SRM VALLIAMMAI ENGINEERING COLLEGE

(An Autonomous Institution)

SRM Nagar, Kattankulathur – 603 203

DEPARTMENT OF MECHANICAL ENGINEERING

LAB MANUAL

III SEMESTER

ME3339 Computer Aided Drafting Laboratory

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INDEX

STUDY OF DRAWING STANDARDS

EXPT NO: 1 DATE:

AIM:

To study about the basic fundamentals of the drawing standards used in machine drawing.

CODE OF PRACTICE FOR ENGINEERING DRAWING:

The representation of any matter by some sign or mark on the drawing is known as convention or Code. These conventions are specified by Bureau of Indian Standards (BIS) and are called BIS specifications.

ISO STANDARDS:

In the 21st century, growing international engineering and technological developments, technical collaboration between mutual countries, exchange and export of technologies and globalization leading to the establishment of various Multi-National Companies (MNCs) have necessitated the inter-nationalization of engineering drawing standards. INTERNATIONAL STANDARDS ORGANIZATION (I.S.O), Geneva, Switzerland, have formulated (1970) international standards for engineering drawing, for which our country has also given the approval and accordingly modified the Indian Standards IS 696:1972.

B.I.S:

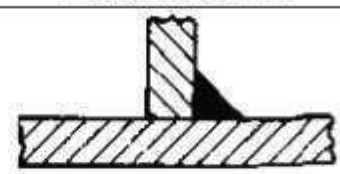
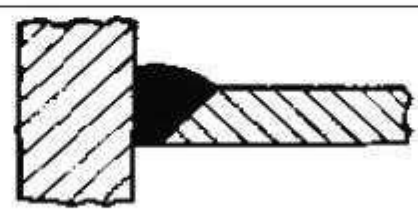
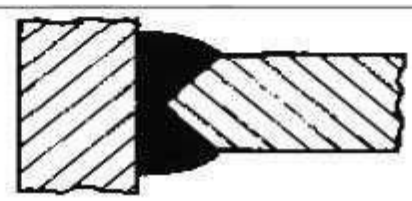
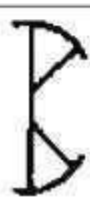
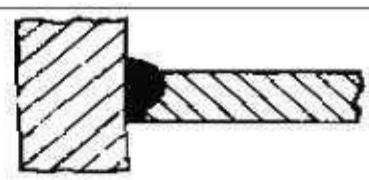
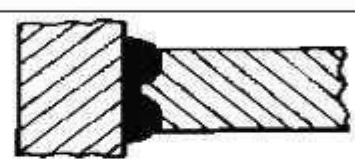
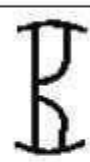
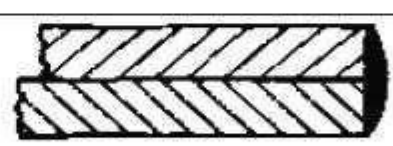
Indian Standards Institution (I.S.I) is renamed as Bureau of Indian Standards (B.I.S) in 1983.Engineering Drawing Committee (E.D.C) of B.I.S has adopted the I.S.O standards in Toto for the various topics given in the following Table.

Sl.No	BIS Codes	Topics
1)	IS: 9609 (Parts 0 & 1) – 2001	Technical product documentation – Lettering
2)	IS: 10711 – 2001	Technical products documentation – size and layout of drawing sheets
3)	IS : 10713 – 1983	Scales for use on Technical Drawing
4)	IS : 10714 – 1983	General Principles of Presentation
5)	IS: 10714 (Part 20) – 2001	Lines
6)	IS : 10715 – 1983	Presentation of threaded parts on technical drawing
7)	IS : 10716 – 1983	Rules for presentation of springs
8)	IS : 10717 – 1983	conventional representation of gears on technical drawing
9)	IS : 11663 – 1986	Conventional representation of common features
10)	IS : 11664 – 1986	Folding of drawing prints
11)	IS : 11665 – 1986	Technical drawing – Title blocks \
12)	IS : 11669 – 1986	Dimensioning on Technical Drawings
13)	IS : 11670 – 1986	Abbreviation's for use in Technical Drawing
14)	IS: 15021 (Parts 1-4) – 2001	Technical Drawings – Projection Methods
15)	IS: 7008 – 1988	Acme or Trapezoidal thread
16)	IS 4218 – 1976 IS 11698 – 1986	Metric screw threads
17)	IS: 2292 – 1974	Taper keys
18)	IS: 2293 – 1974	Gib – head keys
19)	IS: 2048 – 1983	Feather keys
20)	IS: 2294 – 1986	Woodruff keys
21)	IS: 2327 – 1991	Spline shafts
22)	IS 919 – 1963	Limits & Fits for engineering

RESULT:

Thus the basic fundamentals of the drawing standards used in machine drawing was studied.

BASIC WELD SYMBOLS

S.NO	FORM OF WELD	DIAGRAM	SYMBOL
1)	FILLET		
2)	SQUARE BUTT		
3)	SINGLE – V BUTT		
4)	DOUBLE – V BUTT		
5)	SINGLE – U BUTT		
6)	DOUBLE – U BUTT		
7)	SINGLE – BEVEL BUTT		
8)	DOUBLE – BEVEL BUTT		
9)	SINGLE – J BUTT		
10)	DOUBEL – J BUTT		
11)	EDGE		
12)	SPOT		

STUDY OF WELDING SYMBOLS

EXPT NO: 2 DATE:

AIM:

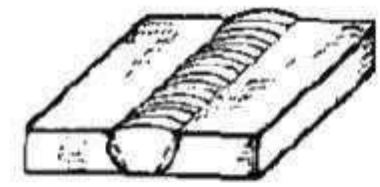
To study about the welding symbols.

WELDING:

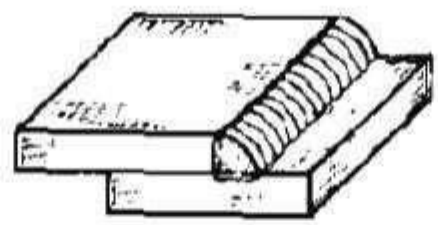
It is a technique of making permanent joint. It is the process of joining two parts of metal by fusing them together.

TYPES OF WELDING:

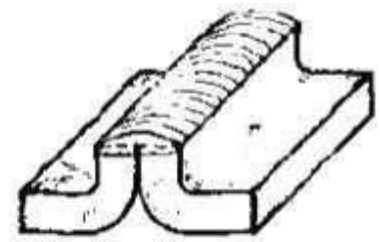
- Pressure welding or Forge welding
  - Forge welding
  - Thermit pressure welding
  - Pressure gas welding
  - Electric resistance welding
- Fusion welding
  - Gas welding
  - Electric arc welding
  - Thermit fusion welding
- Fusion pressure welding
  - Spot welding
  - Seam welding
  - Flash butt welding



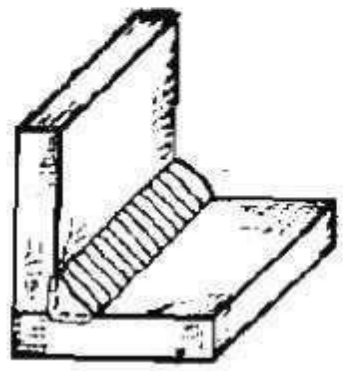
BUTT JOINT



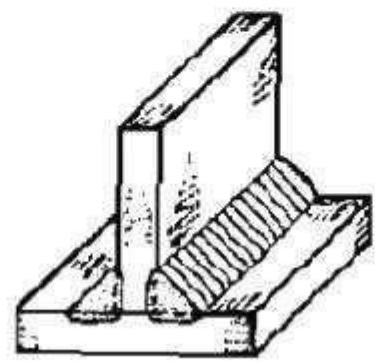
LAP JOINT



EDGE JOINT

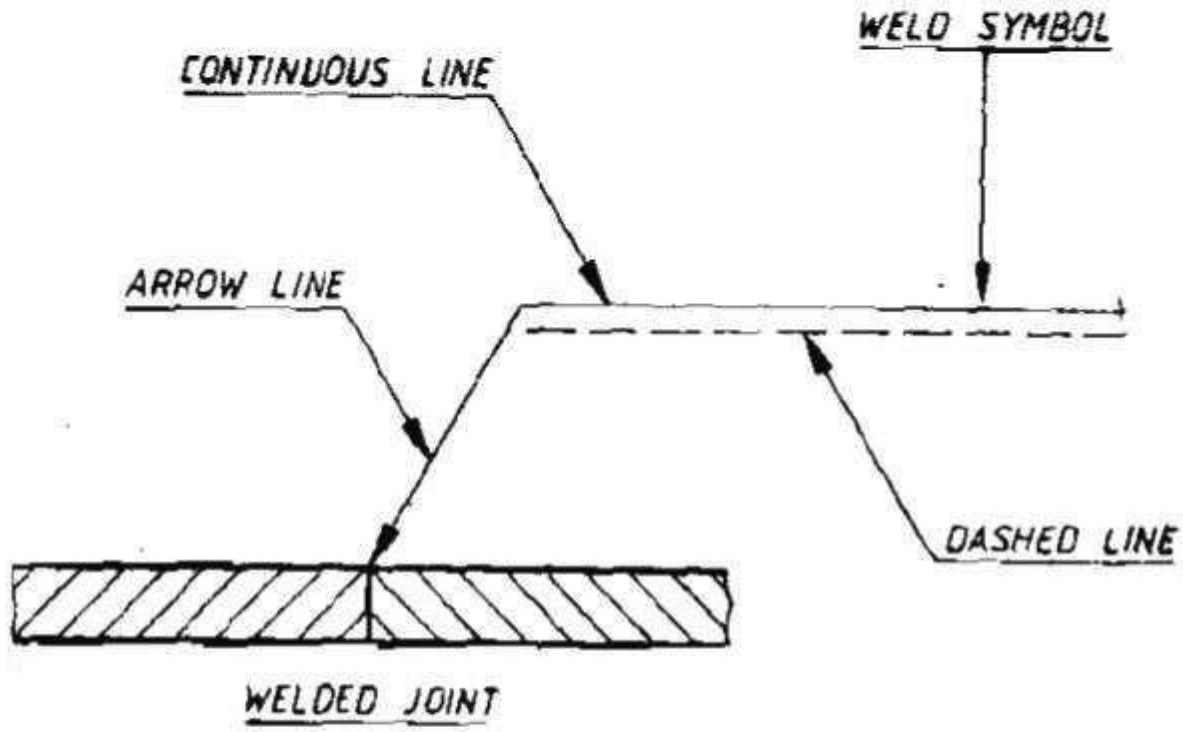


CORNER JOINT

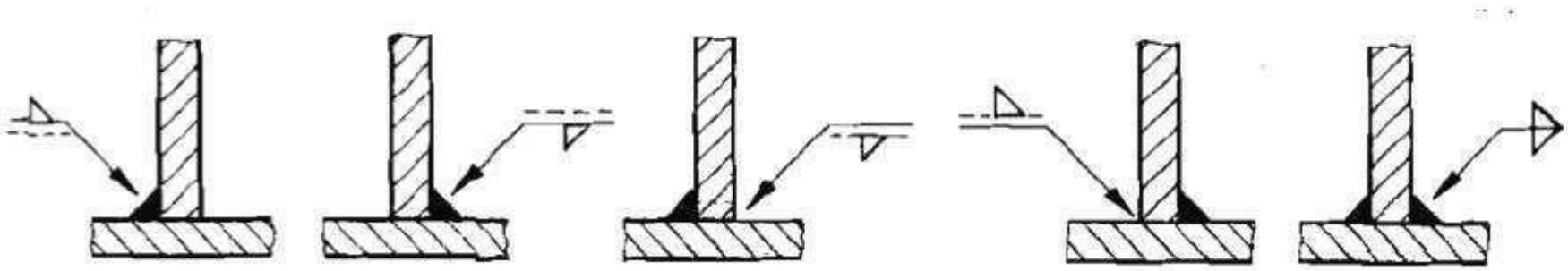


TEE JOINT

TYPES OF WELD JOINTS



WELD JOINT REPRESENTATION



ARROW SIDE

OTHERSIDE

SYMMETRICAL WELDS

TYPES OF WELD JOINTS:

- Butt joint
- Corner joint
- Tee joint
- Lap joint
- Edge joint

SYMBOLIC REPRESENTATION OF WELD:

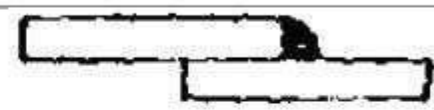
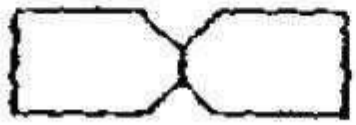
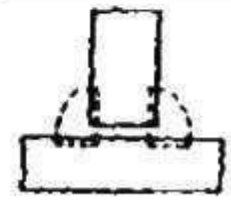
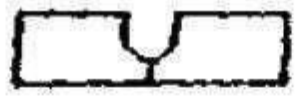
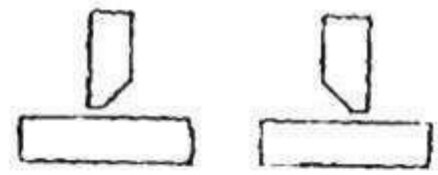
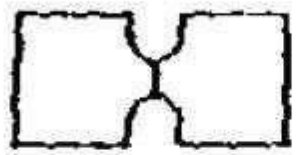
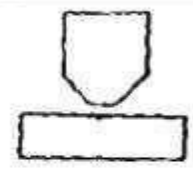
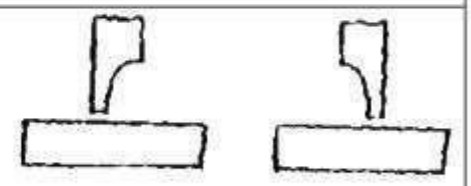
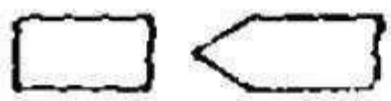
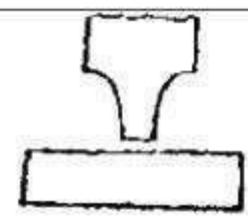
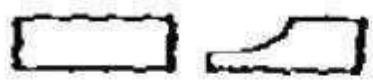
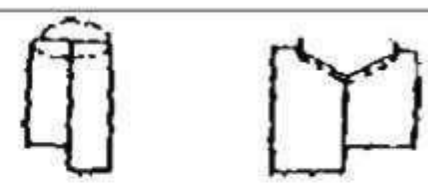
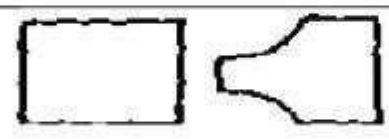
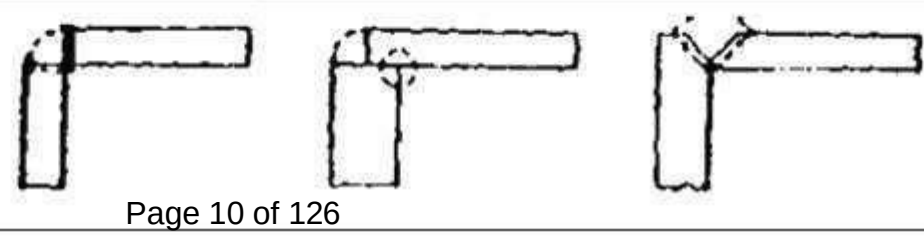
Welded joints are symbolically represented as shown in figure.

- An arrow line pointing to the joint where the weld to be made.
- A dual reference line consisting of two parallel lines, one continuous and one dashed.
- A basic symbol to specify the type of weld.
- A supplementary symbol to characterize the shape of the external surface on the weld.
- Dimension of the weld.

ARROW LINE AND REFERENCE LINE:

The position of the arrow line with respect to the weld is of no special significance. The side of the joint on which the arrow line is drawn is called "arrow side". The side of the joint remote to the arrow line is called "other side". The reference line has significance on the weld side. If the weld symbol is placed BELOW the reference line, the welding should be done in the "ARROW SIDE". If the weld symbol is placed ABOVE the reference line, the welding should be done in the "OTHER SIDE". If the weld symbol is placed both

EDGE PREPARATION

FORM OF WELD	DIAGRAM	FORM OF WELD	DIAGRAM
SQUARE BUTT		SINGLE FILLET	
SINGLE – V BUTT		DOUBLE FILLET	
DOUBLE – V BUTT		STRAIGHT BEVEL	
SINGLE – U BUTT		SINGLE BEVEL	
DOUBLE – U BUTT		DOUBLE BEVEL	
SINGLE – BEVEL BUTT		SINGLE J BEVEL	
DOUBLE – BEVEL BUTT		DOUBLE J BEVEL	
SINGLE – J BUTT		EDGE JOINT	
DOUBEL – J BUTT			
CORNER JOINT			

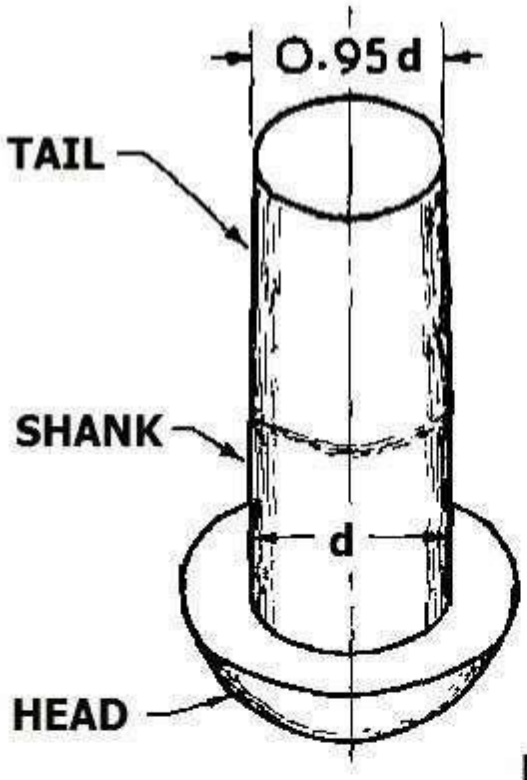
ABOVE and BELOW the reference line, the welding should be done in both the "ARROW AND OTHER SIDES".

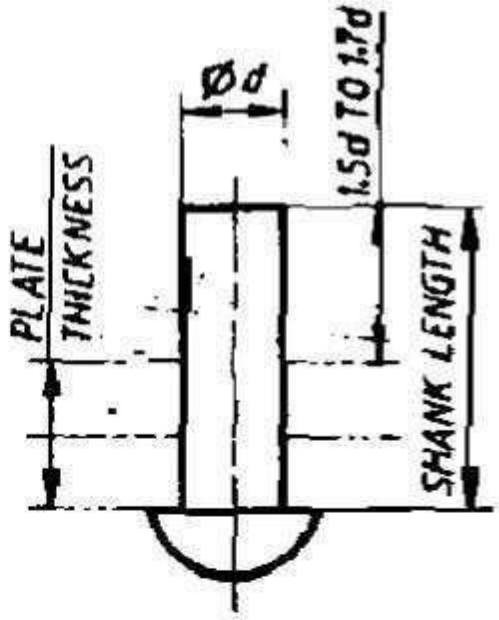
EDGE PREPARATION:

The preparation of the edges of the pieces to be welded depends upon the thickness of metal being welded Edge preparation is necessary when thickness increases so that heat would be able to penetrate the entire depth. This ensures formation of sound welds. The edge preparation is done by beveling the edges of the pieces after the rust, grease, oil or paint are completely removed from their surfaces.

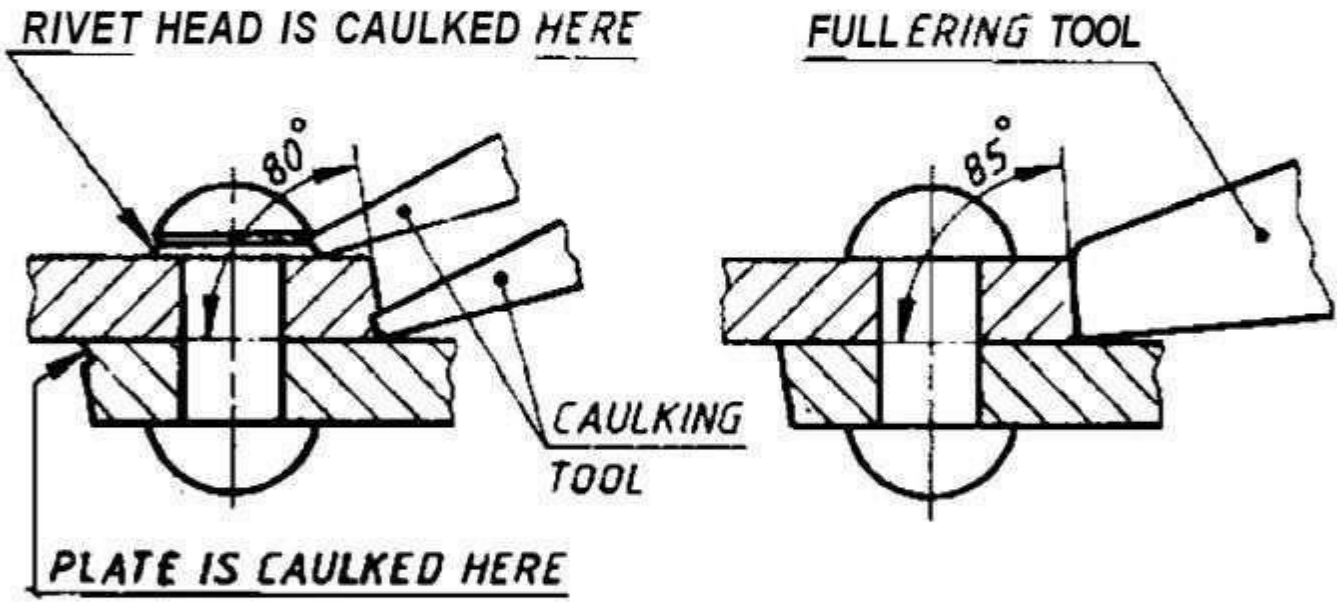
RESULT:

Thus the welding symbols were studied.

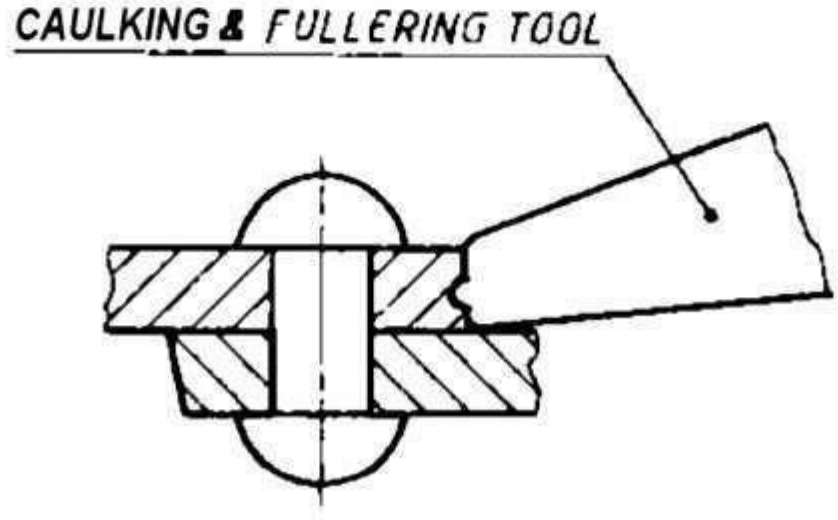




RIVET



Caulking Fullering



Combined Caulking & Fullering

CAULKING & FULLERING

STUDY OF RIVETED JOINTS

EXPT NO: 3 DATE:

AIM:

To study about the riveted joints.

RIVET:

Rivets are short cylindrical pieces, made of mild steel or non – ferrous materials such as copper, aluminum. A rivet consists of a head, cylindrical body or shank and a slightly tapered tail.

If l = Length of the shank of the rivet

d = Diameter of rivet

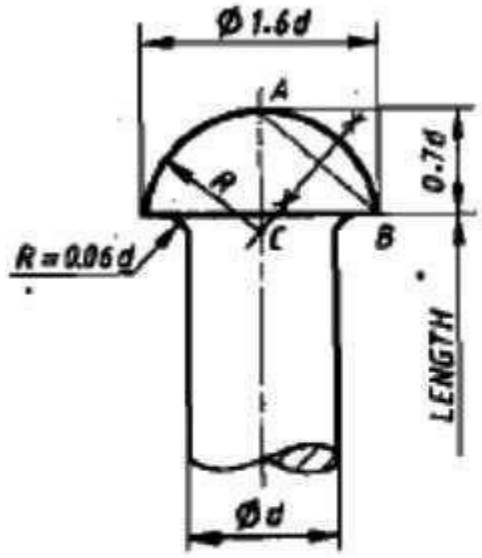
t = Thickness of each of the connecting plates

RIVETING:

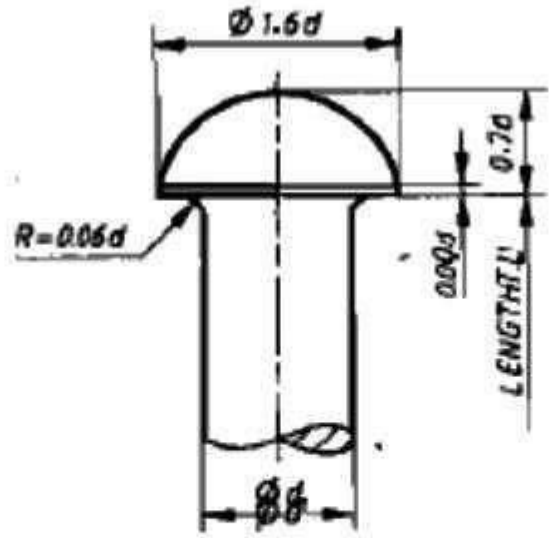
It is the process of forming a well – shaped concentric head from the projection portion of the shank end of the rivet inserted in the holes previously drilled in the plates to be fastened. The riveting process involves making of the holes in the plates and the formation of the rivet head

CAULKING & FULLERING:

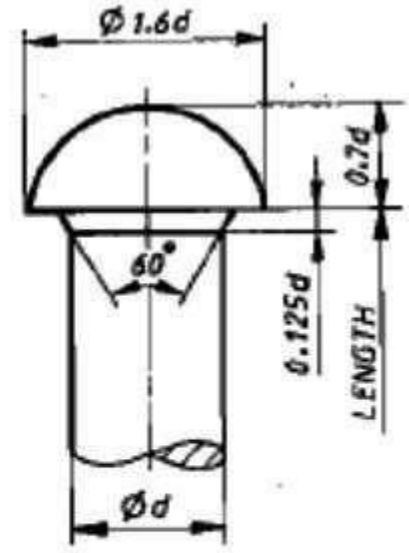
In boilers and hydraulic tanks containing pressurized fluids, there are possibilities of leak at the joints between the two overlapping plates, or between the bearing plate and underneath of the rivet head. Therefore the riveted joints of the boilers and tile hydraulic tanks have to be made leak – proof. This is accomplished by burring down or forcing down the edges of the rivet heads and the plates. This will force the edge of the plate or the rivet head to bite into the bearing plate which makes the joint leak – proof.



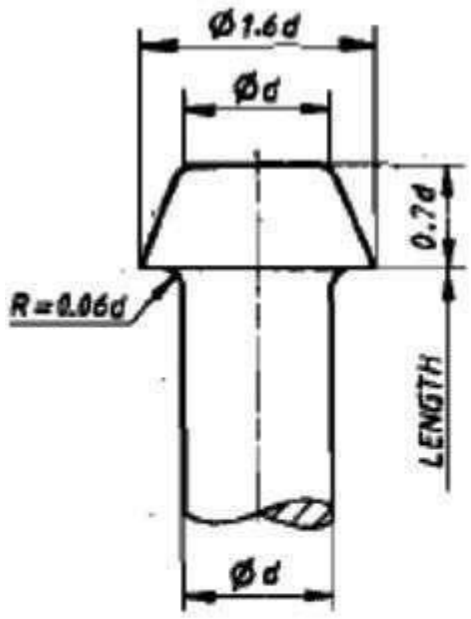
SNAP HEAD
for General Work



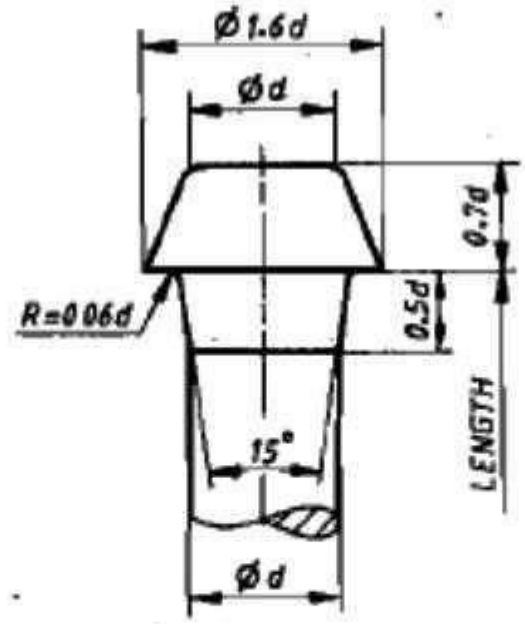
SNAP HEAD
for Ship Building



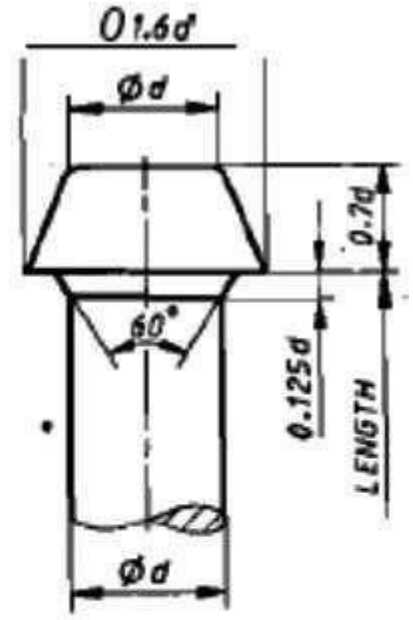
SNAP HEAD
for Boiler Work



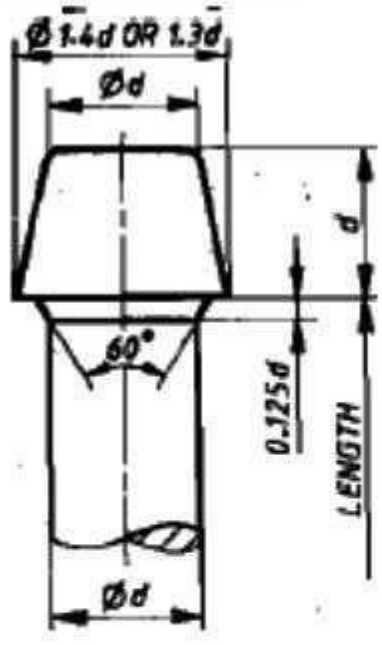
PAN HEAD
for General Work



PAN HEAD WITH TAPERED NECK
for General and Boiler Work



PAN HEAD-TYPE 1
for Boiler Work



PAN HEAD-TYPE 2
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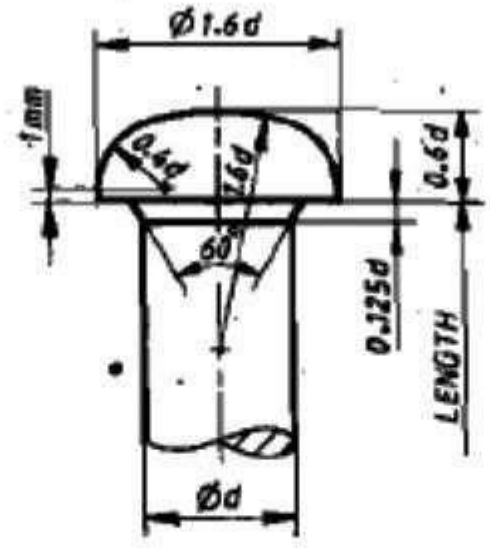
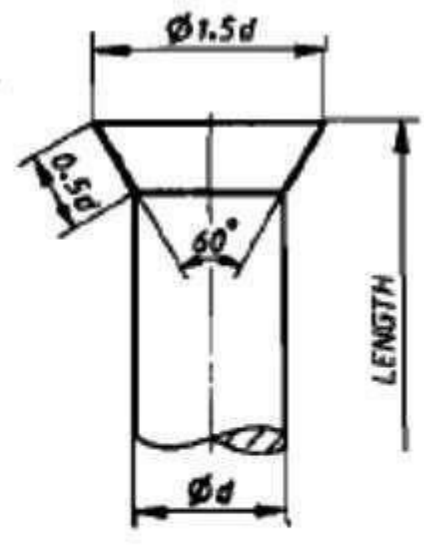


ELLIPSOID
for Boiler Work



FLAT COUNTERSUNK
for General Work

FORMS & PROPORTIONS OF RIVET HEADS

The burring down of the edges of the rivet head or the plates is accomplished by hammering down by a blunt chisel like tool of about 5mm thick and 50 mm in breadth with the edge ground at an angle of 80°, known as caulking tool as shown in Figure, along the edges of the plates and all-round the rivet head. This operation is called caulking.

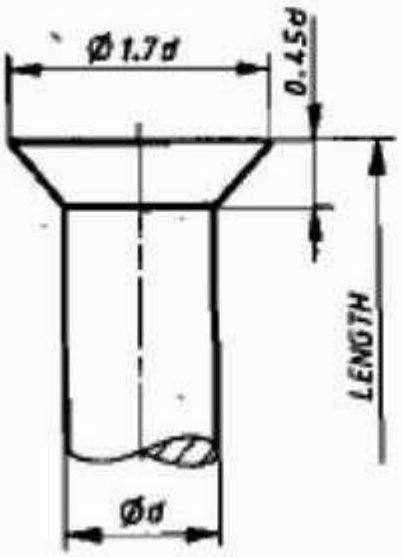
Another, more satisfactory method of burring down the edges of the plates is by the use of fullering tool, similar to the caulking tool but the thickness of the tool is equal to that of the plates as shown in the Figure. To facilitate the fullering operation the edges of the plates will be previously chamfered or bevelled to an angle of 80° before the joint is made. The fullering operation increases this angle to 80°. The Figure shows the use of a tool which combines both the caulking and fullering operations.

APPLICATIONS OF RIVETED JOINTS:

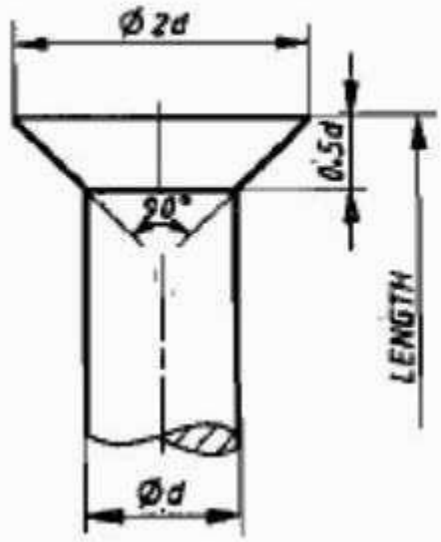
A riveted joint is a permanent type of fastener used to join the metal plates or rolled steel sections together. Riveted joints are extensively used in structural works such as bridges and roof trusses and in the construction of pressure vessels such as storage tanks, boilers, etc. Although welded joints are best suited to several of these applications than the riveted joints, however, riveted joints are ideal in cases where the joints will be subjected to pronounced vibrating loads. Riveted joints are also used when a non-metallic plate and a metallic plate are to be connected together. They are also used when the joints are not expected to be heated while joining as in welding, which may cause warping and tempering of the finished surfaces of the joints. The disadvantage of riveted joints are: (i) more metal is removed while making of the holes, which weakens the working cross sections along the line of the rivet holes, and (ii) weight of the rivets increases the weight of the riveted members.

