SVR ENGINEERING COLLEGE

Experiment No. | | **Date:** | | |

DEPATMENT OF MECHANICAL ENGINEERING

CAD/CAM (15A03710)

LABORATORY MANUAL

Prepared by

DEPATMENT OF MECHANICAL ENGINEERING
SVR ENGINEERING COLLEGE
AYYALURIMETTA, NANDYAL-518501

GENERAL INSTRUCTIONS

1. Students should wear the uniform and closed foot wear. Students inappropriately dressed for lab, at the instructor's discretion, are denied access.
2. Eating, drinking and smoking are prohibited in the laboratory at all times.
3. Never work in the laboratory without proper supervision by an instructor.
4. Never carry out unauthorized experiments. Come to the laboratory prepared. If you are unsure about what to do, please ask the instructor.
5. Except the scientific calculator, any other electronic devices are not permitted to use inside the Laboratory.
6. Any damage to any of the equipment/instrument/machine caused due to carelessness, the cost will be fully recovered from the individual (or) group of students.

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Experiment No. ☐☐ **Date:** ☐☐ ☐☐ ☐☐

List of Experiments:

Experiment No.   **Date:**

Part Modeling

Solid Works

BASICS OF SOLIDS MODELING WITH SOLIDWORKS

Introduction

Solid Works is the state of the art in computer-aided design (CAD). Solid Works represents an object in a virtual environment just as it exists in reality, i.e., having volume as well as surfaces and edges. This, along with exceptional ease of use, makes Solid Works a powerful design tool. Complex three-dimensional parts with contoured surfaces and detailed features can be modeled quickly and easily with Solid-Works. Then, many parts can be assembled in a virtual environment to create a computer model of the finished product. In addition, traditional engineering drawings can be easily extracted from the solids models of both the parts and the final assembly. This approach opens the door to innovative design concepts, speeds product development, and minimizes design errors. The result is the ability to bring high-quality products to market very quickly.

CONSTRAINT-BASED SOLIDS MODELING

The constraint-based solids modeling used in Solid Works makes the modeling process intuitive. The 3-D modeling begins with the creation of a 2-D sketch of the profile for the cross section of the part. The sketch of the cross section begins much like the freehand sketch of the face of an object. The initial sketch need not be particularly accurate; it needs only to reflect the basic geometry of the part’s cross-sectional shape. Details of the cross section are added later. The next step is to constrain the two-dimensional sketch by adding enough dimensions and parameters to completely define the shape and size of the two-dimensional profile. The name constraint-based modeling arises because the shape of the initial two-dimensional sketch is “constrained” by adding dimensions to the sketch. Finally, a three-dimensional object is created by revolving or extruding the two-dimensional sketched profile. Figure 1 shows the result of revolving a simple L-shaped cross section by 270o about an axis and extruding the same L-shaped cross section along an axis.

In either case, these solid bodies form the basic geometric solid shapes of the part. Other features can be added subsequently to modify the basic solid shape. Once the solids model is generated using Solid Works, all of the surfaces have been automatically defined, so it is possible to shade it in order to create a photorealistic appearance. It is also easy to generate two-dimensional orthographic views of the object. Solid modeling is like the sculpting of a virtual solid volume of material. Because the volume of the object is properly represented in a solids model, it is possible to slice through the object and show a view of the object that displays the interior detail (sectional views). Once several solid objects have been created, they can be assembled in a virtual environment to confirm their fit and to visualize the assembled product. Solids models are useful for purposes other than visualization. The solids model contains a complete mathematical representation of the object, inside and out. This mathematical representation is easily converted into specialized computer code that can be used for stress analysis, heat transfer analysis, fluid-flow analysis, and computer-aided manufacturing. Getting Started in Solid Works Introduction and Reference Solid Works Corporation developed Solid Works® as a three-dimensional, feature-based, solids-modeling system for personal computers.

Solid modeling represents objects in a computer as volumes, rather than just as collections of edges and surfaces. Features are three-dimensional geometries with direct analogies to shapes that can be machined or manufactured, such as holes or rounds. Feature-based solid modeling creates and modifies the geometric shapes of an object in a way that represents common manufacturing processes. This makes Solid Works a very powerful and effective tool for engineering design. As per other computer programs, Solid Works organizes and stores data in files. Each file has a name followed by a period (dot) and an extension. There are several file types used in Solid Works, but the most common file types and their extensions are Part files .prt or .sldprt Assembly files .asm or .sldasm Drawing files .drw or .slddrw

Part files are the files of the individual parts that are modeled. Part files contain all of the pertinent information about the part. Because Solid Works is a solids-modeling program, the virtual part on the screen will look very similar to the actual manufacture part. Assembly files are created from several individual part files that are virtually assembled (in the computer) to create the finished product.

Assembly files are the two dimensional engineering drawing representations of both the part and assembly file. The drawings should contain all of the necessary information for the manufacture of the part, including dimensions, part tolerances, and so on. The part file is the driving file for all other file types. The modeling procedure begins with part files. Subsequent assemblies and drawings are based on the original part files. One advantage of Solid Works files is the feature of dynamic links. Any change to a part file will automatically be updated in any corresponding assembly or drawing file.

Tool bars:

The Sketch toolbar contains tools to set up and manipulate a sketch.

• The Sketch Tools toolbar contains tools to draw lines, circles, rectangles, arcs, and so on.

• The Sketch Relations toolbar contains tools for constraining elements of a sketch by using dimensions or relations.

• The Features toolbar contains tools that modify sketches and existing features of a part.

• The Standard toolbar contains the usual commands available for manipulating files (Open, Save, Print, and so on), editing documents (Cut, Copy, and Paste), and accessing Help.

The Standard Views toolbar contains common orientations for a model.

• The View toolbar contains tools to orient and rescale the view of a part.

- **Line**: creates a straight line.

• **Center point Arc**: creates a circular arc from a center point, a start point and an end point.

• **Tangent Arc**: creates a circular arc tangent to an existing sketch entity.

• 3 Pt Arc: creates a circular arc through three points.

• **Circle**: creates a circle.

• **Splines**: creates a curved line that is not a circular arc.

• **Polygon**: creates a regular polygon.

• **Rectangle:** creates a rectangle.

• **Point creates**: a reference point that is used for constructing other sketch entities.

• **Centerline**: creates a reference line that is used for constructing other sketch entities.

Convert Entities: creates a sketch entity by projecting an edge, curve, or contour onto the sketch plane.

• **Mirror**: reflects entities about a centerline.

• **Fillet:** creates a tangent arc between two sketch entities by rounding an inside or an outside corner.

• **Offset:** Entities creates a sketch curve that is offset from a selected sketch entity by a specified distance.

• **Trim**: removes a portion of a line or curve.

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Experiment No. | | **Date:** | | | |

1.Modeling of Component in 3D – V block

AIM: To draw the detail view of part drawing of the simple component (V-BLOCK) as shown in the figure by using Soliworks software.

HARDWARE REQUIRED:

1. CPU
2. A colour monitor with highest 64 bit colour display and with screen resolution 1024 by 768 pixels.
3. A scroll mouse.

SOFTWARE REQUIRED:

1. Windows XP operating system
2. Solid works

COMMANDS USED:

Ex: Line, Circle, Erase, Trim, Mirror, Move, Region, Extrude, Subtract.

PROCEDURE:

Study the given drawing completely and find out the front view of the given Isometric object.
Draw the required front view of the object with specified dimensions.
Extrude the drawn section using extrude command for the given dimension.
Next select the appropriate plane and draw the other sections in similar way.
Also remove the materials where ever needed using subtract command.
Chamfering is done by the chamfer command.
All individual objects are combined together by using union command.

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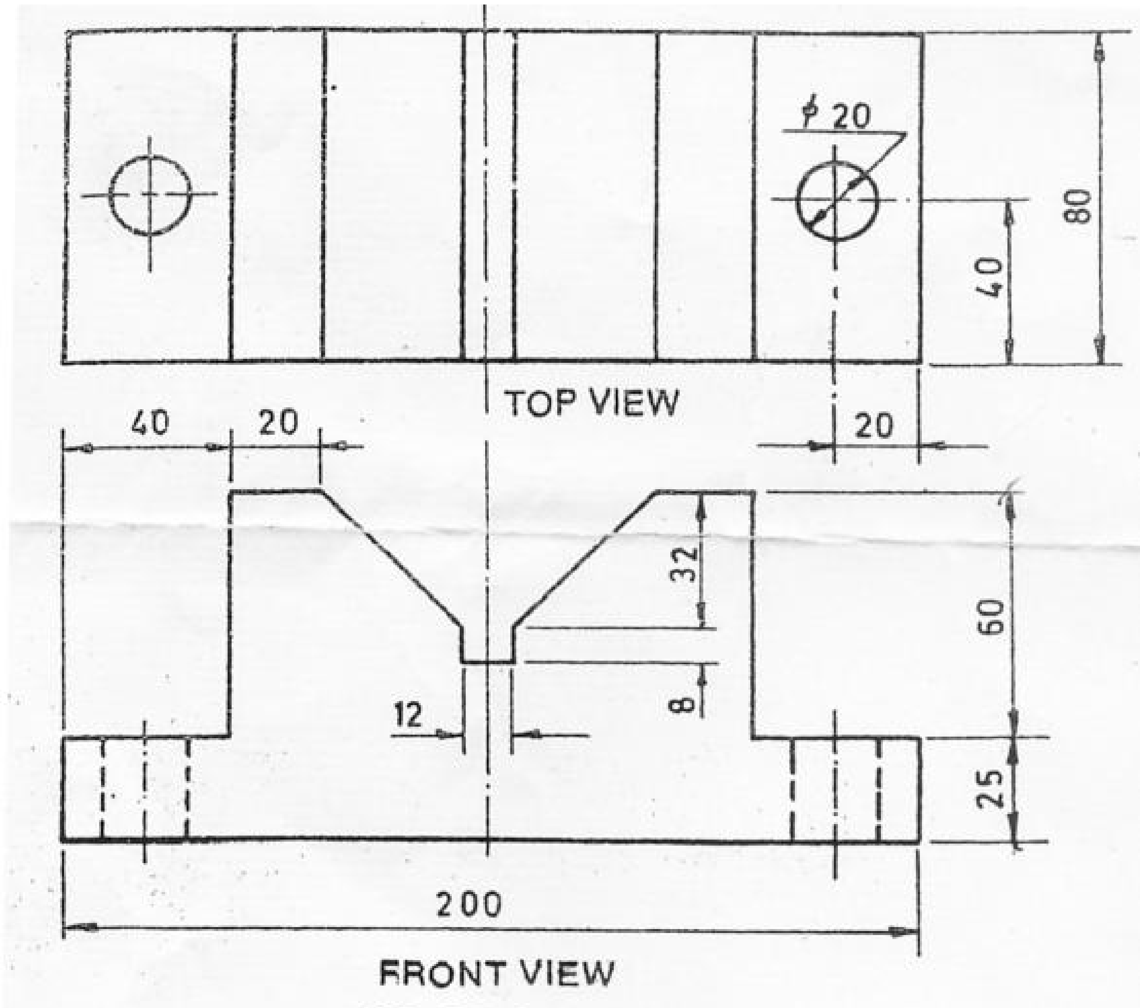

V- BLOCK

RESULT:
Thus the detailed view of part drawing of the simple component (V-BLOCK) is drawn by using the Solidworks software.

2.Modeling of Component in 3D – Open Bearing

OPEN BEARING

AIM:
To draw the detail view of part drawing of the simple component (OPEN BEARING) as shown in the figure by using Solidworks software.

HARDWARE REQUIRED:
1. CPU
2. A colour monitor with highest 64 bit colour display and with screen resolution 1024 by 768 pixels.
3. A scroll mouse.

