COMPUTER AIDED MACHINE DRAWING

(19A03404)

B.TECH

(II YEAR-II SEM)

(2020-21)

DEPARTMENT OF MECHANICAL ENGINEERING

SVR ENGINEERING COLLEGE

Syllabus

Course Objectives:

- Introduce conventional representations of material and machine components
- Train to use software for 2D and 3D modelin
- Familiarize with thread profiles, riveted, welded and key joints.
- Teach solid modeling of machine parts and their sections
- Explain creation of 2D assembly drawings from 3D assemblies
- Familiarize with limits, fits and tolerances in mating components

The foll wing contents o are to be done by any 2D software package Conventional representation of materials and components:

Detachable joints: Drawing of thread profiles, hexagonal and square-headed bolts and nuts, bolted joint with washer and locknut, stud joint, screw joint and foundation bolts.

Riveted joints: Drawing of rivet, lap joint, butt joint with single strap, single riveted , double riveted double strap joints. Welded joints: Lap joint and T joint with fillet, butt joint with conventions.

Keys: Taper key, sunk taper key, round key, saddle key, feather key, woodruff key. Shaft coupling, bushed pin-type flange coupling, universal coupling, Oldhams' coupling.

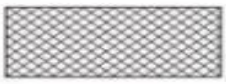
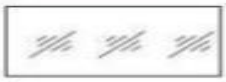
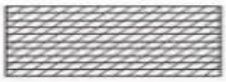
The following contents to be done by any 3D software package Sectional views Creating solid models of complex machine parts and create sectional views.

Assembly drawings: (Any four of the following using solid model software) Lathe tool post, tool head of shaping machine, tail stock, machine vice, gate valve, carburettor, piston, connecting rod, excentric, screw jack, plumber block, axle bearing, pipe vice, clamping device, Geneva cam, universal coupling,

Manufacturing drawing: Representation of limits, fits and tolerances for mating parts. Use any four parts of above assembly drawings and prepare manufacturing drawing with dimensional and geometric tolerances.

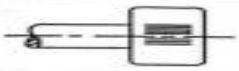
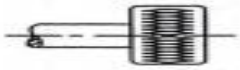
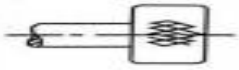
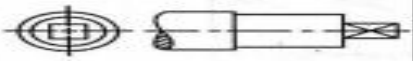
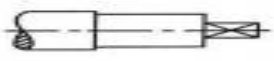
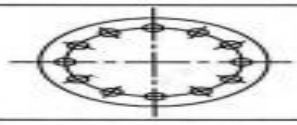
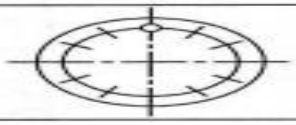
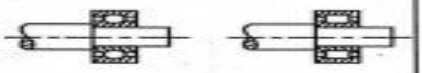
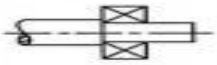
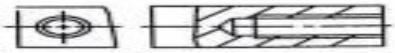
I. DRAWING OF MACHINE ELEMENTS AND SIMPLE PARTS

CONVENTIONAL REPRESENTATION OF MATERIALS

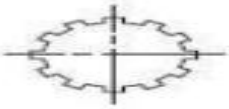
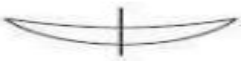
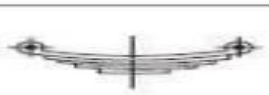
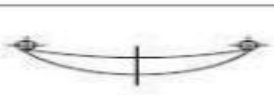
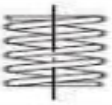
Type	Convention	Material
Metals		Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.
		Lead, Zinc, Tin, White-metal, etc.
Glass		Glass
Packing and Insulating material		Porcelain, Stoneware, Marble, Slate, etc.
		Asbestos, Fibre, Felt, Synthetic resin products, Paper, Cork, Linoleum,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

Conventional Representation of Materials

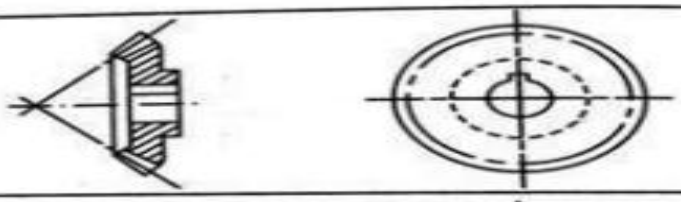
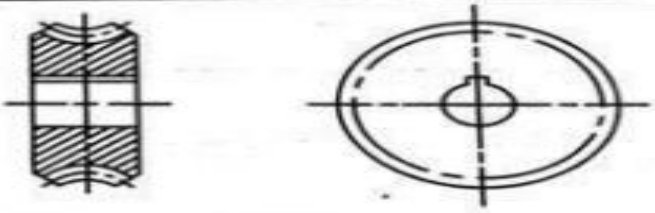
CONVENTIONAL REPRESENTATION OF MACHINE COMPONENTS

Title	Subject	Convention
Straight knurling		
Diamond knurling		
Square on shaft		
Holes on circular pitch		
Bearings		
External screw threads (Detail)		
Internal screw threads (Detail)		
Screw threads (Assembly)		

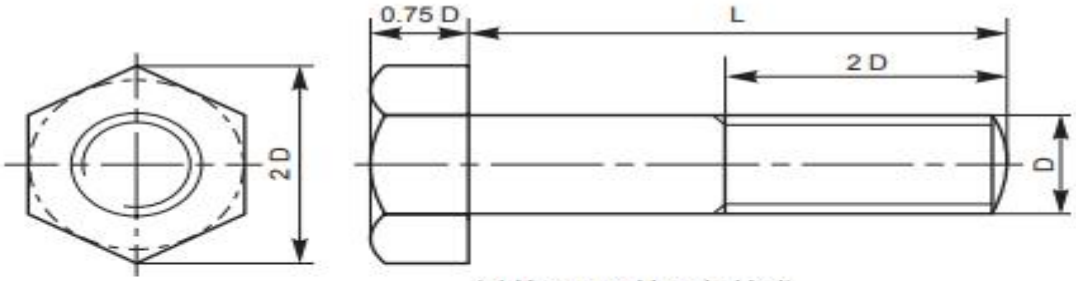
CONVENTIONAL REPRESENTATION OF MACHINE COMPONENTS

Title	Subject	Convention	
Splined shafts			
Interrupted views			
Semi-elliptic leaf spring			
Semi-elliptic leaf spring with eyes			
	Subject	Convention	Diagrammatic Representation
Cylindrical compression spring			
Cylindrical tension spring			

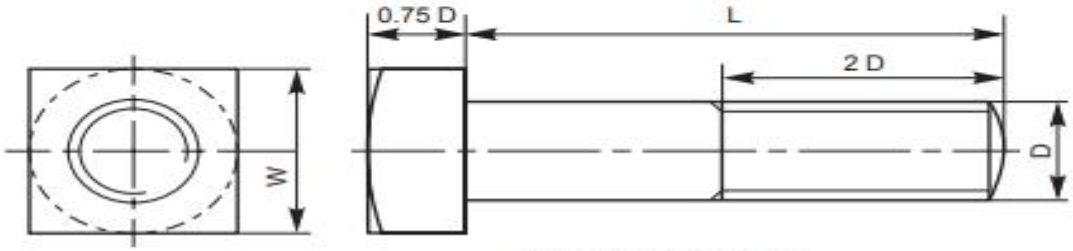
CONVENTIONAL REPRESENTATION OF MACHINE COMPONENTS

Title	Convention
Spur gear	
Bevel gear	
Worm wheel	
Worm	

Hexagonal and square Headed bolts

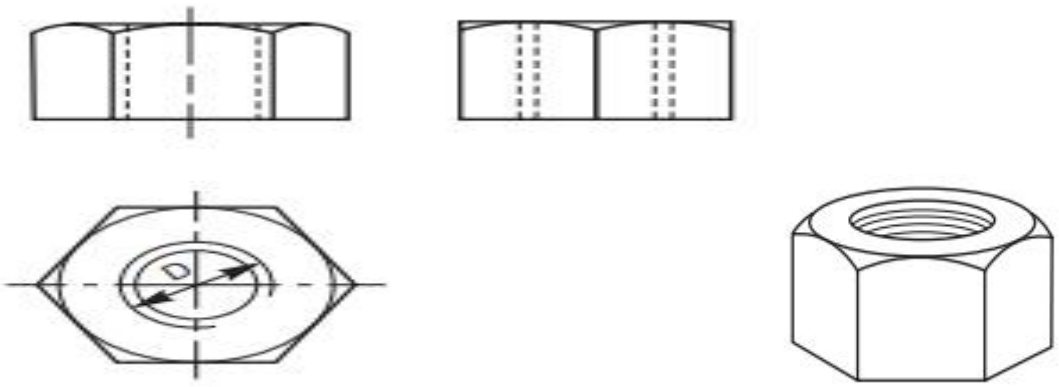


(a) Hexagonal headed bolt



(b) Square headed bolt

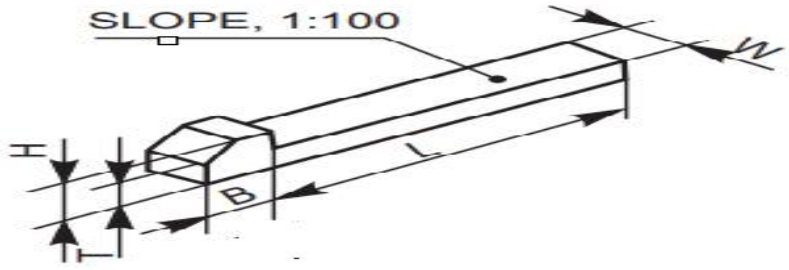
Hexagonal Nut



Gib Head Key

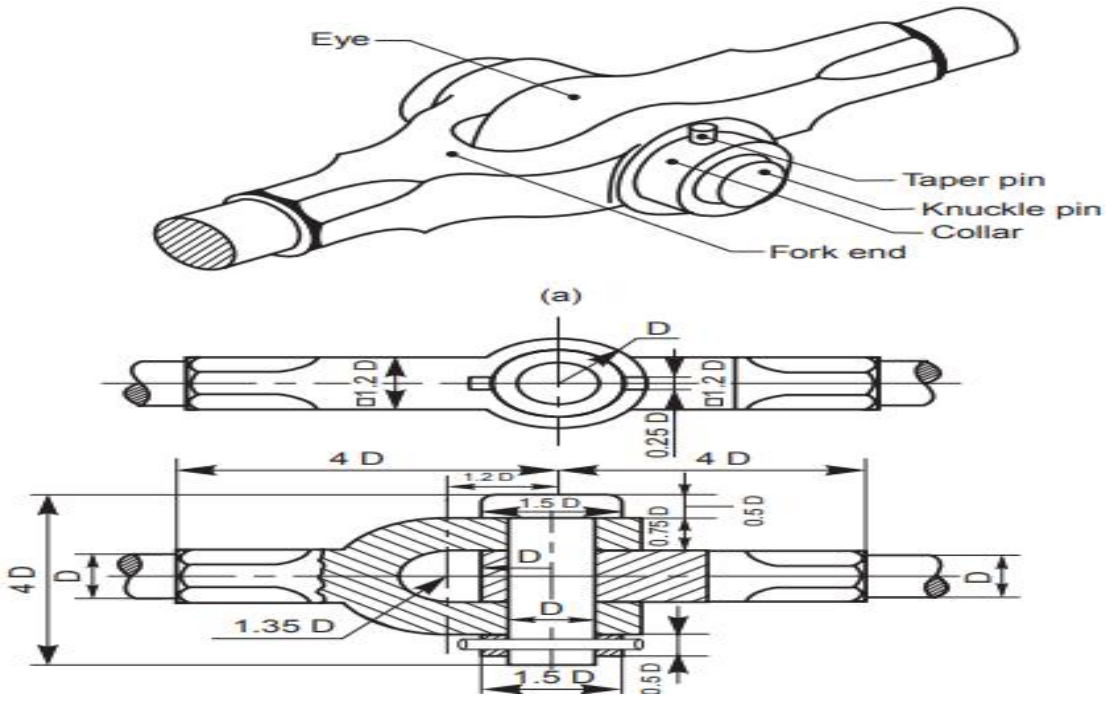
If D is the diameter of the shaft, then,

Width of key, W	= 0.25 D + 2 mm
Thickness of key, T	= 0.67 W (at the thicker end)
Standard taper	= 1:100
Height of head, H	= 1.75 T
Width of head, B	= 1.5 T



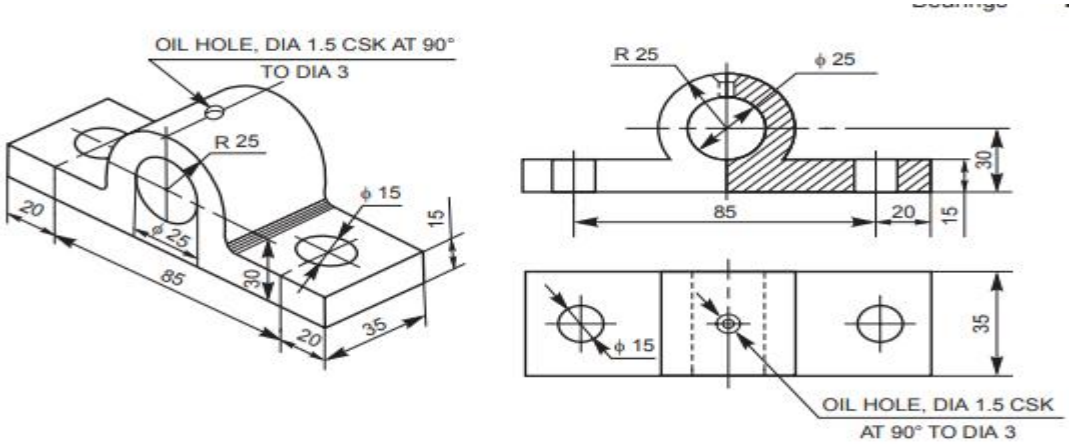
bolted joint with washer and locknut, stud joint, screw joint and foundation bolts. Riveted joints: Drawing of rivet, lap joint, butt joint with single strap, single riveted , double riveted double strap joints.

