I

Name _____

Roll No. _____ Year 20____20____

Exam Seat No. _____

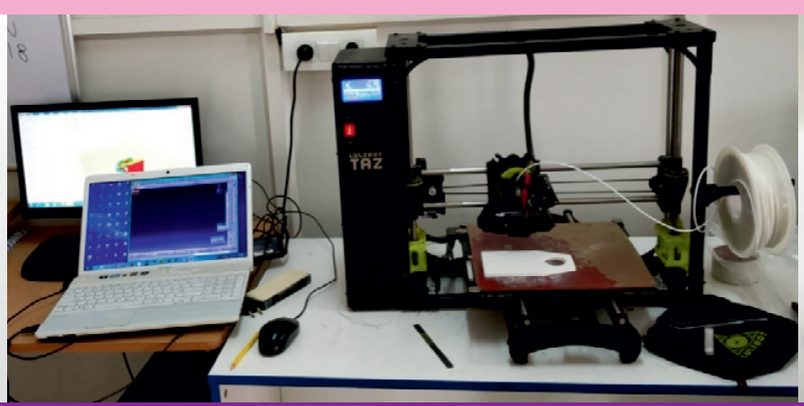
MECHANICAL GROUP | SEMESTER - V | DIPLOMA IN ENGINEERING AND TECHNOLOGY

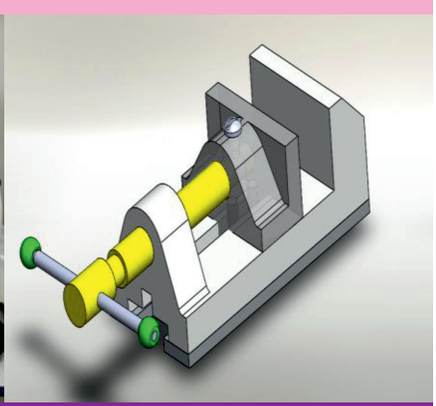
A LABORATORY MANUAL

FOR

SOLID MODELING AND ADDITIVE MANUFACTURING

(22053)





MAHARASHTRA STATE BOARD OF TECHNICAL EDUCATION, MUMBAI

(Autonomous) (ISO 9001 : 2015) (ISO / IEC 27001 : 2013)

VISION

To ensure that the Diploma level Technical Education constantly matches the latest requirements of technology and industry and includes the all-round personal development of students including social concerns and to become globally competitive, technology led organization.

MISSION

To provide high quality technical and managerial manpower, information and consultancy services to the industry and community to enable the industry and community to face the changing technological and environmental challenges.

QUALITY POLICY

We, at MSBTE are committed to offer the best in class academic services to the students and institutes to enhance the delight of industry and society. This will be achieved through continual improvement in management practices adopted in the process of curriculum design, development, implementation, evaluation and monitoring system along with adequate faculty development programmes.

CORE VALUES

MSBTE believes in the followings:

- Education industry produces live products.
- Market requirements do not wait for curriculum changes.
- Question paper is the reflector of academic standards of educational organization.
- Well designed curriculum needs effective implementation too.
- Competency based curriculum is the backbone of need based program.
- Technical skills do need support of life skills.
- Best teachers are the national assets.
- Effective teaching learning process is impossible without learning resources.

A Practical Manual

for

Solid Modeling and Additive Manufacturing

(22053)

Semester– (V)

(ME/PS)

Maharashtra State
Board of Technical Education, Mumbai
(Autonomous) (ISO:9001:2015) (ISO/IEC 27001:2013)

Maharashtra State Board of Technical Education,
(Autonomous) (ISO:9001:2015) (ISO/IEC 27001 : 2013)
4th Floor, Government Polytechnic Building, 49, Kherwadi,
Bandra (East), Mumbai - 400051.
(Printed on May,2019)

Maharashtra State
Board of Technical Education

Certificate

This is to certify that Mr. / Ms …….
Roll No…….of Fifth Semester of Diploma in
…….of Institute
……..
(Code……..) has completed the term work satisfactorily in course **Solid Modeling and Additive Manufacturing(22053)** for the academic year 20…….to 20…..... as prescribed in the curriculum.

Place ……. Enrollment No……

Date:…..... Exam Seat No. …….....

Course Teacher **Head of the Department** **Principal**

Seal of the Institute

Preface

The primary focus of any engineering laboratory/ field work in the technical education system is to develop the much needed industry relevant competencies and skills. With this in view, MSBTE embarked on this innovative 'I' Scheme curricula for engineering diploma programmes with outcome-base education as the focus and accordingly, relatively large amount of time is allotted for the practical work. This displays the great importance of laboratory work making each teacher; instructor and student to realize that every minute of the laboratory time need to be effectively utilized to develop these outcomes, rather than doing other mundane activities. Therefore, for the successful implementation of this outcome-based curriculum, every practical has been designed to serve as a ***'vehicle'*** to develop this industry identified competency in every student. The practical skills are difficult to develop through 'chalk and duster' activity in the classroom situation. Accordingly, the 'I' scheme laboratory manual development team designed the practical to ***focus*** on the ***outcomes***, rather than the traditional age old practice of conducting practical to 'verify the theory' (which may become a byproduct along the way).

This laboratory manual is designed to help all stakeholders, especially the students, teachers and instructors to develop in the student the pre-determined outcomes. It is expected from each student that at least a day in advance, they have to thoroughly read through the concerned practical procedure that they will do the next day and understand the minimum theoretical background associated with the practical. Every practical in this manual begins by identifying the competency, industry relevant skills, course outcomes and practical outcomes which serve as a key focal point for doing the practical. The students will then become aware about the skills they will achieve through procedure shown there and necessary precautions to be taken, which will help them to apply in solving real-world problems in their professional life.

This manual also provides guidelines to teachers and instructors to effectively facilitate student-centered lab activities through each practical exercise by arranging and managing necessary resources in order that the students follow the procedures and precautions systematically ensuring the achievement of outcomes in the students.

Mechanical, Plastic, Automobile and allied Industries need to build computer based models of desired product to perform different analyses before sending them for manufacturing so as to avoid wastage of resources. These models are being created using computer aided design software through 'solid modeling module' of the software. The same solid model can be send to rapid prototype machines and 3D printers for direct additive manufacturing also. This course will enable the students to inculcate solid modeling and additive manufacturing concepts and methodology to solve engineering problems

Although all care has been taken to check for mistakes in this laboratory manual, yet it is impossible to claim perfection especially as this is the first edition. Any such errors and suggestions for improvement can be brought to our notice and are highly welcome.

Programme Outcomes (POs) to be achieved through Practical of this Course:

PO 1. **Basic knowledge:** Apply knowledge of basic mathematics, sciences and basic engineering to solve the broad-based mechanical engineering problems.

PO 2. **Discipline knowledge:** Apply mechanical engineering knowledge to solve broad-basedmechanicalengineering related problems.

PO 3. **Experiments and practice:** Plan to perform experiments and practices to use the results to solve broad-basedmechanicalengineering problems.

PO 4. **Engineering tools:** Apply relevant mechanical technologies and tools with an understanding of the limitations

PO 5. **The engineer and society:** Assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to practice in field of mechanical engineering.

PO 6. **Environment and sustainability:** Apply mechanical engineering solutions also for sustainable development practices in societal and environmental contexts.

PO 7. **Ethics:** Apply ethical principles for commitment to professional ethics, responsibilities and norms of the practice also in the field of mechanical engineering.

PO 8. **Individual and team work:** Function effectively as a leader and team member in diverse/ multidisciplinary teams.

PO 9. **Communication:** Communicate effectively in oral and written form.

PO 10. **Life-long learning:** Engage in independent and life-long learning activities in the context of technological changes also in the mechanical engineering and allied industry.

Program Specific Outcomes (PSOs)

PSO 1: Modern Software Usage: Use latest mechanical related software for simple design, drafting, manufacturing, maintenance and documentation of mechanical components and processes.

PSO 2: Maintenance and selection of machines, equipment, instruments: Maintain and select appropriate machine, equipment and instrument in field of Mechanical Engineering.

PSO 3: Manage Mechanical Process: Manage the mechanical process by selection and scheduling right type of machinery, equipment, substrates, quality control techniques, operational parameters and software for a particular mechanical process or job for economy of operations.

List of Industry Relevant Skills

The following industry relevant skills of the competency Use **Solid Modeling and Additive Manufacturing** are expected to be developed in you by undertaking the practical of this laboratory manual.

a. Prepare 2D Drawings using sketcher workbench of any parametric CAD software.

b. Generate 3D Solid models from 2D sketches using Part workbench of any parametric CAD software.

c. Prepare assemblies of part models using Assembly workbench of any parametric CAD software.

d. Generate orthographic views of 3D solid models/assemblies using drafting workbench of any parametric CAD software.

e. Generate production drawings for given part models/assemblies.

f. Print components using 3D Printer/Rapid prototyping machine.

Practical- Course Outcome matrix

Course Outcomes (COs)

a. Prepare 2D Drawings using sketcher workbench of any parametric CAD software.
b. Generate 3D Solid models from 2D sketches using Part workbench of any parametric CAD software.
c. Prepare assemblies of part models using Assembly workbench of any parametric CAD software.
d. Generate orthographic views of 3D solid models/assemblies using drafting workbench of any parametric CAD software.
e. Generate production drawings for given part models/assemblies.
f. Print components using 3D Printer/Rapid prototyping machine

Sr. No.	Practical Outcome	CO a.	CO b.	CO c.	CO d.	CO e.	CO f.
1.	Prepare drawing template consisting of name plate boundary lines and projection symbol.	√	-	-	-	-	-
2.	Draw and print two simple 2D geometries using sketcher commands	√	-	-	-	-	-
3.	Draw and print two complex 2D geometries using sketcher commands	√	-	-	-	-	-
4.	Draw and print the given two simple 3-D drawings using 3D modeling commands	√	√	-	-	-	-
5.	Develop solid models of individual components of Bench vice / Drill Jig / Screw Jack / Tool Post / anyone assembly consisting of at least five parts. (Problem 1)	√	√	√	-	-	-
6.	Develop solid models of individual components of Bench vice / Drill Jig / Screw Jack / Tool Post / anyone assembly consisting of at least five parts. (Problem 1 continued)	√	√	√	-	-	-
7.	Develop solid models of individual components of Bench vice / Drill Jig / Screw Jack / Tool Post / anyone assembly consisting of at least five parts. (Problem 1 continued)	√	√	√	√	-	-
8.	Develop solid models of individual components of Bench vice / Drill Jig / Screw Jack / Tool Post / anyone assembly consisting of at least five parts. (Problem 1 continued)	√	√	√	√	-	-

9.	Assemble and print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials. (Problem 2)	√	√	√	√	√	
10.	Assemble and print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials. (Problem 2 continued)	√	√	√	√	√	**-**
11.	Assemble and print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials. (Problem 2 continued)	√	√	√	√	√	-
12.	Draw and print the production drawing of all individual components part models of assembly developed in PrO 5 to 8. (Problem 3)	√	√	√	√	√	√
13.	Draw and print the production drawing of all individual components part models of assembly developed in PrO 5 to 8. (Problem 3 continued)	√	√	√	√	√	√
14.	Draw and print the production drawing of all individual components part models of assembly developed in PrO 5 to 8. (Problem 3 continued)	√	√	√	√	√	√
15.	Print one simple component using 3D printer / Rapid prototyping machine.	-	-	-	-	-	√
16.	Print one complex component using 3D printer / Rapid prototyping machine. (Problem 1)	-	-	-	-	-	√

Guidelines to Teachers

1. ***Teacher need to ensure that a dated log book*** for the whole semester, apart from the laboratory manual is maintained by every student which s/he has to ***submit for assessment to the teacher*** in the next practical session.
2. There will be two sheets of blank pages after every practical for the student to report other matters(if any), which is not mentioned in the printed practicals.
3. For difficult practicals if required, teacher could provide the demonstration of the practical emphasizing of the skills which the student should achieve.
4. Teachers should give opportunity to students for hands-on after the demonstration.
5. Assess the skill achievement of the students and COs of each unit.
6. One or two questions ought to be added in each practical for different batches. For this teachers can maintain various practical related question banks for each course.
7. If some repetitive information like data sheet, use of software tools etc. has to be provided for effective attainment of practical outcomes, they can be incorporated in Appendix.
8. For effective implementation and attainment of practical outcomes, teacher ought to ensure that in the beginning itself of each practical, students must read through the complete write-up of that practical sheet.
9. During practical, ensure that each student gets chance and takes active part in taking observations/ readings and performing practical.
10. Teacher ought to assess the performance of students continuously according to the MSBTE guidelines

Instructions for Students

1. For incidental writing on the day of each practical session every student should maintain a ***dated log book*** for the whole semester, apart from this laboratory manual which s/he has to ***submit for assessment to the teacher*** in the next practical session.
2. For effective implementation and attainment of practical outcomes, in the beginning itself of each practical, students need to read through the complete write-up including the practical related questions and assessment scheme of that practical sheet.
3. Student ought to refer the data books, IS codes, Safety norms, Technical Manuals, etc.
4. Student should not hesitate to ask any difficulties they face during the conduct of practical.

Content Page

List of Practical and Progressive Assessment Sheet

10	Assemble and print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials. (Problem 2 continued)	168					
11	Assemble and print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials. (Problem 2 continued)	190					
12	Draw and print the production drawing of all individual components part models of assembly developed in PrO 5 to 8. (Problem 3)	203					
13	Draw and print the production drawing of all individual components part models of assembly developed in PrO 5 to 8. (Problem 3 continued)	220					
14	Draw and print the production drawing of all individual components part models of assembly developed in PrO 5 to 8. (Problem 3 continued)	226					
15	Print one simple component using 3D printer / Rapid prototyping machine.	232					
16	Print one complex component using 3D printer / Rapid prototyping machine. (Problem 1)	249					
Total							

Note: To be transferred to Proforma of CIAAN-2017.

A suggestive list of PrOs is given in the above table. More such PrOs can be added to attain the COs and competency. A judicial mix of minimum 12 or more practical need to be performed, out of which, the practicals marked as '*' are compulsory, so that the student reaches the 'Precision Level' of Dave's 'Psychomotor Domain Taxonomy' as generally required by the industry.

Practical No.1: Prepare drawing template consisting of name plate boundary lines and projection symbol.

I. Practical Significance

The main purpose of engineering drawing is to communicate to other engineers, machinists etc. by placing various views on a drawing sheet of different sizes. In additions to these standard views we also need to show border, title block, tables, and various special notes. To show additional details, either we use standard template formats available in software or we can modify these standard templates as per our requirement or we can create our custom based new template format. In these practical, we are going to create custom based new template format and extension of format file is *.frm.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills for the industry identified competency ***'Draw Title Block, border, projection symbols for engineering drawing.***

IV Relevant Course Outcome(s)

- Create our custom based new template format as per our requirement.

V Practical Outcome

- Create custom based new template as per our requirement and use it to all drawings prepared in this subject.

VI Relative Affective Domain

- Working in team work.
- Practice good housekeeping.
- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Basic knowledge of computer handling.
- Basic knowledge of template content.

- Knowledge of various sheet sizes.
- Knowledge of solid modeling and format environment commands.

VIII Experimental setup

NIL.

IX Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3	Software	Any parametric solid modeling software.	As per batch size
4	Plotter.	Plotter A_2 OR A_3Size.	1

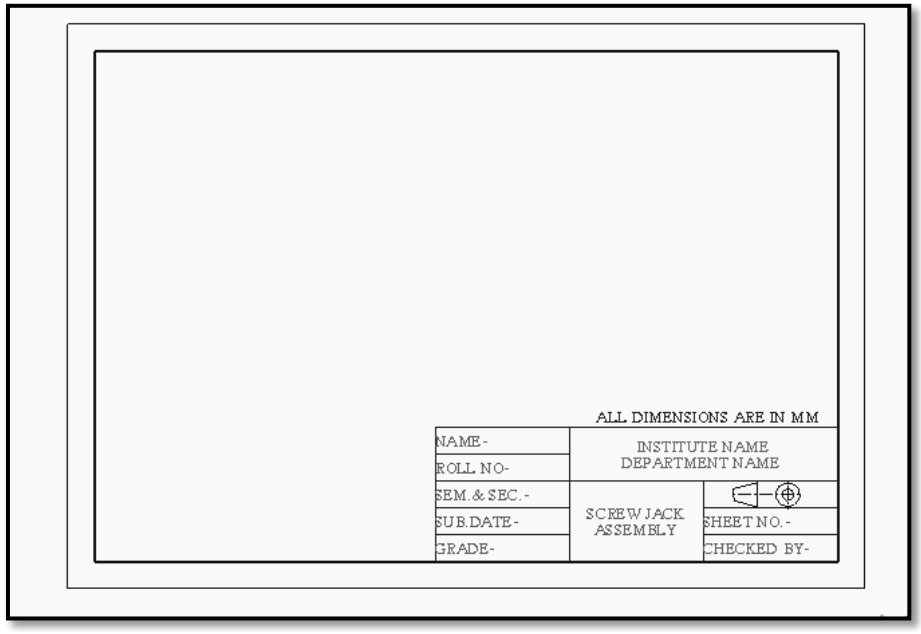
X Precautions to be Followed

.....
.....
.....
.....
.....
.....
.....
.....
.....
.....

XI Procedure

The first things you need to do if you want own customized format is to decide what it is going to look like. The best way to do that is to draw one out roughly on the relevant sized sheet of paper. As an example we are going to create an A4 sized landscape custom based template format (Here Solid Modeling CAD software is used).This is the guide to help you. You now have to determine how each element will be made. Considering following specifications of sample template for exercise as shown in Figure.

Problem Definition: Paper Size- A4 (297 X210 mm)Name Plate Size -150 X50 mm.
Border-277X190mm



ALL DIMENSIONS ARE IN MM		
NAME-	INSTITUTE NAME DEPARTMENT NAME	
ROLL NO-		
SEM.& SEC.-	SCREW JACK ASSEMBLY	
SUB.DATE-		SHEET NO.-
GRADE-		CHECKED BY-

Sr. No.	Particular	Color	Width	Height	Thick	Font/Alignment in cell
1	Border(Inside offset by 10 mm	Black	1	--	--	--
2	Table Lines	Brown	0.5	--	--	--
3	NAME,ROLL NO,SEM.& SEC.,SUB DATE,GRADE,SHEET NO.CECKED BY	Red	--	4	1	CG Times. Horizontally left. Vertically middle
4	SCREW JACK ASSEMBLY	Green	--	4	1	CG Times. Horizontally center. Vertically middle
5	INSTITUTE NAME DEPARTMENT NAME	Light Blue	--	4	1	CG Times. Horizontally center. Vertically middle
6	First Angle Symbol Ød = 9	Black	0.5	--	--	Line font-SOLID FONT,CTRL FONT
7	Note-ALL DIMENSIONS ARE IN MM	Dark Blue	--	4	1	CG Times

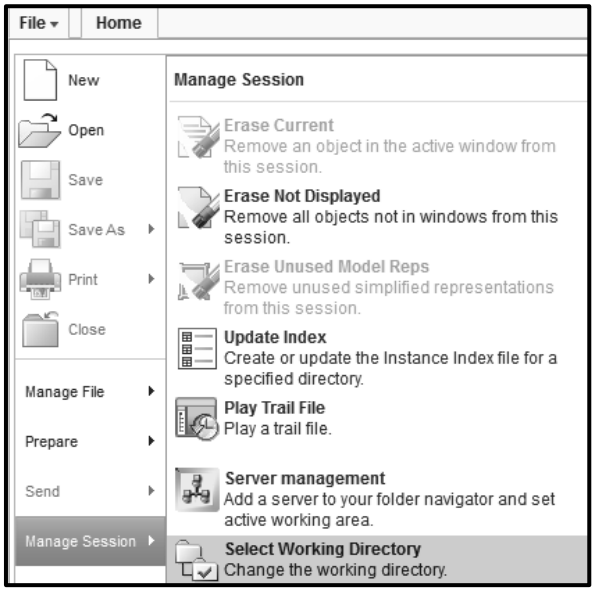
Following steps are required to create an A4 sized landscape custom based template format. (Here you can use any CAD software)

A. Setting the Environment:

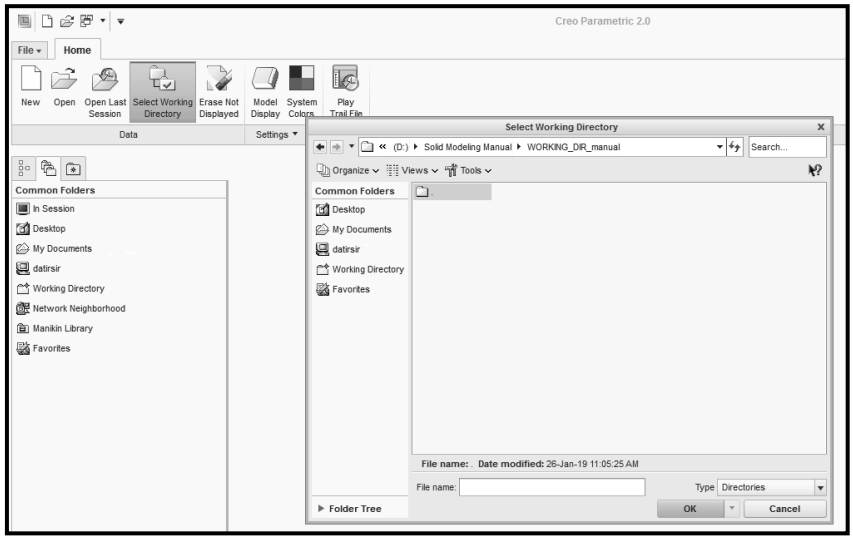
1. Click on icon on the desktop. The software will be launched, and the first window will be displayed as shown in Figure.

B. Set working directory:

To store all the created work content (files) in the current session in a specified folder; initially we have to set the directory. You can set any existing folder as the working directory. To do this, **click File>Manage Session>SelectWorking Directory**, select the particular folder.



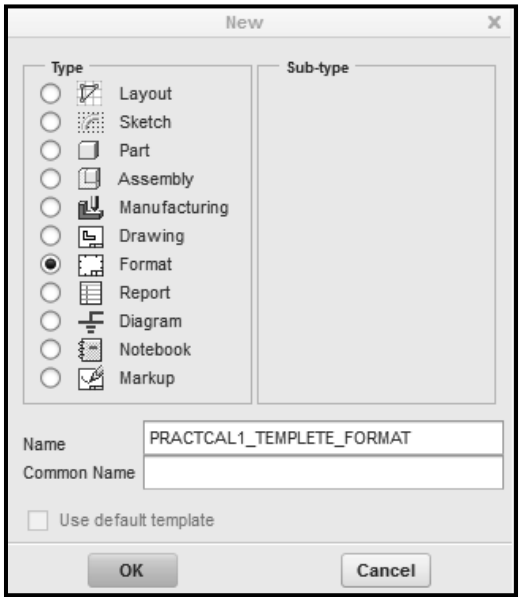
After setting specified directory the screen will appear as shown in Figure.



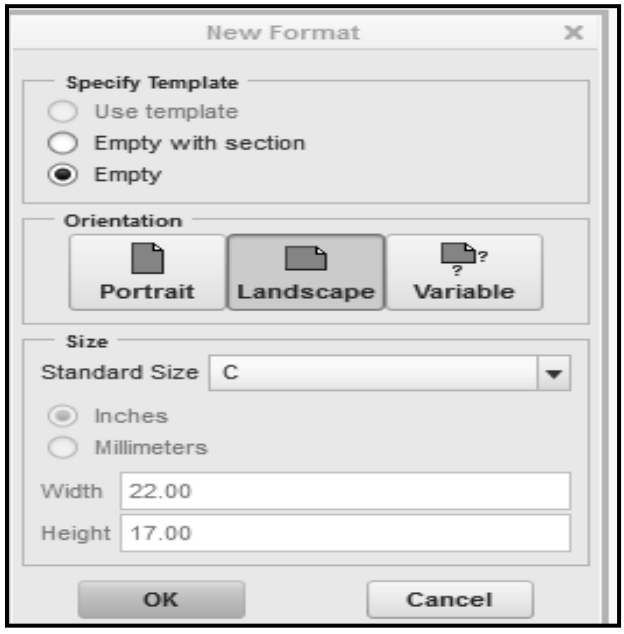
Click **OK as** shown in Figure.

C. To invoke format environment:

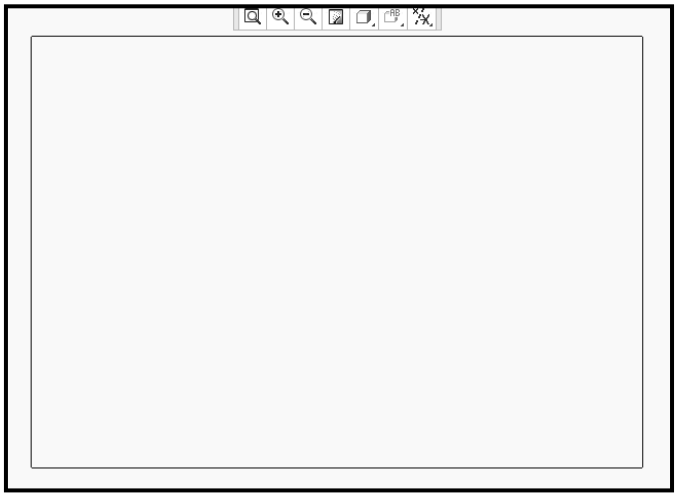
1. Select **New** from the **File** toolbar. The New dialog box is display.
2. Select the **format** radio button.
3. Type name as **PRACTICAL1_TEMPLETE_FORMAT**.
4. Choose the **OK** button.



When you invoke the **Format** mode, the new dialog box will display shown in Figure.



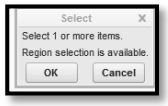
1. Select the **Empty** radio button. Select **landscape** orientation.
2. Specify **A4** (297X210 mm) size sheet by drop down menu.
3. Click on **OK**.The A4 size sheet will appear as shown in Figure.



D. To draw border 10 mm inside the sheet:

1. Select **Sketch** from ribbon menu to draw border to the sheet.
2. Select 

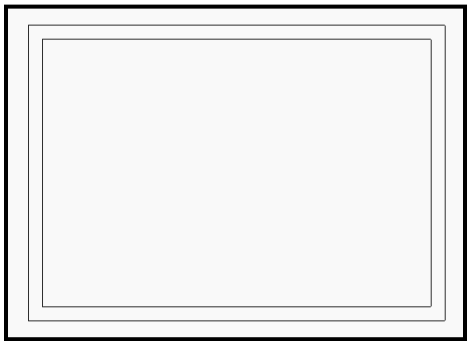
Offset Edge command from sketching tool bar.
3. Select 

Ent. Chain from menu manager.
4. Select the A4 size sheet by pressing Ctrl button of key board.
5. Click **OK** 

from‘select’ dialog box.

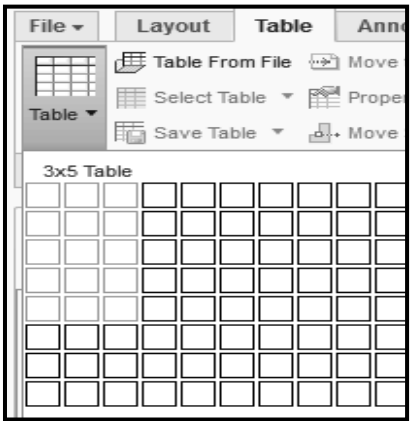
6. Enter off set distance -10 and accept the value.Exit from commands.

Figure show the A4 size sheet with 10 mm border inside the sheet.

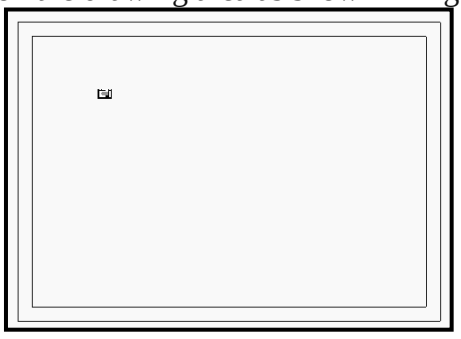


E. Toinsert the table.

1. Choose **Table** from ribbon menu.
2. Select the table by specifying 3 columns and 5 rows as shown in Figure.
3. Click curser inside the border of A4 size sheet.



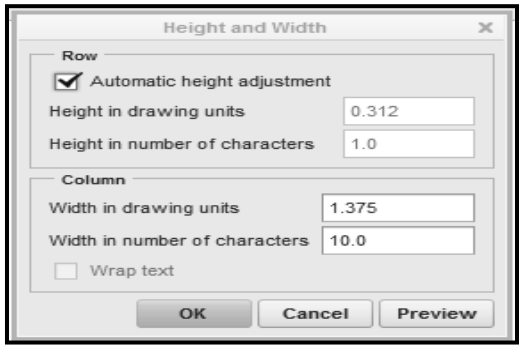
The table will display on the drawing area as shown in Figure.



F. To convert the table to required height and width:

1. Highlight the table by window selection method.
2. Click to 

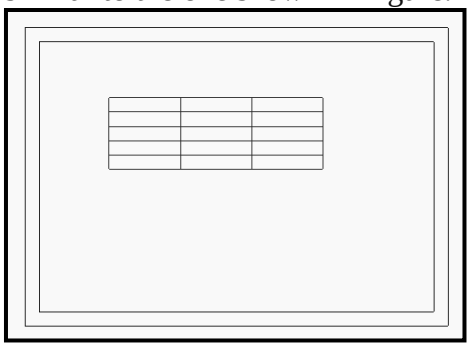
Height and width tool option .The new Height and Width definition dialog box will appear on the screen as shown in figure.



In the same dialog box-

1. Uncheck the '**Automatic height adjustment'** option.
2. Mention the height 10 and width 50 in the same dialog box option.
3. Click **OK,** Click the cursor in drawing area.

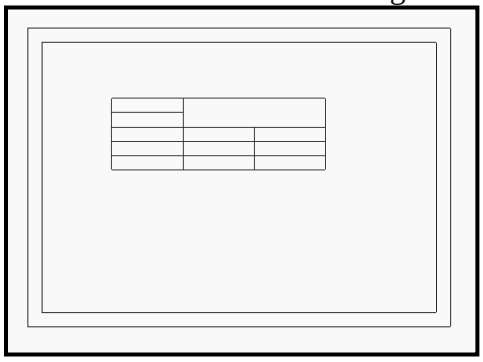
The table appearance is similar to the one shown in Figure.



G. To merge the required cell:

1. Highlight the cells, those you want to merge by pressing Ctrl button of key board.
2. Click on **Merge Cells** options from menu bar.
3. Click the cursor in drawing area.

The table appearance is similar to the one shown in Figure.

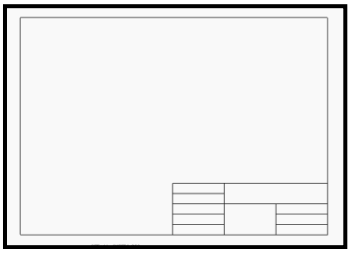


Repeat the same procedure to merge remaining cell as per sample title block given to you.

H. To move entire table to lower right corner of the border :

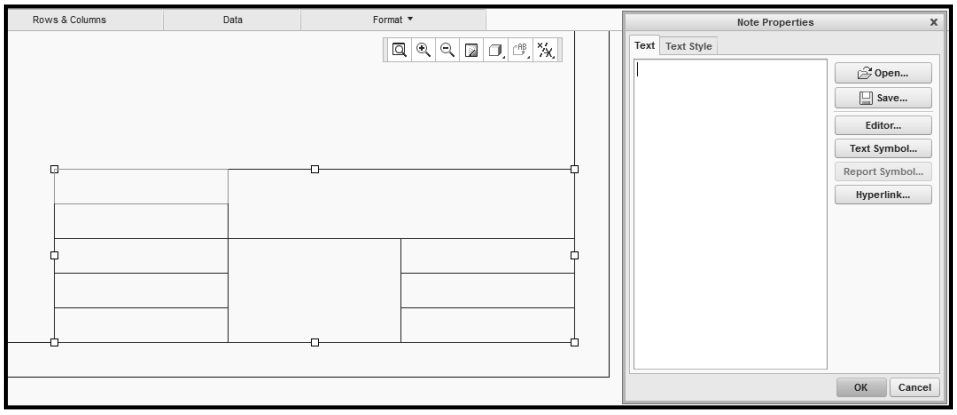
1. Click on **Table** from menu bar.
2. Highlight table by clicking on it.
3. Pick up the table Grib and locate it at lower left corner of border.

The table will look like as shown in Figure.

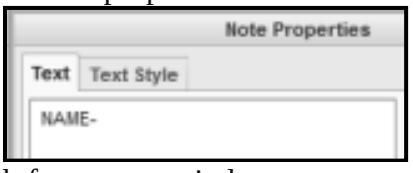


I. To add the text as per the given sample title block:

1. Highlight a cell by double clicking.
2. New 'Note properties' definition dialog box will open.

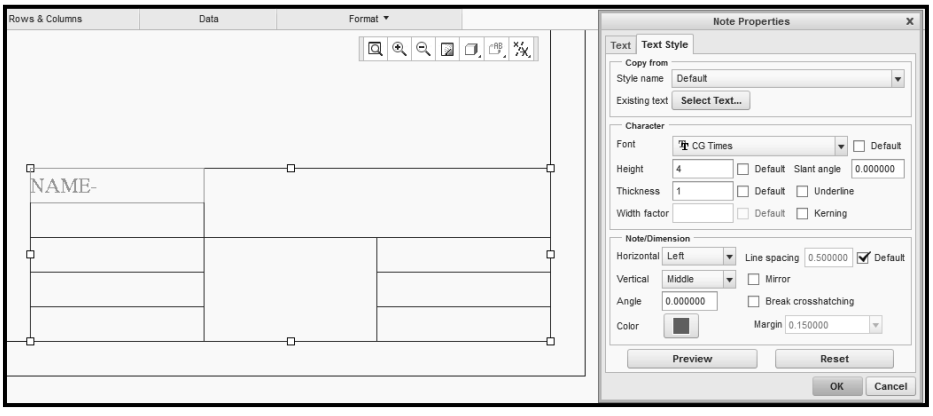


3. Type the text 'NAME-' in note properties window as shown.



4. Select the **Text Style** tab from same window.
5. Mention Height 4, thickness 1, font CG TIMES, left, middle and red color for proper alignment of letter. Click on **Preview** and **OK** tab.

The same screen will look appear as shown in Figure.

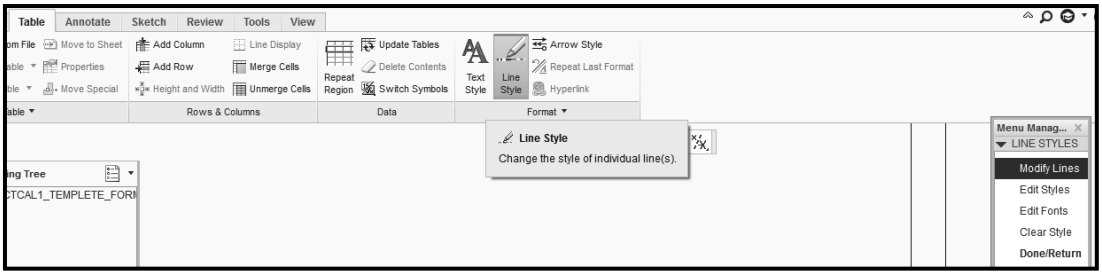


6. Repeat the same procedure for all letters.
   Finally you will get the TITLE BLOCK as shown in Figure.

NAME-	INSTITUTE NAME DEPARTMENT NAME	
ROLL NO-		
SEM.& SEC.-	LINES & LETTERING	
SUB.DATE-		SHEET NO.-
GRADE-		CHECKED BY-

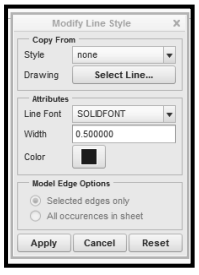
J. To change table line width and color:

1. Select **Table**>Select **Line Style** option button.
2. Keep Modify Lines as default as it is.



3. Select table of name plate by just clicking.
4. Select **OK**

5. New **Modify Line Style**definition dialog box will appear on screen.



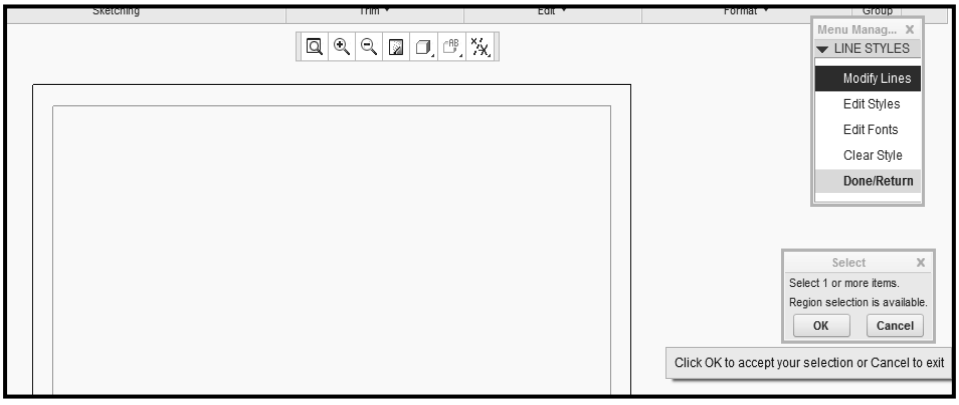
6. Enter required width 0.5 and brown color and click on **Apply** button.

The table line width will get changed and appear is similar to the one shown in Figure.

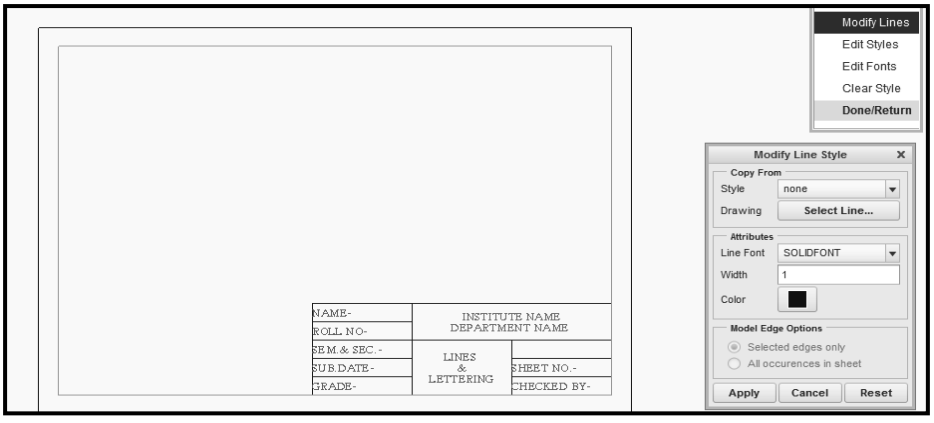
NAME-	INSTITUTE NAME DEPARTMENT NAME	
ROLL NO-		
SEM.& SEC.-	LINES & LETTERING	
SUB.DATE-		SHEET NO.-
GRADE-		CHECKED BY-

K. To change line width and color of border lines:

1. Select **Sketcher> Line Style> Modify Lines**.
2. Then select entire border lines by pressing Ctrl button of key board.
3. Accept your selection by clicking **OK**

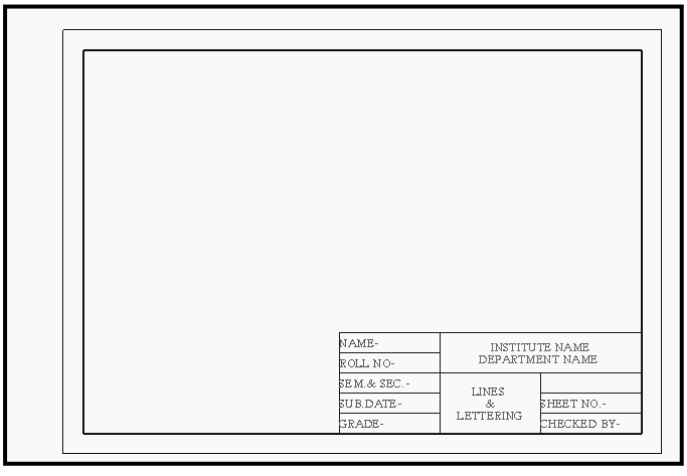


New **Modify Line Style** definition dialog box will appear on screen.



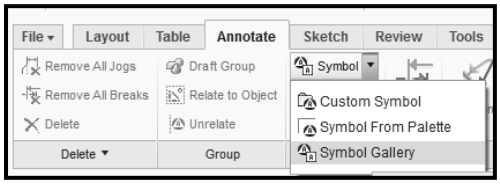
4. Enter the width by 1 and black color. Click to **Apply> Close>OK**

The border line width will get changed and appear is similar to the one shown in Figure.