SOFTWARE REQUIRED:
1. Windows XP operating system
2. Solidworks

COMMANDS USED:

Ex: Line, Circle, Erase, Trim, Mirror, Move, Region, Extrude, Subtract, Union.

PROCEDURE:

• Study the given drawing completely and find out the front view of the given Isometric object.
• Draw the required front view of the object with specified dimensions.
• Extrude the drawn section using extrude command for the given dimension.
• Next select the appropriate plane and draw the other sections in similar way.
• Also remove the materials where ever needed using subtract command.
• Chamfering is done by the chamfer command.
• All individual objects are combined together by using union command.

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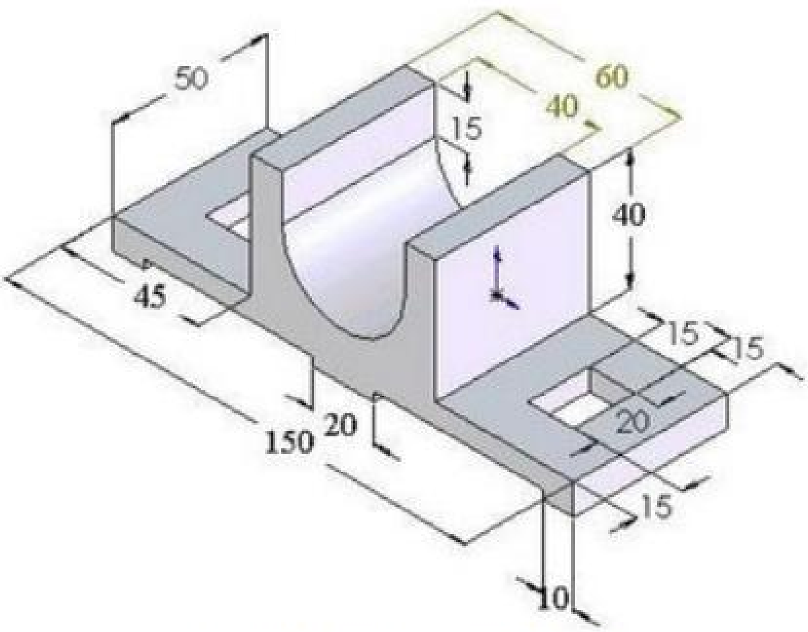
Experiment No. **Date:**



OPEN BEARING

RESULT:
Thus the detailed view of part drawing of the simple component (OPEN BEARING) is drawn by using the Solidworks software.

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Experiment No. | | | **Date:** | | | | | | |

3.Modeling of Component in 3D – Angular block

ANGULAR BLOCK

AIM:
To draw the detail view of part drawing of the simple component
(ANGULAR BLOCK) as shown in the figure by using Solidworks software.

HARDWARE REQUIRED:
1. CPU with Pentium IV processor.
2. A colour monitor with highest 64bit colour display and with screen resolution 1024 by 768 pixels.
3. A scroll mouse.

SOFTWARE REQUIRED:
1. Windows XP operating system
2. Solidworks

COMMANDS USED:
Ex: Line, Circle, Erase, Trim, Mirror, Move, Region, Extrude, Subtract, Union.

PROCEDURE:
• Study the given drawing completely and find out the front view of the given Isometric object.
• Draw the required front view of the object with specified dimensions.
• Extrude the drawn section using extrude command for the given dimension.
• Next select the appropriate plane and draw the other sections in similar way.
• Also remove the materials where ever needed using subtract command.
• Chamfering is done by the chamfer command.
• All individual objects are combined together by using union command.

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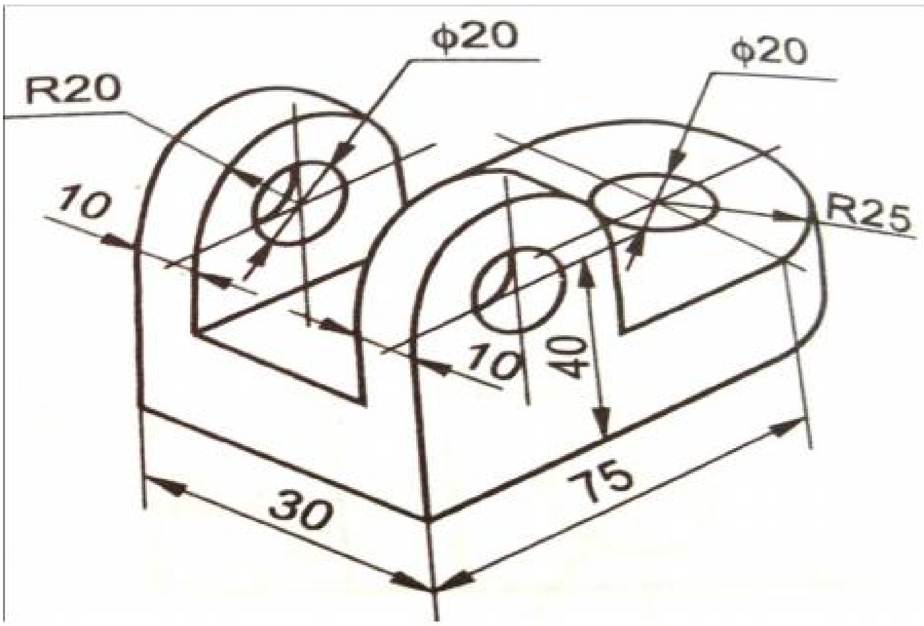
Experiment No. | | | **Date:** | | | | | | |



ANGULAR BLOCK

RESULT:
Thus the detailed view of part drawing of the simple component (ANGULAR BLOCK) is drawn by using the Solidworks software.

4.Modeling of Component in 3D –Dovetail Guide

DOVETAIL GUIDE

AIM:
To draw the detail view of part drawing of the simple component (DOVETAIL Guide) as shown in the figure by using Solidworks.

HARDWARE REQUIRED:
1. CPU.
2. A colour monitor with highest 64 bit colour display and with screen resolution 1024 by 768 pixels.
3. A scroll mouse.

SOFTWARE REQUIRED:
1. Windows XP operating system
2. Solidworks.

COMMANDS USED:
Ex: Line, Circle, Erase, Trim, Mirror, Move, Region, Extrude, Subtract, Union.

PROCEDURE:
- Study the given drawing completely and find out the front view of the given Isometric object.
- Draw the required front view of the object with specified dimensions.
- Extrude the drawn section using extrude command for the given dimension.
- Next select the appropriate plane and draw the other sections in similar way.
- Also remove the materials where ever needed using subtract command.
- Chamfering is done by the chamfer command.
- All individual objects are combined together by using union command.

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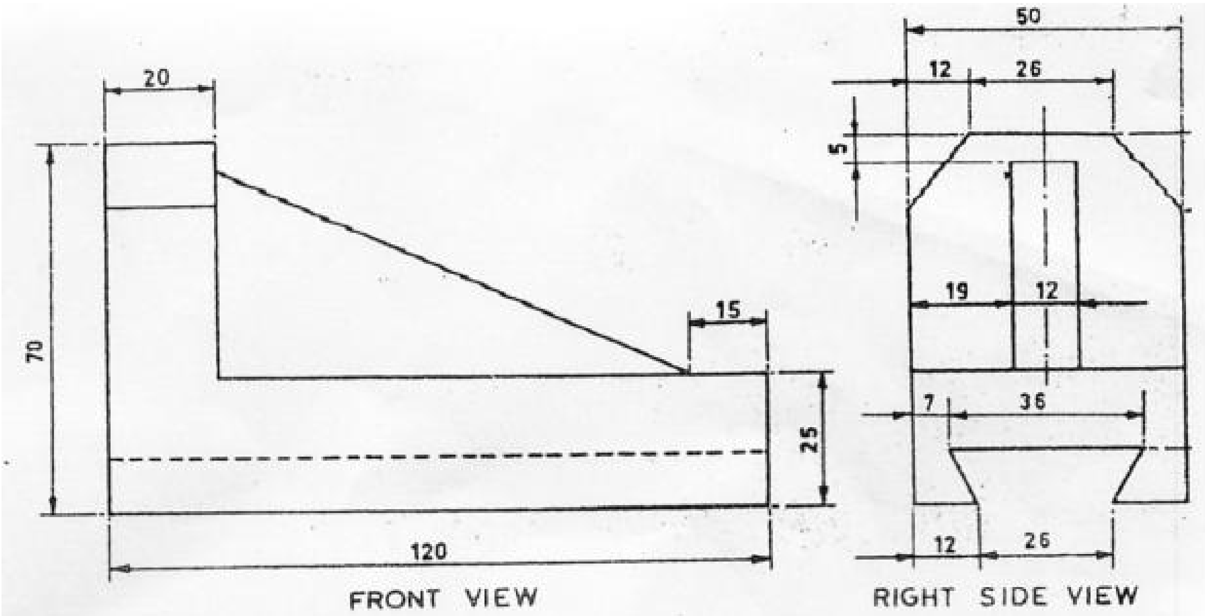


RESULT:
Thus the detailed view of part drawing of the simple component (DOVETAIL GUIDE) is drawn by using the Solidworks software

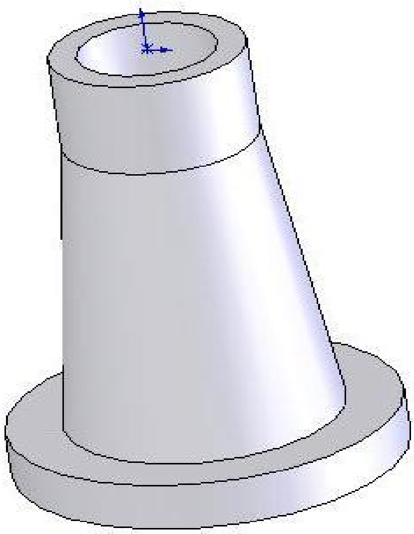
Experiment No. | | | **Date:** | | | | | | |

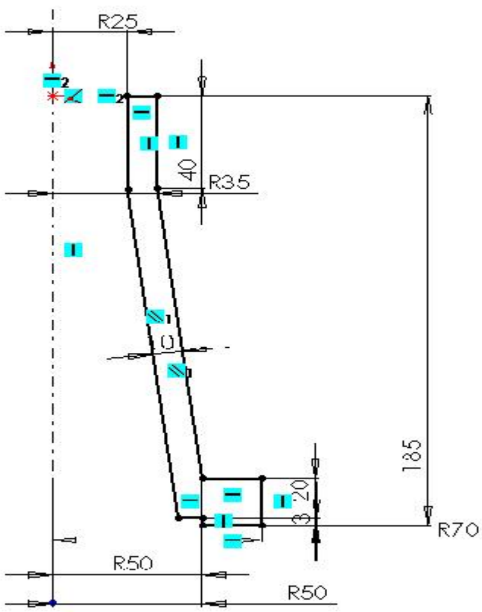
Assembly modeling

5. Assembly of a screw jack parts

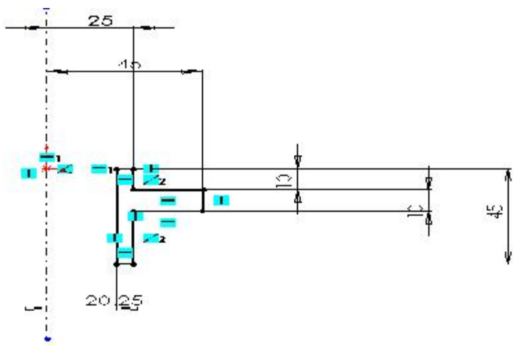
AIM:
To model and assemble the Screw jack as per the dimensions given and also convert the 3D model into different views.

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.

