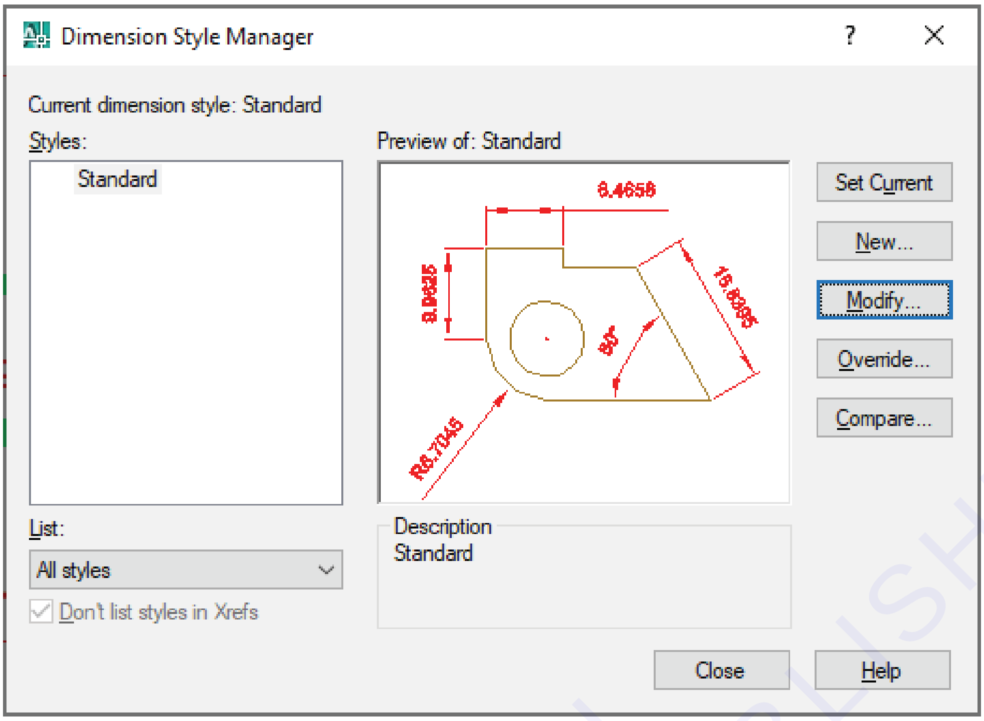
Allows the dimensioning of angles.

Adjusting Dimension Styles

Dimstyle (D) (Figs 2 & 3)

From the dialog box You can create **your own dimension styles** after keeping the base as any one of the default styles namely Annotative, ISO-25, Standard. Click New.