TYPES OF RIVETS:

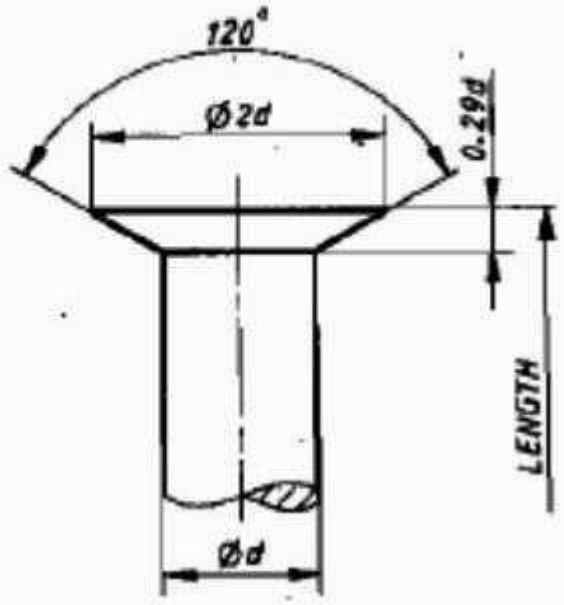
- Snap Head
- Pan Head



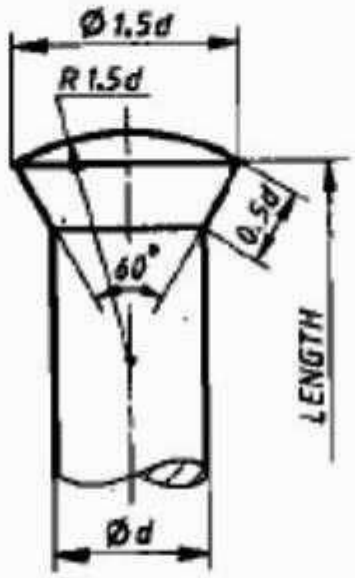
FLAT COUNTERSUNK
for Boiler Work



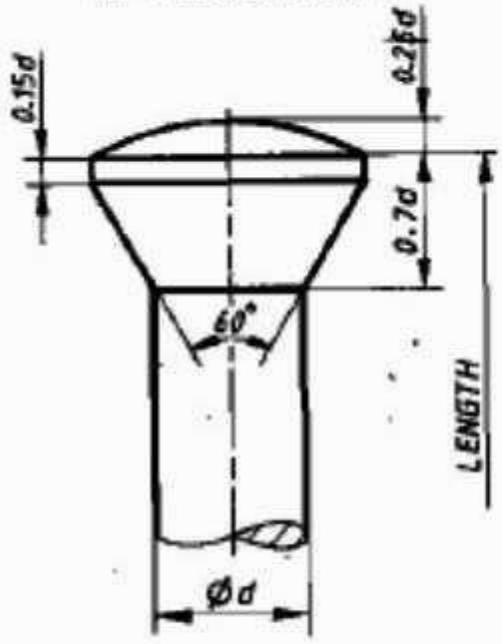
FLAT COUNTERSUNK HEAD 90°
for General Work



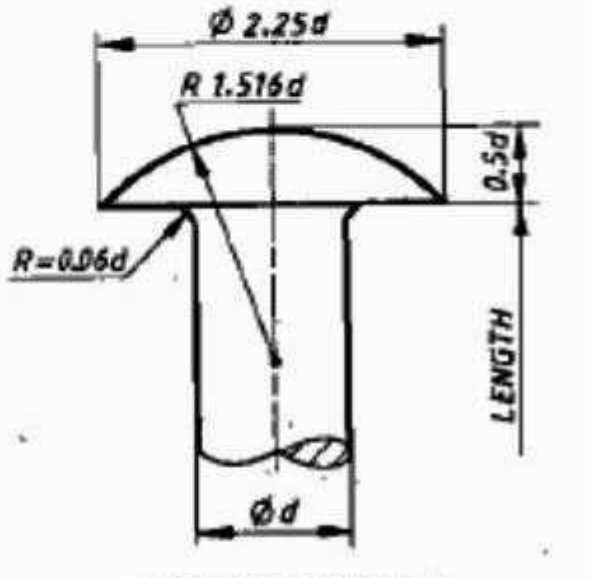
FLAT COUNTERSUNK
for General Work



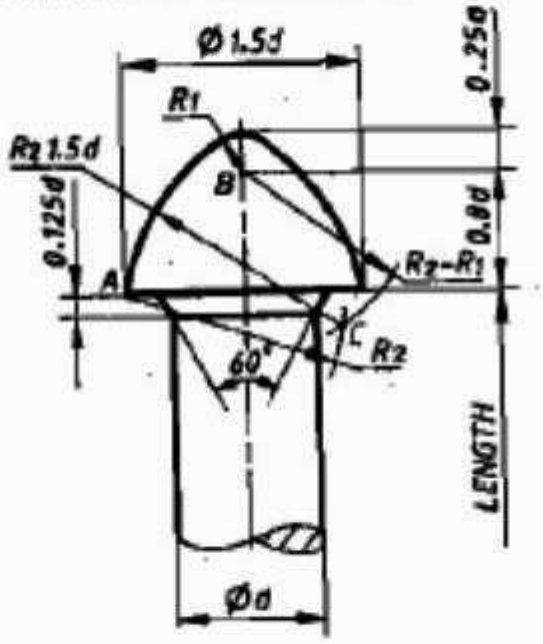
ROUNDED COUNTERSUNK
for General and Boiler Works



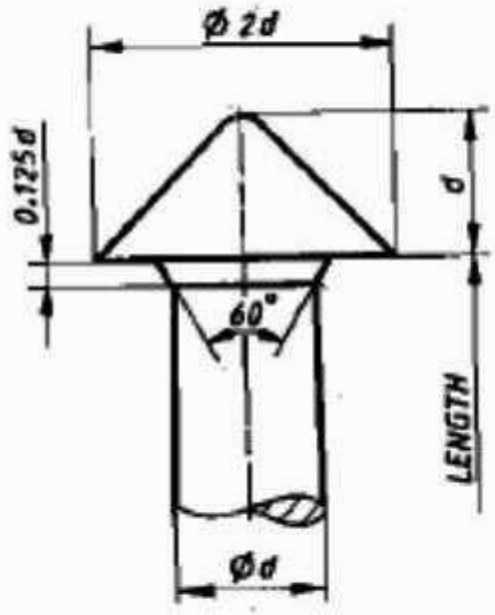
ROUNDED COUNTERSUNK
for Ship Building



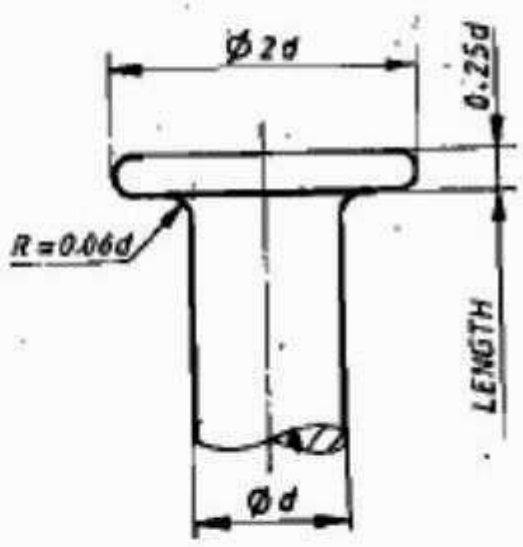
MUSHROOM HEAD
for General Work



CONICAL HEAD
for Boiler Work



STEEPLE HEAD
for Boiler Work



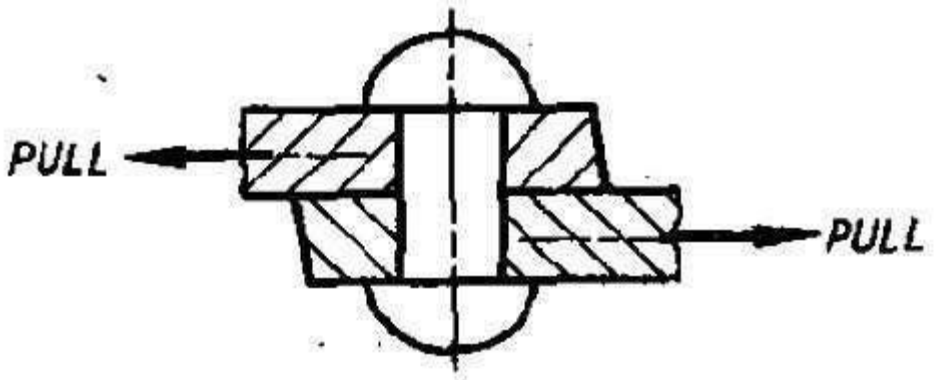
FLAT HEAD
for General Work

FORMS & PROPORTIONS OF RIVET HEADS

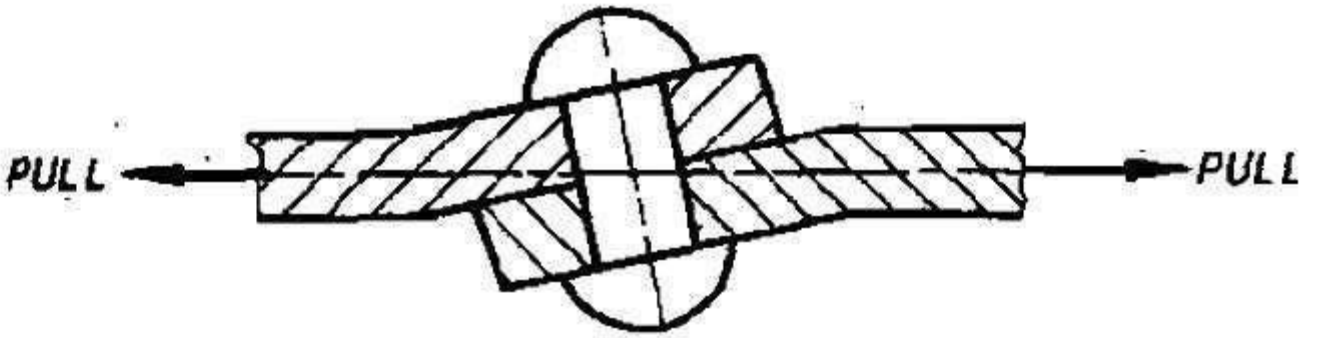
- ❖ Pan Head with Tapper Neck
- ❖ Counter Sunk Head
- ❖ Flat Counter Sunk Head
- ❖ Rounded Counter Sunk Head
- ❖ Mushroom Head
- ❖ Ellipsoidal head
- ❖ Conical head
- ❖ Steeple Head
- ❖ Flat Head

TYPES OF RIVETED JOINTS:

- ❖ Lap joint
- ❖ Butt joint

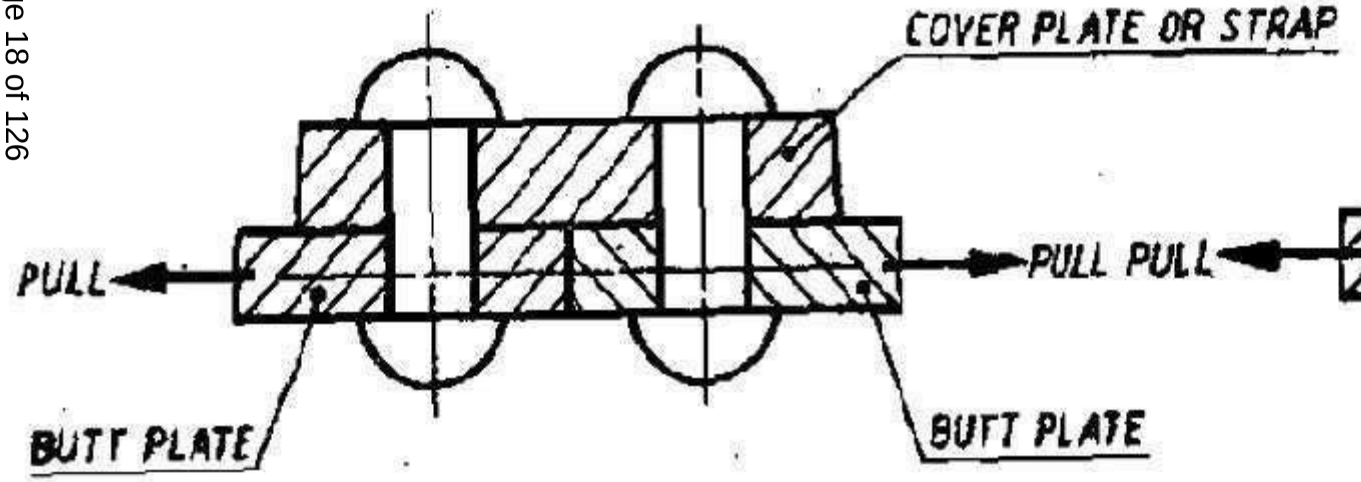


Before applying pull

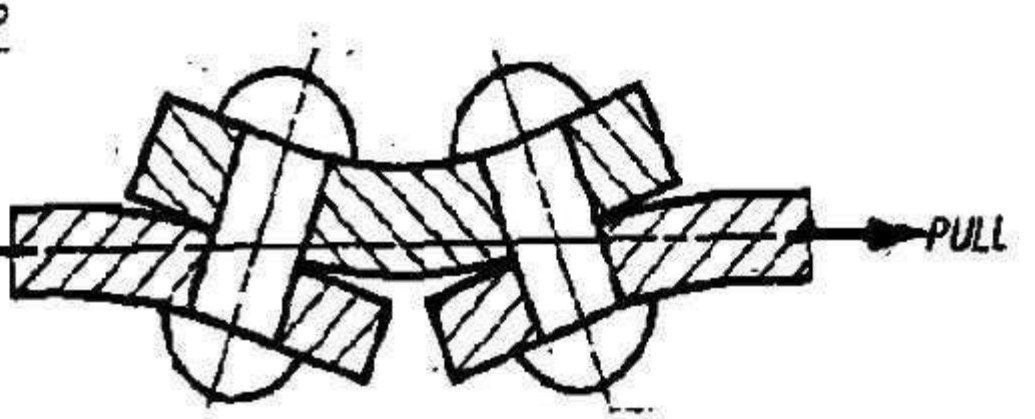


After applying pull

LAP JOINT



Before applying pull



After applying pull

BUTT JOINT

LAP JOINT:

The simplest way to connect two plates by riveting is to overlap them over a short distance along their edges and drill a row of holes though both of them in the overlapped portion and a joint are made by riveting as shown in figure. This type of joint is called a lap joint. When a lap joint is subjected to a pull as shown in Figure. Since the plates are not in the same plane, they develop a tendency to pull themselves along a plane resulting in their buckling as shown Figure which develops an axial pull in the rivets. Since the rivets are not designed to take up axial tensile forces, their heads may 'fly off".

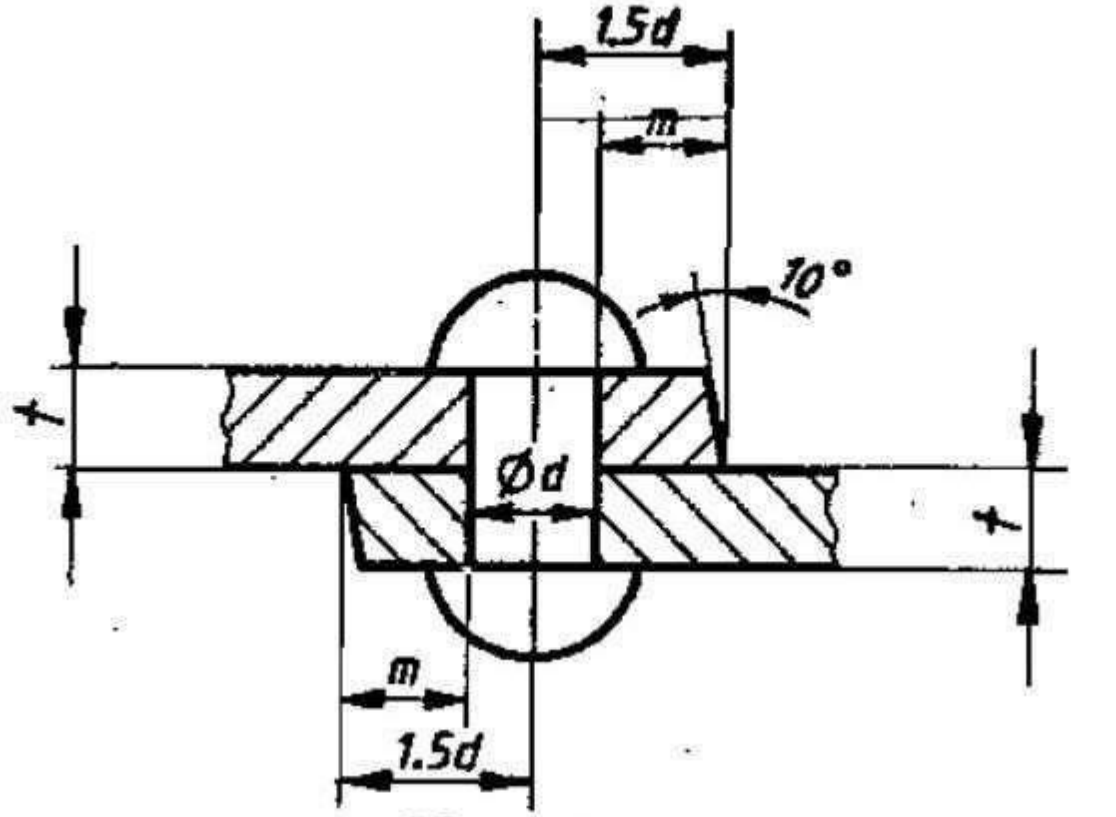
BUTT JOINT:

When the plates are intended to take heavy loads, the buckling action of the plates may be avoided by placing the connecting plates in alignment so as to butt each other and a cover plate called strap is placed over the joint between them and riveted to hold both the plates together as shown in Figure. This type of joint is called butt joint. In a butt joint, the load is transferred from one of the butt plates to the cover plate and then from the cover plate to the other butt plate. With this kind of arrangement, the rivets will be subjected to shear forces for which they are designed. Even the butt joint with a cover plate on only one side, called butt joint with single cover plate, may also be subjected to buckling as shown in Figure. Since this arrangement has cover plates on both the sides of the joint, it is called butt joint with double cover plates. Generally the butt joints are provided with double cover plates.

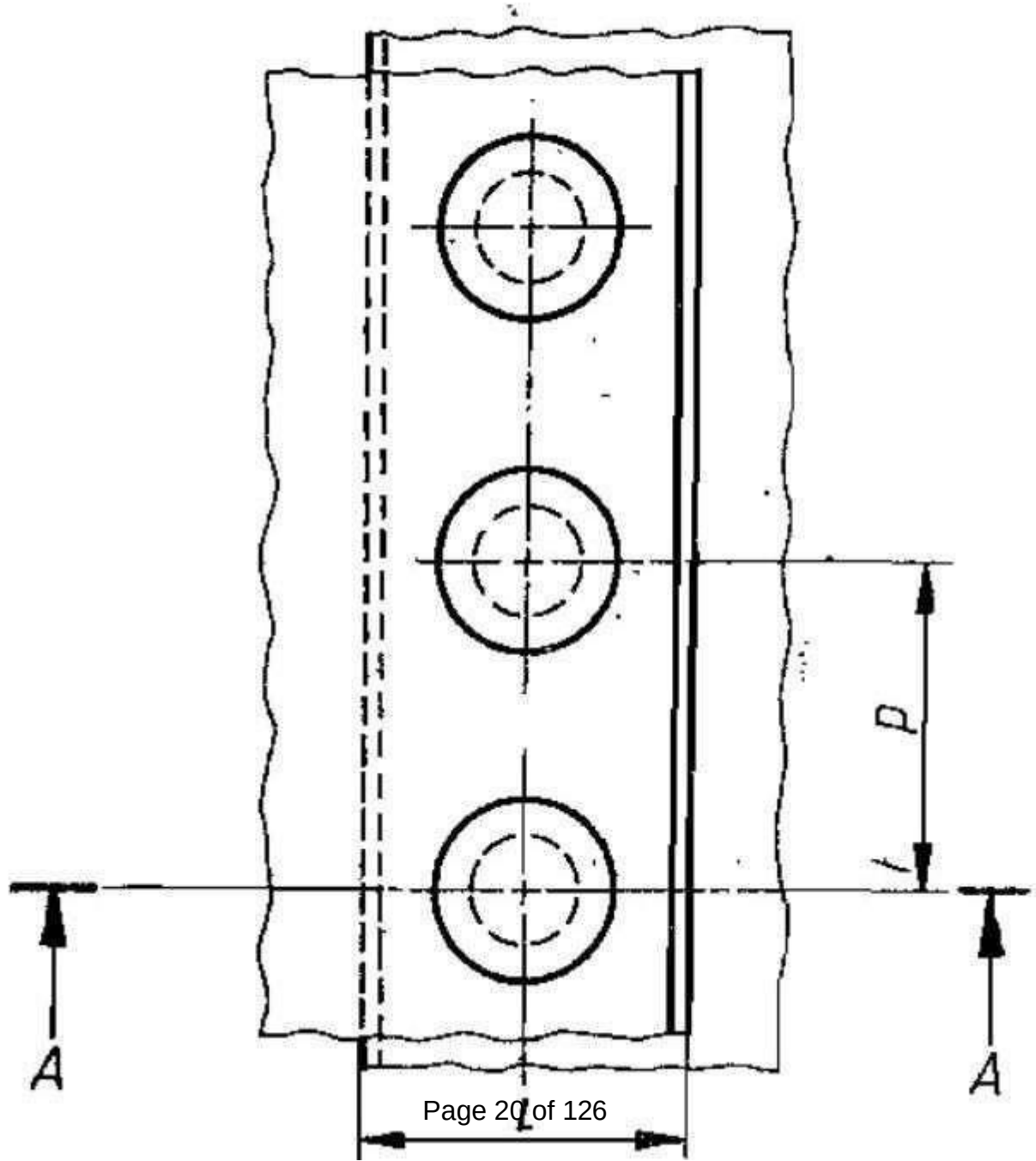
TERMINOLOGY:

The definitions of the different terms used in the riveted joints are given below.

Longitudinal Pitch – P: It is the distance from the centre of one rivet to the centre of the next rivet in the same row measured parallel to the caulking edge of the plate.



SECTION AA



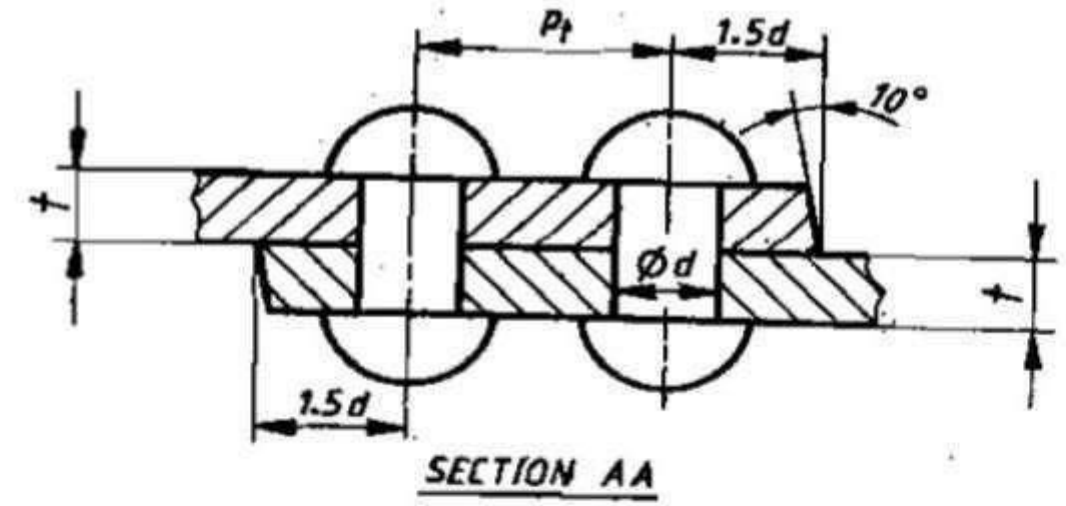
Margin – m: It is the distance between the edge of the plate and the nearest rivet hole. The margin is always taken equal to the rivet diameter. Since m=d, the distance between the centre of the rivet and the caulked or fullered edge of the plate will always be equal to 1.5d.

Transverse Pitch – P_t: It is the perpendicular distance between the rows of rivets. It is called as row pitch.

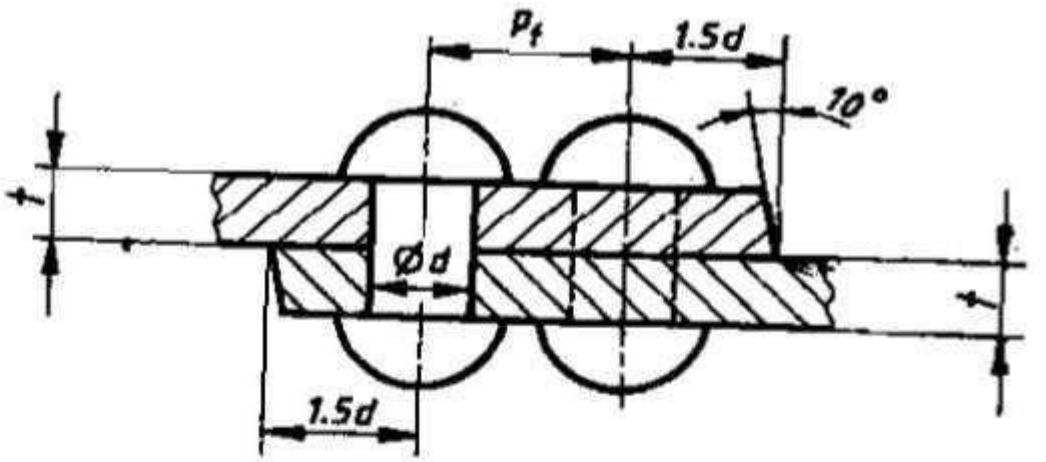
Diagonal Pitch – P_d: It is the distance from the centre of a rivet in a row to the centre of the next rivet in the adjoining row.

Back Pitch– P_b: It is the perpendicular distance between consecutives rows of rivets.

Nominal Diameter: It is the diameter of the shank of the rivet before riveting.



SECTION AA



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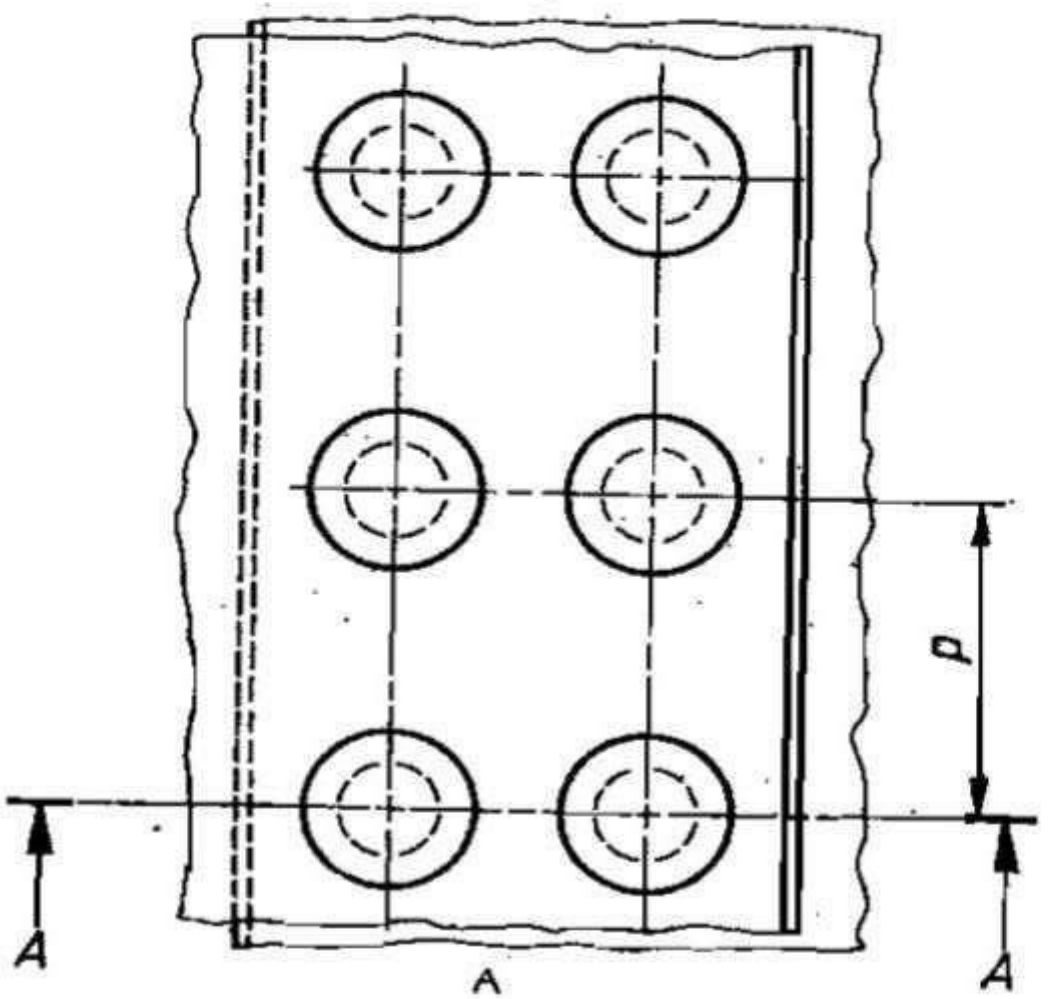
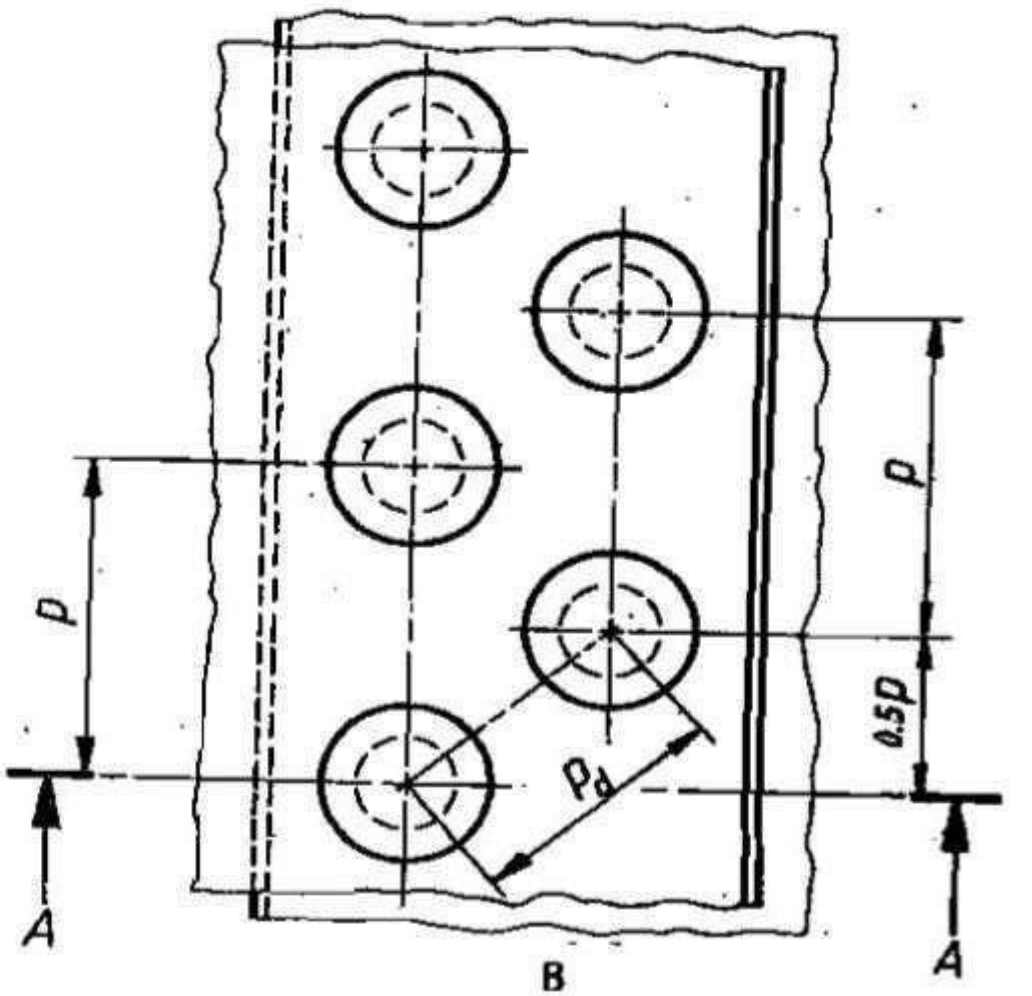


A
Chain Riveting

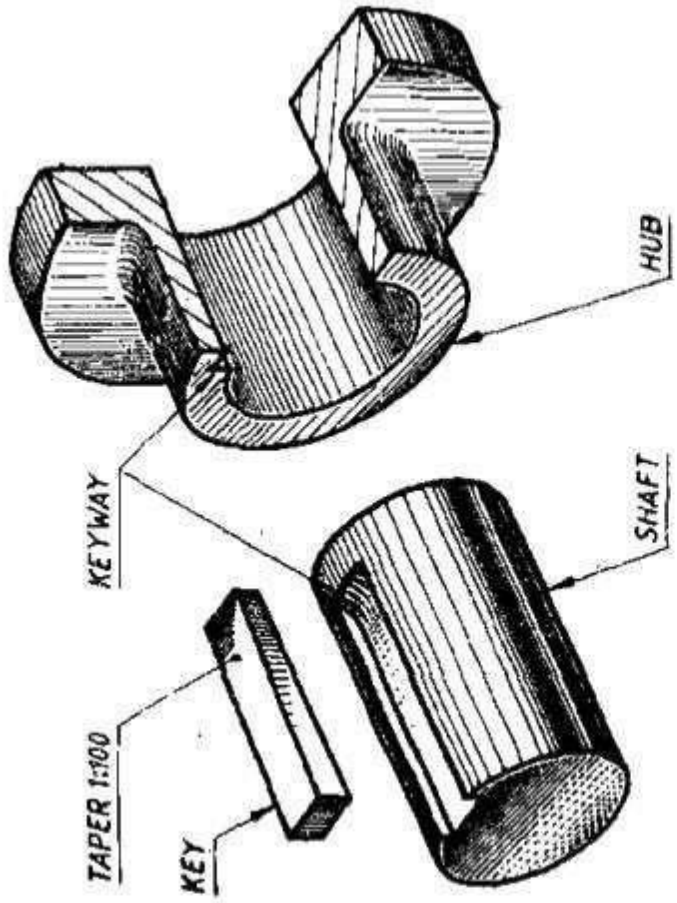


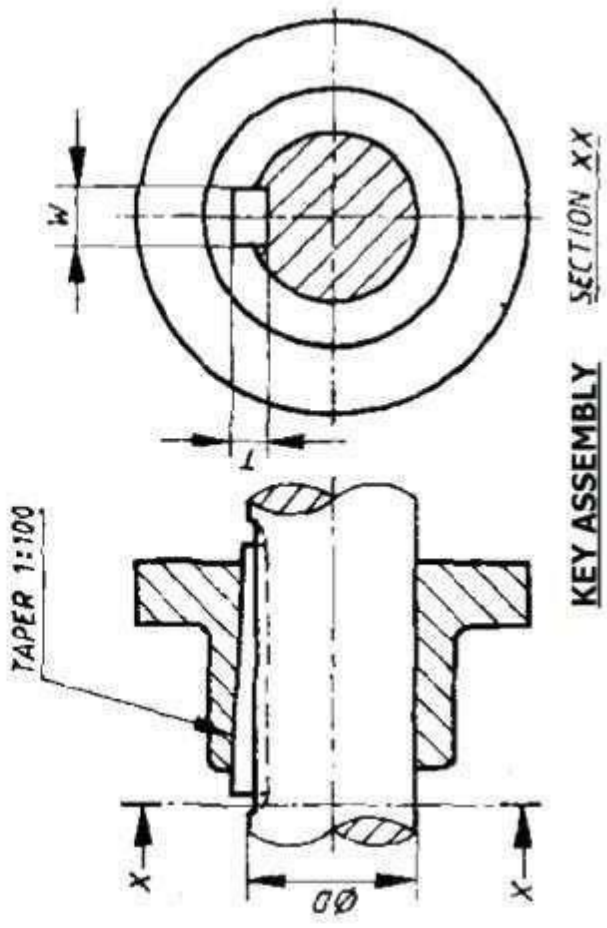
B
Zig-zag Riveting

Double Riveted Lap Joint

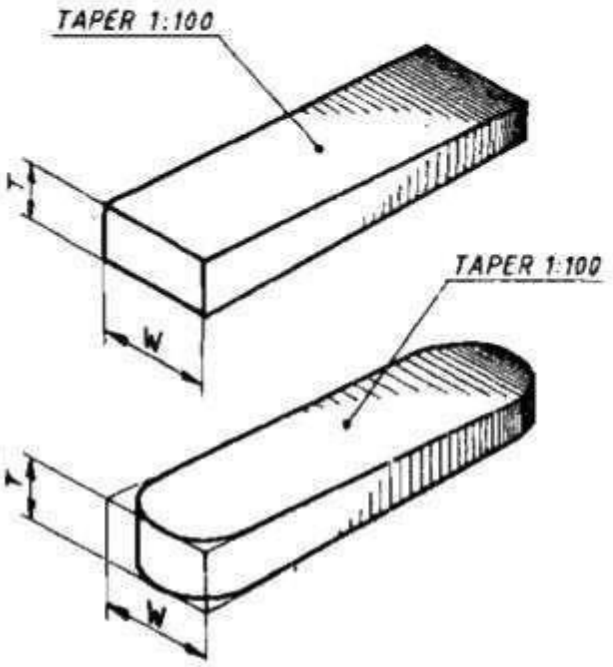
RESULT:

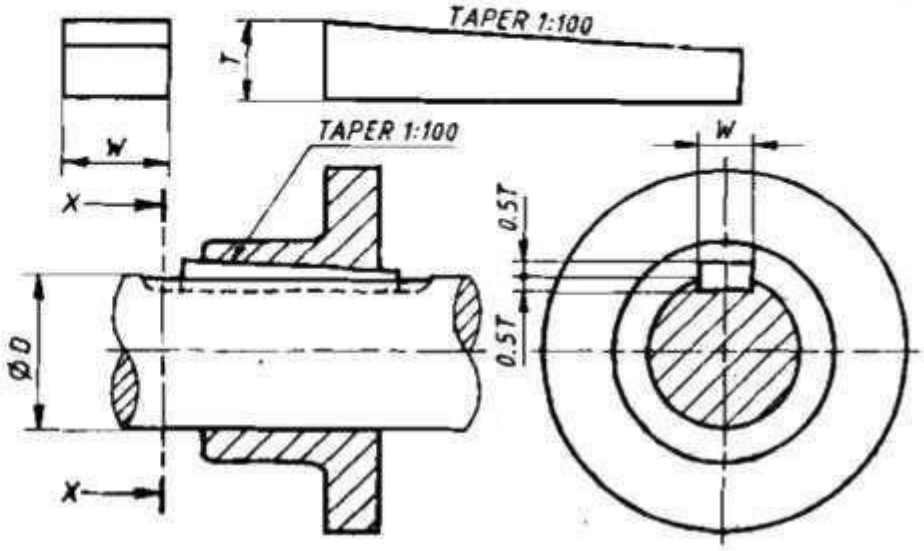
Thus the riveted joints were studied.





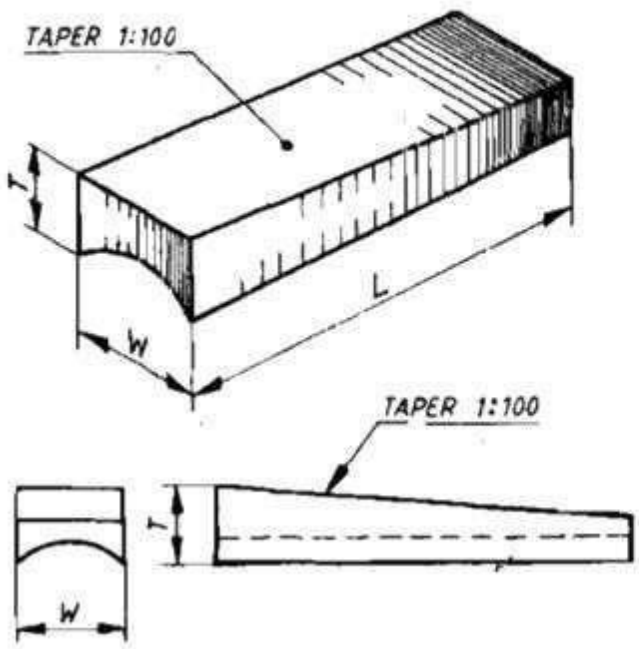
KEY ASSEMBLY

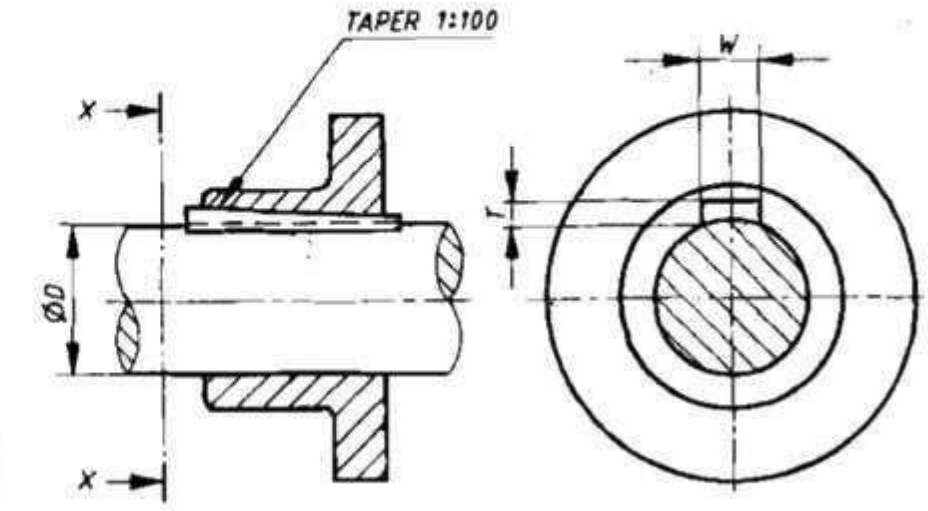




SUNK TAPER KEY

SECTION XX





HOLLOW SADDLE KEY

SECTION XX

STUDY OF KEYS

EXPT NO: 4 DATE:

AIM:

To study about different types of keys.

KEYS:

A key is a metal piece inserted in an axial direction between a shaft and a hub to prevent relative rotation keys mainly used to hold pulley gears wheels etc. The keywill be driven such that it sits partly into the shaft and partly into hub.

SUNK TAPER KEY:

A Sunk taper key shown in Figure is of rectangular or square cross section of uniform width having its bottom surface straight and top surface tapered. The key is driven between the shaft and hub with half of its thickness to fit in the flat key way made in the shaft and the other half having the tapered surface to fit in the tapered key way made in the hub. This type of key is generally used transmit heavy loads.

If D – Diameter of the shaft in mm

W – Width of the key = 0.25D + 2 mm

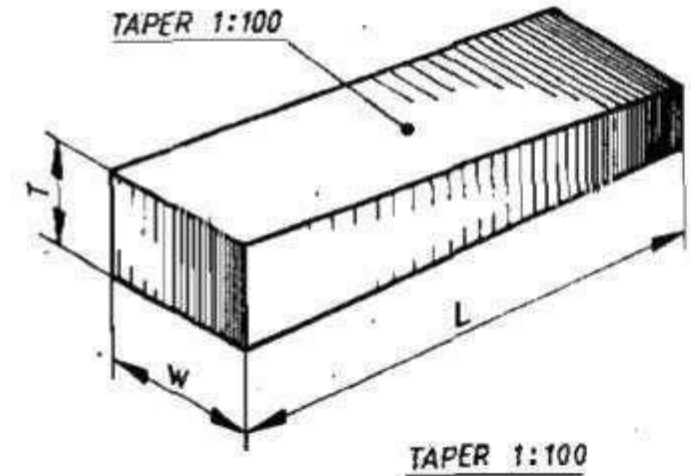
T – Nominal thickness = 0.66W

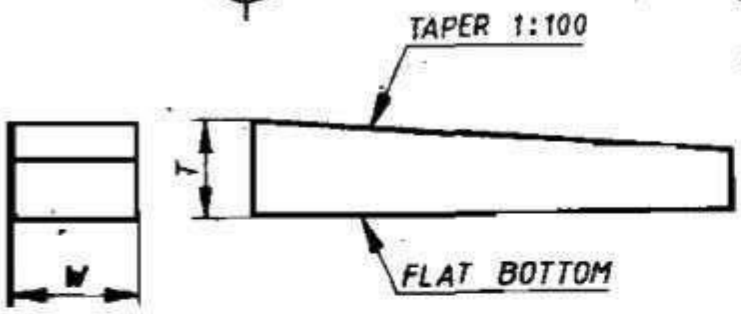
Standard taper = 1 : 100

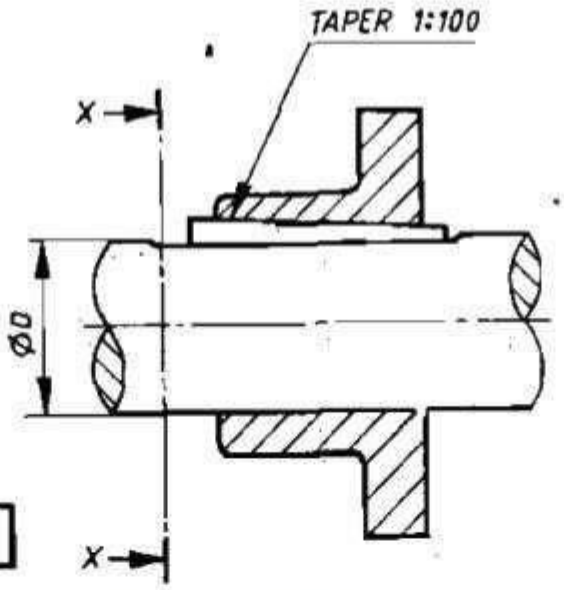
HOLLOW SADDLE KEY:

A hollow saddle key is of uniform width but in thickness its upper side is flat and the underside is hollow. Since the saddle key holds the shaft and the part mounted on it only by friction, it is not suitable for heavy loads. This key is used when there is frequent alternations in the position of the key on the shaft is expected.