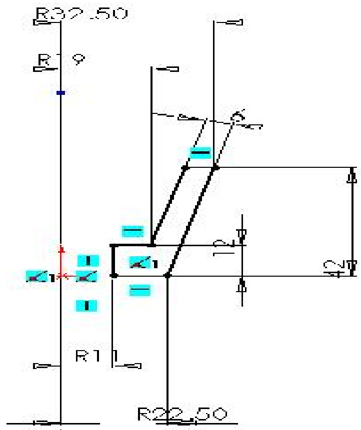




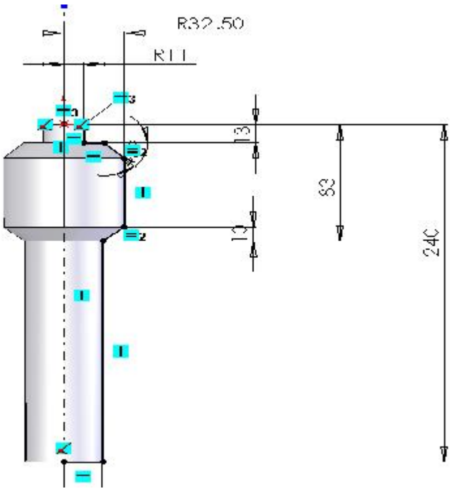
1.Body Use revolve feature



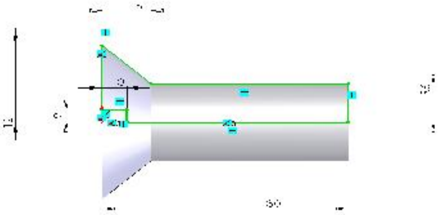
**2.Nut Use Revolve ,
fillet feature,cut sweep**



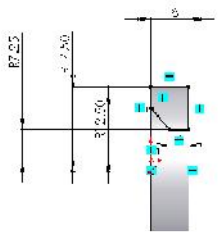
4.Cup Use Revolve



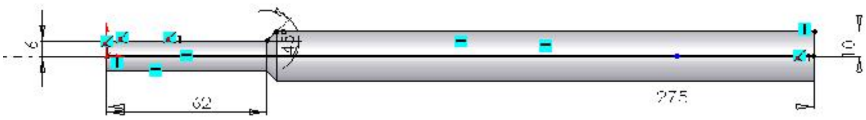
**3. Screw use
Revolve, Thread
feature.**



**6. Screw use
Revolve feature.**



**5. Washer use
Revolve feature.**



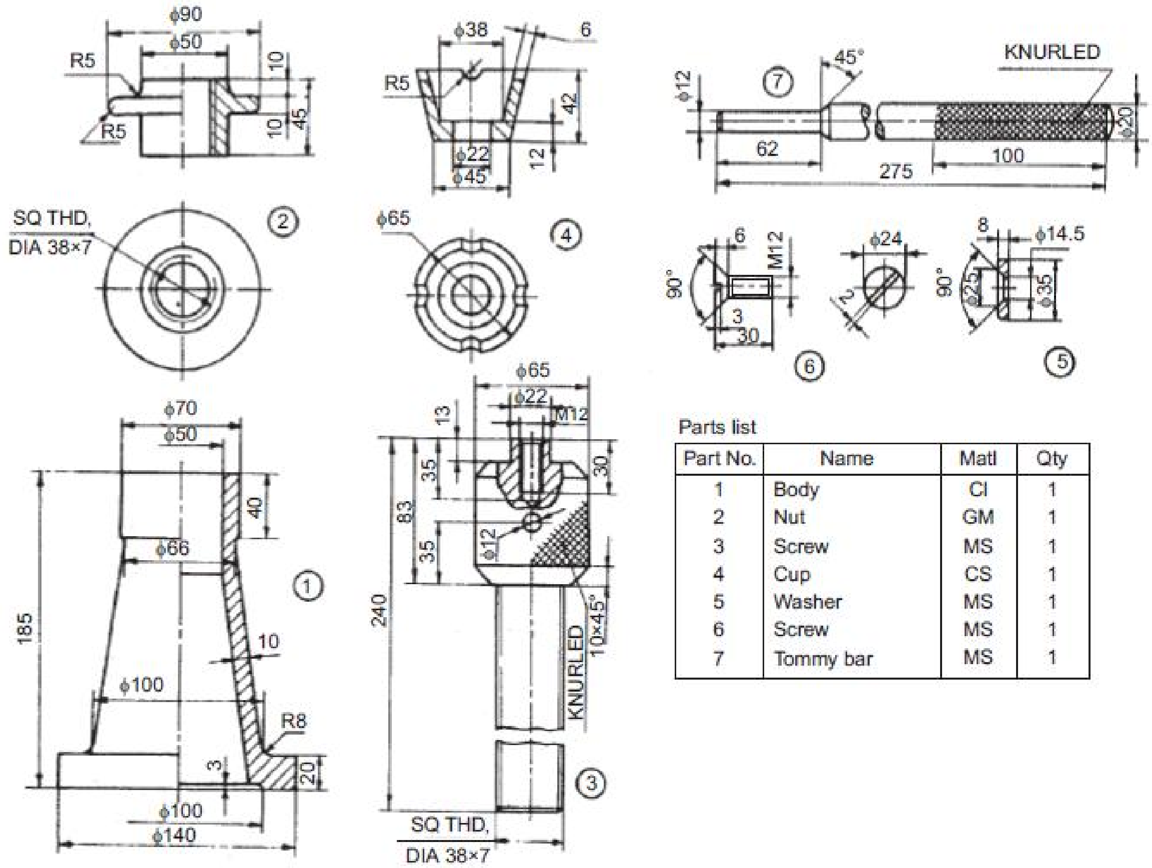
**7. Tommy bar Tommy bar use
Revolve feature.**

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.

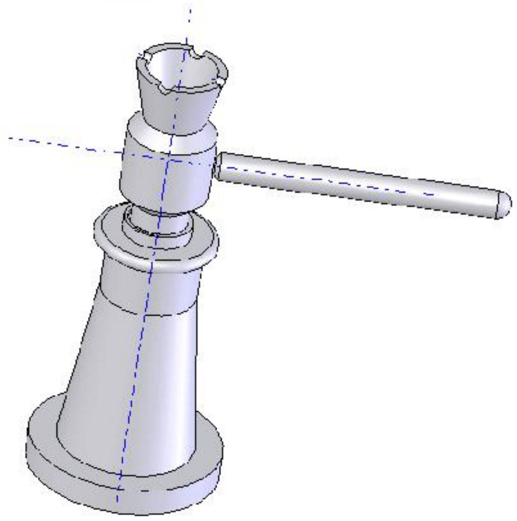
Result:

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

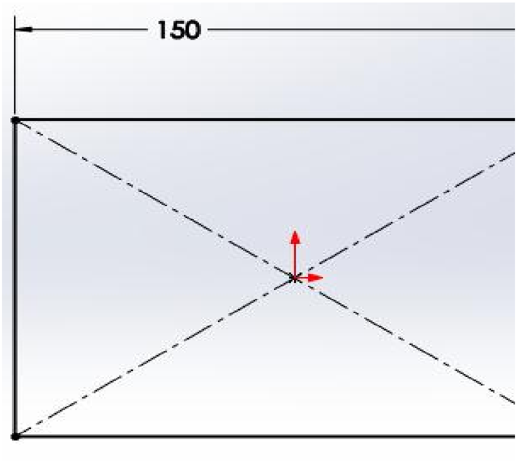


Experiment No. **Date:**

6.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,

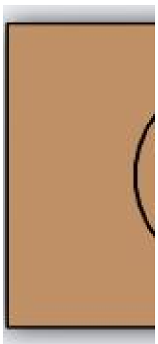


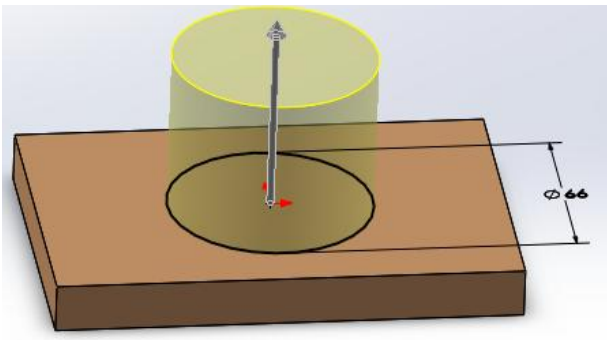
1.Create a 2D sketch on Front Plane as shown in the figure.

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.

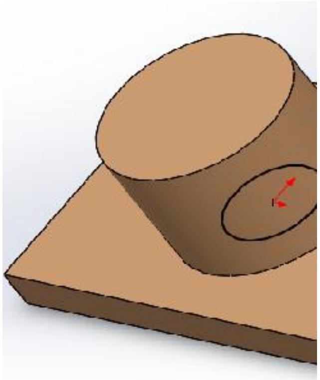
3. extrude to 15 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

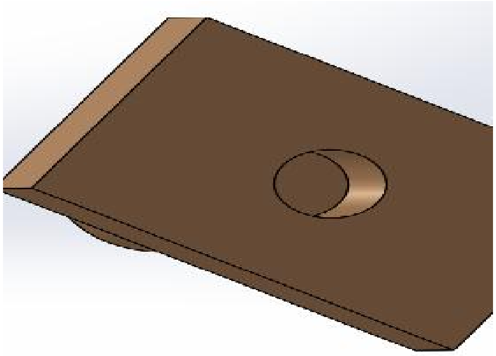




4.extrude to 50 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

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

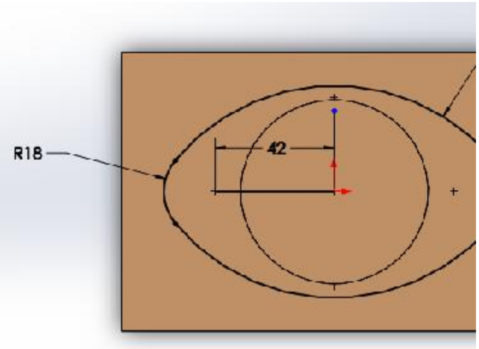


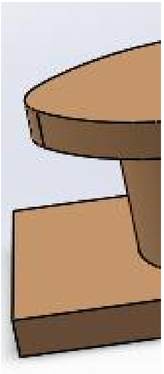


5.Create a 2D sketch on Front Plane as shown in the figure.

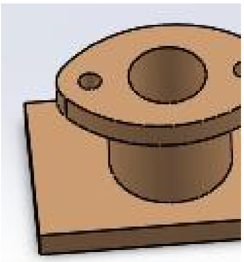
(Right click the Front plane>insert sketch and draw the 2D sketch).

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).

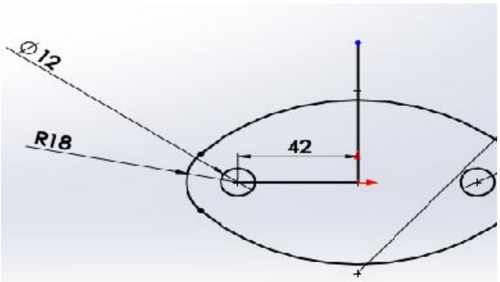
Extrude cut to 51 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above

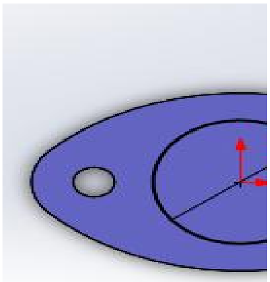
7. Create thread,Take sweep (insert→boss/base →sweep) command and give Select profile and there relative circle, and Select path there relative curve,Options→orientation /twist type (select →along path),Define by→select turns →give the valve of 50 to 100). →Ok done. And mirror it.