Knuckle joints are used in suspension links, air brake arrangement of locomotives.



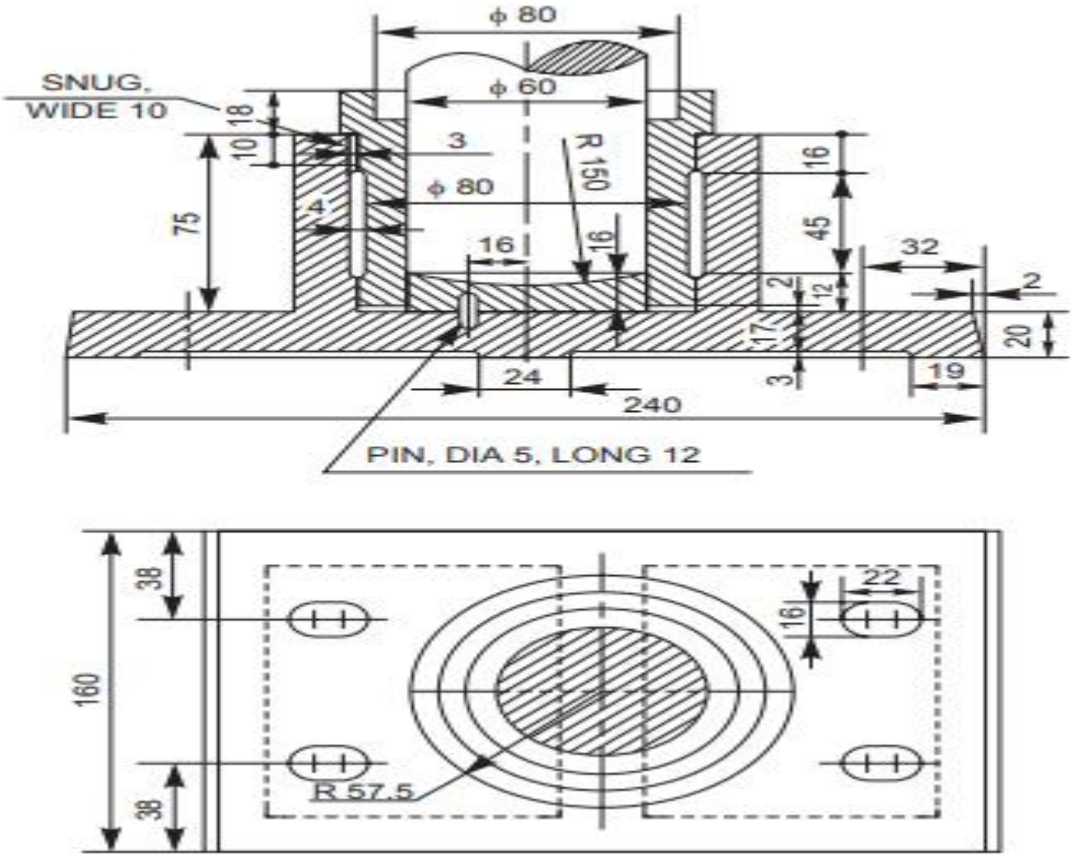
Bushed journal Bearing

This bearing consists of mainly two parts, the body and the bush. The body is usually made of cast iron and the bush of soft materials such as brass, bronze or gunmetal. The bush is press fitted in the body; preventing relative axial and rotary motion. With this arrangement, to renew the bearing, it is only necessary to renew the bush



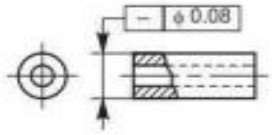
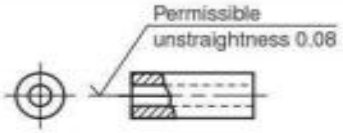
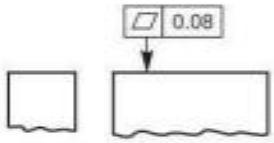
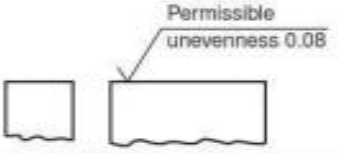
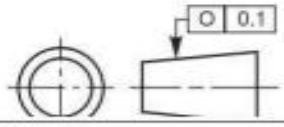
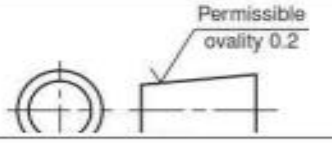
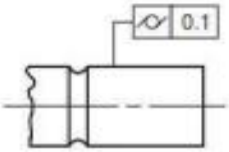
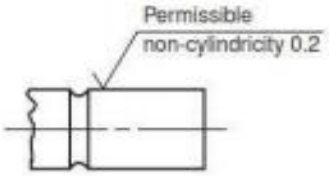
Footstep Bearing:

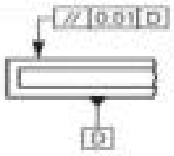
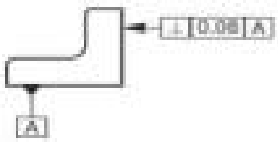
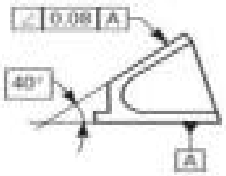
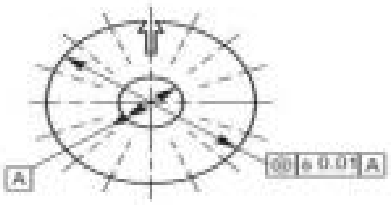
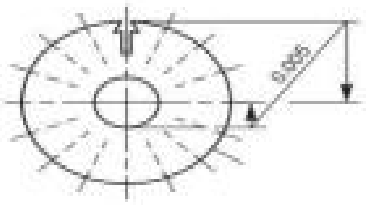
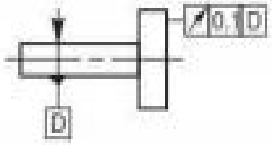
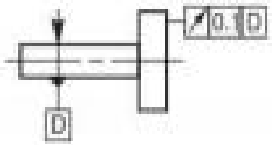
This bearing is used to support a vertical shaft under axial load. Further, in this, the shaft is terminated at the bearing. The bottom surface of the shaft rests on the surface of the bearing which is in the form of a disc. The bush fitted in the main body supports the shaft in position and takes care of possible radial loads coming on the shaft



Characteristics to be toleranced		*Symbols*
Form of single features	Straightness	
	Flatness	
	Circularity (roundness)	
	Cylindricity	
	Profile of any line	
	Profile of any surface	
Orientation of related features	Parallelism	
	Perpendicularity (squareness)	
	Angularity	
Position of related features	Position	
	Concentricity and coaxiality	
	Symmetry	
	Run-out	

FORM AND POSITIONAL TOLERANCES

As per the standard	*As prevalent in industry*
1. Straightness tolerance	
	
2. Flatness tolerance	
	
3. Circularity tolerance	
	
4. Cylindricity tolerance	
	

5. Parallelism tolerance	
	
6. Perpendicularity tolerance	
	
7. Angularity tolerance	
	
8. Concentricity and coaxiality tolerance	
	
9. Symmetry tolerance	
	
10. Radial run-out	
	
11. Axial run-out	
	

Systems of indication of tolerances of form and of position

SURFACE ROUGHNESS AND ITS INDICATION

Surface Roughness: The properties and performance of machine components are affected by the degree of roughness of the various surfaces. The higher the smoothness of the surface, the better is the fatigue strength and corrosion resistance. Friction between mating g parts is also reduced due to better surface finish.

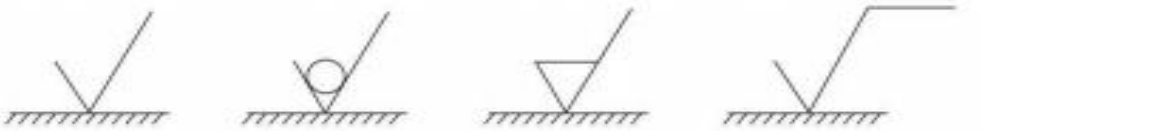
Surface Roughness Number: The surface roughness number represents the average departure of the surface from perfection over a prescribed sampling length and is expressed in microns.

$$R_a = \frac{h_1 + h_2 + h_3 + \ldots + h_n}{n}$$

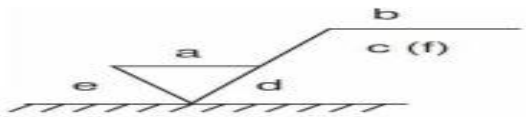
The surface roughness may be measured, using any one of the following:

1. Straight edge
2. Surface gauge
3. Optical flat
4. Tool makers Microscopes
5. Profilometer
6. Profilograph
7. Talysurf

Machine Symbols: The basic symbol consists of two legs of unequal length, inclined at approximately 60° to the line, representing the surface considered. This symbol may be used where it is necessary to indicate that the surface is machined, without indicating the grade of roughness or the process to be used.



a) Basic symbol.
b) Material Removal is Not Allowed
c) emoval Of Material is Allowed.
d) pecial surface characteristics



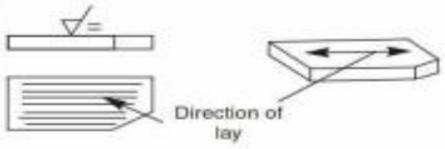
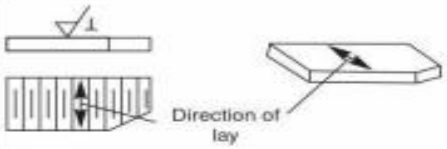
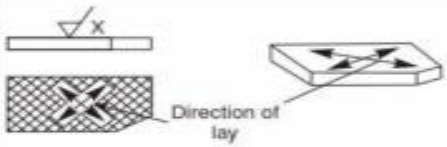
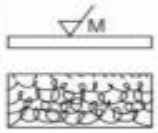
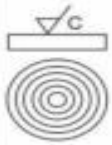
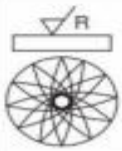
Indication of Machining Allowance

Roughness values R_a *μm*	*Roughness grade number*	*Roughness grade symbol*
50	N12	
25	N11	
12.5	N10	
6.3	N9	
3.2	N8	
1.6	N7	
0.8	N6	
0.4	N5	
0.2	N4	
0.1	N3	
0.05	N2	
0.025	N1	

Equivalent surface roughness symbols

Indication of Special Roughness Characteristics: In certain circumstances, for functional reasons, it may be necessary to specify additional special requirements, concerning surface roughness. If it is required that the final surface texture be produced by one particular

production method, this method should be indicated on an extension of the longer arm of the symbol. Also, any indications relating to treatment of coating may be given on the extension of longer arm of the symbol

Symbol	*Interpretation*	
=	Parallel to the plane of projection of the view in which the symbol is used	
⊥	Perpendicular to the plane of projection of the view in which the symbol is used	
X	Crossed in two slant directions relative to the plane of projection of the view in which the symbol is used	
M	Multi-directional	
C	Approximately circular, relative to the centre of the surface to which the symbol is applied	
R	Approximately radial, relative to the centre of the surface to which the symbol is applied	

Symbols specifying the directions of lay

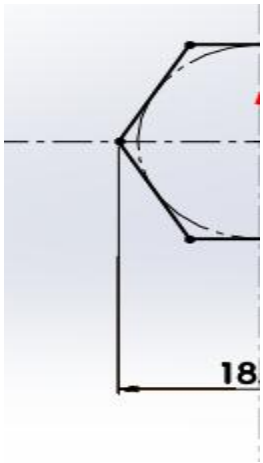
Experiments : 1. Assembly of bolt and nut with different screw thread

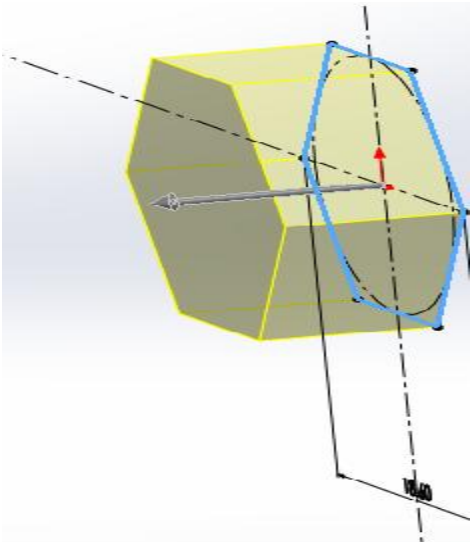
DETAILS AND MODELING OF INTERNAL AND EXTERNAL THREAD OF BOLT AND NUT USING SOLID WORKS

AIM : To model a bolt and nut by creating, modifying assembling and manipulating various features by feature based parametric solid modeliing and detailing .

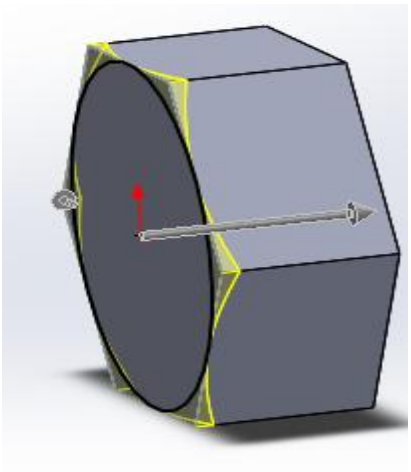
Tools: Personal computer with Pentium IV processor with windows xp/windows-7 and solidworks software.