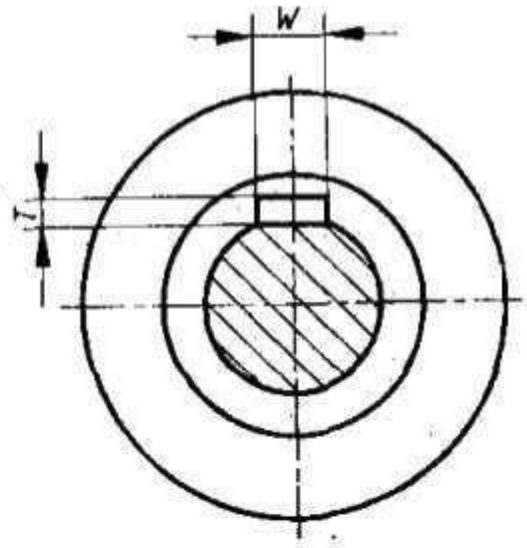
If D – Diameter of the shaft in mm







FLAT SADDLE KEY



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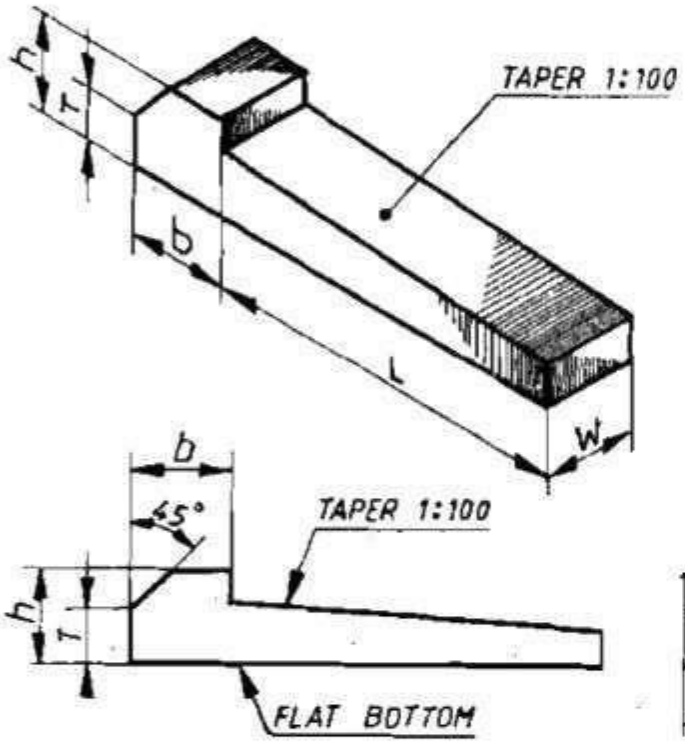
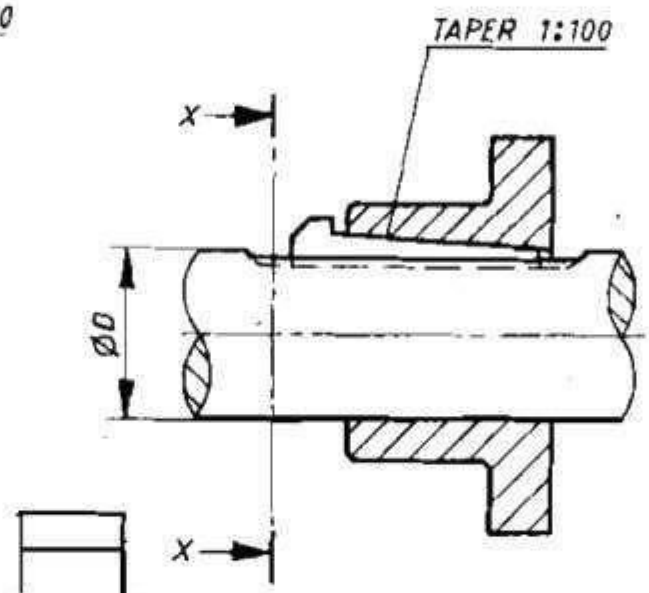
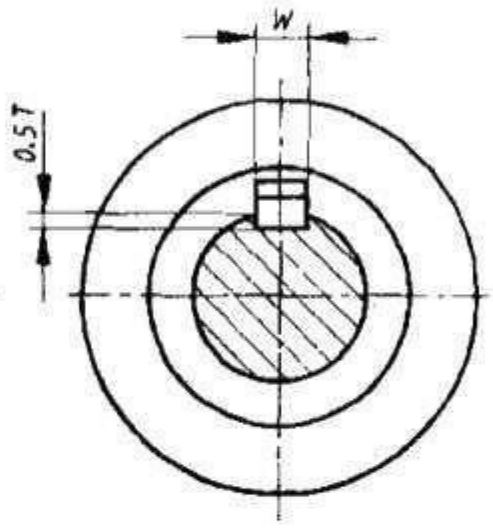
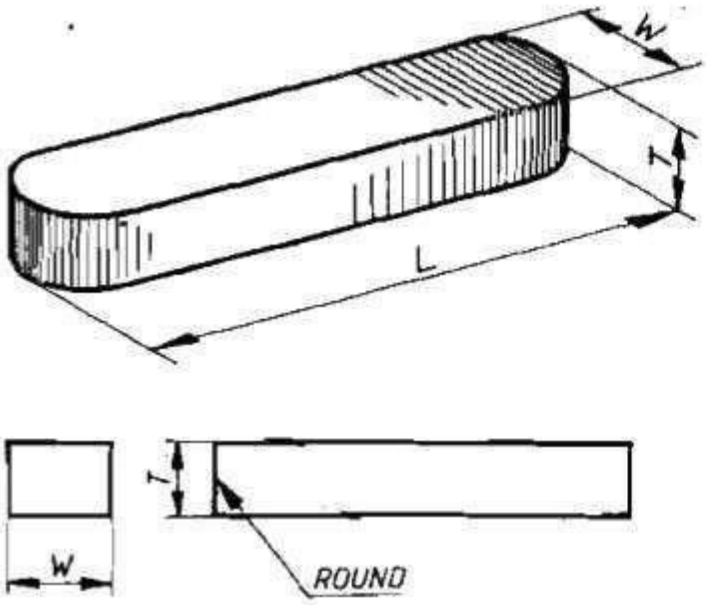


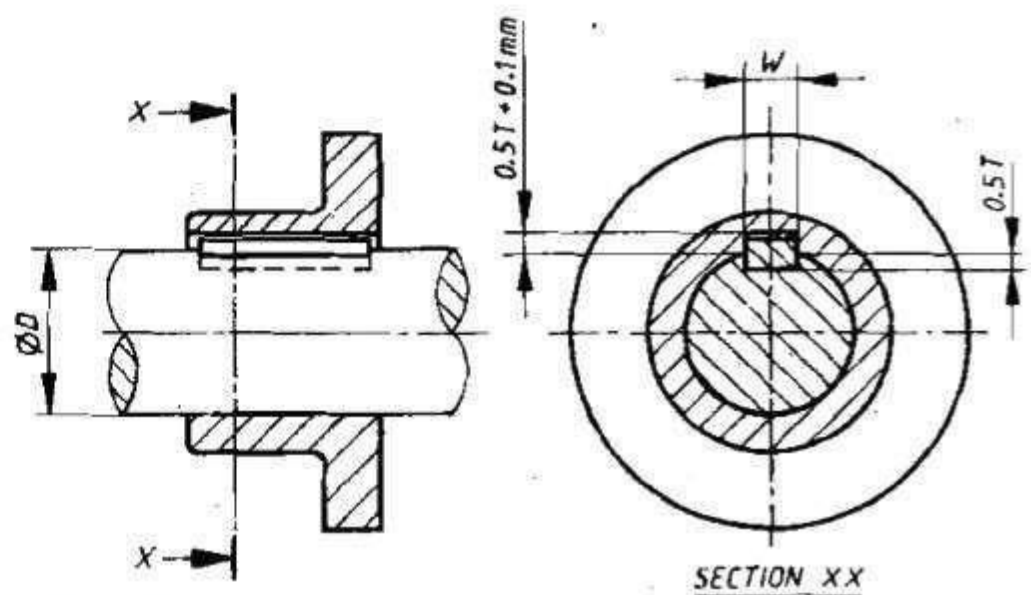


GIB - HEAD KEY



SECTION XX





FEATHER KEY OR PARALLEL KEY

SECTION XX

W – Width of the key = 0.25D + 2 mm

T – Nominal thickness = 0.33W

Standard taper = 1 : 100

FLAT SADDLE KEY:

A flat saddle key is similar to a hollow saddle key, except that it's underneath surface is flat. The key sits over the flat surface formed on the shaft and fits into the key way in the hub. When the shaft rotates, the key will be wedged between the flat surface on the shaft and the keyway in the hub, and thus holds them to rotate together. It is not used for heavy loads and not suitable for shafts which frequently changed their direction of rotation.

If D – Diameter of the shaft in mm

W – Width of the key = 0.25D + 2 mm

T – Nominal thickness = 0.33W

Standard taper = 1 : 100

GIB – HEAD KEY:

When a tapered sunk key is used, it can be removed by striking exposed thin end. If this end is not accessible, a head called gib is provided integral with the sunk taper key at its thicker end as shown figure. When a gib – head key is to be removed, a wedge is forced vertically in the gap between the head of the key and the vertical face of the hub.

If D – Diameter of the shaft in mm

W – Width of the key = 0.25D + 2 mm

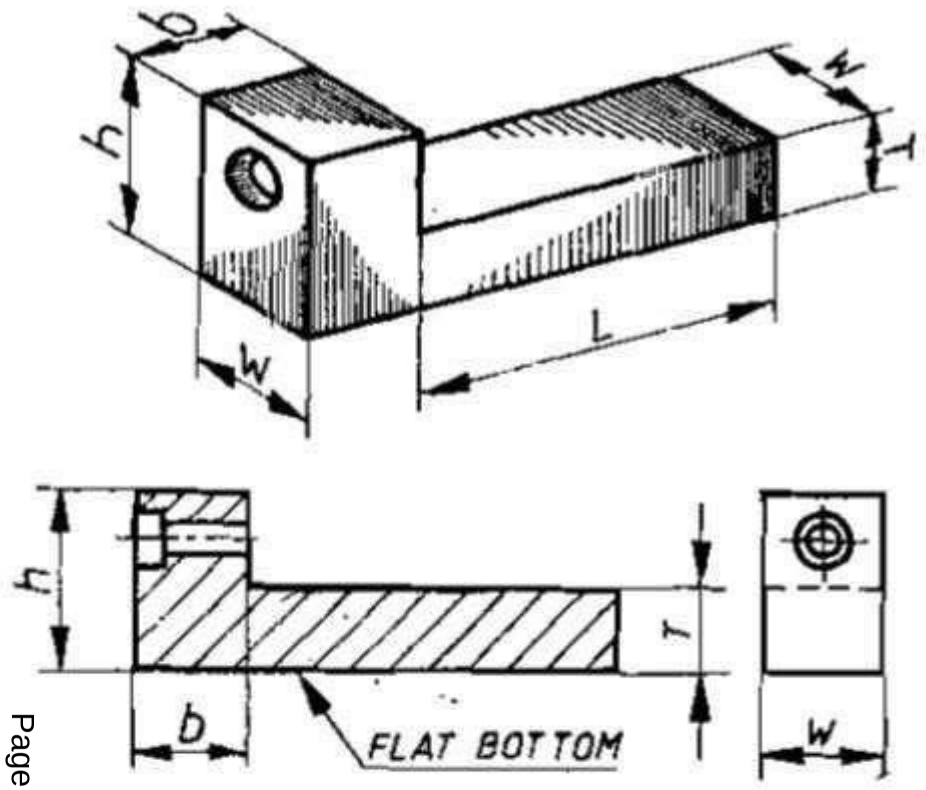
T – Nominal thickness = 0.66W

Height of Gib – head = 1.75T

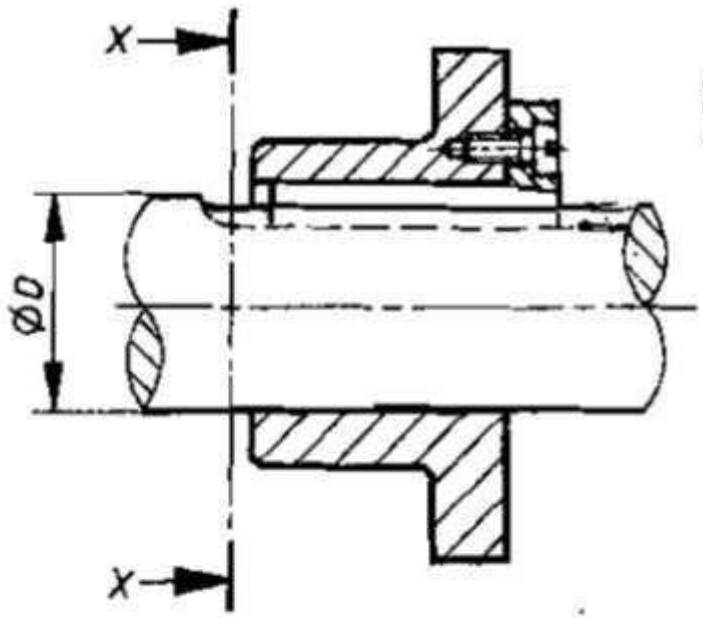
Width of Gib – head = 1.5T

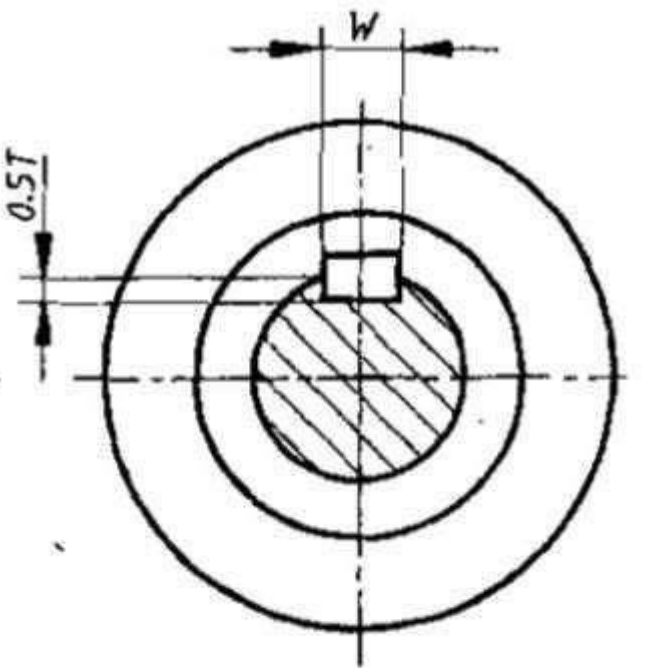
FEATHER KEY OR PARALLEL KEY:

A feather key or a parallel key permits an axial sliding movement for the wheel over a shaft when both of them are rotating together. A feather key is of rectangular or square cross section with uniform width and thickness as shown in figure. The ends of the feather key are



SINGLE HEAD KEY





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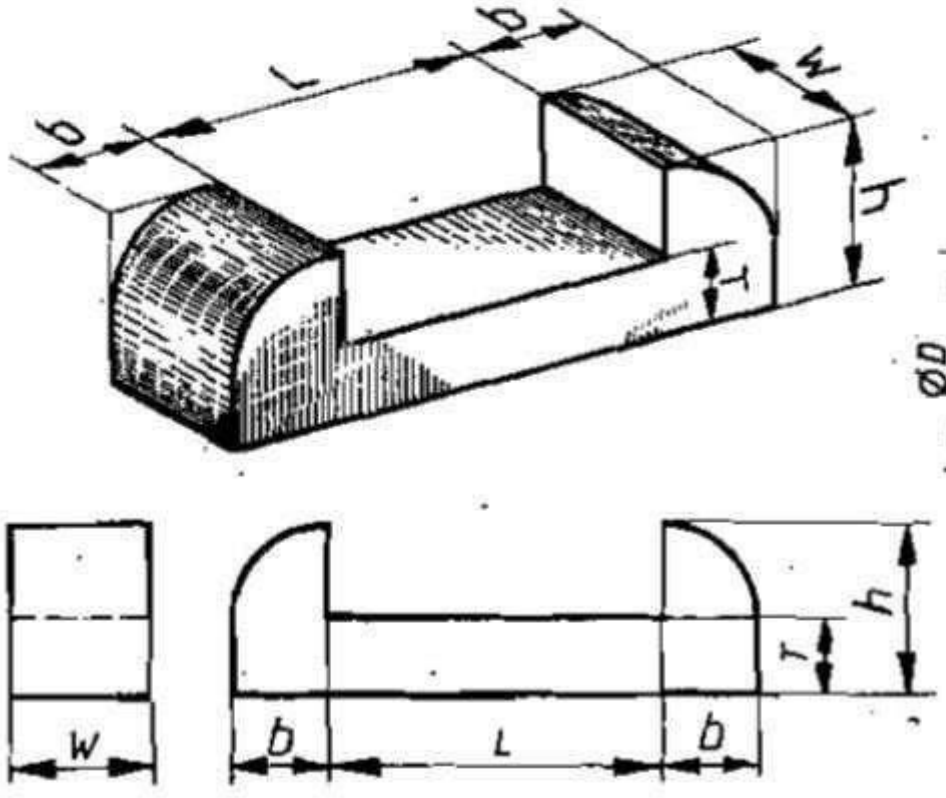
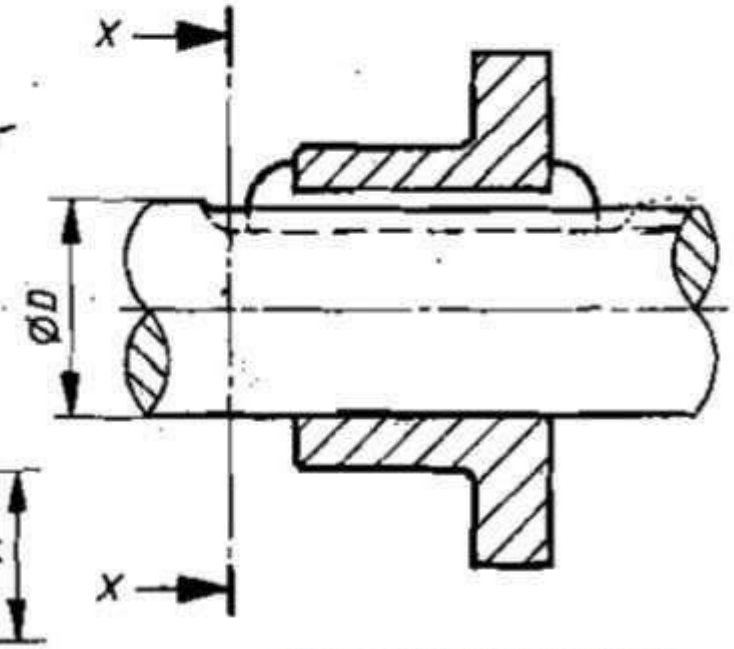
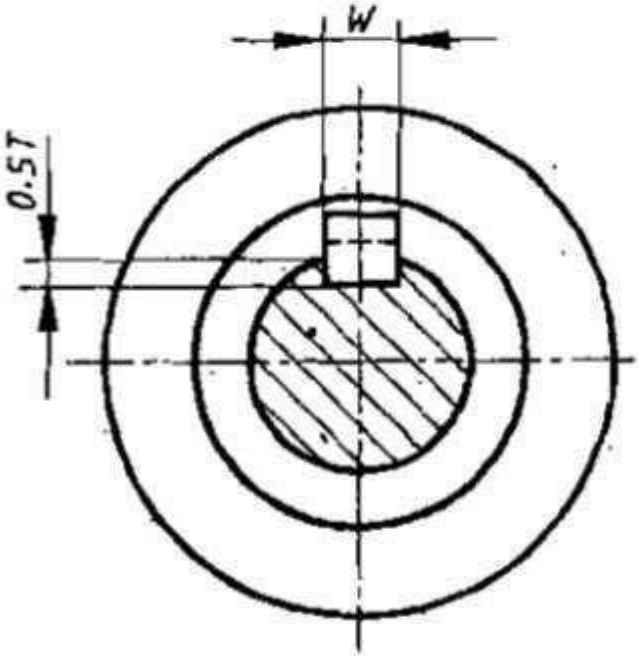


DOUBLE HEAD KEY





SECTION XX

usually rounded and the key will be sunk into the shaft for half of its thickness so that it fits snugly into the key way recess it with press fit.

If D – Diameter of the shaft in mm

W – Width of the key = 0.25D + 2 mm

T – Nominal thickness = 0.66W

SINGLE HEAD KEY:

A single head key is also a feather type key provided with a gib head at one of its ends as shown in figure. The key is connected to the hub by a screw. The key is sliding fot in the shaft. The proportions of the key are as follows

If D – Diameter of the shaft in mm

W – Width of the key = 0.25D + 2 mm

T – Nominal thickness = 0.66W

Height of head = 1.75T

Width of head = 1.5T

DOUBLE HEAD KEY:

A double head key is also a feather type of key having integral gib head as shown in figure. It fits tight in the hub and slides along with it in the key way in the shaft. The proportions of the key are as follows

If D – Diameter of the shaft in mm

W – Width of the key = 0.25D + 2 mm

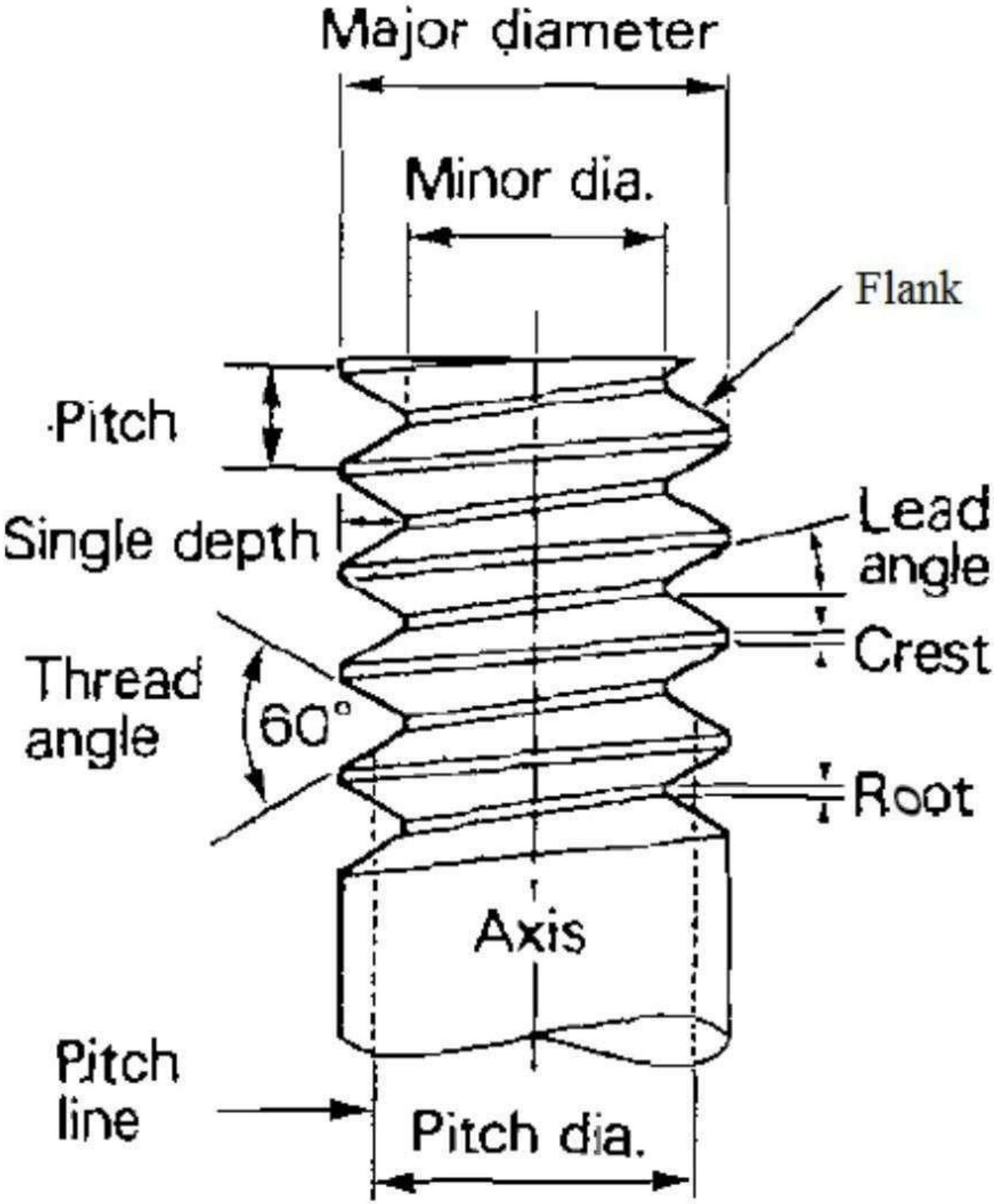
T – Nominal thickness = 0.66W

Height of head = 1.75T

Width of Gib = 1.5T

RESULT:

Thus the different types of keys were studied.



PARTS OF SCREW THREAD

STUDY OF SCREW THREADS AND THREADED FASTENERS

EXPT NO: 5 DATE:

AIM:

To study about screw threads and fasteners.

SCREW AND SCREW THREAD:

A screw is a cylindrical or a conical rod with a helical groove cut on it. Its function is to transform the input motion of rotation into output motion of translation.

A screw thread is a continuous helical ridge formed by cutting a helical groove on a cylindrical or conical shank.

SCREW THREAD TERMINOLOGY

External thread – It is a continuous helical ridge on the external surface of the cylinder.

Internal thread – It is a thread on the internal surface of a cylinder. The thread on the surface of a hole of a nut is an internal thread.

Right hand thread – If a threaded element viewed axially, a point moving clockwise along the thread moves away from the observer the thread is a right hand thread.

Left hand thread – If a threaded element viewed axially, a point moving anticlockwise along the thread moves away from the observer the thread is a left hand thread.

Depth of the thread – It is the distance between the crest and the root of the thread which is measured normal to the axis on an axial plane.

Angle of the thread – It is the angle includes between the sides of the two adjacent threads and is measured on an axial plane.

Thread angle – The distance between the flanks, measured in an axial plane, is known as thread angle.

Nominal Diameter – It is the diameter of the cylindrical rod on which the threads are cut. This diameter specifies the size of the screw.

Major Diameter – It is the diameter of an imaginary coaxial cylinder which bounds the crests of an external thread or the roots of an internal thread. D and d denote the major diameters of the internal and external threads respectively.

Minor, or Core, or Root Diameter – It is the diameter of an imaginary coaxial cylinder which bounds the root so fan external thread, or the crests of an internal thread. D_I and d_3, denote the minor diameters of the internal and external thread respectively.

Pitch Diameter – It is the diameter of the thread at which an imaginary coaxial cylinder that can be passed so as to cut the thread so that the width of the cut thread will be equal to the width of the groove. $D_2$and d_2, denote the pitch diameters of internal and external threads respectively.

Root – It is the bottom portion of the surface of a thread, either flat or rounded which joins the sides of the adjacent threads.

Crest – It is the top portion of the surface of a thread, either flat or rounded which joins the sides of the same thread.

Flank or Side – It is the surface of a thread that connects the crest with the root and also it offers the surface contact with its counterpart.

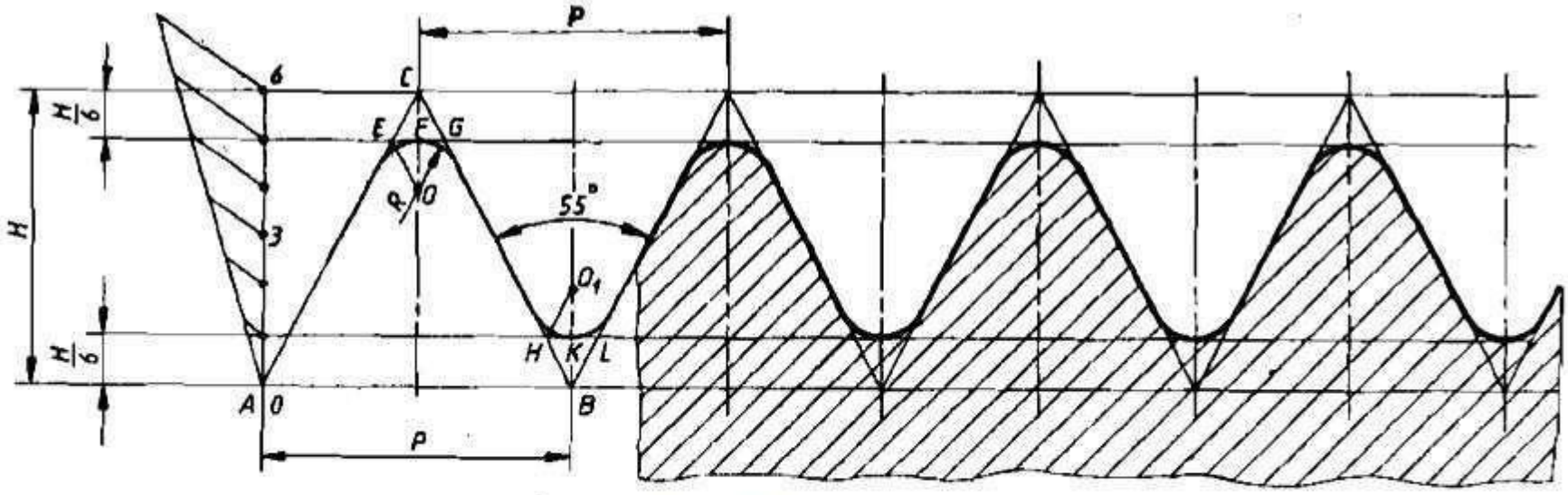
Height of the Fundamental Triangle –The imaginary equilateral triangle which bounds a V – thread is called a fundamental triangle. In Figure, the triangle ABC is called the fundamental triangle. Its height, His measured normal to the axis on an axial plane.

Pitch – It is the distance from a point on a screw thread to a corresponding point on the next thread, measured parallel to the axis. It may be indicated as the distance from crest to crest; or from root to root, but the former is the convention.

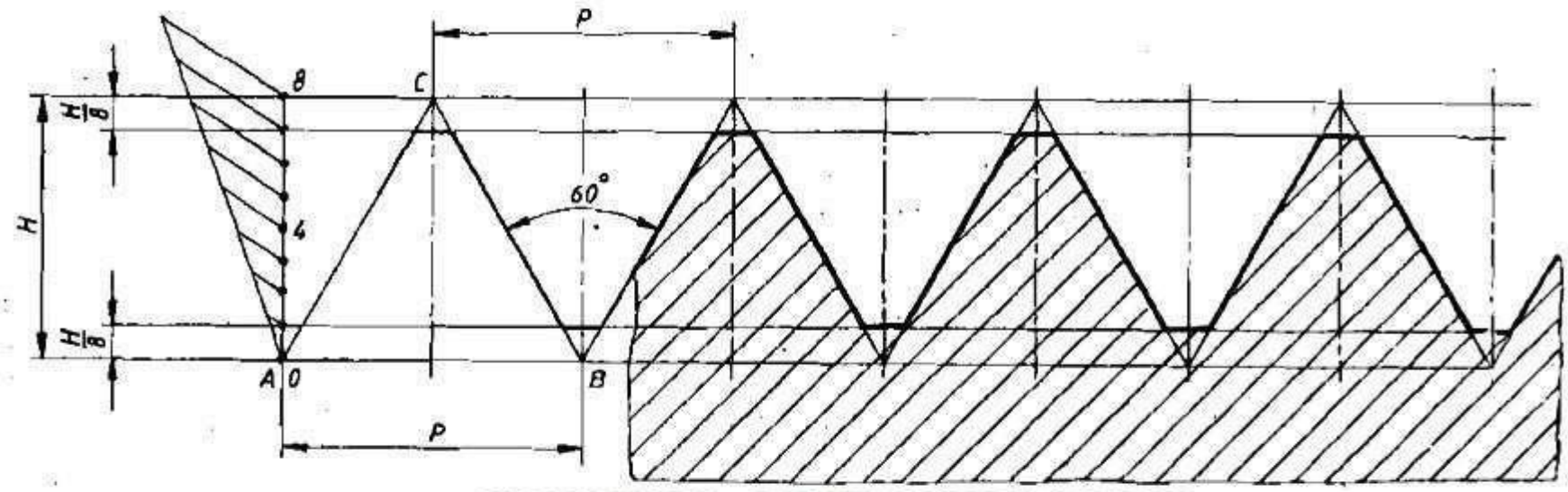
Lead – It is the axial distance advanced by a nut for its one full turn over a threaded rod. On a single start thread, the lead and the pitch are identical. On a double start thread, the lead is twice the pitch, and on a triple start thread, the lead is three times the pitch. Thus, the lead may also be defined as the product of the pitch and number of starts.

DIFFERENCE BETWEEN 'V' AND SQUARE THREADS:

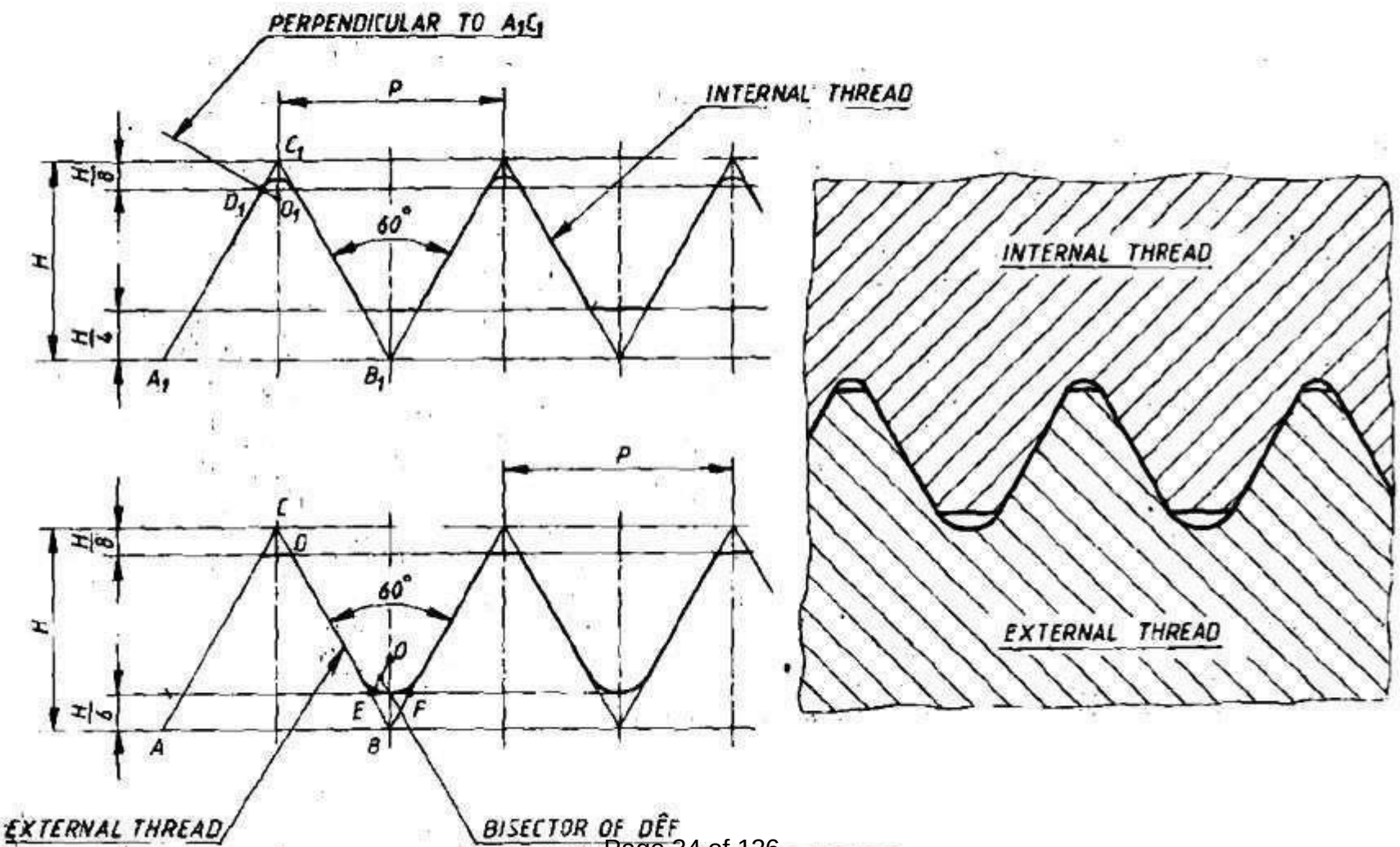
V THREAD	SQUARE THREAD
❖ V - Threads have inclined flanks making an angle between them.	❖ The flanks of square threads are perpendicular to thread axis and parallel to each other.
❖ V – Threads have a larger contact area providing more frictional resistance to motion. So they are more suitable for fastening.	❖ Square threads offer less friction to relative motion. The normal force between the threads acts parallel to the axis with zero radial components. So they are suitable for power transmission.
❖ V – Threads are stronger than the square threads.	❖ Square threads have only half the resisting the power, resting the shearing action.
❖ V – Threads are cheap because they can be cut easily by a die or on machines.	❖ Square threads are costly.
❖ Examples for V – threads are the thread used in bolts, nuts and studs.	❖ Examples for square threads are lead screw of a lathe, screw jack etc.,



WHITWORTH THREAD



SELLERS THREAD - EARLIER AMERICAN STANDARD



UNIFIED THREAD - ISO THREAD

STANDARD FORMS OF V – THREADS:

Whitworth Thread – British Standard

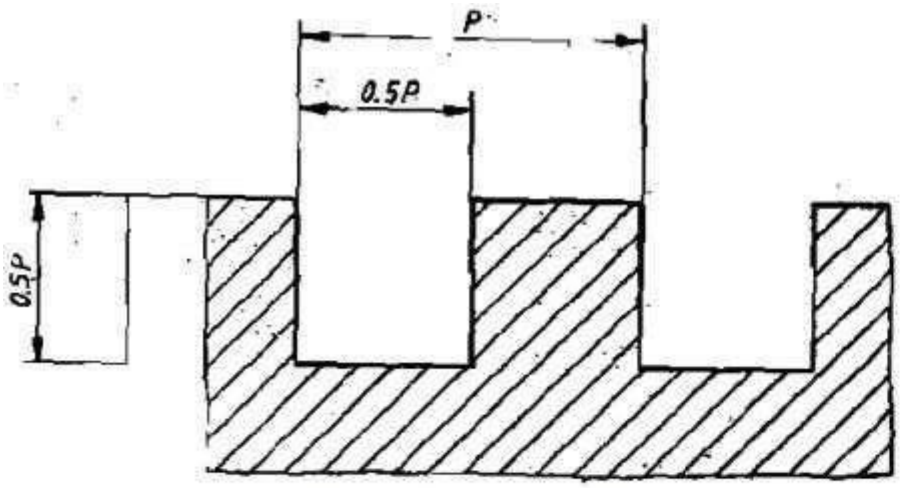
This thread was introduced by Sir Joseph Whitworth and was standardized as British Standard thread, abbreviated as BSW. The profile of the thread is shown in figure. It has a thread angle of 55˚ and is rounded off at the crest and roots.