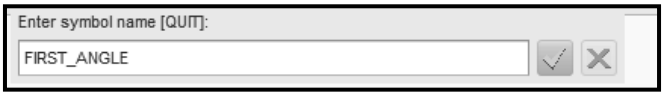


L. To createFIRST ANGLE symbol in template:

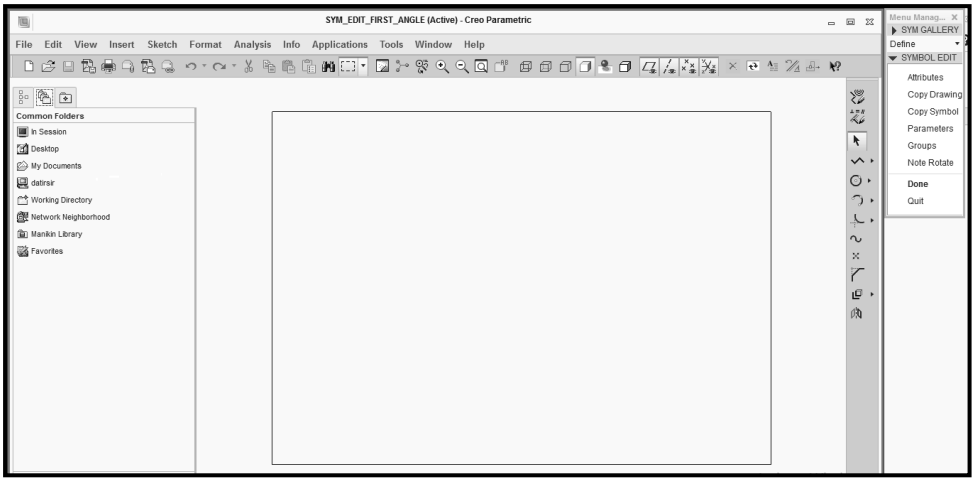
1. Select **Annotate** from menu bar.
2. Select **Symbol Gallery** from Symbol drops down list.



3. Select **Define** from Menu Manager.
4. Specify symbol name as **FIRST_ANGLE** and accept selection.

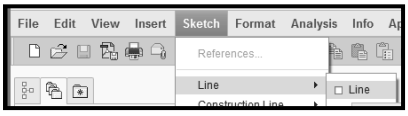


The next new screen will appear shown in Figure.

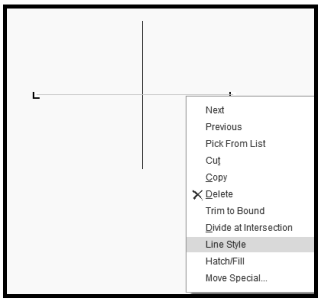


To draw first angle projection symbol. Use tool bar shown in previous Figure.

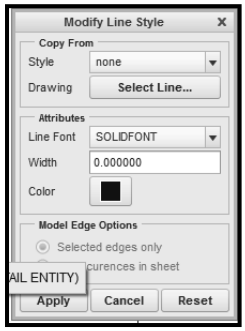
1. Select **Sketcher>Line**



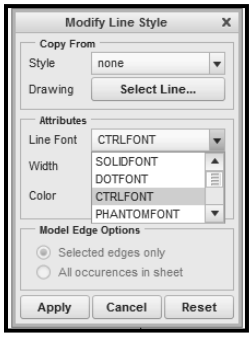
2. Start to draw symbol. First you have to draw horizontal and vertical lines.
3. Change line style by selecting line and click RMB. Select **Line Style.**



New **Modify Line Style**definition dialog box will appear on screen.

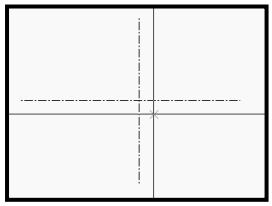


4. Select **CTRL FONT** and **Black**color**.** Click to **Apply** button.

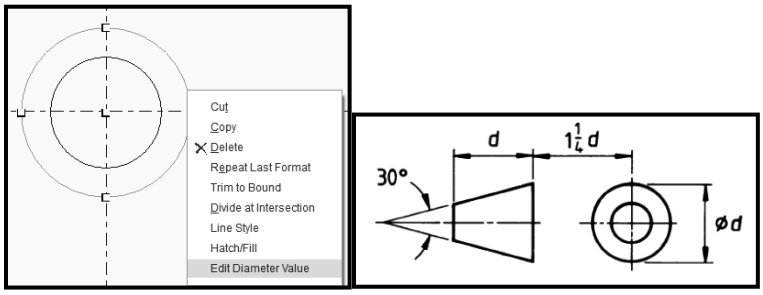


Continue the same procedure whenever you want to change the line style.

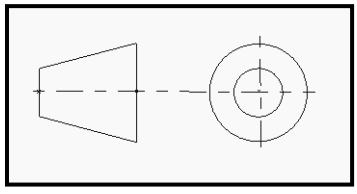
5. Next insert **Point** at intersection of two center lines as shown in Figure.



6. Draw two circles of diameter 9 and 4.5 by using **Edit Diameter Value** by right clicking option and accept it by clicking as shown in Figure.

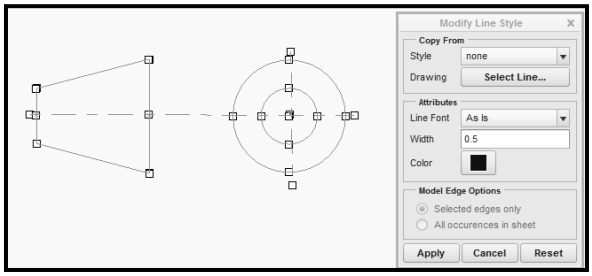


You have to draw remaining entities of symbol according to proportionate mention in Figure. Finally First Angle symbol will look like as shown in Figure.

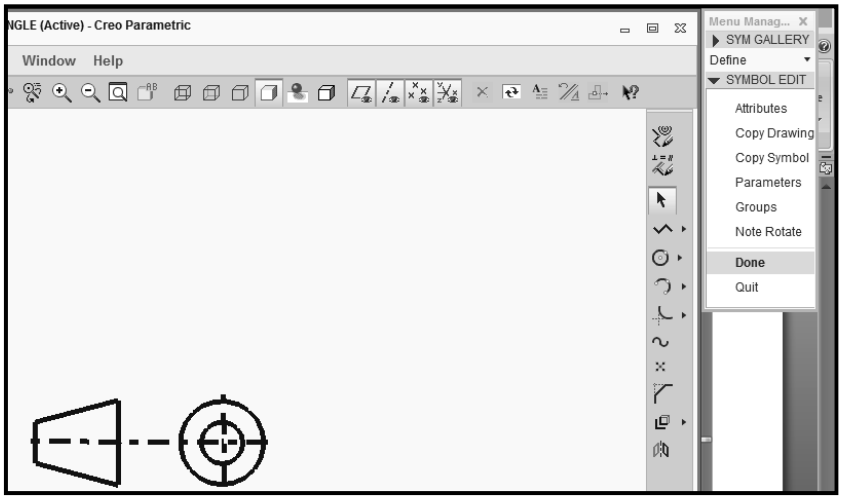


To change symbol width.

1. Select symbol by window method.
2. Right click and choose **Line Style** option.
3. Specify width by 0.5 as shown in Figure. Click **Apply**button.

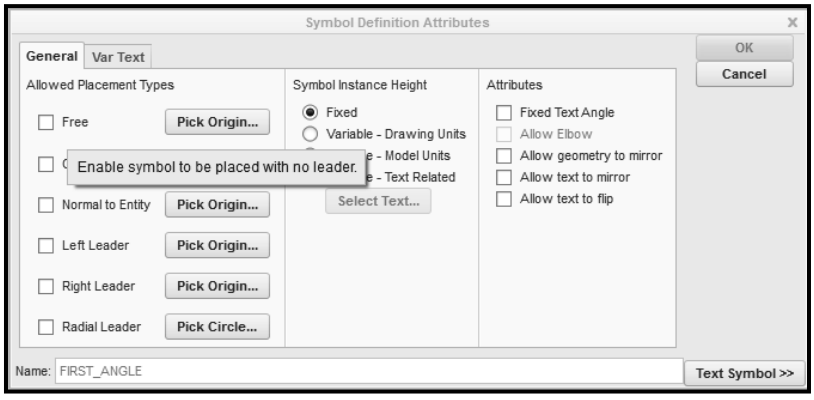


4. The symbol line width will change as shown in Figure. Next click on **Done** from the Menu manager.



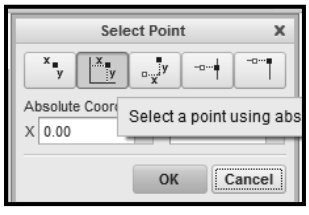
New window **Symbol Definition Attribute** dialog boxwill display on the screen as shown in Figure.

5. Tick the **Free** option and click on **OK.**



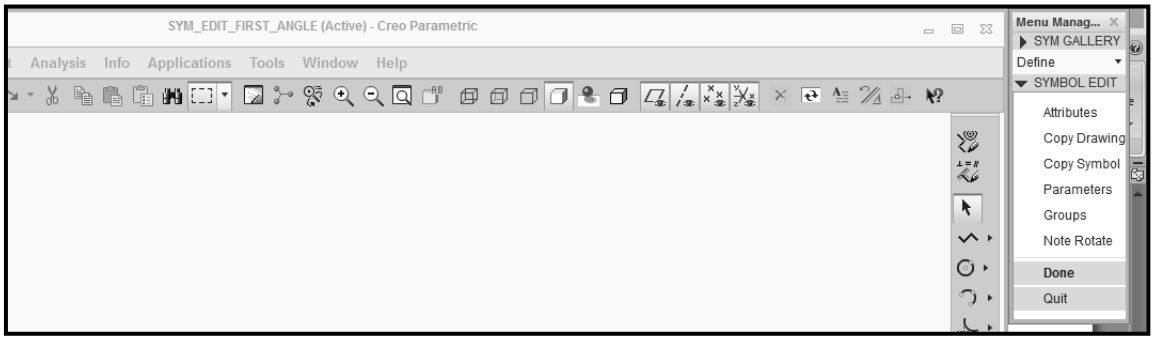
New Select Point definition dialog box will open.

6. Select a point using **absolute co-ordinates** option. Click on **OK**

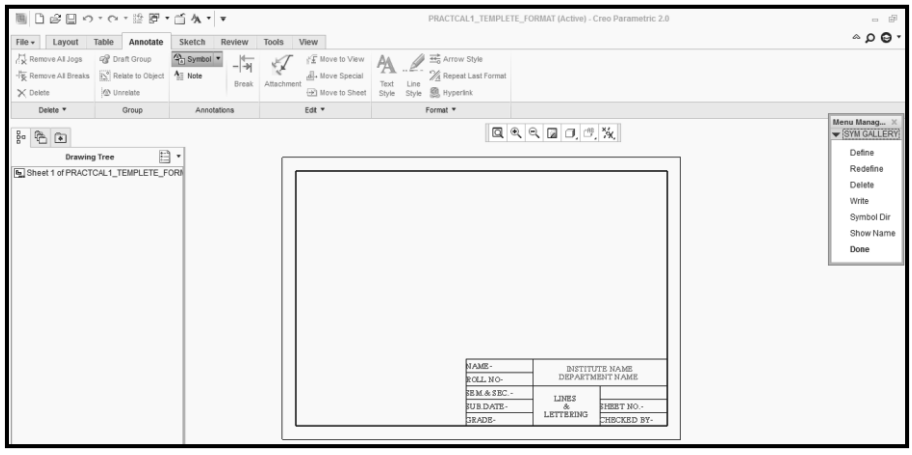


Again window **Symbol Definition Attribute** will display on the screen.Click on **OK**.

7. Select **Done** option from Menu Manager.



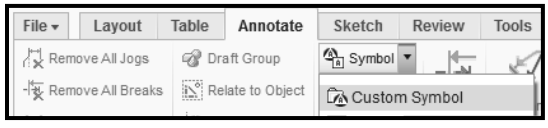
Now you will be at main screen as shown in Figure.



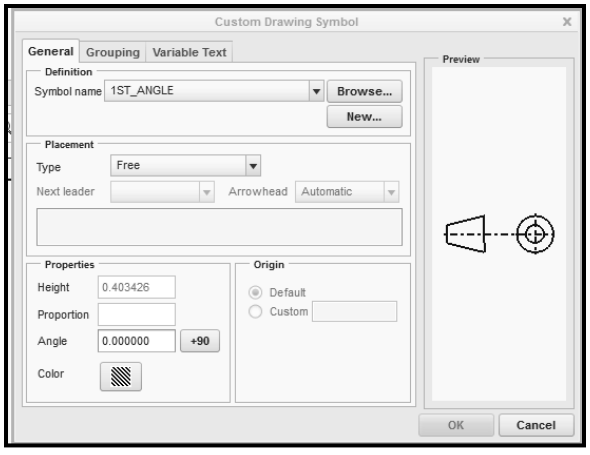
8. Select **Done** option from Menu Manager.
   First Angle symbol is created in **Custom Symbol.**

M. To insert prepared first angle projection symbol in name plate:

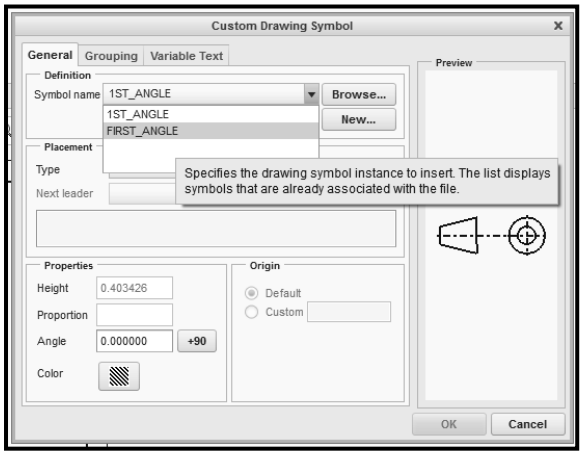
1. Select **Annotate> Symbol>Custom Symbol** from menu bar.



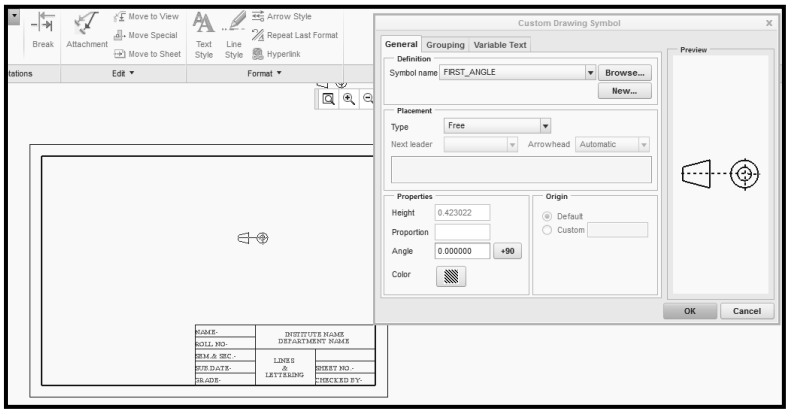
New **Custom Drawing Symbol** window will display on screen.



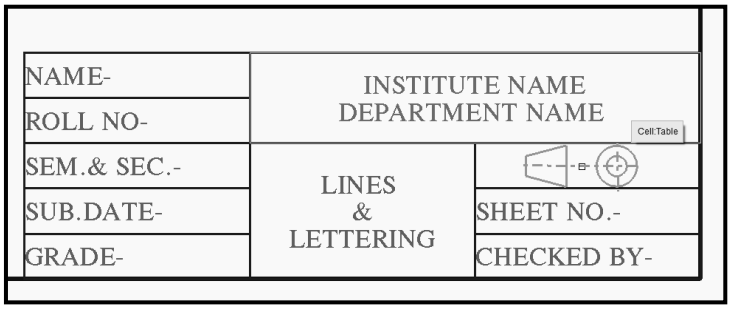
2. Select **FIRST_ANGLE** as the symbol name from the list as shown in Figure.



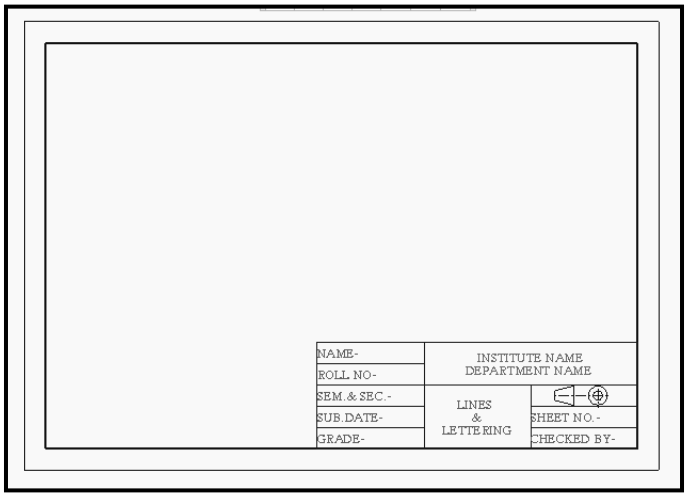
3. Now click on actual symbol and place it in drawing area by moving cursor as shown in Figure. Click on OK.



4. Move the symbol at required position.



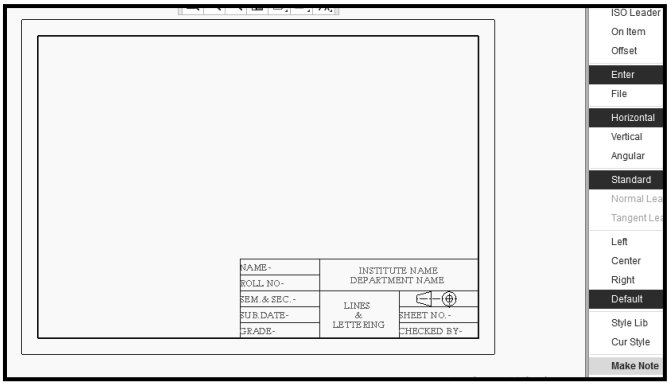
Finally sheet with symbol will look like as shown in Figure.



N. To add note without leader:

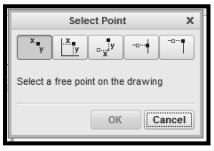
1. Click **Annotate** tab, select Note button 

Note and click **Make Note** options from Menu Manager.



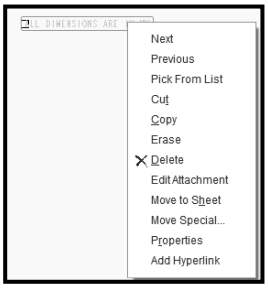
New **Select Point** window will appear on the screen.

1. Choose **Free point** and locate it as per requirement.

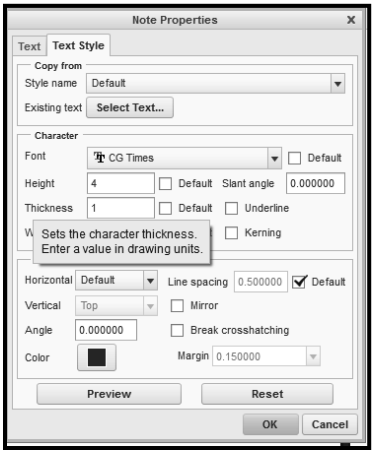


2. Write note as **ALL DIMENSIONS ARE IN MM** and accept it.
3. Select **Done** from Menu Manager.

4. Right click on letters you written. Select **Properties** option.



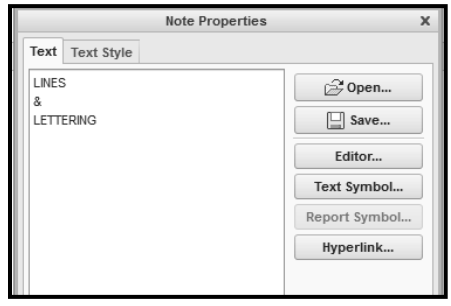
Note Properties window will open.



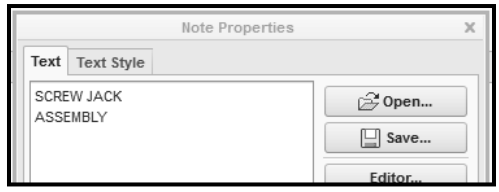
5. Specify font as CG Times, Height 4, Thickness 1, blue color. Select **Preview.** Click **OK** button. Finally mentioned note will look like as shown in Figure.

O. To change sheet name Lines & Lettering by SCREW JACK ASSEMBLY:

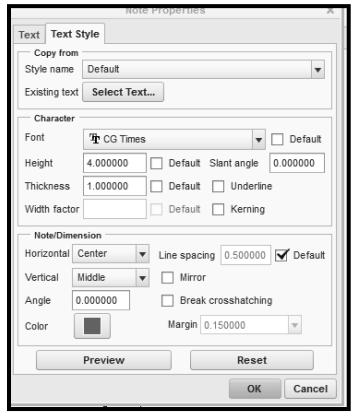
1. Double click to respective cell. **Note Properties** window will display.



2. Type **SCREW JACK ASSEMBY** in Text option.

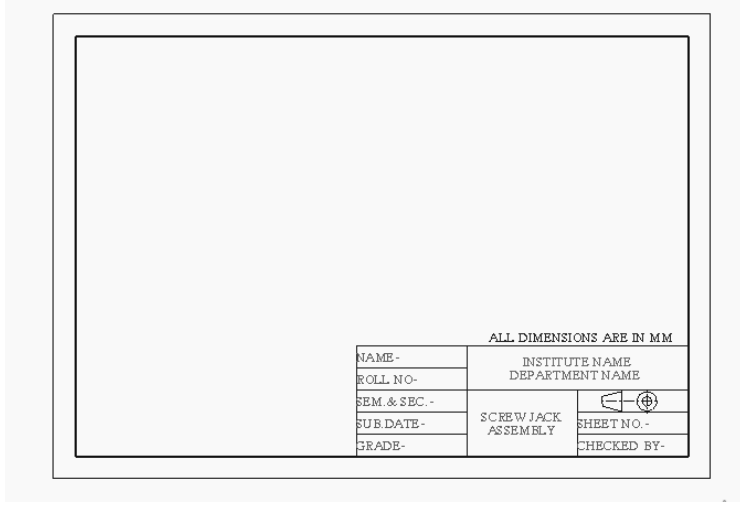


3. Select **Text Style** tab, and enter the font-4, Thickness -1, horizontally center and vertically middle alignment. Color –green.



4. Click **Preview** and **OK**.

Finally the A4 size landscape custom based template format will ready as shown in Figure.



P. Save the work by clicking the **Save** button. The A4 size custom based template format stored in working directory and we can use it for further work.

XII Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XIII Actual Procedure Followed

.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....

XIV Precautions Followed

.....
.....
.....
.....

XV Observations and Calculations

.....

.....

.....

.....

XVI Results

.....

.....

.....

.....

.....

XVII Interpretation of Results

.....

.....

.....

.....

.....

.....

XVIII Course proficiency

.....

.....

.....

.....

.....

.....

XIX Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions so as to ensure the achievement of identified CO.

1. List the different menu bar used in software.
2. Write the purpose of Title block in drawing sheet.
3. Explain information mention in title block.

[Space for Answer]

.....

.....

.....

.....

.....

.....

XX References / Suggestions for Further Reading

- https://www.youtube.com/watch?v=B96jGGkt-zk
- https://www.youtube.com/watch?v=cWCI8Klji80
- https://www.youtube.com/watch?v=zJrea53xtG8

XXI Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. ……..
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.2: Draw and print two simple 2D geometries using sketcher commands.

I. Practical Significance

To create a 3D feature, it is necessary to draw its 2D sketch. In the sketcher environment the sketch of the feature is created, dimensions and constrains are provided to sketch. The designer can make to make sure that the 2D sketch of the product is satisfying the necessary conditions, then continue to create 3D model of the product in the part mode.

II. Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III. Competency and Skills

This practical is expected to develop the following skills for the industry identified competency ***'Apply different sketcher environment commands to draw 2D geometries of the modeling software, apply various constrains and dimensioning to the 2D sketch'***

IV. Relevant Course Outcome(s)

- Prepare 2D Drawings using sketcher workbench of any Parametric Modeling software.

V. Practical Outcome

- Operate available modeling software to draw 2D sketch for engineering product.

VI. Relative Affective Domain-

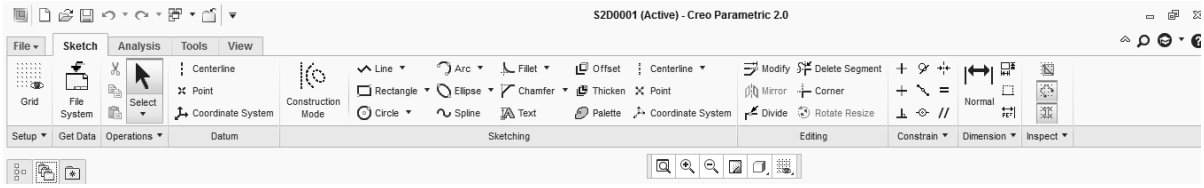
- Working in team work.
- Follow ethical Practices.

VII. Minimum Theoretical Background

- Basic knowledge of computer handling.
- Basic knowledge geometric constructions.
- Reading of engineering drawing.
- Basic knowledge of sketcher environment.

The sketcher environment -

Sketcher is the main creation tool ofparametric CAD software. The sketcher toolbar ribbon is located at the top of the window. This section explores the many options of sketcher. Its basic icons are shown in Figure. The LMB (Left Mouse Button) issued to select geometry or to select a location when creating geometry. When you invoke the **Sketch** mode, the initial screen appearance is similar to the one shown in Figure. This figure shows the **Sketcher Tools** as well as Setup, Get Data, Operations, Datum, Editing, Constrain, Dimension, and Inspect etc. toolbars displayed at the top side of the graphics window.

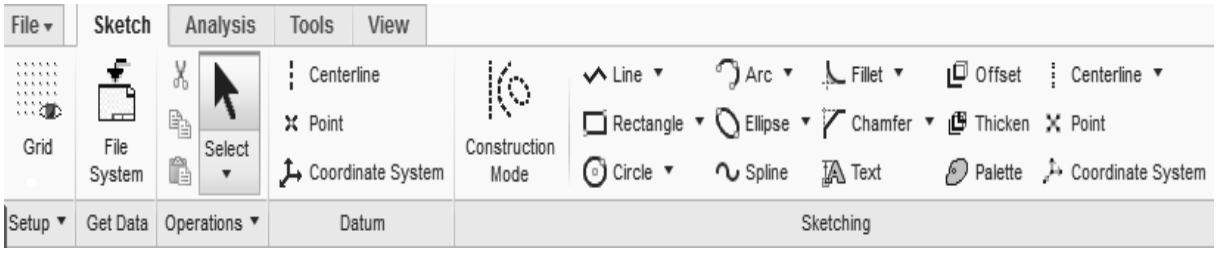


Reminder:

LMB = press the Left Mouse Button down, used to select points or features.
RMB = press the Right Mouse Button down, used to search through a series of features or used to bring up a pop-up menu.MMB = press the Middle Mouse Button down, used to cancel a command, place a dimension, or accept the current value.

Sketcher Tools Explained:

Now let's explain most of the sketcher icons. It is important to note that the LMBactivates each of the sketcher tools.

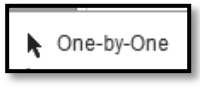


1 **Setup:**

Allows you to reset the sketch plane and references, sketching options, and to reorient your view (note this button is also on the Graphics toolbar).

Tool Name	Symbol	Use
Setup Tool Bar		
Grid		Define the grid settings.

2 Operations:

Operations Tool Bar		
Tool Name	**Symbols**	**Use**
Select		Select Items tool allows you to select features already on the screen by moving the cursor over the item, then pressing the LMB.
		Select one by one entity. For next entity selection press Ctrl and then click the entities.
		Select the chain of entities.

		Select all Geometry entities.
		Select all items the drawn section.

Datum Tool Bar		
Tool Name	**Symbol**	**Use**
Centerline		Draw an infinitely long geometric centerline byselecting two points with the LMB.
Points		Create a geometry point using the LMB.

Sketching Tool Bar		
Tool Name	**Symbol**	**Use**
Line Chain		Draw a solid line from first LMB pick location to secondLMB location pick.
Construction Mode		Toggle button is used to draw sketch in either geometry mode or construction mode.
Line Tangent		Draw a solid line tangent between two arcs or circles, whichare selected using the LMB.
Corner Rectangle		Sketch a rectangle by selecting two opposite corners of the rectangle using the LMB.
Slanted Rectangle		Sketch a slanted rectangle by sketching one side of the slanted rectangle using the LMB twice, and then moving perpendicular to this side to create the slanted rectangle's size.
Center Rectangle		Sketch a rectangle by selecting the center point of the rectangle, then one of its four corners using the LMB
Parallelogram		Sketch a parallelogram by sketching one side of the parallelogram using the LMB twice, and then moving away from this side to create the parallelogram's shape.
Center and Point Circle		Draw a circle by selecting the location of the circle's center with the LMB, then moving away from that point to create its radius.
Concentric Circle		Draw a new circle using the same center point as an existing circle. First the existing circle must be selected with the LMB, and then a new circle appears.
3-Point Circle		Draw a circle through three points which are selected by pressing the LMB three times.
3-Tangent Circle		Draw a circle tangent to three features which are selected by pressing the LMB three times.
3-Point/Tangent End Arc		Draw a circular (constant radius) arc by selecting its two endpoints using the LMB, then moving the cursor to size the arc's radius or make one end of the arc tangent to an existing feature. plus sign (+) will appear at the center of the arc.

Center and Ends Arc		Draw a circular (constant radius) arc by first selecting its center point using the LMB, then moving the cursor to size the arc's radius. Pressing the LMB sets the arc's radius.
3-Tangent Arc		Draw an arc tangent to three other features.
Concentric Arc		Draw a concentric arc using the same center point asan existing circle or arc.
Conic Arc		Draw a conic (variable radius) arc by selecting its two endpoints using the LMB, then moving the cursor to size the conic arc.
Axis Ends Ellipse		Draw an ellipse by selecting the end points of the major or minor axis, then moving perpendicular to this axis to size the other axis.
Center and Axis Ellipse		Draw an ellipse by selecting its center usingLMB, one end of its major or minor axis using LMB, then moving perpendicular to this axis to size the other axis.
Spline		Draw a free-hand spline curve by selecting spline points using the LMB.
Circular Fillet		Draw a circular fillet or round tangent to two features.
Circular Trim Fillet		Draw a circular fillet or round tangent to two features.
Elliptical Fillet		Draw an elliptical fillet or round tangent to two featuresat the two points selected on the features using the LMB.
Elliptical Trim Fillet		Draw an elliptical fillet or round tangent to two features at the two points selected on the features using the LMB.
Chamfer Fillet		Draw a chamfer between two intersecting lines starting at the points selected using the LMB, then remove the line segments in the area of the intersection of the two lines.
Chamfer Trim Fillet		Draw a chamfer between two intersecting lines starting at the points selected using the LMB, then remove the line segments in the area of the intersection of the two lines.
Offset		Create duplicate geometry an offset distance from the selected geometry.
Thicken		Create entities by offsetting an edge or a sketched entity ontwo sides.
Text		Create alpha characters and symbols on the sketch.
Text Along A Curve		To place text along a curve create the start and height points as before, then check the "Place along curve" box followed by selecting the curve to follow using the LMB.
Palette		Provides you with a customizable library of predefined shapes that you can readily import onto the active sketch plane. These shapes are presented in a palette.
Construction Centerline		Draw an infinitely long sketcher centerline by selecting two points with the LMB.

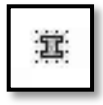
Construction Centerline Tangent		Draw an infinitely long sketcher centerline tangent to two circles or arcs using the LMB.
Construction Point		Create a construction point using the LMB. This point is known only inside sketcher and is not visible outside sketcher.
Construction Coordinate System		Create a construction coordinate system at the specified point by pressing the LMB.

Editing Tool Bar		
Tool Name	**Symbol**	**Use**
Modify		The common way is to highlight the dimensions to be modified using the LMB for the first. The second way is to select the tool first, then select a dimension to be modified using the LMB.
Mirror		Mirror a selection of features about a specified centerline, thus there must be a centerline present in the sketch.
Divide Entity at This Point		Divide a feature at the point of selection located by pressing the LMB. This will break a straight or curved line segment into two parts.
Delete Segment		Will remove any line segment that is drawn through while holding down the LMB.
Corner		This tool is used to trim the intersection of two line segments back to the intersection point.

Constraint Tool Bar		
Tool Name	**Symbol**	**Use**
Vertical Constraint		Force the selected line to be vertical. After selecting this tool, select the desired line using the LMB.
Horizontal Constraint		Force the selected line to be horizontal. After selecting this tool, select the desired line using the LMB.
Perpendicular Constraint		Force two selected lines to be perpendicular to each other. After selecting the tool, select the two lines using the LMB.
Tangent Constraint		Force a line, an arc, or a circle to be tangent to an arc or a circle. After selecting the tool, select the two features using the LMB.
Mid-point Constraint		Force a point to locate itself at the midpoint of a line segment or arc. After selecting the tool, select the approximate location of the. Midpoint using the LMB, then select the line segment or arc using the LMB.
Coincident Constraint		Make two points coincident, that is, the exact same point. After selecting the tool, select the two points using the LMB.
Symmetric Constraint		Force two points to be symmetric about the selected centerline. There must be a centerline present to use this command. After the tool is selected, pick the governing centerline using the

		LMB, then select the two points (one on each side of the centerline) using the LMB.
Equal Constraint		Force two or more features to be equal size. After the tool is selected, select the governing feature (line length or radius) using the LMB,then using the LMB select all features that you want to be the exact same size.
Parallel Constraint		Force two lines to be parallel. After the tool is selected, select the governing line segment using the LMB, then using the LMBselect all other line segments that you want to be the parallel to the governing line segment.

Dimension Tool Bar		
Tool Name	**Symbol**	**Use**
Normal Dimension		Add a strong dimension to the existing sketch.
Perimeter		Add a perimeter dimension to the sketch after selecting a dimension which can vary when the perimeter dimension is modified.
Baseline		Create an ordinate dimension baseline, either vertical or horizontal.
Reference Dimension		Add a reference (driven by other values) dimension to the existing sketch. Reference dimensions are added the same way as regular dimensions.

Inspect Tool Bar		
Tool Name	Symbol	Use
Overlapping Geometry		Pick the Overlapping Geometry icon to highlight the sketcher geometry that overlaps so that you can correct the problem.
Highlight Open Ends		Pick the Highlight Open Ends icon to highlight using green dots, the line segments that are not connected to anything, thus they are open ends.
Shade Closed Loops		Pick the Shade Closed Loops icon to fill in all closed figures so you can see which sections of your sketch are not closed for one reason or another.

Close Sketcher from Part Mode		
Tool Name	**Symbol**	**Use**
Accept		Accept the changes made in the sketcher and exit sketcher.
Cancel		Cancel the changes made in sketcher and exit sketcher.

Sketcher Graphics Tool Bar		
Refit		Adjust the zoom level to fully display the object on the screen.

Zoom In		Zoom in on target geometry to view it in greater detail.
Zoom Out		Zoom out to gain a wider perspective on the geometry.
Repaint		Redraw the current view (refresh).
Sketcher display filter		Define sketcher display.
Select All		To display all sketcher display.
Disp.Dims.		To display dimensions.
Disp.Constr.		To display constraints.
Disp.Grid.		To display grid.
Disp.Verts		To display vertices.

VIII. Experimental setup

.....

.....

.....

.....

IX. Resources Required

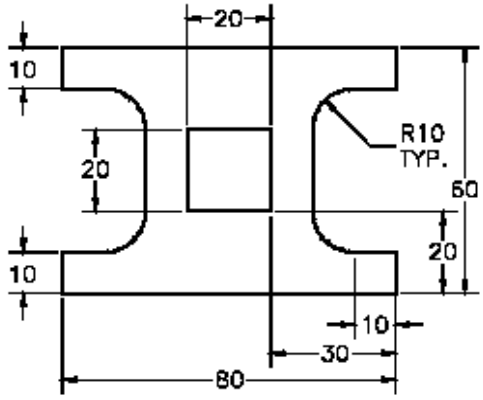
S. No.	Name of Resource	Suggested Broad Specification	Quantity
1	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3	Software	Any parametric solid modeling software.	As per batch size
4	Plotter	Plotter A_2 OR A_3Size.	1

X. Precautions to be Followed

1. Check given drawing for dimensional printing mistakes if any and if dimensions are missing assume proportionate dimensions.
2. While constructing the drawing, periodically save your work.

XI. Procedure-

Exercise No.1-Rerdaw the given 2D geometries.
(Use Grid and snap to grid option)



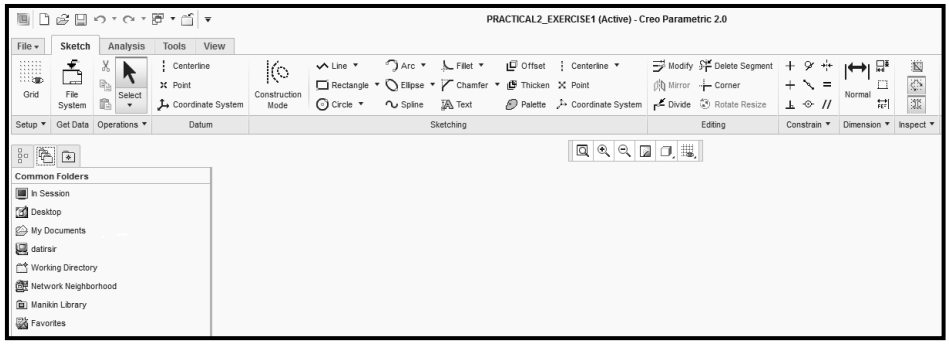
Following steps are required to sketch the given 2D geometries in available software. (Here you can use any CAD software)

A. Setting the Environment: As explained in practical No. 01.

B. Set working directory: As explained in practical No. 01.

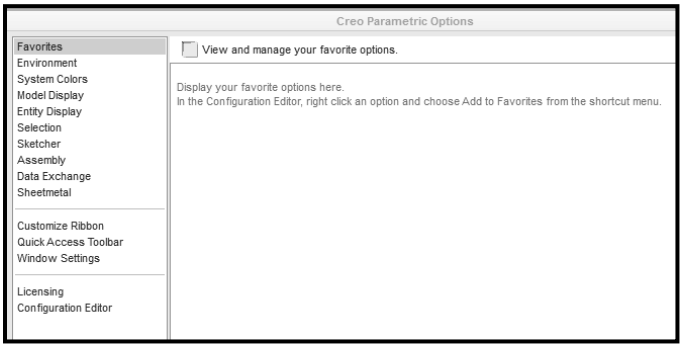
C. To invoke sketcher environment- by either selecting **File>New** from the menu or click on the **New File** icon from the main toolbar. A new window will be displayed as shown in Figure by default. Select Sketch option from the same window to sketch the drawing.Type name as PRACTICAL2_EXERCISE1.Click OK button.

Now you will be in the sketcher environment of the sketch as shown in Figure to sketch the 2D geometries.

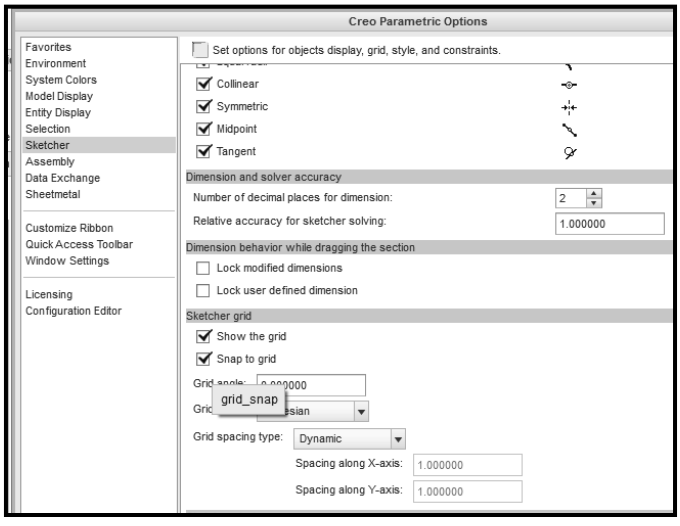


D. To sketch given 2D geometries-

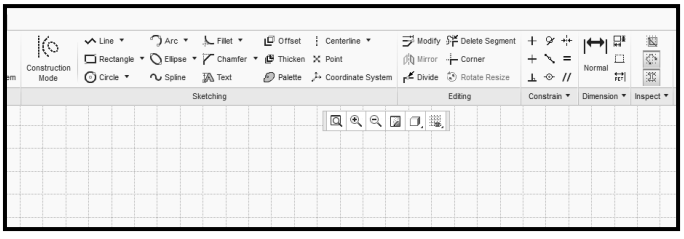
1. Initially set the sketcher workbench by invoking **File >Options** which display a **Solid Modeling Parametric Options** window as shown in Figure.



2. By selecting **Sketcher** from the list of the same window, you can check on **show the Grid** and**Snap to grid**. Also set the **Number of decimal places for dimension-2**. As shown in Figure.