Fig 2



Current dimension style: Standard

Styles:
Standard

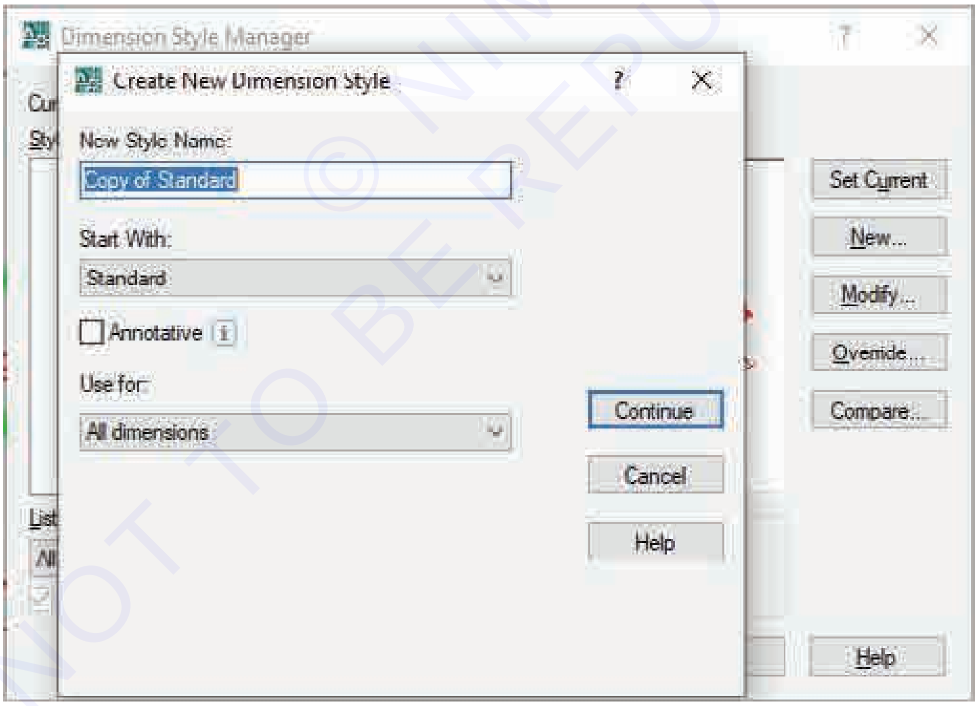
Preview of: Standard

List:
All styles

☑ Don't list styles in Xrefs

Description
Standard

Fig 3



New Style Name:
Copy of Standard

Start With:
Standard

☐ Annotative

Use for:
All dimensions

After saving you can click on modify button to edit your dimension style. (Fig 4)

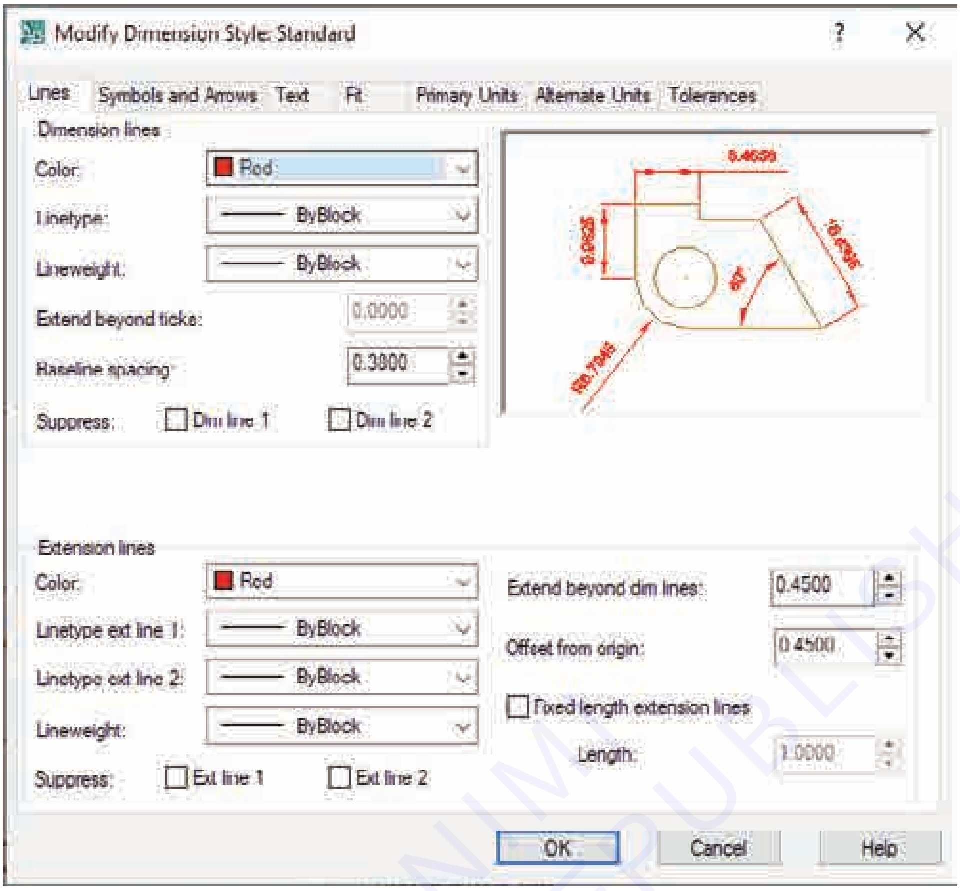
Line tab in dialog box is activated now. You can Change the dimension line and extension line features in this tab. (Fig 5)

In the symbol arrow tab you can change arrow head size, shape and some other features. (Fig 6)

With Text tab you can redefine the properties of dimension text ie; size, color, alignment, font etc.