Sellers Thread – Earlier American Standard

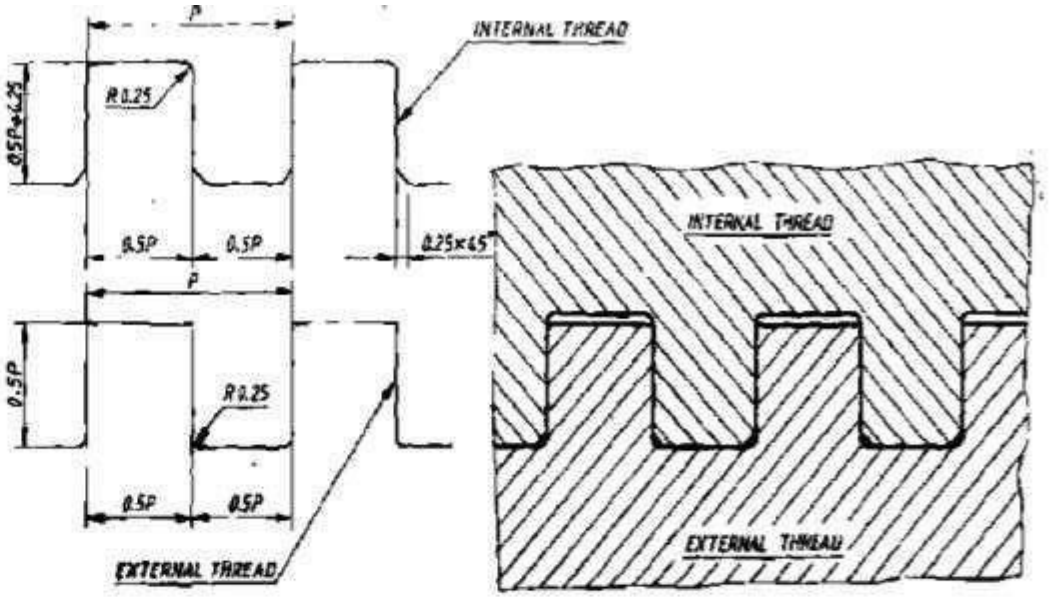
This thread was adopted earlier as the American standard thread by the American Standards Institution. The profile of the thread is shown in figure. It has a thread angle of 60˚ between the crest and roots are kept flat.

Unified Thread – ISO Thread – Indian Standard

The Bureau of Indian Standard has adopted V – thread profile recommended by the International Organizations for Standards, ISO, metric screw thread for use in this country. This thread is also known as Unified Thread. The profile of the thread is shown in figure. It has a thread angle of 60˚

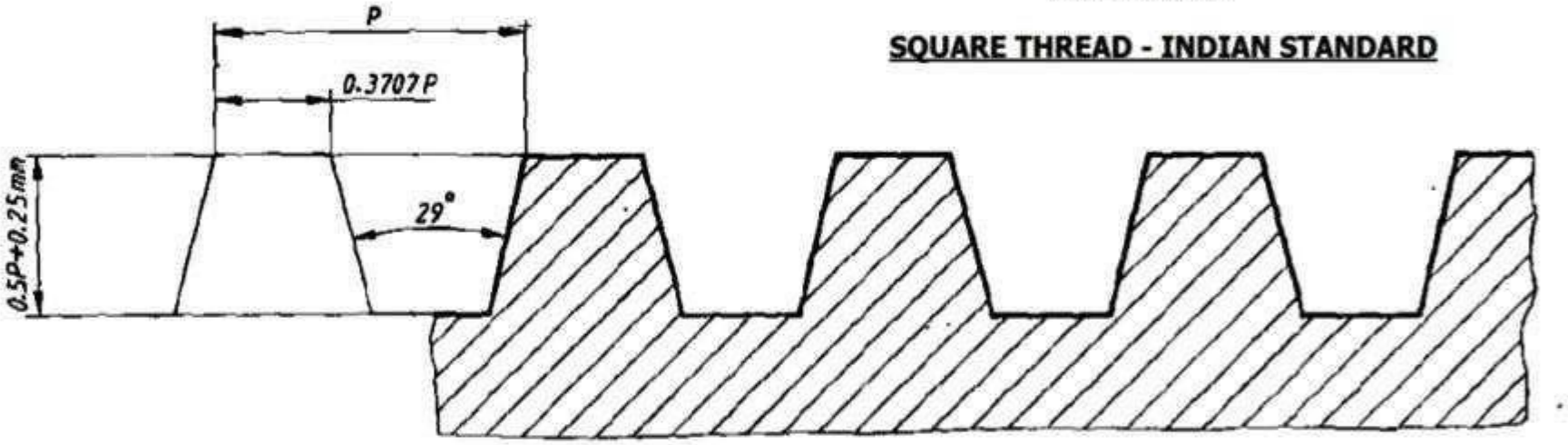


BASIC FORM OF SQUARE THREAD

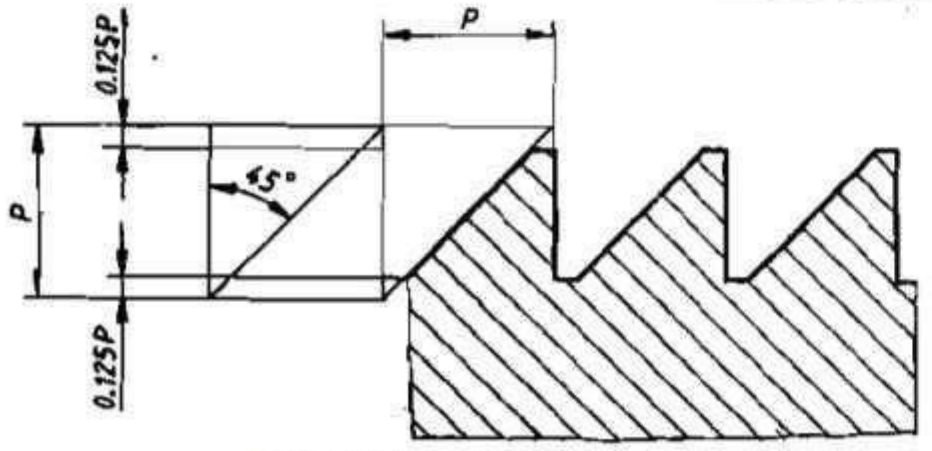


All Dimensions in mm

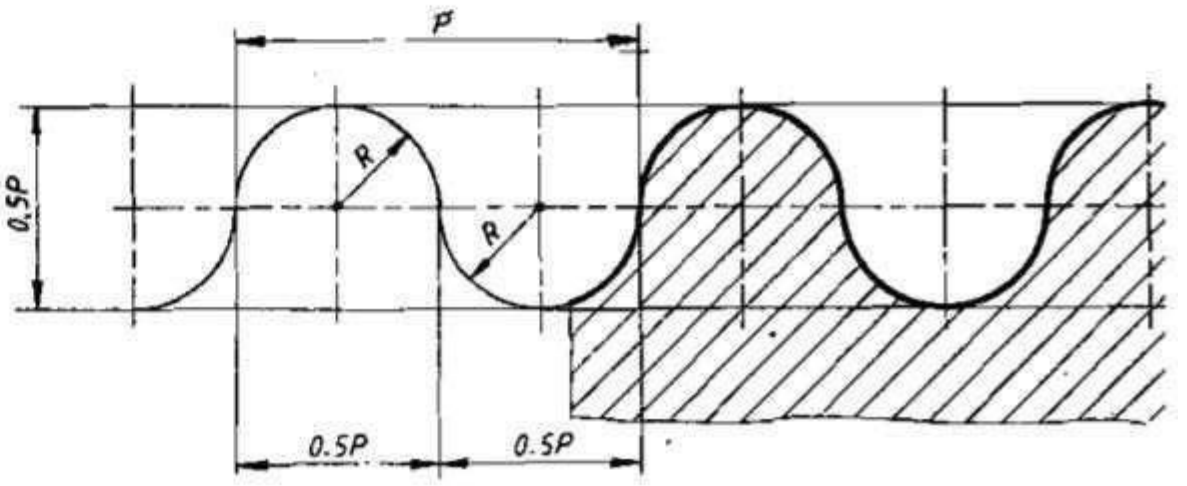
SQUARE THREAD - INDIAN STANDARD



ACME THREAD



BUTTRESS THREAD



BASIC KNUCKLE THREAD

STANDARD FORMS OF SQUARE THREAD:

Basic Form of Square Thread

The basic form square thread is shown in figure. The flanks or the sides of this thread are perpendicular to axis of the thread. The depth and the thickness of the thread equal to half the pitch.

Square Threads – Indian Standard

The profile of the square thread adopted by the Bureau of Indian Standards is shown in figure. The depth of the external thread is equal to half of the pitch. The sharp corners at the root of the external threads are rounded off to the radius R = 0.25mm. The depth of the internal threads is equal to 0.5P + 0.25mm. The sharp corners at the crest of the internal threads are chamfered to 0.25mm * 45°.

MODIFIED FORMS OF SQUARE THREADS:

Acme Thread

This thread is modified from square thread. It is easier to cut and is stronger at the root. The angle of the thread is 29°. The inclined sides of the thread facilitate quick and early engagement and disengagement. It is used for power screws like lead screw of lathe, jackscrews, bench vices and valve operating screws.

Buttress Thread

The profile of this thread is a combination of square and V – threads. It combines the low frictional of square and ability to transmit power of square thread and the strength of V – thread. It is used to transmit load in uni – direction. These threads are used in screw press, vices.

Knuckle Thread

It is also a modification of square thread. The sharp corners of square thread are rounded off. This thread is used where heavy wear rough use is expected. The thread can be rolled or cast

easily. It is used in railway carriage coupling screws, light bulbs and sockets, bottle caps and also objects made of brittle materials as glass, plastic, porcelains etc.

THREADED FASTENERS:

Threaded fasteners are temporary fasteners, which hold the parts together throughthe medium of a screw thread. These are used in pairs for their action (for example, a nut and a bolt). They have the advantage over permanent fasteners of allowing assembly of parts when required. A wide variety of threaded fasteners are in use. Some of them are standardized and others are made for special use.

COMMON TYPE OF THREADED FASTENERS:

The five types of threaded fasteners in common use are

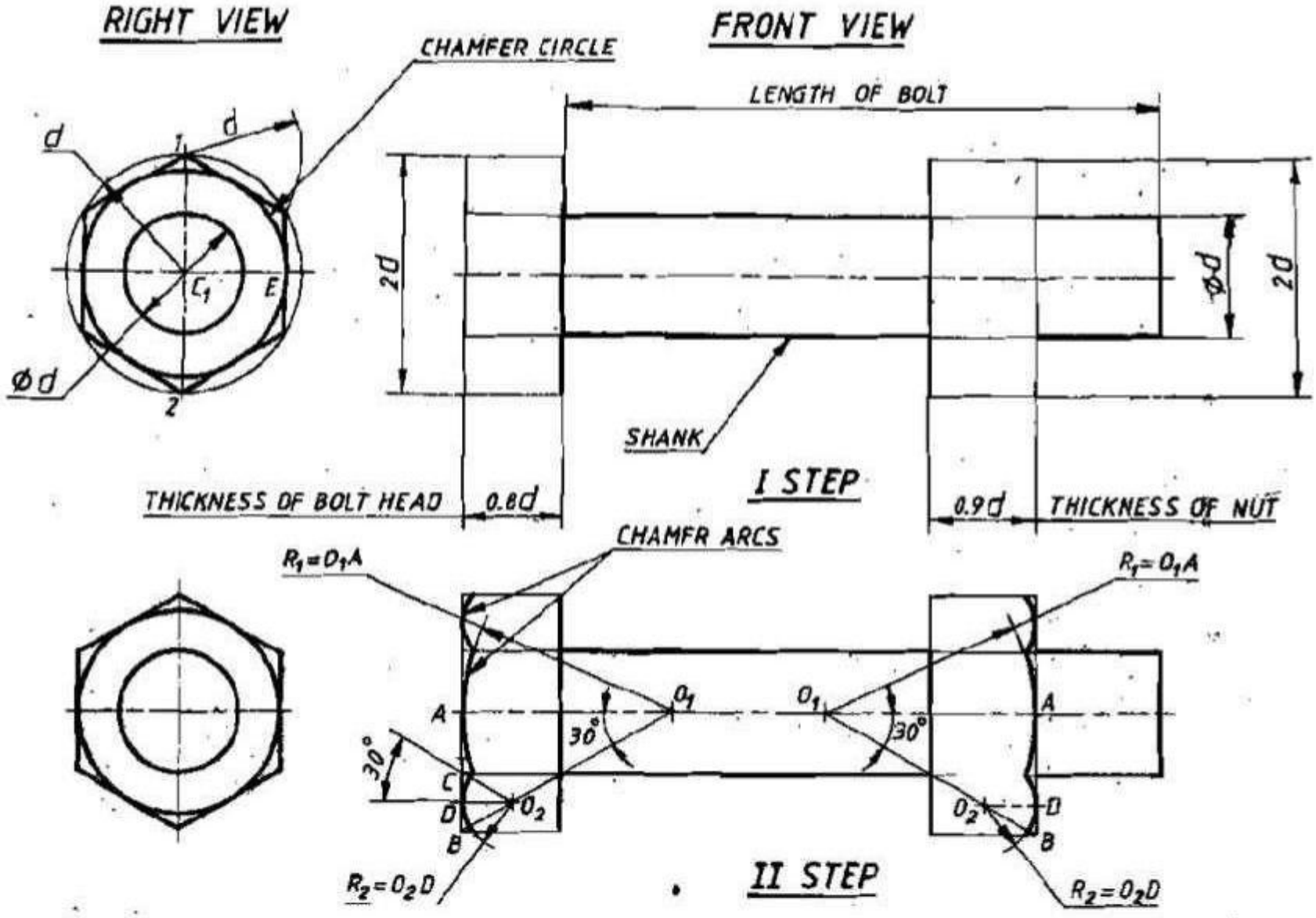
- ❖ Bolt
- ❖ Stud
- ❖ Cap screw
- ❖ Machine screw
- ❖ Setscrew

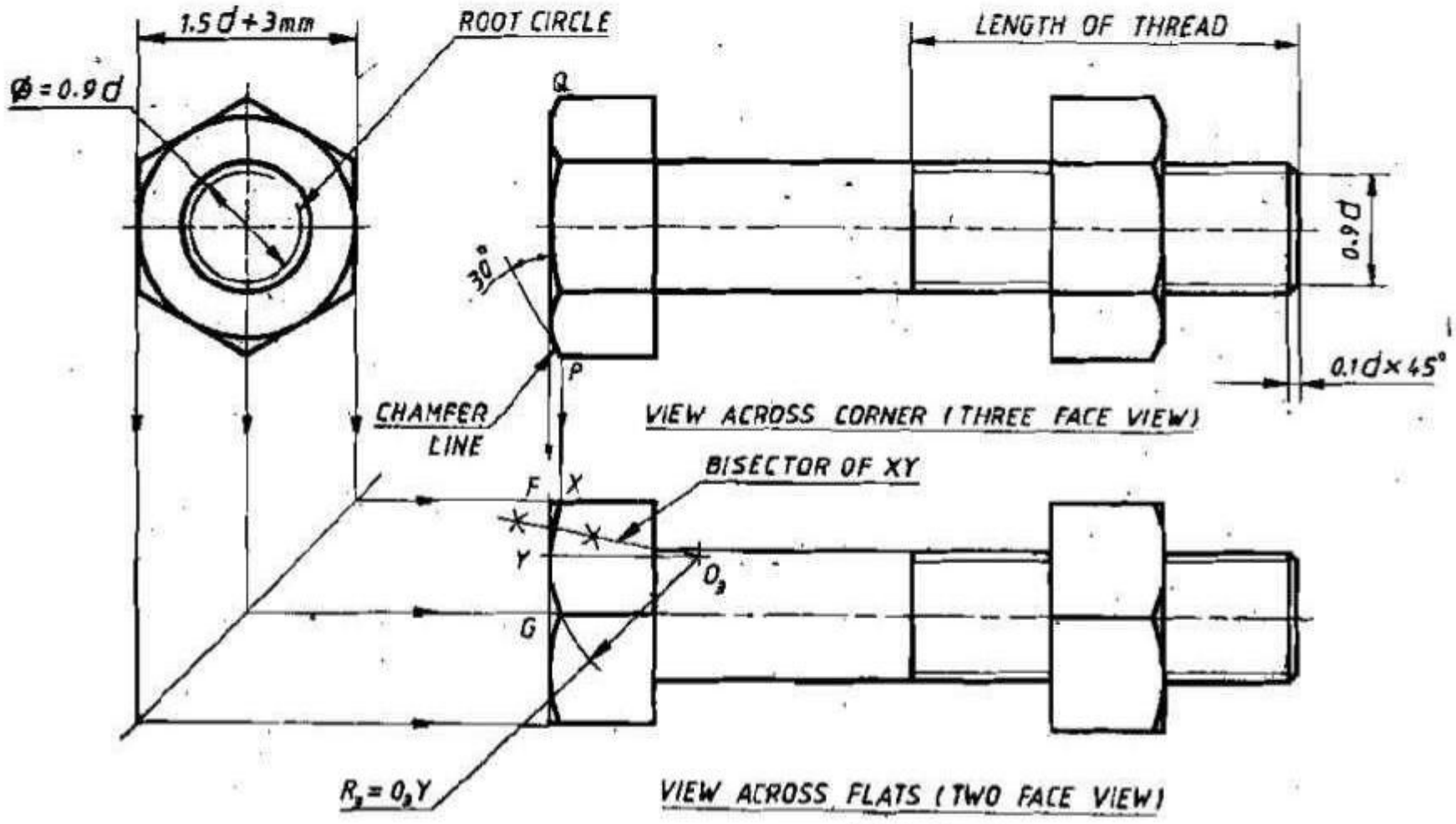
All these with external threaded and used in combination with another having corresponding internal threads (eg) a nut or a tapped hole.

BOLTS:

A bolt is a metal having a head at one end and a threaded portion to a definite length on other end. The head is formed by forging or machining. The bolt is admitted through holes in the parts, which are to be fastened. The projected thread end of the bolt admits a corresponding nut from the other side. Tightening the bolt by turning gives necessary clamping grip to hold the parts together.

Bolts and nuts of various shapes are used for different purpose but the hexagonal head and square head are very common. Although, the square shape provides better spanner grip than the hexagon, but needs one fourth of a turn to bring it into the same position for inserting spanner again, whereas a hexagon need only one sixth of a turn and hence provided.





STEPS IN DRAWING HEXAGON HEAD BOLT & NUT

The sharp corners on the external flat end faces of bolt heads and nuts are chamfered conically at 30° to ensure safety of the user. To facilitate early insertion of the nut over the bolt, the threaded holes in the nut are countersunk. Three dimensions are usually sufficient for simplified representation of a bolt

The bolt shank diameter **(d)**

The bolt length **(l)**

The length of a threaded portion of the shank **(b)**

EMPIRICAL PORTIONS OF HEXAGON AND SQUARE HEAD BOLT & NUT

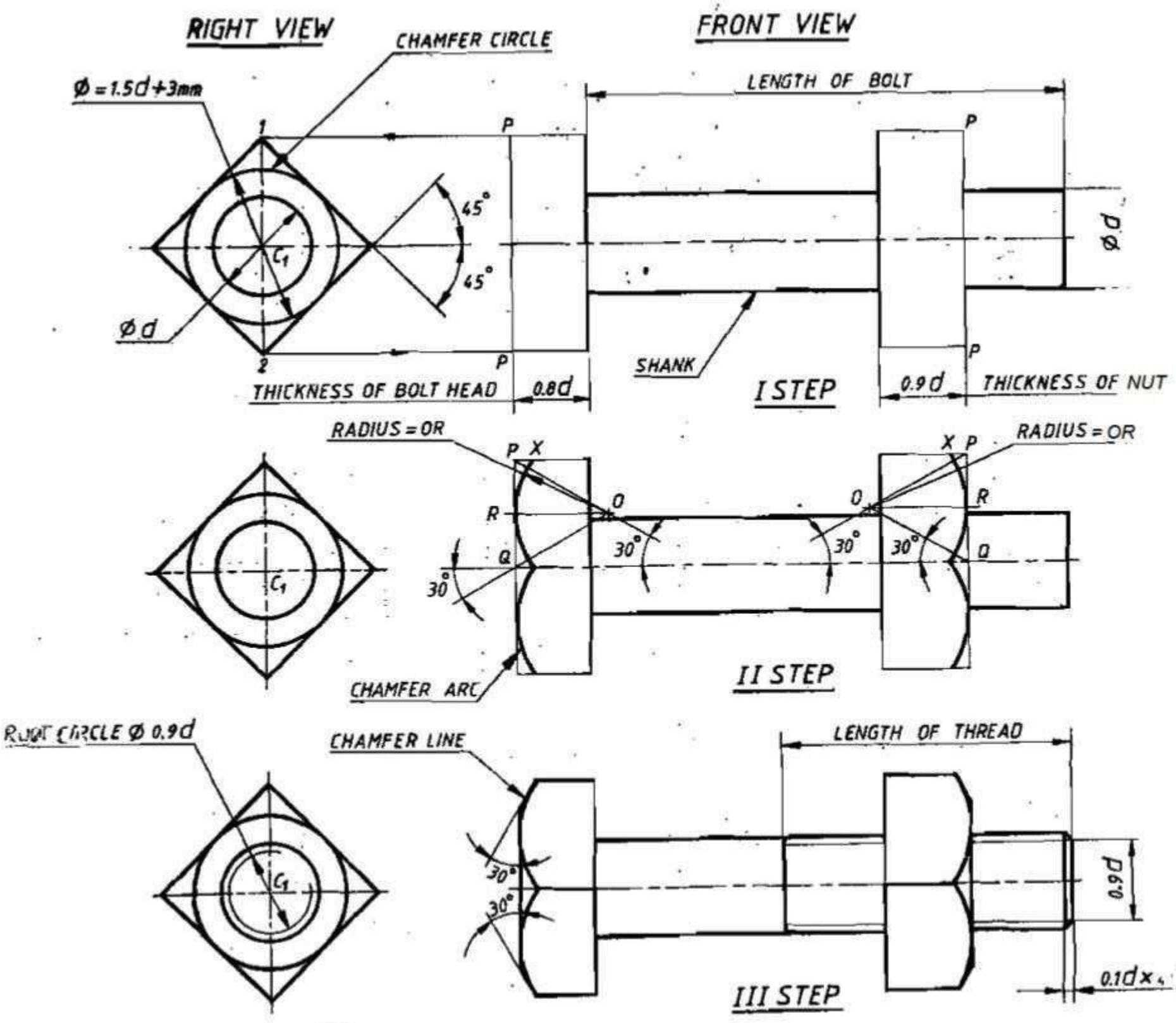
DETAIL PROPORTION:

Nominal diameter	d	= size of bolt or nut mm
Width across flats	s	=1.5d+3mm
Width across corners	e	=2d
Thickness of bolt head	k	=0.8d
Thickness of nut	n	=0.9d
Root diameter	d_1	=d–(2*depth of thread)
Length of the bolt	l	=as specified
Thread length	b	=2d+6mm (for l<150mm)
		=2d+12mm (for l>150mm)
Chamfer of bolt end	Z	=depth of thread*45(degree) (or) =0.1d
Chamfer angle of bolt head & nut		= 30°

DRAWING OF HEXAGONAL NUT AND BOLT

STEP: 1

Draw the shank of the bolt equal to the given diameter (d) and length (l).The thickness of the bolt head equals to 0.8d and the thickness of nut equal to 0.9d are marked. Measure the width across corners equal to 2d and complete the three faces of the bolt head and nut in these lines.


RIGHT VIEW
CHAMFER CIRCLE
FRONT VIEW
Ø = 1.5d + 3mm
LENGTH OF BOLT
1
P
45°
45°
Ød
2
P
SHANK
P
Ød
P
THICKNESS OF BOLT HEAD
0.8d
I STEP
0.9d
THICKNESS OF NUT
RADIUS = OR
P X
RADIUS = OR
X P
O
O
R
R
Q
Q
30°
30°
30°
30°
CHAMFER ARC
II STEP
ROOT CIRCLE Ø 0.9d
CHAMFER LINE
LENGTH OF THREAD
30°
30°
0.9d
0.1d x
III STEP

The right hand view of the bolt and nut assembly is drawn as follows with any point on the axis as centre and radius equal to draw a thin circle. A hexagon is inscribed inside this circle. The chamfer circle is drawn as a thick circle with the centre C_1 and radius C_1E.

STEP: 2

The chamfer arcs in three face view of bolt head and nut are drawn as follows. Through the corner B, draw a line 30° to the axis of the bolt or nut to cut it at O_1. With O_1 as centre O_1A draw the chamfer arc in the centre face.The chamfer arcs on the two side faces are drawn as follows. Draw the perpendicular bisector of BC to cut BO_1 at O_2 as centre and radius O_2D draw the chamfer arc. Repeat the construction on the other side face.

STEP: 3

The chamfer lines on the side faces of the three face views of the bolt head and nut are drawn through the points P and Q inclined at 30° to the flat faces of the bolt. The end of the bolt is chamfered 0.1d*45°.

The threaded portion of the shank is indicated, by drawing two thin lines at a distance equal to d_1=0.9d.the root circle in the right view is represented by a thin three-fourth of a circle of diameter 0.9d.

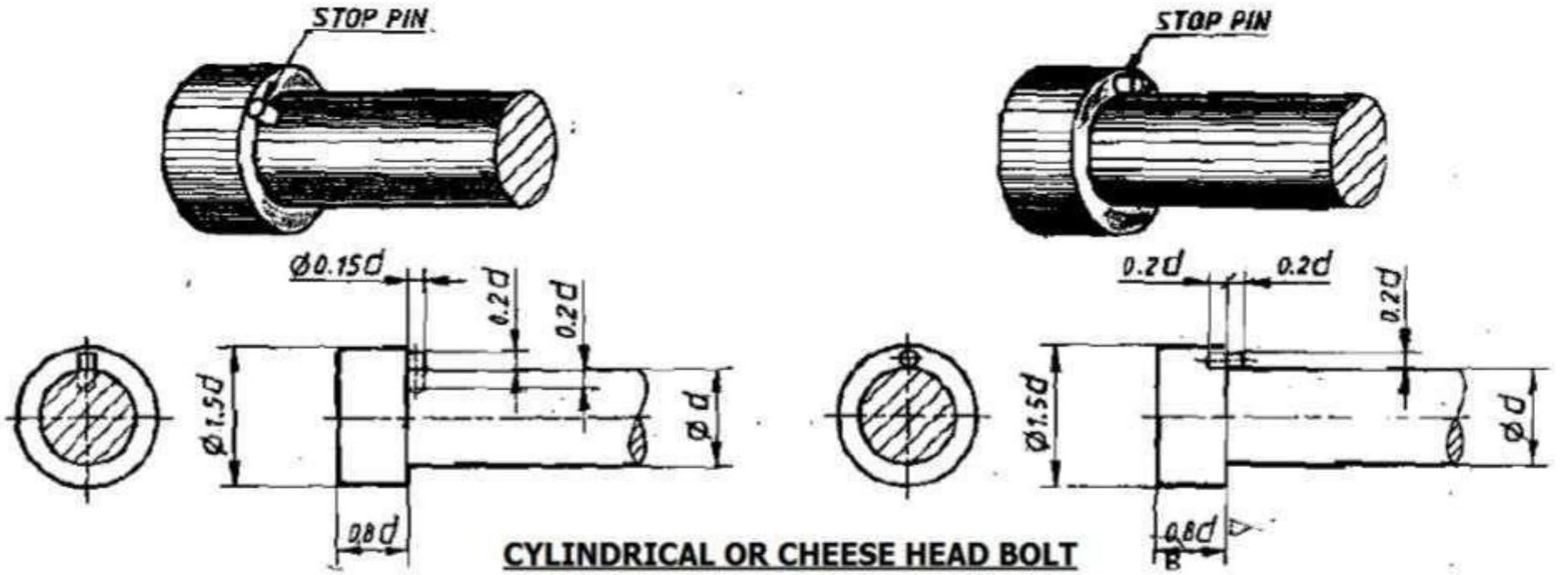
STEP: 4

The two face view of the bolt head and nut is projected from the side view. If the side view is not drawn, then the distance across the flats is measured equal 1.5d + 3mm. The chamfer arcs in the two face view are drawn as follows. Project P to get X. Mark Y the midpoint of FG. Draw the perpendicular bisector of XY and FG to intersect each other at O_3. With centre O_3 and radius O_3Y draw the chamfer arc. Repeat the construction on the other face

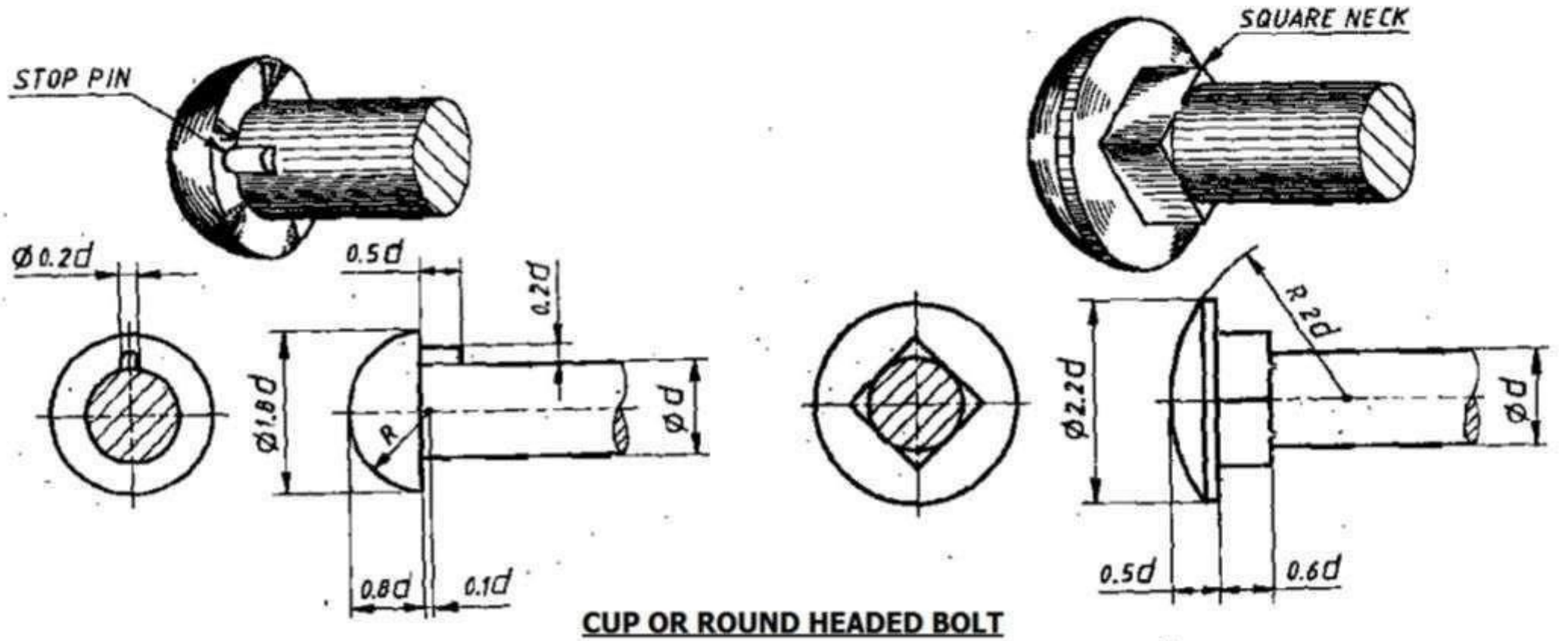
DRAWING OF SQUARE HEAD BOLT AND NUT:

STEP: 1

Draw the shank of the bolt equal to the given diameter d and the length of the bolt. The thickness of the bolt head is equal to 0.8d and the thickness of the nut is equal to 0.9d are marked. The right hand view of the bolt and nut assembly is drawn as follows. With any



CYLINDRICAL OR CHEESE HEAD BOLT



CUP OR ROUND HEADED BOLT

A B

pointC_1 on the axis as centre and diameter equal to 1.5d+3mm draw a chamfer circle with its sides inclined at 45° to the axis. Project the corners 1 and 2 to get to get point P.Drawa thick circle with diameter d to indicate the nominal diameter.

STEP: 2

The chamfer arcs in the view across the corners of the bolt and nut are drawn as follows. Through the corner P, draw a line inclined at 30° to the axis. Draw the perpendicular bisector of PQ to intersect the 30° line at O. With O as centre and radius OR draw the chamfer arc. Repeat the construction on the other face.

STEP: 3

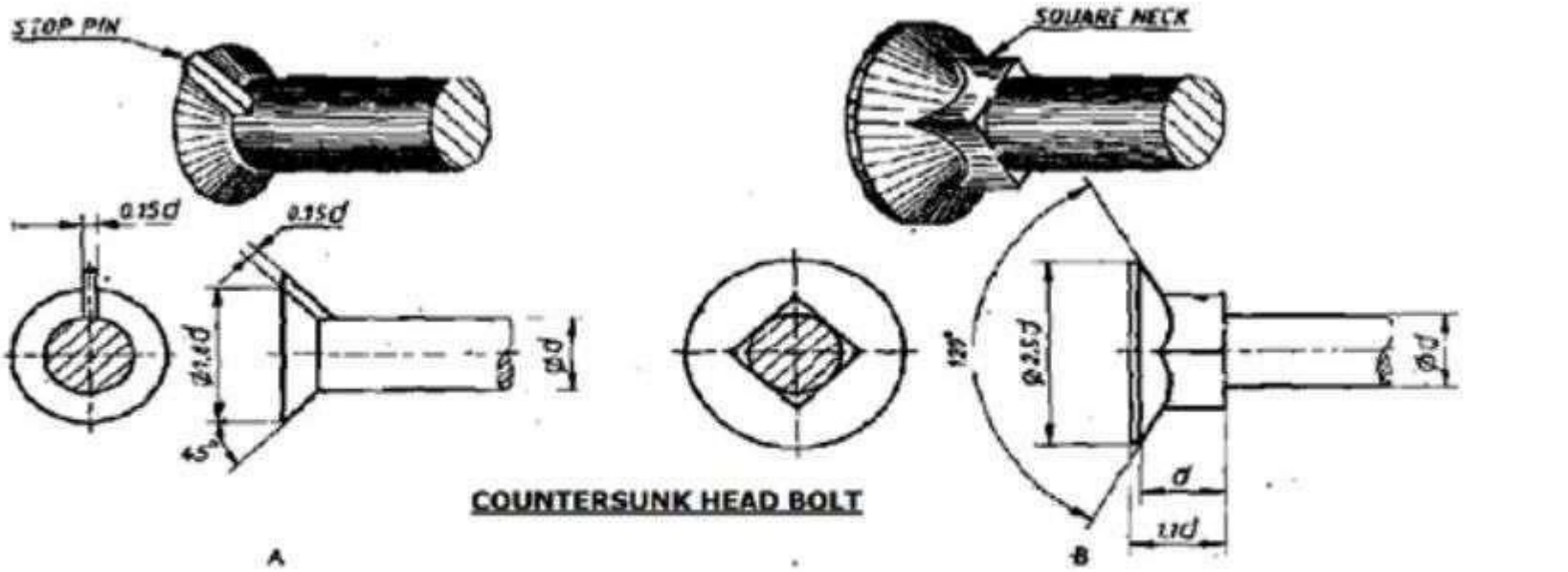
The chamfer line is drawn at 30° to the flat face of the bolt head and nut. The threaded portion on the shank of the bolt is indicated by drawing two thin line spaced at a distance equal to the root diameter d1=0.9d. The root circle in the right view is represented by a thin three-fourth of a circle with centre C_1 and diameter 0.9. The end of the bolt is chamfered to 0.1d*45°.

SPECIAL TYPES OF BOLTS:

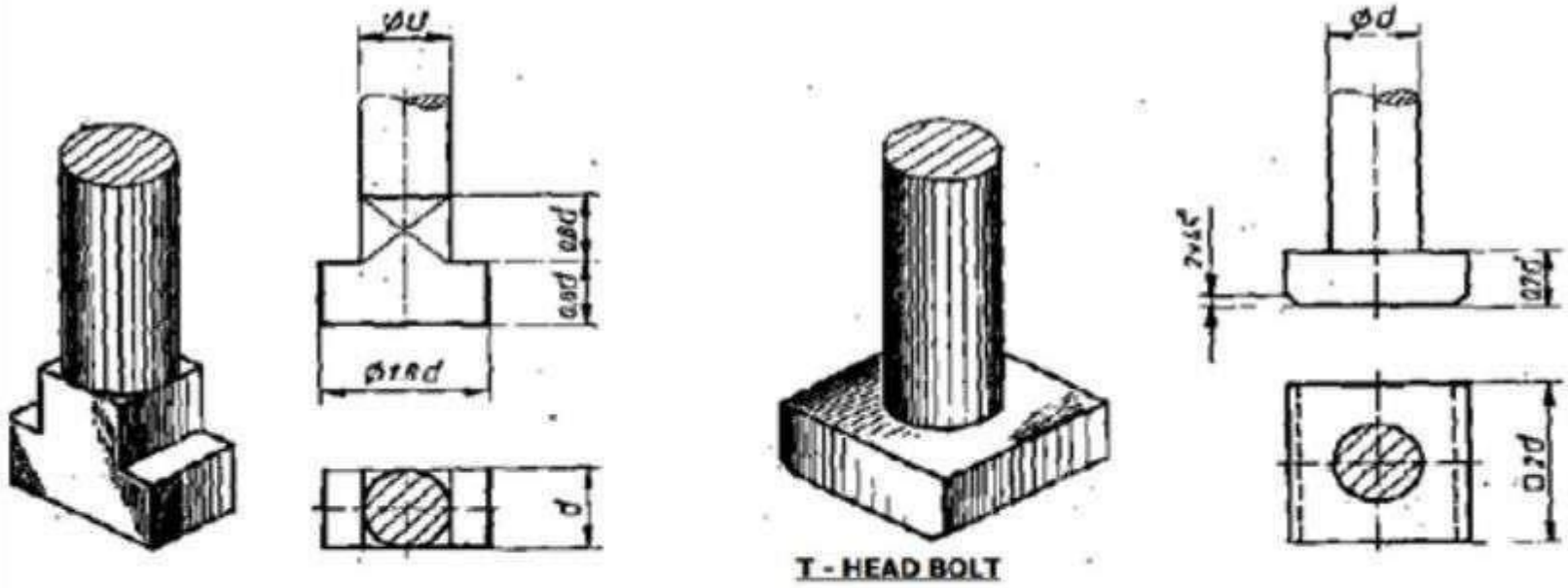
In practice various types of bolts than the hexagon and square head bolts are used in where the bolt head cannot be held by the spanner when the nut is turned on or off the bolt. The rotation of the bolt prevented by a stop pin or a snug or a square neck provided below the head.