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

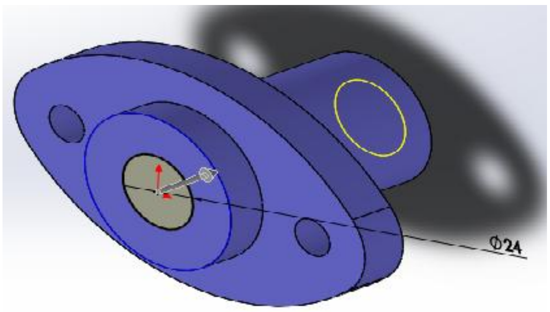
(Right click the Front plane>insert sketch and draw the 2D sketch).

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





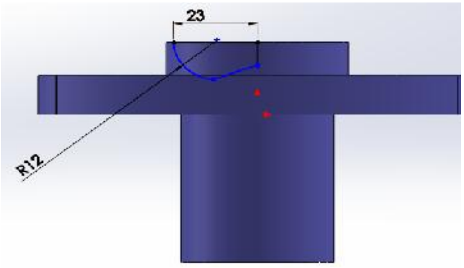
Extrude to 45 mm (Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given above, Extrude cut use through all.(Select the face by (Enter Space bar> double click the plane) and Draw the 2D sketch as given below,

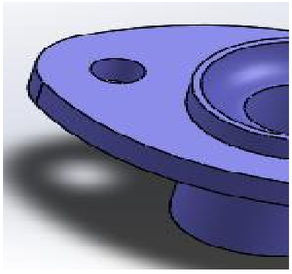


7.Create a 2D sketch on Front Plane as shown in the figure.

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

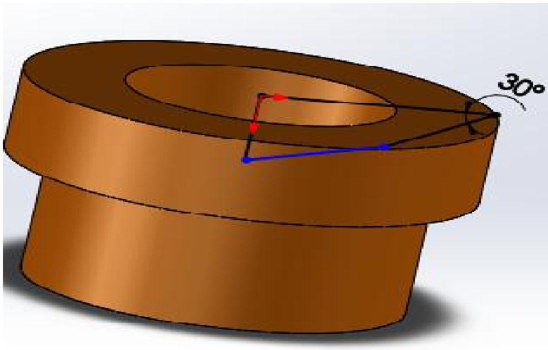
Revolve, the sketch to 360 degree on top sketched line, by (Insert> Boss/Base>Revolve) ok. As per given below figure.



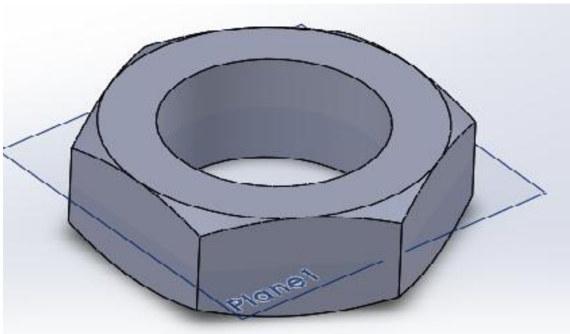


Below figures use as per the dimensions

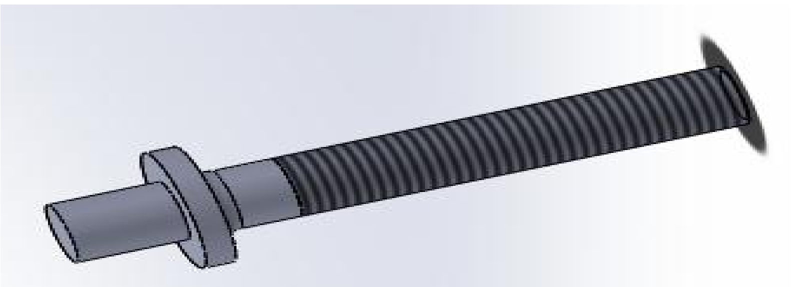
III Neck bush:



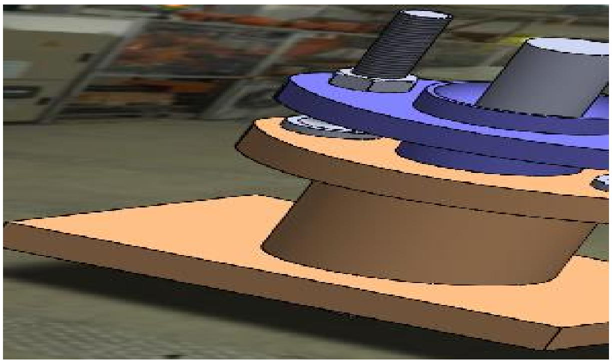
IV M12 Nut



V. Stud



Assembly model as per the dimensions:



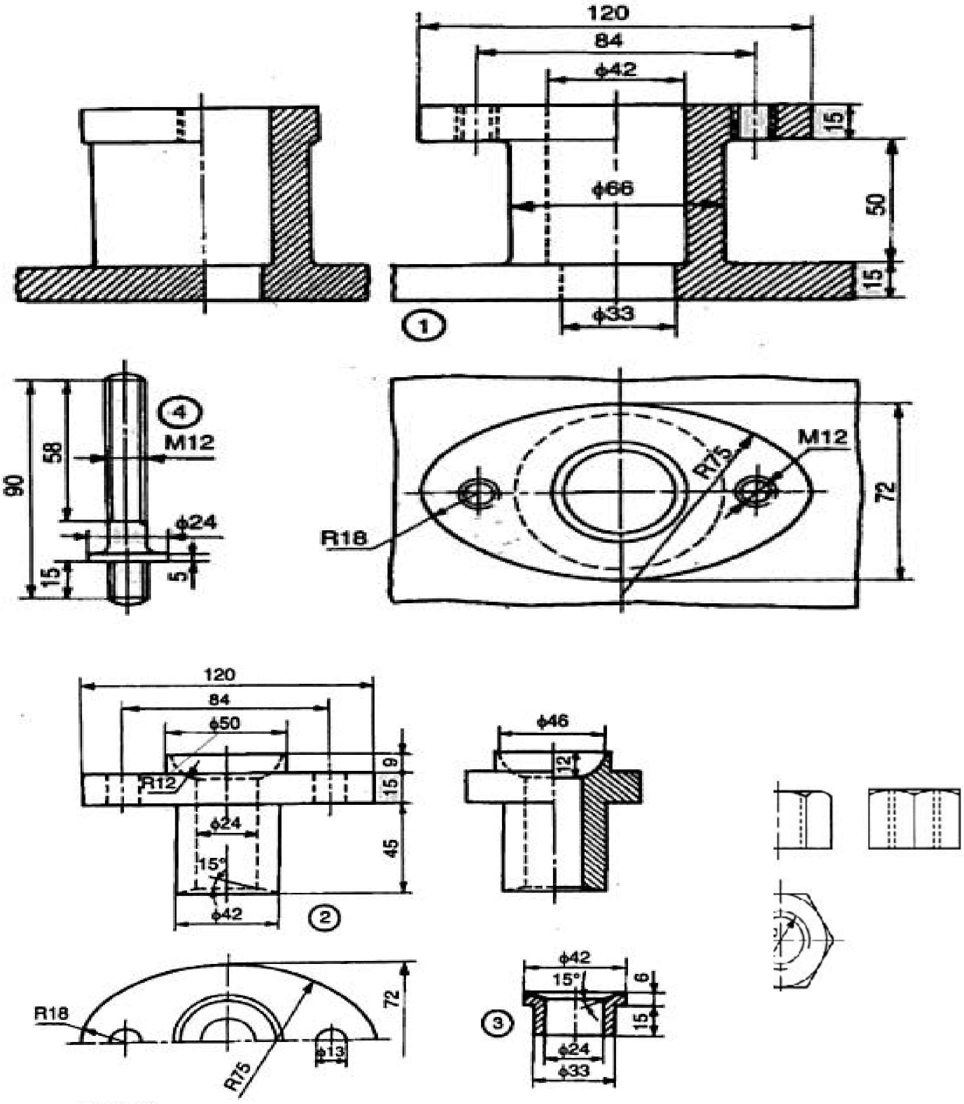
PROCEDURE: PART DRAWING:

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

Experiment No.   **Date:**



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

Experiment No. **Date:**

7.DETAILS AND ASSEMBLY OF FOOT STEP BEARING USING SOLIDWORKS SOFTWARE

AIM - To develop the part drawing of foot step bearing in the orthographic representation using Auto cad.

HARDWARE REQUIRED:
1. CPU
2. A colour monitor with highest 64 bit colour display and with screen resolution 1024 by 768 pixels.
3. A scroll mouse.

SOFTWARE REQUIRED:
1. Windows XP operating system
2. Solidworks

COMMANDS USED:

Ex: Line, Circle, Erase, Trim, Mirror, Move, Region, Extrude, Subtract, Union.

PROCEDURE:

- Study the given drawing completely and find out the front view of the given Isometric object.
- Draw the required front view of the object with specified dimensions.
- Extrude the drawn section using extrude command for the given dimension.
- Next select the appropriate plane and draw the other sections in similar way.
- Also remove the materials where ever needed using subtract command.
- Chamfering is done by the chamfer command.
- All individual objects are combined together by using union command.

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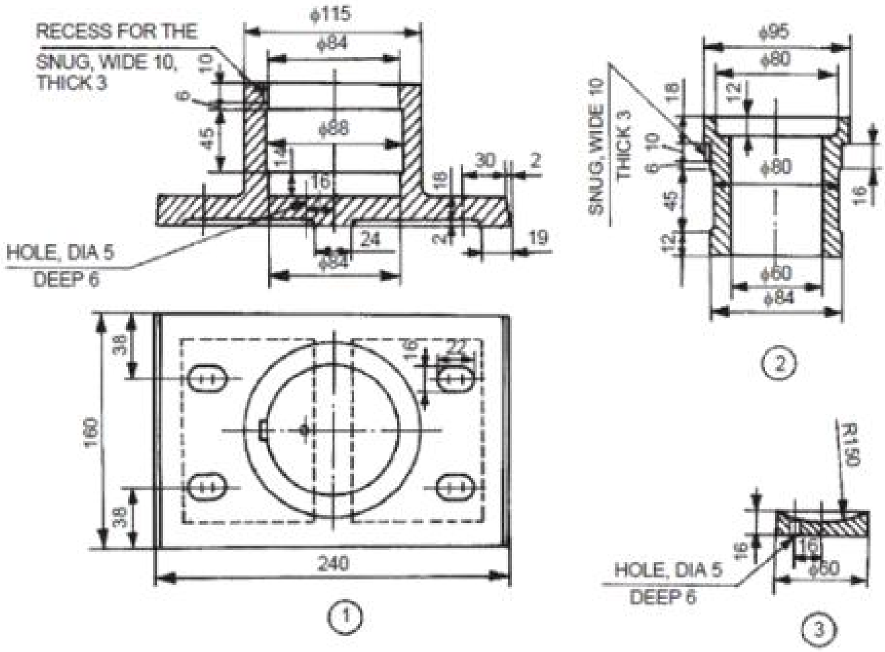
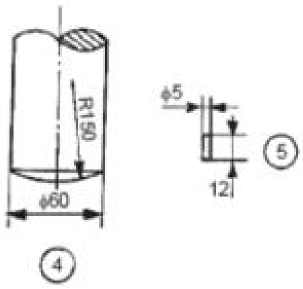
Experiment No. | | | **Date:** | | | | | |



Parts list

Sl. No.	Name	Matl.	Qty.
1	Body	Cast iron	1
2	Bush	Brass	1
3	Disc	P Bronze	1
4	Shaft	Mild steel	1
5	Pin	Mild steel	1



RESULT:
Thus the Detail View of the Footstep bearing and then its respective views have been drawn

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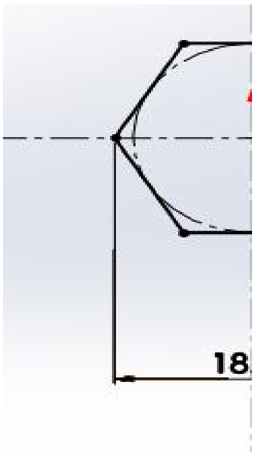
Experiment No. **Date:**

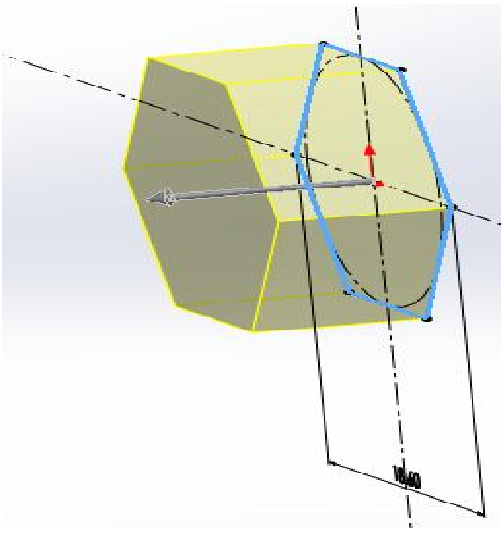
1. DETAILS AND MODELING OF INTERNAL AND EXTERNAL THREAD OF BOLT AND NUT USING SOLID WORKS(EXTRA)

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.