With fit Tab we can select the best fit for the dimension text in the dimension object. (Fig 7)

Fig 4



Dimension lines

Color: Red

Linetype: ByBlock

Lineweight: ByBlock

Extend beyond ticks: 0.0000

Baseline spacing: 0.3800

Suppress: ☐ Dim line 1 ☐ Dim line 2

Extension lines

Color: Red

Linetype ext line 1: ByBlock

Linetype ext line 2: ByBlock

Lineweight: ByBlock

Suppress: ☐ Ext line 1 ☐ Ext line 2

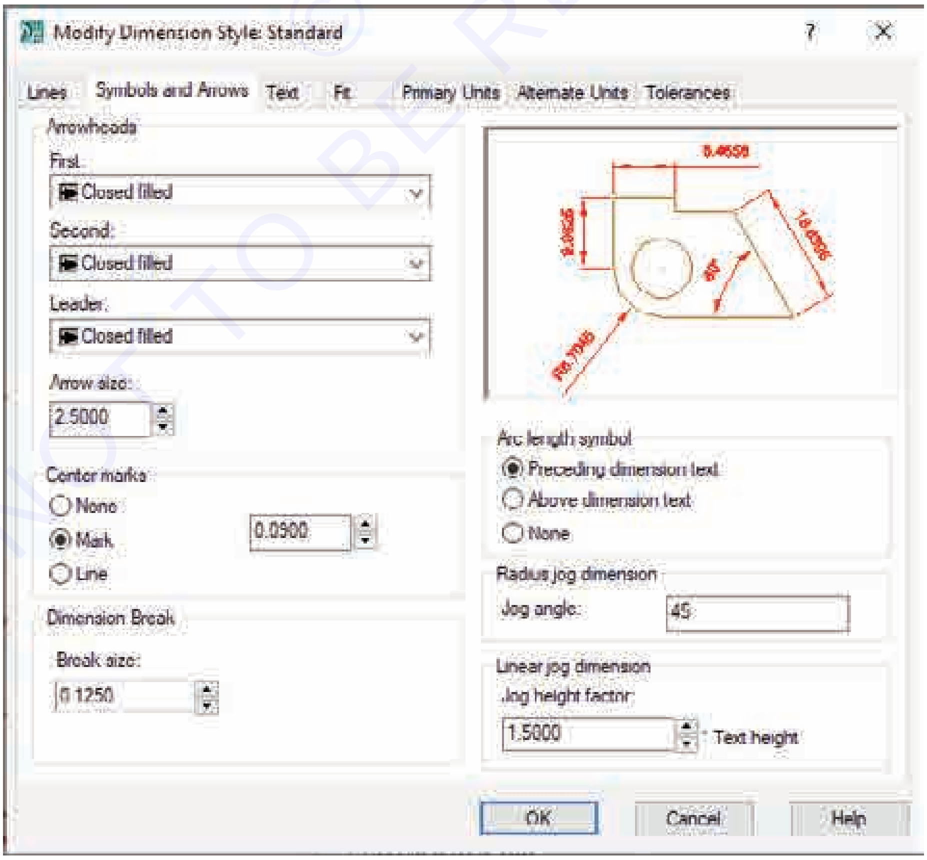
Extend beyond dim lines: 0.4500

Offset from origin: 0.4500

☐ Fixed length extension lines

Length: 1.0000

Fig 5



Arrowheads

First: Closed filled

Second: Closed filled

Leader: Closed filled

Arrow size: 2.5000

Center marks

○ None

● Mark 0.0900

○ Line

Dimension Break

Break size: 0.1250

Arc length symbol

● Preceding dimension text

○ Above dimension text

○ None

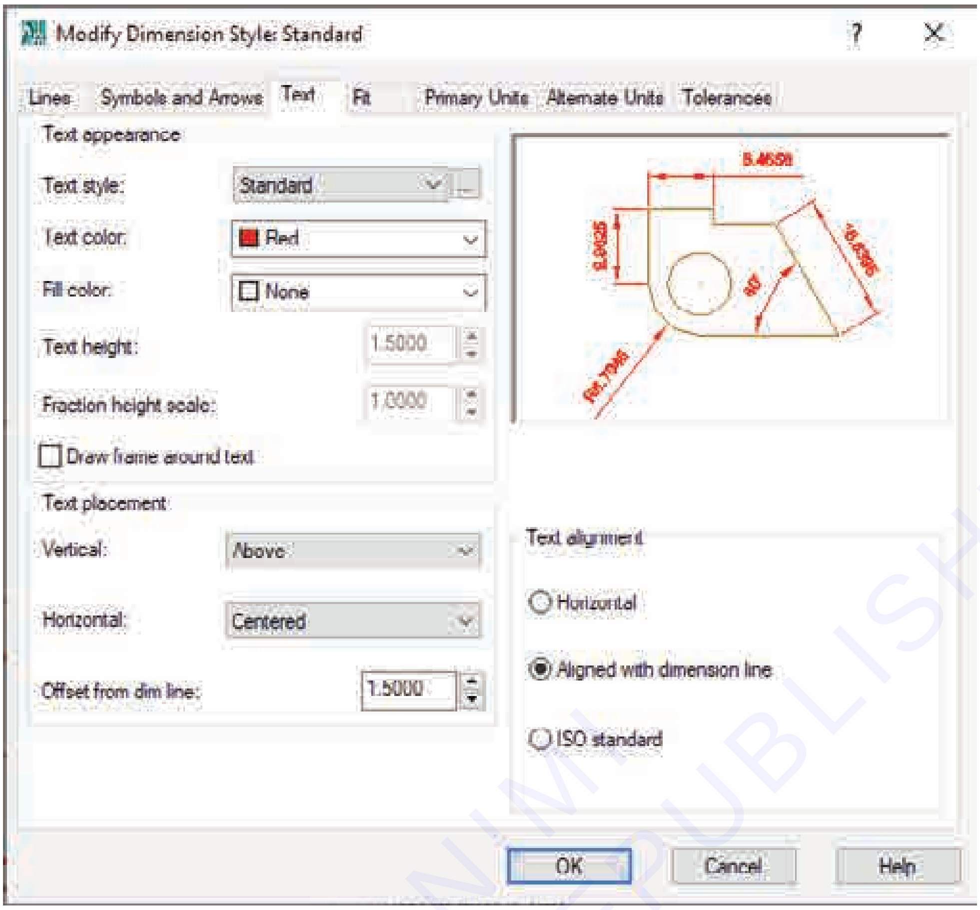
Radius jog dimension

Jog angle: 45