1. Procedure: Create a 2D sketch on Front Plane as shown in the figure.
2. (Right click the Front plane>insert sketch and draw the 2D sketch)
3. Note: All the 2D sketches drawn should be fully Defined and there should not be any under defined) and use (click Add Relation and Smart Dimensions)

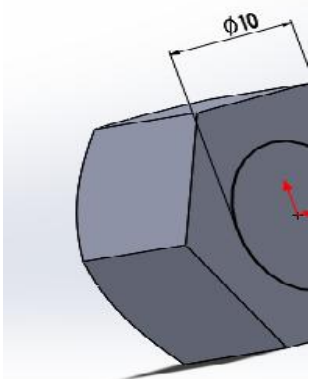


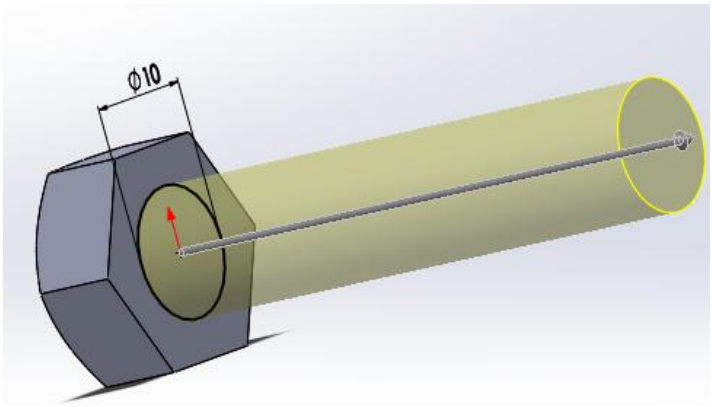


4. Create circle of 2D sketch of Hexagonal width of 11.6 mm, on right plane and cut extrude to 7mm, (Select the face by (Enter Space bar> double click the Normal plane)



5. Create circle of 2D sketch of Diameter of 11.6 mm, on right plane and extrude cut to 7mm taper 60º, flip side to cut, draft inward. (Select the face by (Enter Space bar> double click the Normal plane) and Draw the 2D sketch as given above.
   Extrude cut by (Insert>Boss/Base>Extrude)) ok.





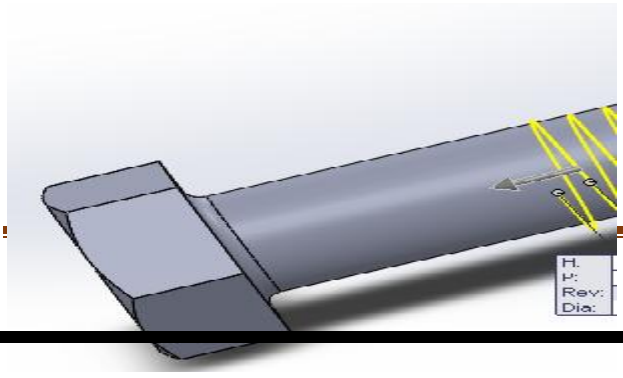
6.Create circle of 2D sketch of Diameter of 10 mm, on right plane and extrude to 7mm (Enter Space bar> double click the Normal plane)

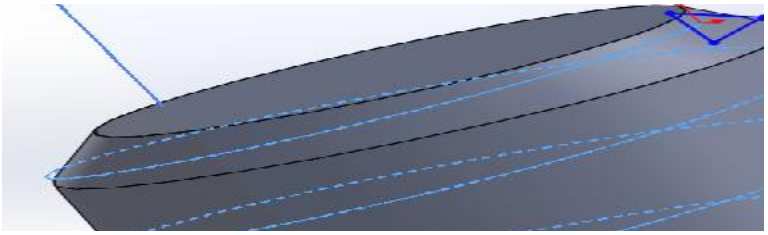
7.Create fillet and chamfer at corner of bolt at size of 1mm

8.Create external thread, Click Insert>Curve>Helix/Spiral
Press F to zoom fit, set Parameters Constant Pitch , Pitch suitable dimensions Revolutions 4 , Start angle 0.0deg.
Click Sketch , click Circle . Sketch circle at start point, then click Smart dimension.

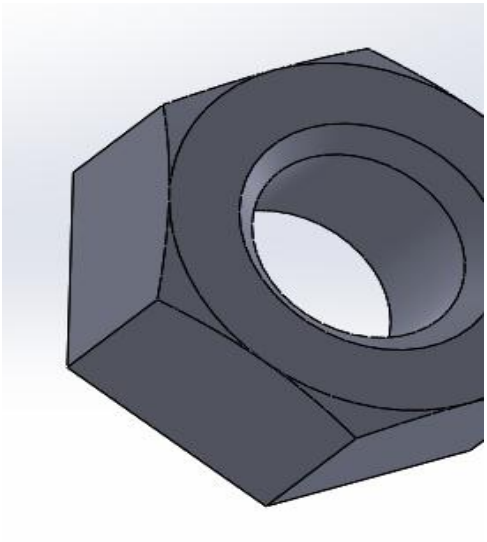
set circle diameter to 1 mm.

9. Take sweep (insert→boss/base →sweep) command and give Select profile and there relative circle, and Select path there relative curve,Options→orientation /twist type (select →along path),Define by→select turns →give the valve of 50 to 100). →Ok done.

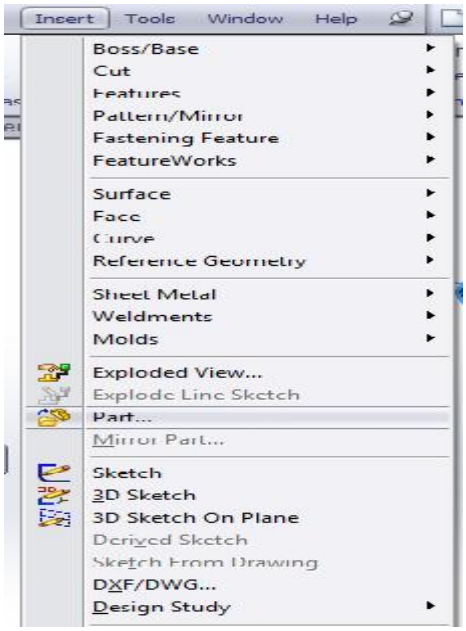


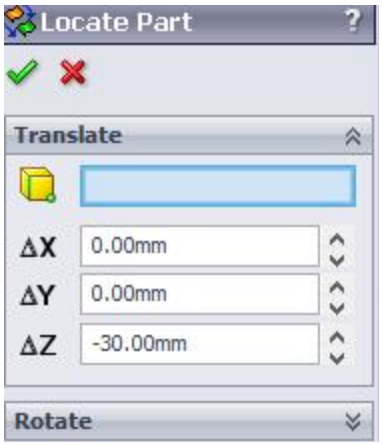


Similarly create Nut also as above said. Dimensions of nut as per bolt.

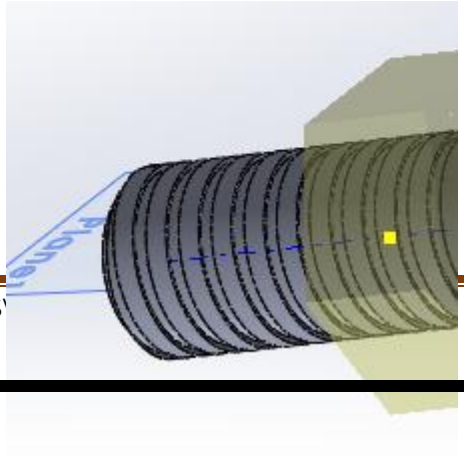


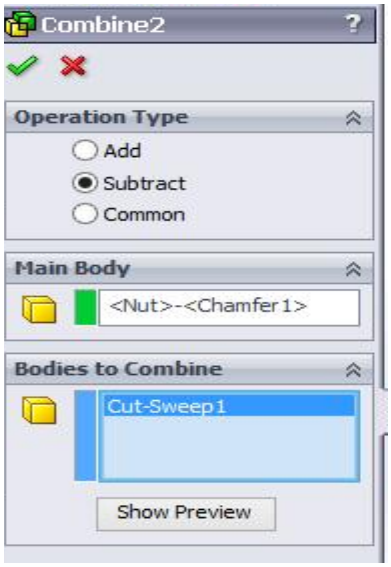
After create Nut part and save it.
And open bolt file and select
Select Insert> part> nut part>ok.





10. Locate part with center of bolt.



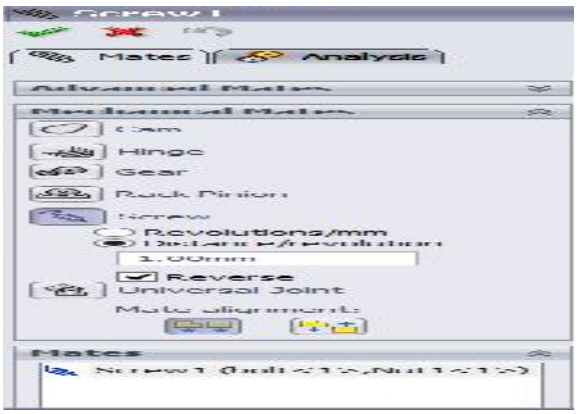


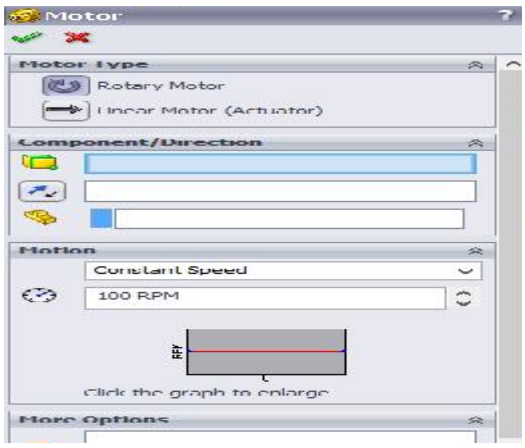
11.Insert>Feature>combine> Select Main body of nut and bolt >OK and save the file with different name.

Assembly: Open new assembly file. Import bolt and nut file in assembly mode.

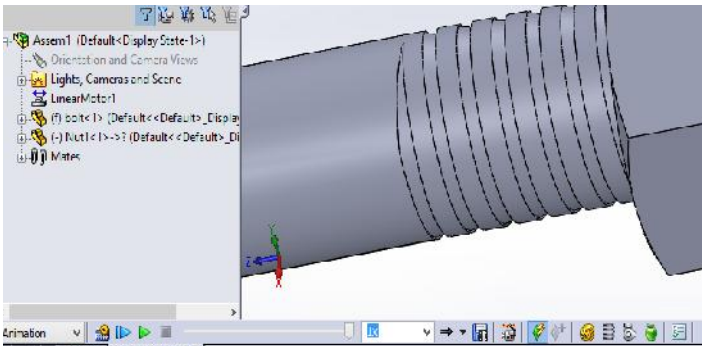
12. Mate the components using concentric Mate (select both bolt and nut thread faces).

13. Mate the components using screw Mate and select both faces of bolt and nut.





Animation: open the motion study>switch of the orientation and camera views.

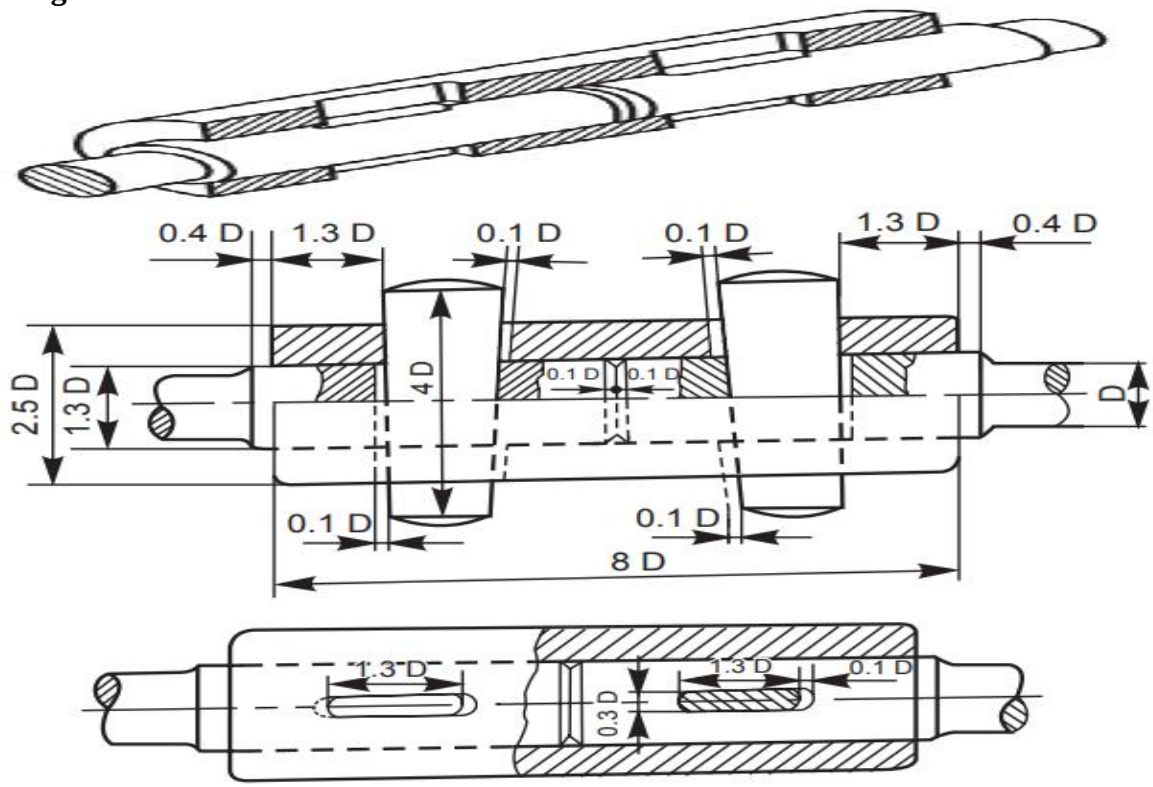


Start the motor> select the linear motor>motor location(select nut face) and component to move relative to(select bolt) ,motion at constant speed of 10mm/s>OK. Calculate and play.

2. Assembly of keys and cotter joints

Cotter joint: Draw 3D assembly component

Cotter joints are used to connect two rods, subjected to tensile or compressive forces along their axes.