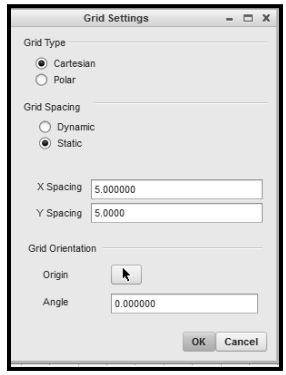


3. Now click **OK** button. The screen will look like as shown in Figure.
4. To set the X-spacing and Y-spacing for current exercise, follow the following procedure.



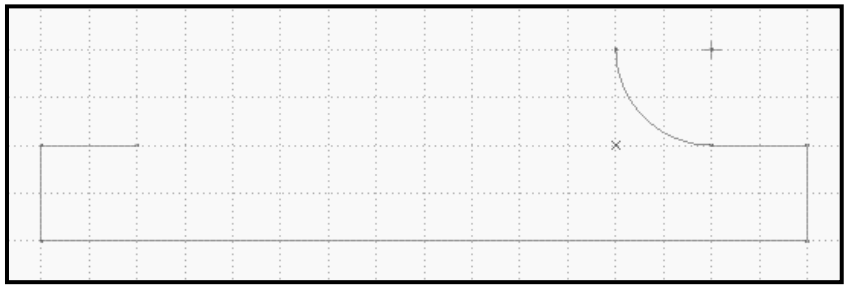
5. Click on 

Grid icon and specify the X& Y spacing is as 5 units as shown in Figure. Then click **OK** button of same window. Selection of grid lines helps to draw entities easily.

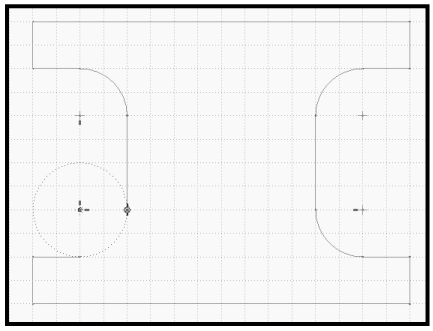


6. Choose  **Line** button from the sketcher tool bar. Click LMB on the drawing area and move the cursor left by 2 grid units to draw line of 10 units long as per the given drawing. When the cursor is moved horizontally left, horizontal constraint will applied automatically to the line as shown in Figure. Press **LMB.**

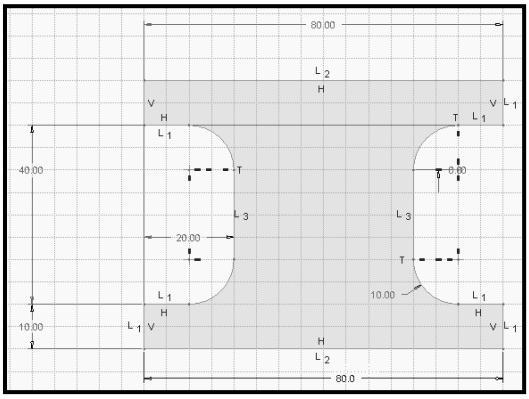
7. Continue the same procedure till the point as shown in Figure.
8. Now draw the arc by selecting **Arc >Center and End**  option as shown in Figure.



9. In similar manner complete the entire close loop drawing as shown in Figure.



10. Press the MMB. Weak dimensions will display on the sketch as shown in Figure.

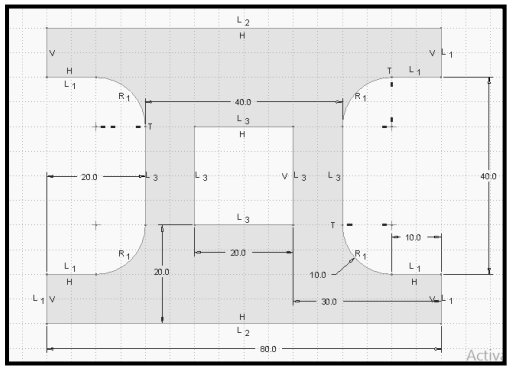


E. Modify the Dimensions:

1. To modify a dimension by **double-clicking** on its value using the LMB, and then type a new value as per the drawing.
2. Use 

Normal Dimension option to provide additional dimensions if required by **resolving** other dimensions.
3. You can also use 

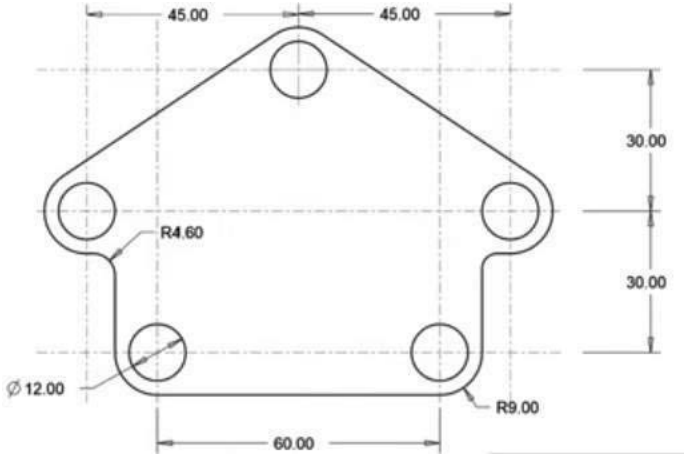
Modify Dimensions to change the dimensions.
4. Continue the same procedure for sketching square inside the drawing. In this way, all sketch dimensions is changed as per the given drawing as shown in Figure



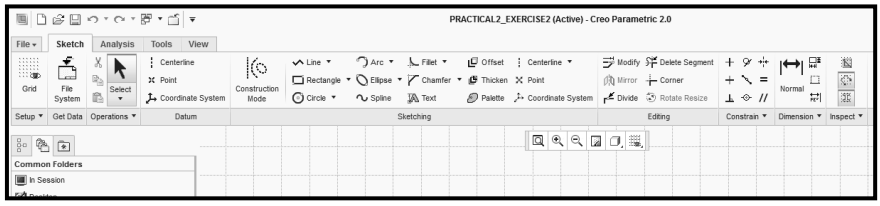
F. Click**Save** button which saves sketch by **PACTICAL2_EXERCISE1**with **.sec** file ext.

G. Print the created sketch in MS Word by Print Screen Shot.

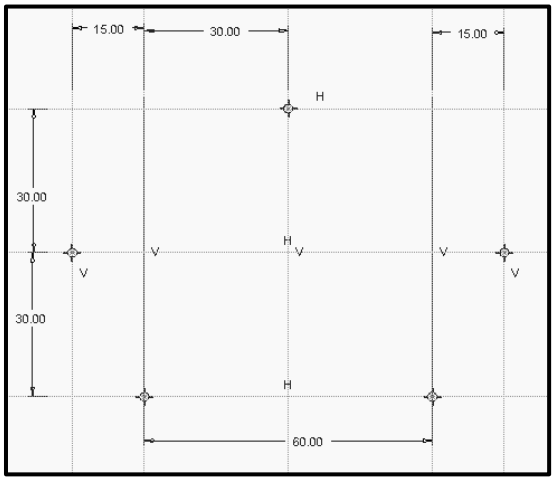
Exercise No.2-Rerdaw the given 2D geometries using sketcher workbench.(Without Grid)



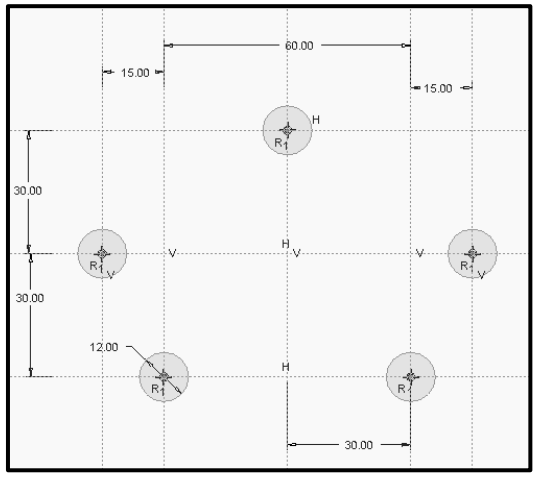
1. **To invoke sketcher environment**-As explained in practical No.2, exercise 1**.**Type name as **PRACTICAL2_EXERCISE2**.Click **OK** button.
2. Now you will be in the sketcher environment of the sketch as shown in Figure to sketch the 2D geometries.



3. Draw the center lines, circle center points  as shown in figure.
4. To modify a dimension by **double-clicking** on its value using the LMB, and then type a new value as per the drawing.
5. You can use  **Modify** option to change the dimensions as shown in Figure.

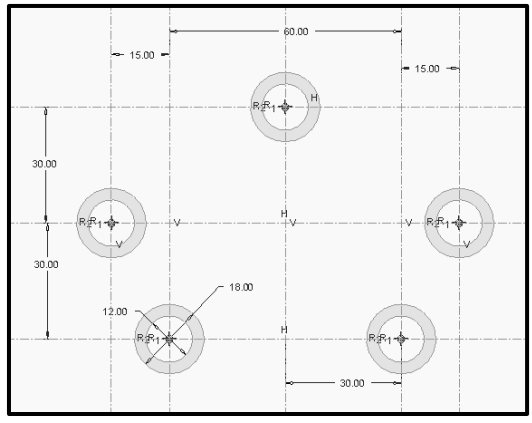


6. Draw internal circles of given diameters at different center pointsas shown below by using  **Circle** command from sketching tool bar.

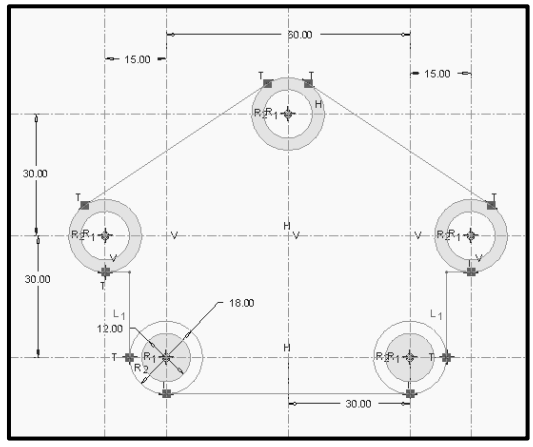


7. Draw the outer circles at the same center points using 

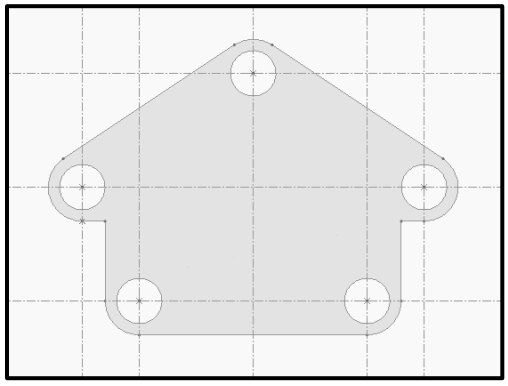
commands as per given dimensions.

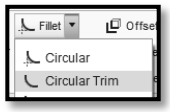


8. Draw tangent lines to outer circles by using  **Line Tangent** command from sketching tool bar as per the given drawing.

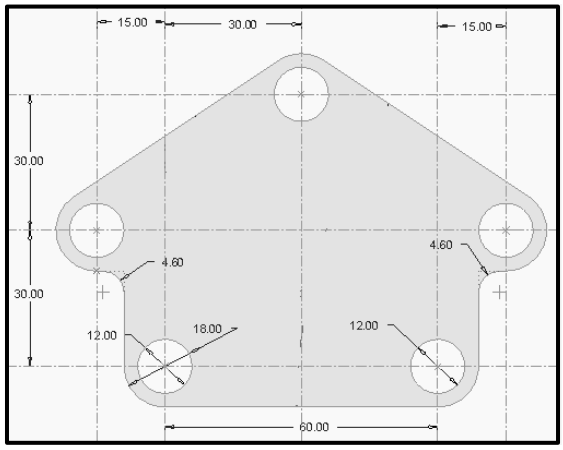


9. Trim the unwanted parts of circles by using  **Trim** command from editing tool bar.



10. Give the fillets at respective position by using 

Circular Trim command from sketching tool bar and complete the sketch as shown in Figure.



11. Click**Save** button which saves sketch by **PACTICAL2_EXERCISE2**inworking directory with **.sec** file extension.

12. Print the created sketch in MS Word by Print Screen Shot.

XII. Resources Used

S. No	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XIII. Actual Procedure Followed

.....

.....

.....

.....

XIV. Precautions Followed

.....

.....

.....

.....

XV. Course proficiency

.....

.....

.....

.....

XVI. Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Explain purpose of working directory in software.
2. Enlist sketcher toolbar to draw basic 2D entities.

[Space for Answer]

.....

.....

.....

.....

.....

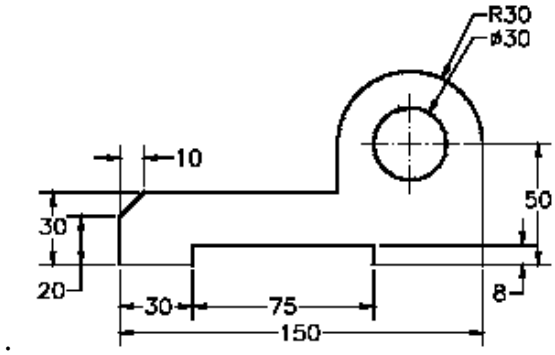
.....

.....

.....

.....

XVII. Questions for Practice.



XVIII. References / Suggestions for Further Reading

- https://www.youtube.com/watch?v=lpH4ZUUD9N0
- https://www.youtube.com/watch?v=qFIev5cRlW4
- https://www.youtube.com/watch?v=lpH4ZUUD9N0
- https://www.youtube.com/watch?v=sVWsUS_7V6s&list=PLrOFa8sDv6jfVMccV28fssFut0EG0NNb6

XIX. Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. …….
4. ……..

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.3: Draw and print two given complex 2D geometries using sketcher commands.

I. Practical Significance

To create a 3D feature, it is necessary to draw its 2D sketch. In the sketcher environment the sketch of the feature is created, dimensions and constrains are provided to sketch. The designer can make to make sure that the 2D sketch of the product is satisfying the necessary conditions, then continue to create 3D model of the product in the part mode.

II. Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III. Competency and Skills

This practical is expected to develop the following skills for the industry identified competency ***'Apply different sketcher environment commands to draw 2D geometries of the modeling software, apply various constrains and dimensioning to the 2D sketch'***

IV. Relevant Course Outcome(s)

- Prepare 2D Drawings using sketcher workbench of any parametric modeling software.

V. Practical Outcome

- Operate available modeling software to draw 2D sketch for engineering product.

VI. Relative Affective Domain

- Working in team work.
- Follow ethical practices.

VII. Minimum Theoretical Background

- Reading of engineering drawing.
- Basic knowledge of CAD software and commands.

VIII. Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3	Software	Any parametric solid modeling software.	As per batch size
4	Plotter	Plotter A2 OR A3 Size.	1

IX. Experimental setup

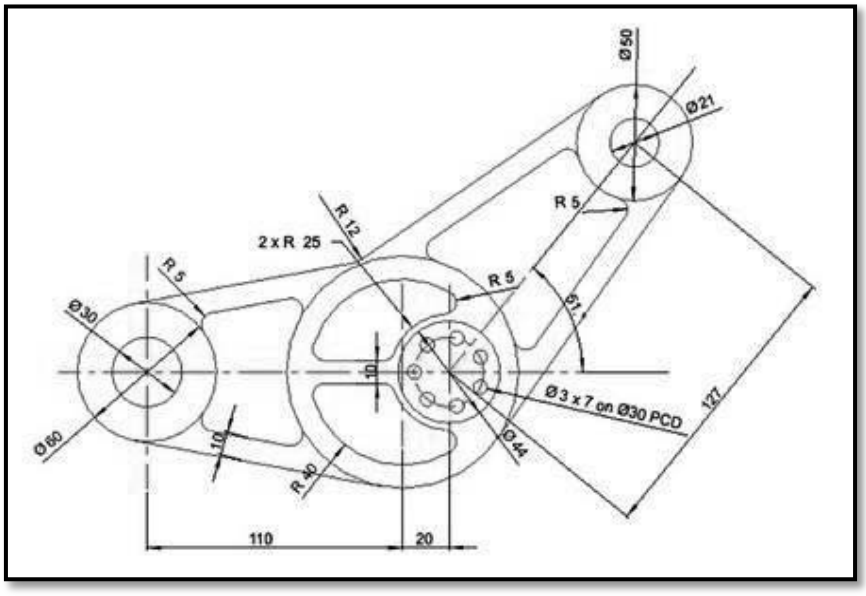
.....
.....
.....
.....

X. Precautions to be Followed

1. Check given drawing for dimensional printing mistakes if any and if dimensions are missing assume proportionate dimensions.
2. While constructing the drawing, periodically save your work.

XI. Procedure-

Exercise No.1-Redraw the following given 2D geometries using sketcher workbench as shown in Figure.



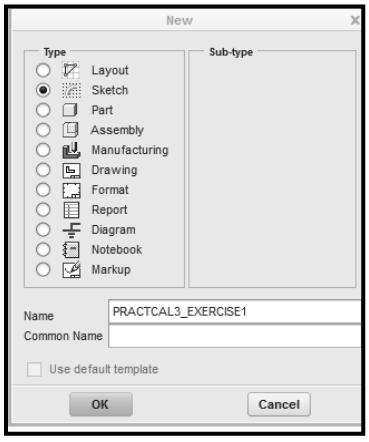
Following steps are required to sketch the given 2D geometries.

A. Start Solid Modeling Parametric CAD software: As explained in practical No. 01.

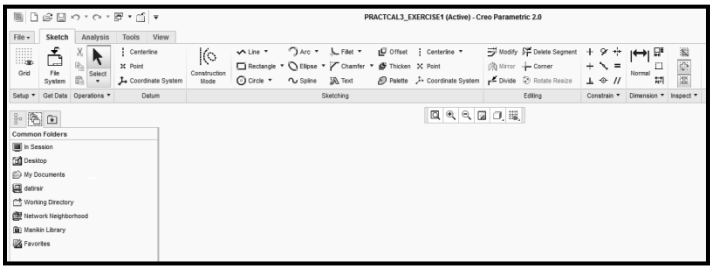
B. Set working directory: As explained in practical No. 01.

C. To create sketcher environment: As explainedin practical No.02.

1 Type name as **PRACTICAL3_EXERCISE1**.Click **OK** button.

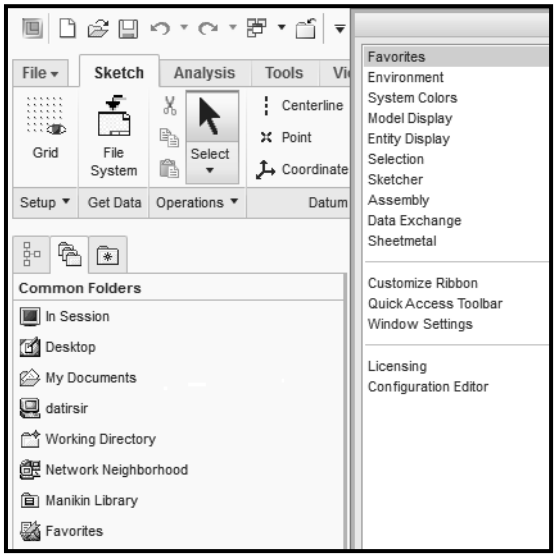


Now you will be in the sketcher environment as shown in Figure to sketch the 2D geometries.

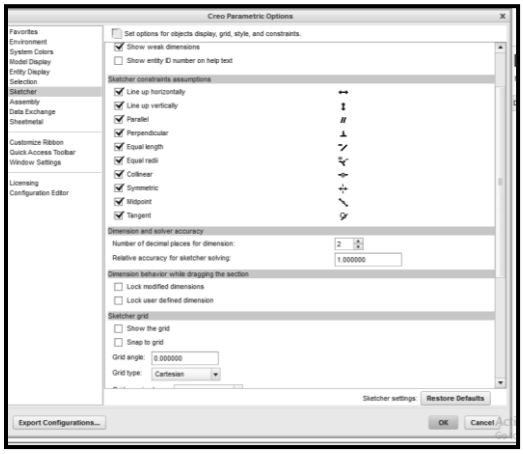


D. To sketch given 2D geometries:

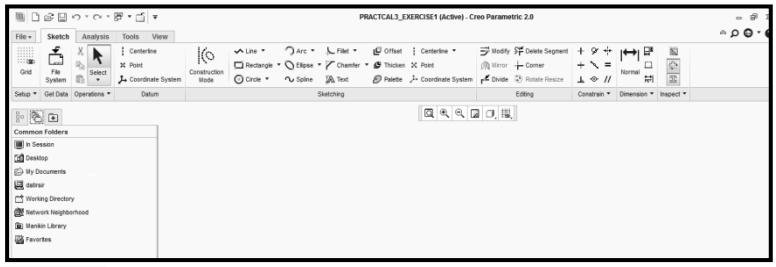
1. Initially set the sketcher workbench by invoking **File > Options**which display a **Solid Modeling Parametric Options** window as shown in Figure.

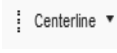


2. By selecting **Sketcher** from the list of the same window, specify the number of decimal places for dimension. For the current exercise, 2 decimal places for dimension needed. As shown in Figure.



3. Now click **OK** button. The screen will look like as shown in Figure.



4 Use 

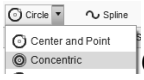
Centerlineoption to draw vertical and horizontal centerlines.

5 To modify a dimension bydouble-clickon its value using the LMB, and then type a new value as per the drawing.

6 Draw one more center line inclined at an angle 51º using same procedure.

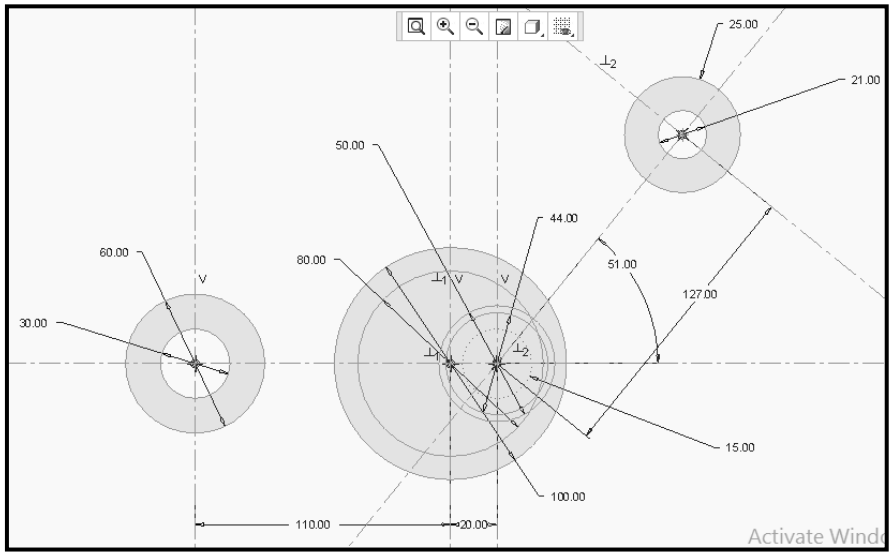
7 Use 

Point option to locate the center point of the circles.

8 Use 

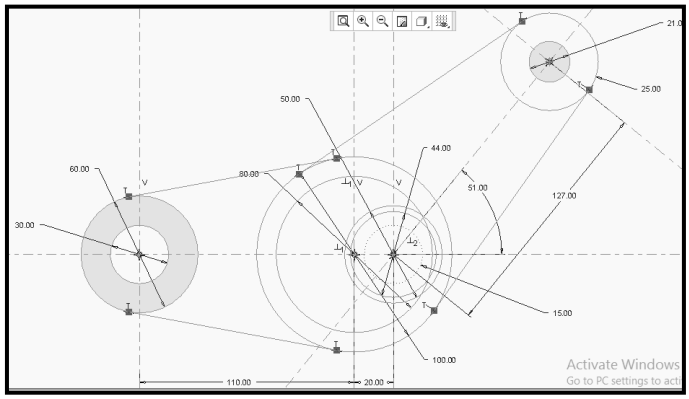
Concentric option to draw circles of different diameter as per the given drawing.

9 To modify a dimension of the circle by double-clickon its value using the LMB, and then type a new value Ø 30,60,100,80,15,44,50,25 & 21 as per the drawing.



10 To draw tangent lines to the circles. Use 

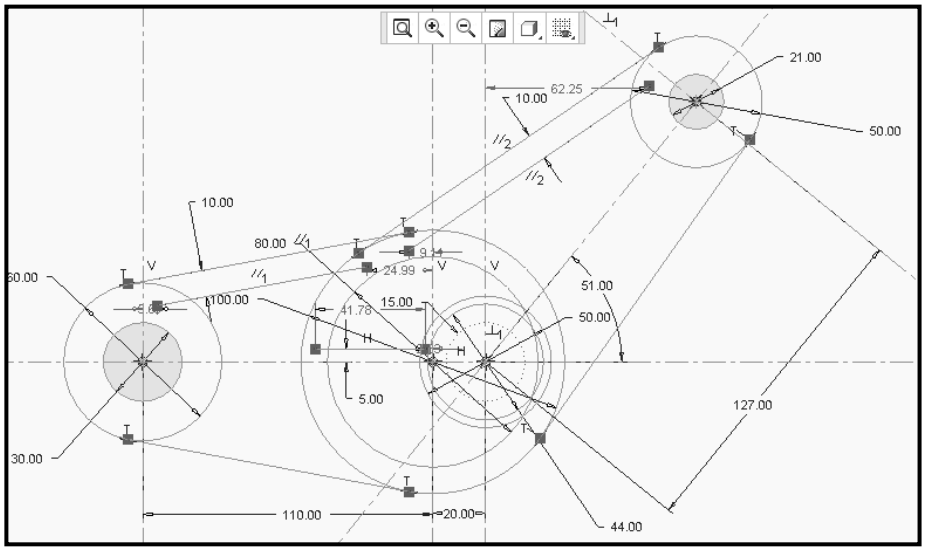
LineTangent option.Click **Line>Line Tangent** button, and select the tangent point of first circle and tangent point of second circle as shown in Figure. Continue the same procedure to draw four tangent lines.

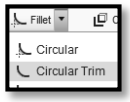


11 To draw lines parallel to the tangent lines as shown in above Figure, Choose 

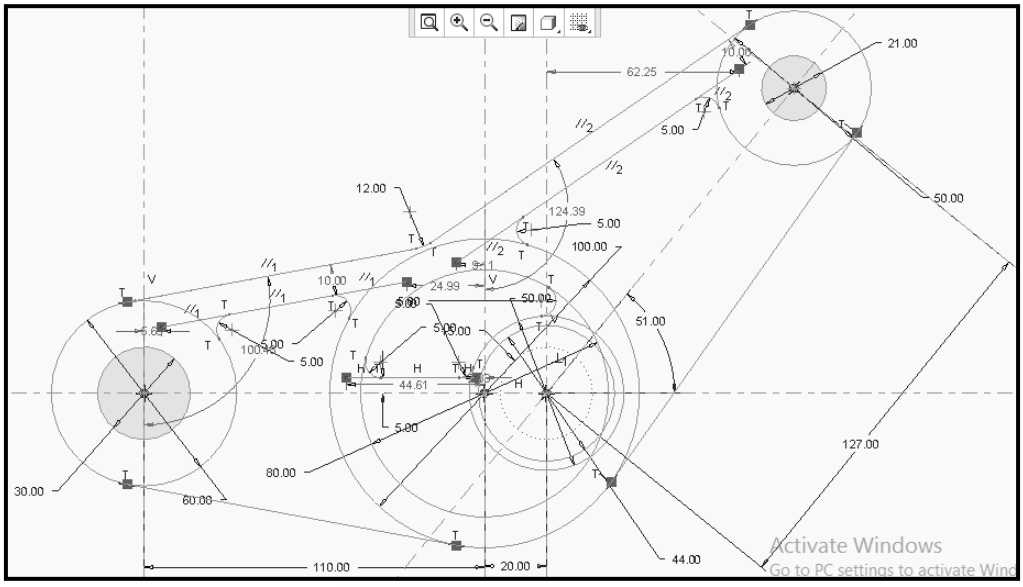
Line button from sketcher tool bar. Pick the first and second point near to the tangent line. Continue the same procedure for next parallel line. Press MMB to exit.(Here parallel constraint is applied automatically)

12 To modify a dimension of the parallel lines by double-click on its value using the LMB, and then type a new value 10 mm as per the drawing.



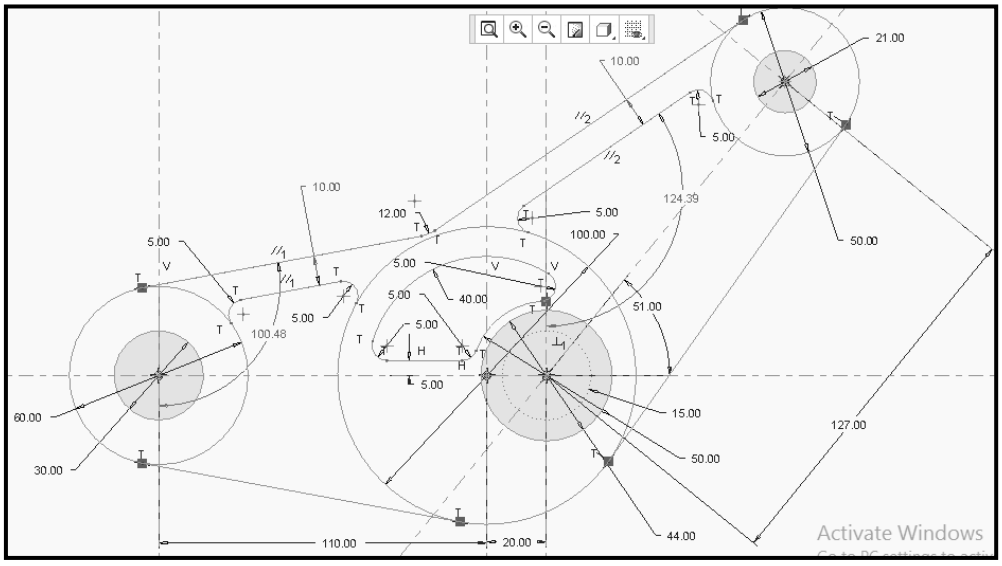


1. To draw fillet of 5 mm. Use **Fillet** from sketcher tool bar. Click **Fillet>Circular Trim.** Select line as first entity and Ø 60 circle as second entity. Repeat the same procedure for all fillets as shown in Figure.
2. To modify a dimension of the fillet radius by double-click on its value using the LMB, and then type a new value 5mmas per the drawing.



3. To trim unwanted lines, use 

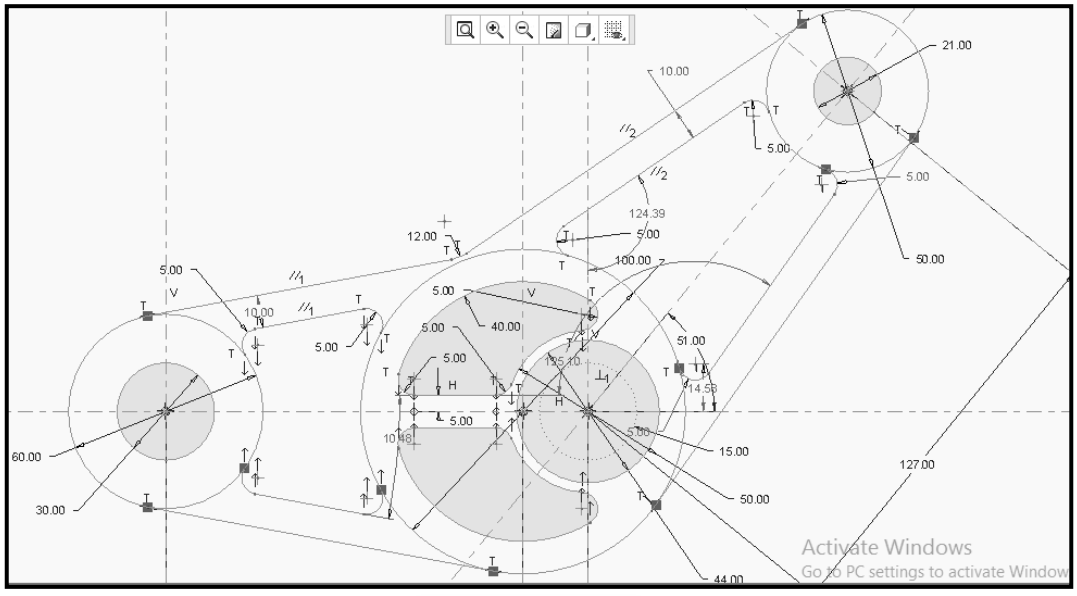
Delete Segment option of editing tool bar. Click on the **Delete Segment** button and then choose unwanted lines one by one as shown in Figure.



4. Use 

Mirror option from the editing tool bar to mirror fillet and line. Click 

button, select all entities to be mirrored by keep holding **Ctrl** key board button and select entities to be mirrored one by one. Once the selection complete, release **Ctrl** button and click center line. The selected entities will get mirrored as shown in Figure.

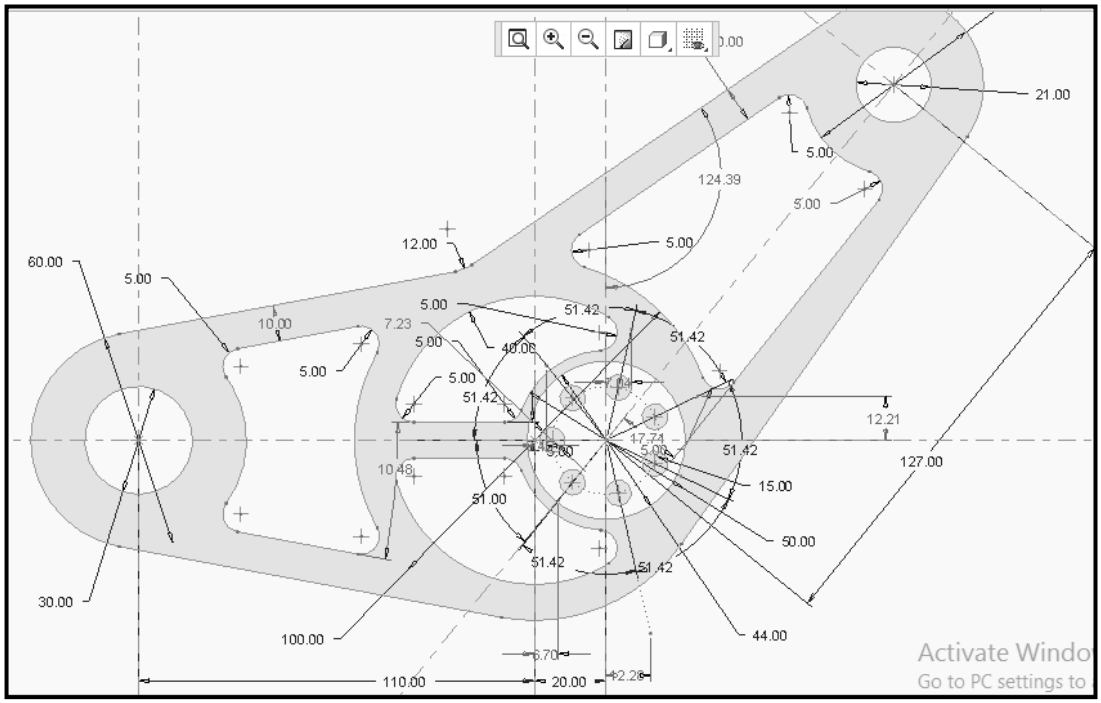


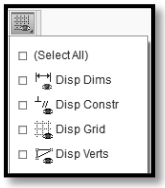
5. To trim unwanted lines, use 

Delete Segmentoption of editing tool bar. Click on the 

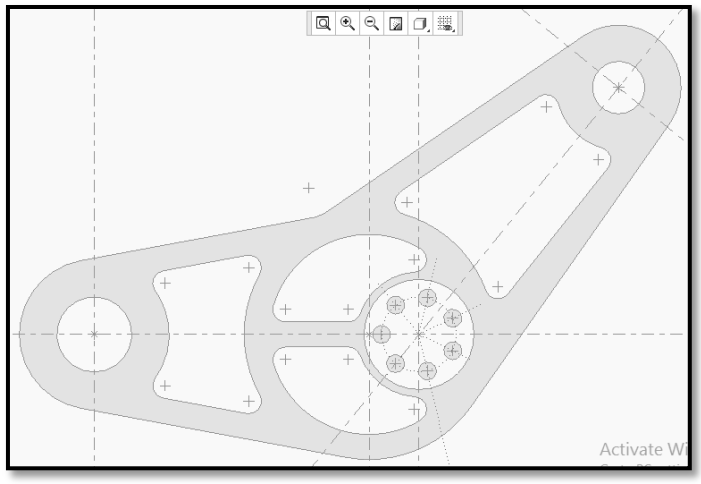
Delete Segmentbutton and then choose unwanted lines one by one as shown in Figure. Draw ϕ3 x 7 on ϕ30 PCD as per the given drawing.

6. Use  **Shade Closed Loop** button in active mode to appear shaded closed loop portion as shown in Figure.



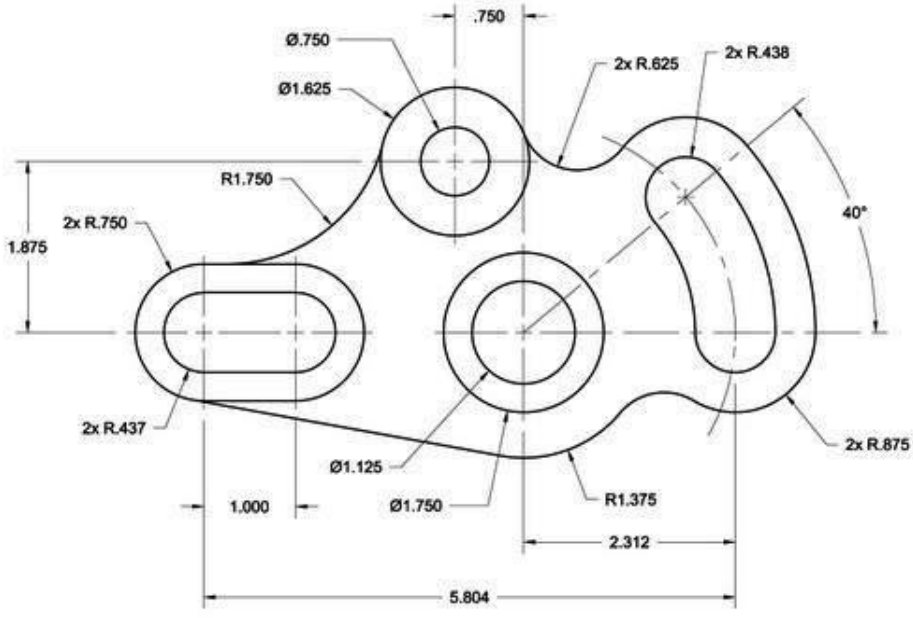


7. Use **Sketcher Display Filter** option to see final sketch as shown in Figure.



E. Click **Save** button which saves sketch by **PACTICAL3_EXERCISE1** with **.sec** file extension.

Exercise No.2-Rerdaw the given 2D geometries using sketcher workbench as shown in Figure.



Following steps are required to sketch the given 2D geometries.

A. Start Solid modeling Parametric: As explained in practical No.01.

B. Set working directory: As explained in practical No.01.

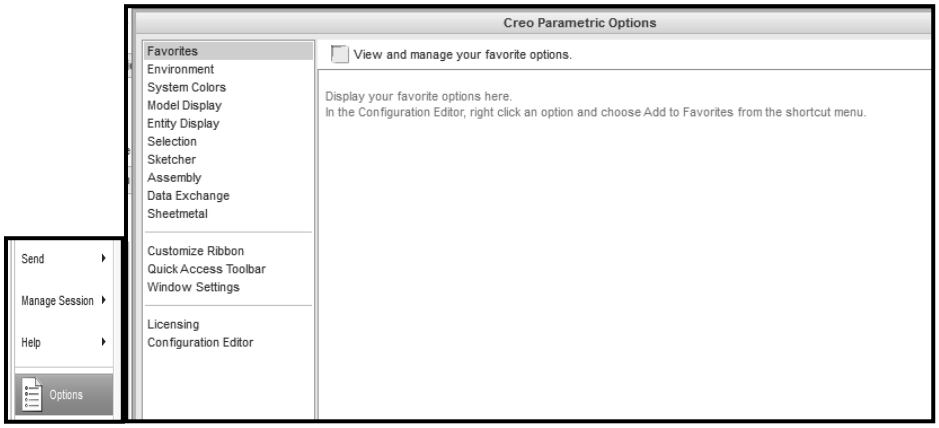
C. To create sketcher environment: As explained in practical No.02.