Linear jog dimension

Jog height factor: 1.5000 * Text height

Fig 6



Modify Dimension Style: Standard

Text appearance

Text style: Standard

Text color: 🟥 Red

Fill color: ☐ None

Text height: 1.5000

Fraction height scale: 1.0000

☐ Draw frame around text

Text placement

Vertical: Above

Horizontal: Centered

Offset from dim line: 1.5000

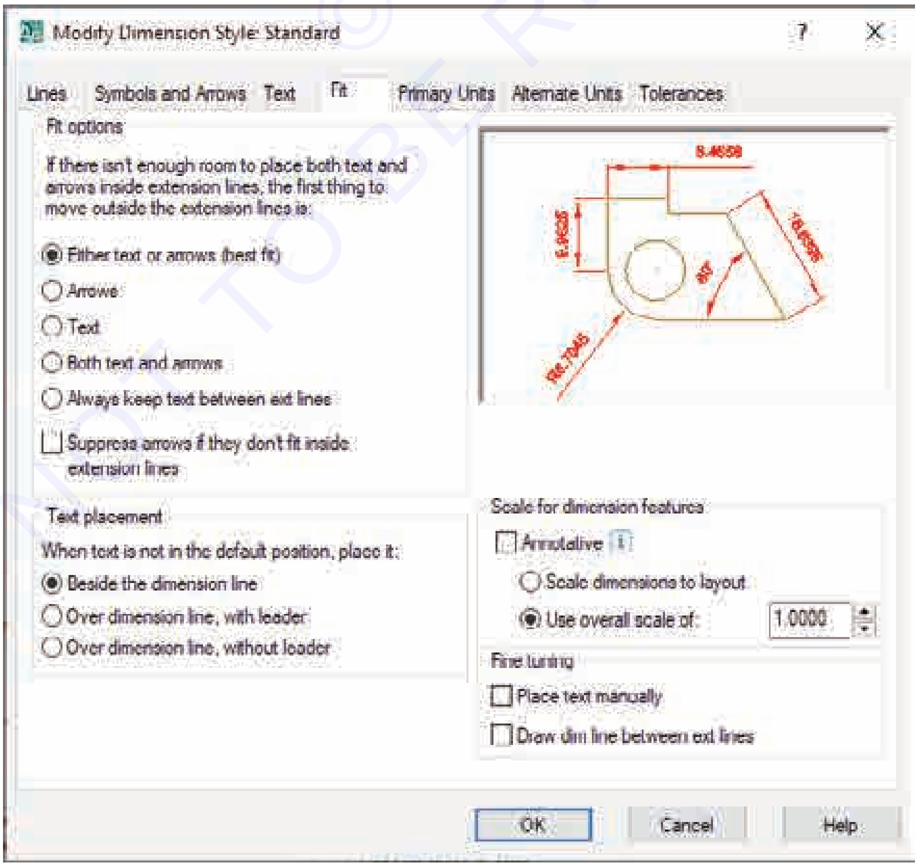
Text alignment

○ Horizontal

● Aligned with dimension line

○ ISO standard

Fig 7



Modify Dimension Style: Standard

Fit options

If there isn't enough room to place both text and arrows inside extension lines, the first thing to move outside the extension lines is:

● Either text or arrows (best fit)

○ Arrows

○ Text

○ Both text and arrows

○ Always keep text between ext lines

☐ Suppress arrows if they don't fit inside extension lines

Text placement

When text is not in the default position, place it:

● Beside the dimension line

○ Over dimension line, with leader

○ Over dimension line, without leader

Scale for dimension features

☐ Annotative

○ Scale dimensions to layout

● Use overall scale of: 1.0000

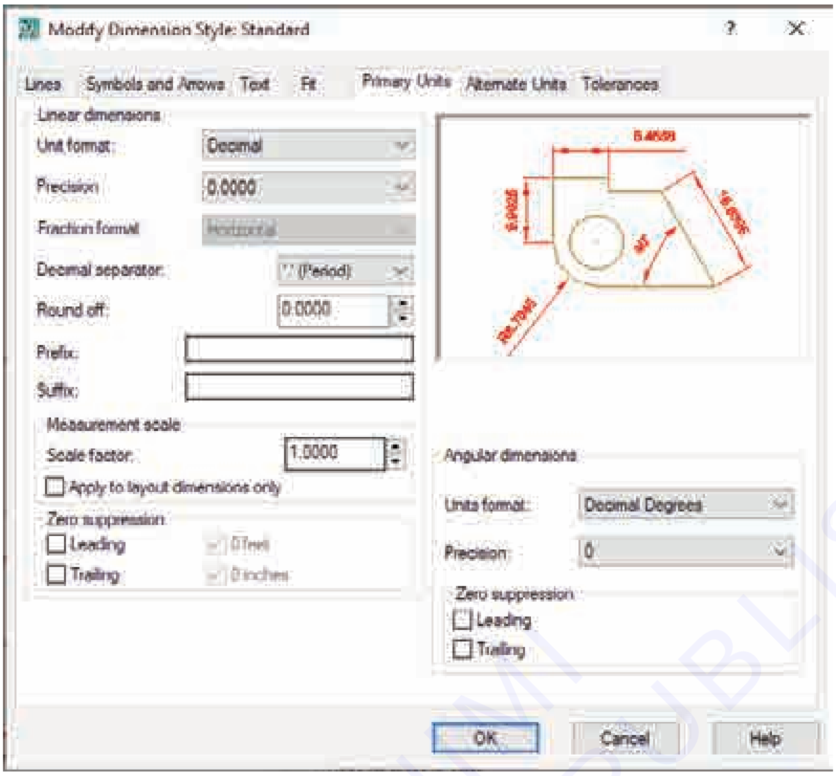
Fine tuning

☐ Place text manually

☐ Draw dim line between ext lines

AutoCAD allows primary and alternate unit for a drawing. In this tab we can adjust primary unit features such as precision round off, Decimal separator symbol, angular measurement unit and its precision. (Figs 8,9 & 10).

Fig 8



Linear dimensions

Unit format: Decimal

Precision: 0.0000

Fraction format: Horizontal

Decimal separator: '.' (Period)

Round off: 0.0000

Prefix:

Suffix:

Measurement scale

Scale factor: 1.0000

☐ Apply to layout dimensions only

Zero suppression

☐ Leading ☑ 0 feet

☐ Trailing ☑ 0 inches

Angular dimensions

Units format: Decimal Degrees

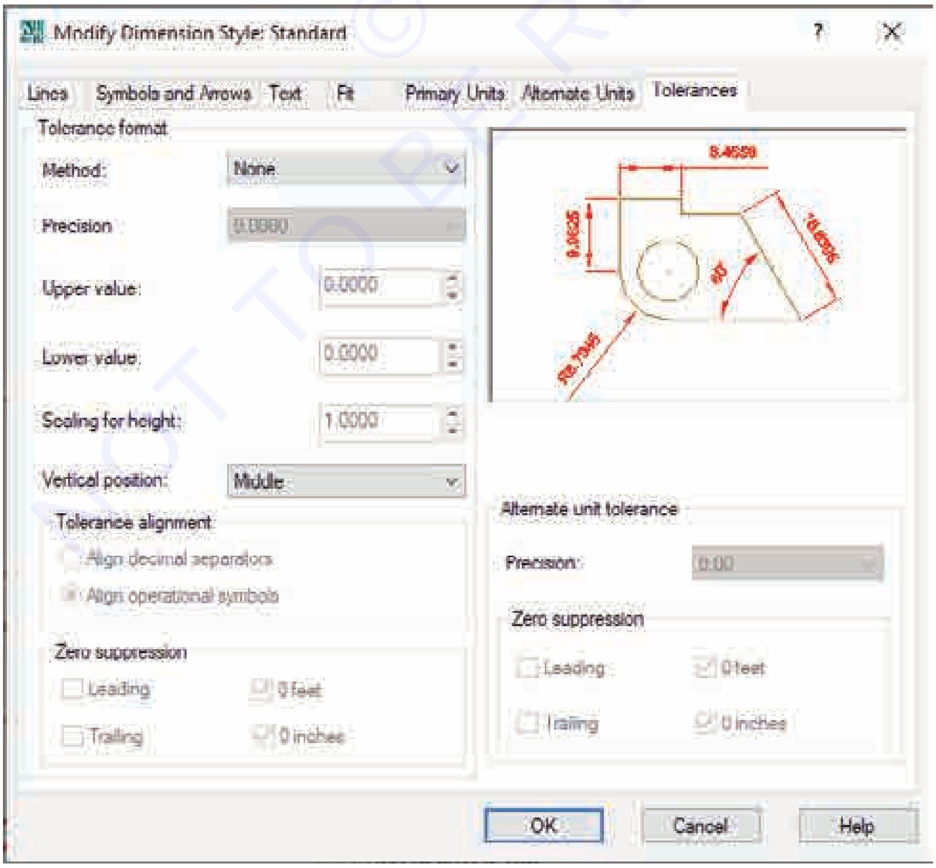
Precision: 0

Zero suppression

☐ Leading

☐ Trailing

Fig 9



Tolerance format

Method: None

Precision: 0.0000

Upper value: 0.0000

Lower value: 0.0000

Scaling for height: 1.0000

Vertical position: Middle

Tolerance alignment

○ Align decimal separators

● Align operational symbols