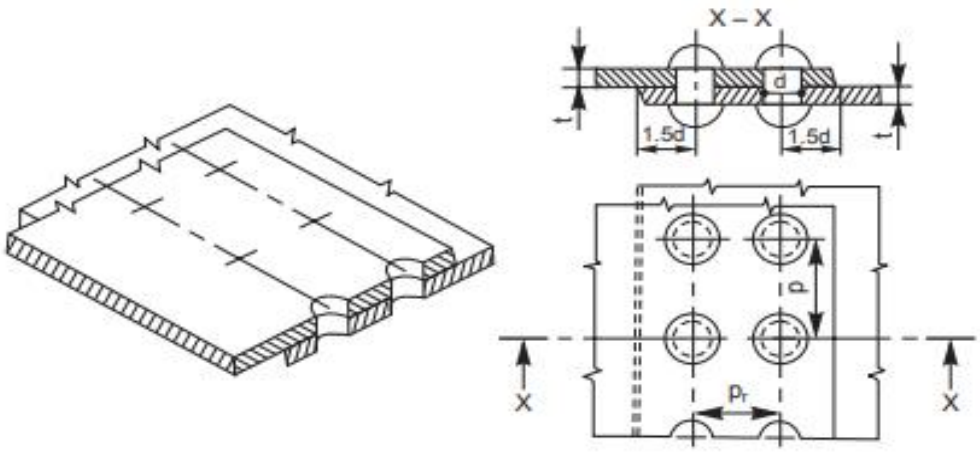


3. Riveted joints for plates:

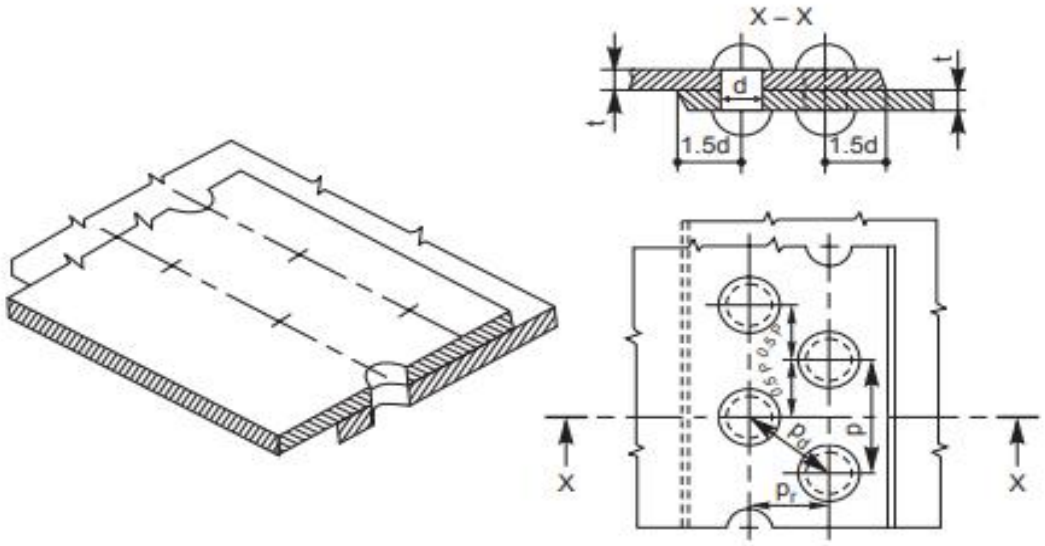
In a lap joint, the plates to be riveted, overlap each other. The plates to be joined are first bevelled at the edges, to an angle of about 80° Depending upon the number of rows Riveted Joints of rivets used in the joint, lap joints are further classified as single riveted lap joint, double riveted lap joint and so on.

were d=$6\sqrt{t}$ mm

Double riveted chain lap joint



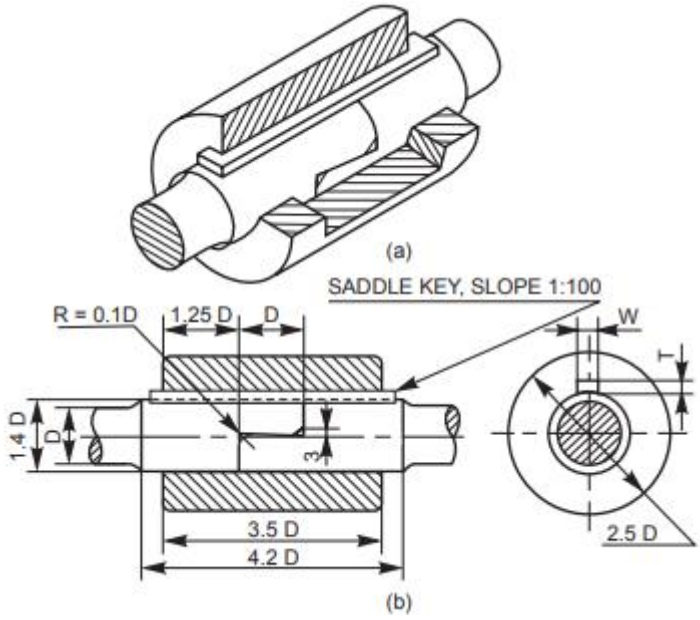
Double riveted chain lap joint



Double riveted zig-zag lap joint

4.Half lap muff coupling

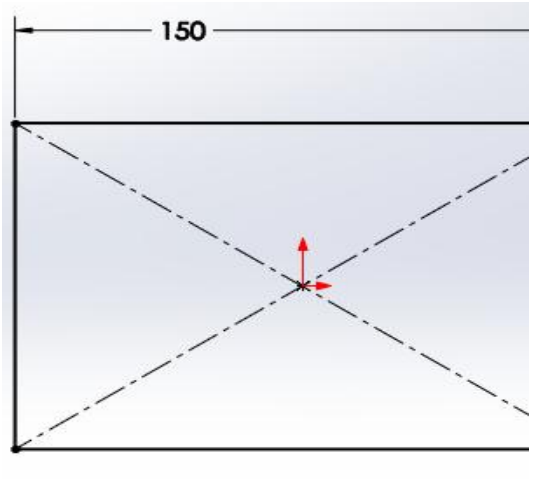
In this, the ends of the shafts overlap each other for a short length. The taper provided in the overlap prevents the axial movement of the shafts. Here too, after placing the muff over the overlapping ends of the shafts, a saddle key(s) is(are) used to make the coupling



5.DETAILS AND ASSEMBLY OF STUFFING BOX USING SOLIDWORKS SOFTWARE

AIM:
To draw the detail view of the Stuffing Box and assemble the parts by using the Solidworks software and obtain its respective views.

COMMANDS USED: Sketch, extrude , Shaft, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

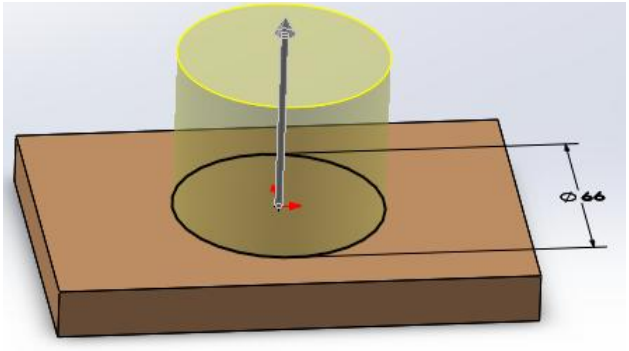


1.Create a 2D sketch on Front Plane as shown in the figure.

2.(Right click the Front plane>insert sketch and draw the 2D sketch).

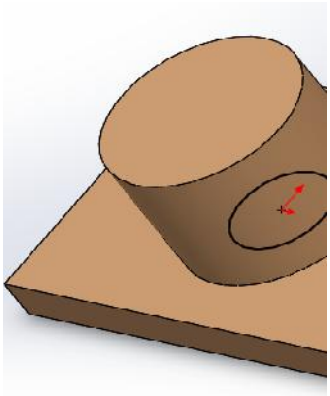
Note: All the 2D sketches drawn should be fully Defined and there should not be any under defined) and use (click Add Relation and Smart Dimensions.

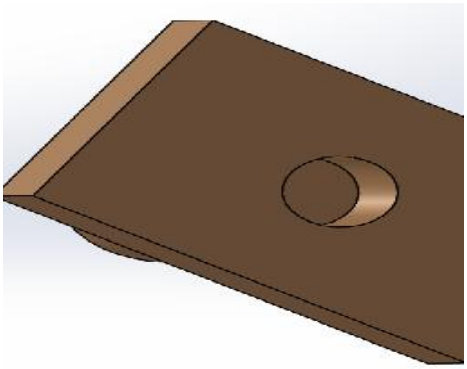
3. extrude to 15 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above



4.extrude to 50 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

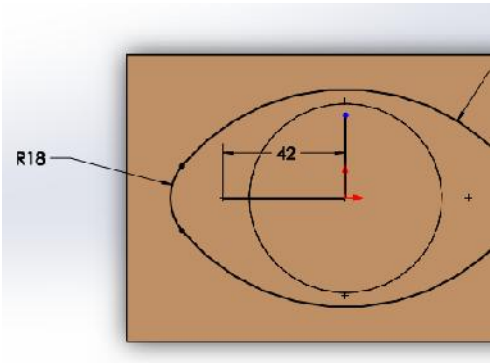
Inner diameter 34 mm size and use extrude cut and remove material up to end of block as shown below.





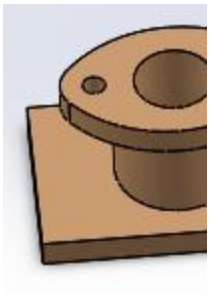
5.Create a 2D sketch on Front Plane as shown in the figure.

(Right click the Front plane>insert sketch and draw the 2D sketch).



extrude to 15 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

1. **Use Extrude cut with dimensions of 42 mm size circle as per below figure.**



Create a hole as per the dimensions of 12mm size both sides.

6.Create a 2D sketch on Front Plane as shown in the figure.

(Right click the Front plane>insert sketch and draw the 2D sketch).

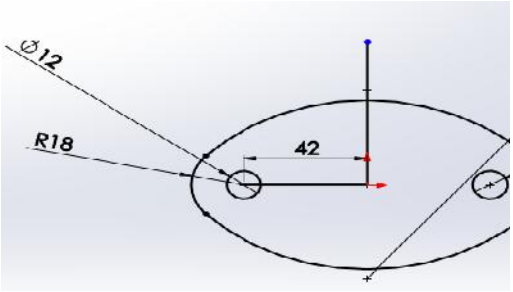
Extrude cut to 51 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

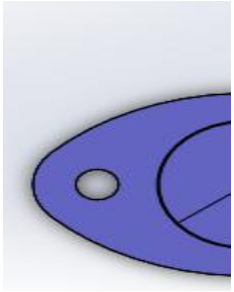
7. Create thread,Take sweep (insert→boss/base →sweep) command and give Select profile and there relative circle, and Select path there relative curve,Options→orientation /twist type (select →along path),Define by→select turns →give the valve of 50 to 100). →Ok done. And mirror it.

II Gland: 6.Create a 2D sketch on Front Plane as shown in the figure.

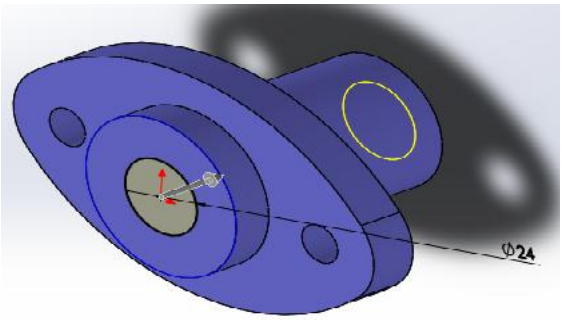
(Right click the Front plane>insert sketch and draw the 2D sketch).

Extrude to 12 mm and 10mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above





xtrude to 45 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above, Extrude cut use through all.(Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given below,

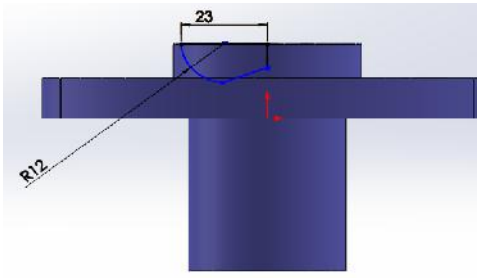


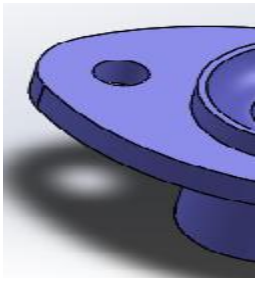
7.Create a 2D sketch on Front Plane as shown in the figure.

(Right click the Front plane>insert sketch and draw the 2D sketch).

Revolve, the sketch to 360 degree on top sketched line, by (Insert> Boss/Base>Revolve)

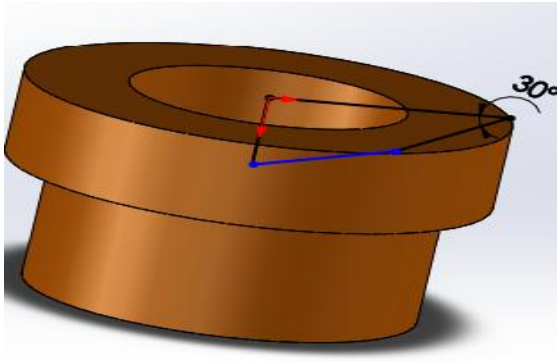
ok. As per given below figure.