1 Type name as **PRACTICAL3_EXERCISE2**.Click **OK** button.

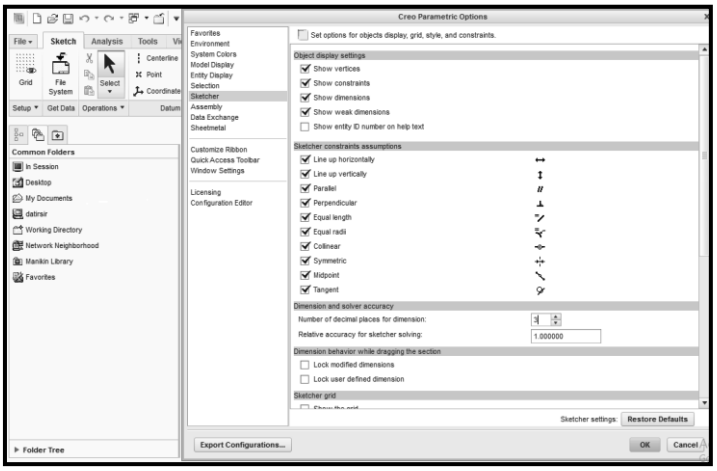
2 Now you will be in the sketcher environment to draw sketch the 2D geometries.

D. To sketch given 2D geometries:

1. Initially set the sketcher workbench by invoking **File >Options** which display solid modeling parametric options window as shown in Figure.



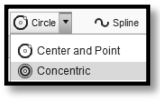
2. By selecting **Sketcher**optionfrom the list of the same window.
3. Specify the number of decimal places for dimension. For the current exercise, 3 decimal places of dimension needed. As shown in Figure.



4. Now click **OK** button. The screen will look like as shown in Figure. Now you are in sketcher environment to draw the given sketch.
5. Use 

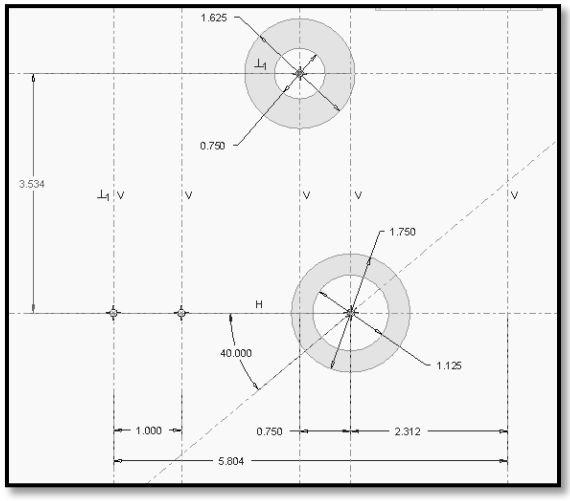
Centerlineoptionsto draw vertical and horizontal centerlines.
6. To draw 5 vertical center lines, Select 

and click first point and second point vertically. Continue the same procedure for next vertical centerlines.
7. To draw 2 horizontal centerlines, continue the same procedure clicking two points horizontally for each center line.
8. Draw one more center line inclined at an angle 40º using same procedure.
9. To modify a linear and angular dimension of centerlines, double-click on its value using the LMB, and then type a new value according to given drawing.
10. Use 

to locate the center point of the circles.
11. Use 

Concentric option to draw circles of different diameter as per the given drawing.

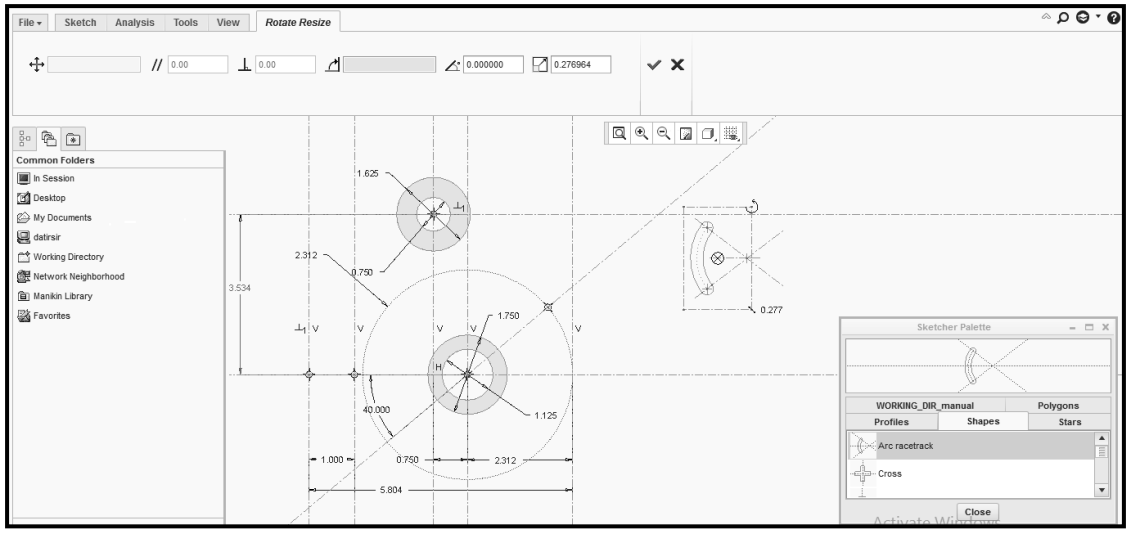
12. To modify a dimension of circles, double-click on its value using the LMB, and then type a new value according to given drawing.



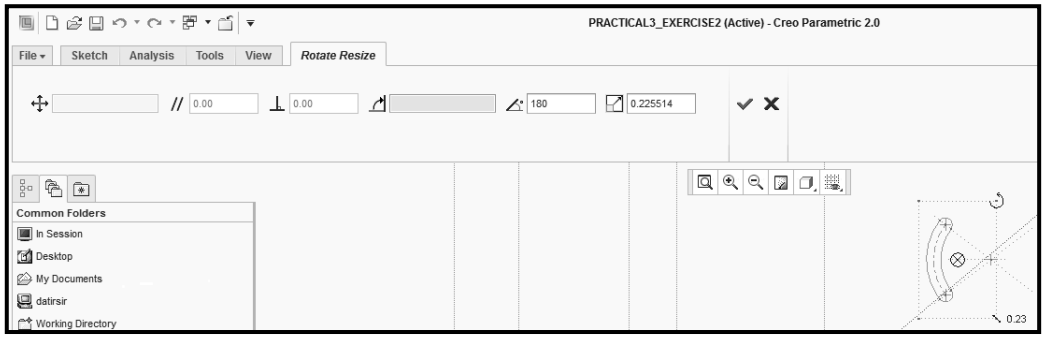
13. To draw elongated cylindrical hole, use 

Palette tool. Click on 

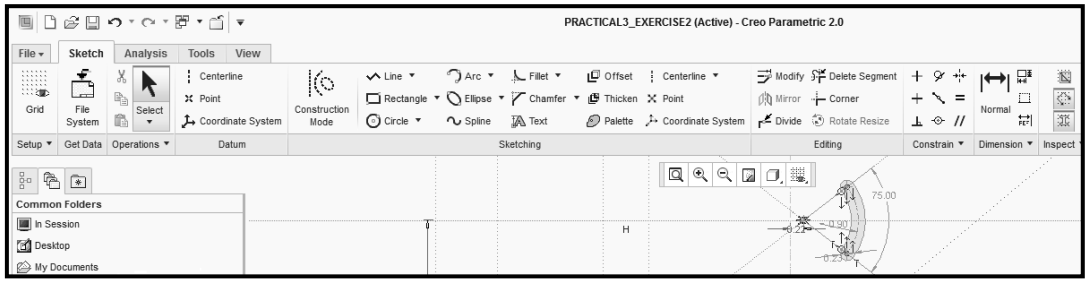
Palette icon. Palette sketcher window will appear on the screen.
14. Select **Arc racetrack** from the **Shapes** option. Drag it in drawing area as shown in Figure.



15. To rotate it by 180º, enter 180 values.



16. Accept by clicking on green colored check window as shown in Figure.



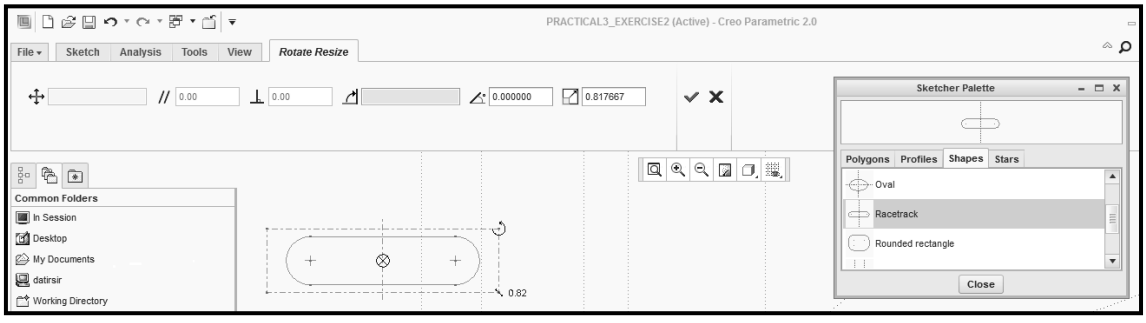
17. To modify a dimension of arc racetrack by double-click on its value using the LMB, and then type a new value according to given drawing.

18. To locate arc racetrack to correct position as per the given drawing. Use  **Coincident** tool from the sketcher tool bar.

19. First select  coincident button from sketcher tool bar. Then select first point as center point of the circle R 2.312 and second point will be center point of arc racetrack.
20. Similarly coincide the remaining two points of arc racetrack with respect to circle R2.312.Out of that one point will be coincident to intersection of point of the circle radius R2.312 and center line which making an angle 40º with horizontal center line. The second point will be coincident to intersection of point of the circle R 2.312 and horizontal center line of the circleR 2.312.
21. To draw elongate hole, use 

Palette tool. Click on 

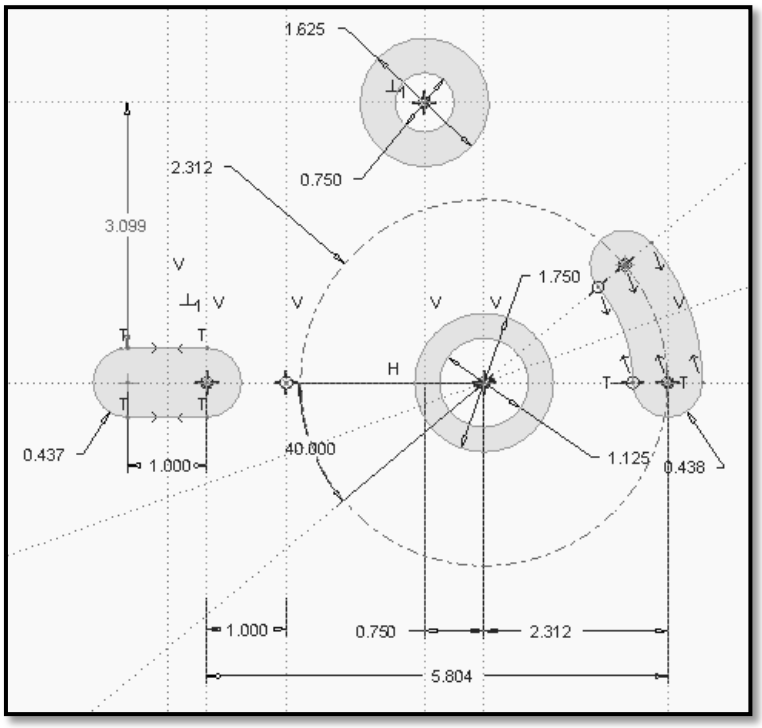
Palette icon. Palette sketcher window will appear on the screen. Select **racetrack** from the **Shapes** option. Drag it in drawing area as shown in Figure.
22. Accept it by clicking on green colored check window as shown in Figure.



23. To modify a dimension of racetrack by double-clickon its value using the LMB, and then type a new value according to given drawing.

24. To locate racetrack to correct position as per the given drawing. Use  **Coincident**tool from the sketcher tool bar.

25. First select  button from sketcher tool bar. Applying same method explained earlier case to coincident the two points of racetrack with the main sketch as shown in Figure.

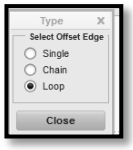


26. To Offset the arc racetrack by 0.437 and offset race track by 0.313 using 

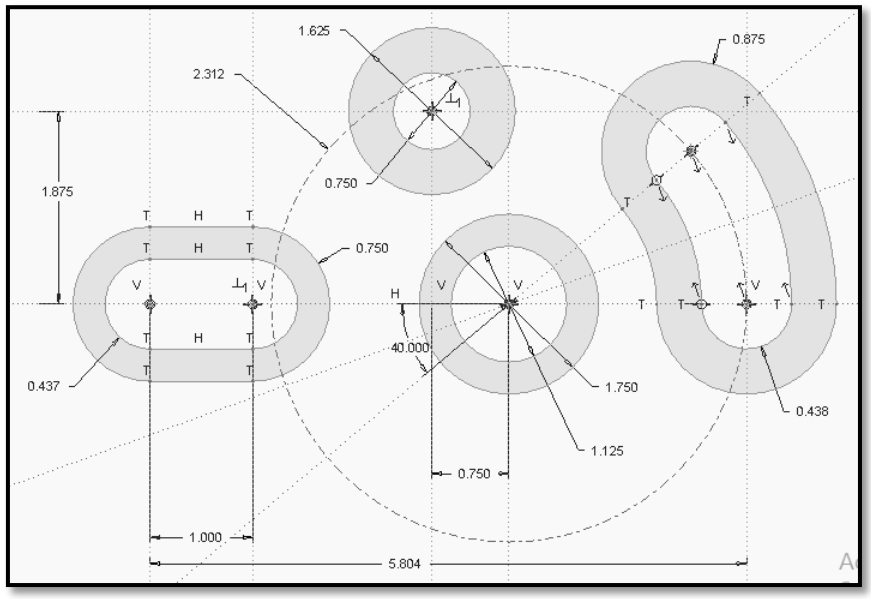
Offset tool.

27. Click on **offset** 

from sketcher tool bar and select entity to be offset.

28. Select 

loop option. Enter off set distance 0.437 for arc racetrack. Accept the entered value. Continue the same procedure for racetrack as shown in Figure.

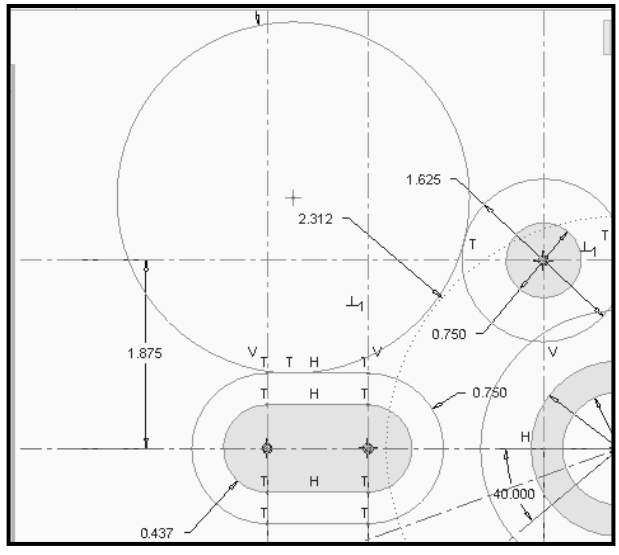


29. To draw the tangent circles. Use  **Tangent** constraint tool from constrain tool bar.

30. First draw circle using  tool. Take care that drawn circle should not be touch to any other entity.
31. Modify a dimension of above circle by double-click on its value using the LMB, and then type a new value R1.750 according to given drawing.

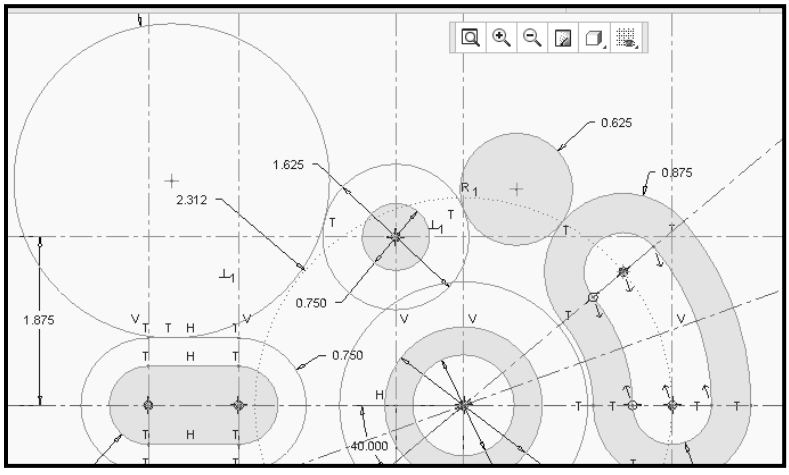
32. To tangent the circleR1.750 and circleØ1.625.Click on  **Tangent** tool from constraint tool bar. Then select the circle R1.750 and circle Ø1.625.

33. To tangent the circleR1.750 and outer racetrack of R 0.750. As the  in selected mode. Select first the circleR1.750 and then select outer racetrack of R 0.750. As shown in Figure.



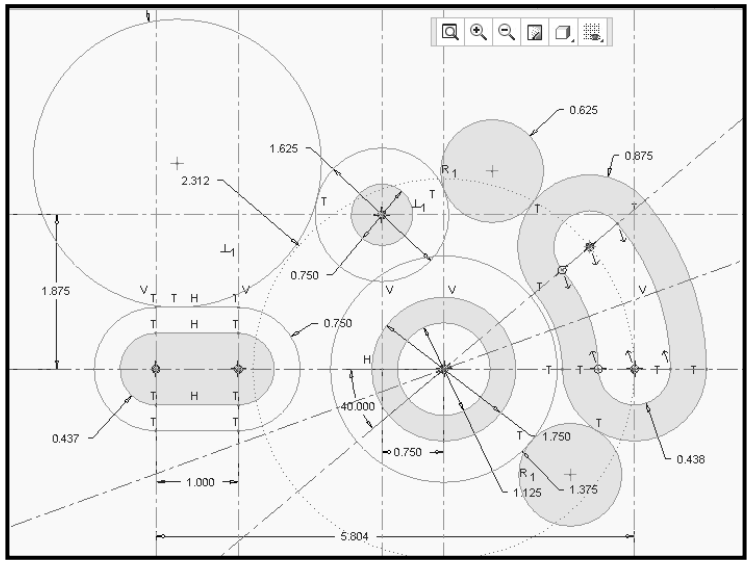
34. Now we have to draw a tangent circle to circle of Ø1.625 and to the outer racetrack of R 0.875 from upper side of the drawing.
35. First draw circle using 

tool. Take care that drawn circle should not touch to any other entity.
36. Modify a dimension of above circle by double-clickon its value using the LMB, and then type a new value R 0.625 according to given drawing.
37. To tangent the circle Ø1.625 and circle R 0. 625.Click on  **Tangent** tool from constraint tool bar. Then select the first circle Ø1.625and then select circle R 0. 625.
38. To tangent the circleR 0.625 and outer racetrack of R 0.875. As the  in selected mode. Select first the circleR 0.625 and then select outer racetrack R 0.875.As shown in Figure.



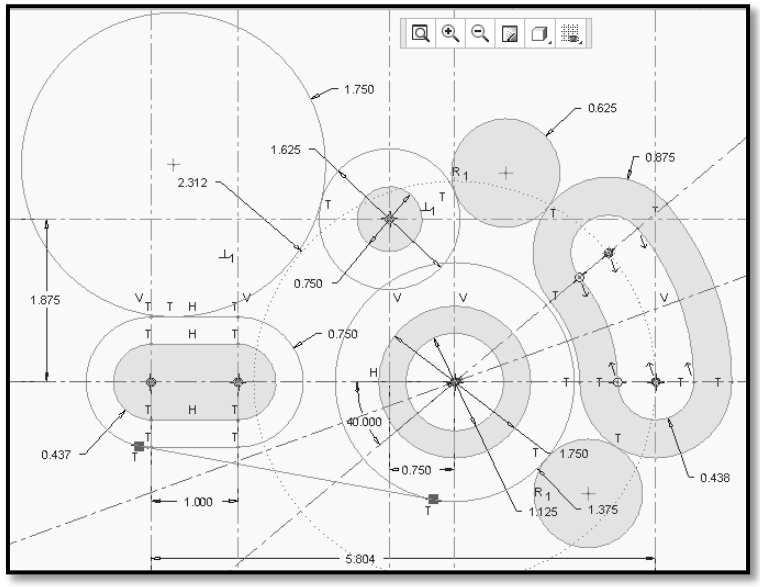
39. Now we have to draw tangent circle to the circle R1.375 and racetrack of R 0.875 from lower side of the sketch.
40. First draw circle using 

tool. Take care that drawn circle should not touch to any other entity.
41. Modify a dimension of above circle by double-click on its value using the LMB, and then type a new value R 0.625according to given drawing.
42. To tangent the circle R1.375 and circle R 0. 625.Click on  **Tangent** tool from constraint tool bar. Then select the first circle R1.375 and then select circle R 0. 625.
43. To tangent the circleR 0.625 and outer racetrack of R 0.875. As the  in selected mode. Select first the circleR 0.625 and then select outer racetrack R 0.875.As shown in Figure.



44. We have to draw the tangent line to circle R1.375 and to the outer racetrack of R 0.750 from lower side of the drawing.
45. To draw tangent lines to the circlesR1.375 andto the outer racetrack of R 0.750. Use

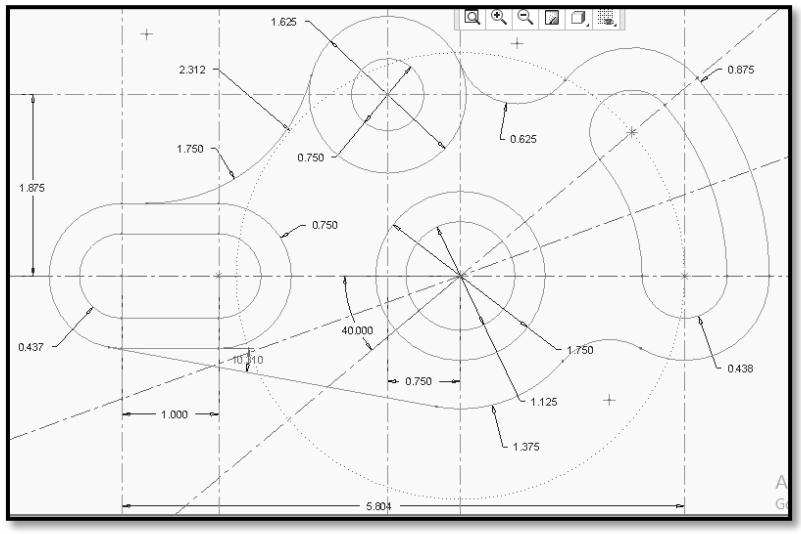
Line Tangentoption. Click**Line >Line Tangent** button, and select the tangent point of first circleR1.375 and then select tangent point of theouter racetrack of R 0.750. As shown in Figure.



46. To trim unwanted lines, use 

option of editing tool bar. Click on the 

Delete Segmentbutton and then choose unwanted lines one by one as shown in Figure.

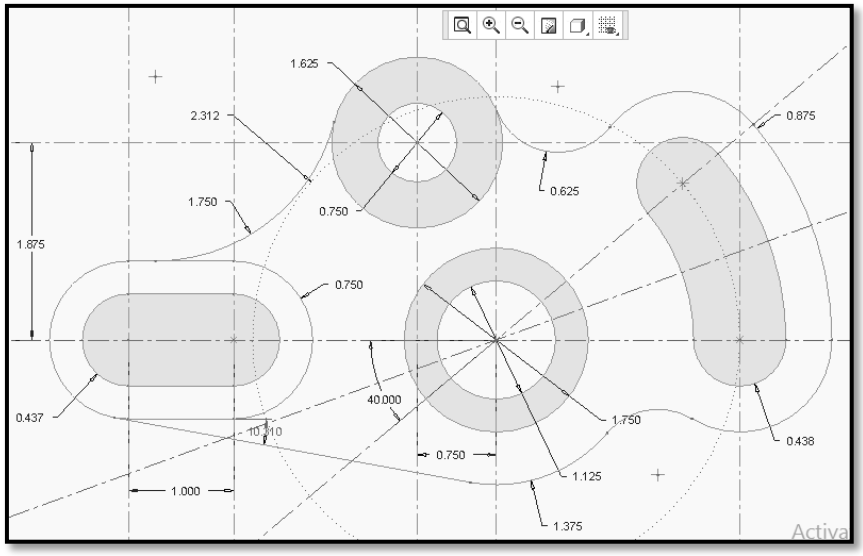


47. To trim unwanted lines, use 

option of editing tool bar. Click on the 

Delete Segmentbutton and then choose unwanted lines one by one as shown in Figure.

48. Use  **Shade Closed Loop** button in active mode to appear shaded closed loop portion as shown in Figure.



E. Finally save your work by clicking Save button which saves sketch by**PACTICAL3_EXERCISE2** with **.sec** file extension.

XII. Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					

3.					

XIII. Actual Procedure Followed

.....
.....
.....
.....

XIV. Precautions Followed

.....
.....
.....
.....

XV. Course proficiency

.....
.....
.....
.....

XVI. Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Describe the Palette feature used in this session.
2. Write advantages and limitations of Palette feature used in this session.

[Space for Answer]

.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

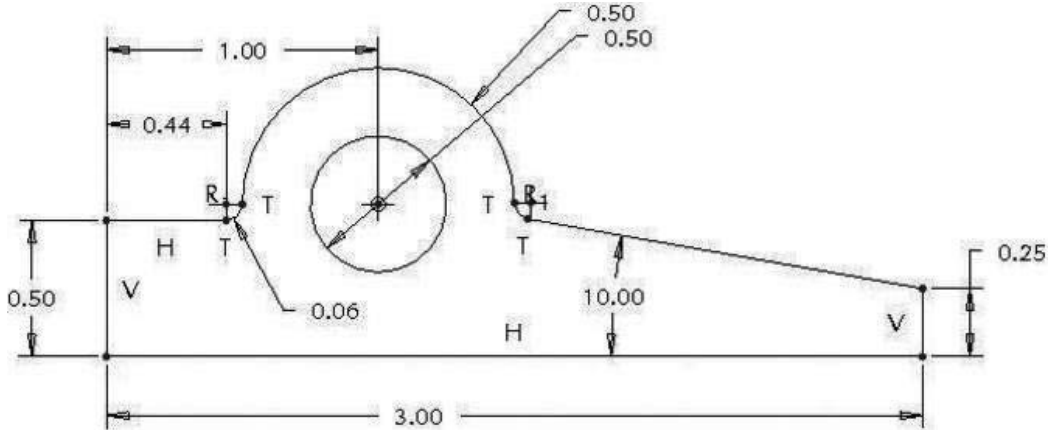
.....

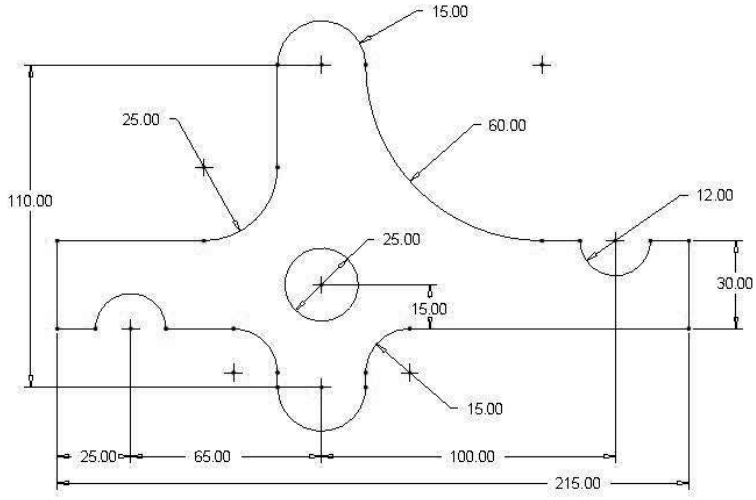
.....

.....

.....

XVI Questions for Practice.





XVII References / Suggestions for Further Reading

- https://www.youtube.com/watch?v=KQk-hn5DZVg
- *https://www.youtube.com/watch?v=lpH4ZUUD9N0*

XVIII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40%**
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60%**
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……
4. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.4: Draw and Print Two Simple 3-D Drawings using 3D Modeling Commands.

I Practical Significance

To create solid models of any mechanical components. To learn different sketching and modeling commands. Also understand datum features and datum plane theory. Study different geometric and modeling constraints. From design engineers point of view, can be seen the object from various directions and in various views. It helps to be sure that the object looks exactly as wanted. It also gives additional vision as to what more changes can be done in the object.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills –

- Geometric constraints – overview, apply equal length.
- Dimension constraints, weak, strong, locked.
- Viewing the model – default, flat sketch view, spin.
- Datum plane - visibility.
- Dashboard interface.

IV Relevant Course Outcome(s)

- Generate 3D models from 2D sketches using Part workbench of any parametric solid modeling software.

V Practical Outcome

- Operate available modeling software to draw 3D Models of any engineering product.

VI Relative Affective Domain-

- Working in team work.
- Practice good housekeeping.
- Demonstrate working as a leader/a team member.
- Follow ethical practices.

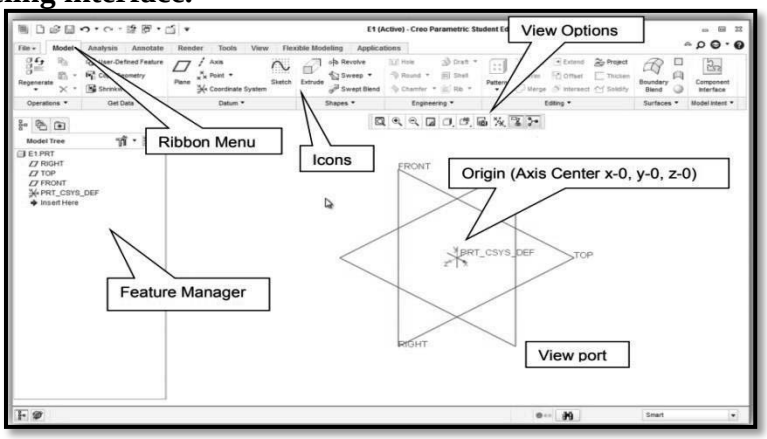
VII Minimum Theoretical Background

- Basic knowledge of reading of 3D objects.
- Knowledge of creating working directory.

Mouse Buttons

- **Left Button -**Most commonly used for *selecting* objects on the screen or sketching.
- **Right Button –**Used for activating pop-up *menu* items, typically used when editing. (Note: you must hold the down button for 2 seconds)
- **Center Button –** (option) Used for model *rotation*, *dimensioning*, zoom when holding Ctrl key, and pan when holding Shift key. It also *cancels* commands and line chains.
- **Center Scroll Wheel –** (option) same as Center Button when depressed, only it activates Zoom feature when scrolling wheel.

Solid modeling interface.



In the Graphics toolbar at the top of the graphics area, disable or enable the display of any datum features whenever you want.

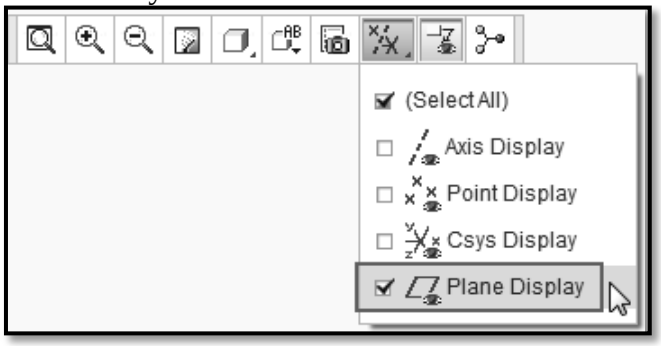
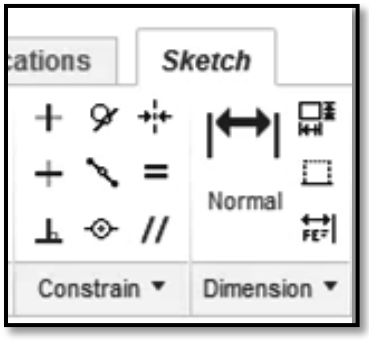


Figure- Datum Features.

Sketch Constraints & Relations:



Constraint	Geometric entities to select	Resulting Constraint
Horizontal or Vertical	One or more lines or two or more points.	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two or more lines.	The items lie on the same infinite line.
Perpendicular	Two lines.	The two items are perpendicular to each other.
Parallel	Two or more lines. A line and a plane (or a planar face) in a 3D sketch.	The items are parallel to each other. The line is parallel to the selected plane.
Tangent	An arc, ellipse, or spline, and a line or arc.	The two items remain tangent.
Concentric	Two or more arcs, or a point and an arc.	The arcs share the same centerpoint.
Midpoint	Two lines or a point and a line.	The point remains at the midpoint of the line.
Coincident	A point and a line, arc, or ellipse.	The point lies on the line, arc, or ellipse.
Equal	Two or more lines or two or more arcs.	The line lengths or radii remain equal.
Symmetric	A centerline and two points, lines, arcs, or ellipses.	The items remain equidistant from the centerline, on a line perpendicular to the centerline.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

IX Experimental setup:

.....

.....

.....

.....

.....

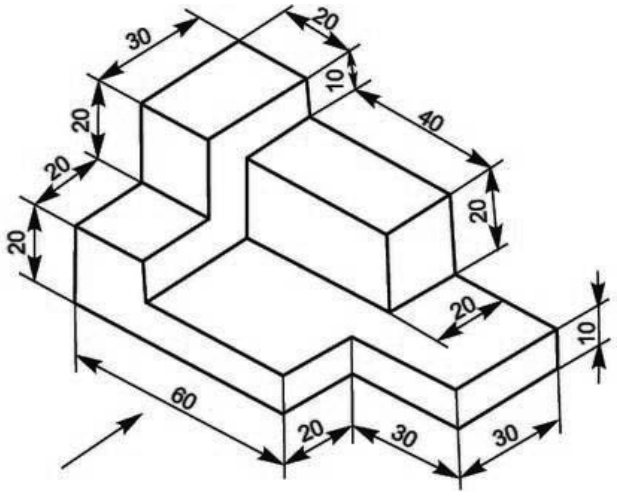
.....

X Precautions to be Followed

1. Student should understand and can draw at least two Orthographic views of any model.
2. While constructing 2D sketch, boundary (Area) of any profile should be enclosed.
3. While specifying dimensions, carefully select the entity or end points of entity and click the middle button (roller) of mouse.
4. Check given drawing for dimensional printing mistakes if any and if dimensions are missing assume proportionate dimensions.
5. While constructing the drawing, periodically save your work.

XI Procedure

Exercise No.1-Createfollowinggiven 3D part using part modeling workbench of CAD software.

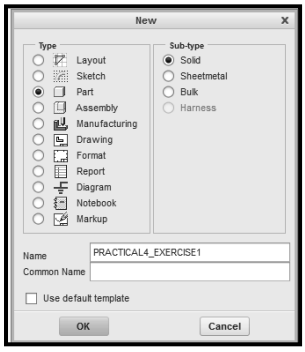


Following steps are required to create given solid 3D part in part modeling workbench.

A. Start Solid modeling.
B. Set the working directory.
C. Start part model environment.
D. Selecting the Sketching Plane for the Base Feature.
E. Creating and Dimensioning the Sketch for the Base Feature.
F. Selecting the Sketching Plane for the Second Feature.
G. Creating and Dimensioning the Sketch for the Second Feature.
H. Selecting the Sketching Plane for the Third Feature.
I. Creating and Dimensioning the Sketch for the Third Feature.
J. Selecting the Sketching Plane for the Forth Feature.
K. Creating and Dimensioning the Sketch for the Forth Feature.
L. Selecting the Sketching Plane for the Fifth Feature.
M. Creating and Dimensioning the Sketch for the Fifth Feature.
N. Save the part and close the file.
O. Print drawing of part.

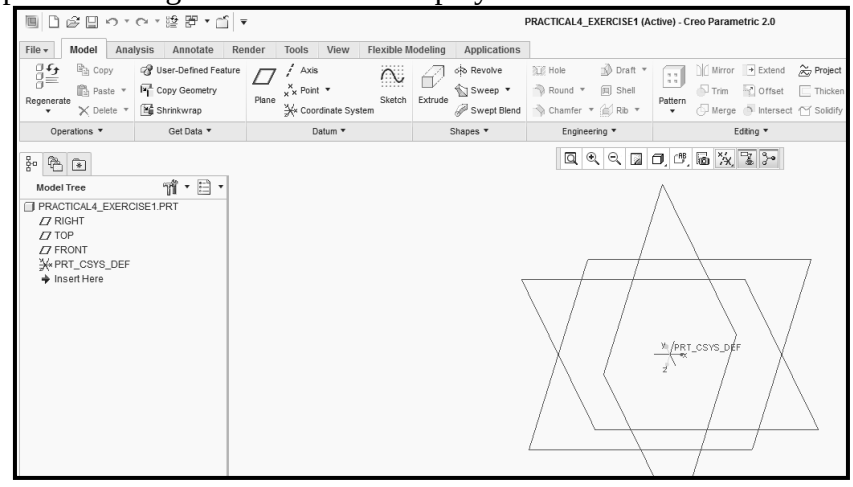
A. Start Solid modeling parametric: As explained in practical No.01.
B. Set Working Directory: As explained in practicalNo.01.
C. Start part model environment-

1. Selecting **File>New**  from the menu or click on the **New File** icon from the main toolbar. A new window will be displayed as shown in Figure by default.
2. Select **Part**  option from the same window to sketch the drawing.Type name as **PRACTICAL4_EXERCISE1**.Click **OK** button.
3. In the New dialog box, notice the default object Type is **Part** and Sub-type is **Solid**, these are the correct options for creating a solid part. Give the suitable Name for the model. **Uncheck** the Use default template and click OK.



4. **New File Box window** will display on the screen.
5. Then from **New File** options dialog box select **solid** part**mmns_part_solid** or **solid_part_mmks**and click OK.

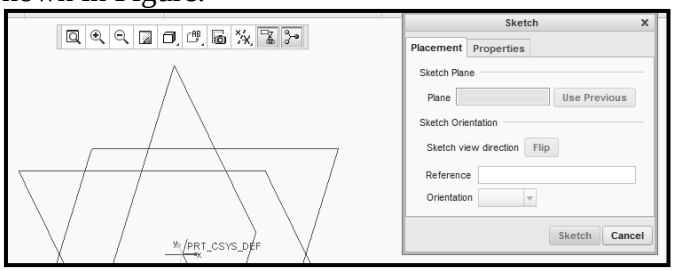
6 The part modeling workbench will display on the screen as shown in Figure.



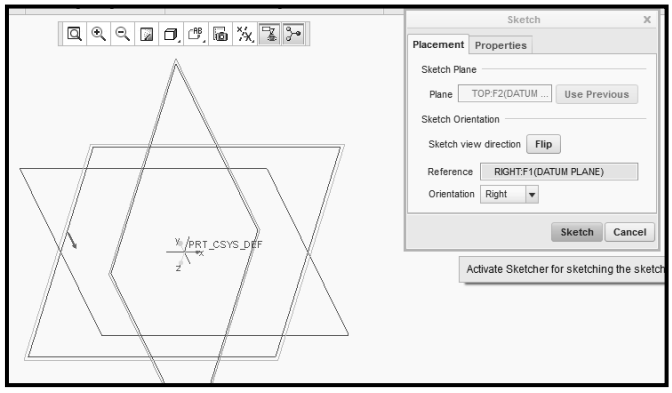
D. Selecting the Sketching Plane for the Base Feature:

1 Now we have to create first base feature of the given part. So we need to draw a sketch of first feature on any one of the default plane.

2 Click on  **Sketch** button from part modeling tool bar. Then **Sketch** window will appear as shown in Figure.



3 Select the correct plane as per the part orientation by just clicking on the default plane. Click on **Sketch** button of the sketch window.

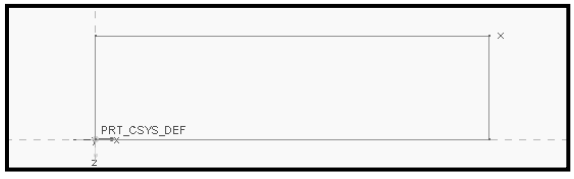


4 Click on  **Orient View** button to orient the view parallel to screen.

E. Creating and Dimensioning the Sketch for the Base Feature:

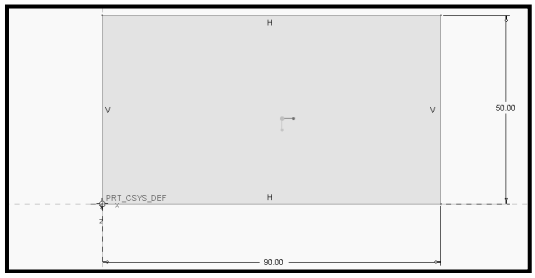
1 Select 

from Sketching tool bar to draw rectangle. Giving first corner of rectangle as the origin and second point as shown in Figure. Click MMB to exit.