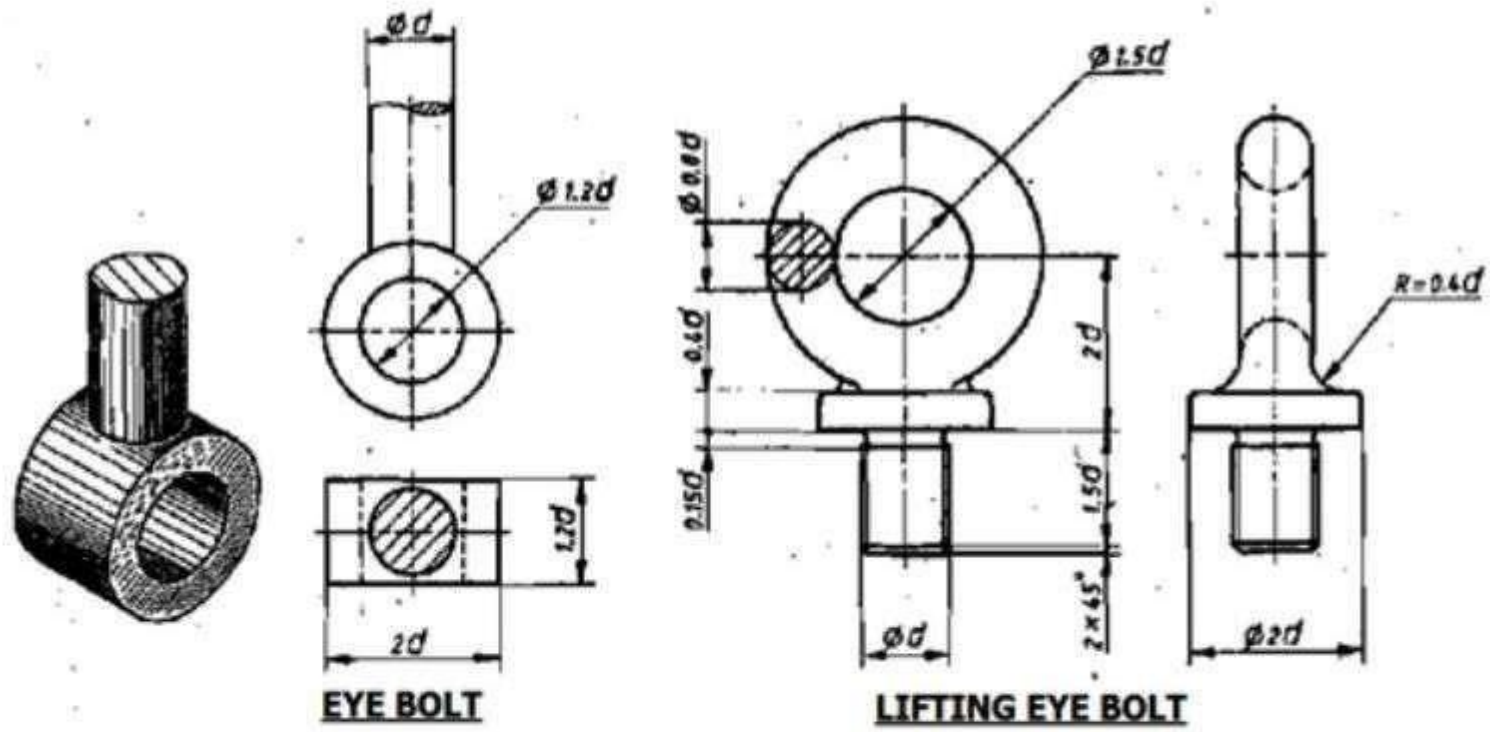
CYLINDRICAL OR CHEESE HEADED BOLT:

The head of this type of bolt is of cup shape and the rotation of the bolt head is prevented by a stop pin. The stop pin may be driven into the shank with its axis perpendicular to the axis of the bolt. The stop pin may also be driven into the head adjacent to the shank with its axis parallel to the axis of the bolt. These types of bolt heads are used in the big ends of the connecting rods, eccentrics, cross heads etc...



COUNTERSUNK HEAD BOLT





COUNTER SUNK HEAD BOLT:

CUP OR ROUND HEADED BOLT:

Two types of cup head bolts are available. In one type, a snug is provided which prevents the rotation of bolt head. The other type, a square neck is provided which will fit into the square hole provided in the bearing surface and thus prevents the rotation of the bolt head. The counter sunk head bolts are used when the bolt head must not project and foul with surfaces. The counter sunk bolt is provided with a stop pin of square cross section integral with the head. The other type of counter sunk bolt is provided with the square neck below the head. This type of bolt is also called as "coach bolt".

T – HEAD BOLT:

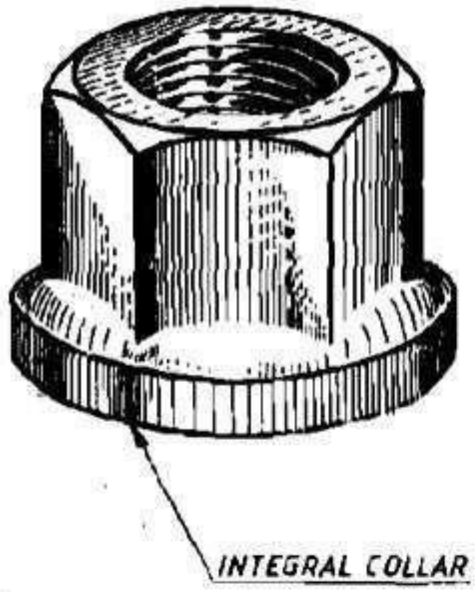
Two types of T bolts are shown in figure. The T head bolt is provided with a square neck underneath the head to prevent the rotation of the bolt. This type of the bolt is used to connect the covers or the flanges where there is sufficient space to accommodate the hexagon head blot. The other T – head bolt is used for securing vices, jigs, work pieces to the tables of the machine tools like shaping, planning, drilling machine, where frequent tightening and loosening are carried.

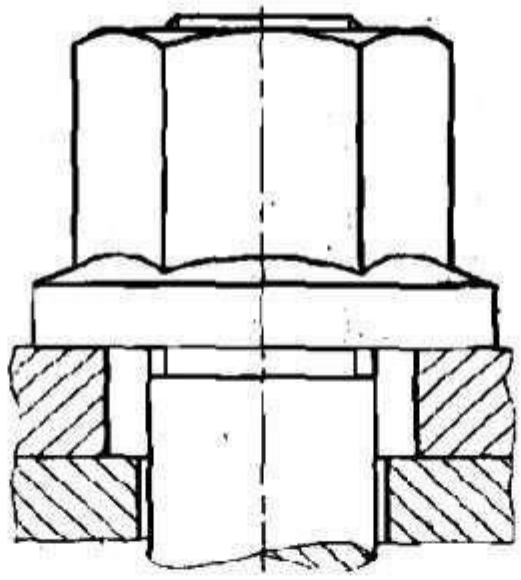
EYE BOLT:

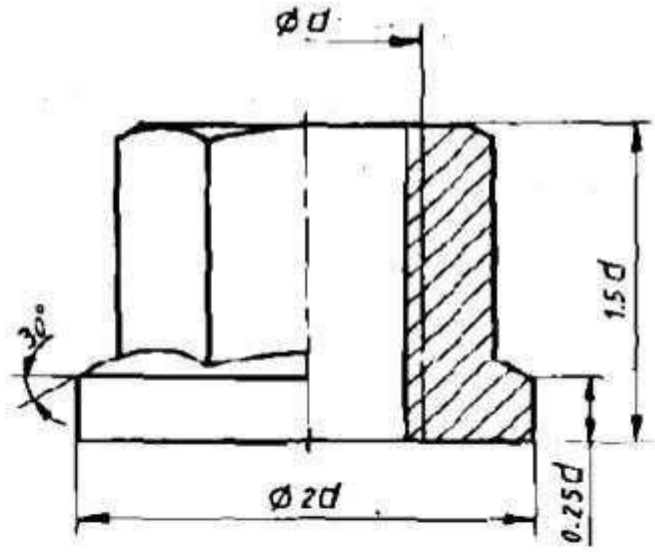
The head of the bolt is in the form of circular form of rectangular cross section. It is generally used in the inspection covers, lids etc..., which have to be opened and closed frequently.

LIFTING EYE BOLT:

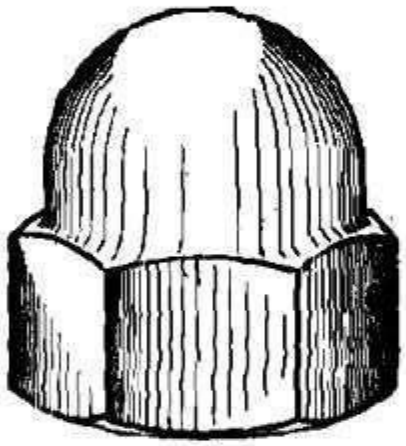
The lifting eye bolt, having a circular ring of circular cross section as head. A flat circular portion, integral with a head is also provided. This type of bolt is used for lifting a heavy machine such as motors, pumps, turbine, electrical generators etc., This bolt is screwed in a threaded hole provided for this purpose , on the top of the machine directly

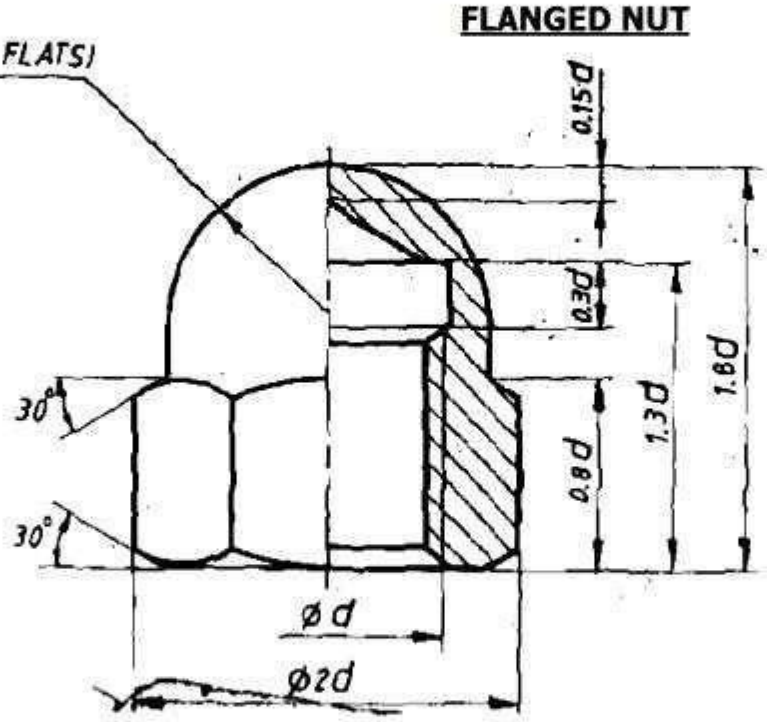


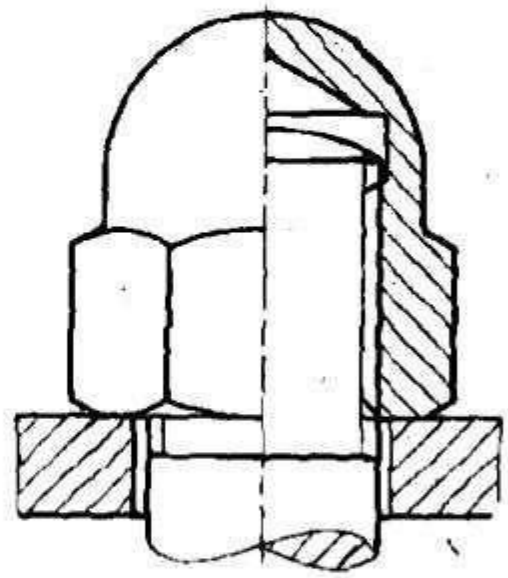




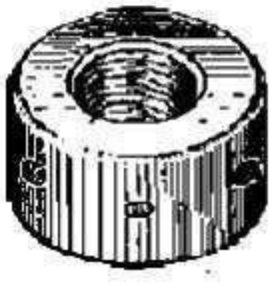
FLANGED NUT

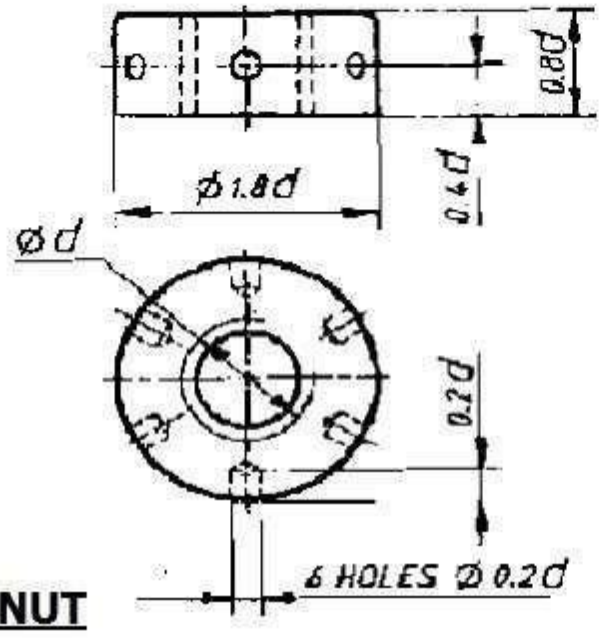




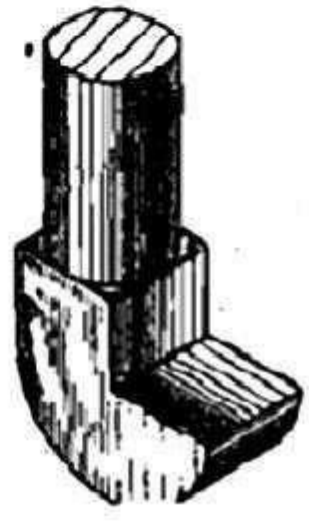


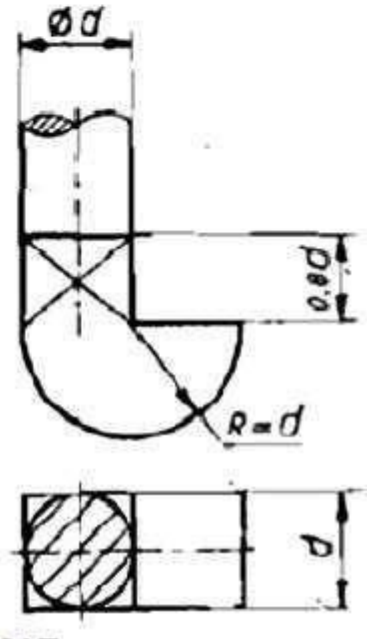
DOMED CAP NUT





CAPSTUN NUT





HOOK BOLT

above the centre of gravity so that while lifting the machine does not change from its usual working position.

HOOK BOLT:

The hook bolt has its head comprising of a square neck and projection. The shank of the bolt passes through a hole in one of the fastening pieces and the other piece comes under the bolt head and is supported by it.

NUTS:

A nut is a device having internal threads used in combination with a bolt or stud, having external threads to fasten parts together. It is screwed on the threaded end of the bolt or stud and the head of the bolt is drawn closer to hold and tighten the parts to be joined. Nuts are usually made in form of hexagonal or square prism, however various other types of nuts are also used for the specified purposes, which are suitable for a particular type of work. These special types of nuts are described here.

FLANGED NUT:

A hexagonal nut having a collar or a circular flange provided integral with it at if its bottom as shown in Figure is called a flanged nut. The collar increases the bearing aka of the nut and also permits the use of a bolt in a larger hole as shown in Figure

DOMED CAP NUT:

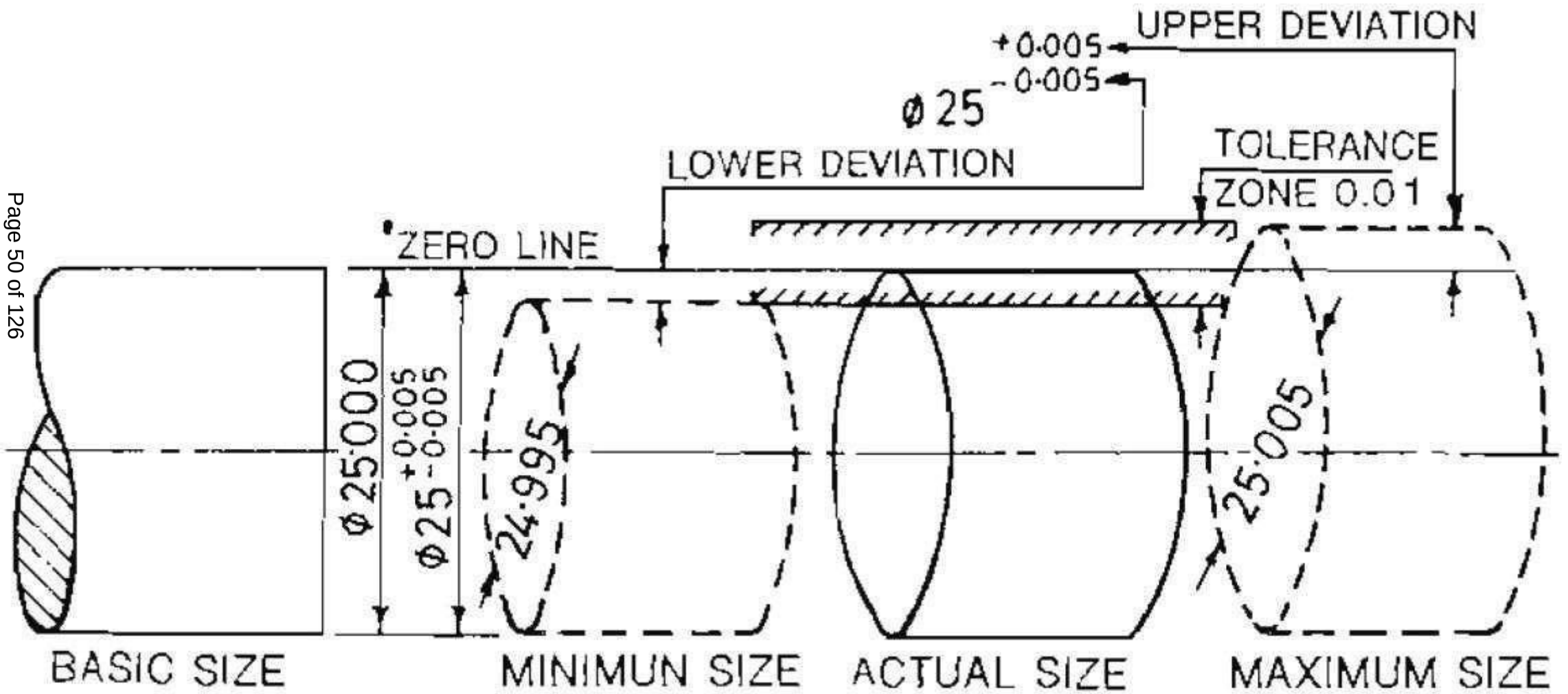
The domed cap nut shown in figure is a blind hexagonal nut used when the ends of the bolts or studs are not exposed. Thus the ends of the bolts are protected from corrosion. These nuts are usually made of brass, bronze, etc.

CAPSTUN NUT:

It is cylindrical in shape. Six holes are drilled in the curved surface of the nut for turning it with a tommy bar is introduced in these holes. Sometimes holes are also drilled in the upper flat face of the nut, for turning with a tommy bar.

RESULT:

Thus the screw threads and fasteners were studied.



TERMINOLOGY FOR DIMENSIONAL TOLERANCE

STUDY OF LIMITS, TOLERANCE AND FITS

EXPT NO: 6 DATE:

AIM:

To study about limits, tolerance &fits.

LIMITS:

Limits are the two extreme permissible dimension of any part. There are two extreme permissible sizes for a dimension. The largest permissible size is called as upper limit or high limit (H) whereas the smallest permissible size is called as lower limit (L).

TOLERANCE:

The difference between the maximum and minimum limits of dimension is known as tolerance.

TERMINOLOGY FOR DIMENSIONAL TOLERANCE:

Size – A number expressing the numerical value of a dimension expressed in appropriate units indicates the size of the component on its drawing.

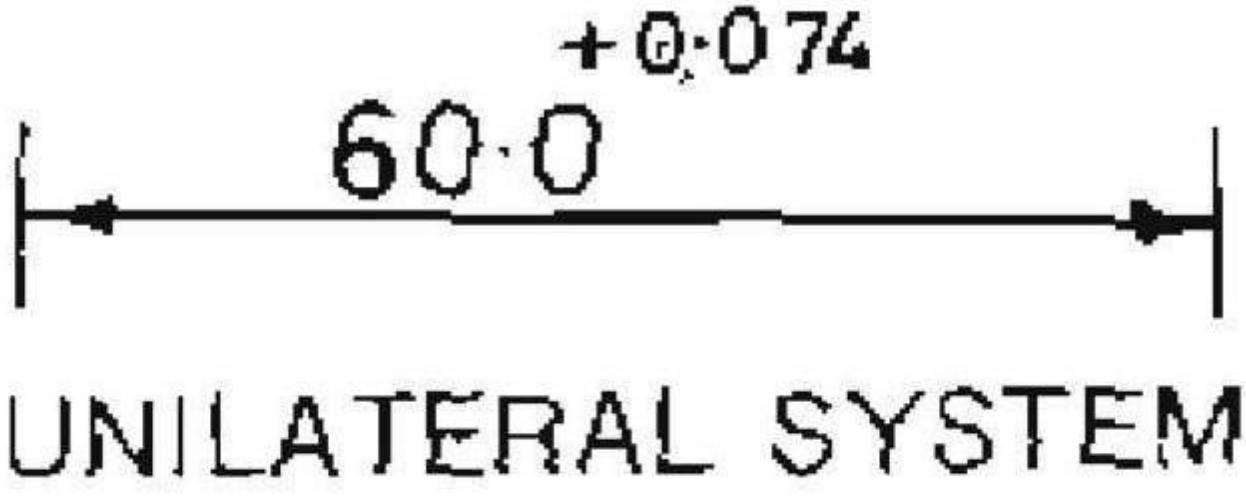
Actual Size – The size of a finished component which is actually measurable with instruments is known as the actual size of the component.

Nominal Size – The nominal size is the designation which is used for the purpose of general identification of a component.

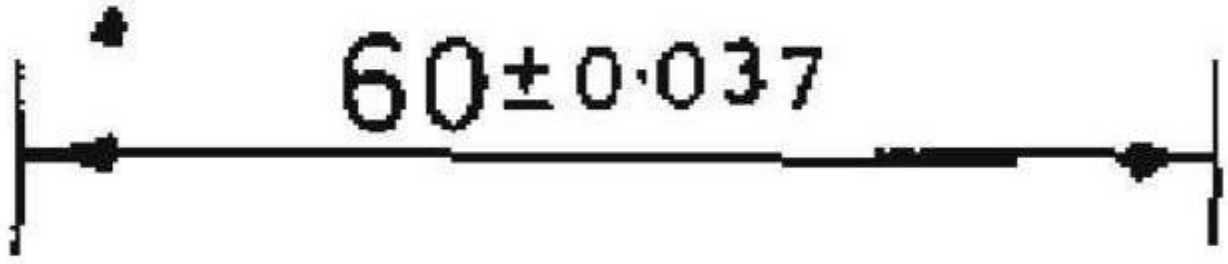
Basic Size – It is the design size of the component from which the limits are delivered by the application of tolerance.

Basic Dimension – It is the dimension of the machine part obtained by design calculations.

For example, diameter of a pin is found to be 23.48 mm by design calculation, and approximated to 25 mm. This dimension (25 mm) is known as the basic dimension or the



UNILATERAL SYSTEM



BILATERAL SYSTEM

nominal dimension. The allowable maximum and minimum limits are decided with reference to the basic size.

Actual Size– It is the size of component as may be found by actual measurement. For a component to be acceptable actual size should be within maximum limit and minimum limit.

Zero Line–In the graphical representation of tolerance system, the zero line represents the basic size. The upper deviation and the lower deviation are measured from the zero line.

Unilateral Limits– It is the method of representing limits. When both the limits of size we on the same side of zero line, the component dimension has unilateral limits. One of the limits of the size can be the basic size,

$$\text{e.g. } \phi\ 50\ ^{+\ 0.05}_{+\ 0.00} \text{ or } \phi\ \underline{50}\ ^{+\ 0.00}_{-\ 0.05}.$$

Bilateral Limits– Here, one of the limits of the size is on onside of the zero line and the other limit of size is on the other side of the zero line,

$$\text{e.g. } 60\ ^{+\ 0.05}_{-\ 0.05}.$$

Upper Deviation– It is an algebraic difference between the maximum size and basic size.

Upper deviation = Maximum Limit – Basic size.

Consider that in the above problem, the maximum limit is 25.05mm and minimum limit is 24.95 mm.

Upper deviation = 25.05 –25.00 = 0.05 mm.

Lower Deviation – It is an algebraic difference between minimum limit and the basic size.

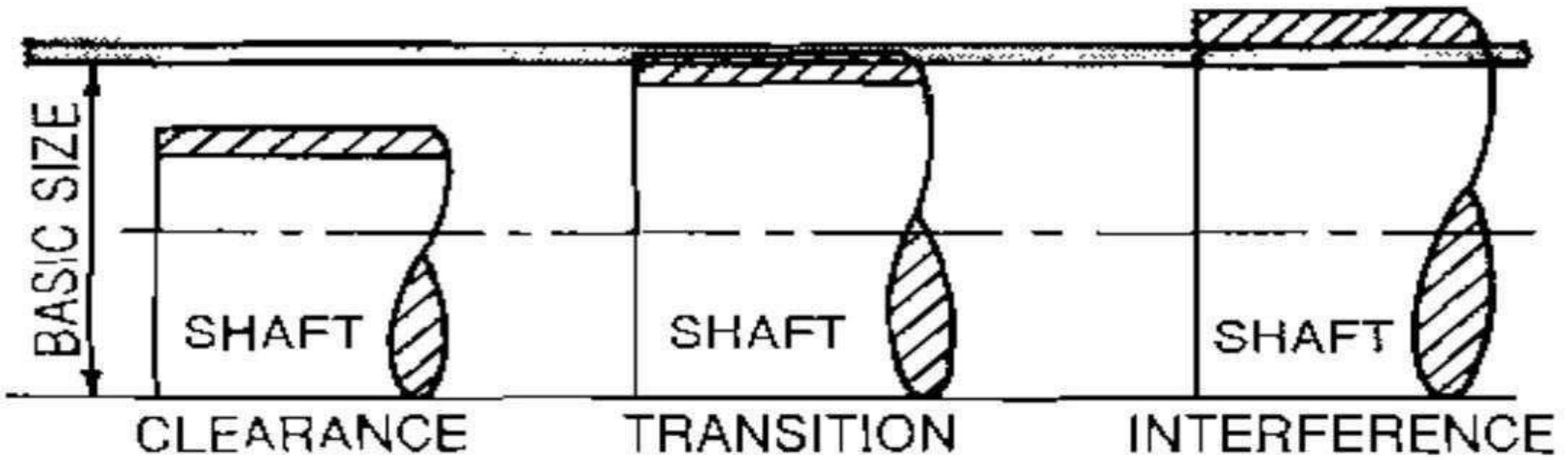
Lower deviation = Minimum Limit – Basic size

Lower deviation = 24.95 –25.00

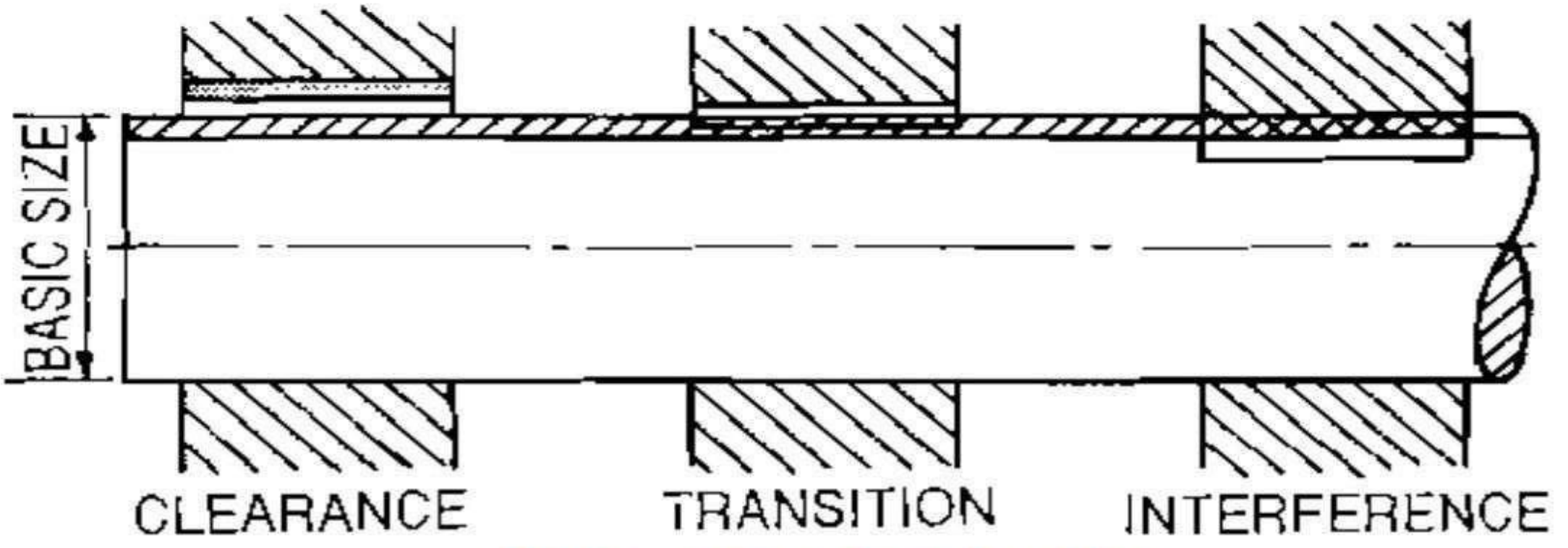
Lower deviation = –0.05 mm.

Tolerance Zone – It is an algebraic difference between the maximum limit and minimum limit.

Tolerance zone = Maximum Limit – Minimum Limit



HOLE BASIS SYSTEM



SHAFT BASIS SYSTEM

Tolerance zone = 25.05 –24.95 = 0.1 mm

Or

Tolerance zone = Upper Deviation – Lower Deviation

Tolerance zone = 0.05 –(–0.05) = 0.10 mm.

Maximum Material Condition–This is defined as the upper limit of shaft and the lower limit of hole.

Minimum Material Condition–This is defined as the lower limit of shaft and the upper limit of hole.

These two conditions are useful in the design of limit gauges.

Allowance– It is an intentional difference between the maximum material limits of mating parts.

Mating Surfaces and Mating Dimensions–When two components are assembled, the contact surfaces are known as mating surfaces and their dimensions are termed as mating dimensions.

Basic Shaft– A basic shaft is one whose upper deviation is zero e.g. shaft h.

Basic Hole– A basic hole is one whose lower deviation is zero e.g. hole *H*.

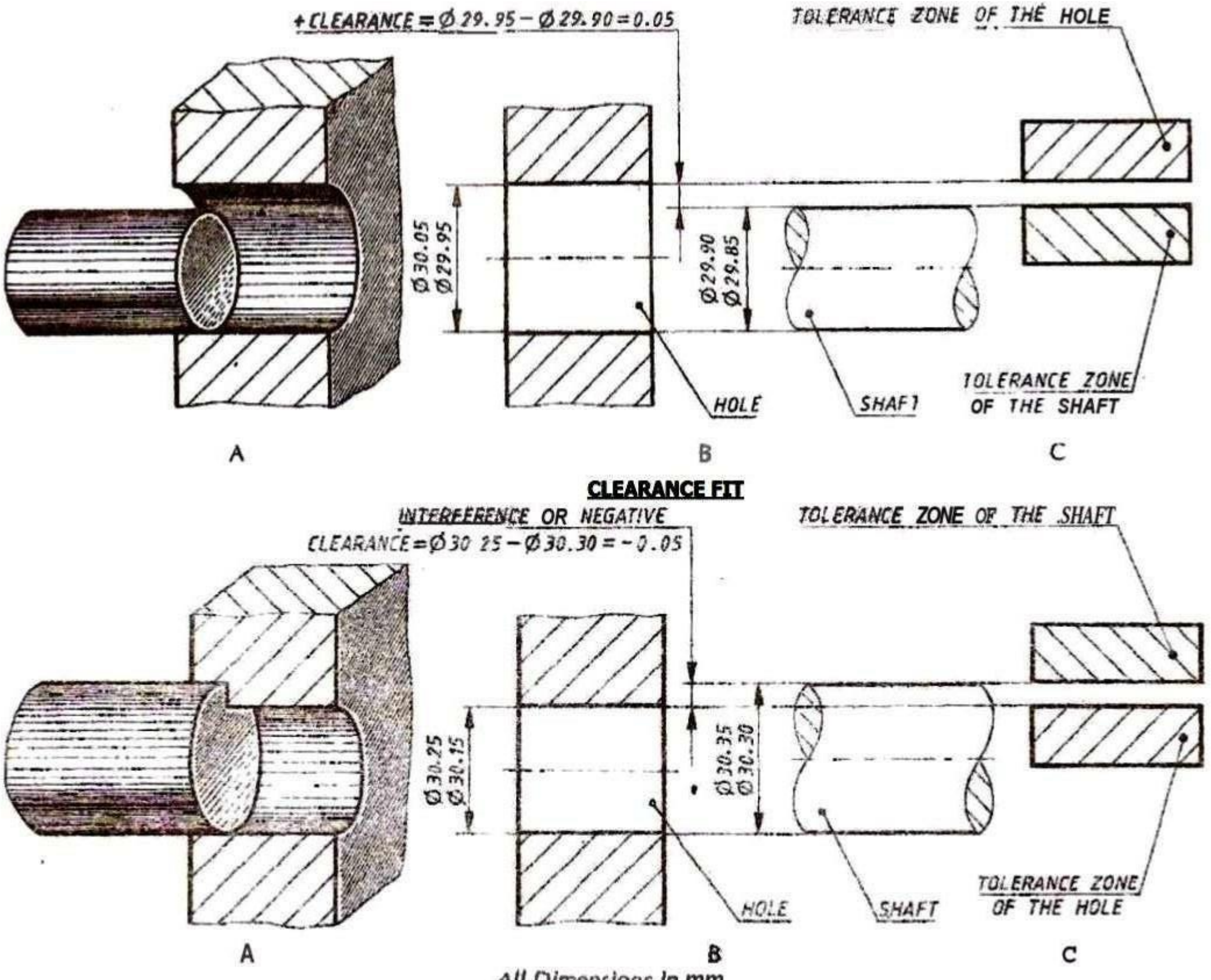
HOLE BASIS SYSTEM:

The nominal size of hole is taken same as the design size. The tolerance zone of hole is taken as constant, while the tolerance zone of the shaft is varied above or below the zero – line according to the required fits as shown in figure. This system is popular in industries due to availability of standard tools for producing holes such as drills and reamers etc.

A shaft of varying tolerance for zone can be easily manufactured. For the hole, the lower deviation is zero and minimum size is equal to the design size. The hole basis system is used for locomotive construction, machine and engine building.

SHAFT BASIS SYSTEM:

A nominal size is taken as a design size. The tolerance zone of a shaft is adopted as constant while a tolerance zone of the hole is changed above or below zero – line according to the fits as shown in figure



All Dimensions In mm

INTERFERENCE FIT

FIT:

It is defined as the degree of tightness or looseness between two mating parts. The type of fit depends upon the amount of clearance and interference.

ALLOWANCE:

An allowance may be either a positive (+) or a negative (–) according to the type of fit required. If the condition is such that the shaft is smaller than the hole, we say that there is a positive allowance, but if the shaft is larger than the hole, we say that there is a negative allowance.

TYPES OF FIT:

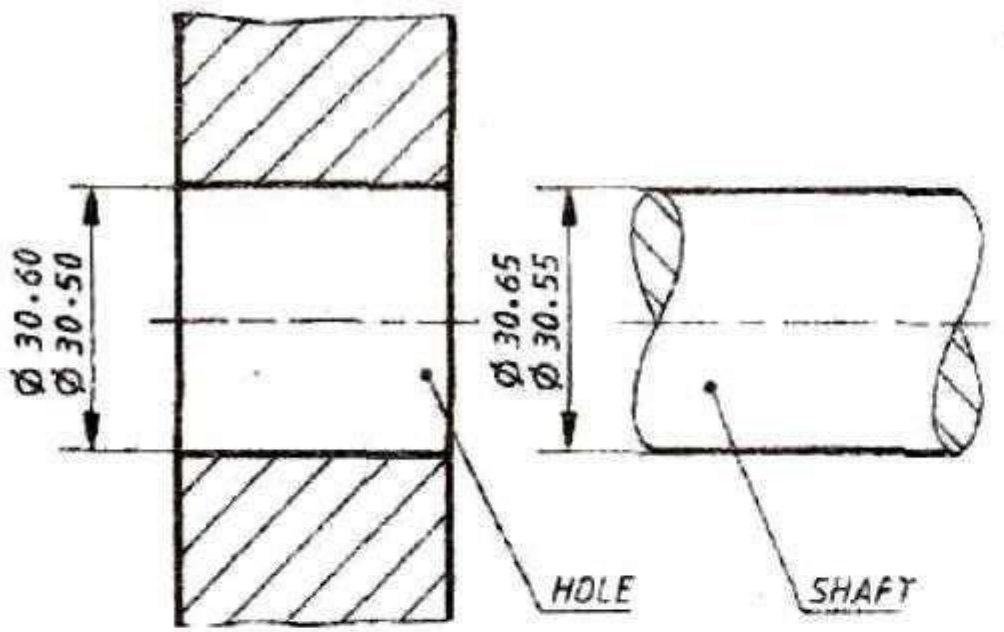
CLEARANCE FIT:

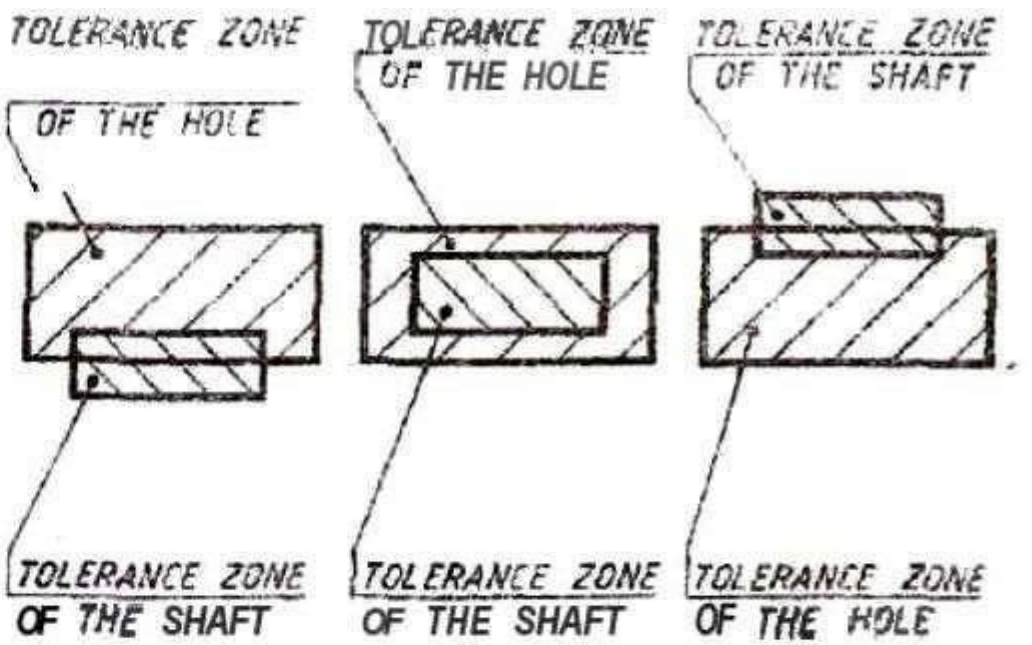
It is defined as the fit established when a positive clearance exists between the hole and the shaft. It is obtained by selecting the maximum and minimum limits of the shaft and the hole so that the clearance due to the difference between the dimensions of the smallest possible hole and the largest possible shaft is always positive. There are different classes in this type of fit depending on the clearance and the specific operating conditions of the giving mating parts. They vary with the shaft speed, shaft bearing load, lubricating oil grade, temperature and the length of the mating surfaces.

Figure shows a clearance fit. The clearance between the smallest possible hole and the largest possible shaft is = ϕ 29.95 –ϕ 29.80 = 0.05 mm. Figure shows the conventional representation of a clearance fit, where the tolerance zone of the hole lies above that of the shaft.