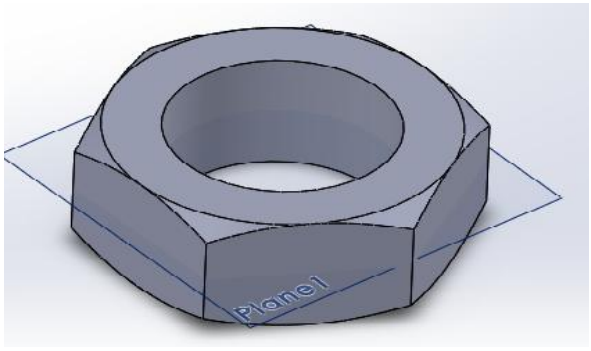


Below figures use as per the dimensions

III Neck bush:

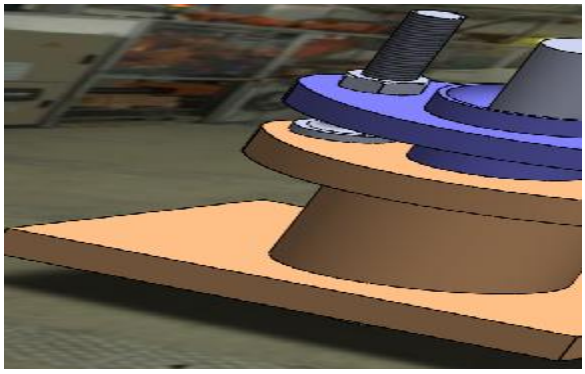


IV M12 Nut



V.Stud

Assembly model as per the dimensions:



PROCEDURE: PART DRAWING:

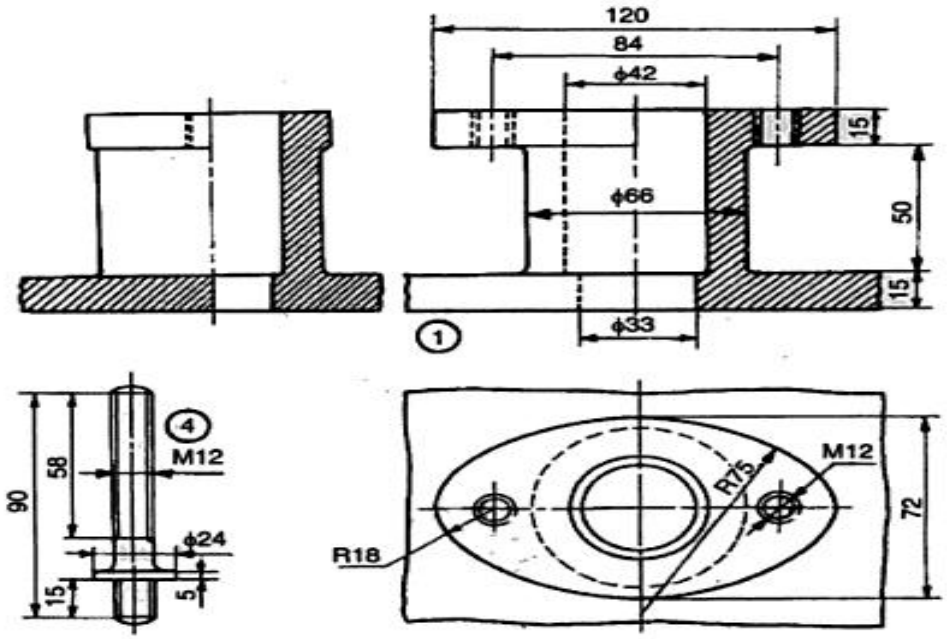
CYLINDER: ¬Using Pad, Cut and Round Commands the cylinder has been drawn.
NUT: ¬Using extrude, Cut and Round Commands the nut has been drawn.
GLAND BUSH: ¬Using extrude and Cut Commands the gland bush has been drawn.
PISTON ROD: ¬Using extrude and Cut Commands the piston rod has been drawn.
PACKING: ¬Using Shaft command the packing has been drawn.

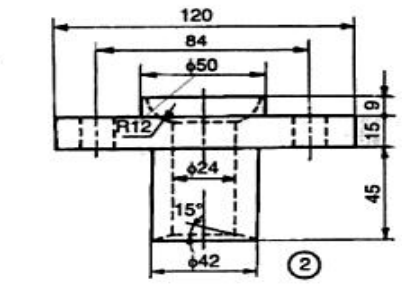
ASSEMBLY AND DETAILED DRAWING: ÷Ė

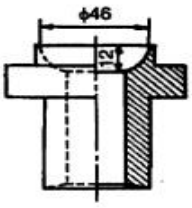
Using the Assembly and Drawing mode to make the respective views and bill of materials.

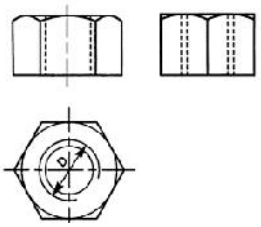
RESULT:

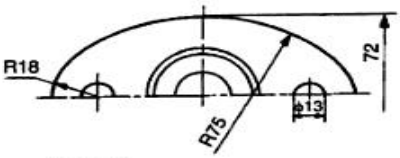
Thus the Detail View of the Stuffing Box and then its respective views have been drawn

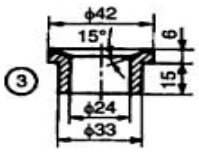












Parts list

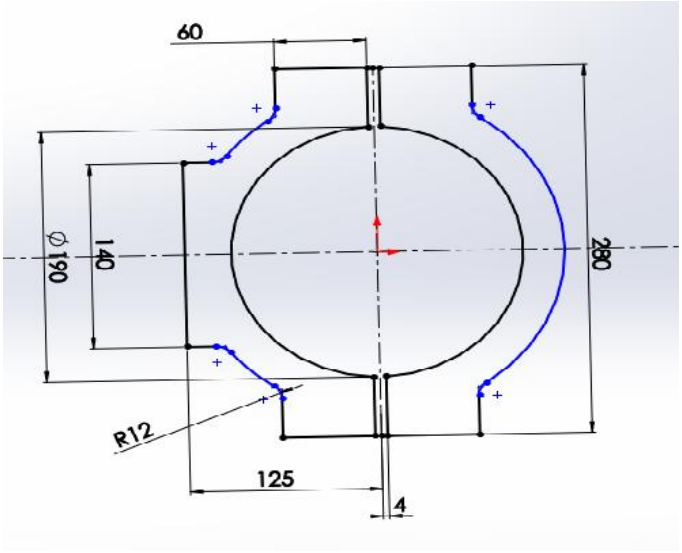
Part No.	Name	Matl	Qty
1	Body	CI	1
2	Gland	Brass	1
3	Bush	Brass	1
4	Stud	MS	2
5	Nut, M12	MS	2

6.Details and assembly of Eccentric using solid works software

AIM:
To model and assemble the Eccentric as per the dimensions given and also convert the 3D models into different views with Bill of materials.

Tools: Personal computer with Pentium IV processor with windows xp/windows-7 and solidworks software, Sketch, extrude , Shaft, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

1.Strap:



2D sketch as given above
Extrude by (Insert>Boss/Base>Extrude)) ok.

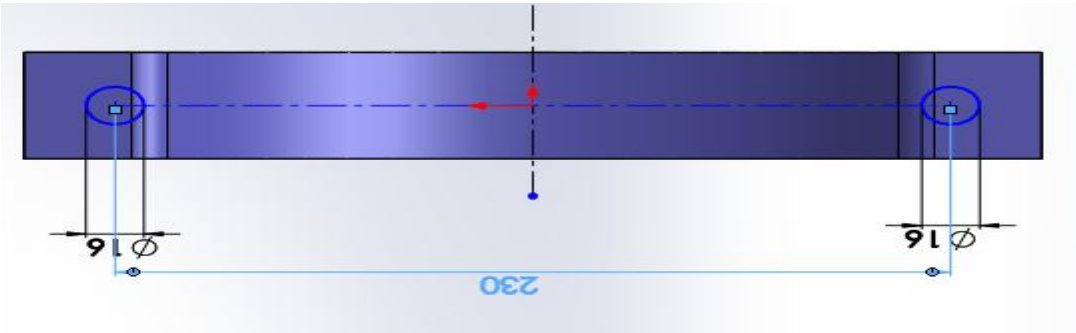
1.Create a 2D sketch on Front Plane as shown in the figure.

2.(Right click the Front plane>insert sketch and draw the 2D sketch).

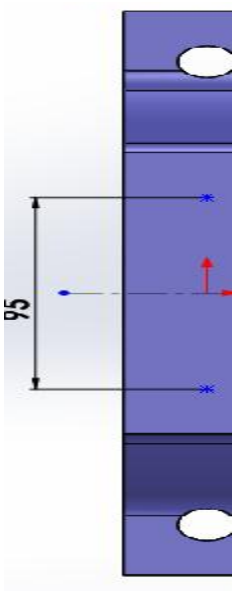
Note: All the 2D sketches drawn should be fully Defined and there should not be any under defined) and use (click Add Relation and Smart Dimensions.

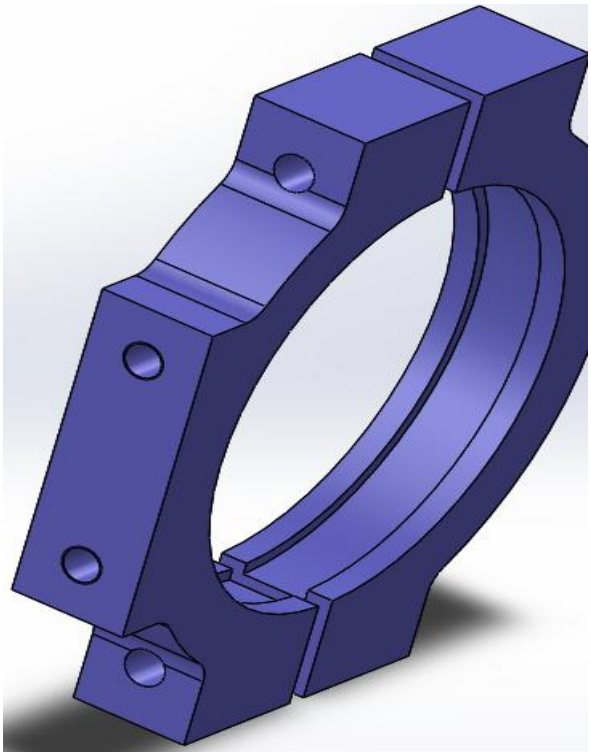
3. extrude to 45 mm (Select the face by (Enter Space bar> double click the Mid plane) and Draw the

4.select right plane and draw the 2D Sketch circles for hole both ends sides.



6. Create tapped hole M16X1.5mm one end side Insert>features>hole>wizard

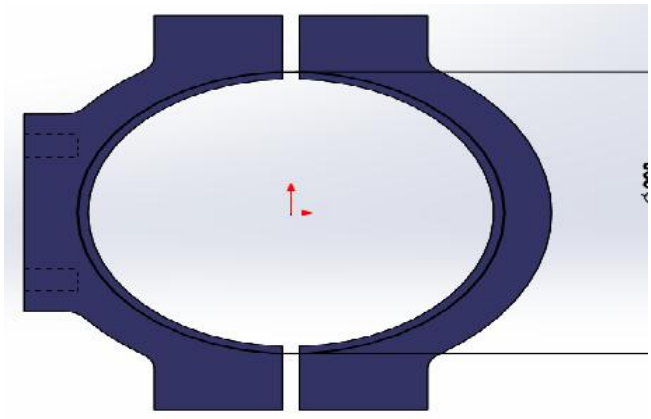


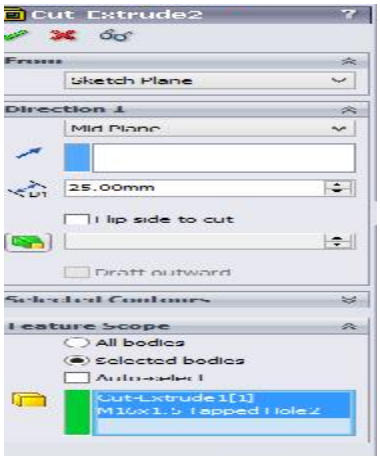


7. Create a 2D sketch on Front Plane as shown in the figure.

(Right click the Front plane>insert sketch and draw the 2D sketch).

Circle of 200 mm diameter and use cut extrude(select mid plane 25 mm.



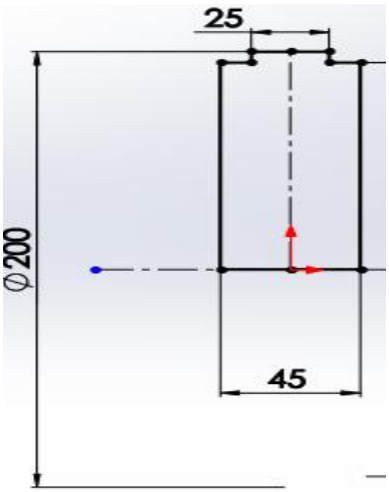


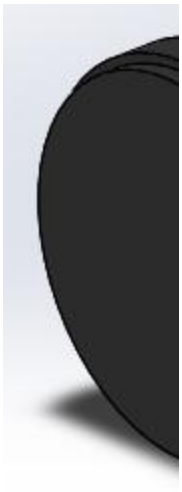
2.Sheave: Step.1 1.Create a 2D sketch on Front Plane as shown in the figure.

2.(Right click the Front plane>insert sketch and draw the 2D sketch).

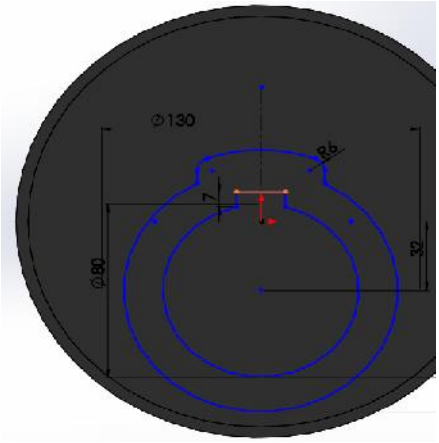
Note: All the 2D sketches drawn should be fully Defined and there should not be any under defined) and use (click Add Relation and Smart Dimensions.

Revolve, the sketch to 360 degree on top sketched line, by (Insert> Boss/Base>Revolve) ok.