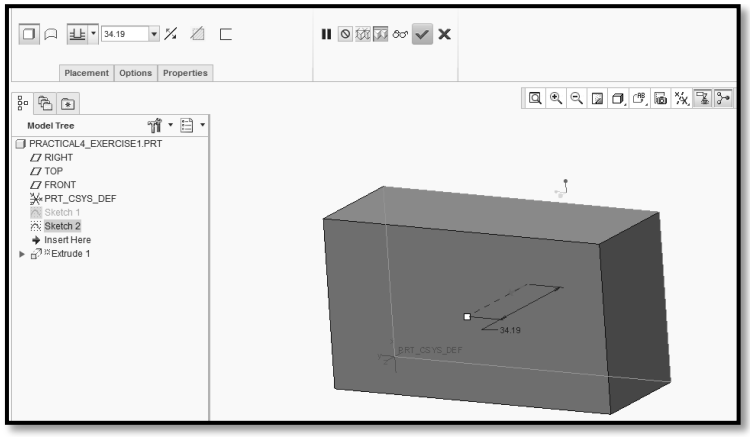
2 To modify a dimension double-clickon its value using the LMB, and then type a new value 90 X 50 mm as per the drawing. 

Accept it by clicking on green colored Check mark.



3 Select 

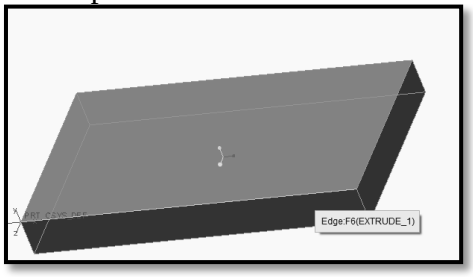
button of part modeling tool bar.



4 Mention the height of 

10 mm,Click on  to change depth direction of extrude. Click on  for preview, and finally click on  accept button if extrude correct.

5 Finally first feature of the part will look like as shown in Figure.



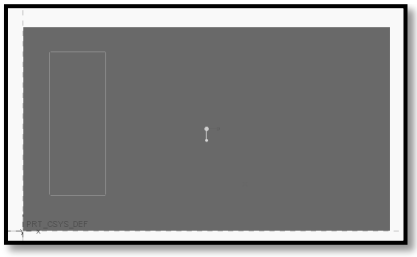
F. Selecting the Sketching Plane for the Second Feature:

1 Click on  **Sketch** button from part modeling tool bar. Then **Sketch** window will appear as shown in Figure.
2 Take the cursor on the top plane, and click on it.
3 Click on **Sketch** of the sketch window.
4 Click on  **Orient View** button to orient the view parallel to screen. Then top plane of the first feature will ready for sketching of the second feature.

G. Creating and Dimensioning the Sketch for the Second Feature.

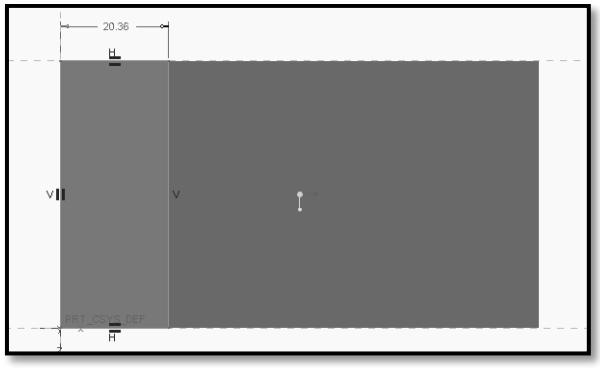
1 Select 

from Sketching tool bar to draw rectangle.

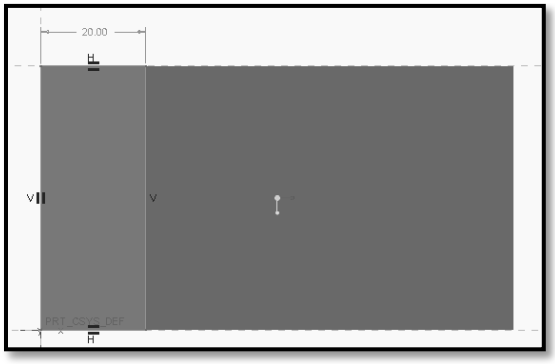


2 Use  constraint tool to coincident three sides of the above rectangle.

3 Select  and then select line and edge of the first features as shown in Figure.

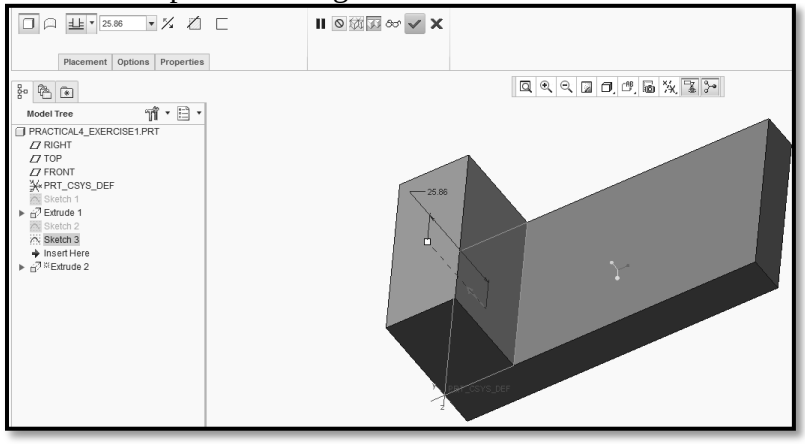


4 To modify a dimension double-click on its value using the LMB, and then type a new value 20 mm as per the drawing. Accept it by clicking on green colored **Check** mark.



5 Select 

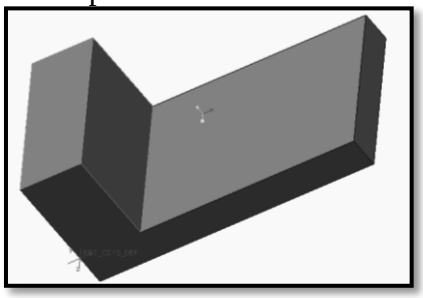
button of part modeling tool bar.



6 Mention the height of 

10 mm, Click on  to change depth direction of extrude. Click on  for preview, and finally click on  **Accept** button if extrude correct.

7 Finally first feature of the part will look like as shown in Figure.



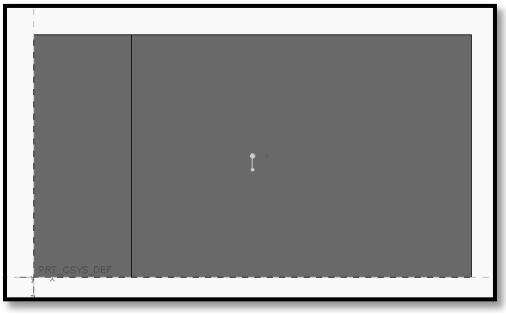
H. Selecting the Sketching Plane for the Third Feature:

1 Click on  sketch button from part modeling tool bar. Then **Sketch** window will appear.

2 Take the cursor on the appropriate top plane according to given part, and click on it.

3 Click on **Sketch** of the sketch window.

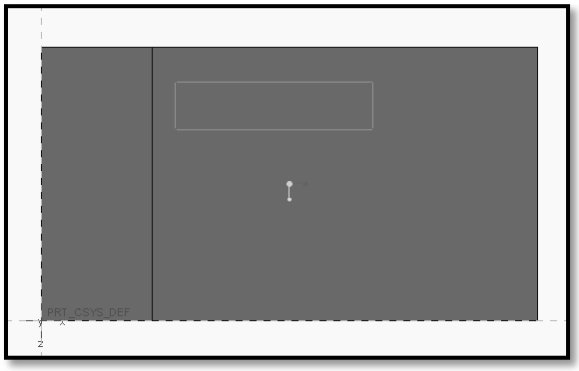
4 Click on  Orient View button to orient the view parallel to screen. Then top plane of the first feature will ready for sketching of the third feature as shown in Figure.



I. Creating and Dimensioning the Sketch for the Third Feature.

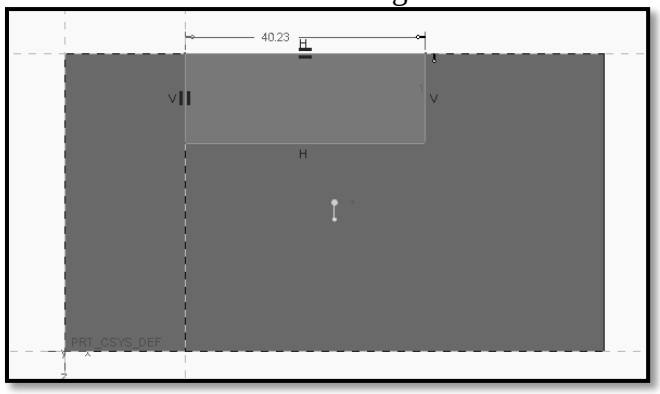
1 Select 

from Sketching tool bar to draw rectangle.

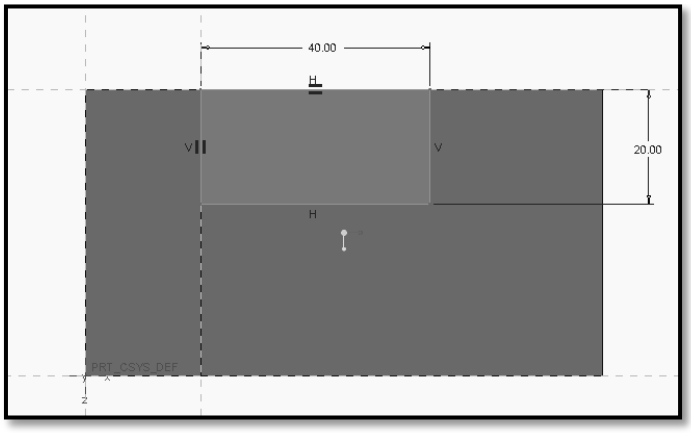


2 Use  constraint tool to coincident two sides of the above rectangle.

3 Select  and then select line and edge first features as shown in Figure.

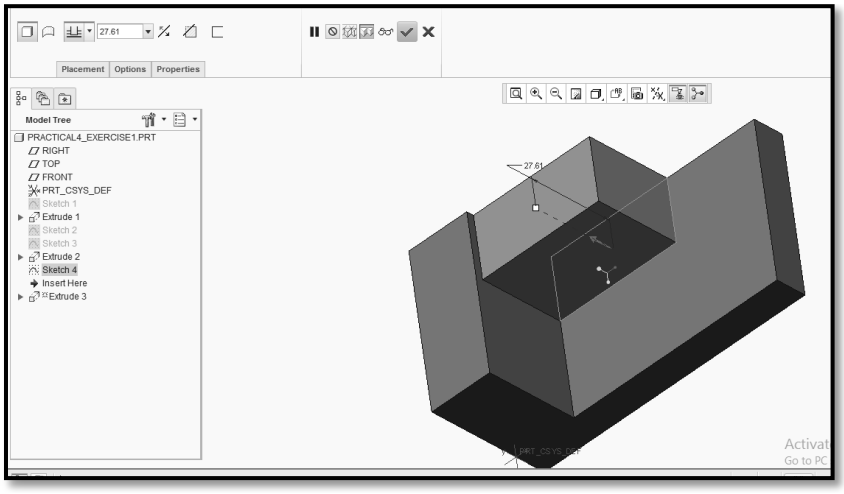


4 To modify a dimension double-clickon its value using the LMB, and then type a new value 40 X 20 mm as per the drawing. Accept it by clicking on green colored **Check** mark.



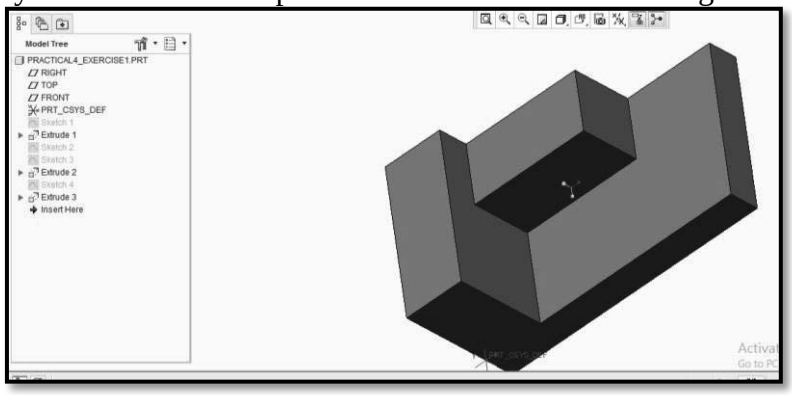
5 Select 

button of part modeling tool bar.



6 Mentionthe height of  20 mm , Click on  to change depth direction of extrude. Click on  for preview, and finally click on  **Accept** button if extrude correct.

7 Finally first feature of the part will look like as shown in Figure.



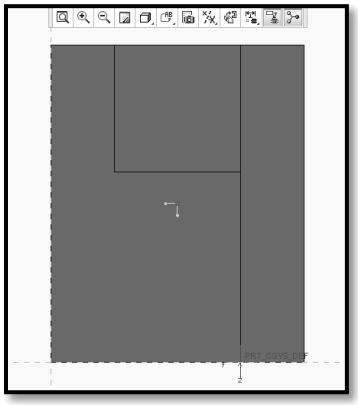
J. Selecting the Sketching Plane for the Forth Feature:

1 Click on  **Sketch** button from part modeling tool bar. Then **Sketch** window will appear.

2 Take the cursor on the appropriate plane according to given part, and click on it.

3 Click on **Sketch** of the sketch window.

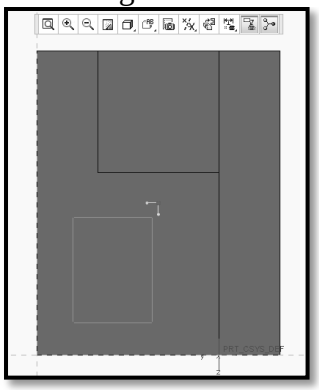
4 Click on  **Orient View** button to orient the view parallel to screen. Then plane of the second feature will ready for sketching of the forth feature as shown in Figure.



K. Creating and Dimensioning the Sketch for the Forth Feature:

1 Select 

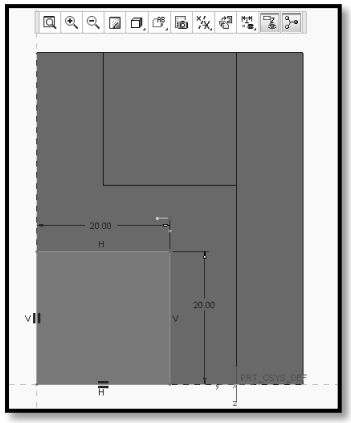
from Sketching tool bar to draw rectangle.



2 Use  **Constraint** tool to coincident two sides of the above rectangle.

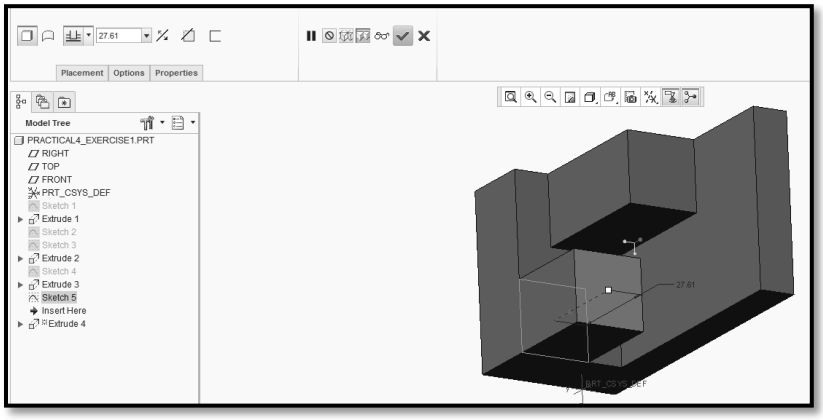
3 Select  and then select line and edge second features as shown in Figure.

4 To modify a dimension double-clickon its value using the LMB, and then type a new value 20 X 20 mm as per the drawing. Accept it by clicking on green check mark.



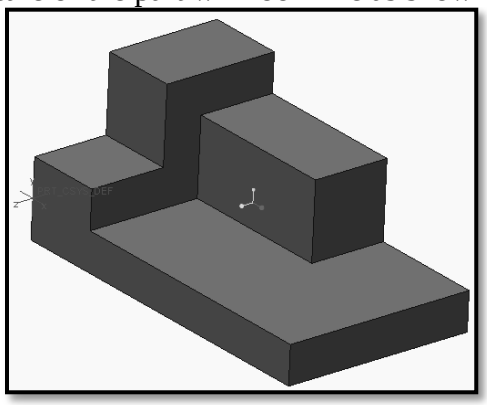
5 Select 

button of part modeling tool bar.



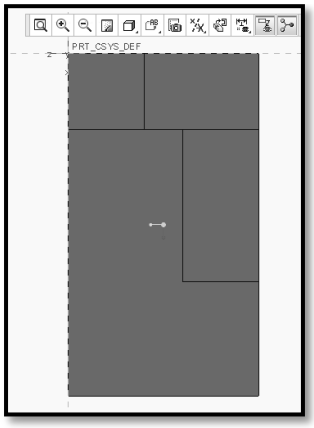
6 Mention the height of  20 mm, Click on  remove material. Click on  to change depth direction of extrude. Click on  for preview, and finally click on  accept button if extrude correct.

7 Finally forth feature of the part will look like as shown in Figure.



L. Selecting the Sketching Plane for the Fifth Feature:

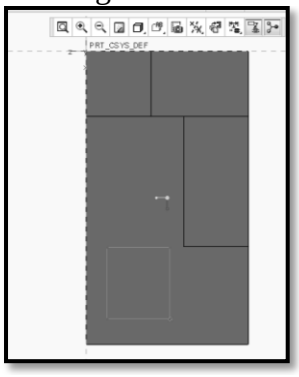
1 Click on  **Sketch** button from part modeling tool bar. Then **Sketch** window will appear.
2 Take the cursor on the first feature top plane according to given part, and click on it.
3 Click on **Sketch** of the sketch window.
4 Click on  **Orient View** button to orient the view parallel to screen. Then plane of the first feature will ready for sketching of the fifth feature as shown in Figure.



M. Creating and Dimensioning the Sketch for the Fifth Feature:

1 Select 

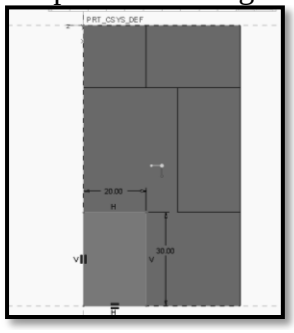
from sketching tool bar to draw rectangle.



2 Use  **Constraint** tool to coincident two sides of the above rectangle.

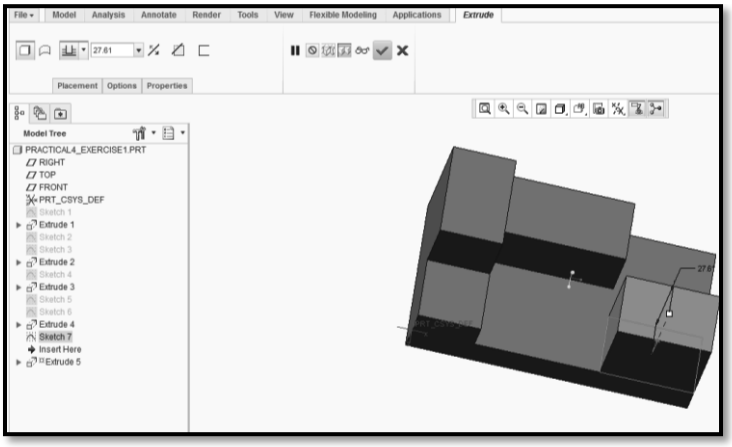
3 Select  and then select line and edge second features as shown in Figure.

4 To modify a dimension double-clickon its value using the LMB, and then type a new value 20 X 30 mm as per the drawing. Accept it by clicking on green



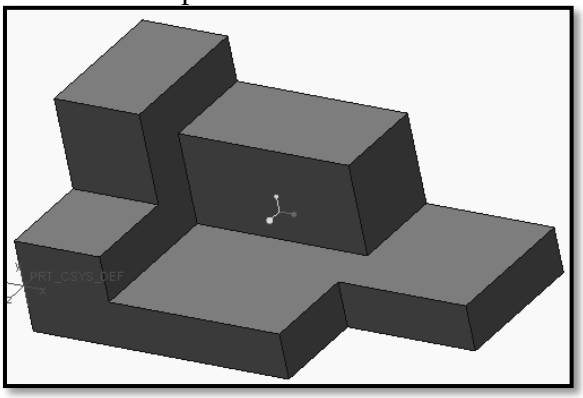
5 Select 

button of part modeling tool bar.



6 Mention the height of  10 mm, Click on  remove material. Click on  to change depth direction of extrude.Click on  for preview, and finally click on  accept button if extrude correct.

7 Finally fifth feature of the part will look like as shown in Figure.

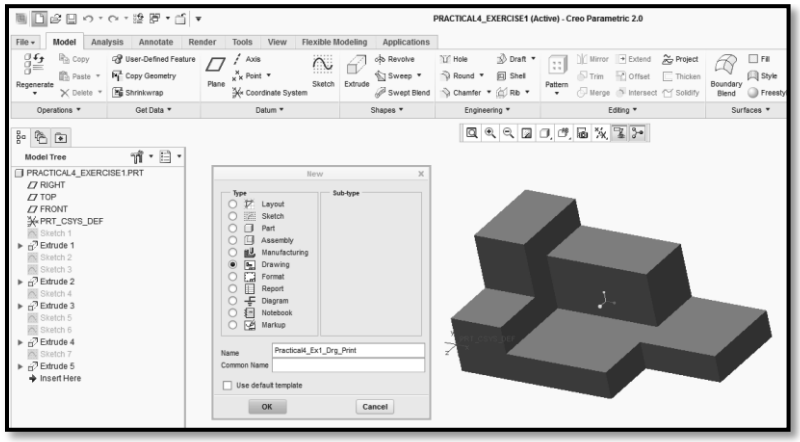


N. Save the part and close the file.

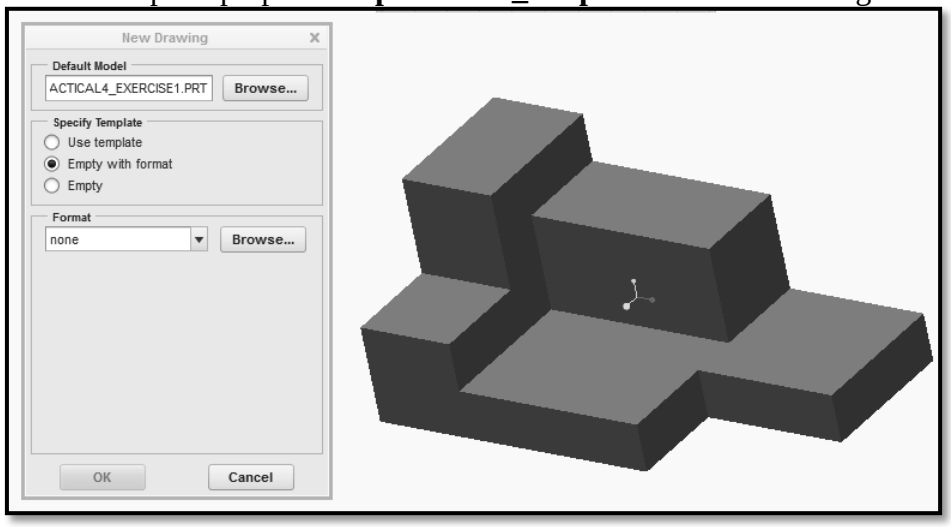
O. Print drawing of the part.

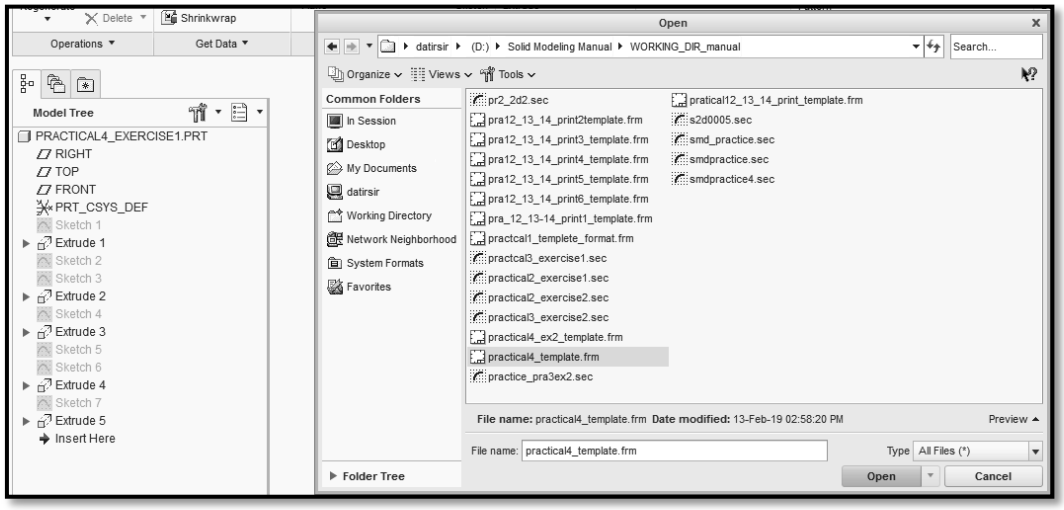
1 Open part model to be print.

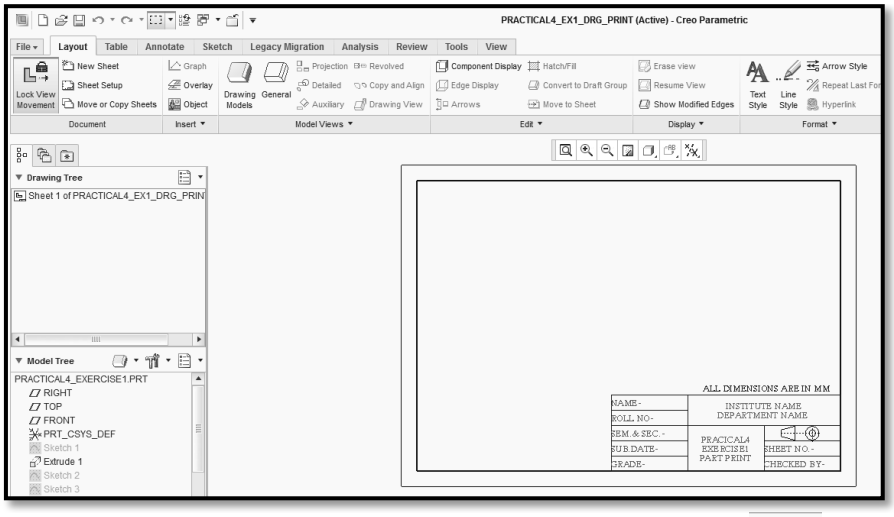
2 Start new drawing file. Click **File> Drawing .**Uncheck on **Use default template.** Click **OK**as shown in Figure.



3 New definition dialog box will display, choose **Empty with format**option. Browse format of the template prepared as **practical4_template**from the working directory.







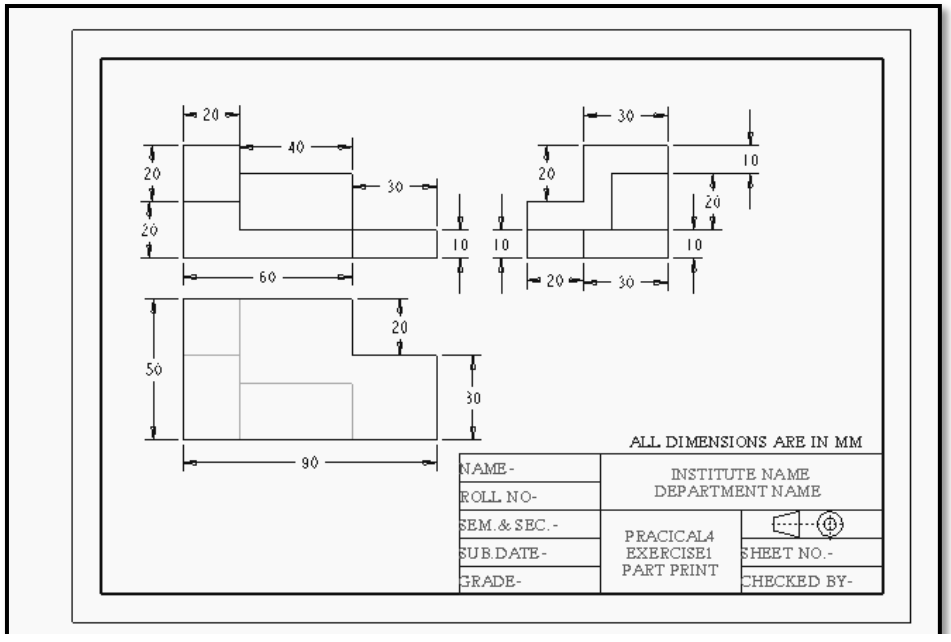
4 Select **Layout** option from drawing tool bar. Choose **General** 

.Select **Default views and OK.**

6 Move the cursor in A4 size sheet, and double click. Choose the views according to First Angle Method and locate it correct position.

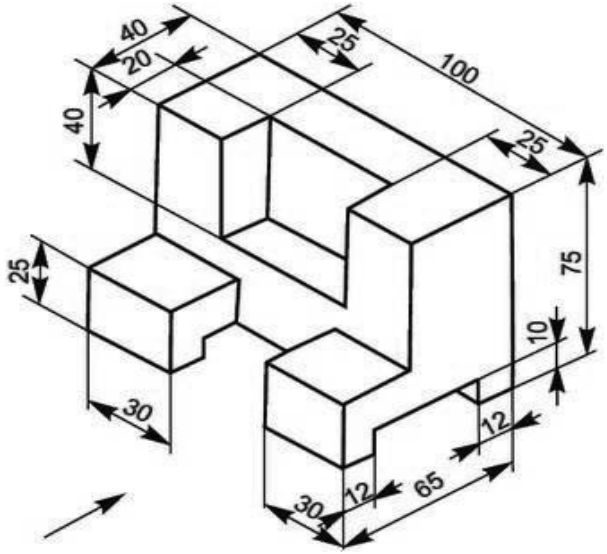
7 Click on **Annotate** option from drawing tool bar and give the dimensions as shown in Figure.

8 Finally the front ,top and side view as per the First angle Method will seen as shown in Figure with dimension.



9 Click Save button to save the drawing in working directory.

Exercise No.1- Create followinggiven 3D part using part modeling workbench of CAD software.



Following steps are required to create given solid 3D part in part modeling workbench.

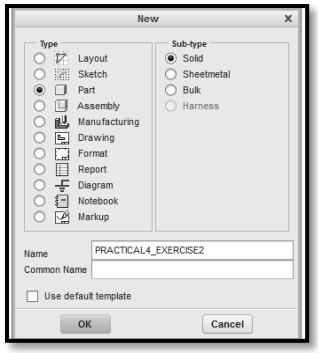
A. Start Solid modeling.
B. Set the working directory.
C. Start part model environment.
D. Selecting the Sketching Plane for the Base Feature.
E. Creating and Dimensioning the Sketch for the Base Feature.
F. Selecting the Sketching Plane for the Second Feature.
G. Creating and Dimensioning the Sketch for the Second Feature.
H. Selecting the Sketching Plane for the Third Feature.
I. Creating and Dimensioning the Sketch for the Third Feature.
J. Save the part and close the file.
K. Printing of part drawing.

A. Start Solid modeling parametric: As explained in practical No.01.

B. Set Working Directory: As explained in practical No.01.

C. Start part model environment: As explained in practical No.04 Exercise1.

1 Type name as **PRACTICAL4_EXERCISE2**.Click **OK** button as shown in figure.



2 **New File Box window** will display on the screen.

3 Then from **New File** options dialog box select ‘solid_part_mmks’ and click OK.

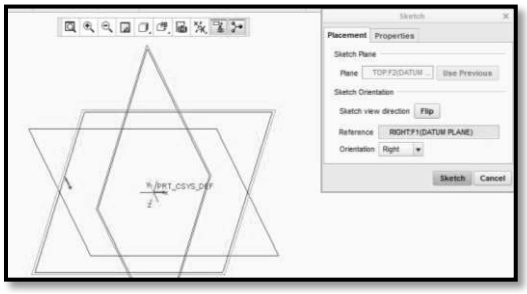
4 Part modeling workbench will display on the screen.

D. Selecting the Sketching Plane for the Base Feature:

5 Now we have to create first base feature of the given part. So we need to draw a sketch of first feature on any one of the default plane.

6 Click on  **Sketch** button from part modeling tool bar. Then **Sketch** window will appear as shown in Figure.

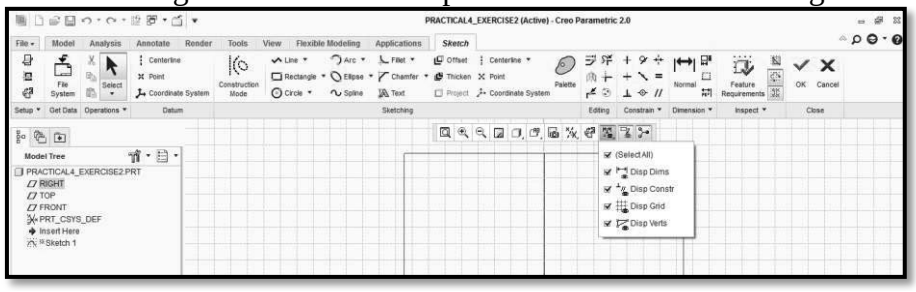
7 Select the correct plane as per the part orientation by just clicking on the default plane. Click on **Sketch** button



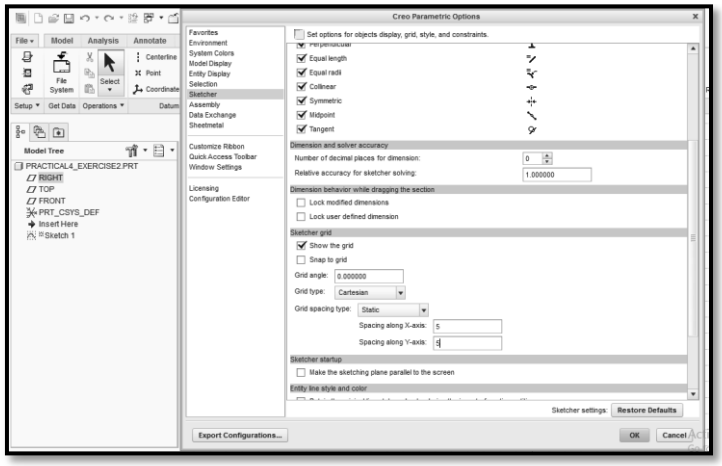
8 Click on  Orient View button to orient the view parallel to screen.

E. Creating and Dimensioning the Sketch for the Base Feature.

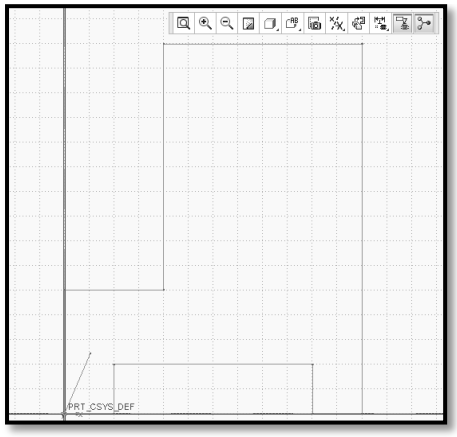
1 Activate the grid from the View option tool bar as shown in Figure.



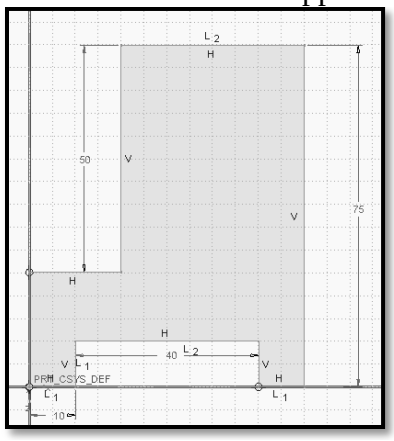
2 **Click File>Options>Sketcher.** Set the grid spacing by 5 mm as shown in Figure. Click OK button.



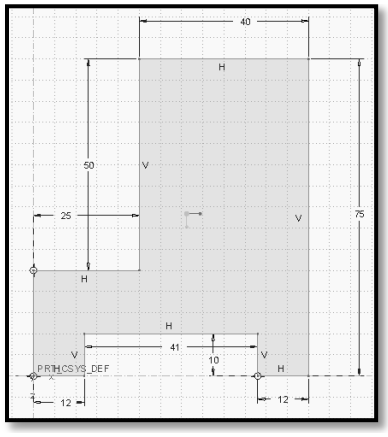
6 Select  from sketching tool bar to draw line close loop profile as shown in figure.



7. Click MMB to exit. Weak dimensions will appear on the drawing.

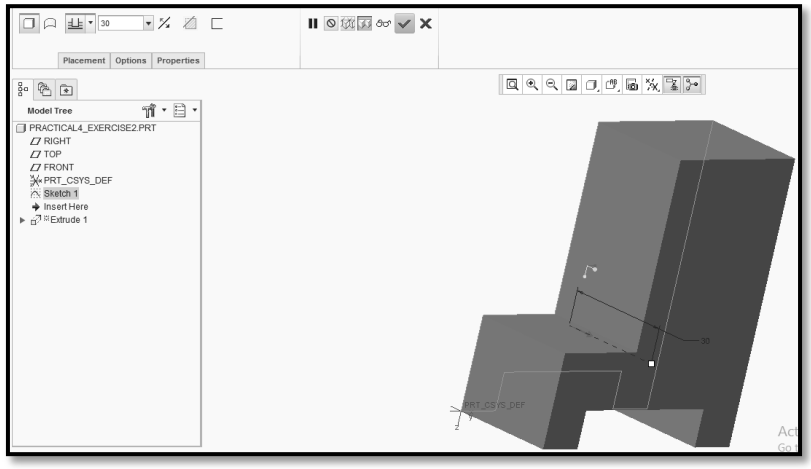


F. To modify a dimension double-clickon its value using the LMB, and then type a new value as per the drawing. Accept it by clicking on green colored Check mark.



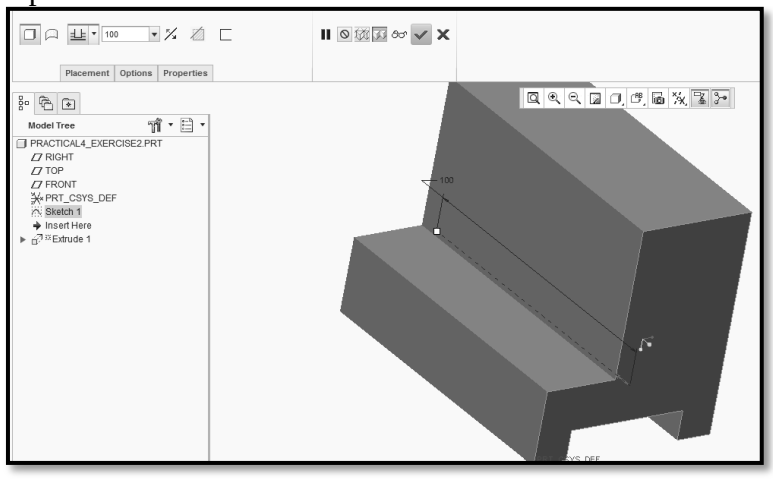
7 Select 

button of part modeling tool bar.

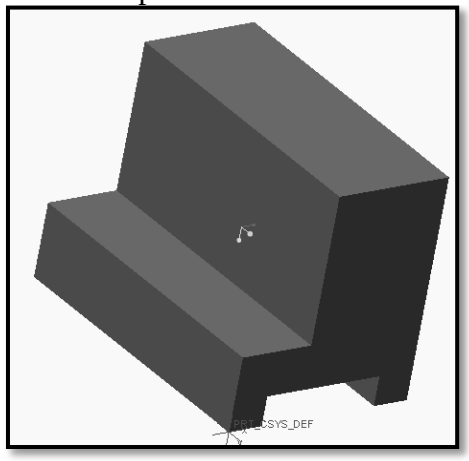


8 Mention the height of 

100 mm, Click on  to change depth direction of extrude. Click on  for preview, and finally click on  accept button if extrude correct.