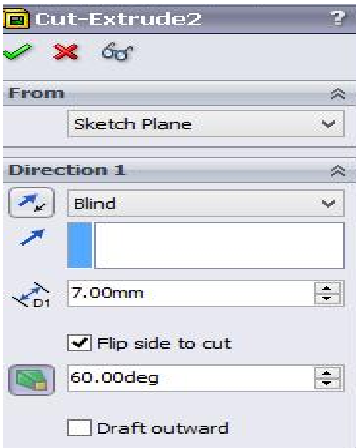
1. Procedure: Create a 2D sketch on Front Plane as shown in the figure.
2. (Right click the Front plane>insert sketch and draw the 2D sketch)
3. Note: All the 2D sketches drawn should be fully Defined and there should not be any under defined) and use (click Add Relation and Smart Dimensions)

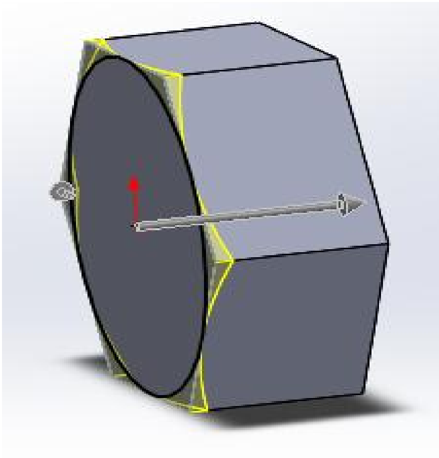




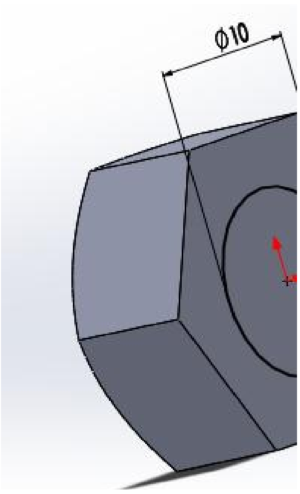
4. Create circle of 2D sketch of

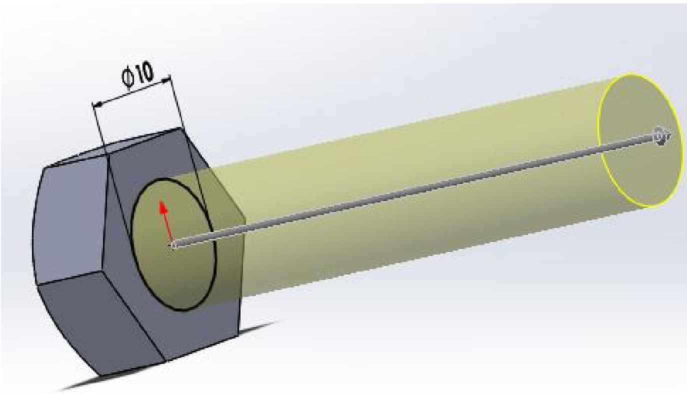
   Hexagonal width of 11.6 mm, on right plane and cut extrude to 7mm, (Select the face by (Enter Space bar> double click the Normal plane)





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)

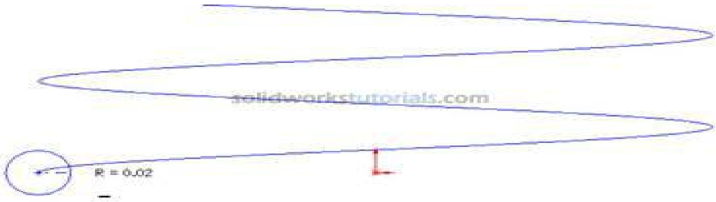
7.Create fillet and chamfer at corner of bolt at size of 1mm

8.Create external thread, Click Insert>Curve>Helix/Spiral

Press F to zoom fit, set Parameters Constant Pitch , Pitch suitable dimensions

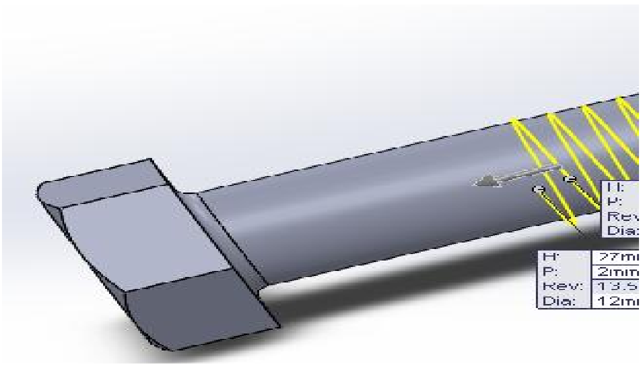
Revolutions 4 , Start angle 0.0deg.

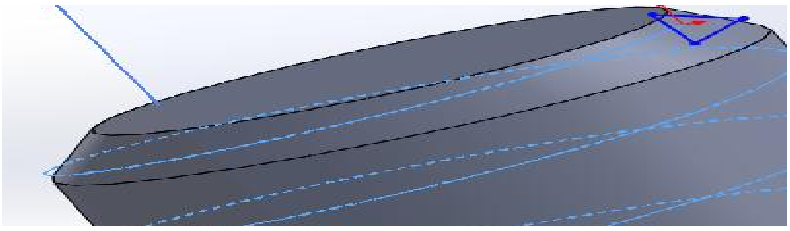
Click Sketch , click Circle . Sketch circle at start point, then click Smart dimension.



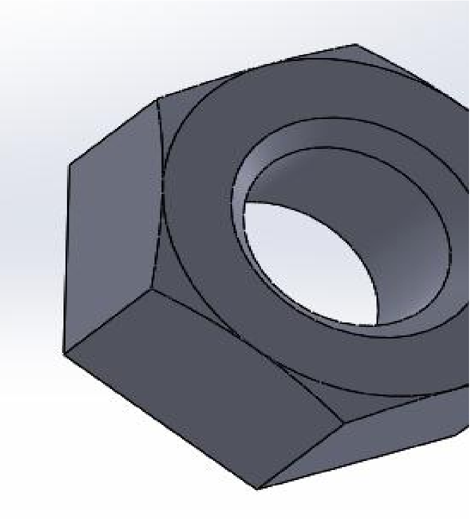
set circle diameter to 1 mm.

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.

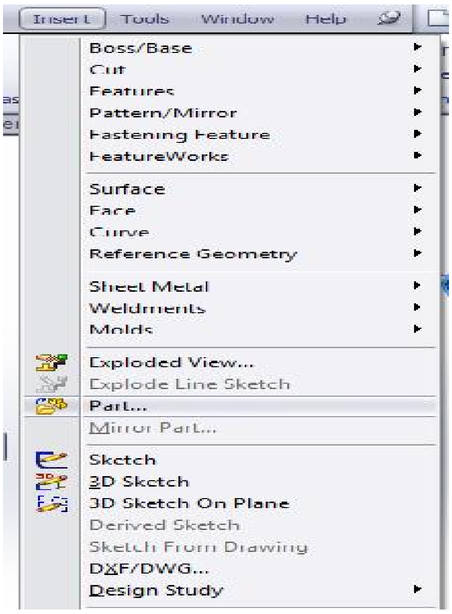




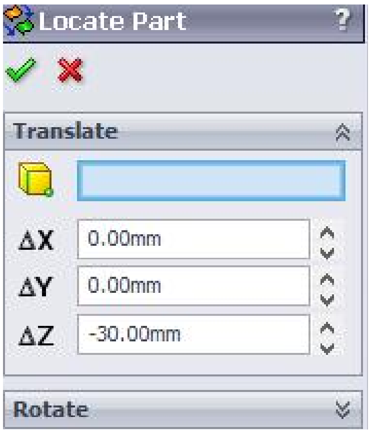
Similarly create Nut also as above said. Dimensions of nut as per bolt.

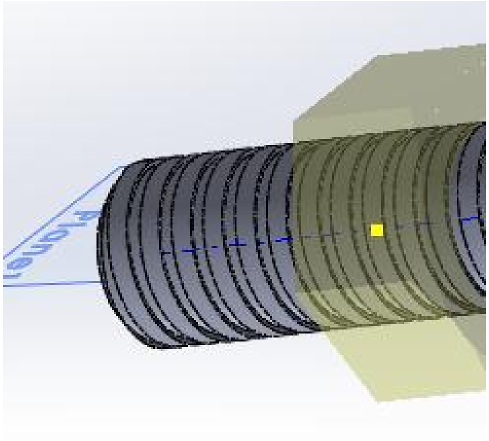


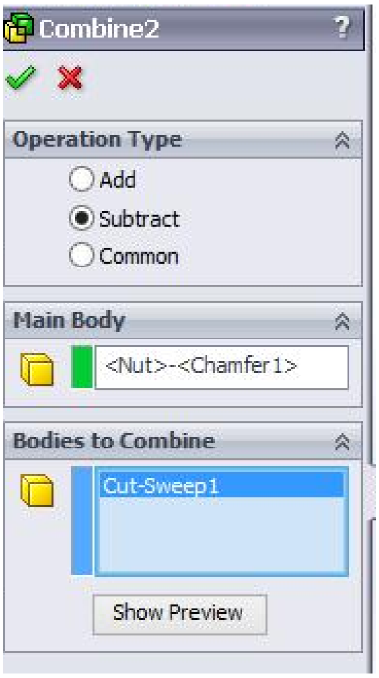
After create Nut part and save it.
And open bolt file and select
Select Insert> part> nut part>ok.



10. Locate part with center of bolt.





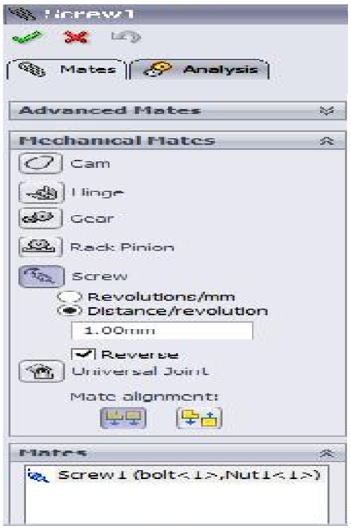


11.Insert>Feature>combine> Select Main body of nut and bolt >OK and save the file with different name.