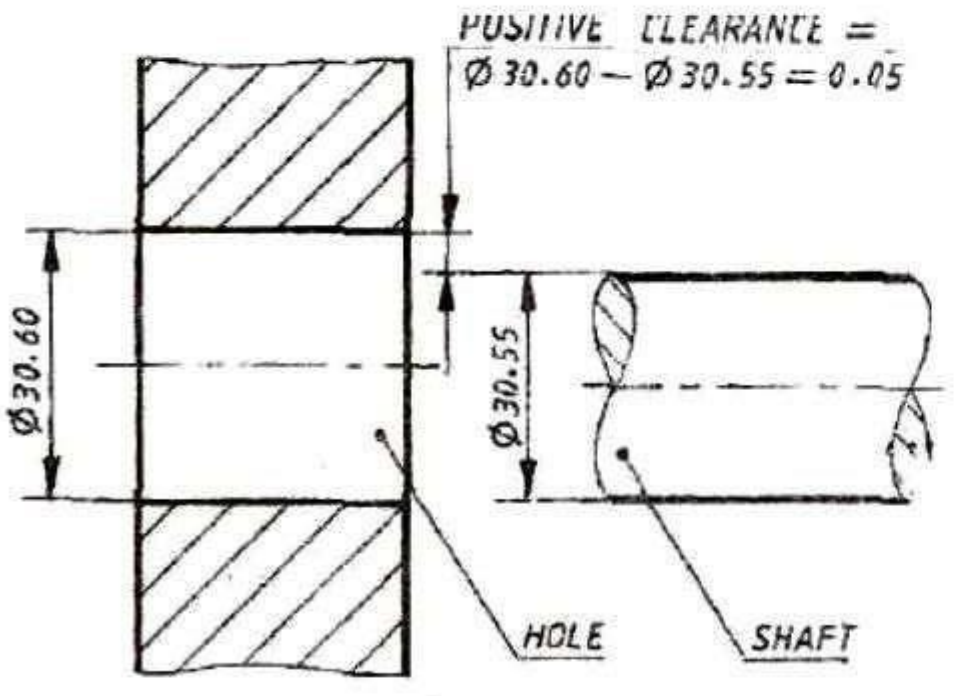
INTERFERENCE FIT:

It is defined as the fit established when a negative clearance exist between the sizes of the hole and the shaft. It is obtained by selecting the maximum and minimum limits of the shaft and the hole so that there is an interference of the surfaces and the clearance due to the



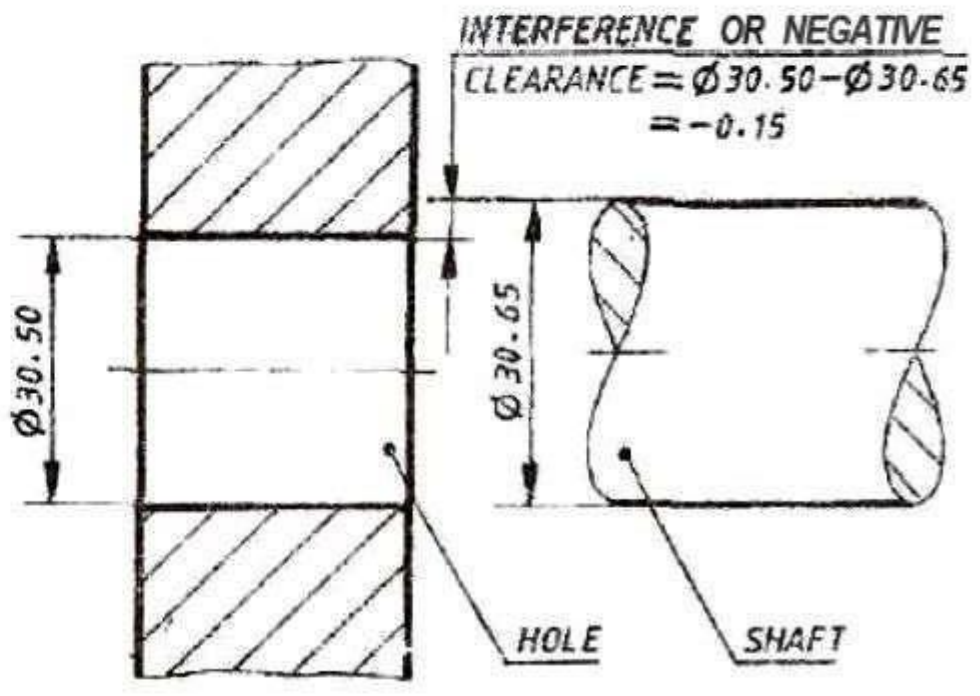


A



C

B



D

All Dimensions in mm

TRANSITION FIT

difference between the dimensions of the largest possible and the smallest possible shaft is always negative. Interference fits are obtained by several methods, for instance, a shaft may be driven into the hole with a considerable force, or heating the part having the hole in order to increase the diameter of the hole, or by cooling the shaft and thus decreasing its diameter.

Figure shows in interference fit. The difference between the dimensions of the largest possible hole and the smallest possible shaft is = ϕ30.25– ϕ30.30 = –0.05mm. Figure shows the conventional representation of an interference fit, where the tolerance zone of the hole lies entirely below that of the shaft.

The interference fit is obtained by driving a shaft into the hole with a considerable force. When the force applied is heavy the interference fit is called heavy force fit, and when a lighter force is used to drive the shaft into the hole, it is called light force fit. The interference fit can also be obtained by heating and subsequent cooling. The part containing the hole is heated so that the diameter of the hole will increase due to material expansion, and then after inserting the shaft in the hole, on cooling the hole will shrink to hold the shaft rigidly.

TRANSITION FIT

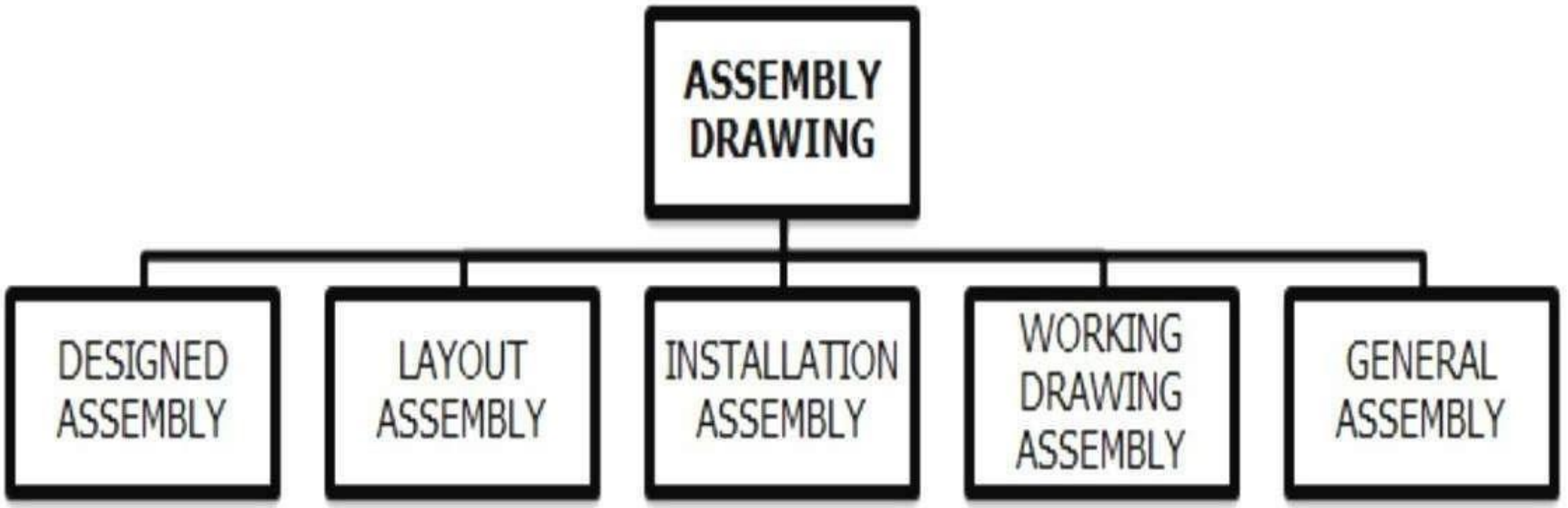
It is defined as the fit established when the dimensions of the hole and the shaft are such that there exists a positive clearance or a negative clearance when the shaft is fined into the hole. It is obtained by selecting the maximum and minimum limits for the shaft and the hole such that there exist a positive clearance when the smallest possible shaft is fitted into the largest possible hole, or a negative clearance when the largest possible shaft is forced into the smallest possible hole.

Figure A shows a transition fit, *Figure C* shows the fitting of the smallest possible shaft of ϕ 30.55 mm in the largest possible hole of ϕ30.60mm allowing a positive clearance of ϕ30.60 – ϕ30.55 = –0.05 mrn. Figure D shows the fitting of the largest possible shaft of ϕ30.65mm in the smallest possible hole of 930.50mm gives an interference fit of ϕ30.50 –

30.65 = – 0.15mm.Figure B shows conventional representation of transition fits in which the tolerance zones of the hale and the shaft overlap.

RESULT:

Thus the limits, tolerance and fits were studied.


ASSEMBLY DRAWING
DESIGNED ASSEMBLY
LAYOUT ASSEMBLY
INSTALLATION ASSEMBLY
WORKING DRAWING ASSEMBLY
GENERAL ASSEMBLY

STUDY OF ASSEMBLY DRAWINGS

EXPT NO: 7 DATE:

AIM:

To study about basics of assembly drawing.

INTRODUCTION:

A drawing which displays the parts of a machine or a machine unit assembled in their relative working positions is known as *assembly drawing*. The assembly drawing should be such that it should satisfy:

(i) Manufacturing requirements

(ii) Operational requirements

(iii) Maintenance requirements.

TYPES OF ASSEMBLY DRAWINGS

The assembly drawings are classified according to their use as shown below:

Designed Assembly –This assembly drawing is prepared at the design – stage on a larger scale.

Layout Assembly –This is an assembly drawing showing how the parts are assembled with their basic proportions (dimensions).

Installation Assembly –This is prepared for the installation or erection of a machine. This is also sometimes known as an outline assembly.

Working Drawing Assembly – A complete set of working drawings of a machine comprises of detailed drawings, giving all necessary information for the production of individual parts and assembly drawing showing the location of each part. The assembly

drawing should be ready before the detailed drawings are accepted as finished and the blue-prints are made.

General Assembly –It comprises of the detailed drawings of the individual parts, subassembly and the assembly drawings of the machine.

ACCEPTED NORMS TO BE OBSERVED FOR ASSEMBLY DRAWINGS:

Selection of Views –The main or important view which is usually in section should show all the individual parts and their relative locations. Additional views are shown only when they add necessary information.

Sectioning –The parts should be sectioned according to the requirements (i.e. half section or partial section) to show important assembly details. Code of the BIS for general engineering drawings must be observed (refer to SP:46-1988).

Dotted Lines –The dotted lines should be omitted from the assembly drawing when a proper section is taken. If the view of a part is drawn by the half-section, then in un section portion of the view, the dotted lines may be drawn to clarify details of the part.

Dimensions–The overall dimensions and centre-to-centre distances showing the relationship of parts to the machine as a whole are sometimes shown. Detailed dimensions are given on working assembly drawings when the detailed drawings are not prepared.

Bill of Materials –Each part of the machine's identified on assembly drawing by the leader line and number which are used in the detail drawing and in the bill ofmaterial. The height of the number may be approximately 5 mm and encircled by9 mm diameter leader lines are drawn radially touching the respective parts.

The bill of materials also shows the following:

- Number of parts
- Material of parts required for one unit
- Standard norm for standard components and checked by
- Scale

- ❖ Method of projection
- ❖ Shop processes
- ❖ Name of the company
- ❖ Designed by, drawn by
- ❖ Any special remark.

SEQUENCES OF PREPARING THE ASSEMBLY DRAWING:

- ❖ Study functional requirements of each component and their interrelationship learn the actual working of a machine.
- ❖ Study carefully, the views of each component in the detailed drawing and decide the relative location of each part for the proper functioning of the machine.
- ❖ Decide the mating dimensions between two components which are required to be assembled.
- ❖ Prepare free-hand sketch of the main view or an .important view (generally front elevation). Add, additional views, if necessary.
- ❖ Select a suitable scale for the entire assembly drawing.
- ❖ Layout the views of the 'assembly drawing so that it becomes easier to' understand.
- ❖ Prepare the bill of materials.
- ❖ Label each component by the leader-line and number it.
- ❖ Show overall dimensions.
- ❖ Draw the section-lines according to the convention adopted (refer to .SP:46-1988).
- ❖ Use proper grade of pencil to make the, drawing fair; maintain uniform thickness of lines and follow SP: 46-1 988.
- ❖ Show required fits and tolerances between the two mating components.

RESULT:

The basic of assembly drawing was studied.

STUDY OF DRAFTING SOFTWARE (AUTOCAD)

EXPT NO: 8 DATE:

AIM:

To study the AUTOCAD Software.

Sl.No	Command	Description
1.	OPEN	Opens an existing drawing file
2.	ARC	Creates an arc
3.	ARRAY	Creates multiple copies of objects in a pattern
4.	BHATCH	Fills an enclosed area or selected objects with a hatch pattern
5.	BLOCK	Creates a block definition from objects you select
6.	BREAK	Erase parts of object or splits an object in two
7.	CHAMFER	Bevels the edges of object
8.	CHANGE	Changes the properties of existing objects
9.	CIRCLE	Creates a circle
10.	COLOR	Defines color for new objects
11.	COPY	Duplicates objects
12.	DIVIDE	Places evenly spaced point objects or blocks along the length or perimeter of an object
13.	DONUT	Draws filled circles and rings
14.	ELLIPSE	Creates an ellipse or an elliptical arc
15.	ERASE	Removes objects from a drawing
16.	HATCH	Fills a specified boundary with a pattern
17.	HATCHEDIT	Modifies an existing hatch object
18.	EXTEND	Extends an object to meet another object
19.	INSERT	Places a named block or drawing into the current drawing
20.	LAYER	Manages layers and layer properties
21.	LINE	Creates straight line segments
22.	LINETYPE	Creates, loads, and set line types
23.	OFFSET	Creates concentric circles, parallel lines, and parallel curves
24.	FILLET	Rounds and fillets the edges of objects
25.	MIRROR	Creates a mirror image copy of objects
26.	MOVE	Displaces objects a specified distance in a specified direction
27.	MSLIDE	Creates a slide file of the current view port in model space, or of all view ports in paper space.

28.	LTSCALE	Sets the line type scale factor
29.	PAN	Moves the drawing display in the current view port
30.	OOPS	Restores erased objects
31.	PLINE	Creates two-dimensional polylines
32.	POINT	Creates a point object
33.	POLYGON	Creates an equilateral closed polyline
34.	PROPERTIES	Controls properties of existing objects
35.	MTEXT	Multiline text
36.	ORTHO	Constrains cursor movement
37.	OSNAP	Sets object snap modes
38.	REDRAW	Refreshes the display in the current view port
39.	REGEN	Regenerates the drawing and refreshes the current view port
40.	ROTATE	Rotate
41.	SCALE	Enlarges or reduces selected objects equally in the X,Y, and Z directions
42.	SCRIPT	Executes a sequence of commands from a script
43.	SKETCH	Creates a series of freehand line segments
44.	SPLINE	Creates a quadratic or cubic spine (NURBS) curve
45.	TEXT	Displays text on screen as it is entered
46.	UNDO	Reverse the effect of commands
47.	ZOOM	Increases or decreases the apparent size of objects in the current view port
48.	AREA	Calculates the area and perimeter of objects or of defined areas
49.	LTSCALE	Sets the line type scale factor
50.	BACKGROUND	Sets up the background for your scene
51.	BASE	Sets the insertion base point for the current drawing
52.	BLIPMODE	Controls the displays of marker blips
53.	BLOCKICON	Generates preview images for blocks created with release 14 or earlier
54.	CHPROP	Changes the color, layer, line type, scale factor, line weight, thickness, and plot style of an object
55.	CLOSE	Closes the current drawing
56.	DBLIST	Lists database information for each object in the drawing
57.	DDEDIT	Edits text and attribute definitions
58.	DDPTYPE	Specifies the display mode and size of point objects

59.	DELAY	Provides a timed pause within a script
60.	DIM AND DIM	Accesses dimensioning mode
61.	DIMALIGNED	Creates an aligned linear dimension
62.	DIMANGULAR	Creates an angular dimension
63.	DIMBASELINE	Creates a linear, angular, or ordinate dimension from the baseline of the pervious dimension or a selected dimension
64.	DIMDIAMETER	Creates diameter dimensions for circles and arcs
65.	DIMEDIT	Edit dimensions
66.	DIMLINEAR	Creates linear dimension
67.	DIMORDINATE	Creates ordinate point dimensions
68.	DIMOVERRIDE	Overrides dimension system variables
69.	DIMRADIUS	Creates ordinate point dimensions
70.	DIMSTYLE	Creates and modifies dimension styles
71.	DIST	Measures the distance and angle between two points
72.	DWGPROPS	Sets and displays the properties of the current drawing
73.	FILL	Controls the filling of multi-lines, traces, solids, all hatches and wide polylines
74.	FILTER	Creates reusable filters to select objects based on properties
75.	ID	Displays the coordinate values of a location
76.	LIST	Displays database information for selected objects
77.	MASSPROP	Calculate and displays the mass properties of regions or solids
78.	MENU	Loads a menu file
79.	MENULOAD	Loads partial menu files
80.	MENUUNLOAD	Unloads partial menu files
81.	OPTIONS	Customizes the AutoCAD settings
82.	PLAN	Displays the plan view of a user coordinate system
83.	PLOT	Plots a drawing to a plotting device of file
84.	SHADEMODE	Shades the objects in the current view port
85.	SNAP	Restricts cursor movement to specified intervals
86.	SPELL	Checks spelling in a drawing
87.	VLISP	Displays the Visual LISP interactive development environment(IDE)

RESULT:

Thus the AutoCAD software was studied.

BASICS OF 2D DRAWING - AUTOCAD

EXPT NO: 9 DATE:

AIM:

1. To understand drawing standards

2. Draw basic sketches and constraint them using Geometrical and Dimensional constraints

PROCEDURE:

1. Open Auto CAD; Draw the Shape given in the Fig.1.

2. Add relations and Smart dimensions and make sure that the Sketch is Fully constraint

3. Change the dimensions according to Fig.1

4. Repeat the same for Fig.2 to Fig.6

Commands used:

Line, Circle, Arc, Fillet, Trim, Smart Dimension, Relations, Show, View

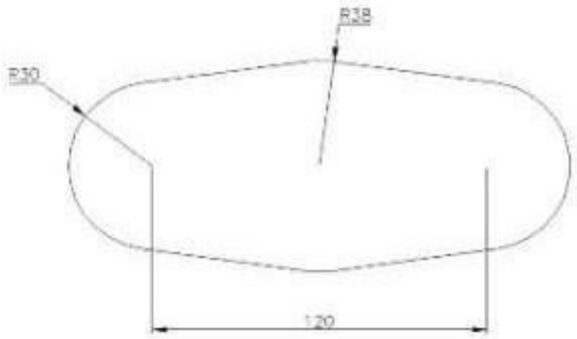


Fig.1

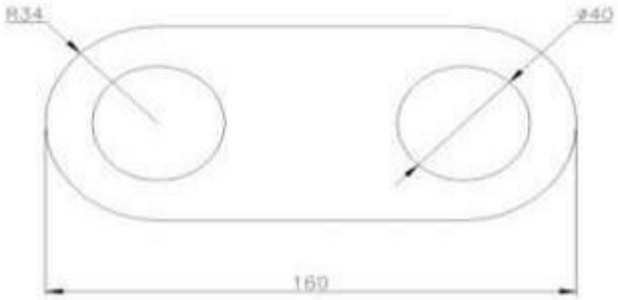


Fig.2

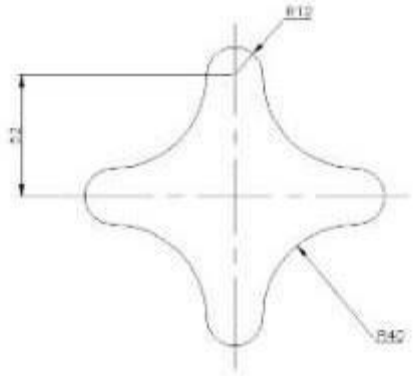


Fig.3

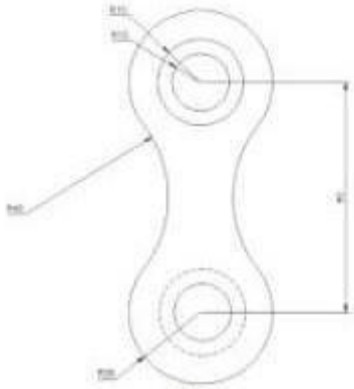


Fig.4

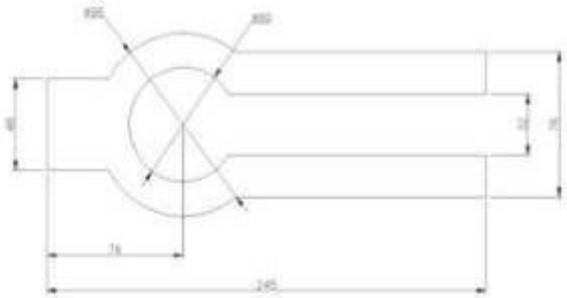


Fig.5

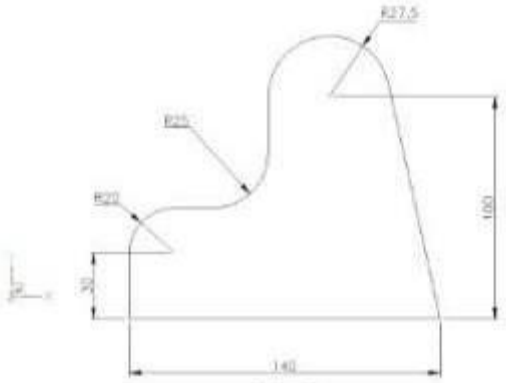


Fig.6

RESULT:

The basic sketches were drawn using AUTOCAD as shown in Figures and the required parameters were added to modify the dimensions at later stage if necessary.

BUSH BEARING

EXPT. NO: 10 DATE:

AIM:

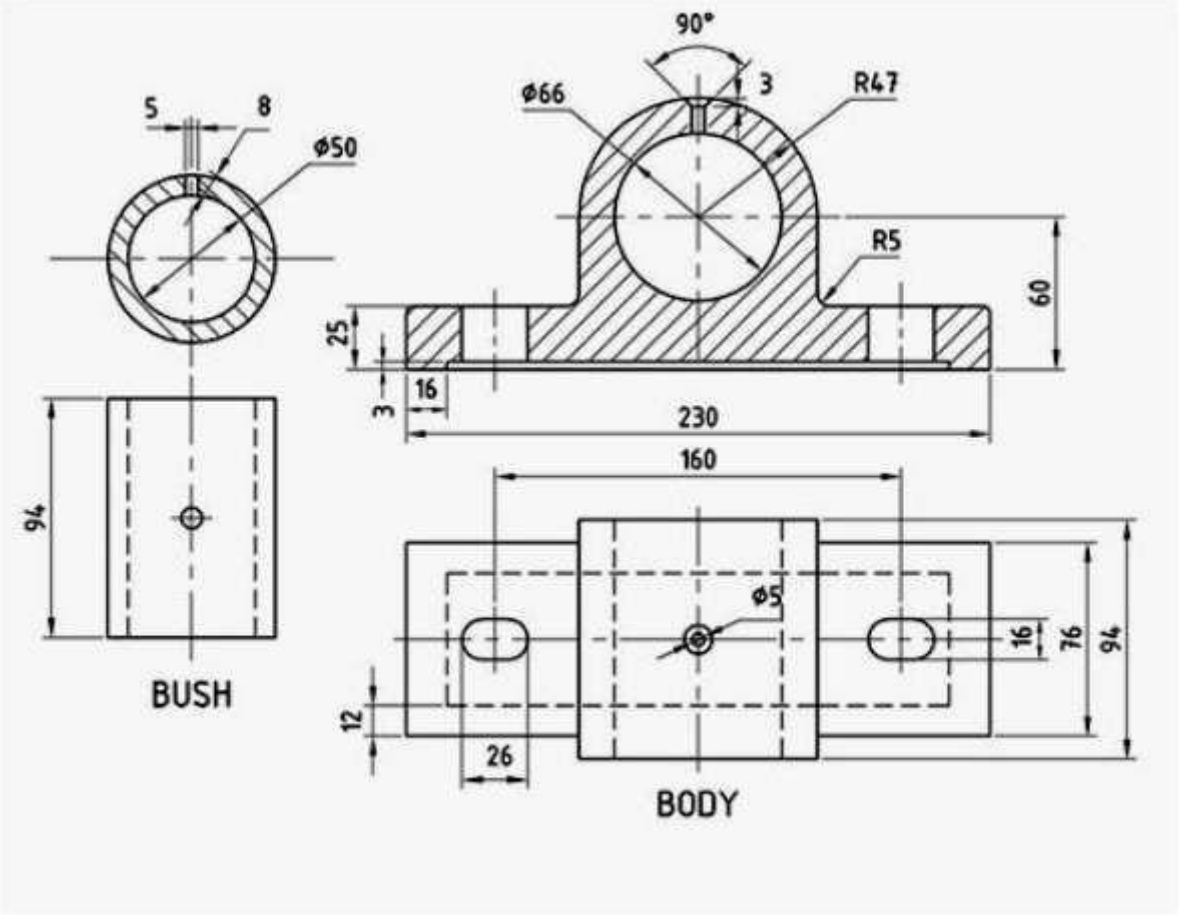
To create 2D models of Bush Bearing parts using AutoCAD.

PROCEDURE :

1. The drawings of Bush Bearing are studied.
2. 2D models of Bush Bearing are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



RESULT:

The 2D models of Bush Bearing parts are created using Auto CAD.

SAFETY VALVE

EXPT. NO: 11 DATE:

AIM:

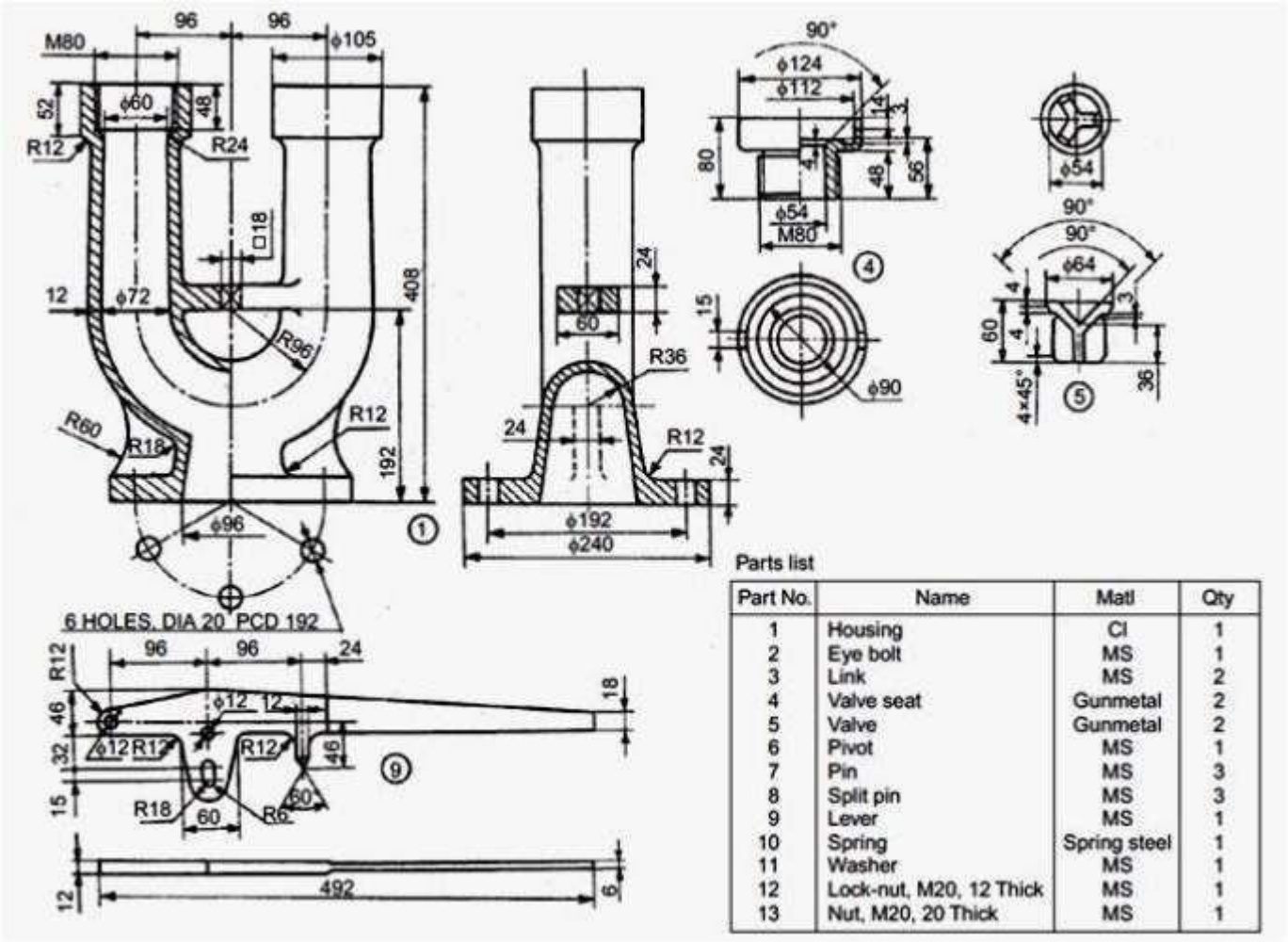
To create 2D models of Safety Valve parts using AutoCAD.

PROCEDURE :

1. The drawings of Safety Valve are studied.
2. 2D models of Safety Valve are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Part No.	Name	Matl	Qty
1	Housing	CI	1
2	Eye bolt	MS	1
3	Link	MS	2
4	Valve seat	Gunmetal	2
5	Valve	Gunmetal	2
6	Pivot	MS	1
7	Pin	MS	3
8	Split pin	MS	3
9	Lever	MS	1
10	Spring	Spring steel	1
11	Washer	MS	1
12	Lock-nut, M20, 12 Thick	MS	1
13	Nut, M20, 20 Thick	MS	1

RESULT:

The 2D models of Safety Valve parts are created using Auto CAD.

NON RETURN VALVE

EXPT. NO: 12 DATE:

AIM:

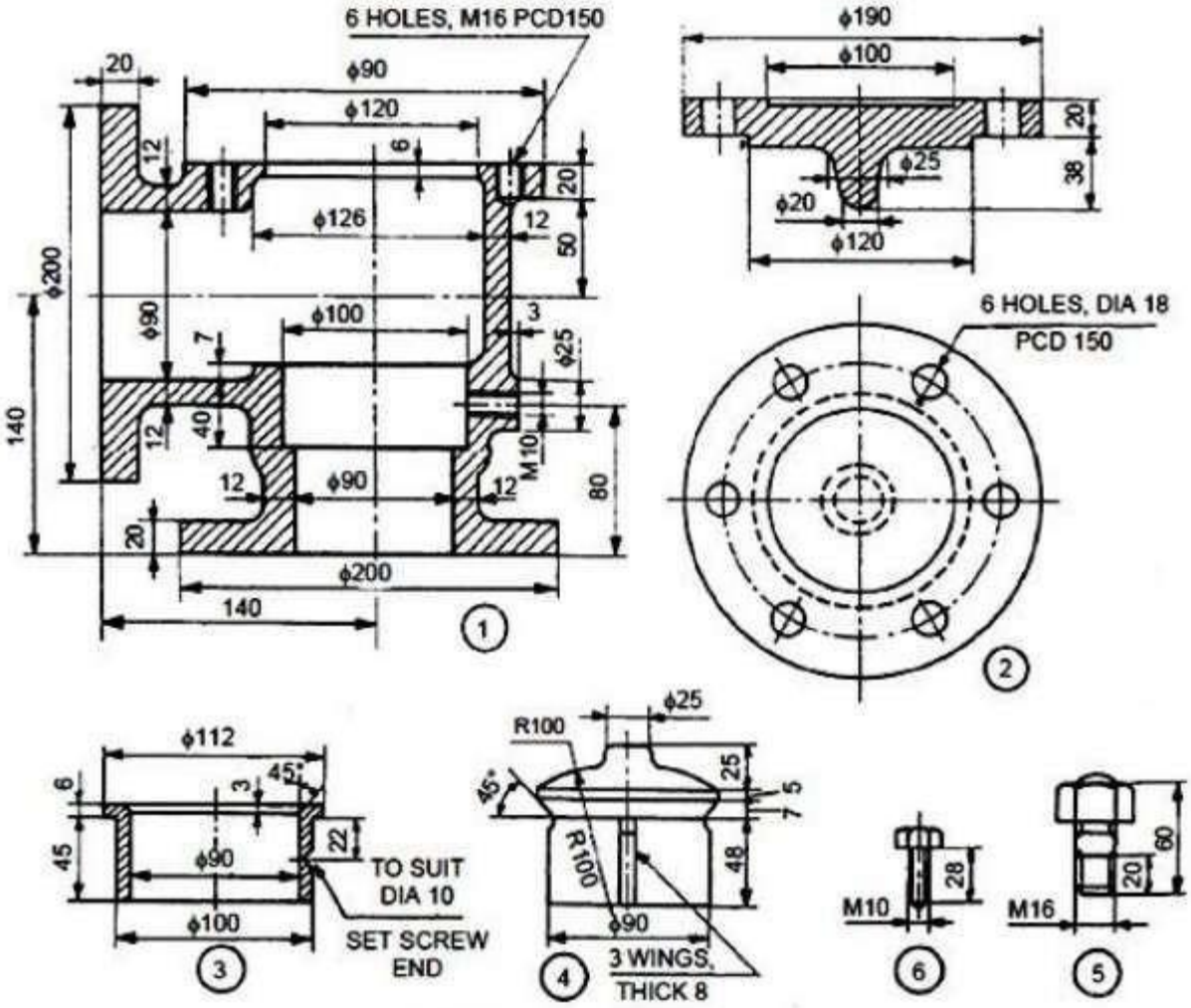
To create 2D models of Non Return Valve parts using AutoCAD.

PROCEDURE :

1. The drawings of Non Return Valve are studied.
2. 2D models of Non Return Valve are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

No.	Name	Matl	Qty
1	Body	Brass	1
2	Cover	Brass	1
3	Valve seat	Bronze	1
4	Valve	Brass	1
5	Stud with nut	MS	6
6	Set screw	MS	1

RESULT:

The 2D models of Non Return Valve parts are created using Auto CAD.

FLANGE COUPLING

EXPT. NO: 13 DATE:

AIM:

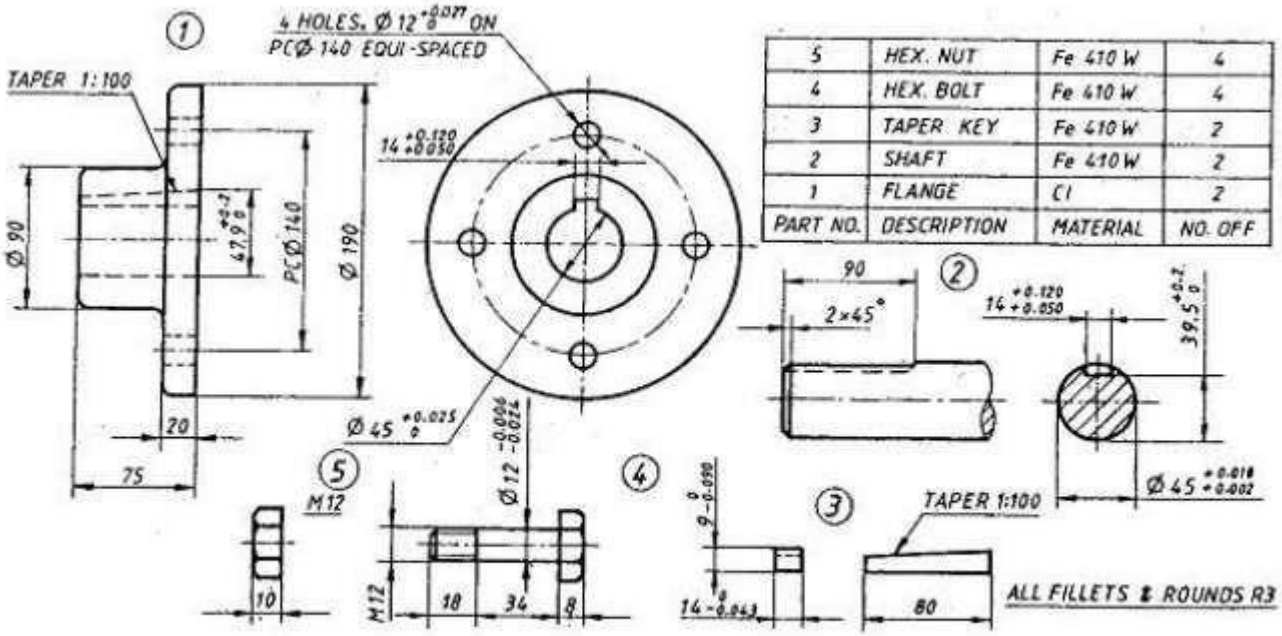
To create 2D models of Flange Coupling parts using AutoCAD.

PROCEDURE :

1. The drawings of Flange Coupling are studied.
2. 2D models of Flange Coupling are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



PART NO.	DESCRIPTION	MATERIAL	NO. OFF
5	HEX. NUT	Fe 410 W	4
4	HEX. BOLT	Fe 410 W	4
3	TAPER KEY	Fe 410 W	2
2	SHAFT	Fe 410 W	2
1	FLANGE	CI	2

All Dimensions in mm

Details of Flanged Coupling — Unprotected Type

RESULT:

The 2D models of Flange Coupling parts are created using Auto CAD.

OLDHAM COUPLING

EXPT. NO: 14 DATE:

AIM:

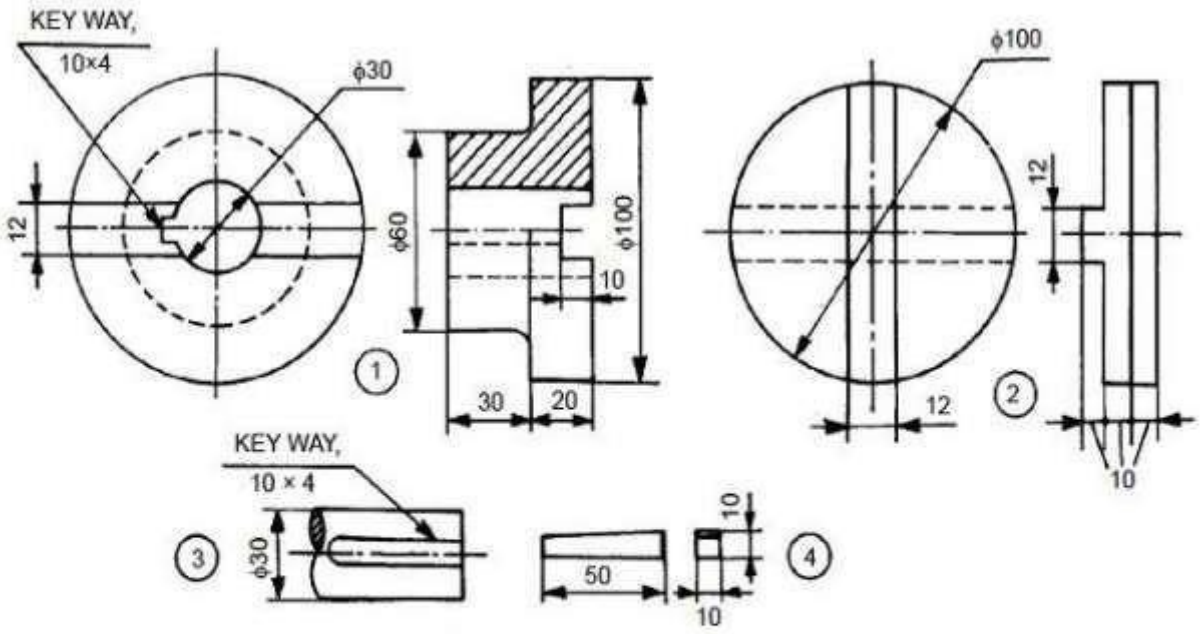
To create 2D models of Oldham Coupling parts using AutoCAD.

PROCEDURE :

1. The drawings of Oldham Coupling are studied.
2. 2D models of Oldham Coupling are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Sl. No.	Name	Matl.	Qty.
1	Flange	MS	2
2	Disc	MS	1
3	Shaft	MS	2
4	Key	MS	2

RESULT:

The 2D models of Oldham Coupling parts are created using Auto CAD.