9 Finally base feature of the part will look like as shown in Figure.



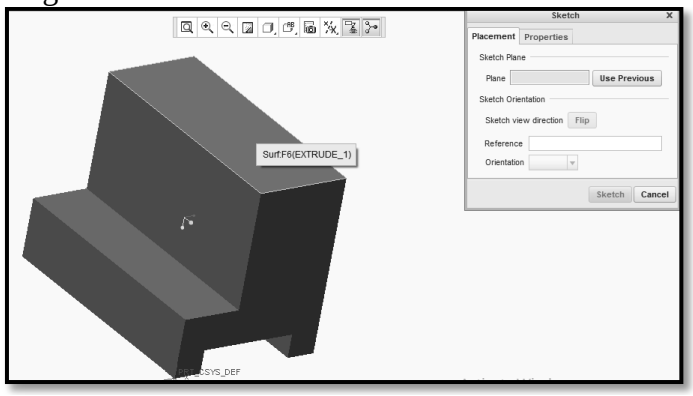
G. Selecting the Sketching Plane for the second Feature:

5 Click on  **Sketch** button from part modeling tool bar. Then **Sketch** window will appear.

6 Take the cursor on the appropriate plane according to given part, and click on it.

7 Click on **Sketch** of the sketch window.

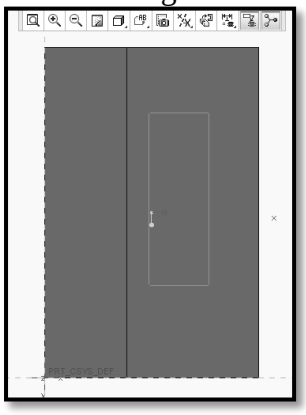
8 Click on  **Orient View** button to orient the view parallel to screen. Then plane of the second feature will ready for sketching of the second feature as shown in Figure.



H. Creating and Dimensioning the Sketch for the second Feature.

2 Select 

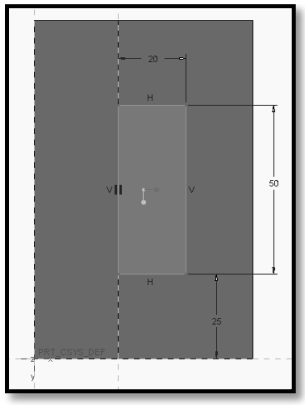
from Sketching tool bar to draw rectangle.



6 Use  constraint tool to coincident two sides of the above rectangle.

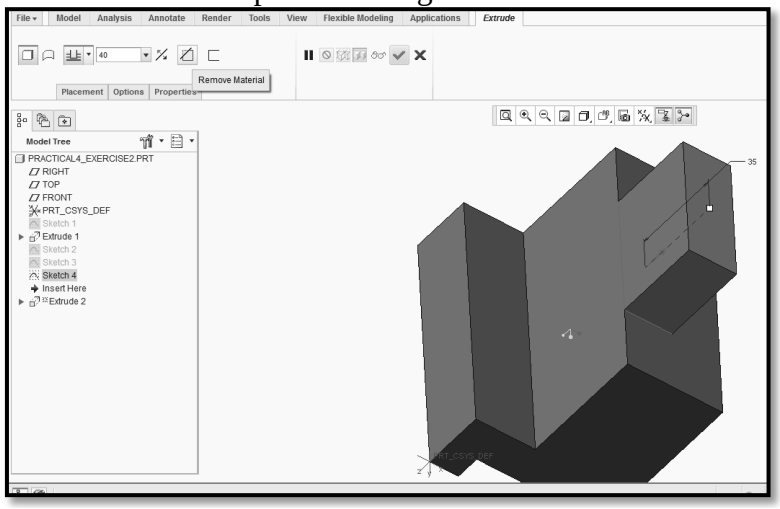
7 Select  and then select line and edge of the first features as shown in Figure.

8 To modify a dimension double-clickon its value using the LMB, and then type a new value 20 X 50 mm as per the drawing. Accept it by clicking on green check mark.



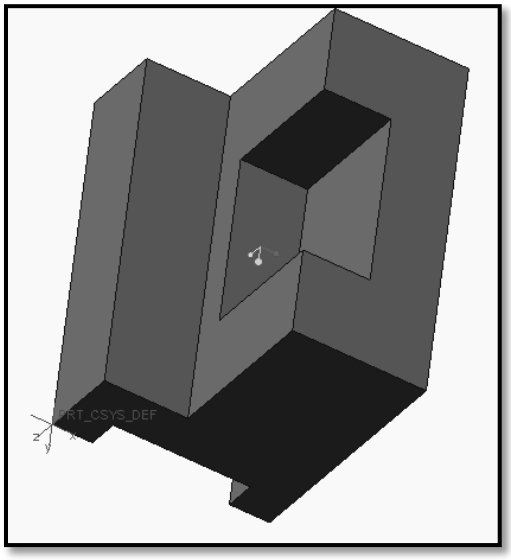
9 Select 

button of part modeling tool bar.



8 Mention the height of 

40 mm, Click on  remove material. Click on  to change depth direction of extrude. Click on  for preview, and finally click on  accept button if extrude correct. You will get second feature of the part as shown in Figure.

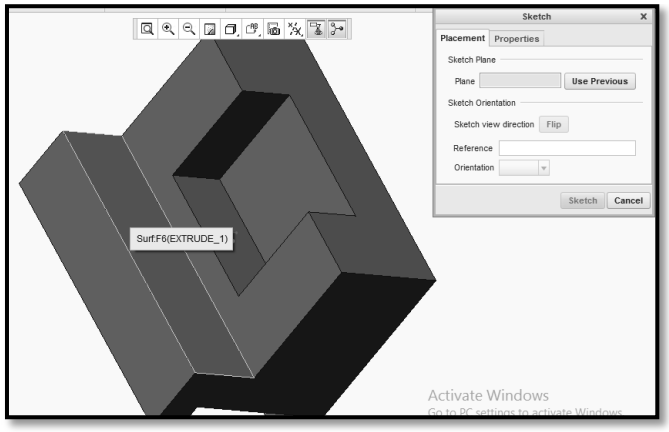


I. Selecting the Sketching Plane for the Third Feature.

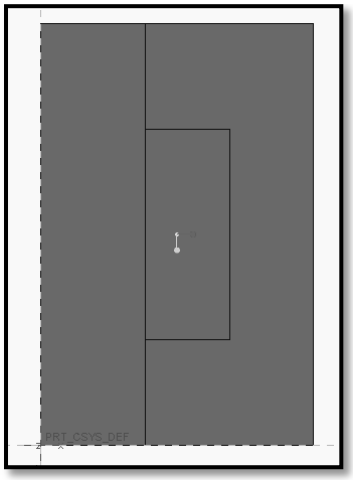
10 Click on  **Sketch** button from part modeling tool bar. Then **Sketch** window will appear.

11 Take the cursor on the first feature top plane according to given part, and click on it.

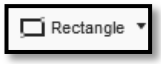
12 Click on **Sketch** of the sketch window.



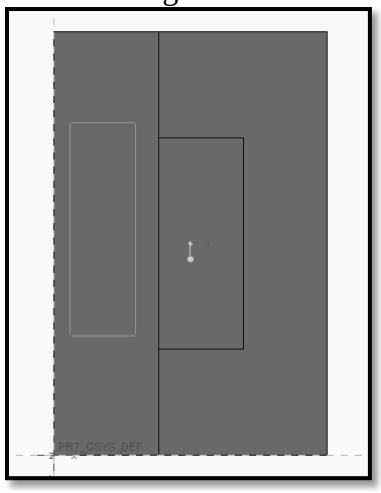
1 Click on  Orient View button to orient the view parallel to screen. Then plane of the first feature will ready for sketching of the third feature as shown in Figure.



J. Creating and Dimensioning the Sketch for the Third Feature.

1 Select 

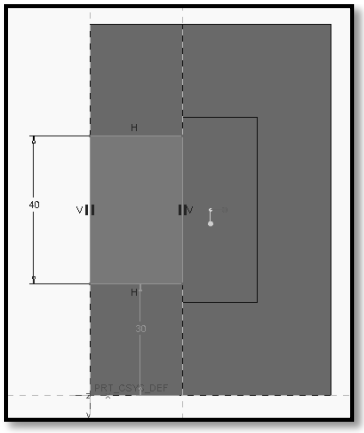
from Sketching tool bar to draw rectangle.



2 Use  constraint tool to coincident two sides of the above rectangle.

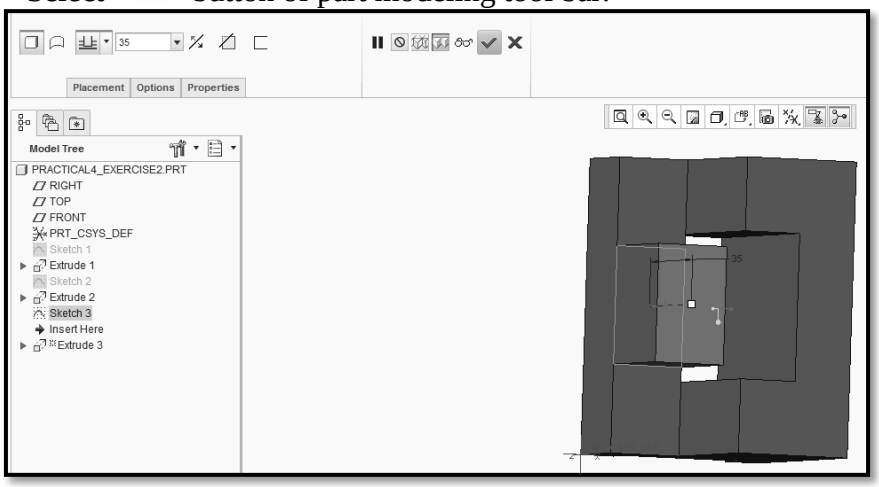
3 Select  and then select line and edge first features as shown in Figure.

4 To modify a dimension double-click on its value using the LMB, and then type a new value 40 X 25 mm as per the drawing. Accept it by clicking on green



2 Select 

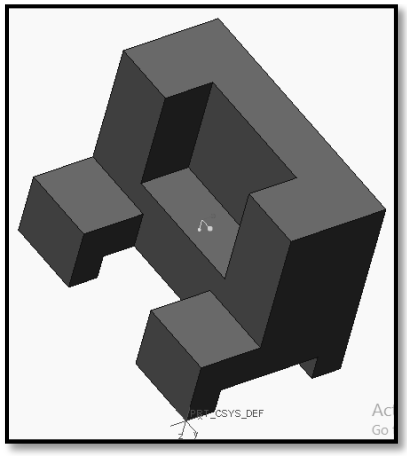
button of part modeling tool bar.



8 Mention the height of 

25 mm, Click on  remove material. Click on  to change depth direction of extrude.Click on  for preview, and finally click on  accept button if extrude correct.

9 Finally third feature of the part will look like as shown in Figure.



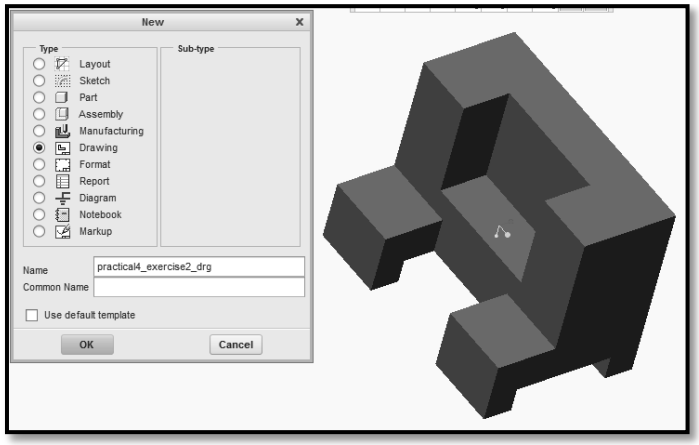
5. Save the part and close the file.

K. Print drawing of the part:

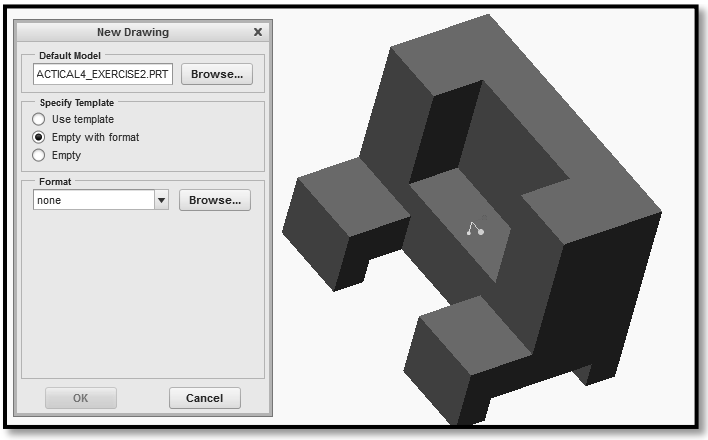
1 Open part model to be print.

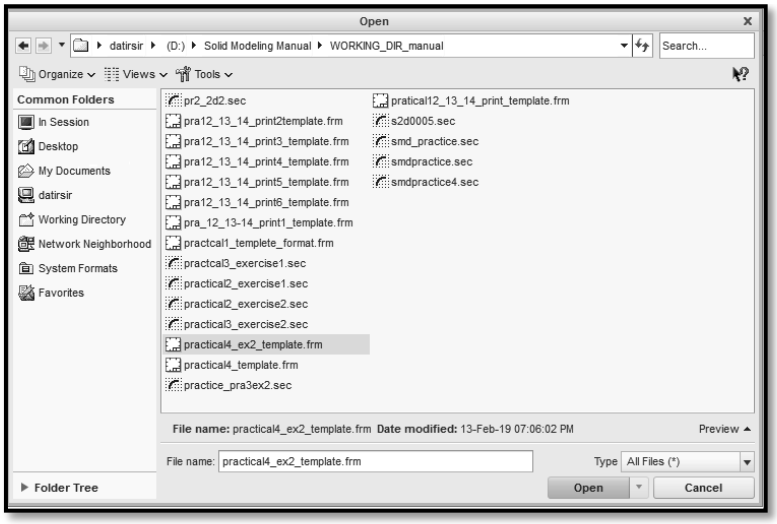
2 Start new drawing file. Click **File> Drawing .**Uncheck on **Use default template.**

3 Type the name **practical4_exercise2.**Click **OK** as shown in Figure.

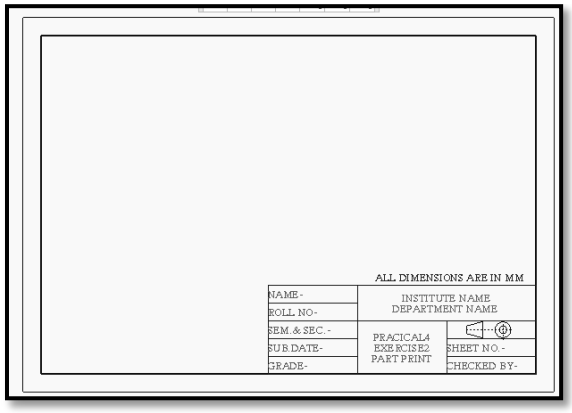


4 New definition dialog box will display, choose **Empty with format**option. **Browse** format of the template prepared as **practical4_ex2_template**from the working directory.



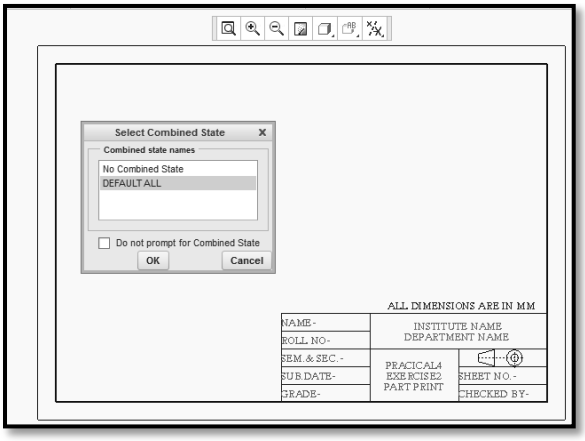


5 A4 size template will available for part drawing as shown in Figure.



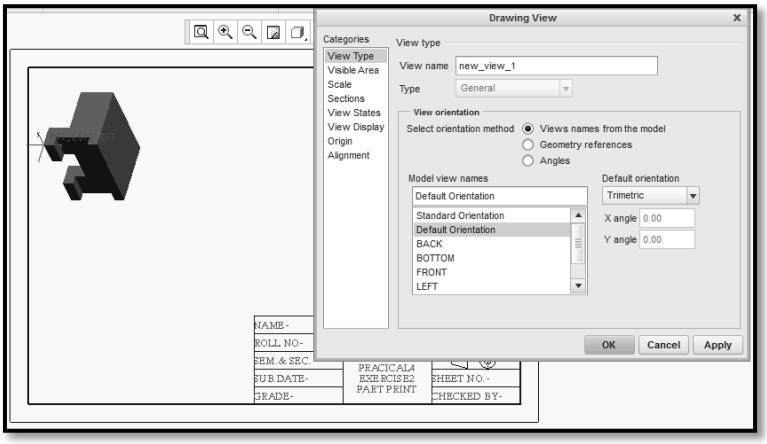
5 Select **Layout** option from drawing tool bar. Choose **General** 

.Select **Default views**andclick **OK.**

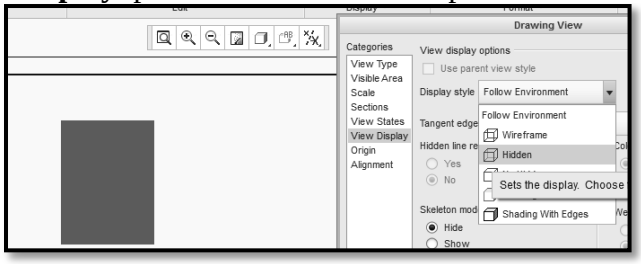


1 Move the cursor in A4 size sheet, and double click. Choose the views according to First Angle Method and locate it correct position.

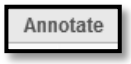
2 Choose **FRONT** from **Model view names** window. Select **Apply> Close**



1. Select**View Display**option. **Choose Hidden** option.

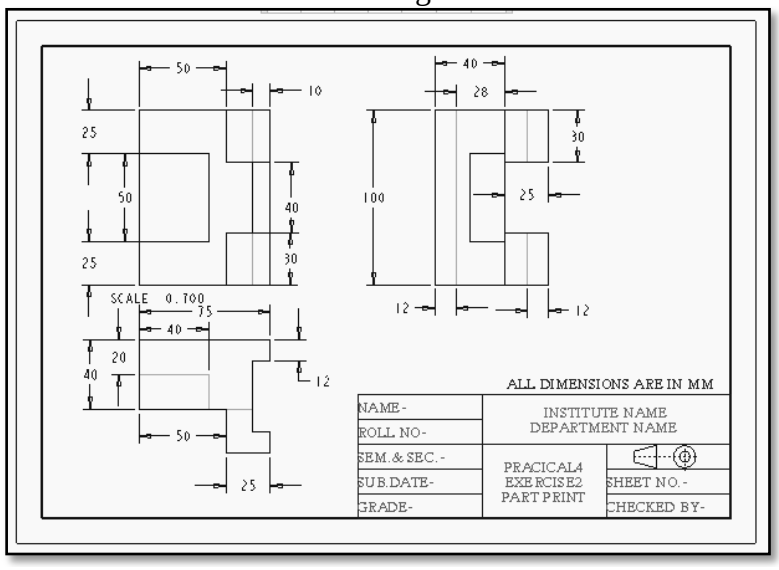


2. Select 

button from drawing tool bar.
3. Repeat the same procedure for next view according to First Angle of Projection as shown in Figure.
4. Click on 

Annotate option from drawing tool bar. Choose 

dimensions button and give the dimensions as shown in Figure.



5. Finally the front ,top and side view as per the First angle Method will seen as shown in Figure with dimension.
6. Click **Save** button to save the drawing in working directory.

XII Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XIII Actual Procedure Followed

.....
.....
.....
.....

XIV Precautions Followed

.....
.....
.....
.....

.....

.....

XV Course proficiency

.....

.....

.....

.....

.....

.....

XVI Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

Questions:

1. Explain application of solid modeling in industry.
2. List different types of tool bar used in part modeling.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XVII References / Suggestions for Further Reading

- https://www.youtube.com/watch?v=ZUPRT95V4Ek
- https://www.youtube.com/watch?v=OPmjH5QfiRs
- https://www.youtube.com/watch?v=O78bWdxjqWs

XVIII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.5: Develop Solid Models Of Individual Components Of Bench Vice / Drill Jig / Screw Jack / Tool Post Assembly Consisting Of At Least Five Parts.

I Practical Significance

To create solid models of any mechanical components. To learn different sketching and modeling commands. Also understand datum features and datum plane theory. Study different geometric and modeling constraints. From design engineers point of view, can be seen the object from various directions and in various views. It helps to be sure that the object looks exactly as wanted. It also gives additional vision as to what more changes can be done in the object.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills –

- Starting and creating new parts.
- Controlling the display of datum features datum plane theory.
- Datum plane - visibility.
- Dashboard interface.
- Revolve Protrusion.

IV Relevant Course Outcome(s)

- Generate 3D models from 2D sketches using Part workbench of any parametric Modeling software.

V Practical Outcome

- Operate available modeling software to draw 3D Models of any engineering product.

VI Relative Affective Domain-

- Working in team work.
- Practice good housekeeping.
- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Basic knowledge of reading of 3D objects.
- Knowledge of creating working directory.
- Basic knowledge of preparation of 2D sketches.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

IX Precautions to be Followed

1. Student should understand and can visualize at least two Orthographic views of any model.
2. Check given drawing for dimensional printing mistakes if any and if dimensions are missing assume proportionate dimensions.
3. While using Sweep or Helical sweep command, need to draw reference axis, reference line, trajectory and helical sweep section. These activity should do carefully.
4. While constructing the drawing, periodically save your work.

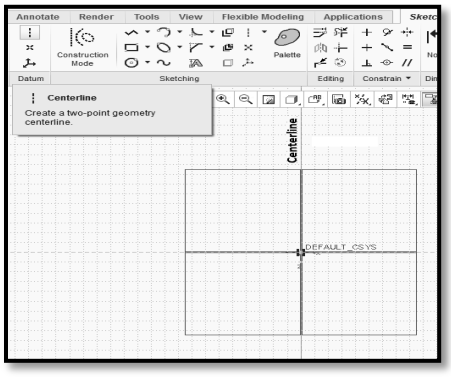
X Procedure

For Setting the Working Directory and Selection of Sketching Plane (Step 1 & Step 2): Follow the steps as explained in Previous Practical's.

Step 3: Create a sketch to define the shape of Body of the Screw Jack.

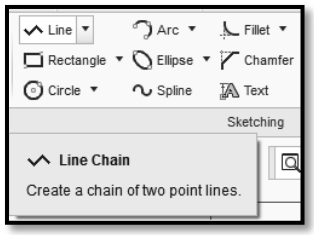
1. Draw a *Centerline* 

by picking from 'Datum' group. Place the centerline by picking two co-linear points as shown in Figure.

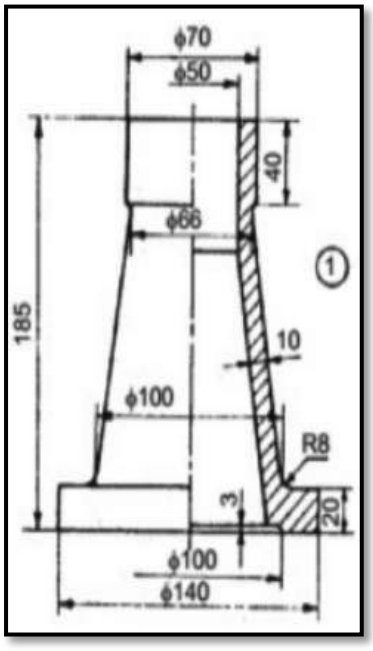


Construction of centerline

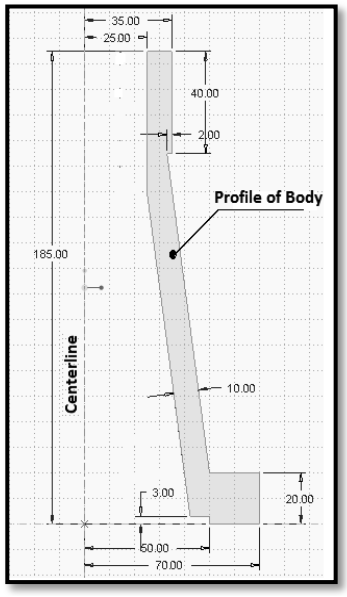
2. Create a profile shown in the Figure with the help of Line chain command from sketching group.



Line Dashboard.

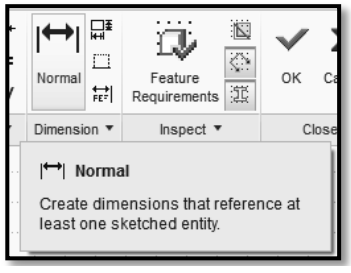


View of Screw Jack Body.



Profile of Screw Jack Body without Fillet.

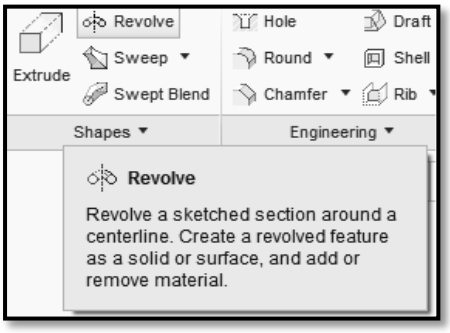
3. Create and Edit the dimensions by using Normal  from Dimension group.



Dashboard of Normal Dimensions.

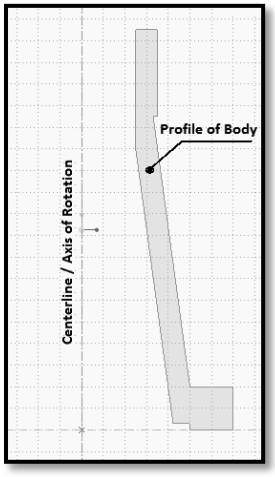
4. Click *OK*  from the Close group of the Sketch tab to complete the sketch and return to the *Revolve / Extrude* dashboard.

Then click on Revolve  command from Shapes group then select axis / centerline.



Dashboard of Revolve command

Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.

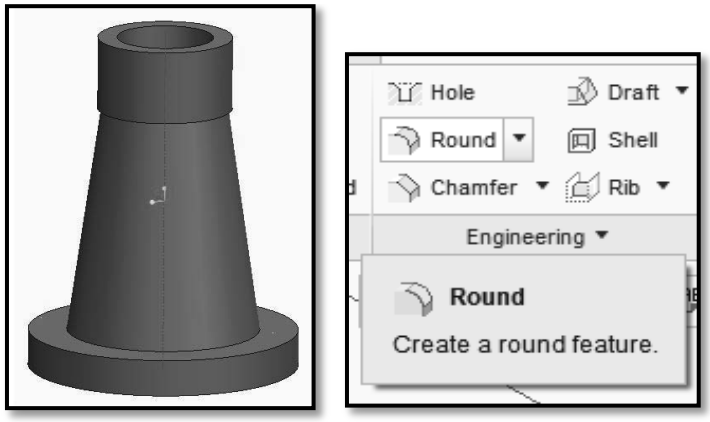


Axis of revolution for body profile.

Angle of rotation and Accept Dashboard.

5. Enter an angle value as 360° and click Accept button. 

We will get body of the screw jack without fillets as shown in the Figure.

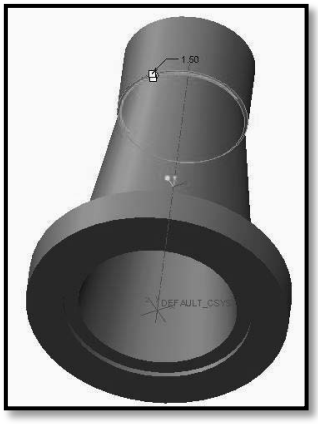


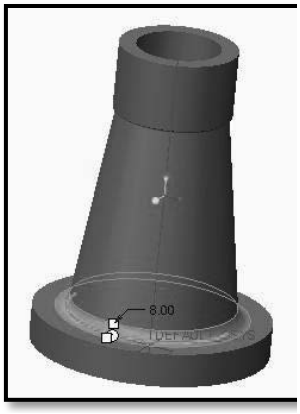
Body of Screw Jack without fillets.

6. Then provide the fillets with the help of **Round** 

command from engineering group at required corners / edges and specify respective radius and click on Accept  button. Do this activity separately for every edge.

Radius and Accept dashboard.



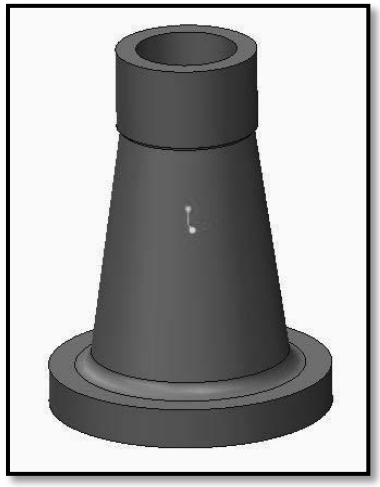


(a) **(b)**

(c)

Fig. (a), (b), (c) – Fillets at different edges and corners

We will get final shape of Screw Jack Body with Fillets as shown in the Figure.



Body of Screw Jack with fillets.

7. Saving your work

- In the Quick Access toolbar, click Save  to save your model.
- In the **Save Object** dialog, click **OK** to specify that the model will be saved to your working directory.

XI Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XII Actual Procedure Followed

.....
.....
.....
.....
.....

XIII Precautions Followed

.....
.....
.....
.....
.....
.....

XIV Course Proficiency

.....
.....
.....
.....
.....

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

XVI Questions:

1. State the meaning of mmns, mmks.
2. List different uses of mouse buttons.

[Space for Answer]

.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....

.....

.....

.....

.....

.....

.....

.....

.....

XVII References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=XlsaSe444AE
2. https://www.youtube.com/watch?v=b6b9FY14PKw
3. https://www.youtube.com/watch?v=X0AMdUMNsDI

XVIII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40%**
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60%**
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.6: Develop solid models of individual components of Bench Vice / Drill Jig / Screw Jack / Tool Post / anyone assembly consisting of at least five parts

I Practical Significance

To create solid models of any mechanical components.To learn different sketching and modeling commands. Also understand datum features and datum plane theory. Study different geometric and modeling constraints. From design engineers point of view, can be seen the object from various directions and in various views. It helps to be sure that the object looks exactly as wanted. It also gives additional vision as to what more changes can be done in the object.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills

- Starting and creating new parts.
- Datum plane - visibility.
- Dashboard interface.
- Revolve Protrusion.

IV Relevant Course Outcome(s)

- Generate 3D models from 2D sketches using Part workbench of any parametric Modeling software.

V Practical Outcome

- Operate available modeling software to draw 3D Models of any engineering product.

VI Relative Affective Domain-

- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Basic knowledge of reading of 3D objects.
- Knowledge of creating working directory.
- Basic knowledge of preparation of 2D sketches.
- Basic knowledge of computer handling.
- Basic knowledge of geometric and dimensional constructions.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

XVI Precautions to be Followed

1. Student should understand and can visualize at least two Orthographic views of any model.
2. While constructing 2D sketch, boundary (Area) of any profile should be enclosed.
3. While specifying dimensions, carefully select the entity or end points of entity and click the middle button (roller) of mouse.
4. Check given drawing for dimensional printing mistakes if any and if dimensions are missing assume proportionate dimensions.
5. While using Sweep or Helical sweep command, need to draw reference axis, reference line, trajectory and helical sweep section. These activity should do carefully.
6. While constructing the drawing, periodically save your work.

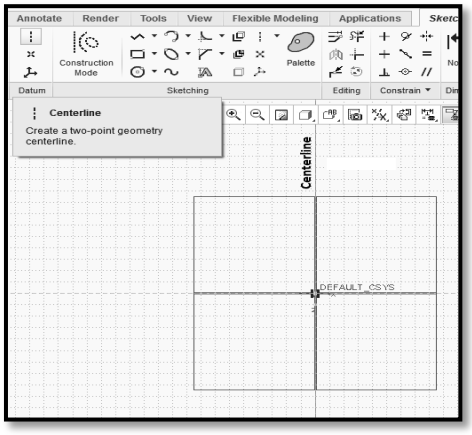
IX Procedure

For Setting the Working Directory and Selection of Sketching Plane (Step 1 & Step 2): Follow the steps as explained in First Practical.

Step 3: Create a sketch to define the shape of Nut of the Screw Jack**.**

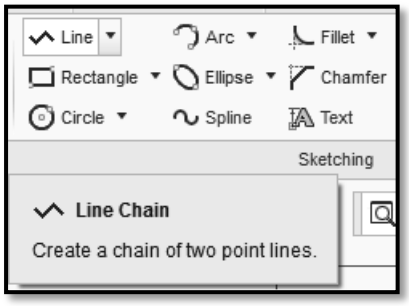
1. Draw a Centerline by picking 

from ‘Datum’ group. Place the centerline by picking two co-linear points as shown in Figure.

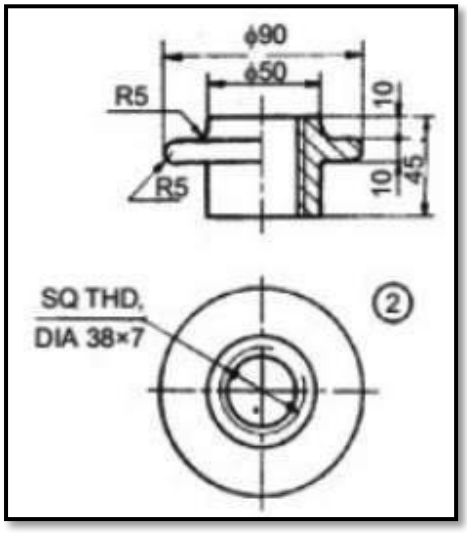


Construction of centerline

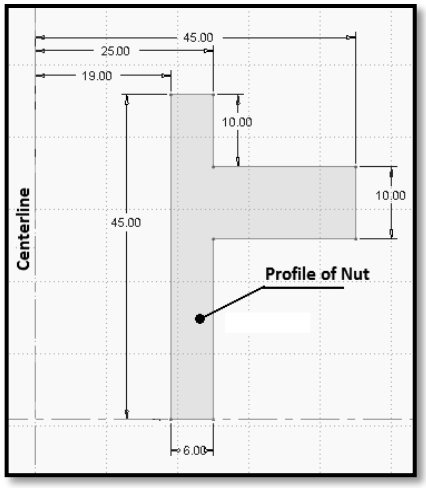
2. Create a profile shown in the Figure with the help of Line chain command from sketching group.



Line Dashboard.

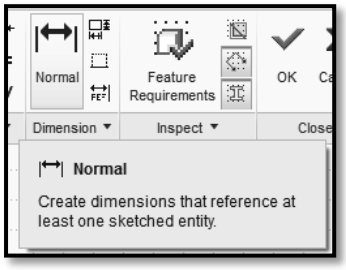


View of Screw Jack Nut.



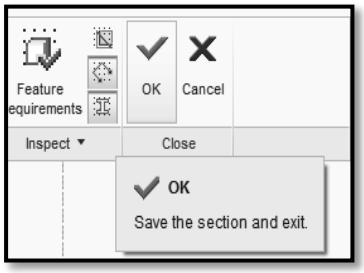
Profile of Screw Jack Nut without Fillet.

3. Create and Edit the dimensions by using Normal from Dimension group.



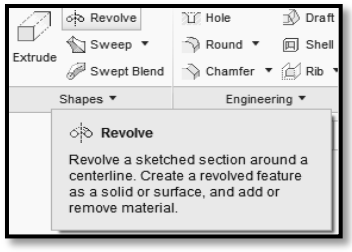
Dashboard of Normal Dimensions.

4. Click OK  from the Close group of the Sketch tab to complete the sketch and return to the Revolve / Extrude dashboard.



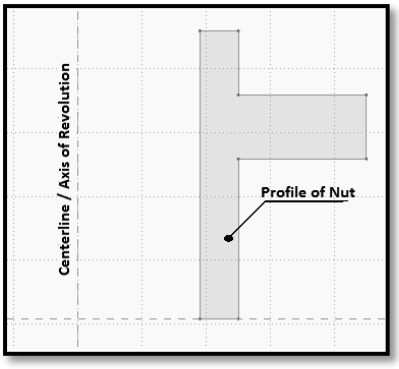
Dashboard of Accept.

5. Then click on **Revolve**  command from Shapes group then select axis / centerline.



Dashboard of Revolve command

Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.

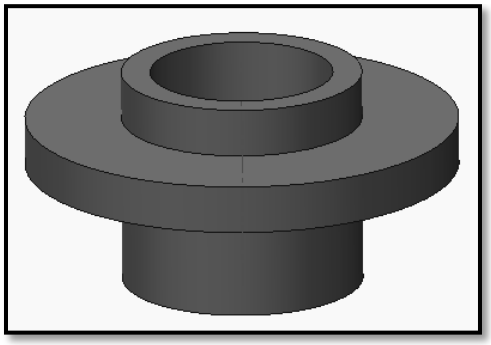


Axis of revolution for Nut profile.

6. Enter an angle value as 360° and click Accept  button.

Angle of rotation and Accept Dashboard.

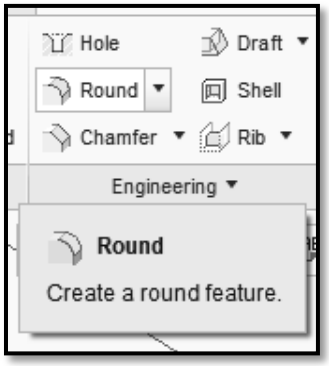
We will get body of the screw jack without fillets as shown in the Figure.



Nut of Screw Jack without fillets.

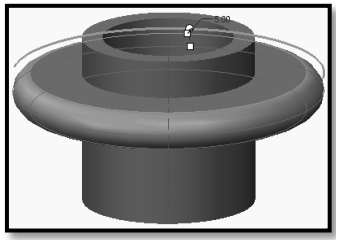
7. Then provide the fillets with the help of **Round** 

command from Engineering group at required corners / edges and specify respective radius and click on Accept  button. Do this activity separately for every edge.



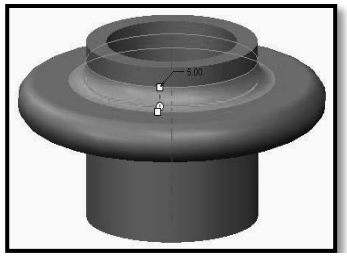
Radius dashboard.

Round command dashboard.



(a)

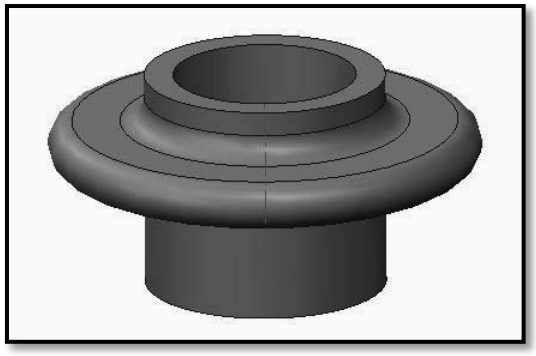
(b)



(c)

(a), (b), and (c) – Fillets at different corners and edges.

We will get final shape of Screw Jack Nut with Fillets as shown in the Figure.



Nut of Screw Jack with fillets.

8. Saving your work

- In the Quick Access toolbar, click **Save**  to save your model.
- In the **Save Object** dialog, click **OK** to specify that the model will be saved to your working directory.

X Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XI Actual Procedure Followed

.....

.....

.....

.....

XII Precautions Followed

.....

.....

.....

.....

XIII Course Proficiency

.....

.....

.....

.....

XIV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

Questions:

1. State any two elements required for Revolve Protrusion.
2. Explain different methods of selection of sketching planes.
3. List five uses of datum planes.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XV References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=XlsaSe444AE
2. https://www.youtube.com/watch?v=b6b9FY14PKw
3. https://www.youtube.com/watch?v=X0AMdUMNsDI

XVI Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.7: Develop Solid Models Of Individual Components Of Bench Vice / Drill Jig / Screw Jack/ Tool Post / Anyone Assembly Consisting Of At Least Five Parts.

I Practical Significance

To create solid models of any mechanical components.To learn different sketching and modeling commands. Also understand datum features and datum plane theory. Study different geometric and modeling constraints. From design engineers point of view, can be seen the object from various directions and in various views. It helps to be sure that the object looks exactly as wanted. It also gives additional vision as to what more changes can be done in the object.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills –

- Starting and creating new parts.
- Dashboard interface.
- Revolve Protrusion.
- Sweep and Helical Sweep.
- Concept of Trajectory.

IV Relevant Course Outcome(s)

- Generate 3D models from 2D sketches using Part workbench of any parametric Modeling software.

V Practical Outcome

- Operate available modeling software to draw 3D Models of any engineering product.