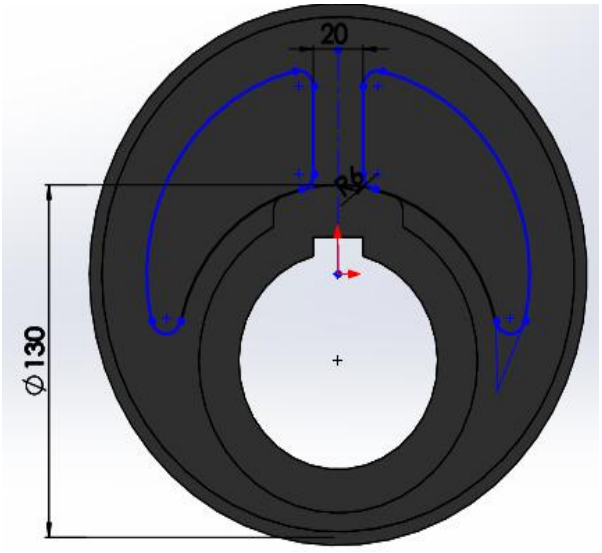


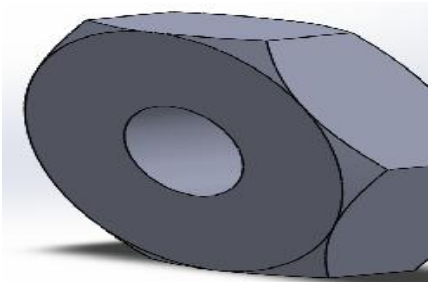


Create circle of 2D sketch as per the dimensions, on right plane and extrude to 20mm (Select the face by (Enter Space bar> double click the Normal plane) and Draw the 2D sketch as given above. Extrude by (Insert>Boss/Base>Extrude)) ok.



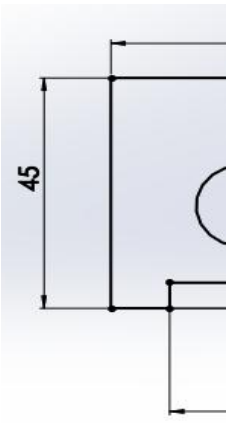
And select inner sketch use extrude cut through all.





3.Hexaganal Nuts as per the dimensions as per above first experiment.

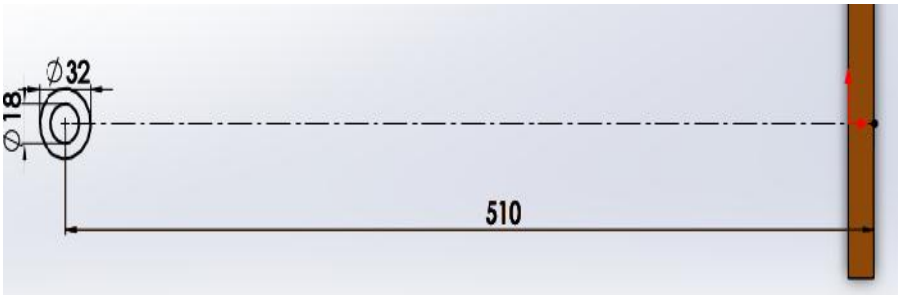
4.Packing strap as per the dimensions. and extrude 8 mm.



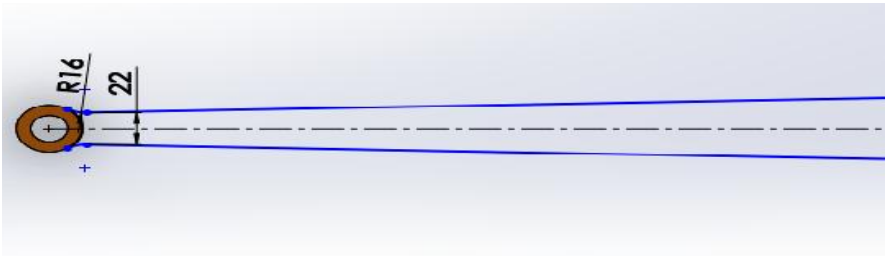
5. Rod:

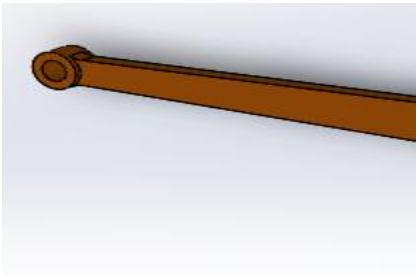
Use extrude option and select mid plane 30mm.

Use extrude option and select mid plane 20mm



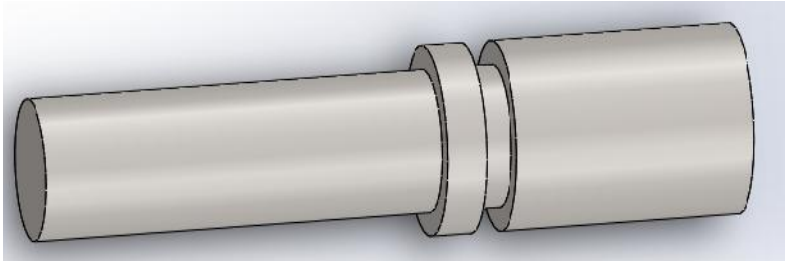
Use extrude option and select mid plane 12mm and center hole of 17mm ,distance of holes 95 mm size.



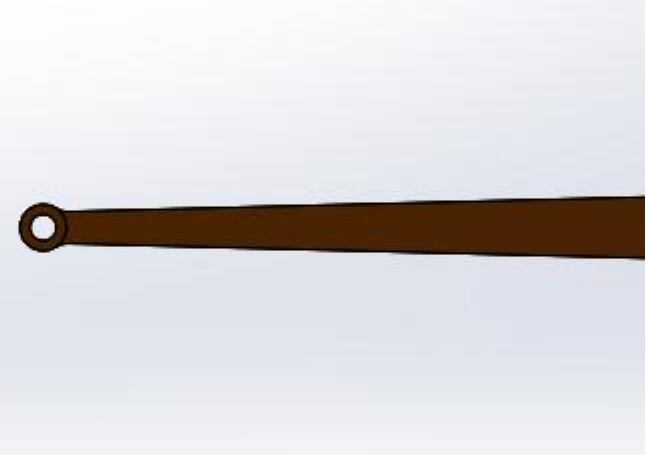


6.Bolt: Create bolt as per the dimensions:

7.Stud as per the dimensions:

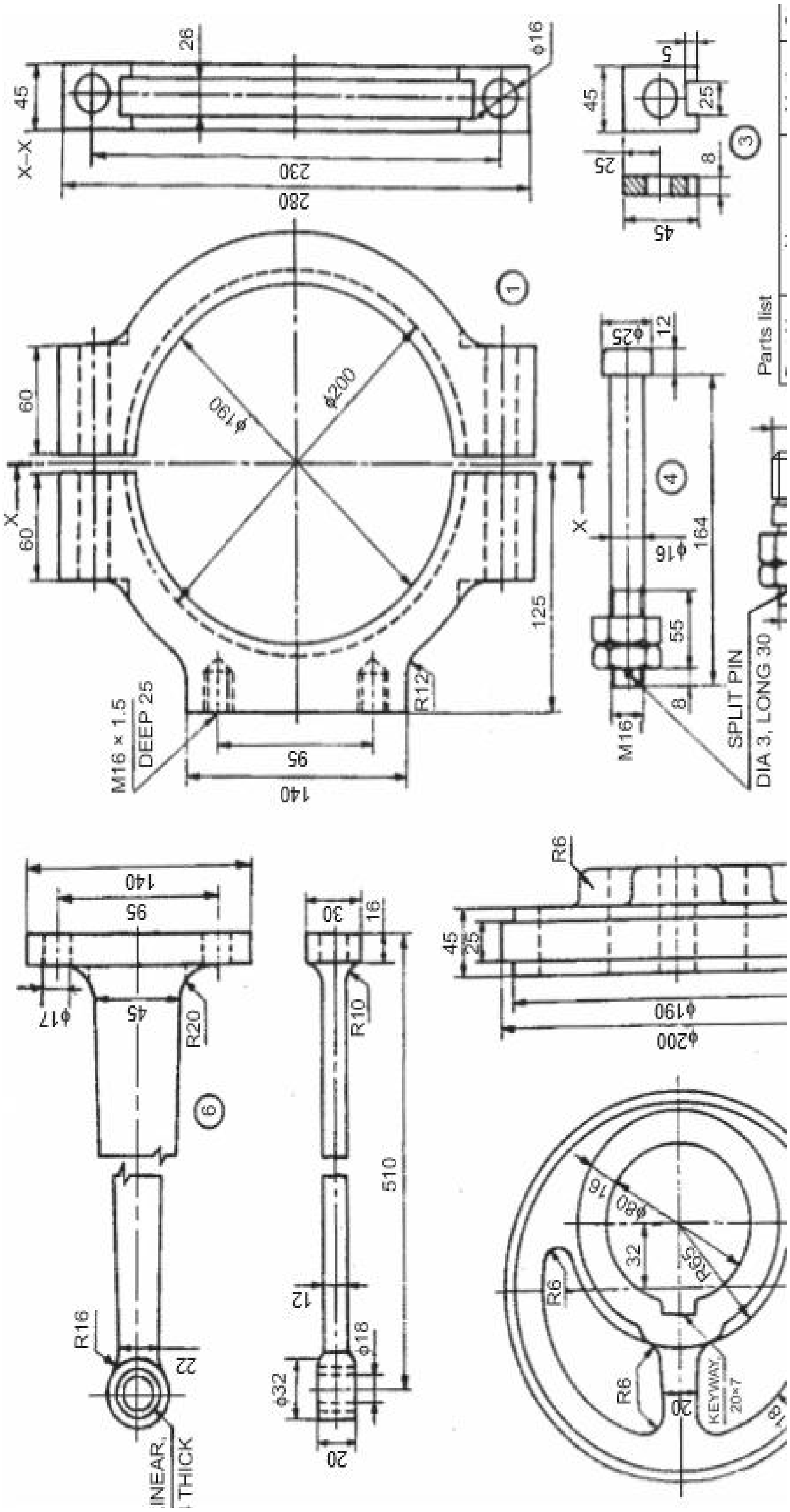


Assembly: Insert the components.



Procedure:

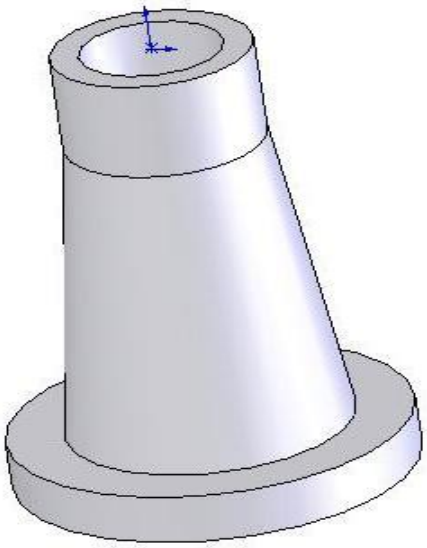
1. Model different parts of a eccentric using Extrude, Revolve and features.
2. Select the assembly in solid works main menu.
3. Using Insert component icon of property manager, insert base component & next components to be assemble.
4. Assemble using MATE Feature.
5. Continue the inserting the component & mating until the entire component are assembled.
6. Save the assembly.
7. From the main menu of solid works select the drawing option.
8. Drawing icon in main menu of Solid works
9. Select the drawing sheet format size as – A4 Landscape.
10. Using the model view manager browse the document to be open.
11. Click the view orientation from the model view manager & place the drawing view in the proper place in the sheet.
12.Using the placed view as parent view project the other or needed views
13. Move cursor to any one view and right click the mouse button.
14. Select the Table – BOM.
15. Place the BOM in the proper place in the drawing sheet.
16. Save the drawing sheet.


X–X
Parts list
SPLIT PIN
DIA 3, LONG 30
M16 × 1.5
DEEP 25
M16
KEYWAY
20×7
R6
R10
R12
R16
R20
ϕ16
ϕ17
ϕ18
ϕ25
ϕ32
ϕ80
ϕ190
ϕ200
164
230
280
510

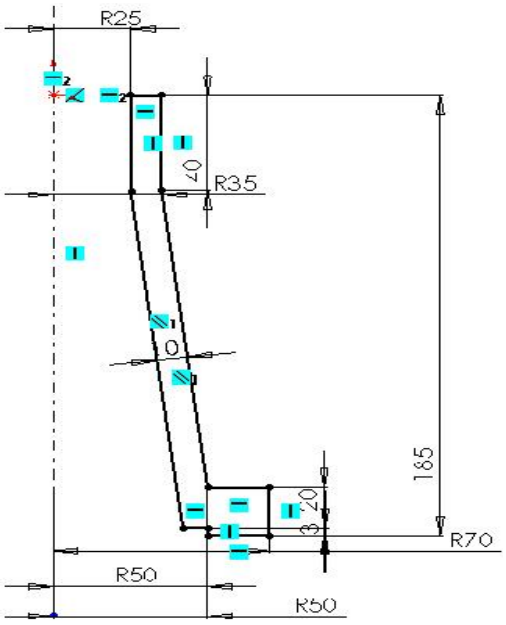
7.DETAILS AND ASSEMBLY OF SCREW JACK USING SOLIDWORKS SOFTWARE

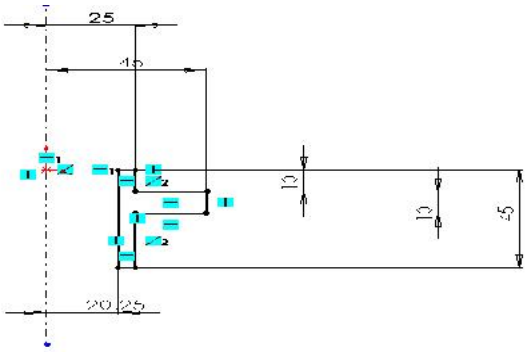
AIM:
To model and assemble the Screw jack as per the dimensions given and also convert the 3D model into different views with Bill of materials.
Tools: Personal computer with Pentium IV processor with windows xp/windows-7 and solid works software, Sketch, extrude , Shaft, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

Description about Screw jack:
A Screw Jack, manually operated is a contrivance to lift heavy object over a small height with a distinct Mechanical Advantages. It also serves as a supporting aid in the raised position. A screw Jack is actuated by a square threaded screw worked by applying a moderate effort at the end of a Tommy bar inserted into the hole of the head of the screw. The body of the screw jack has an enlarged circular base which provides a large bearing area. A gun metal nut is tight fitted into the body at the top. A screw spindle is screwed through the nut. A load bearing cup is mounted at the top of the screw spindle and secured to it by a washer and a CSK screw. When the screw spindle is rotated, the load bearing cup moves only up or down along with the screw spindle but will not rotate with it. The Tommy bar is inserted into the hole in the head of the screw spindle only during working and will be detached when not in use.