MUFF COUPLING

EXPT. NO: 15 DATE:

AIM:

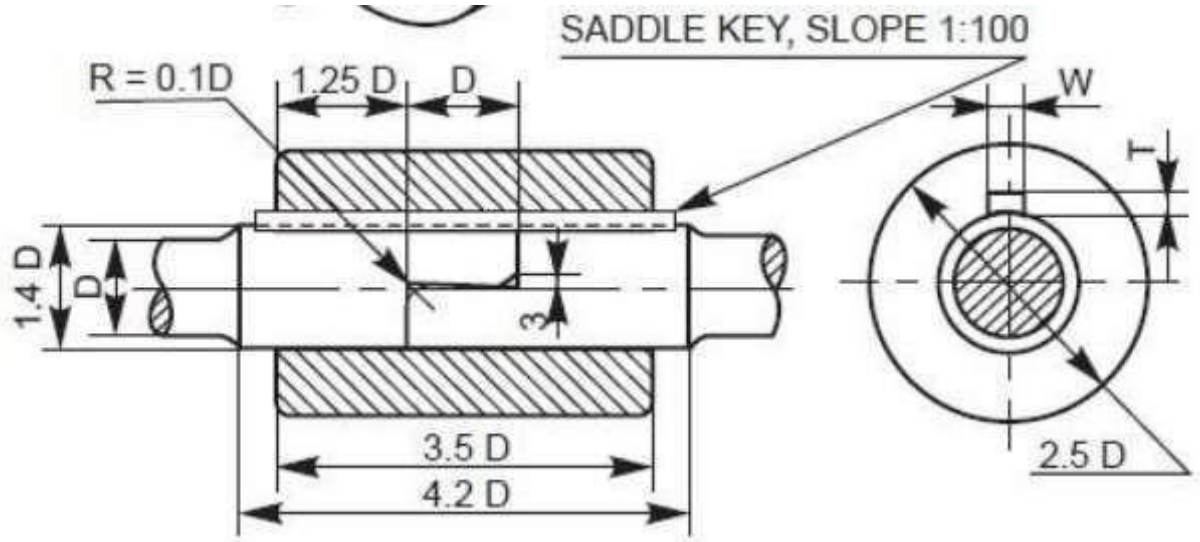
To create 2D models of Muff Coupling parts using AutoCAD.

PROCEDURE :

1. The drawings of Muff Coupling are studied.
2. 2D models of Muff Coupling are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



RESULT:

The 2D models of Muff Coupling parts are created using Auto CAD.

UNIVERSAL JOINT

EXPT. NO: 16 DATE:

AIM:

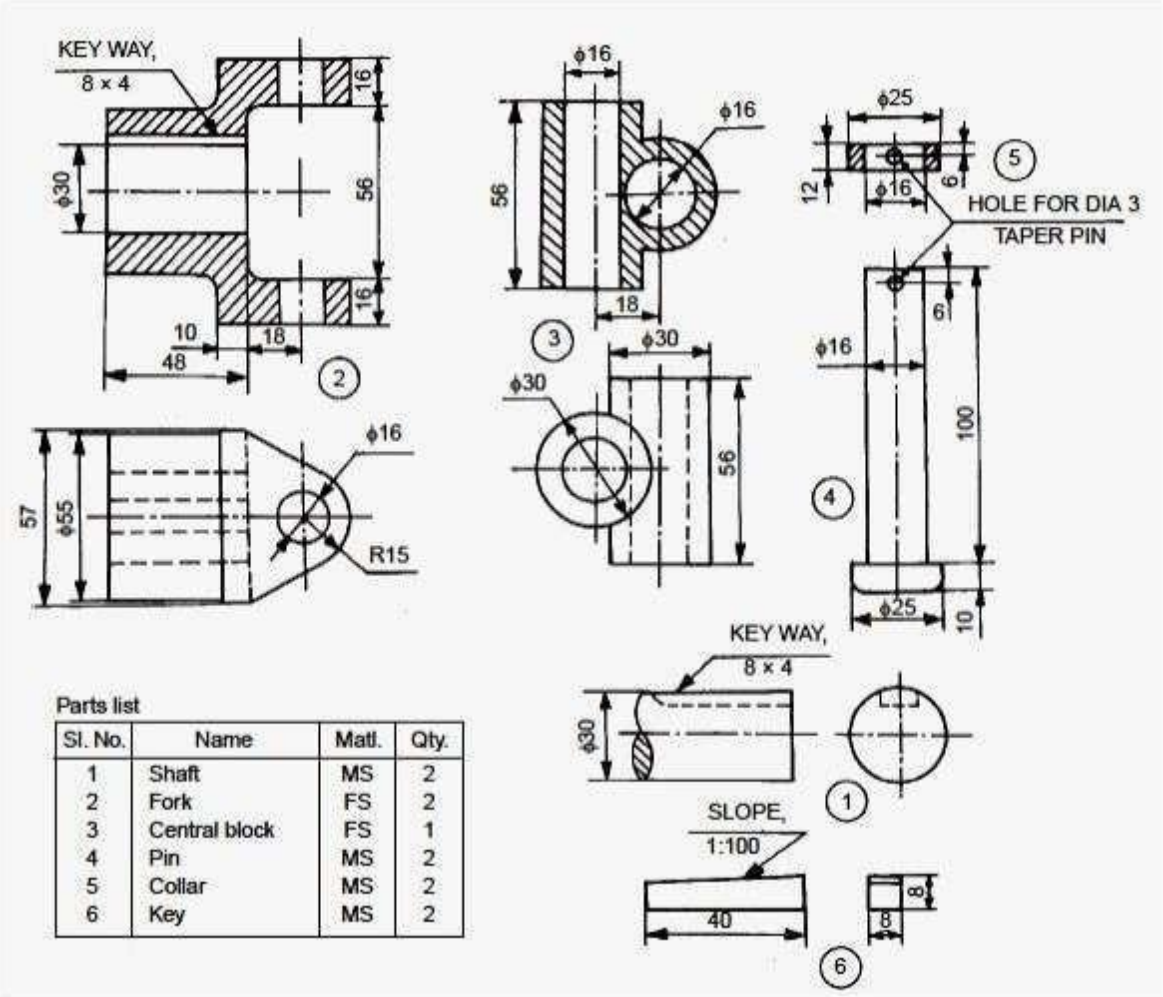
To create 2D models of Universal Joint parts using AutoCAD.

PROCEDURE :

1. The drawings of Universal Joint are studied.
2. 2D models of Universal Joint are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Sl. No.	Name	Matl.	Qty.
1	Shaft	MS	2
2	Fork	FS	2
3	Central block	FS	1
4	Pin	MS	2
5	Collar	MS	2
6	Key	MS	2

RESULT:

The 2D models of Universal Joint parts are created using Auto CAD.

KNUCKLE JOINT

EXPT. NO: 17 DATE:

AIM:

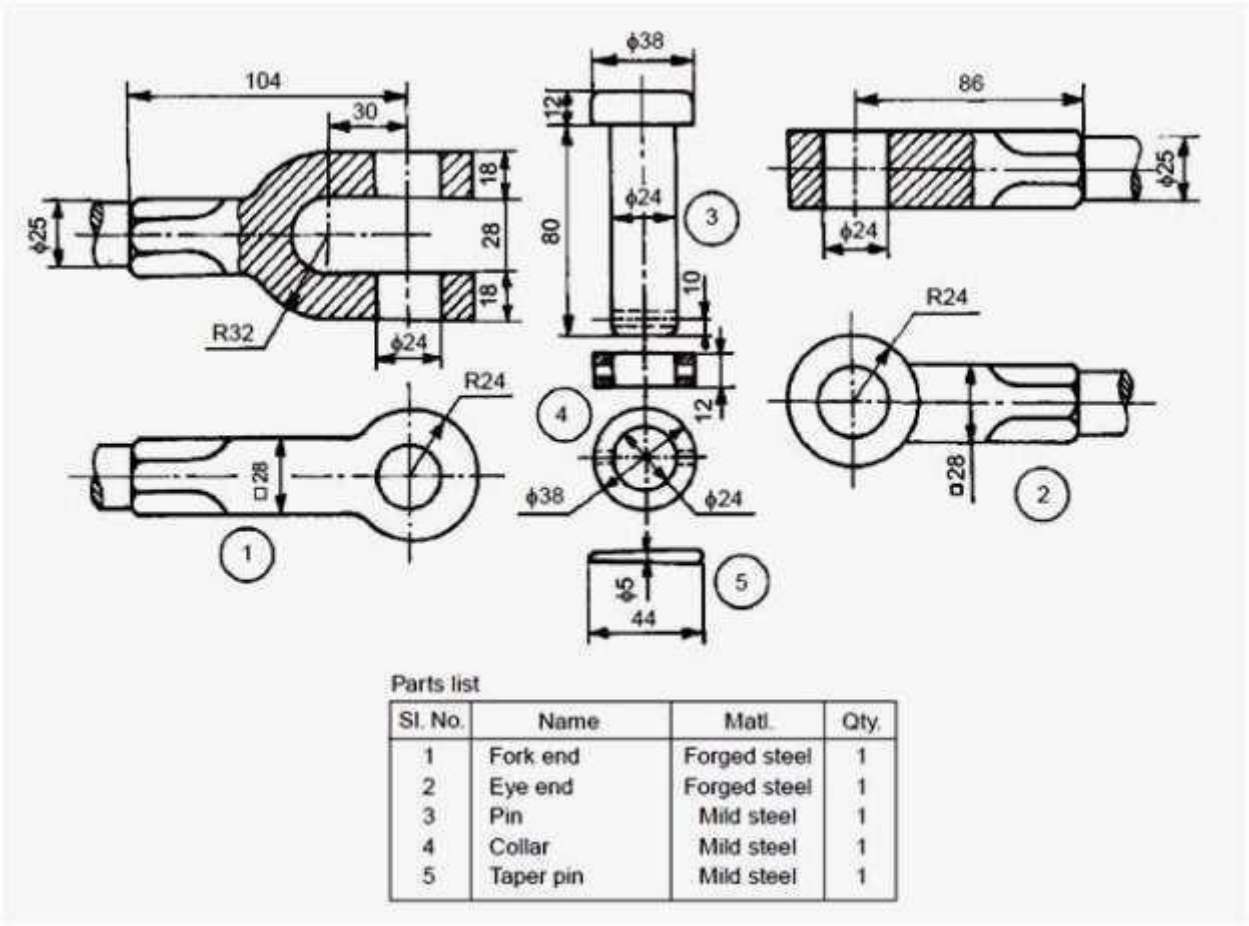
To create 2D models of Knuckle Joint parts using AutoCAD.

PROCEDURE :

1. The drawings of Knuckle Joint are studied.
2. 2D models of Knuckle Joint are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Sl. No.	Name	Matl.	Qty.
1	Fork end	Forged steel	1
2	Eye end	Forged steel	1
3	Pin	Mild steel	1
4	Collar	Mild steel	1
5	Taper pin	Mild steel	1

RESULT:

The 2D models of Knuckle Joint parts are created using Auto CAD.

GIBB AND COTTER JOINT

EXPT. NO: 18 DATE:

AIM:

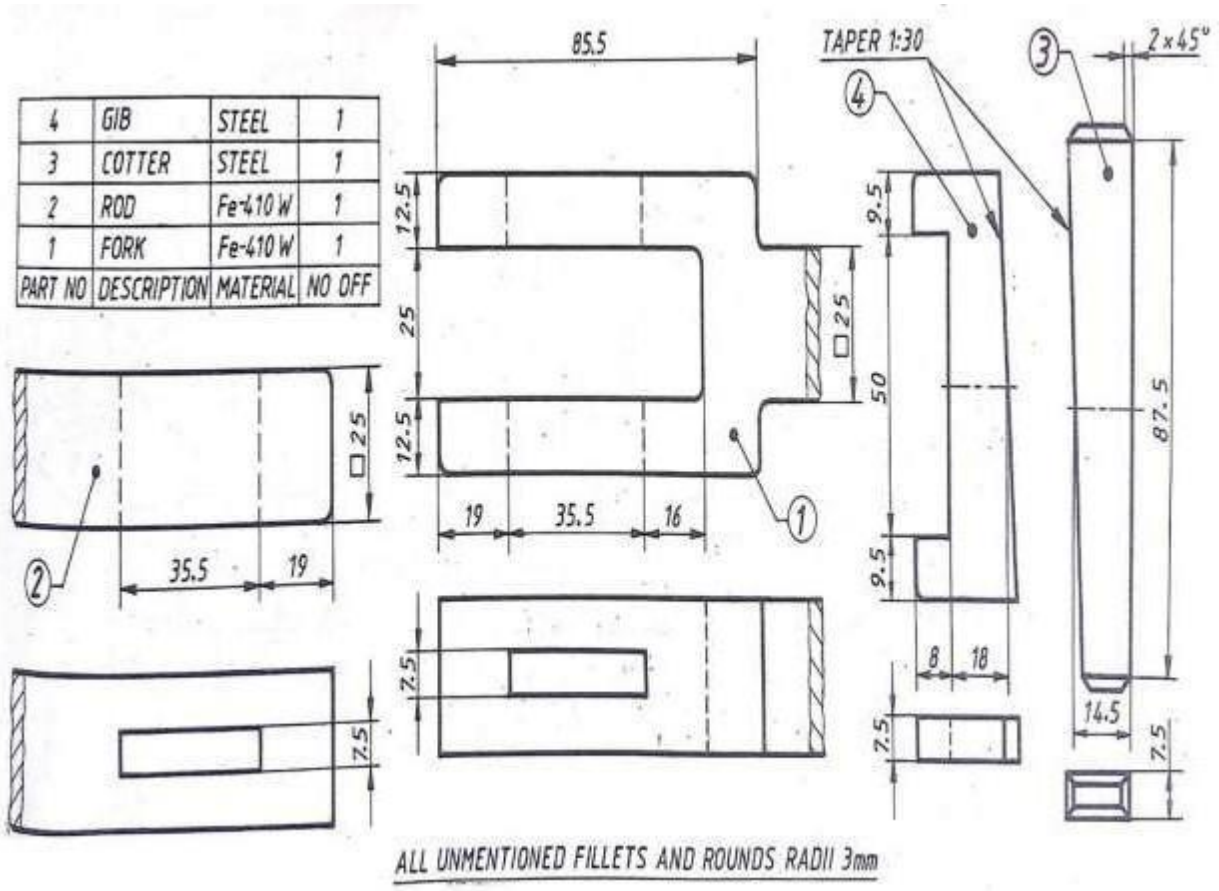
To create 2D models of Gibb and Cotter Joint parts using AutoCAD.

PROCEDURE :

1. The drawings of Gibb and Cotter Joint are studied.
2. 2D models of Gibb and Cotter Joint are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



RESULT:

The 2D models of Gibb and Cotter Joint parts are created using Auto CAD.

SLEEVE AND COTTER JOINT

EXPT. NO: 19 DATE:

AIM:

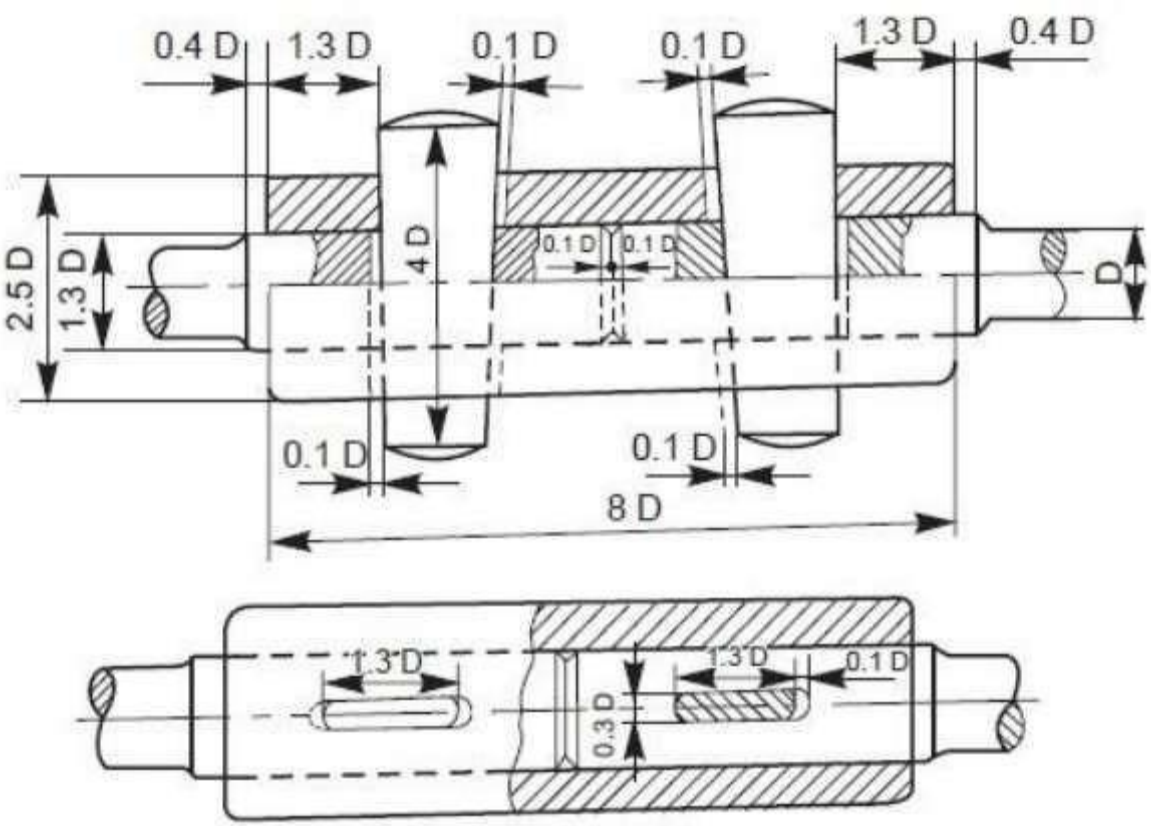
To create 2D models of Sleeve and Cotter Joint parts using AutoCAD.

PROCEDURE :

1. The drawings of Sleeve and Cotter Joint are studied.
2. 2D models of Sleeve and Cotter Joint are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



RESULT:

The 2D models of Sleeve and Cotter Joint parts are created using Auto CAD.

PISTON

EXPT. NO: 20 DATE:

AIM:

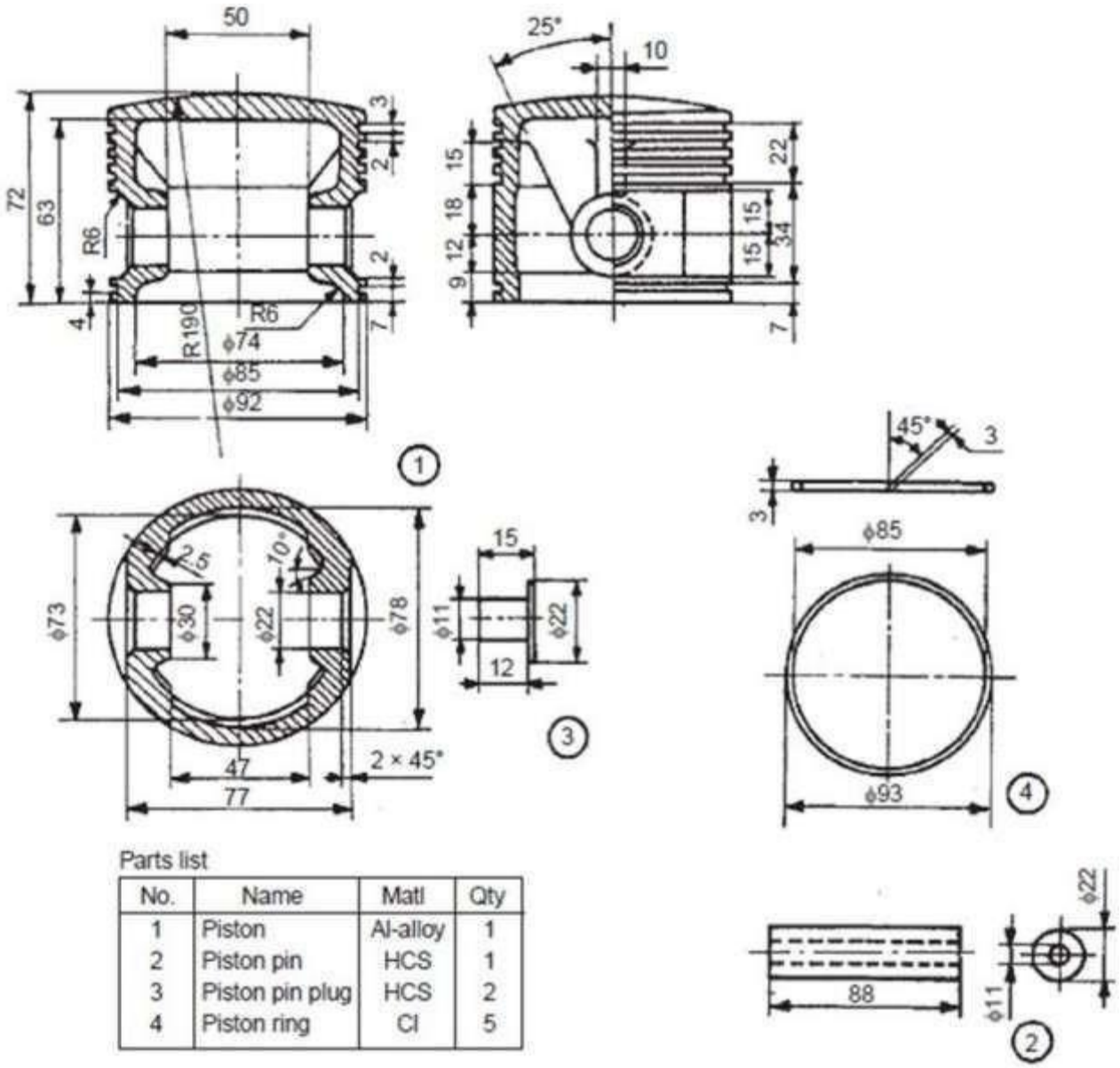
To create 2D models of Piston parts using AutoCAD.

PROCEDURE :

1. The drawings of Piston are studied.
2. 2D models of Piston are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

No.	Name	Matl	Qty
1	Piston	Al-alloy	1
2	Piston pin	HCS	1
3	Piston pin plug	HCS	2
4	Piston ring	CI	5

RESULT:

The 2D models of Piston parts are created using Auto CAD.

CONNECTING ROD

EXPT. NO: 21 DATE:

AIM:

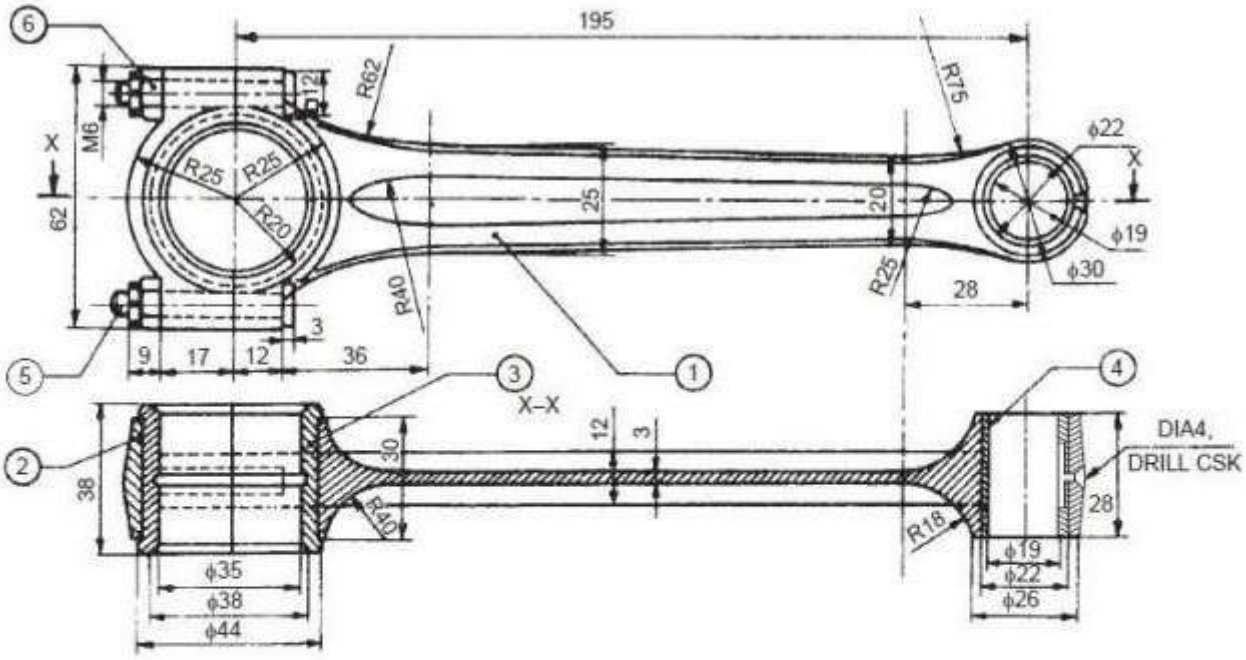
To create 2D models of Connecting Rod parts using AutoCAD.

PROCEDURE :

1. The drawings of Connecting Rod are studied.
2. 2D models of Connecting Rod are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Part No.	Name	Matl.	Qty.
1	Rod	FS	1
2	Cap	FS	1
3	Bearing brass	GM	2
4	Bearing bush	P Bronze	1
5	Bolt	MCS	2
6	Nut	MCS	2

RESULT:

The 2D models of Connecting Rod parts are created using Auto CAD.

CROSSHEAD

EXPT. NO: 22 DATE:

AIM:

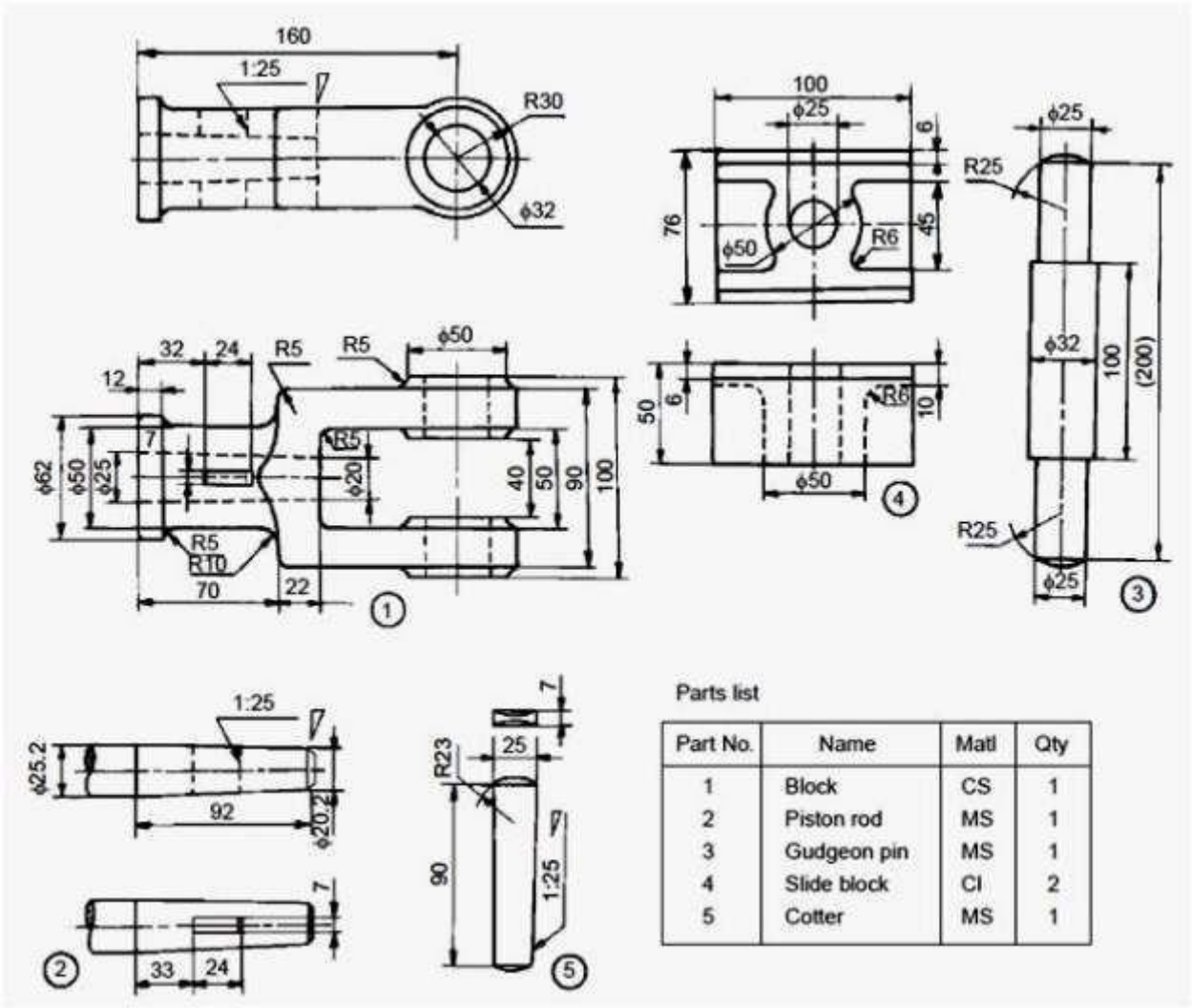
To create 2D models of Crosshead parts using AutoCAD.

PROCEDURE :

1. The drawings of Crosshead are studied.
2. 2D models of Crosshead are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Part No.	Name	Matl	Qty
1	Block	CS	1
2	Piston rod	MS	1
3	Gudgeon pin	MS	1
4	Slide block	CI	2
5	Cotter	MS	1

RESULT:

The 2D models of Crosshead parts are created using Auto CAD.

STUFFING BOX

EXPT. NO: 23 DATE:

AIM:

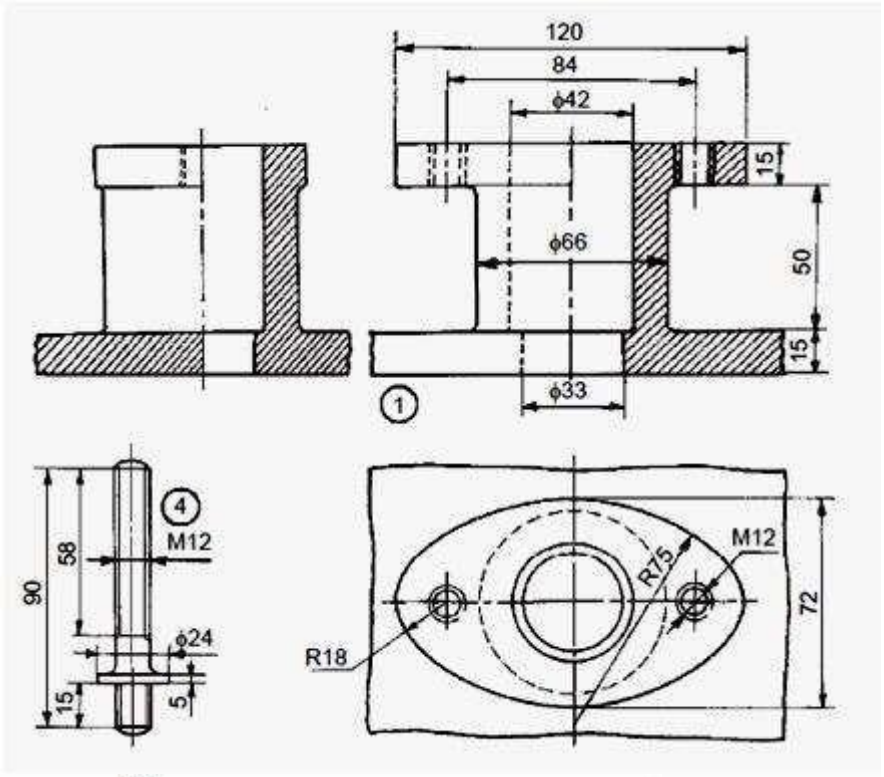
To create 2D models of Stuffing Box parts using AutoCAD.

PROCEDURE :

1. The drawings of Stuffing Box are studied.
2. 2D models of Stuffing Box are created using AutoCAD.

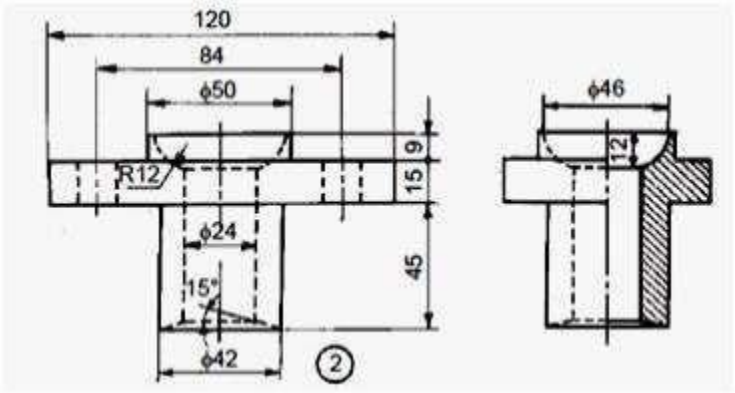
COMMANDS USED:

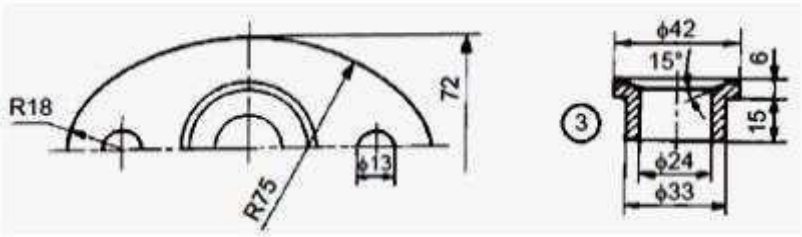
Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Gland	Brass	1
3	Bush	Brass	1
4	Stud	MS	2
5	Nut, M12	MS	2





RESULT:

The 2D models of Stuffing Box parts are created using Auto CAD.

SCREW JACK

EXPT. NO: 24 DATE:

AIM:

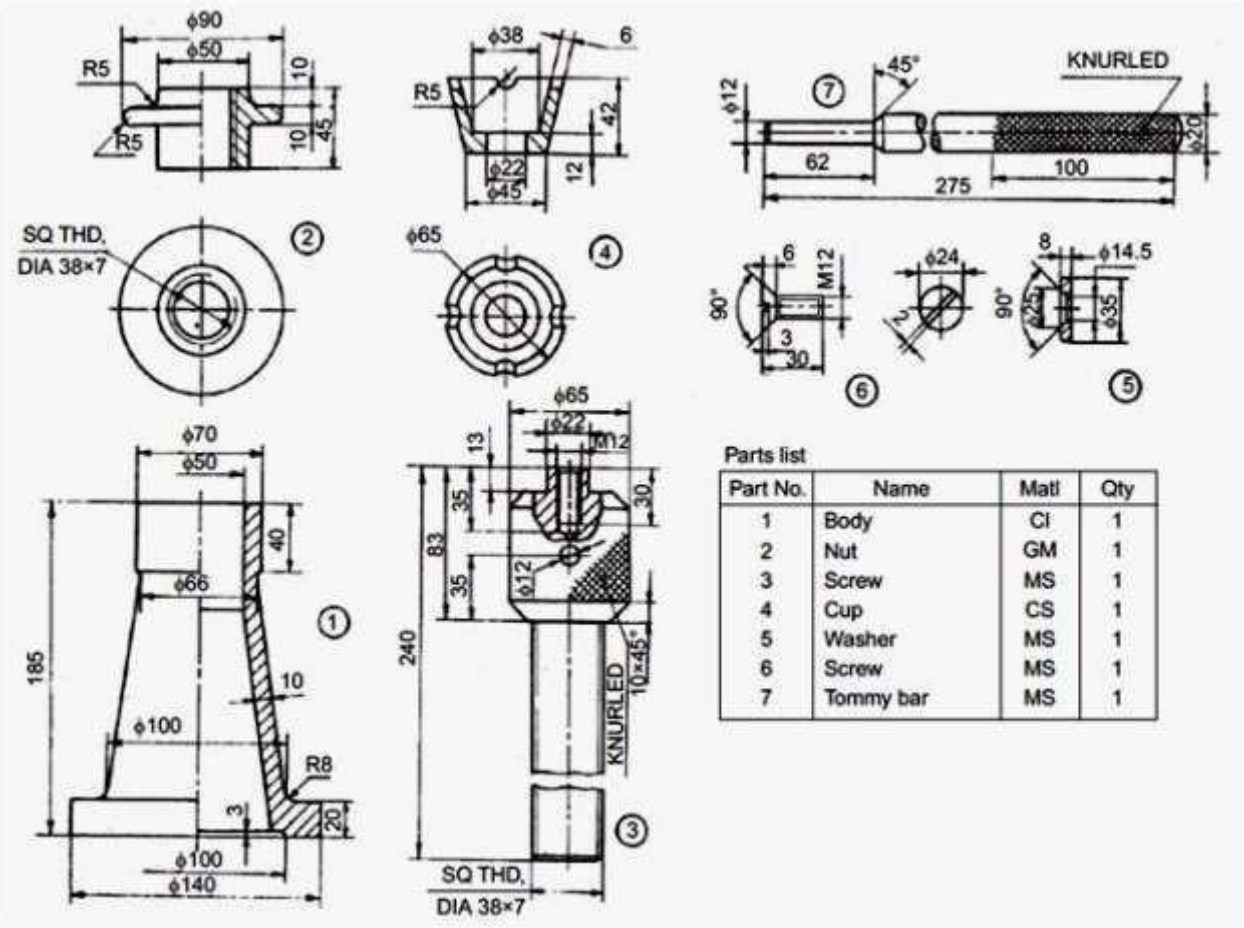
To create 2D models of Screw Jack parts using AutoCAD.

PROCEDURE :

1. The drawings of Screw Jack are studied.
2. 2D models of Screw Jack are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Nut	GM	1
3	Screw	MS	1
4	Cup	CS	1
5	Washer	MS	1
6	Screw	MS	1
7	Tommy bar	MS	1

RESULT:

The 2D models of Screw Jack parts are created using Auto CAD.

MACHINE VICE

EXPT. NO: 25 DATE:

AIM:

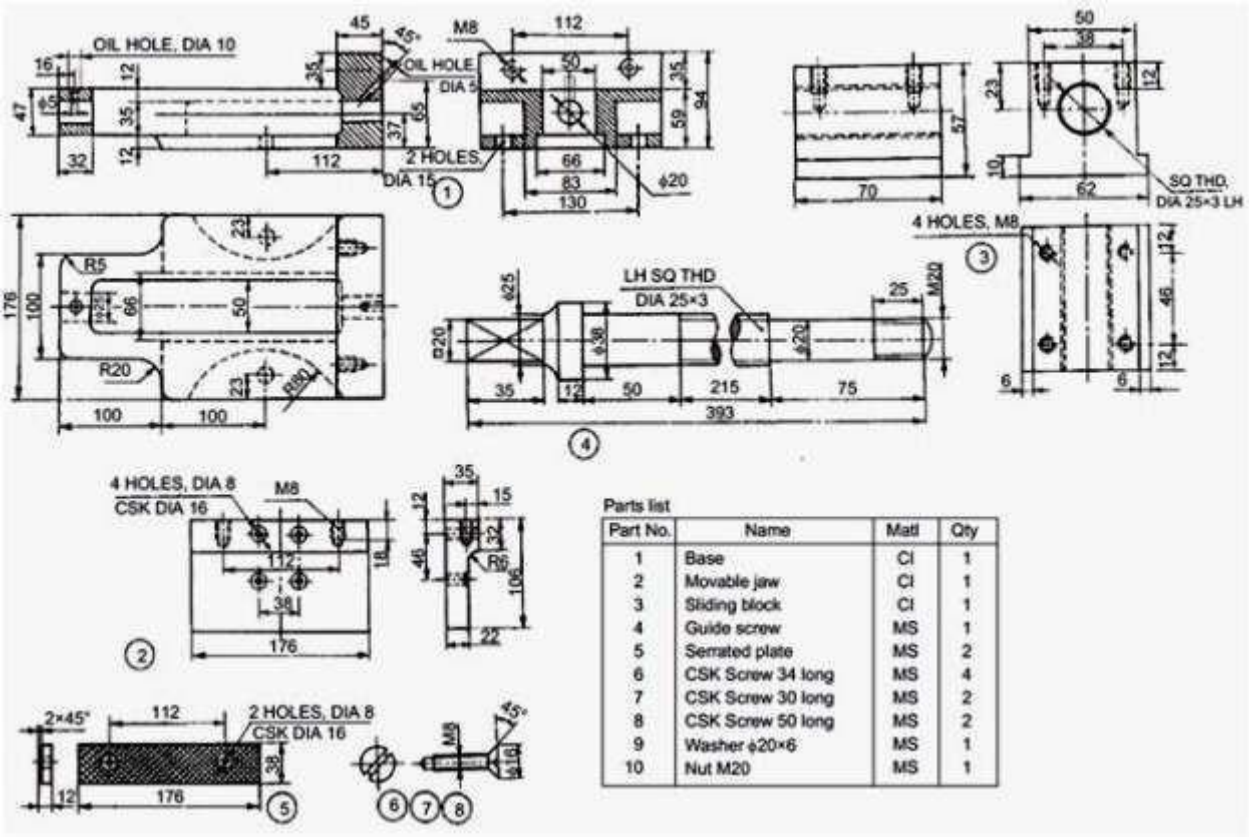
To create 2D models of Machine Vice parts using AutoCAD.