Zero suppression

☐ Leading ☑ 0 feet

☐ Trailing ☑ 0 inches

Alternate unit tolerance

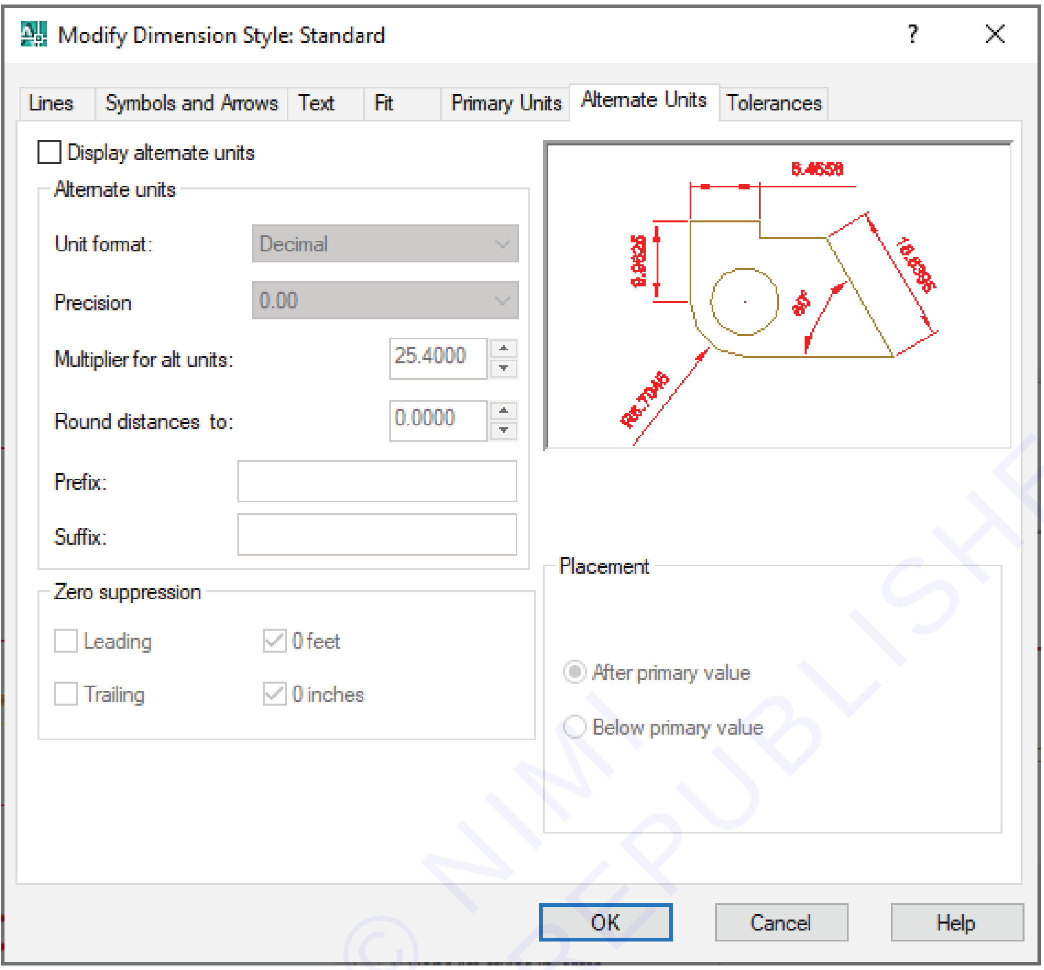
Precision: 0.00

Zero suppression

☐ Leading ☑ 0 feet

☐ Trailing ☑ 0 inches

Fig 10



☐ Display alternate units

Alternate units

Unit format: Decimal

Precision 0.00

Multiplier for alt units: 25.4000

Round distances to: 0.0000

Prefix:

Suffix:

Zero suppression

☐ Leading ☑ 0 feet

☐ Trailing ☑ 0 inches

Placement

● After primary value

○ Below primary value