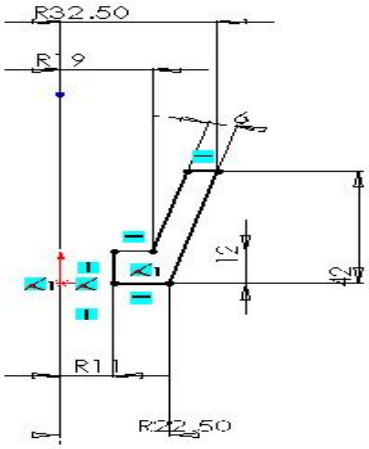


1.Body Use revolve feature

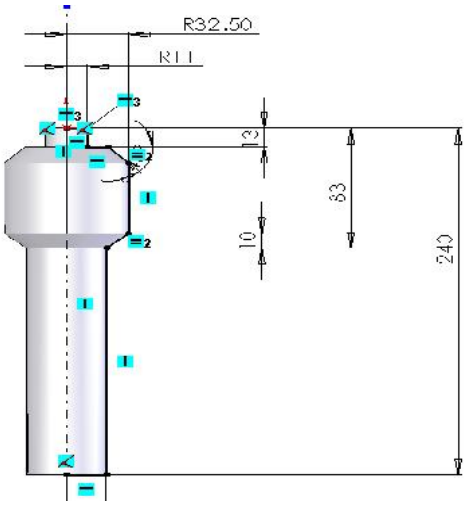




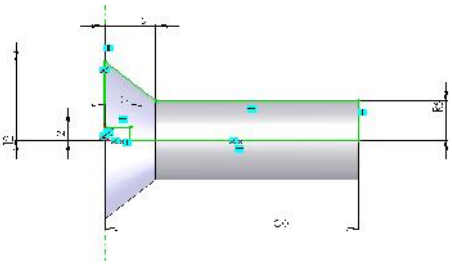
2.Nut Use Revolve ,
fillet feature,cut sweep



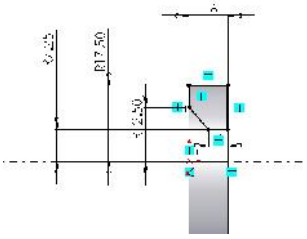
4.Cup Use Revolve feature



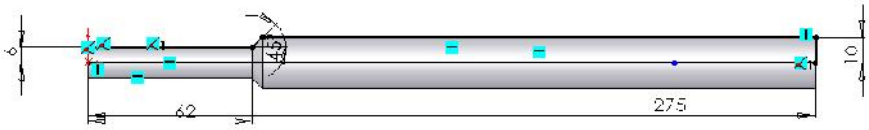
3. Screw use Revolve, Thread feature.



6. Screw use Revolve feature.



5. Washer use Revolve feature.

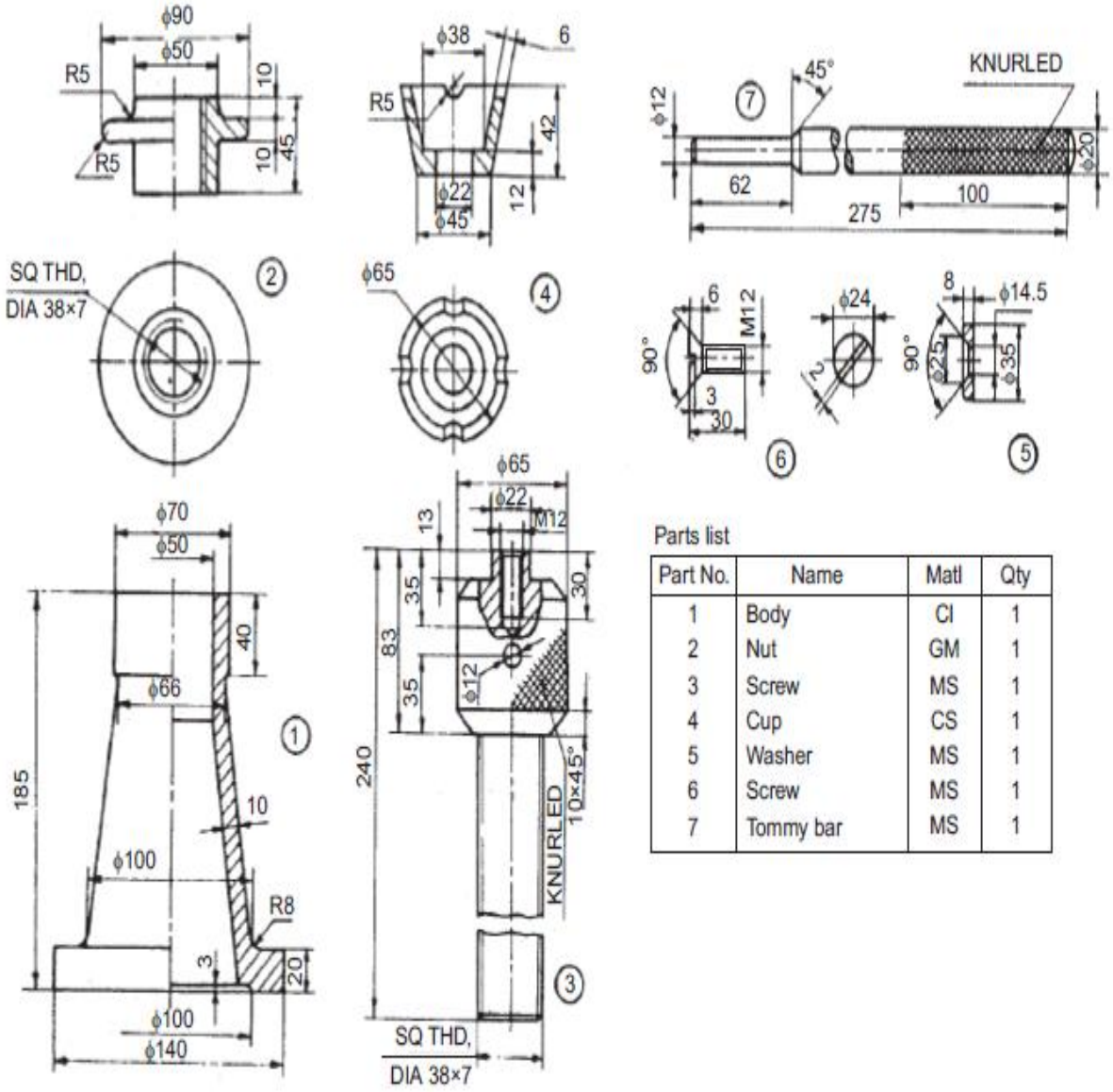


7. Tommy bar Tommy bar use Revolve feature.

Procedure:

1. Model different parts of a Screw Jack using Extrude, Revolve and features.
2. Select the assembly in solid works main menu.
3. Using Insert component icon of property manager, insert base component & next components to be assemble.
4. Assemble using MATE Feature.
5. Continue the inserting the component & mating until the entire component are assembled.
6. Save the assembly.
7. From the main menu of solid works select the drawing option.
8. Drawing icon in main menu of Solid works
9. Select the drawing sheet format size as – A4 Landscape.
10. Using the model view manager browse the document to be open.
11. Click the view orientation from the model view manager & place the drawing view in the proper place in the sheet.
12.Using the placed view as parent view project the other or needed views
13. Move cursor to any one view and right click the mouse button.
14. Select the Table – BOM.
15. Place the BOM in the proper place in the drawing sheet.
16. Save the drawing sheet.

Result:
Thus the given Screw Jack is modeled; assembled & different views are taken

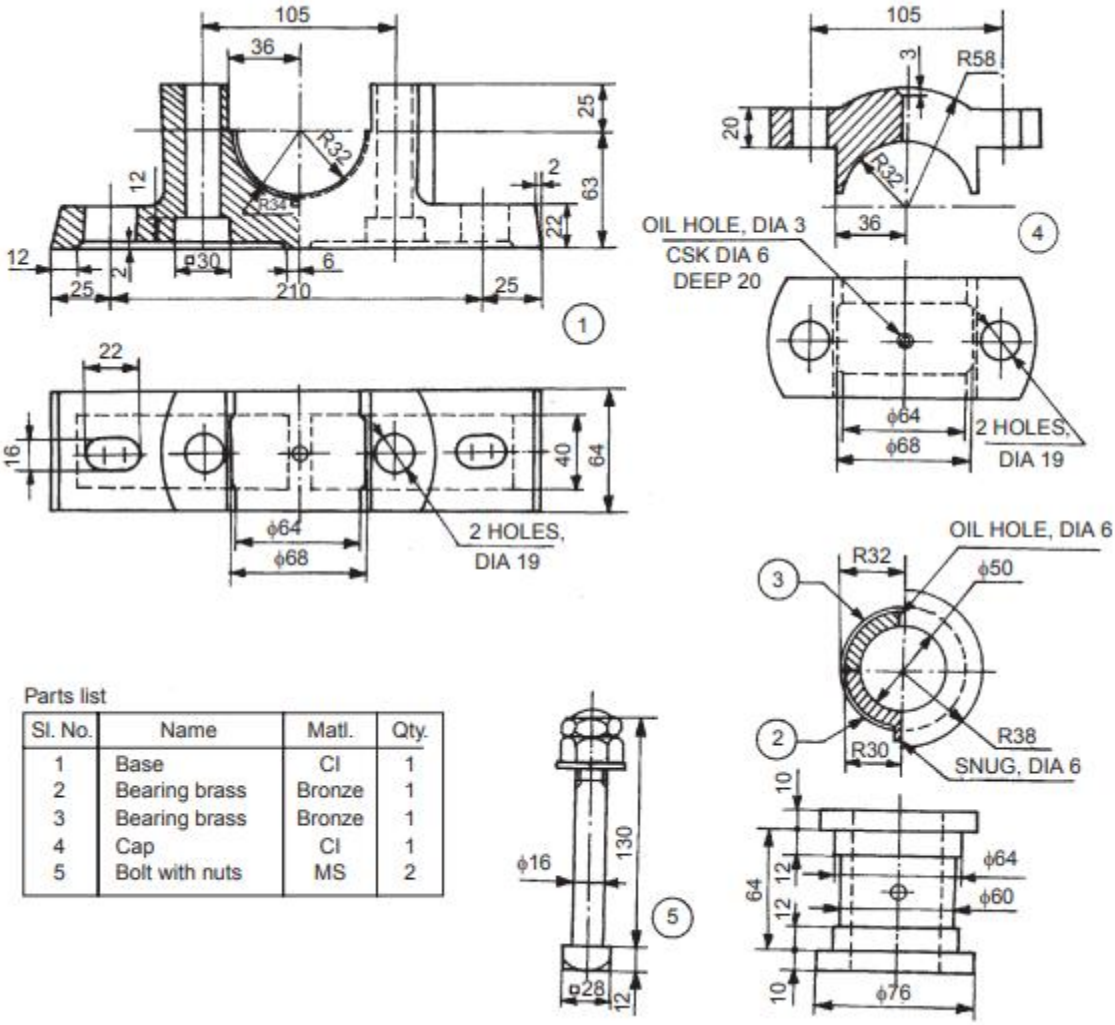


Parts list

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

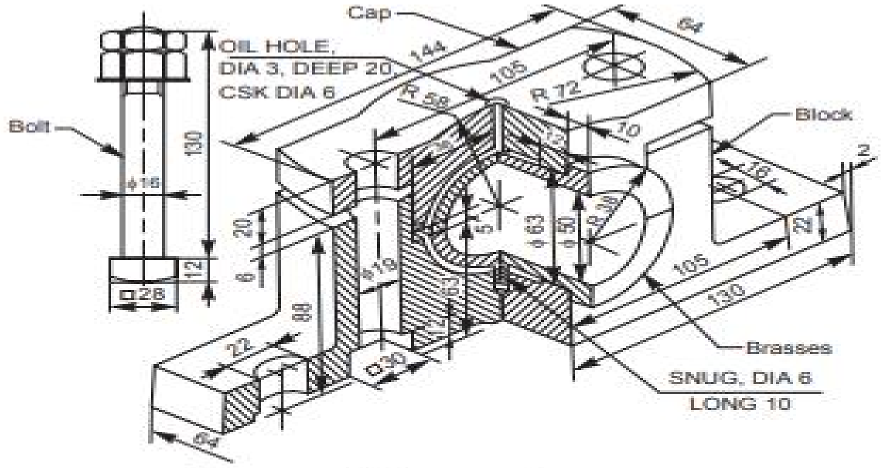
8.Draw and assembled Plummer block components with shuitable dimentions

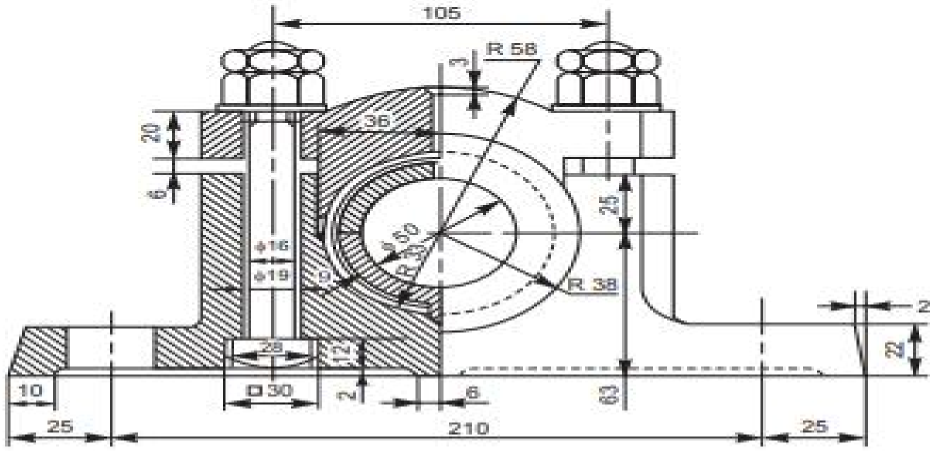
This bearing is used for long shafts, requiring intermediate support, especially when the shaft cannot be introduced into the bearing, end-wise.