VI Relative Affective Domain-

- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Basic knowledge of reading of 3D objects.
- Knowledge of creating working directory.
- Basic knowledge of preparation of 2D sketches.
- Basic knowledge of computer handling.
- Basic knowledge of geometric and dimensional constructions.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

IX Precautions to be Followed

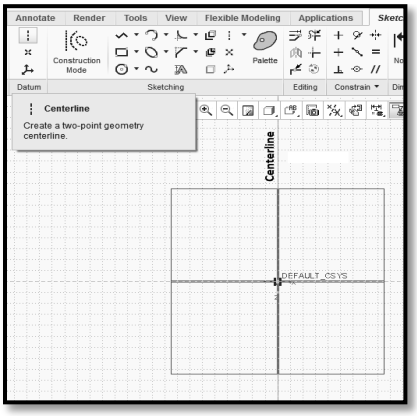
1. Student should understand and can visualize at least two Orthographic views of any model.
2. While constructing 2D sketch, boundary (Area) of any profile should be enclosed.
3. While specifying dimensions, carefully select the entity or end points of entity and click the middle button (roller) of mouse.
4. Check given drawing for dimensional printing mistakes if any and if dimensions are missing assume proportionate dimensions.
5. While using Sweep or Helical sweep command, need to draw reference axis, reference line, trajectory and helical sweep section. These activities should do carefully.
6. While constructing the drawing, periodically save your work.

X Procedure

For Setting the Working Directory and Selection of Sketching Plane (Step 1 & Step 2): Follow the steps as explained in First Practical.
Step 3: Create a sketch to define the shape of Screw Spindle of the Screw Jack.

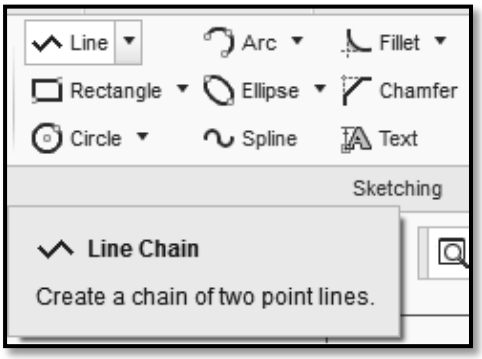
1. Draw a Centerline 

by picking from ‘Datum’ group. Place the centerline by picking two co-linear points as shown in alongside Figure.

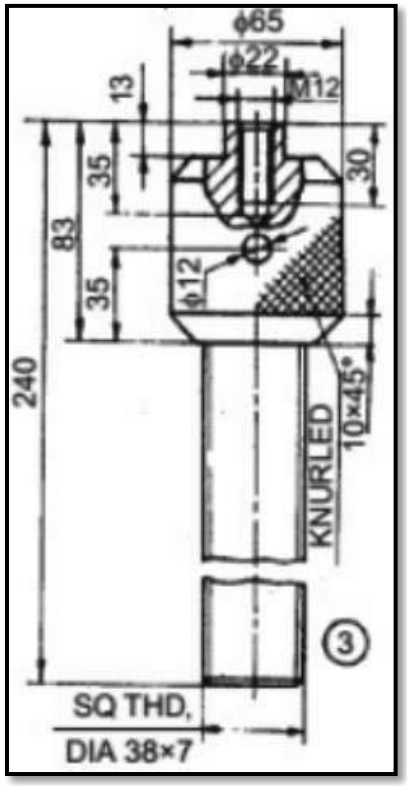


Construction of centerline.

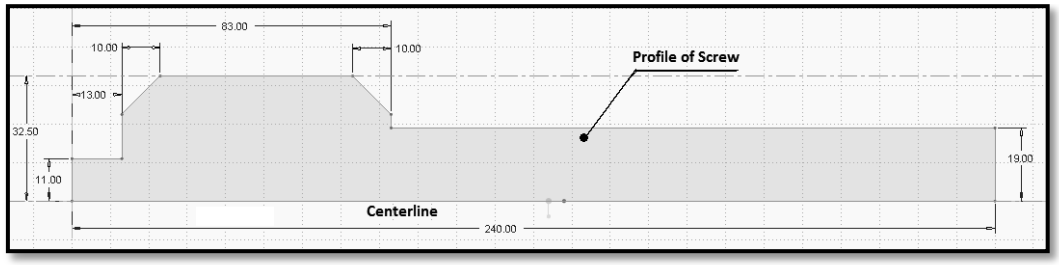
2. Create a profile shown in the Figure with the help of Line chain command from sketching group.



Line Dashboard.

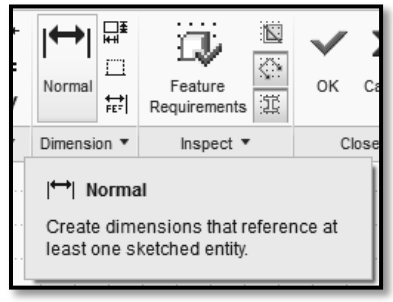


View of Screw Spindle.



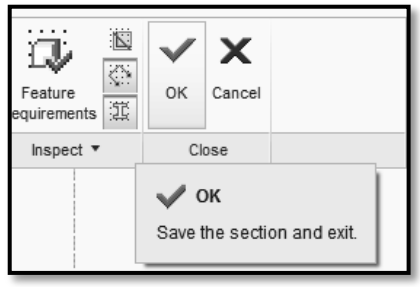
Profile of Screw Spindle.

3. Create and Edit the dimensions by using Normal  from Dimension group.



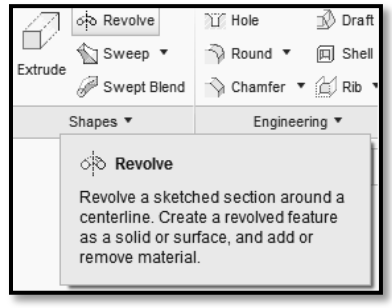
Dashboard of Normal Dimension.

4. Click OK  from the Close group of the Sketch tab to complete the sketch and return to the Revolve / Extrude dashboard.



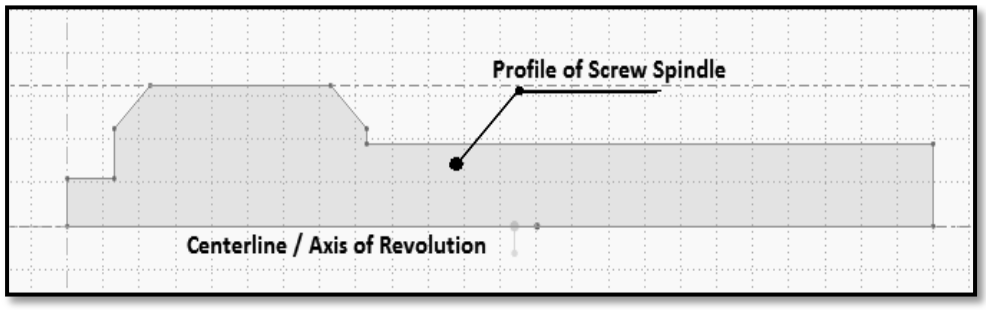
Dashboard of Accept.

5. Then click on **Revolve**  command from Shapes group then select axis / centerline.



Dashboard of Revolve command.

Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.



Axis of revolution for Screw Spindle.

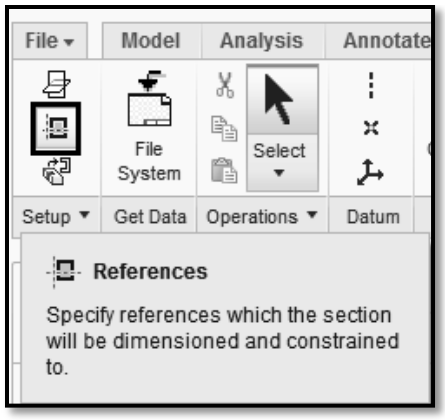
6. Take revolution angle as 360°. And click on Accept  button.

Angle of rotation and Accept Dashboard.

We will get Screw Spindle of the screw jack without threads as shown in the Figure.

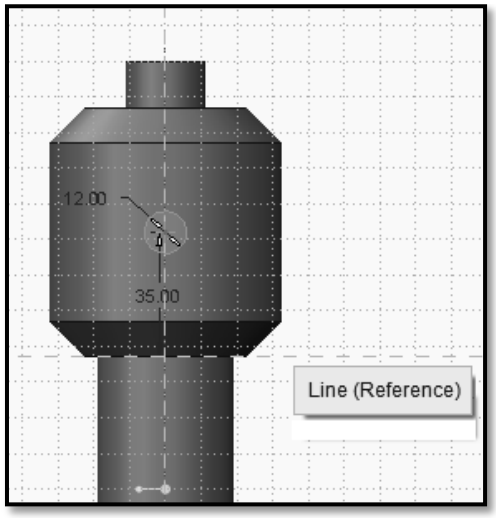
Screw Spindle without Hole & Threads.

7. Create a throughout hole on central reference plane (Shank portion) of Screw Spindle at specified distance.
   i) Go to Sketch menu.
   ii) Set a line reference by clicking **Setup group > References** and select the edge as shown in the Figure[Line (Reference)]



References Dashboard.

iii) Draw circle on the axis as shown and edit dimensions, diameter as 12mm and distance as 35mm from reference line.



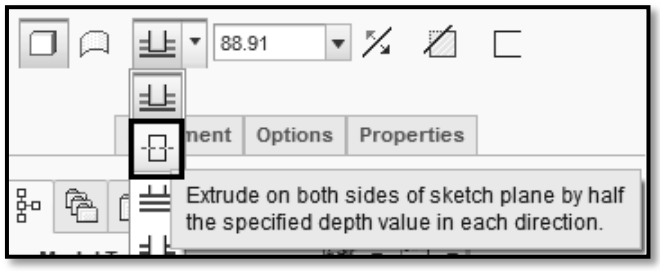
Circular Sketch for circular Hole.

iv) Click Accept / *OK*  from the Close group of the Sketch tab to complete the sketch and return to the *Revolve / Extrude* dashboard.

v) Click on 'Extrude from sketch plane by a specified depth value' feature.

(a)–Extrude Dashboard.

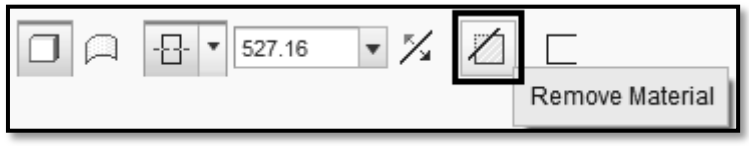
OR click on 'Extrude on both sides of sketch plane by half the specified depth value in each direction' feature.



(b)– Extrude Dashboard.

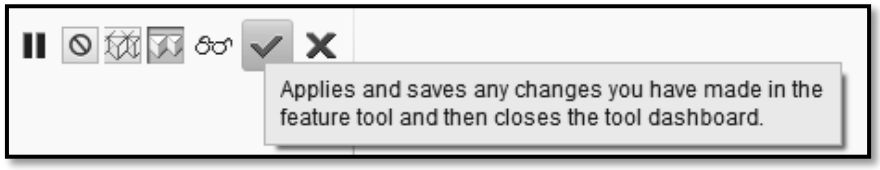
vi) Drag the reference handles and set the dimension of the hole and its depth, in the extrude dashboard.

vii) Then click on 'Remove Material' feature as shown below.



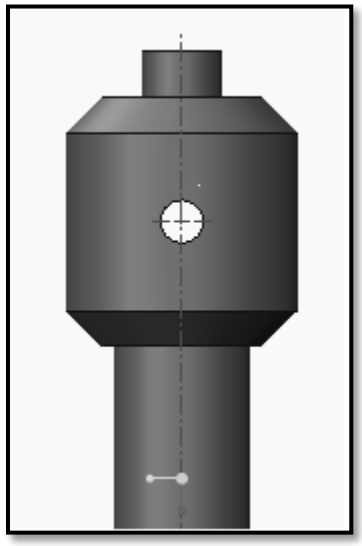
Remove material Dashboard.

viii) Then click Accept  to finish the feature creation.



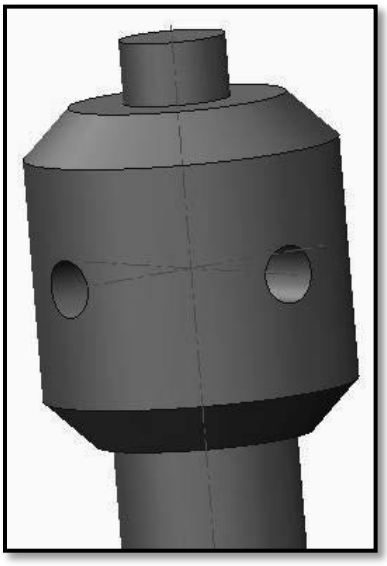
Accept Dashboard.

We will get the throughout hole as shown in Figure below.



(a)-Completed Hole feature.

ix) By selecting another reference plane, follow the same procedure to create another throughout hole on Shank Portion of Screw Spindle as shown in the Figure.

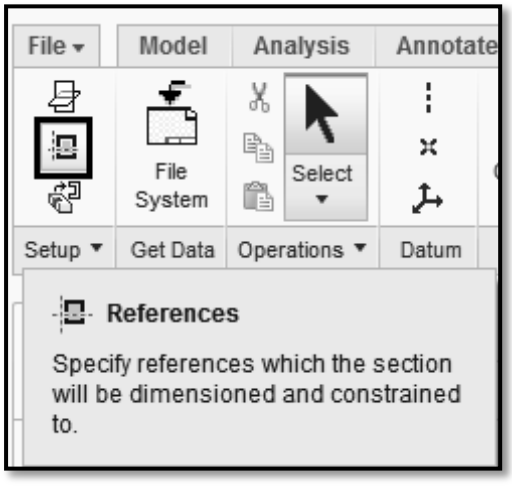


(b)– Completed Hole feature from both side.

8. Create Threads on Screw Spindle.

i) Go to Sketch menu.

ii) Set a reference by clicking **Setup group** > **References** and select left side edge as a reference shown in the Figure.



Reference Dashboard. Reference line, Trajectory & Center line

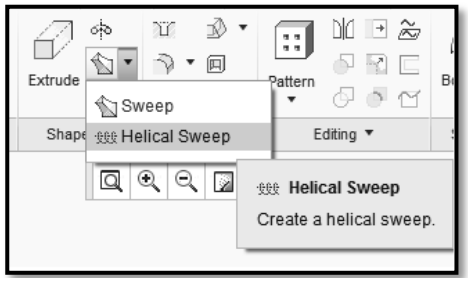
iii) Draw Trajectory by using line command on earlier created reference i.e. on left side of spindle surface.

iv) Then draw geometrical centerline at the axis of spindle by using centerline command from datum group as shown in the Figure.

v) Then click Accept  button.

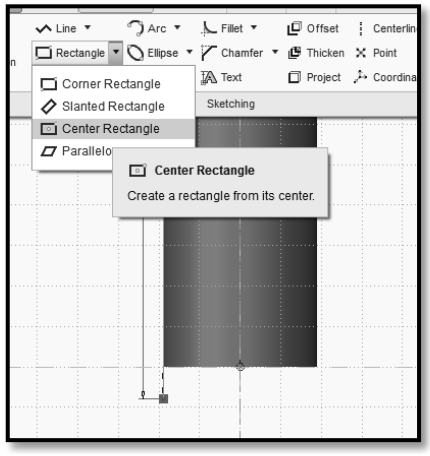
Note:-Start point or end point of the Trajectory should be somewhat away from the bottom surface of the object as shown in the Figure.

vi) By taking Helical Sweep Command, then select create or edit sweep section command and draw section of screw thread (i.e. Sweep Section) at the End Point of the Trajectory / Line, by using Center Rectangle command considering pitch of screw thread as 7mm.

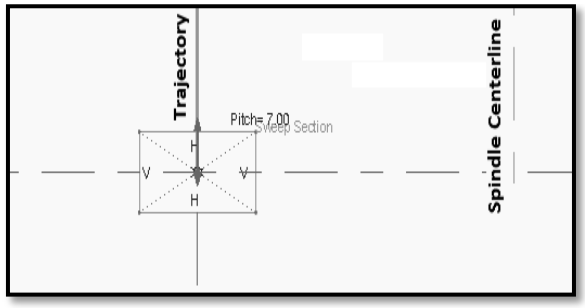


Helical Sweep Dashboard

Pitch of Screw Dashboard.

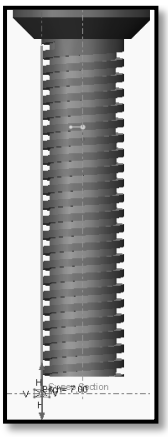


Construction of Center Rectangle



Rectangular Thread Section at the end of Trajectory.

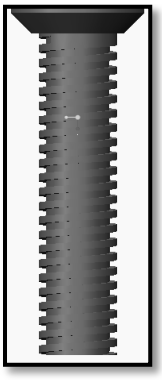
vii) Click on Remove Material button.



Threaded Screw

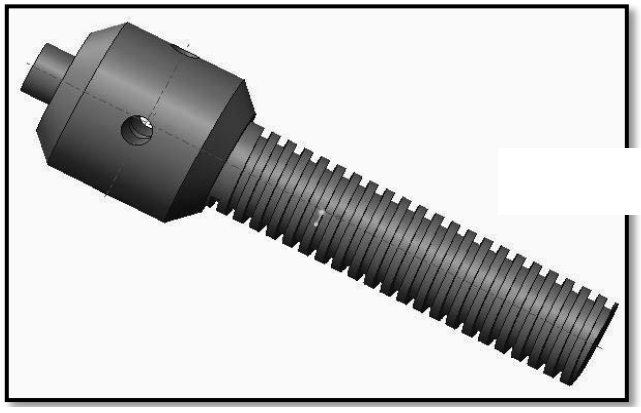
viii) Then click on Accept  button.

We will get threads on Spindle as shown in the Figure.



Completed threaded Screw Spindle.

Finally we will get final shape of Screw Spindle with all features as shown in the Figure.



9. Saving your work

• In the Quick Access toolbar, click **Save**  to save your model.

• In the **Save Object** dialog, click **OK** to specify that the model will be saved to your working directory.

XI Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XII Actual Procedure Followed

.....

.....

.....

.....

.....

XIII Precautions Followed

.....

.....

.....

.....

.....

XIV Course Proficiency

.....

.....

.....

.....

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

Questions:

1. List three uses of datum axes.
2. Explain Normal dimension feature.
3. Explain the model tree.

4. Use of Reference line during threading process.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

XVI References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=XlsaSe444AE
2. https://www.youtube.com/watch?v=b6b9FY14PKw
3. https://www.youtube.com/watch?v=X0AMdUMNsDI

XVII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40%**
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60%**
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.8 : Develop Solid Models of Individual Components of Bench Vice / Drill Jig / Screw Jack / Tool Post Assembly Consisting Of At Least Five Parts.

I Practical Significance

To create solid models of any mechanical components. To learn different sketching and modeling commands. Also understand datum features and datum plane theory. Study different geometric and modeling constraints. From design engineers point of view, can be seen the object from various directions and in various views. It helps to be sure that the object looks exactly as wanted. It also gives additional vision as to what more changes can be done in the object.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills –

- Starting and creating new parts.
- Controlling the display of datum features datum plane theory.
- Sweep and Helical Sweep.
- Concept of Trajectory.

IV Relevant Course Outcome(s)

- Generate 3D models from 2D sketches using Part workbench of any parametric Modeling software.

V Practical Outcome

- Operate available modeling software to draw 3D Models of any engineering product.

VI Relative Affective Domain-

- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Basic knowledge of reading of 3D objects.
- Knowledge of creating working directory.
- Basic knowledge of preparation of 2D sketches.
- Basic knowledge of computer handling.
- Basic knowledge of geometric and dimensional constructions.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

IX Precautions to be Followed

1. Student should understand and can visualize at least two Orthographic views of any model.
2. While constructing 2D sketch, boundary (Area) of any profile should be enclosed.
3. While specifying dimensions, carefully select the entity or end points of entity and click the middle button (roller) of mouse.
4. Check given drawing for dimensional printing mistakes if any and if dimensions are missing assume proportionate dimensions.
5. While using Sweep or Helical sweep command, need to draw reference axis, reference line, trajectory and helical sweep section. These activity should do carefully.
6. While constructing the drawing, periodically save your work.

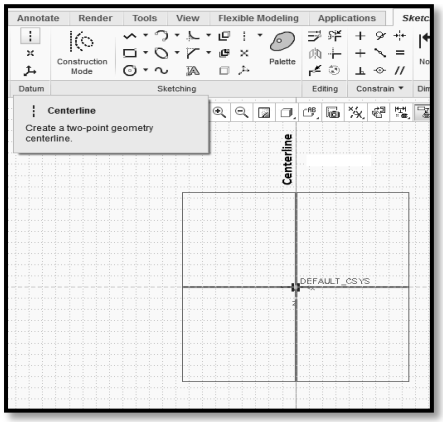
X Procedure

For Setting the Working Directory and Selection of Sketching Plane (Step 1 & Step 2): Follow the steps as explained in Previous Practical's.

Step 3: Create a sketch to define the shape of Cup of the Screw Jack.

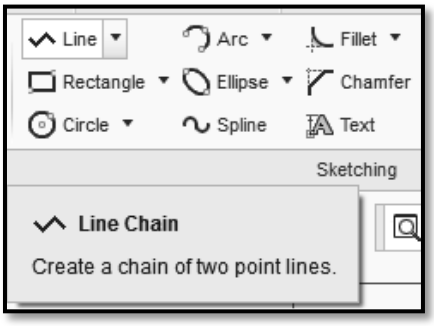
1. Draw a Centerline 

by pickingfrom 'Datum' group. Place the centerline by picking two co-linear points as shown in Figure.

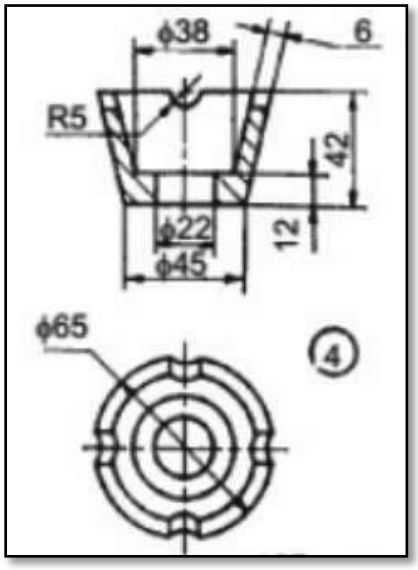


Construction of centerline

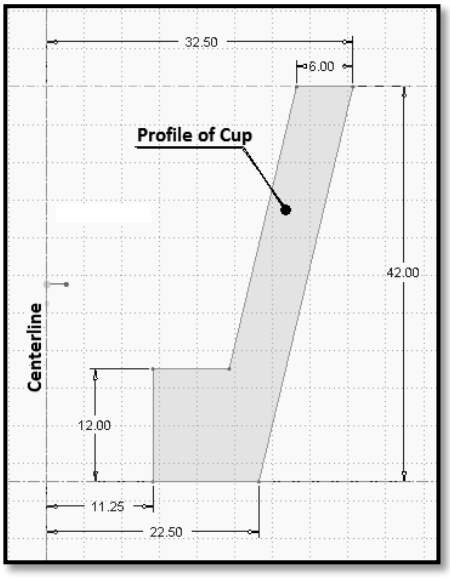
2. Create a profile shown in the Figure with the help of Line chain command from sketching group.



Line Dashboard

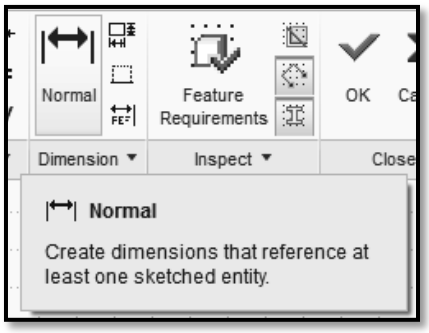


Viewof Cup.



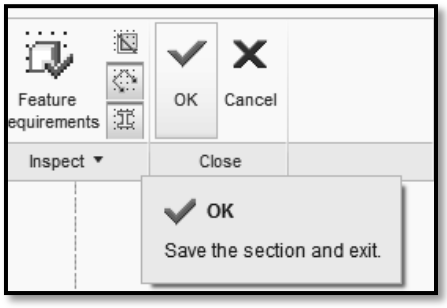
Profileof Cup.

4. Create and Edit the dimensions by using Normal  from Dimension group.



Dashboard of Normal Dimension.

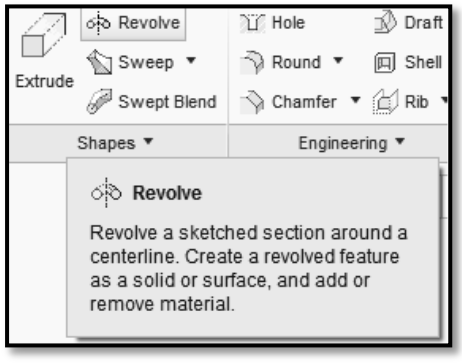
5. Click *OK*  from the Close group of the Sketch tab to complete the sketch and return to the *Revolve / Extrude* dashboard.



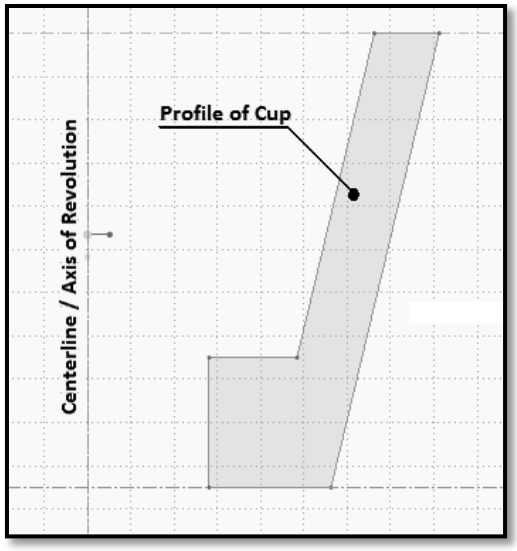
Dashboard of Accept.

6. Then click on **Revolve**  command from Shapes group then select axis / centerline.

Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.



Dashboard of Revolve command.

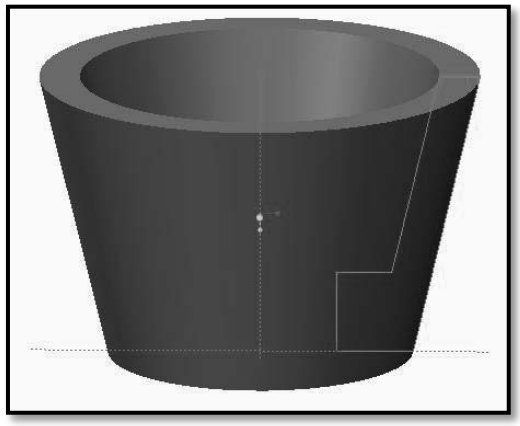


Axis of revolution for Screw Jack Cup.

7. Take revolution angle as 360° and click on Accept  button.

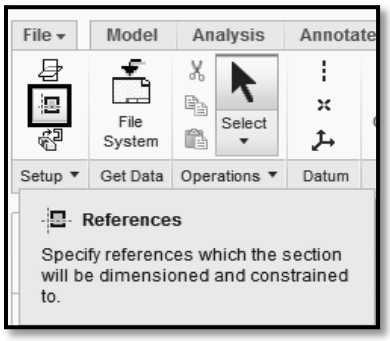
Angle of rotation and Accept Dashboard.

We will get Screw Jack Cup as shown in the Figure.



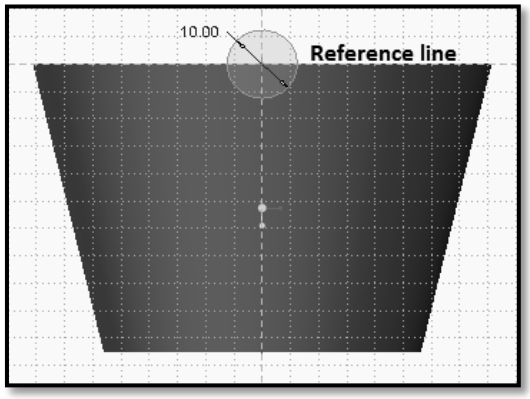
Modelof Screw Jack Cup without Slots.

8. Create a Semicircular Slots.
   i) Go to Sketch menu.
   ii) Set a line reference by clicking **Setup group > References** and select the edge as shown in the Figure [Line (Reference)]



References Dashboard.

iii) Draw circle of diameter 10mm at the intersecting point of reference lines.



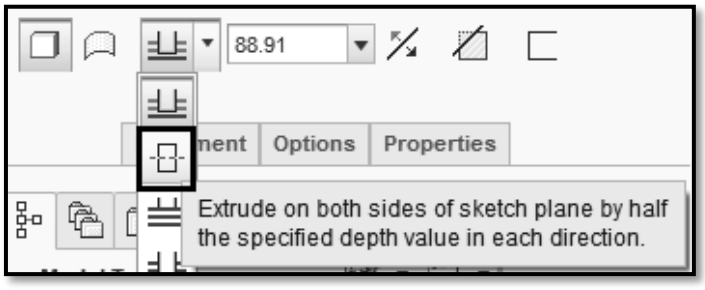
Circular Sketch for Semicircular Slot.

iv) Click OK  from the Close group of the Sketch tab to complete the sketch and return to the Extrude dashboard.

v) Click on 'Extrude from sketch plane by a specified depth value' feature.

(a)– Extrude Dashboard.

OR click on 'Extrude on both sides of sketch plane by half the specified depth value in each direction' feature.



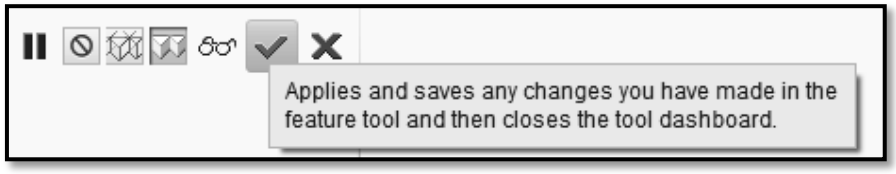
(b)– Extrude Dashboard.

vi) Drag the reference handles and set the diameter of the hole and its depth, in the dashboard.

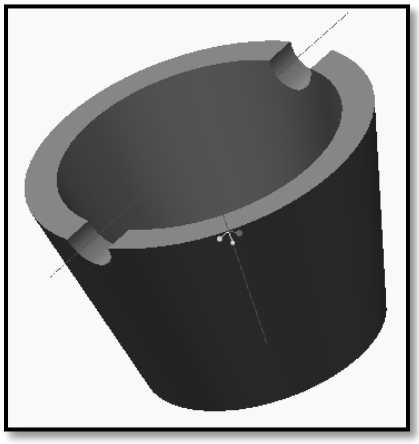
vii) Then click on 'Remove Material' feature.

Remove material Dashboard.

viii) Then click Accept  to finish the feature creation.

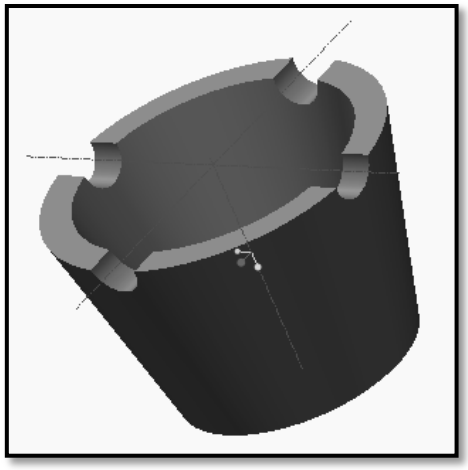


Accept Dashboard.



Completed Slot feature.

ix) By selecting another reference plane, follow the above procedure to createotherSemicircular Slot as shown in the Figure.



Completed Screw Jack Cup with Semicircular Slots.

9. Saving your work

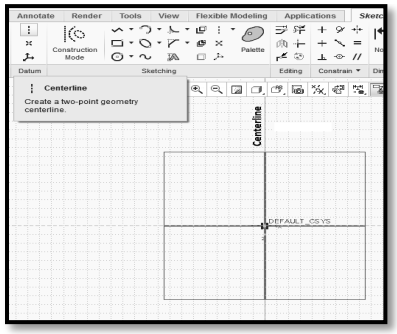
• In the Quick Access toolbar, click **Save**  to save your model.

• In the **Save Object** dialog, click **OK** to specify that the model will be saved to your working directory.

Step 4: Create a sketch to define the shape of Washer of the Screw Jack.

1. Draw a Centerline 

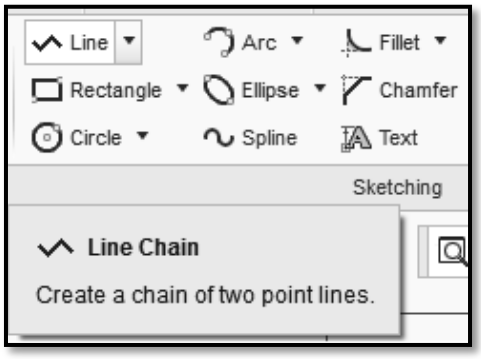
by picking from 'Datum' group. Place the centerline by picking two co-linear points as shown in Figure.



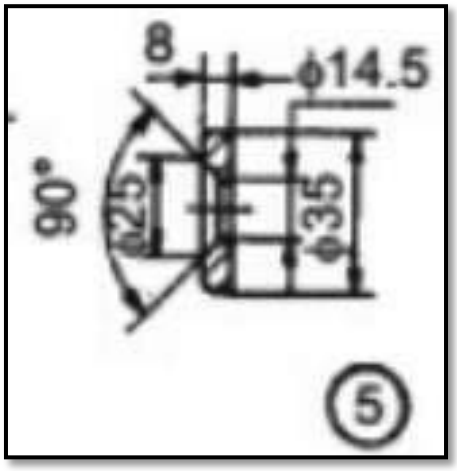
Construction of centerline

2. Create a profile shown in the Figure with the help of Line chain 

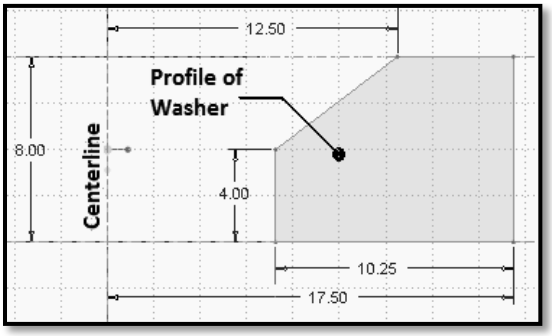
command from sketching group.



Line Dashboard.

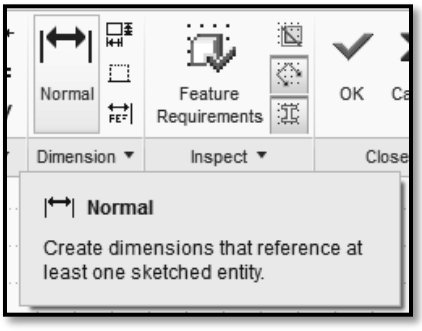


View of Screw Jack Washer.



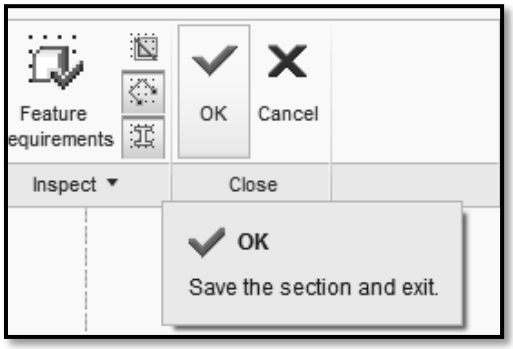
Profile of Screw Jack Washer without Fillet.

3. Create and Edit the dimensions by using Normal  from Dimension group.



Dashboard of Normal Dimension.

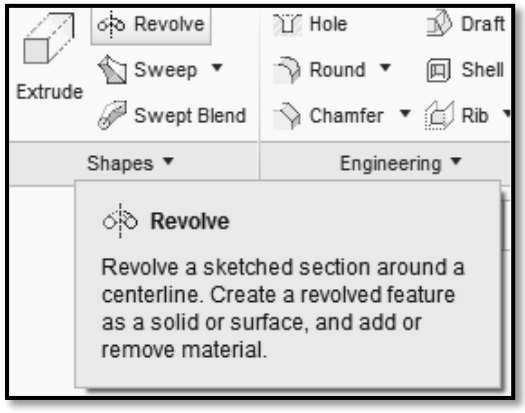
4. Click OK  from the Close group of the Sketch tab to complete the sketch and return to the Revolve / Extrude dashboard.



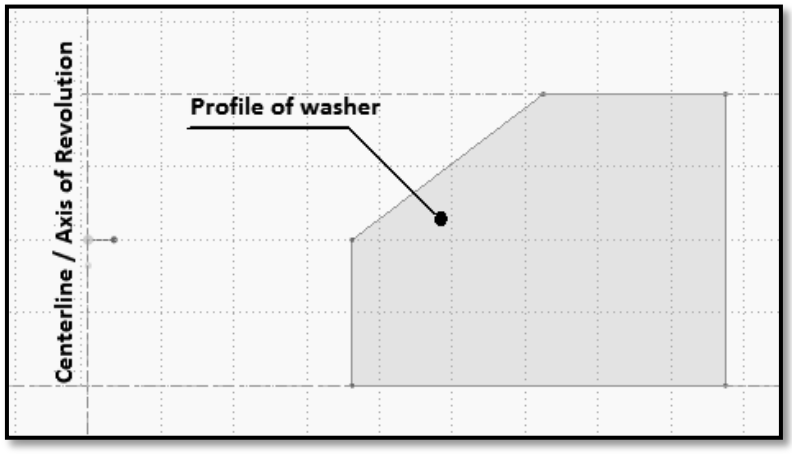
Dashboard of Accept.

5. Then click on **Revolve**  command from Shapes group then select axis / centerline.

   Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.

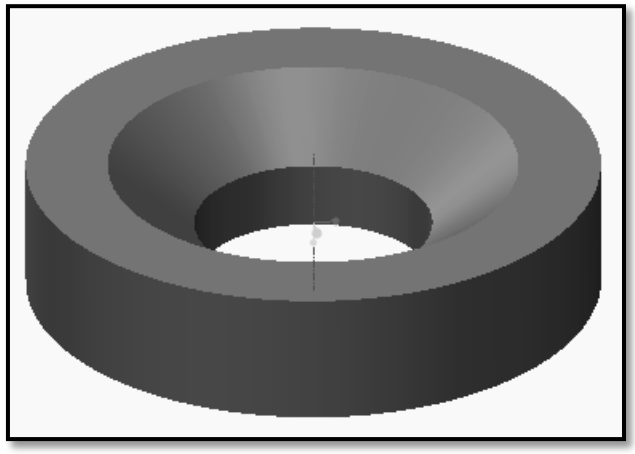


Dashboard of Revolve command.



Axis of revolution for Screw Jack Washer.

We will get Screw Jack Washer as shown in the Figure.

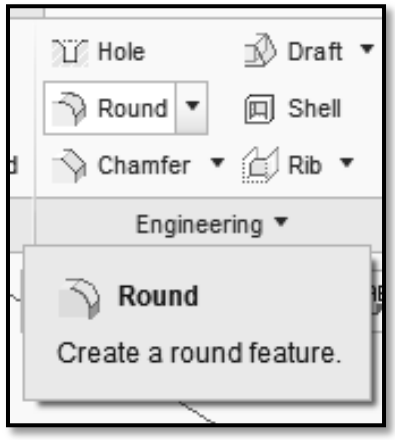


Model of Screw Jack Washer without Fillets.

6. Then provide the fillet with the help of Round 

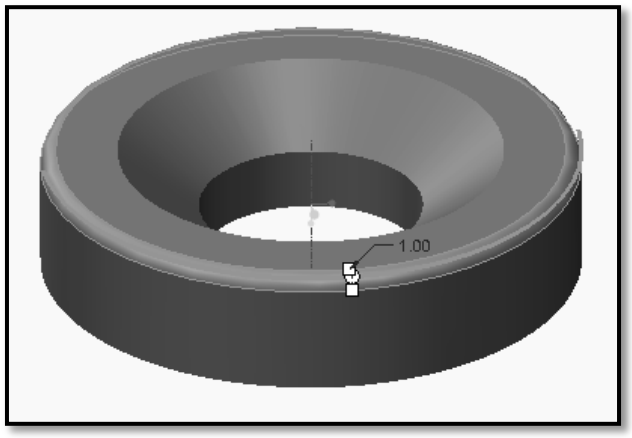
command from Engineering group at the corner / edge and specify respective radius.

   Specify the proportionate value of radius, as it is not given in the sketch.



Round Dashboard.

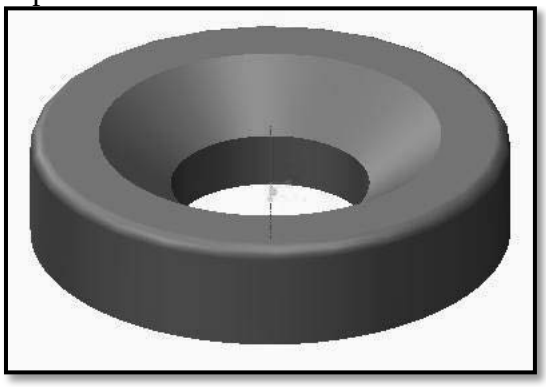
Radius Dashboard.



Washer of Screw Jack with fillet.