PROCEDURE :

1. The drawings of Machine Vice are studied.
2. 2D models of Machine Vice are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Part No.	Name	Matl	Qty
1	Base	CI	1
2	Movable jaw	CI	1
3	Sliding block	CI	1
4	Guide screw	MS	1
5	Serrated plate	MS	2
6	CSK Screw 34 long	MS	4
7	CSK Screw 30 long	MS	2
8	CSK Screw 50 long	MS	2
9	Washer ϕ20×6	MS	1
10	Nut M20	MS	1

RESULT:

The 2D models of Machine Vice parts are created using Auto CAD.

LATHE TAILSTOCK

EXPT. NO: 26 DATE:

AIM:

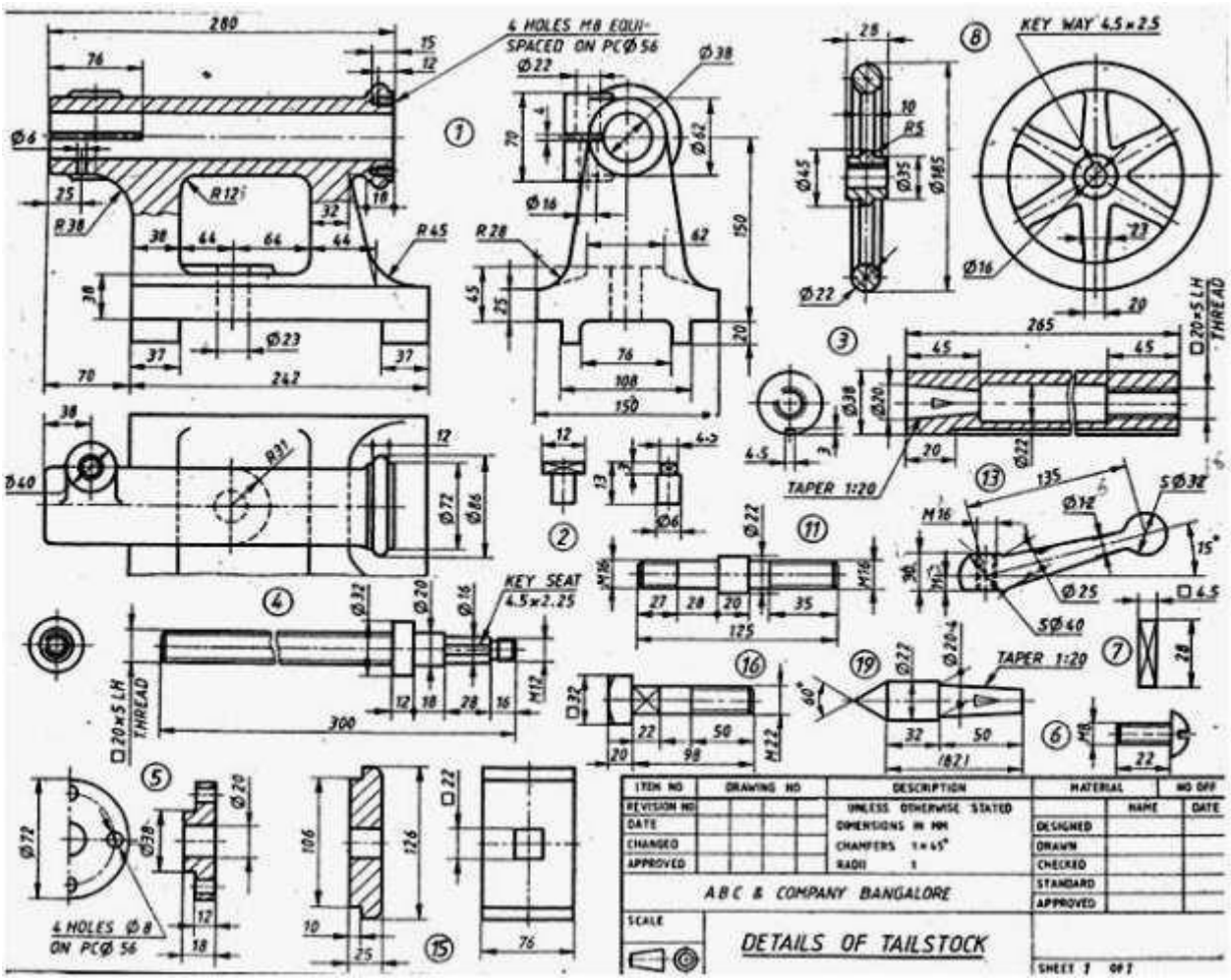
To create 2D models of Lathe Tailstock parts using AutoCAD.

PROCEDURE :

1. The drawings of Lathe Tailstock are studied.
2. 2D models of Lathe Tailstock are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



PART NO.	DESCRIPTION	MATERIAL	NO. OFF
1	BODY	CASTIRON	1
2	FEATHER	Fe 410 W	1
3	BARREL	CAST IRON	1
4	SCREW SPINDLE	Fe 410 W	1
5	FLANGE	CAST IRON	1
6	SCREW	Fe 410 W	4
7	FEATHER KEY	Fe 410 W	1
8	HAND WHEEL	CAST IRON	1
9*	WASHER M 12 STD	Fe 410 W	1
10*	HEX. NUT M12	Fe 410 W	1

PART NO.	DESCRIPTION	MATERIAL	NO. OFF
11	STUD	Fe 410 W	1
12*	WASHER M16 STD	Fe 410 W	2
13	HANDLE	CAST IRON	1.
14*	HEX. NUT M16	Fe 410 W	1.
15	CLAMPING PLATE	CAST IRON	1
16	SQ. HEAD BOLT	Fe 410 W	1
17*	WASHER M22 STD	Fe 410 W	1
18*	HEX. NUT M22	Fe 410 W	1
19	CENTRE	CAST STEEL	1

RESULT:

The 2D models of Lathe Tailstock parts are created using Auto CAD.

PLUMMER BLOCK

EXPT. NO: 27 DATE:

AIM:

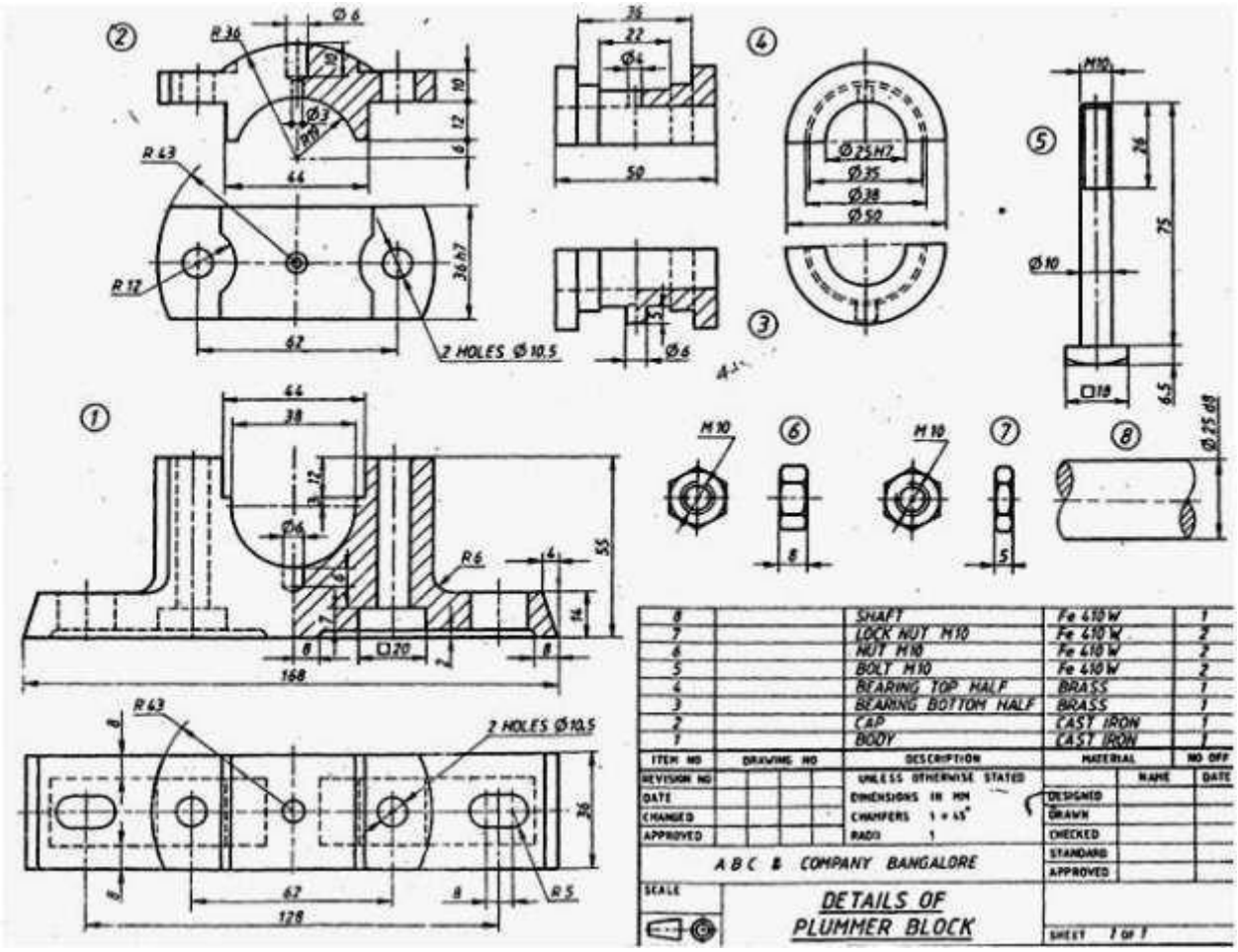
To create 2D models of Plummer Block parts using AutoCAD.

PROCEDURE :

1. The drawings of Plummer Block are studied.
2. 2D models of Plummer Block are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



RESULT:

The 2D models of Plummer Block parts are created using Auto CAD.

VANE PUMP

EXPT. NO: 28 DATE:

AIM:

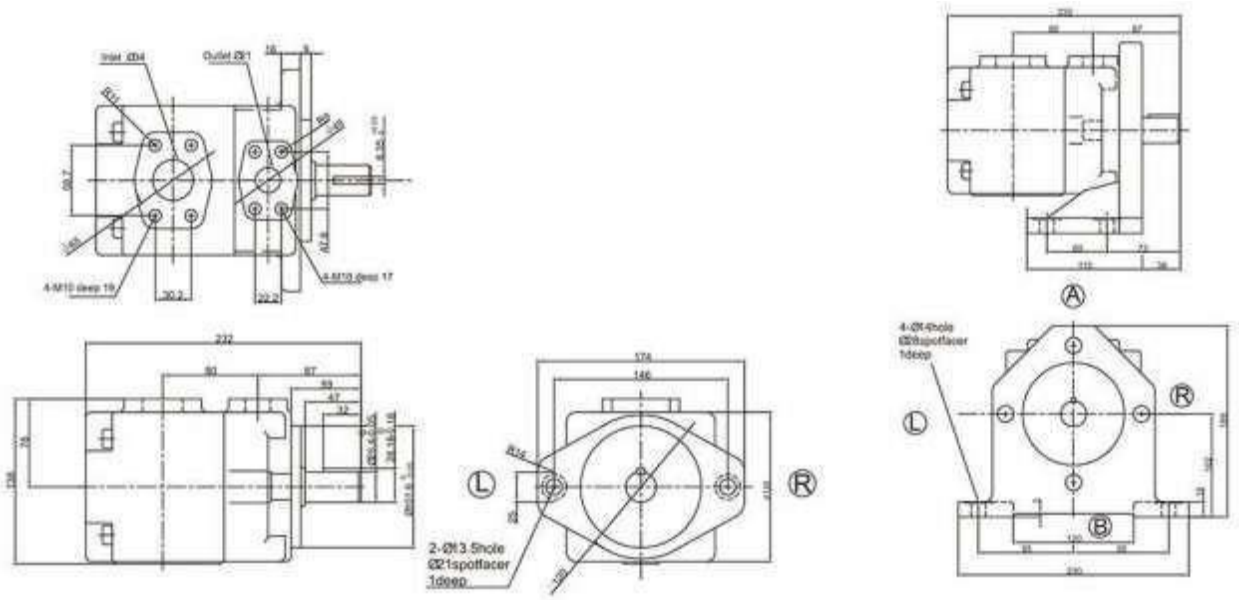
To create 2D models of Vane Pump parts using AutoCAD.

PROCEDURE :

1. The drawings of Vane Pump are studied.
2. 2D models of Vane Pump are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



RESULT:

The 2D models of Vane Pump parts are created using Auto CAD.

GEAR PUMP

EXPT. NO: 29 DATE:

AIM:

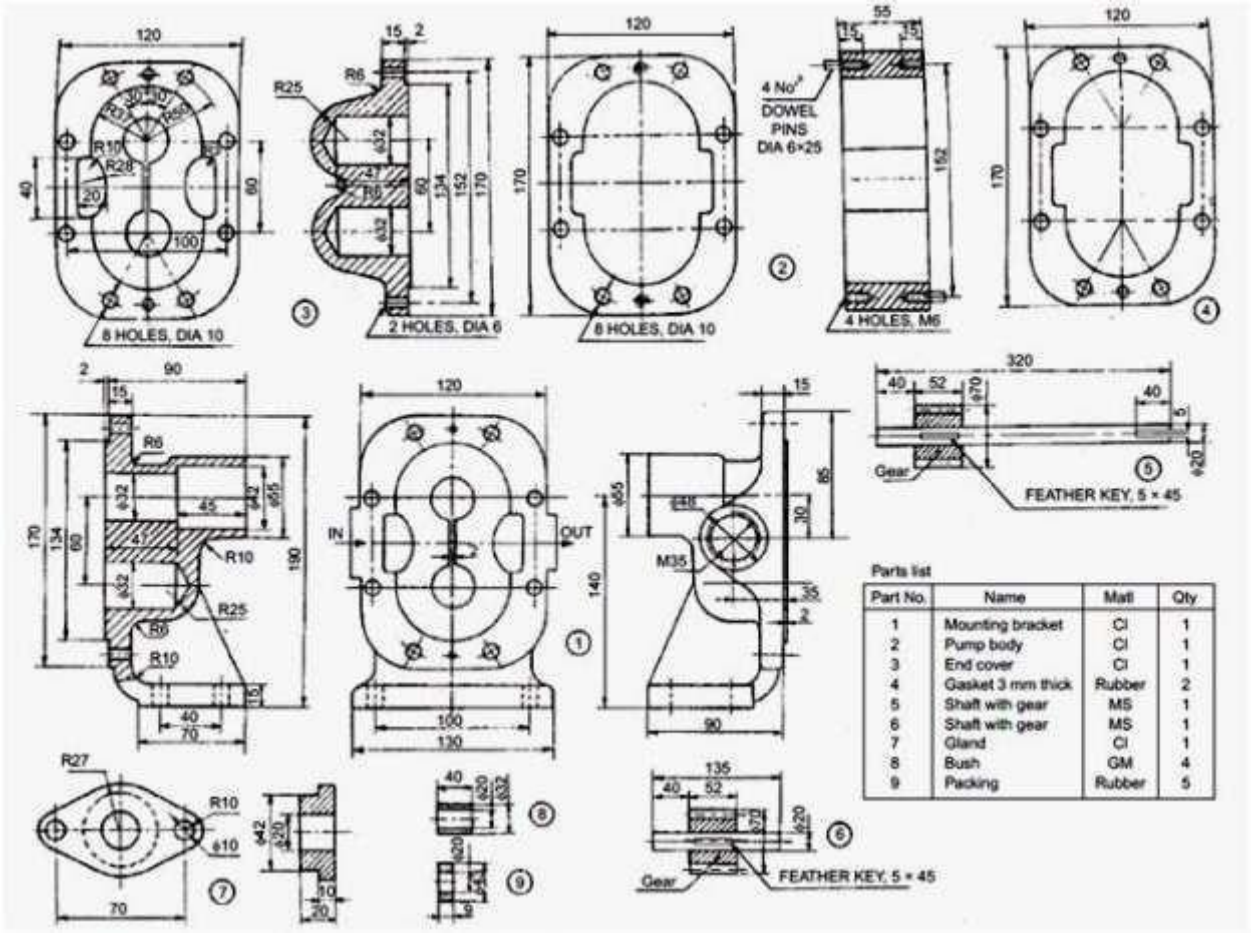
To create 2D models of Gear Pump parts using AutoCAD.

PROCEDURE :

1. The drawings of Gear Pump are studied.
2. 2D models of Gear Pump are created using AutoCAD.

COMMANDS USED:

Sketcher commands : Line , Circle, Arc, Fillet, Trim, Smart Dimensions, Relations, Mirror, Copy, Move, Rotate, Trim, Array, Scale, Dimlinear, Dimaligned, Dimangular, Dimarc, Dimradius, Dimdiameter, Mleaderedit, Mtext, Mleaderalign, Mleadercollect



Parts list

Part No.	Name	Matl	Qty
1	Mounting bracket	CI	1
2	Pump body	CI	1
3	End cover	CI	1
4	Gasket 3 mm thick	Rubber	2
5	Shaft with gear	MS	1
6	Shaft with gear	MS	1
7	Gland	CI	1
8	Bush	GM	4
9	Packing	Rubber	5

RESULT:

The 2D models of Gear Pump parts are created using Auto CAD.

AUTOCAD VIVA QUESTIONS

1) What is Autocad?

Autocad is a software program built to design and shape the 2-D and 3-D images. It provides the tools by which a detail design of the product can be done. It also has the option to create detailed design layout, which can be automatically drawn by using source model.

2) What are the uses of Autocad?

Autocad can be used by the professionals to visualize the imaginary view of the product on a computer system. In Autocad, it is possible by the drafter to make the changes in the product before it gets finalized for design. It also gives the freedom for the designer to implement their various ideas and represent them to the suppliers or their clients.

3) What are the fields where you see maximum use of Autocad?

Autocad is more popular among the architects, engineers and builders for developing their building layouts.

4) What is the file formats used in design?

In Autocad, .dwg file format is used for design, it can be an interchangeable format. The file format which is interchangeable has the extension as DXF and operates data operability. It provides different languages which can be used as per the requirement.

5) How you can create a user interface in Autocad?

User interface can be created by using the command prompts to draw the plots and dialog boxes. The dialog boxes can be displayed by the use of PLOT command and the external database commands (ASE). Setting of CMDDIA to 1, allows the dialog boxes to run the command. The user interface creation also needs the command line to display the entire file so that it can be edited or customized easily.

6) What is the function of vertical integration?

To enhance the architectural designing of 3D object Autocad uses the vertical integration program. The 3D objects can include walls and other things that are associated with the data having information and simple objects like lines and circles. The data is programmed in such away that it represents only the architectural products and the extracted files, and can be modified according to the requirement.

7) What is the use of variant in Autocad?

In Autocad variants are used to help in creation, visualizing and rendering the 3D models that include 3D printing as well. Variants allows you to use the functionality of different application according to the requirement.

8) What are the benefits of using Autocad?
Autocad has replaced the traditional method of drafting and designing which was made by pencil, drafting boards, triangles and compass with just a set of a computer program. The benefits are immense like:

- Saves time and helps to increase the productivity
- It helps to streamline your design and documentation workflow
- Physical „3D" prototype of the design can be quickly created by using Autocad
- 3D models can be directly imported into Autocad by using application like SolidWorks
- Tedious work of drafting can be done easily and you can design and re-design the product in short span of time.

9) What is the process to draw a line more than one time and save it automatically?
When a need arises to draw a new line the process opens up a new file in a new session to write the file. Autocad allows saving multiple drawing for each session. The files are saved by using the file extension .dwg and it can be modified by using the browser.

10) What are the steps that enable the drag and drop feature in Autocad?
Autocad provides a way to drag and drop the elements by the use of "NOUN" and "VERB" in a dialog box. It allows the object to move from one place to another. Likewise, the remove or editing function can be done by using "MOVE and ERASE".

11) What are the features corrected by Autocad?
Autocad detects the problem and correct it by removing the corruption with the drawing parts. With the available option of adding additional vertices, it also adds vertices to the poly-lines. The error can be corrected or neglected by finding out the exact location of the poly-line that has zero vertices. The object can be deleted or removed after there is no use of it in the system.

12) How to set up a default drawing directory?
Default drawing directory has sub directories having the information using the windows commands. The applications, which are going to be used, are highlighted, and by using drag and drop features those application will be placed in the default directory. The properties for the application are selected from the menu and dialog boxes displayed on the front.

13) How you can copy a closed drawing?
The copying of the closed drawing can be done by the designer center in the toolbar of the Autocad. By using the tree view option the copy of closed drawing can be done easily. The modification of the drawing can be done by using graphical interface.

14) How you can hide the specific layers when plotting in Autocad?
To hide the specific layers while plotting, you can use various options like turning off the layers for plotting , freezing the layers and turning off. Turning off for plotting will show the layers on the screen but won‟t output on printing. The layers that are turned off will hide the layers and also it will not appear on the screen.

15) What is the process of copying the dimension styles from one drawing to another in Autocad?
Copying of dimension styles require setting up the particular dimension style. To copy a dimension style, a new document has to be created. Once it is created, this document will be saved as a drawing template. A new reference will be created by new drawing template document and it will show all the options like layer style, units and blocks. Drawing can be done by seeing the current drawing and dimension style would be same as the original picture. By using the design center, Autocad tools can be used to copy the dimension styles from one drawing to another.

16) How can you remove the empty layers from drawing?
The layers can be removed only when the object resides in the layers will be removed, once it is removed it is an empty layer. The empty layers cannot be deleted by purge. It might be because the layer is frozen on a viewport or referenced by an object in a block definition. By using EXPORT command it is possible to remove the layer from drawing, which results in creating a DXF file of the drawing. You can edit the dxf file in a text editor and rename all the instances of layers in the file, except the layer definition.

17) Why Autocad WS is more popular among mobile users?
Autocad WS provides many option for mobile application developer like edit, view and share. They can easily share the application wherever they go and can develop an application in a matter of time. The application can be downloaded and installed from anywhere in the world, ignoring the licensing problem. The users can save file in any format and can run the application on any platform with ease.

18) How can you make a spring, spiral or screw thread?

To make a spiral or screw thread use an AutoLISP routine such as spiral.1sp, it will create a spiral path according to your need. Then you can use EXTRUDE command with a reference object, using the spiral as the path. Also, there is another way you can do this, by using Mechanical Desktop (MDT) or Autosurf by using augmented lines as path.

19) Tell me how you assign the keyboard characters or function keys to Autocad commands?
By editing the ACCELERATORS section of the Autocad menu file we can assign keyboard characters to Autocad commands.

20) How you can open a drawing file that was created with the automatic save features?
Autocad files have an extension of ".dwg" and will only open those files having this extension. To open the drawing files you have to rename it, by using Explorer or DOS prompt you can do that. If you are using EXPLORER you have to make sure that the option of "Hide file extensions for known file types" is not enabled. After the file is copied you can rename and use the OPEN command in Autocad to open the drawing.

21) What will you do when command prompts appear on the command line instead of ASE dialog box and plot dialog box?
In Autocad, CMDDIA variable controls the display of dialog boxes. To enable these dialog boxes you have to set CMDDIA to 1.

22) In Autocad, what is the command that is used to rotate the grid at 45 degress?
To rotate the grid at 45 degrees, command UCS is used.

23) In what situation command prompt appears instead of a dialog boxes?
If a file command is imported from a script or AutoLISP/ ObjectARX/ ADSRX a command prompt appears instead of a dialog box.

24) What are grips?
Grips are small boxes that appear on the objects you select. You can move or edit an object by dragging a grip.

25) What you are supposed to do when "Enter Attributes" dialogue doesn"t display?
In this case, two variable controls this function, ATTREQ and ATTDIA , to see the "Enter Attributes" dialogue , make sure both the variables are set to 1 and not 0.

26. Describe Procedure To Create User Interface?

- Command prompts are used to sketch dialog boxes and plots to create user interface.
- The dialog boxes display is seen by making use of some external commands or PLOT command.
- Set CMDDIA to 1 takes place that allows dialog boxes to run command and enter use spaces.
- The creation of user interface needs the command line to show complete file so that it can be edited or customized very easily.

27. Which Key Combinations Issues The Isoplane Command?
CONTROL and E or F5

28. The Effect Of Qtext Being On Is To?
speed up drawing regeneration by replacing text with rectangles

29. Which Is An Autocad Font For Mathematical Symbols?
MATHC

30. Which One Key Combinations Will Toggle Coords?
Either F6 or CONTROL and D

31. The Special Character String That Causes Text To Be Underlined Is?
%%u

32. Which One Is An Autocad Relative Polar Coordinate?
30<81

33. How Many Entity Colours Does Autocad Support?
255

34. Which Shapes Cannot Be Drawn By The Polygon Command?
rectangle

35. Which One Best Defines A "prototype" Or "template" Drawing?
a means of setting the initial drawing environment for new drawings

36. When Using Autocad For The Design Of A Mechanical Component Measuring 1000 Mm By 500 Mm, The Dimensions Would Normally Be Entered At A Scale Of?
1:10 in A4 size drawing

37. Which One Options Of The Undo Command Allows The Undo Facility o Be Disabled?
CONTROL

38. Which One Commands AllowsSimultaneous Pan And Zoom?
ZOOM DYNAMIC

39. When Is A Space Not Interpreted As The Enter Key?
The space is always an alternative to the enter key.

40. The Special Character String That Is Used To Print The Diameter Sign Is?
%%c

41. In Order For The Grid Command To Work Correctly Two Settings Have o Be Established?
Gridmode and osmode both set to 1

42. Which One Commands Can Be Used To Turn The Grid Through 45 Degrees ?
UCS

43. Which One Key Combinations Will Toggle Snap?
Either F9 or CONTROL and B

44. When Producing A Drawing Of A Site Plan Measuring 300m X 275m imensions Should Be Entered To A Scale Of?
1:1000

45. The Command To Scale A Paper Space Viewport To Half The Drawn ize Is?
Zoom 0.5x

46. Which Statements Is True For Autocad?
ZOOM ALL makes all objects visible irrespective of the layer visibility state.

47. Which One Is The Maximum Number Of Layers Which Can Be Defined In An Autocad Drawing?

256 layers

48. Which One Commands Can Be Used To Change The Value Of A System Variable?

SETVAR

49. If The System Variable Mirrtext Is Set To 1, Text Is?
Mirrored

50. Which One Commands Forces Autocad To Revise The Entire Drawing Taking Into Account The Effect Of Changes Within The System?
REGEN

51. The Shortcuts Or Command Alias That Allows Us To Key In L And Autocad Responds With Line Are Stored In A File Named
ACAD.PGP

52. When Designing An Integrated Circuit Measuring 5 Mm X 5mm The Measurements Should Be Entered Using A Scale Of
1:5

53. A House Measures 15000 Mm X 8000 Mm In Plan View. When Drawing The Plan View Only Which One Sets Of Limits Would Be Best To Use?
0,0/20000,15000

54. Which Commands Will Not Place Some Form Of Text Within The Drawing?
STYLE

55. The "select Object" Option "previous" Is Best Described By
Picks the previously picked group of objects

56. The Maximum Number Of Commands Whose Effect Can Be Undone By The Undo Command Is?
all commands issued since the drawing was last loaded into the drawing editor

57. The Effect Of The Explode Command When Applied To A Block Is To?
reduce a block to it's constituent entities to allow editing

58. Which One Is The Autocad Filename Extension Used To Indicate A Compiled Text Font?

TXT

59. Which One Is A Correct Statement Regarding Autocad Blocks?

Blocks can only be inserted into a drawing if they already exist in that drawing.

60. The Effect Of Freezing A Layer Is To?
make it invisible and prevent regeneration

61. Causes Attributes To Be Set To Their Defaults?
ATTDIA

62. Autocad Has Had The Dreaded Model Space And Paper Space Since R12. What Is The Main Advantage Of The System?
it converts 2D drawings to 3D

63. One Effect Of Setting The Value Of Mirrtext=0 Is That?
text is displayed but not mirrored

64. Which Statements Is True?
When a drawing is saved, a DWG file is created and an existing DWG file of the existing name and location is renamed with the extension .bak. A backup file is one save older than the drawing file of the same name. To ensure that your BAK files are the same as the DWG file, always save twice.

65. Which System Variables Controls The Accuracy Of Approximation Of A Spline Curve?
SPLINESEGS

66. Which One Would Be An Autocad Drawing File?
PLAN.DWG

67. To Continue Drawing A Line From The Last Known Point, Enter
(@)

68. Which One Best Describes A "user Coordinate System"?
the pair of coordinate points which define the lower left and upper right corners of the drawing boundary

69. The Setvar Command Is Used To?
change the values of the system variables

70. What Is The Purpose Of Auto Cad Software?

- AutoCAD software provides the design and the shape for the products that needs to be created.

- It provides flexible and user friendly features with the tools to design the applications and document the workflows.
- This involves aggregate and import models for the formats and usually allows the design to get created without any change in source model.
- It provides tools to provide the formats by detailed designing the layouts and drawings using the views automatically.
- It also has the provision to create detailed design layouts and views can be drawn automatically using the source model.

71. What Are The Uses Of Auto Cad?
    - AutoCAD software is used to draw and design the documents and the applications with easy customization options.
    - AutoCAD provides a platform to be used by professionals to create the designs and 3D models.
    - It allows the creation of the professional technical drawings and conceptual designs used for representation of the logics.
    - It allows the drafter to provide the finishing touches and designing with the detailing and linking to the online data.
    - It provides suppliers or operational professionals to review the drawings and modify it according to the requirements.

72. What Is The Main Purpose Of Auto Cad?
    - AutoCAD is used to create the computer aided designs or software applications including drafting.
    - AutoCAD develops the application in both the 2D and 3D formats and provide the information to the application.
    - AutoCAD provides tools to design the softwares used in the industry, architects and project management.
    - It provides an easy way to design the software with the designs and architect it according to the need.

73. What Are The File Formats Used In Design?
    - There is a use of native format like .dwg for the AutoCAD and it provides an interchangeable format.
    - The file format that is interchangeable has the extension as DXF and it provides the data operability.
    - AutoCAD includes the .dwg format and provide the support as well. It provides a way to use the files that are active.
    - It provides a way to estimate the total number of active .dwg files that can be made in collaboration with other files.
    - It provides languages that can be used and the localization part of the software can be modified to meet the requirements.

75. What Is The Function Of Vertical Integration?

- AutoCAD uses the vertical integration program to enhance the architectural designing of 3D objects.
- The 3D objects can include walls and other things that are associated with the data having intelligence and association with the simple objects like lines and circles.
- The data is programmed such that it represents only the architectural products and the extracted files.
- The information for the data and the actions on the like modifies and creates the relationship of an object.
- It consists of the tools that allow the estimation and other objects related representation in the 2D drawings.
- The elevations and sections used as a 3D architectural model specifies the standard calculations used in creating applications.

76. What Is The Use Of Variants In Auto Cad?

- AutoCAD uses lots of variants including the releases that provide capabilities to build an application.
- The 3D capability that is being provided by the AutoCAD gives fewer rises to the releases of the applications.
- Variants are used to increase the variations present in the application or the package used in particular program.
- AutoCAD provides variants that help in creation, visualization and rendering of the 3D models that provide the 3D printing as well.
- Variants provide a way to use different functionalities of the function in the application and execute it according to the needs.

7 7. What Are The Differences Present In The Software's Features?

The difference in the software‟s features is shown as:

- 3D Capabilities: this is a way through which AutoCAD can view the overall progress of creation, visualization and rendering of the 3D objects.
- Network Licensing: is being provided by the AutoCAD to be used for the multiple machines over the network.
- Customization: is another feature of AutoCAD that doesn‟t support LISP, ARX, and VBA for the customization part.
- Management and automation capabilities: is being provided with the management and automate the whole process of creation.

78. What Are The Features Involved With The Auto Cad Ws?

- AutoCAD is a platform that provides Mobile Apps (iOS) to be created by providing certain options for view, edit and share.
- The user can easily share the applications wherever they go on fly and it makes creating application less time consuming.
- The applications can be downloaded and installed from many other places that require licensing for the mobile applications.
- The users can save the file in whichever way they want and the application can run on any platform.
- AutoCAD applications provide a way to let the user share information on cloud and use the technology to enhance more features.

79. What Are The Different Ports Used In Different Os?

- AutoCAD doesn‟t have major improvements and some changes can be made to improve the capabilities of it.
- It requires the improvements to be made on the ports so to provide the easy accessibility to the applications and devices.
- The changes have been made to suit the environment and it included:
- Canvas control: this is being done in an increased manner and it controls the area that is being defined by the canvas.
- Associated arrays: this provides the arrays that are being associated with other arrays or other parts.
- 3D model manipulation: that is being done by the tools used in the operating systems and allows easy portability of the application.

80. What Is The Procedure To Create User Interface?

- The user interface can be created by using the command prompts to draw the plots and dialog boxes.
- The displaying of the dialog boxes can be seen by the use of PLOT command and the external database (ASE) commands.
- Setting of CMDDIA to 1 takes place that enables the dialog boxes to run the command and enter the spaces of use.
- The user interface creation also requires the command line to display the entire file so that it can be easily customized or edited.

81. What Is The Procedure To Draw A Line More Than One Time And Save t Automatically?

- AutoCAD allows the file to be written in multiple instances of the processes and it limit the resources that needs to be used as well.
- When a new line needs to be drawn the process opens up another file in a new session to write the file.
- AutoCAD allows saving multiple drawings to be saved for each session and it can be used to create the application.
- The files are saved by using the file extension as .dwg and it can be modified using the browser.
- The file extension needs to be hidden and it needs to show allowing the selection of the option as well.
- The file modification takes place by checking the AutoCAD operations and opening up of the drawing.

82. What Is The Procedure To Remove The Empty Layers?

- The empty layers can be removed from the drawing by using the all objects residing at once place.
- The references of the layers are also removed so that it appears to be empty and can easily be created.
- If the layer is empty then the PURGE will not be able to remove the layers from the drawing.
- The layer can be made frozen on the viewport that is visible to the object and provide the definition of the object too.
- The removing of the layer is possible due to the drawing that can be made using the EXPORT command and the result can be created in the DXF file.

83. What Are The Steps Need To Be Taken To Replace The Buttons On Toolbars With Smiley?

- AutoCAD have the place for button icons in the toolbar and tool boxes that it gets replaced with the smiley.
- AutoCAD provides the buttons for the template files that can be edited after customizing the toolbars.
- The changes are being provided by the template files that can cause the menu resources and allow the creation of files according to the requirements.
- To use smiley instead of buttons following steps are required and needed:
- Open the AutoCAD menu and edit the button file using the bitmap file that is saved in the support path.
- Move or copy the icon files that are used for different menus toolbars and open it according to the menus chosen.

- The button properties can be seen and an editor dialog box will be given according to the save as button.
- The directory is specified with the supported path and the Apply button properties are also being given for the AutoCAD properties.

84. What Are The Features Being Corrected By Autocad?

- AutoCAD identifies the problem and correct it by removing the corruption with the drawing parts.
- It involves adding of the vertices to the poly-lines with a provision to add additional vertices.
- The poly-line provides only the provision of using one vertex and the vertex can be added on run.
- The corruption can be neglected or corrected by finding out the exact location of the poly-line that has zero vertices.
- The objects can be deleted after there is no use of it or can be removed if there is no use of them in the system.

85. What Are The Steps To Enable The Drag And Drop Feature In Autocad?

- AutoCAD provides a way to drag and drop the elements by the use of noun and verb that allows the object to move from one place to another.
- AutoCAD provides an option to rotate or erase the selected object and allows the action to be taken on the application.
- The editing functions like MOVE and ERASE can be used to provide the erasing of the objects and noun/verb.
- Open the system option dialog box and choose from there Noun/Verb and turn it on from the menu bar.
- The selection tab can be selected by using the noun/verb selection of the options with the mouse clicks and opening of the menus.

86. What Are The Steps Involved In Setting Up The Default Drawing Directory?

- Default drawing directory is the one that involves sub directories containing the information using the windows commands.
- AutoCAD uses the navigation paths to travel the directories and it adds a shortcut for the directories that doesn‟t have a shortcut.
- The application can be highlighted that needs to be built in and then drag and drop features are used to place the application.

- The properties of the application are selected from the menu and dialog boxes used to display it on the front.
- AutoCAD uses the directory system to use the commands like OPEN that allows the selection of the files and create the shortcut in default directories.

87. What Is The Procedure To Copy The Dimension Styles From One rawing To Another?

- Copying the dimension of styles from one drawing to another requires setting up the particular dimension style.
- Setting up the dimension style also requires the use of documents and creation of the styles while using the blank document.
- The document is saved by using the document as a Drawing template and then new references gets created by the template document.
- Templates define the system variables that can be consistent across all the platforms and provides the drawings like layer styles, units and blocks.
- It can be copied by copying the styles on single case basis and the drawings can be done by seeing the current drawing.
- Using of the design center makes it easy for the AutoCAD tools to allow the browsing and copying of the styles and other drawings activities.

88. What Are The Steps Involved In Copying A Closed Drawing?

- The copying of the closed drawing takes place from the toolbar provide in the design center of the AutoCAD.
- Design center allows the modification of the drawing to be done using the graphical interface.
- Open drawings are given to display the content and copy the closed drawings easily by using the Tree View option.
- The drawing is involved with the drawing elements that can be copied with the defined objects that have the similar type.
- Dim style gets copied from one to another and it provides a drag and drop feature that gets opened into an open document.

89. What Is The Way To Hide The Specific Layers When Plotting?

- The prevent layers used using the plotters for plotting provides the layer manager that allow easy options to customize the application.
- The layers used in layer manager have specific options like:

- Turning off the layers
- Freezing the layers
- Turning off the layers for plotting
- Layers used in the plotting remain visible on the screen and doesn‟t provide the output to the printer.
- Individual layers can also be set up in specific viewports that allows drawing of the layers that contains the systems applications and options.
- It uses the viewport concept that allows the different layers to be frozen on the view port and this way it can be hidden from other layers.

90. Why Autocad Software Is Used?
    - The shape and design of the products that is required to be created is offered by AutoCAD software.
    - This software offers features that are user friendly and very flexible with some specific tools to develop applications and the workflow is documented.
    - Aggregating and importing models is involved in it and most commonly permits to develop design without making any changes in the original model.
    - It offer some specific tools to present formats by designing layouts in detail and views can be used in drawings automatically.
    - Also, it has some specifications to develop detail design layouts and automatically views are drawn by making use of original model.

91. Give Autocad Uses?
    - AutoCAD software is responsible for designing and drawing the documents and applications with simple customization options.
    - A platform is offered by AutoCAD for professionals to generate attractive designs and some 3D models.
    - It permits conceptual designs and qualified technical drawings formation that is effectively used for logical representations.
    - It permits drafter to offer designing and finishing by connecting to online information.
    - It offers operational experts or suppliers to evaluate drawings and change it according to the necessity.

92. What Is The Main Reason For Using Autocad?

- Mainly, AutoCAD is responsible for developing designs that are computer aided or some software application that contain drafting.
- Application can be developed in 2D and 3D formats using AutoCAD and offer information to application.
- Some specific tools are offered by AutoCAD to develop software used in the market, project management and architectures.
- It offers best way to develop software with designs and build it according to the requirement.

93. What File Formats Are Used In Design?
    - There is native format use such as .dwg for AutoCAD and it offers format that can be exchanged.
    - The format of file that can be exchanged has DXF extension and it offers data operability.
    - dwg format is included in AutoCAD and give the support. It gives a method to use active files.
    - It gives a way to approximate total file having extension .dwg that is created in association with other files.
    - It offers languages that are used and software localization part can be changed to meet the needs.

94. Give The Variants Usage In Autocad?
    - Multiple variants are used by AutoCAD that include releases offering abilities to create an application.
    - The 3D ability offered by AutoCAD provides few rises to the application releases.
    - Variations are increased by variants that are in the application or the package used in specific program.
    - AutoCAD offers variants that help in formation, visualizing and depicting 3D models that offer even 3D printing.
    - Variants offer best method to make use of various functionalities of function and evaluate it according to the requirements.

95. Give Features Involved In Autocad Ws?
    - A platform that offers Mobile Apps to be developed by giving some options to view, share and edit is AutoCAD
    - Applications can be easily shared by user when they go on fly and the applications can be developed in less time.
    - Application are downloaded and installed from different places that need license for all the applications developed for mobiles.
    - Files can be saved by the user according to their desire and that application can be run on any different platform.

- The application based on AutoCAD offer best way to let information shared by the user on cloud and use technology to improve features.