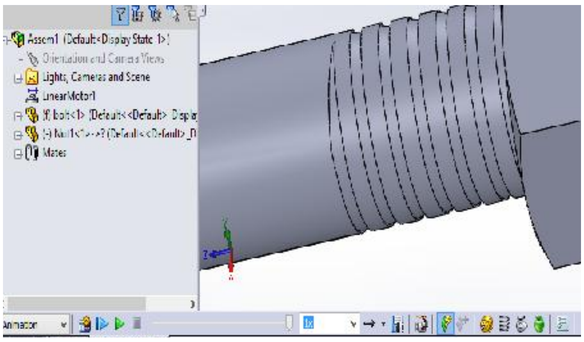
Assembly: Open new assembly file. Import bolt and nut file in assembly mode.

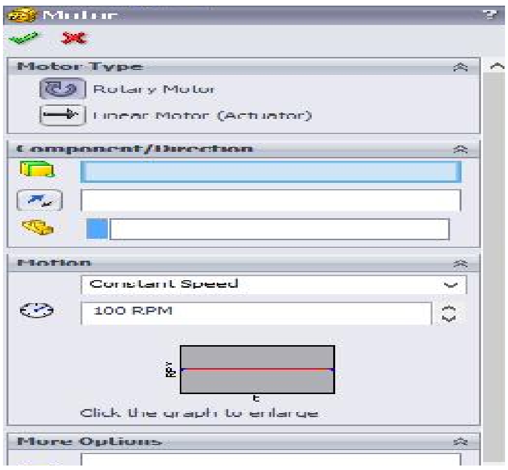
12. Mate the components using concentric Mate (select both bolt and nut thread faces).

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.

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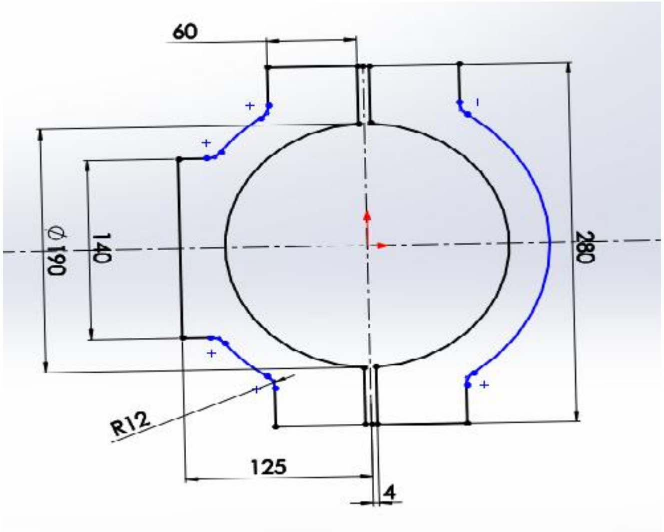
Experiment No. ☐☐ **Date:** ☐☐ ☐☐ ☐☐

2. Details and assembly of Eccentric using solidworks software(Extra)

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:



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

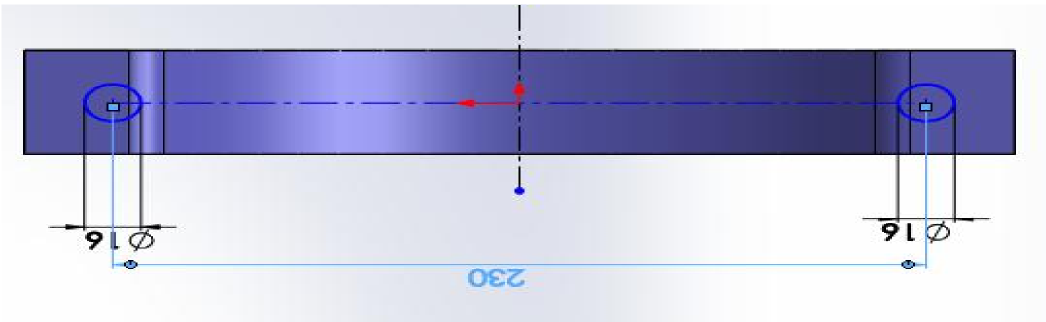
2.(Right click the Front plane>insert sketch and draw the 2D sketch).

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.

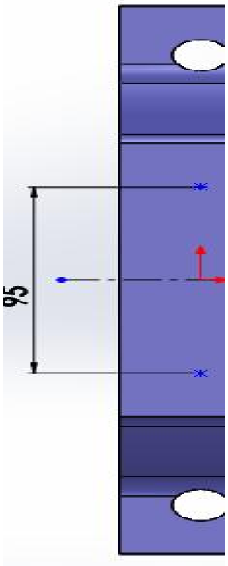
3. extrude to 45 mm (Select the face by (Enter Space bar> double click the Mid plane) and Draw the 2D sketch as given above

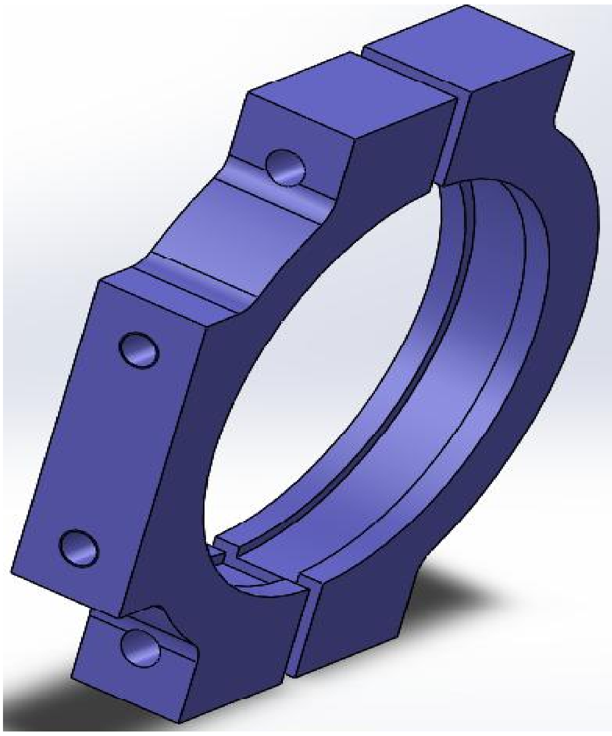
Extrude by (Insert>Boss/Base>Extrude)) ok.

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**

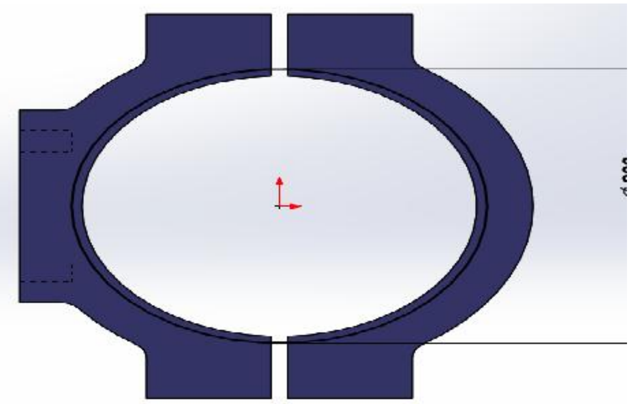




7. Create a 2D sketch on Front Plane as shown in the figure.

(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.

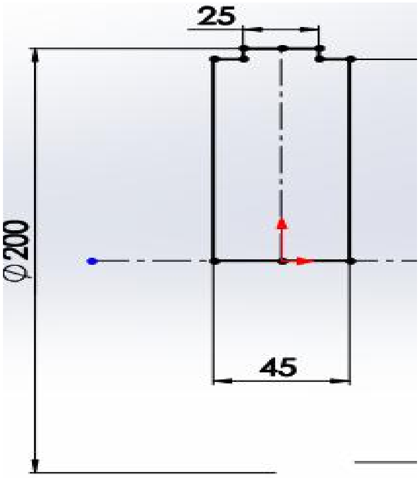


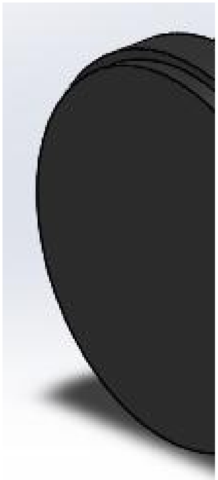
2.Sheave: Step.1 1.Create a 2D sketch on Front Plane as shown in the figure.

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

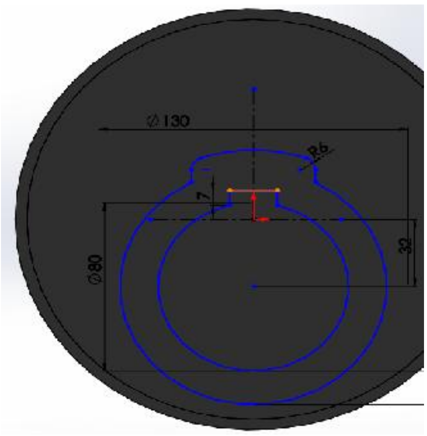
Note: All the 2D sketches drawn should be fully Defined and there should not be any under defined) and use (click Add Relation and Smart Dimensions.

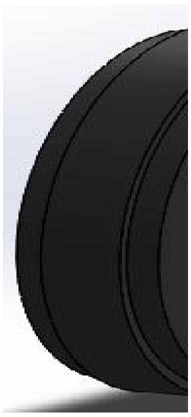
Revolve, the sketch to 360 degree on top sketched line, by (Insert> Boss/Base>Revolve) ok.



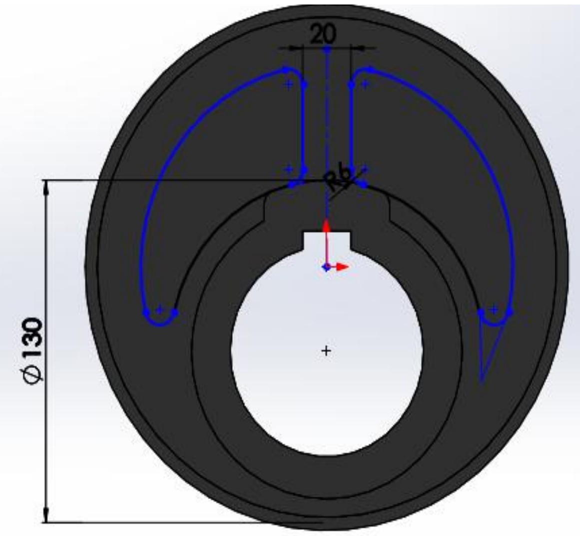


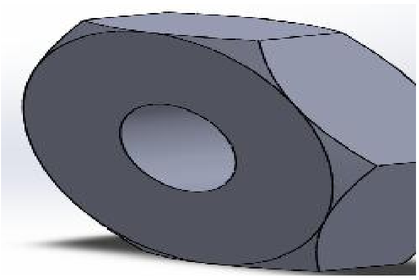
Create circle of 2D sketch as per the dimensions, on right plane and extrude to 20mm (Select the face by (Enter Space bar> double click the Normal plane) and Draw the 2D sketch as given above. Extrude by (Insert>Boss/Base>Extrude)) ok.





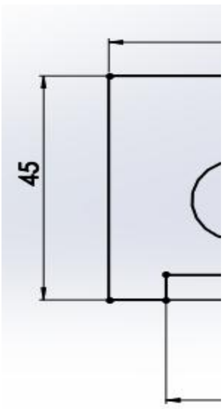
And select inner sketch use extrude cut through all.