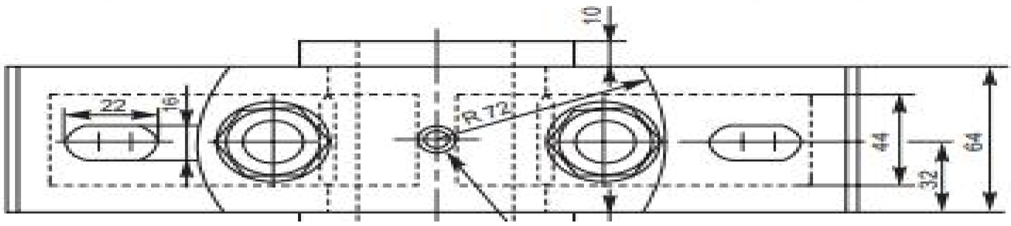


Parts list

Sl. No.	Name	Matl.	Qty.
1	Base	CI	1
2	Bearing brass	Bronze	1
3	Bearing brass	Bronze	1
4	Cap	CI	1
5	Bolt with nuts	MS	2


Cap
OIL HOLE,
DIA 3, DEEP 20
CSK DIA 6
Bolt
Block
Brasses
SNUG, DIA 6
LONG 10





9.DETAILS AND MODELING OF TAIL STOCK USING SOLIDWORK

AIM: To model the parts of a tail stock by creating modifying and manipulating various features and assemble them to create an assembled model using a feature solid modelling and detailing.

COMMANDS USED: Sketch, extrude , Shaft, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

PROCEDURE STEPS:

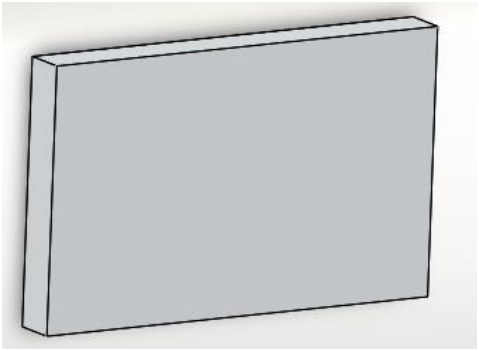
- The drawing with required dimensions are thoroughly observed.
- The tail stock body is first created by using protrusion lofted protrusion commands.
- The cutouts for barrels, clamping nuts lock nut inserts screw spindle are made on the body using cutout sweep cutout command.
- The screw spindle is created by protrusion helical cutout commands including in screw thread.
- The clamping plate, clamping plate, clamping bolt, spindle bearing, washers, nuts are created by using protrusion cutout helical cutout commands.
- The tail stoke center is created by using protrusion and lofted protrusion as per taper dimensions provided the hand wheel model is created by protrusion revolved protrusion cutout commands and saved.
- An assembly file is opened and the parts are imported into the file for an assembly.
- As tailstock body, the base part, the barrel screw, spindle, lamp plate, spindle bearing ,tailstock centre, hand wheel are assembled by assigning various relationship. options like Make axial align plane align Insert convert
- Thus the assembled model of tailstock is created and the parts,models are drafted with dimensions in a draft file and saved.

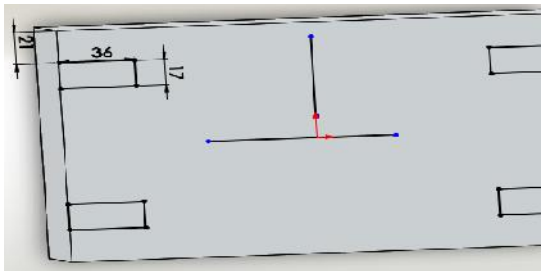
I.Body:

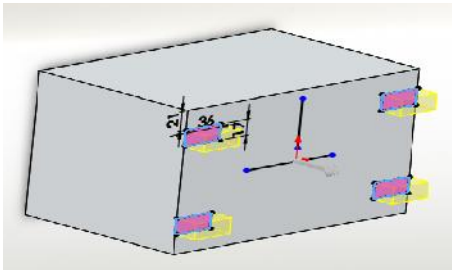
Create a 2D sketch on Front Plane as shown in the figure.

(Right click the Front plane>insert sketch and draw the 2D sketch).

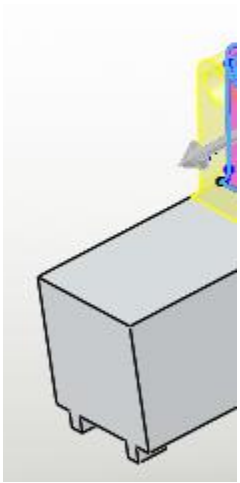
Extrude to 25 mm,20mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

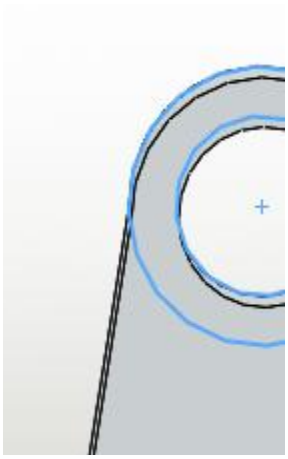




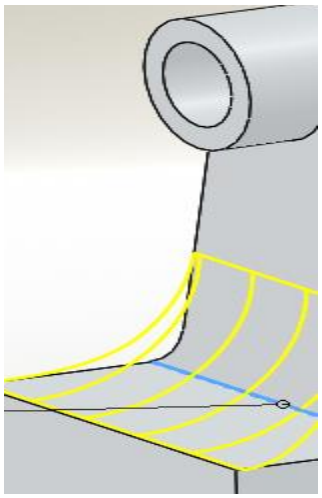


2.Take right plane draw the 2D sketch and extrude 38mm





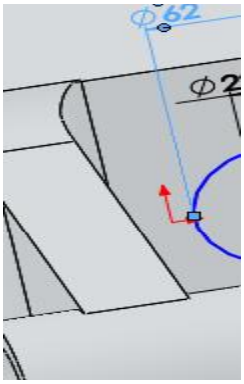
Extrude of 75 mm one side and 167 mm other side.

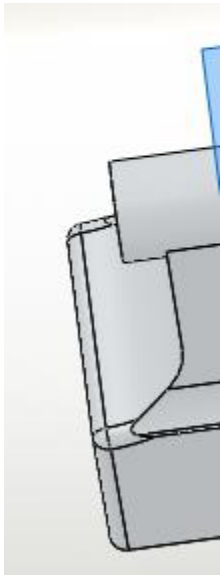


Select plane and offset of 146mm,after that create sketch. And extrude to 30 mm.

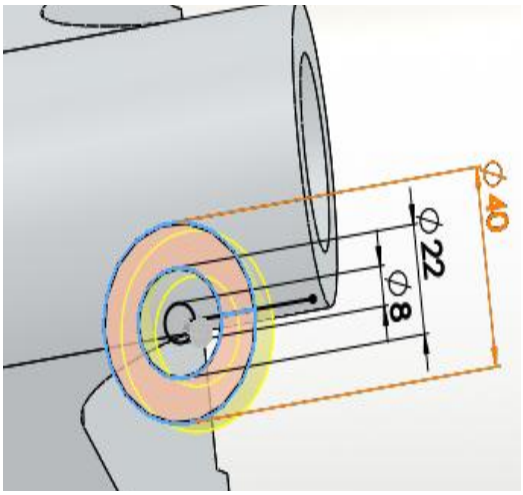
Once gain offset plane 190 mm. and fillet 60mm

Extrude 13 mm.





Create plane offset of 195 mm.

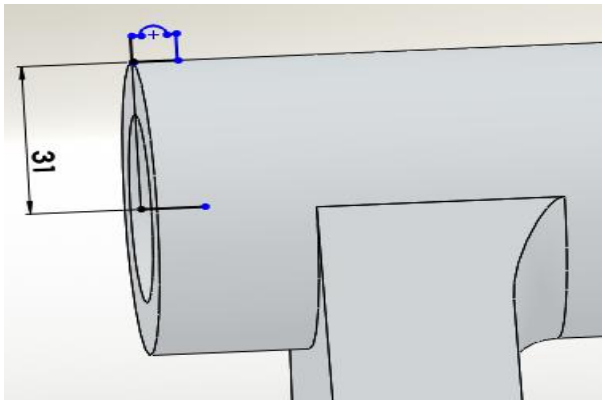


Create a 2D sketch on Front Plane as shown in the figure.

(Right click the Front plane>insert sketch and draw the 2D sketch).

Extrude to 33 mm, (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

Create Same size of boss at 4 mm distance with opposite direction.

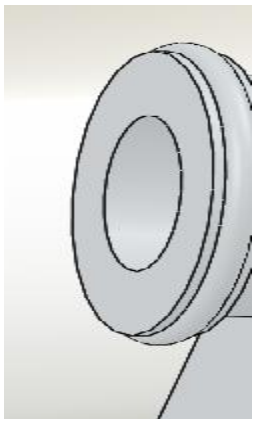


Create a 2D sketch on Front Plane as shown in the figure.

(Right click the Front plane>insert sketch and draw the 2D sketch).

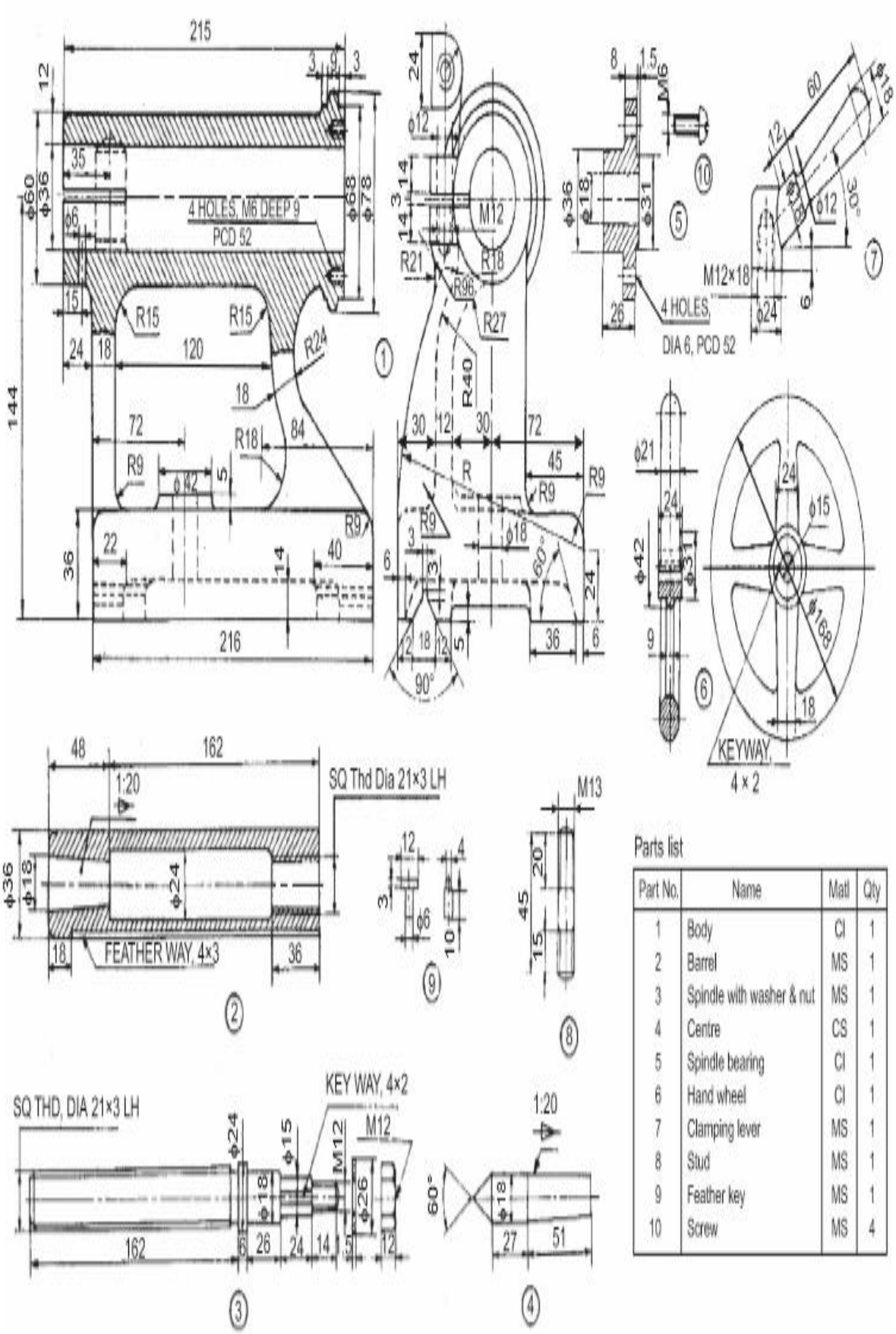
Revolve, the sketch to 360 degree on top sketched line, by (Insert> Boss/Base>Revolve)

ok. As per given below figure.



Remaining parts create as per the above said.

RESULT: Thus the parametric feature based sold model of tailstock parts are created, assembled and the assembled model is saved for details drawing.



Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Barrel	MS	1
3	Spindle with washer & nut	MS	1
4	Centre	CS	1
5	Spindle bearing	CI	1
6	Hand wheel	CI	1
7	Clamping lever	MS	1
8	Stud	MS	1
9	Feather key	MS	1
10	Screw	MS	4