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

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



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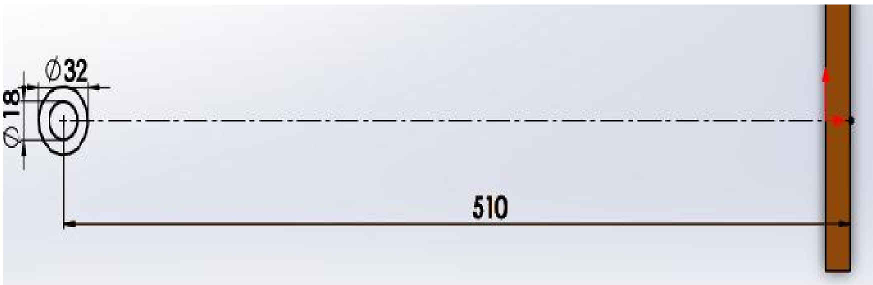
Experiment No.   **Date:**

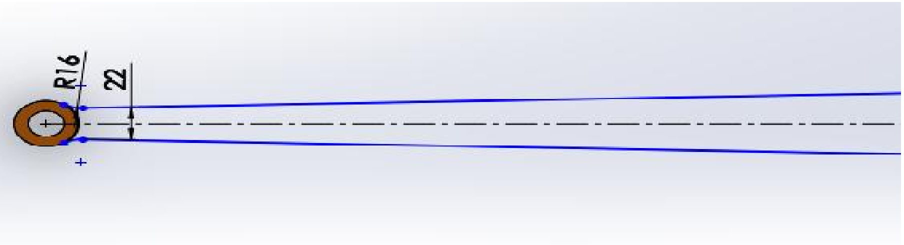
5. Rod:

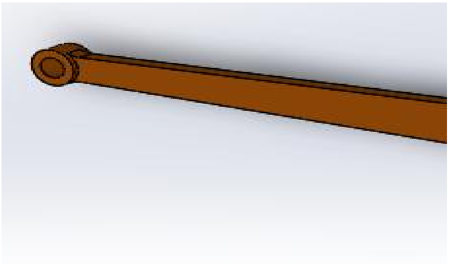
Use extrude option and select mid plane 30mm.

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.





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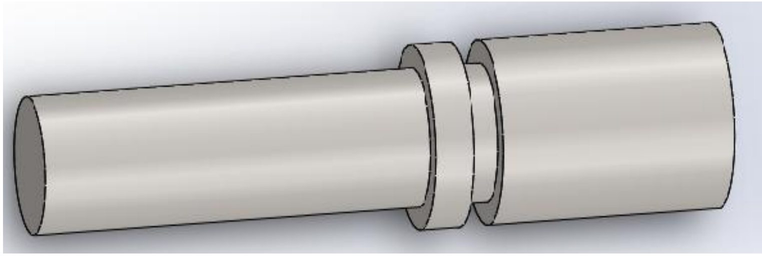
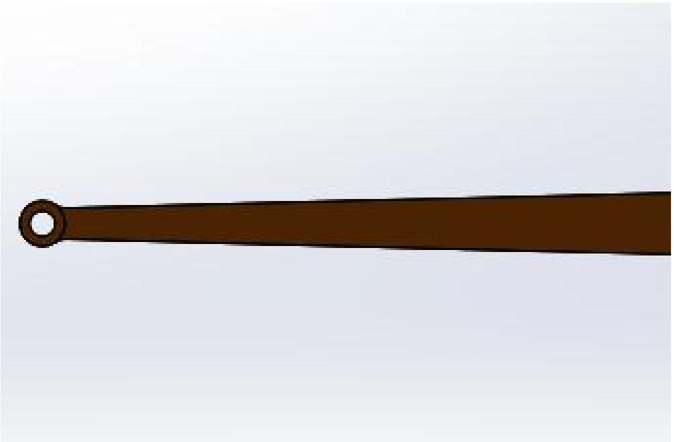
6.Bolt: Create bolt as per the dimensions:
7.Stud   as per the dimensions:

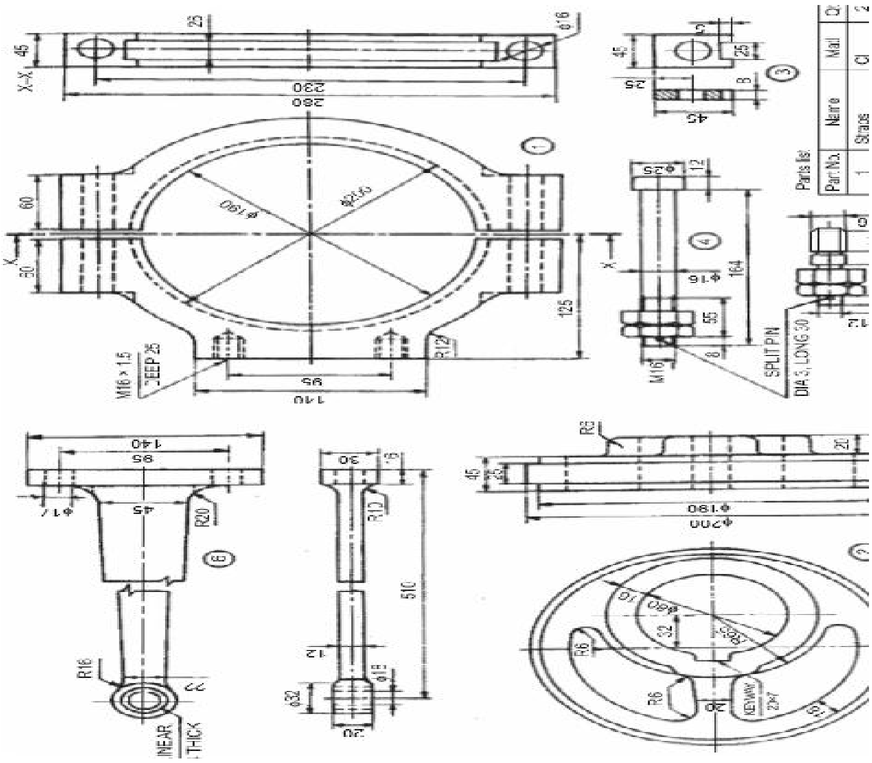


Assembly: Insert the components.



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.



Eccentric Part 2 - eccentric rod Solidwo

CAM

INTRODUCTION

WORD DETAILS:

Although the control will, in general, accept part programming words in any sequence, it is recommended that the following word order for each block is used.

N; G; X or U; Z or W; I; K; F; S; T;

O: PROGRAM NUMBER

The "O" followed by a 4 digit numeral value is used to assign a program number.

Example: O1002

N: SEQUENCE NUMBER

The N word may be omitted. When programmed, the sequence number following the N address is a four digit numerical value and is used to identify a complete block of information. Although ascending, descending, or duplicate numbering is allowed, it is best to program in ascending order in increments of 10. This allows for future editing and simplified sequence number search.

G: PREPARATORY COMMAND:

The two digit G command is programmed to set up the control to perform an automatic machine operation. A full list of G codes are given, one G word from each modal group and one non modal G word can be programmed on the same block.

Example:

Valid N 100 G00 G40 G41 G90 G95

*G40 & G41 are from the same group.

A retained G word (Modal) from one group remains active until another G word from the same group is programmed.

Experiment No. **Date:**

One-shot G word (Non-Modal) must be programmed in every block when required.

G-CODES LISTING FOR DENFORD FANUC LATHES:

Note: - NOT ALL G CODES APPLY TO EACH MACHINE.

Group 1	**G00**	**Positioning (Rapid Traverse)**
1	**G01**	**Liner Interpolation (Feed)**
1	**G02**	**Circular Interpolation CW**
1	**G03**	**Circular Interpolation CW**
0	**G04**	**Dwell**
0	**G10**	**Offset Value Setting By Program**
6	**G20**	**Inch Data Input**
6	**G21**	**Metric Data Input**
9	**G22**	**Stored Stroke Check On**
9	**G23**	**Stored Stroke Check Off**
0	**G27**	**Reference Point Return Check**
0	**G28**	**Reference Point Return**
0	**G29**	**Return from Reference Point**
0	**G30**	**Return to 2nd Reference Point**
0	**G31**	**Skip Function**
1	**G32**	**Thread Cutting**
1	**G34**	**Variable Lead Thread Cutting**
0	**G36**	**Automatic Tool Compensation X**
0	**G37**	**Automatic Tool Compensation Z**
7	**G40**	**Tool Nose Radius Compensation cancels**
7	**G41**	**Tool Nose Radius Compensation Left**
7	**G42**	**Tool Nose Radius Compensation Right**
0	**G50**	**Work Co-ord. Change/Max. Spindle Speed setting**
0	**G65**	**Macro call**
12	**G66**	**Macro Modal Call Cancel**

12	**G67**	**Macro Modal Call Cancel**
4	**G70**	**Finishing Cycle**
4	**G71**	**Stock Removal in Turning**
0	**G72**	**Stock Removal in Turning**
0	**G73**	**Pattern Repeating**
0	**G74**	**Peck Drilling in Z Axis**
0	**G75**	**Grooving in X Axis**
0	**G76**	**Thread Cutting Cycle**
1	**G90**	**Cutting Cycle A**
1	**G92**	**Thread Cutting Cycle**
1	**G94**	**Cutting Cycle B**
2	**G96**	**Constant surface Speed Control**
2	**G97**	**Constant Surface Speed Control Cancel**
11	**G98**	**Feed per Minute**
11	**G99**	**Feed per Revolution**

NOTES FOR G CODE LISTING:

Note 1:-

G Codes of 0 group represent those non modal and are effective to the designed block.

Note 2:-

G Codes of different groups can be commanded to the same block. If more than one G codes from the same group are commanded, the latter becomes effective.

AXIS DEFINITIONS:-

Z AXIS:-

The Z axis is along a line between the spindle and the tailstock, or the center line of rotation of the spindle. Minus (-) movements of the tool are left toward the head stock; positive (+) movements are right towards the tailstock.

X AXIS:-

The X axis is 90 degrees from the Z axis (perpendicular to the Z axis). Minus (-) movements of the tool are toward the center-line of rotation, and positive (+) movements are away from the center –line of rotation.

X: X AXIS COMMAND:-

The X word is programmed as a diameter which is used to command a change in position perpendicular to the spindle center-line.

U: X AXIS COMMAND:-

The U word is an incremental distance (diameter value) which is used to command a change in position perpendicular to the spindle center-line. The movement is the programmed value.

Z: Z AXIS COMMAND:-

The Z word is an absolute dimension which is used is used to command a change in position parallel to the spindle center-line.

W: Z AXIS COMMAND:-

The W word is an incremental distance which is used to command a change of position parallel to the spindle center-line.

Do not program X & U or Z & W in the same block. If an X axis command calls for no movement it may be omitted.

X, U or P: DWELL:-

The X word is used with G04 to command a dwell in seconds.

The P word is used with G04 to command a dwell in milliseconds.

I WORD:-

For arc programming (G02 or G03) , the K Value (with sign) is programmed to define the incremental distance parallel to the Z axis, between the start of the arc and the arc center.

K WORD:-

For arc programming (G02 or G03), the K value (with sign) is programmed to define the incremental distance parallel to the Z axis, between the start of the arc and the arc center.

The maximum arc for I & K programming is limited to the quadrant. If I or K is zero, it must be omitted.

F WORD:-

- **a)** In G99 mode the F word is used to command feed/rev.
- **b)** In G98 mode the F word is used to command feed/min.
- **c)** In G32 mode the F word specifies the lead (pitch) of the thread.

P WORD:-

- **a)** Used in automatic cycles to define the first block of a contour.
- **b)** Used with M98 to define a subroutine number.

Q WORD:-

Q words are used in automatic cycles to define the last block of a contour.

R WORD:-

For circular interpolation (G02 or G03) the R word defines the arc radius from the center of the tool nose radius (G40 active) - or the actual radius required (G41/ G42 active).

S WORD:-

- **a)** In the constant surface speed mode (G96) the four digit S word is used to command the required surface speed in either feet or meters per minute.
- **b)** In the direct R.P.M mode (G97), the four digit S word is used to command the spindle speeds incrementally, in R.P.M between the ranges available for the machine.
- **c)** Prior to entering constant surface speed mode (G96) the S word is used to specify a speed constraint, the maximum speed you wish the spindle to run at. To set this restraint the S word is programmed in conjunction with the G50 word.

T WORD:-

The T words are used in conjunction with “M06”. Those are used to call up the required tool on an automatic indexing turret machine, and to activate its tool offsets.

M WORD:-

An M word is used to initiate auxiliary functions particular to the machine. One M code can be programmed with in one program block together with other part program information.

Experiment No.   **Date:**

M- CODE LIST FOR DENFORD FANUC LATHES:-

All M Codes marked with an asterisk will be executed at the end of a block (i.e., after the axis movement).

*	M00	PROGRAM STOP
*	M01	OPTIONAL STOP
*	M02	PROGRAM RESET
	M03	SPINDLE FORWARD
	M04	SPINDLE REVERSE
*	M05	SPINDLE STOP
	M06	AUTO TOOL CHANGE
	M07	COOLANT "B" ON
	M08	COOLANT "A" ON
*	M09	COOLANT OFF
	M10	CHUCK OPEN
	M11	CHUCK CLOSE
	M13	SPINDLE FORWARD & COOLANT ON
	M14	SPINDLE REVERSE & COOLANT ON
	M15	PROGRAM INPUT USING."MIN P" (SPECIAL FUNCTION)
	M16	SPECIALTOOL CALL (TOOL CALL IGNORES TURRET)
	M19	SPINDLE ORIENTATE
	M20	SPINDLE INDEX A
	M21	SPINDLE INDEX 2A
	M22	SPINDLE INDEX 3A
	M23	SPINDLE INDEX 4A
	M25	QUILL EXTEND
	M26	QUILL RETRACT
	M29	SELECT "DNC" MODE

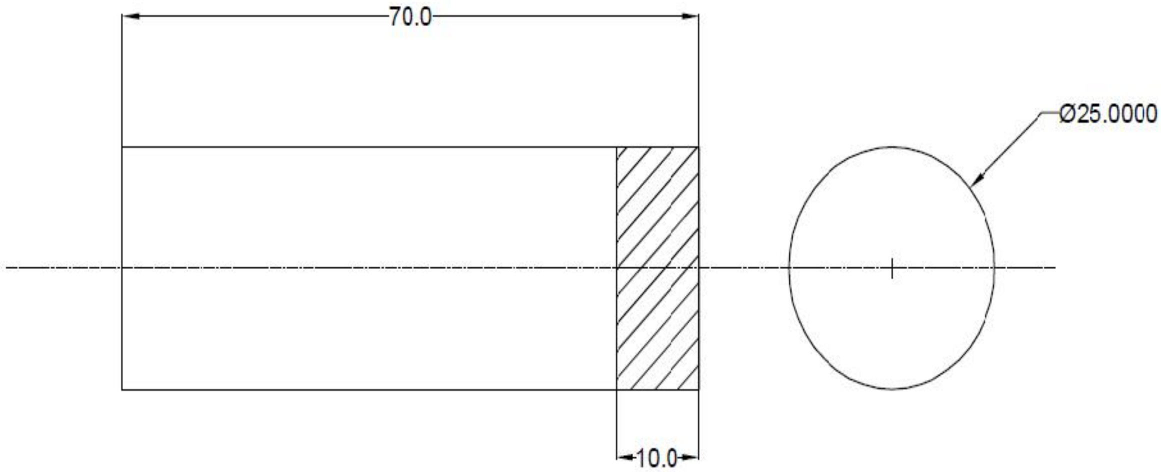
M30	PROGRAM RESET & REWIND
M31	INCREMENT PARTS COUNTER
M37	DOOR OPEN TO STOP
M38	DOOR OPEN
M39	DOOR CLOSE
M40	PARTS CATCHER EXTEND
M41	PARTS CATCHER RETRACT
M43	SWARF CONVEYOR FORWARD
M44	SWARF CONVEYOR REVERSE
M45	SWARF CONVEYOR STOP
M48	LOCK % FEED AND % SPEED AT 100%
M49	CANCEL M48 (DEFAILT)
M50	WAIT FOR AXIS IN POSITION SIGNAL (CANCELS CONTINUOUS PATH)
M51	CANCEL M50 (DEFAILT)
M52	PULL-OUT IN THREADING = 90 DEGRESS (DEFAILT)
M53	CANCEL M52
M54	DISABLE SPINDLE FLUCTUATION TESTING (DEFAILT)
M56	SELECT INTERNAL CHUCKING (FROM PLC EDITION "F")
M57	SELECT EXTERNAL CHUCKING (FROM PLC EDITION "F")
M62	AUX.1 ON
M63	AUX.2 ON
M64	AUX.1 OFF
M65	AUX.2 OFF
M98	SUB PROGRAM CALL
M99	SUB PROGRAM END

8.FACING CYCLE

```
[BILLET X25 Z70]
G21 G98;
G28 U0W0;
M06 T1 ;( FACING TOOL)
M03 S1200;
G00 X26 Z0;
G94 X0 Z-0.5 F50;
    Z-1.0
    Z-1.5
    Z-2.0
    Z-2.5
    Z-3.0
    Z-3.5
    Z-4.0
    Z-4.5
    Z-5.0
    Z-5.5
    Z-6.0
    Z-6.5
    Z-7.0
    Z-7.5
    Z-8.0
    Z-8.5
    Z-9.0
    Z-9.5
    Z-10.0
G28 U0W0;
M05;
M30;
```

SVR ENGINEERING COLLEGE

Experiment No. | | **Date:** | |



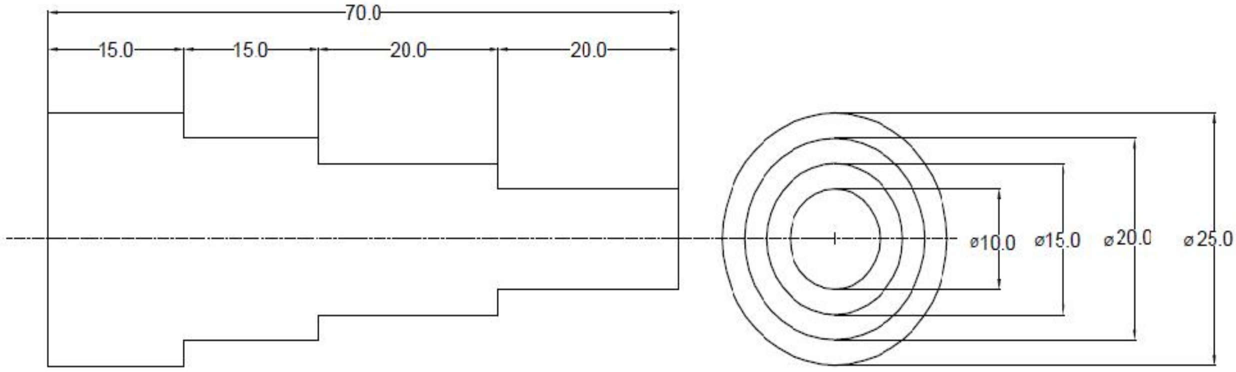
FACING CYCLE

ALL DIMENSIONS ARE IN MM

9.TURNING CYCLE

```
[BILLET X28 Z70]
G21 G98;
G28 U0W0;
M06 T1 ;( FACING TOOL)
M03 S1000;
G00 X25 Z1;
G94 X24 Z45 F50;
    X23
    X22
    X21
    X20
    X19 Z-40
    X18
    X17
    X16
    X15
    X14 Z-20
    X13
    X12
    X11
    X10

G28 U0W0;
M05;
M30;
```



TURNING CYCLE

ALL DIMENSIONS ARE IN MM

Experiment No. **Date:**

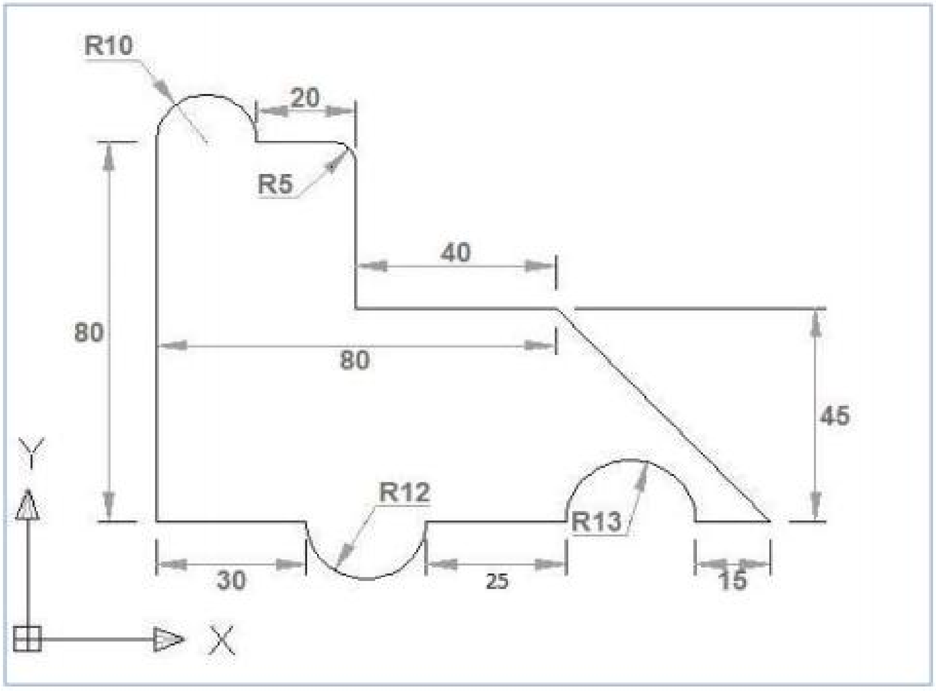
10.LINEAR AND CIRCULAR INTERPOLATION

AIM: To write a program to obtain linear and circular interpolation on the given work piece.

SOFTWARE REQUIRED: CNC XMILL Software with FANUC Language.

PROGRAM: G21 G94

```
G91 G28 Z0
G28 X0 Y0
M06 T06
M03 S1300
 G90 G00 X0 Y0 Z5
G90 G01 X0 Y0
 X30
G03 X54 R12
G01 X82
G02 X108 R13
G01 X123
X80 Y45
X40
Y75
G03 X35 Y80 R5
G01 X20
G03 X0 Y80 R10
G01 Y0
M30
```



LINEAR AND CIRCULAR INTERPOLATION

11.ENGRAVE

AIM: To write a program to engrave the letters “SVREC” on the given work piece.

SOFTWARE REQUIRED: CNC XMILL Software with FANUC Language.

PROGRAM:

```
G21 G94
G91 G28 Z0
G28 X0 Y0
M06 T06
M03 S1300
G90 G00 X0 Y0 Z5
[S]
G00 X2 Y30
 G01 Z-1 F60
G01 X10 Y30
G03 X15 Y35 R5
G01 X15 Y 37.5
G03 X10 Y42.5 R5
G01 X07 Y42.5
G02 X2 Y47.5 R5
G01 X2 Y50
G02 X7 Y55 R5
G01 X15 Y55
G00 Z2
[V]
G00 X20 Y55
G01 Z-1 F60
G01 X27.5 Y30
G01 X33 Y55
```

Experiment No. **Date:**

```
G00 Z2
[E]
G00 X69 Y55
G01 Z-1F60
G01 X56 Y55
G01 X56 Y42.5
 G01 X69 Y42.5
G01 X56 Y42.5
G01 X56 Y30
G01 X69 Y30
G00 Z2
G91 G28 Z0
 G28 X0Y0
M05
M30
```