7. Click OK  to complete the Fillet.

   We will get final shape of Screw Jack Nut with Fillets as shown in the Figure.



Washer of Screw Jack with fillet.

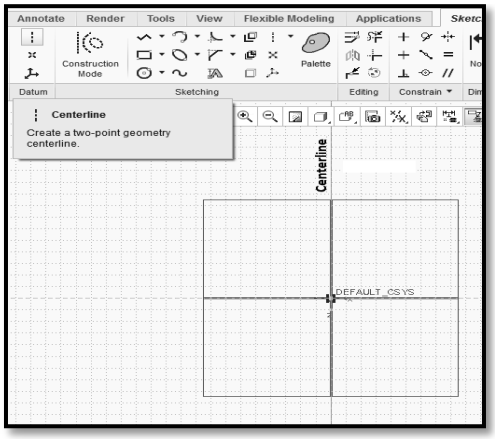
8. Saving your work

- In the Quick Access toolbar, click **Save**  to save your model.
- In the **Save Object** dialog, click **OK** to specify that the model will be saved to your working directory.

Step 5: Create a sketch to define the shape of Screw of the Screw Jack.

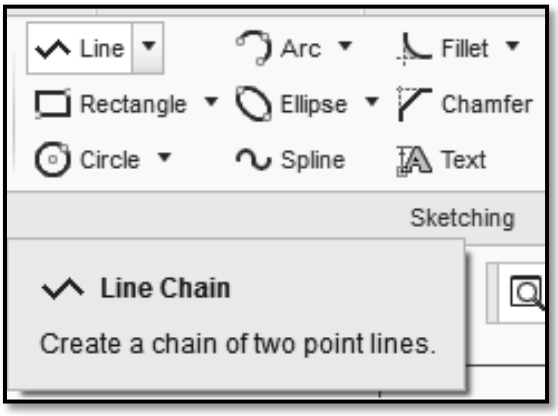
1. Draw a Centerline 

by picking from 'Datum' group. Place the centerline by picking two co-linear points as shown in Figure.

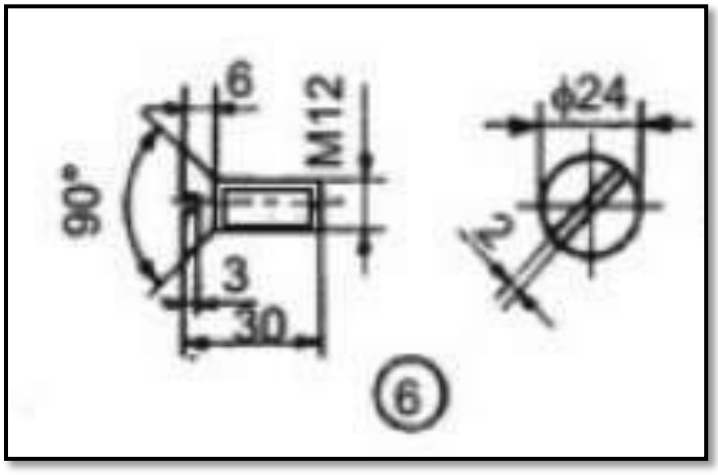


Construction of centerline

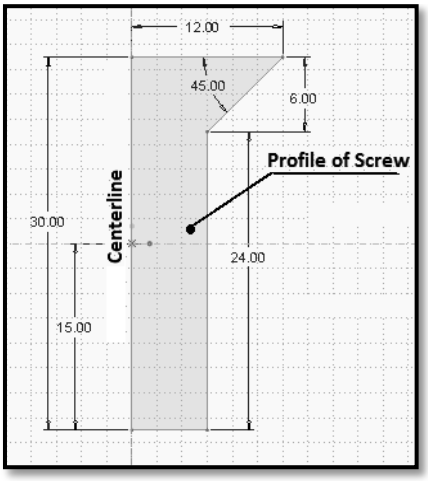
2. Create a profile shown in the Figure with the help of Line chain command from sketching group.



Line Dashboard.

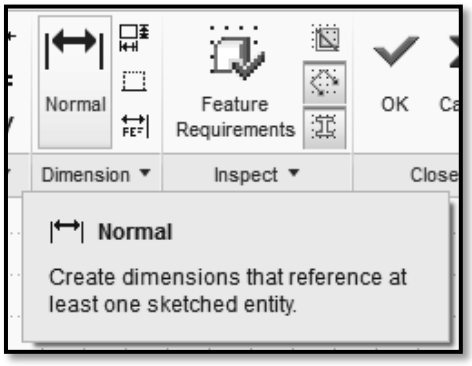


View of Screw of Screw Jack.



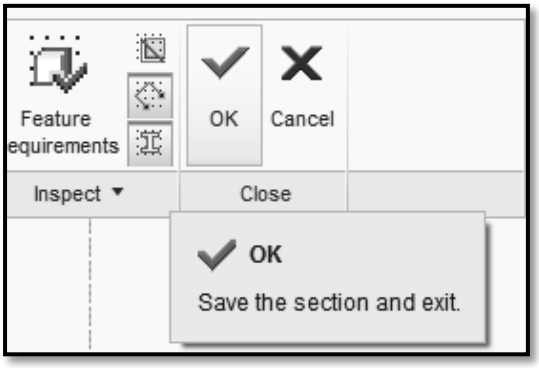
Profile of Screw.

3. Create and Edit the dimensions by using Normal  from Dimension group.



Dashboard Normal Dimension.

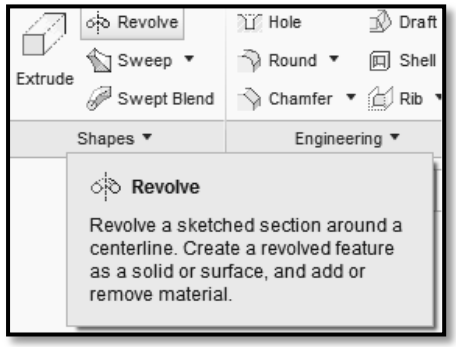
4. Click OK  from the Close group of the Sketch tab to complete the sketch and return to the Revolve / Extrude dashboard.



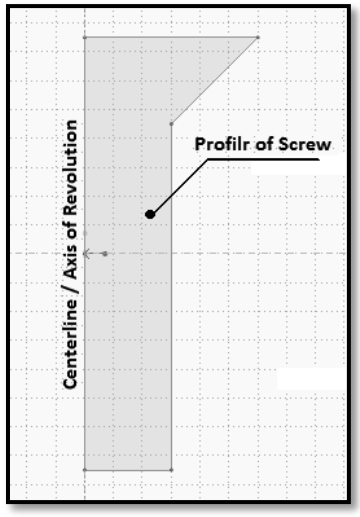
Dashboard for Accept / OK Tool

5. Then click on **Revolve**  command from Shapes group then select axis / centerline.

Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.

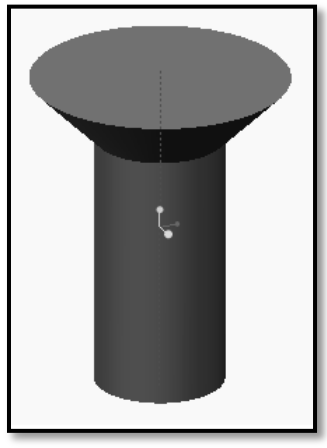


Dashboard for Revolve command.



Axis of revolution for Screw Profile.

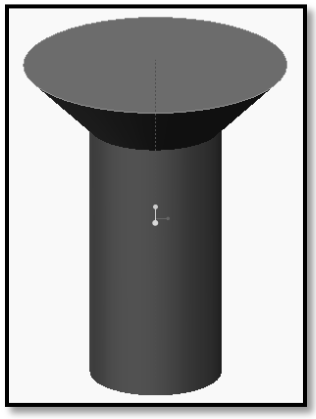
We will get Screw as shown in the Figure.



Model of Screw without Slot & Chamfer.

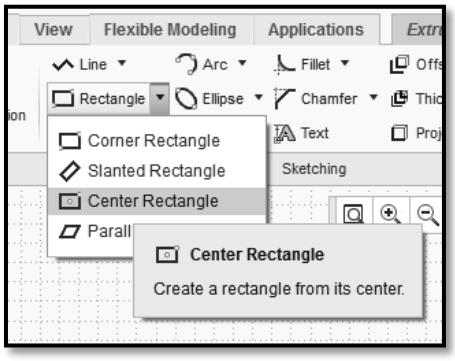
6. To create rectangular slot at the top surface of the screw.

(i) Select the surface as a sketching plane shown in the Figure and then click on the Sketching toolbar.



Selected surface for creating slot.

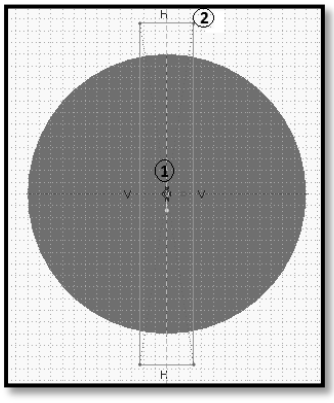
(ii) Click on the Center Rectangle icon as shown in Figure to activate the Rectangle command.



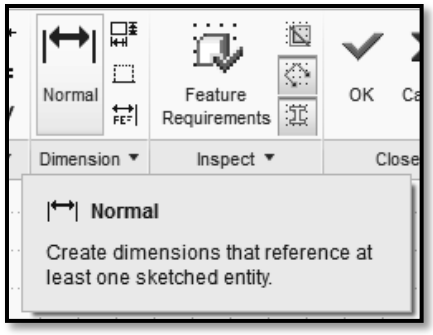
Dashboard for Rectangle command.

(iii) Create a rectangle by clicking Point (1) as center point of rectangle and point (2) as upper right corner of rectangle as shown below.

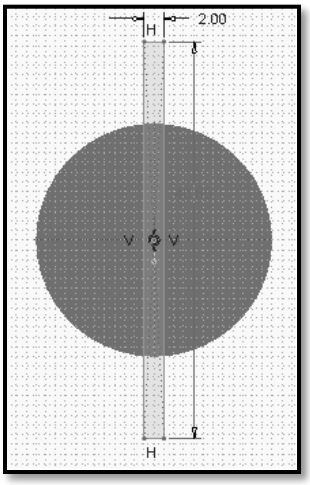
(iv) Create and Edit the dimensions by using Normal  from Dimension group. Edit width as 2 mm.



Creation of Rectangular feature.



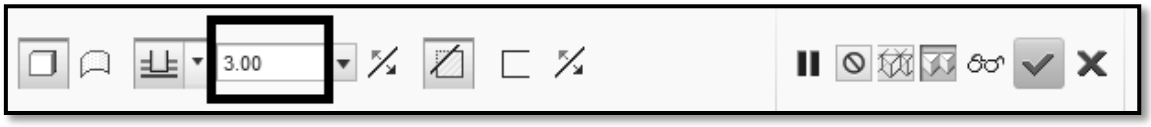
Dashboard of Normal Dimension.



Dimension Edition

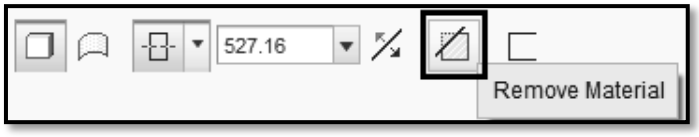
(v) Then click on Accept  buttonfrom the Close group of the Sketch tab to complete the sketch and return to the *Extrude* dashboard.

(vi) Click on 'Extrude command then specify depth value as 3mm.



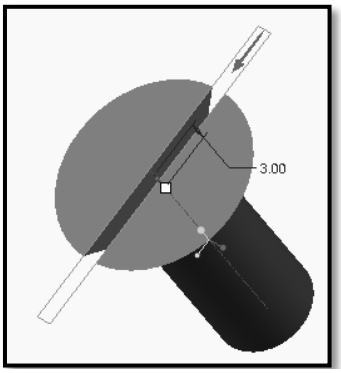
Dashboard for dimension edition.

(vii) Then click on 'Remove Material' feature.

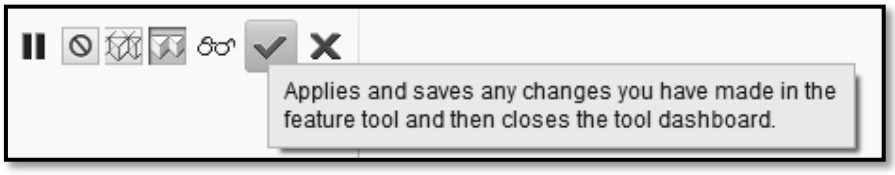


Dashboard for Material Removal.

(viii) To complete the feature clickOK/ Accept  button shown below.

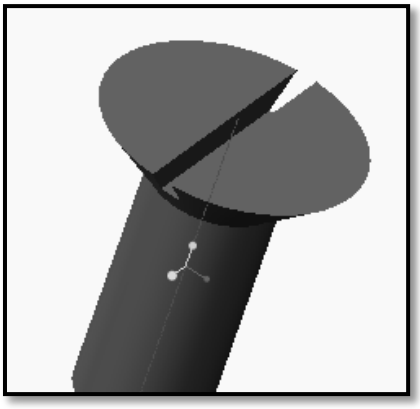


Rectangular Slot Feature



Dashboard for Accept button.

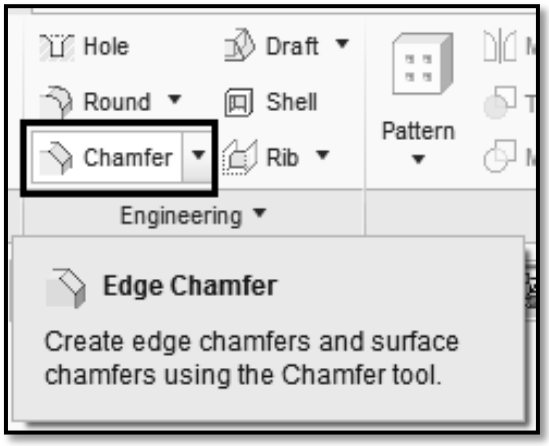
We will get the screw with rectangular slot as shown below.



Screw with Rectangular slot.

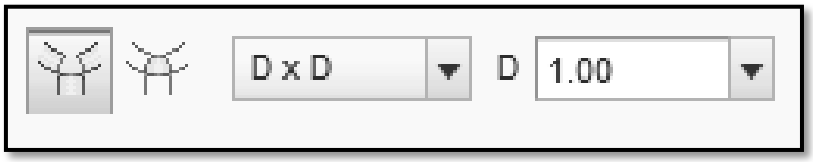
7. To create Chamfer at bottom circular edge.

   (i) Click on the Chamfer command from engineering group as shown below.



Dashboard for Chamfer command.

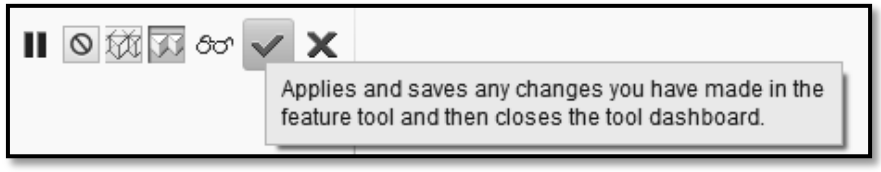
(ii) Select the edge as shown in the Figure and provide the chamfer value as 1mm.



Dashboard for Dimension edition.

Selection of edge for chamfering.

(iii) To complete the feature click *OK / Accept*  button shown below.



Dashboard for Accept button.

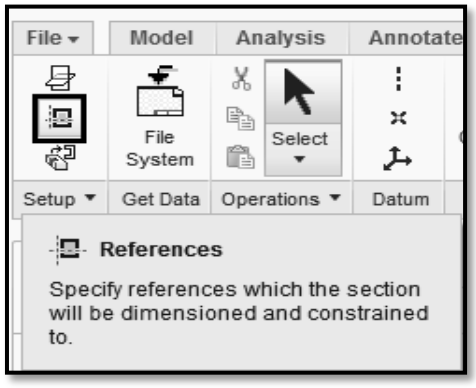
We will get completed Solid Screw with Slot and Chamfer as shown in the Figure.

Screw with Rectangular Slot and Chamfer

8. Create Threads on Screw

i) Go to Sketch menu.

ii) Set a reference by clicking **Setup group > References** and select left side edge as a reference.



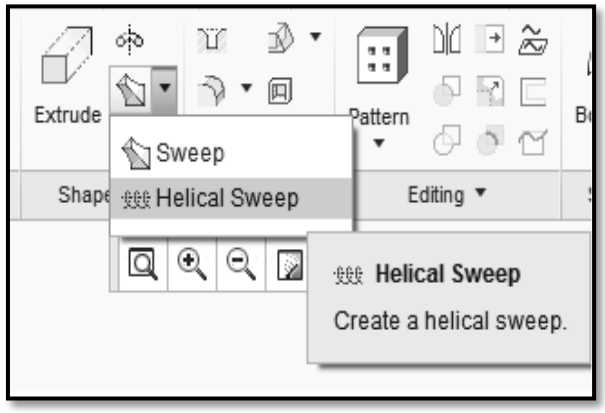
Reference Dashboard.

iii) Draw Trajectory by using line command on earlier created reference i.e. on one side of screw surface.

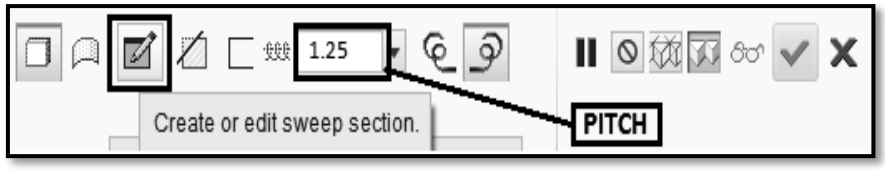
iv) Then draw geometrical centerline at the axis of screw by using centerline command from datum group.

v) Then click Accept  button.

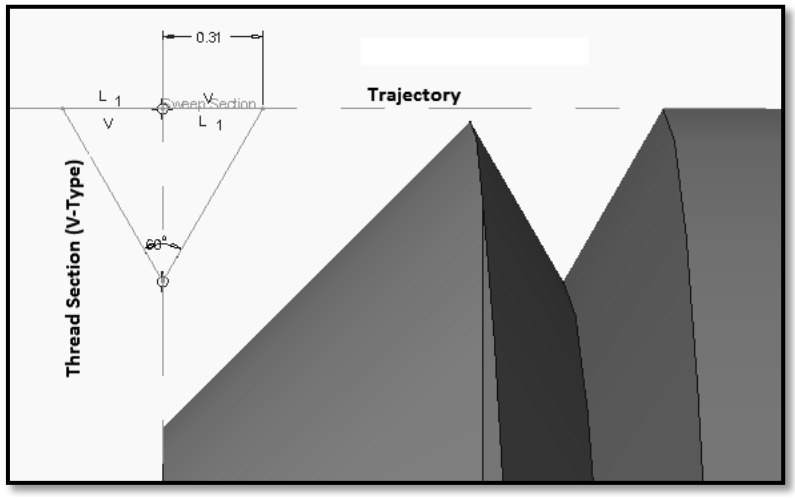
vi) By taking Helical Sweep Command, draw section of screw thread (i.e. Sweep Section – V shape) at the End Point of the Trajectory / Line, by using line command considering pitch of screw thread as 1.25mm.



Helical Sweep Dashboard

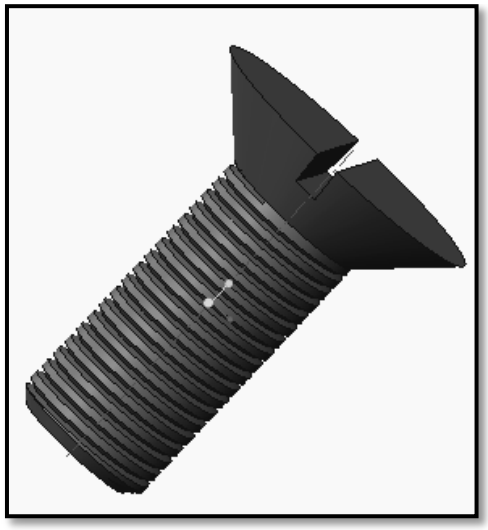


Pitch of Screw Dashboard.



V shape Thread section.

vii) After creating a sweep section (V shape) click on Accept  button.



Completed Screw with All features.

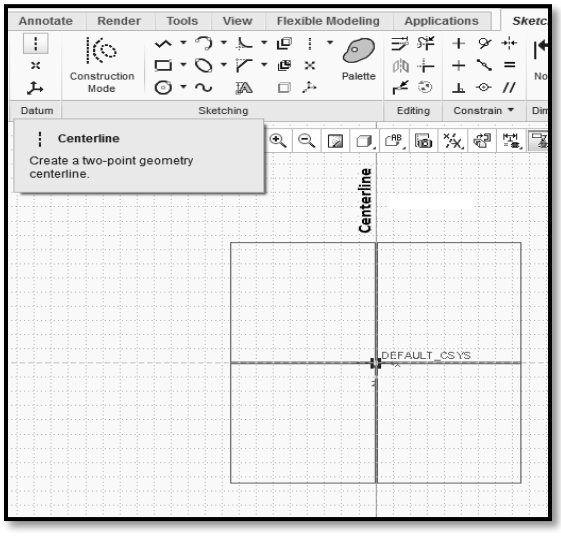
9. Saving your work

- In the Quick Access toolbar, click **Save**  to save your model.
- In the **Save Object** dialog, click **OK** to specify that the model will be saved to your working directory.

Step 6: Create a sketch to define the shape of Tommy Bar of the Screw Jack.

1. Draw a Centerline 

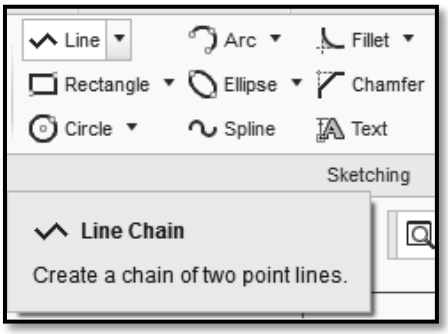
by picking from 'Datum' group. Place the centerline by picking two co-linear points as shown in Figure.



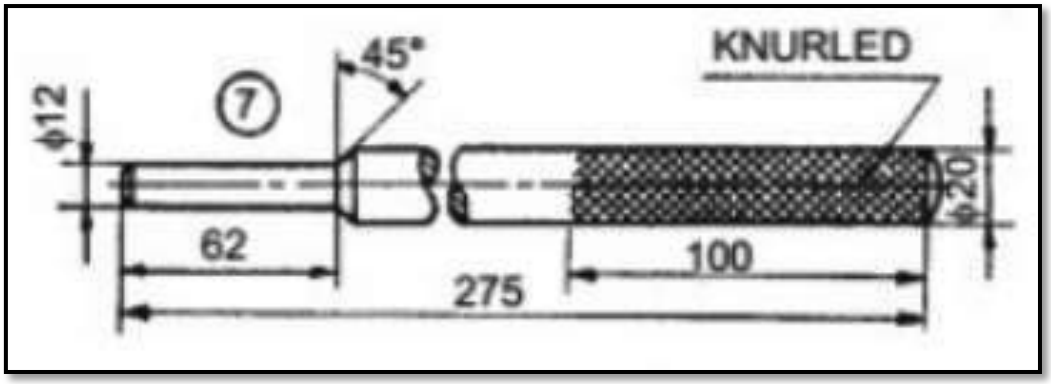
Construction of centerline

2. Create a profile shown in the Figure with the help of Line chain 

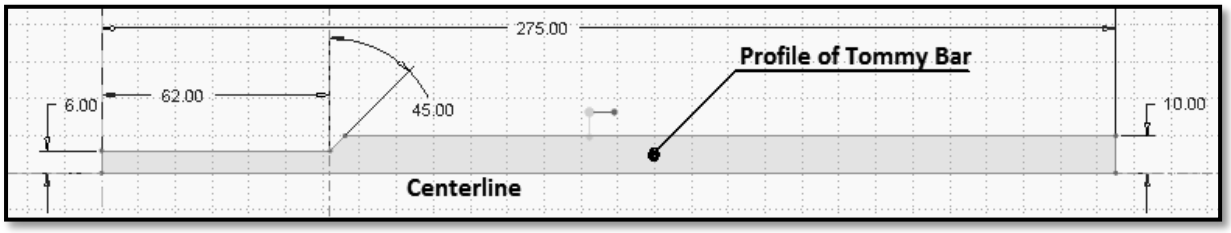
command from sketching group.



Line Dashboard.

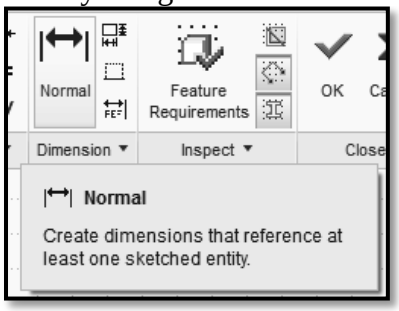


View of Tommy Bar of Screw Jack.



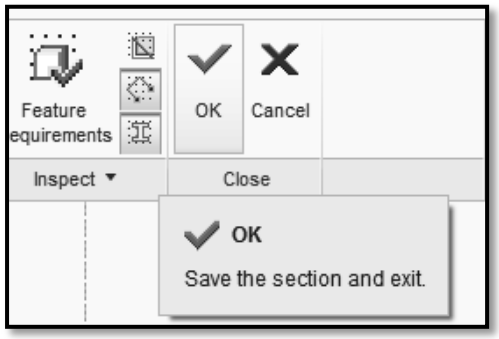
Profile of Tommy Bar of Screw Jack without rounds at the end.

3. Create and Edit the dimensions by using Normal  from Dimension group.



Dashboard of Normal Dimension.

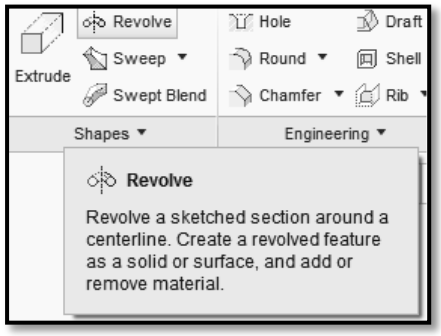
4. Click *OK*  from the Close group of the Sketch tab to complete the sketch and return to the *Revolve / Extrude* dashboard.



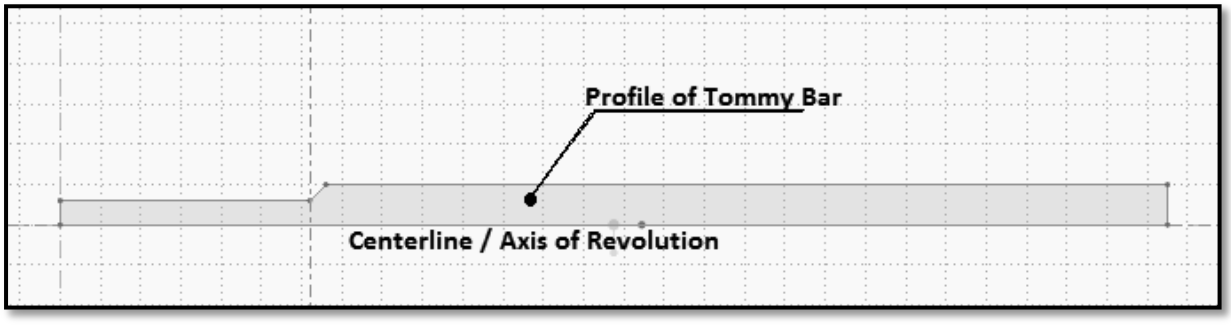
Dashboard of Accept.

5. Then click on **Revolve**  command from Shapes group then select axis / centerline.

Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.



Dashboard of Revolve command.

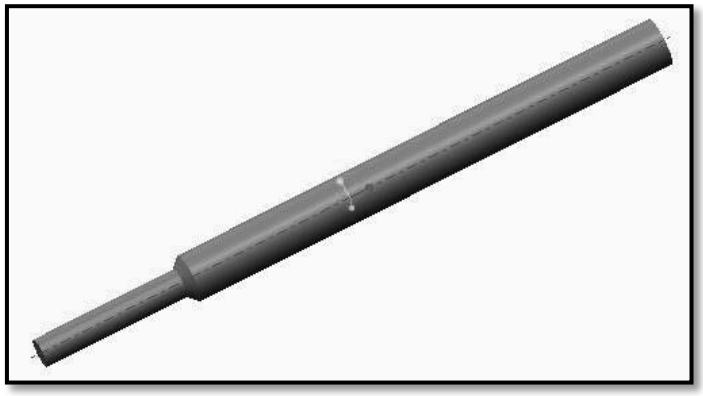


Axis of revolution for Tommy Bar.

6. Take revolution angle as 360°. And click on Accept  button.

Angle of rotation and Accept Dashboard.

We will get Tommy Bar as shown in the Figure.



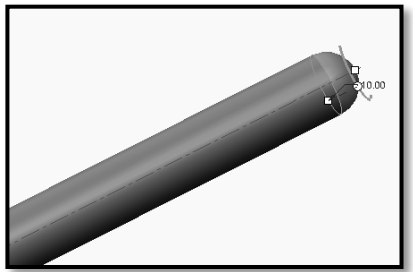
Model of Tommy Bar without Rounds.

7. Then provide the fillets / rounds with the help of **Round** 

command from Engineering group at required corners / edges and specify respective radius.

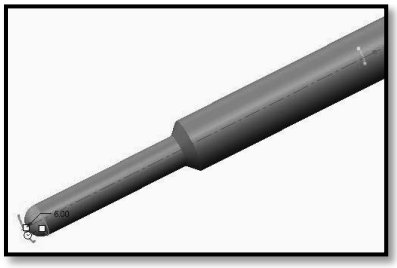
Round command dashboard.

8. Specify the radius of round as 10mm. after specifying the value of radius click on Accept button.



Radius dashboard.

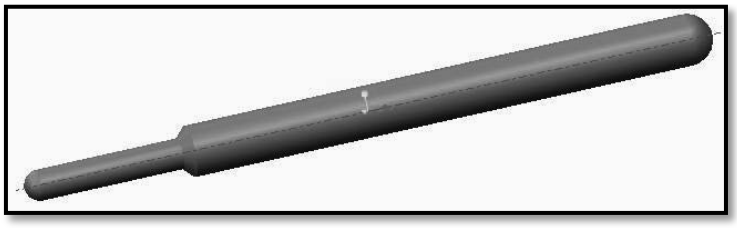
9. Specify the radius of round at other end as 6mm. After specifying the value of radius click on Accept button.



(b)

Radius dashboard.

We will get final shape of Tommy Bar with Rounds as shown in the Figure.



Tommy Bar of Screw Jack with Rounds.

10. Saving your work

- In the Quick Access toolbar, click **Save**  to save your model.
- In the **Save Object** dialog, click **OK** to specify that the model will be saved to your working directory.

XVI. Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XVII. Actual Procedure Followed

.....

.....

.....

XVIII. Precautions Followed

.....

.....

.....

.....

XIX. Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Explain the procedure to construct a Center Rectangle.
2. Explain the Chamfering procedure.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XX. References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=XlsaSe444AE
2. https://www.youtube.com/watch?v=b6b9FY14PKw
3. https://www.youtube.com/watch?v=X0AMdUMNsDI

XXI. Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.9: Assemble and Print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials.

I. Practical Significance

To create a 3D feature assemblies of relative components. Just as you can combine features into parts, you can also combine parts into assemblies. Any Modeling Parametric CAD Software enables you to place component parts and subassemblies together to form assemblies. It also helps you modify, analyze, or reorient the resulting assemblies. Such a virtual designed models and assemblies can be used to easily visualize and evaluate your design before costly prototypes are manufactured.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills for the industry identified competency –

- Create new Modeling Parametric assembly.
- Assemble the components of an assembly using the Default constraint.
- Assembly constraints – Automatic, fully constrained.

IV Relevant Course Outcome(s)

- Prepare assemblies of part models using assembly workbench of any parametric CAD software.

V Practical Outcome

- Use any available parametric CAD modeling software to draw and assemble the different parts at their respective working position for engineering products.

VI Relative Affective Domain-

- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Methods for Assembly.
- 3-D Dragger.
- Position and relationship in between the parts to be assembled.

- Bill of materials.
- Basic knowledge of geometric constructions.
- Constrain Components.

Methods of Assembly

There are two main approaches of assembly in CAD modeling software –

1. Bottom-up Assembly.
2. Top-down Assembly.

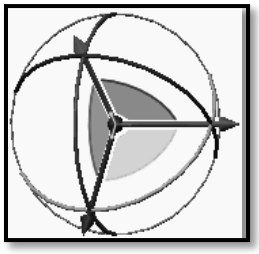
The **bottom-up** approach is the traditional approach. In **bottom-up assembly**, designing / modeling of parts done separately and then call them in assembly environment and apply constraints for them.

In **top-down assembly,**designing / modeling and assembly of the parts are done in assembly environment itself and no need to constrain top down assembly parts.

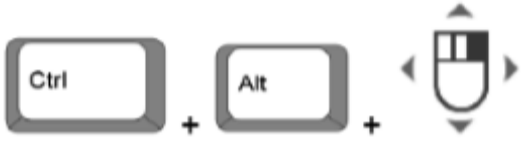
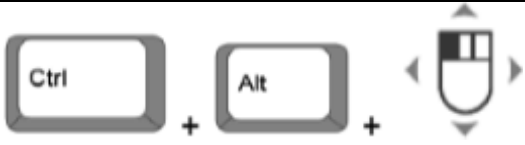
3-D Dragger : -

The color coded 3-D dragger is used to orient the component being assembled within the assembly. As constraints are added and the degrees of freedom are reduced, you will notice that those corresponding portions of the dragger are grayed out.

- The shaded arcs of the dragger control rotation about the three axes.
- The shaded arrows translate the component along those axes.
- These small translucent (shining) quadrants move the component in a 2-D plane.
- The small sphere at the center is used to pan the component in any direction.



3D-Dragger.

Operation	Keyboard and Mouse Selection
Spin – The component will spin within the assembly. Partially constrained components only spin in unconstrained directions.	
Pan – The component will pan about the assembly. Partially constrained components only spin in unconstrained directions.	
Component Drag – The component will spin and pan about the assembly. Partially constrained components can be dragged only in unconstrained directions.	

Constrain Components

After you have placed and oriented a component it is important that you add assembly constraints to define its final design position.

- A **Coincident**  constraint is applied to cylindrical surface of each part. This constraint type makes the center axis of each model coincident.

- A second **Coincident**  constraint is applied to datum planes FRONT of each model. Making the two planes coincident.

- Constraints such as **Coincident**  , **Parallel**  , **Distance**  and so on can be explicitly selected from the constraint drop-down menu in the dashboard under **Automatic**, however, it is often easier to let Solid Modeling Parametric select them based on the references you select. In this case, selecting the two cylindrical surfaces caused a **Coincident**  constraint to be automatically applied. If the planes were farther apart when selected, a **Distance**  constraint may have been applied.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

IX Precautions to be Followed

1. Units used while designing the individual models and units used while assembly should same. (eg. mm or inch).
2. While assembling first part (base part), assemble the first component at default constraint location.
3. Each and every part, which are get assembled at respective working position should fully constrained.

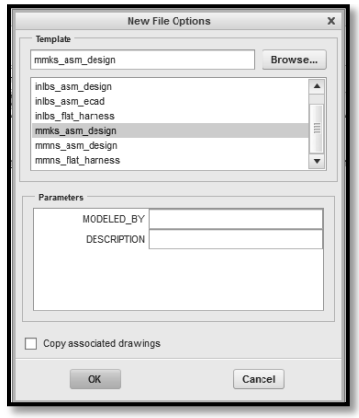
X Procedure

Step 1: Set working directory and create a new assembly

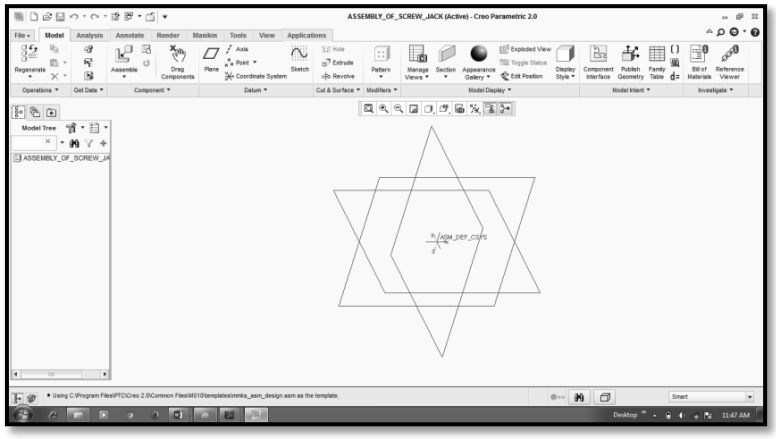
1. Start Solid Modeling Parametric.
2. Set the working directory as explained in earlier practical's.
3. Creating the new assembly model:
   - From the Quick Access toolbar or **Home** tab, click **New**.
   - In the New dialog box, click to select Type as **Assembly**and Sub-type as **Design** as the new model type.
   - Type **Assembly_of_Screw_Jack** in the **Name** field, uncheck the 'Use default template' and click **OK**.

New dialog box.

   - A 'New File Options' dialog box will pop up, from this dialog box select 'mmns_asm_design' OR 'mmks_asm_design' option and click OK.



New File Options dialog box.

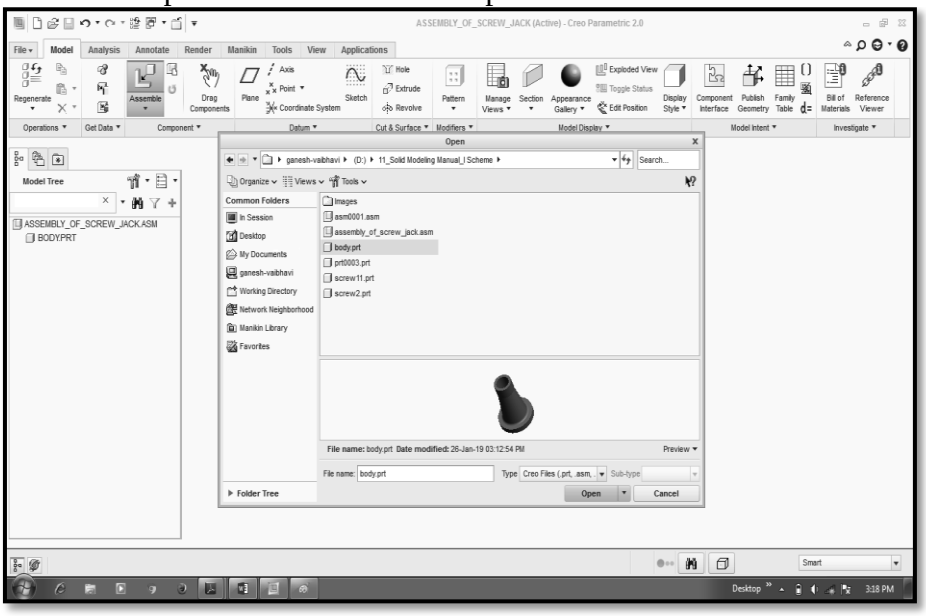


Assembly Environment.

Step 2: Adding the first component to the assembly

1. Selecting the component to assemble:

- Click 'Assemble' tool from Component group of Model tab.
- In the lower-right corner of the Open dialog box, click to expand the Preview pane.
- Browse the location where you have saved Screw Jack **Body** model and select it.
- Click Open to assemble this component.



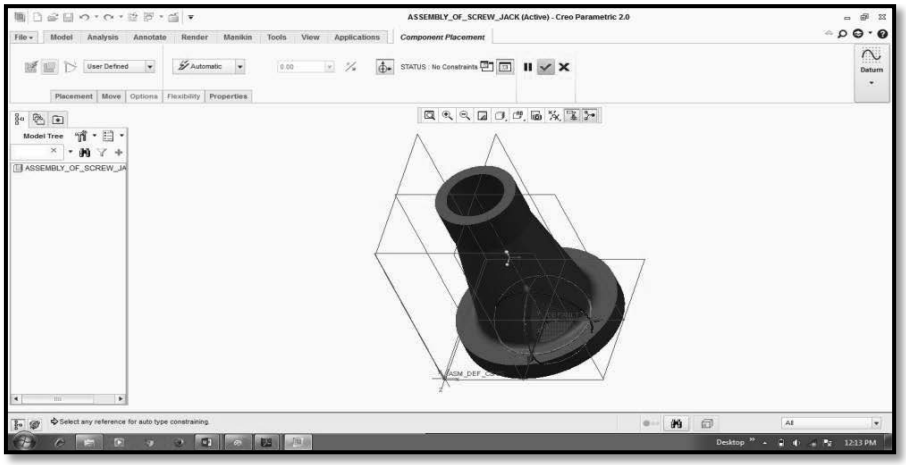
Calling of First object i.e. Screw Jack Body.

- The part will be attached to the cursor and the Assembly dashboard will open.

Assembly dashboard.

Locating the part temporarily, before final placement:

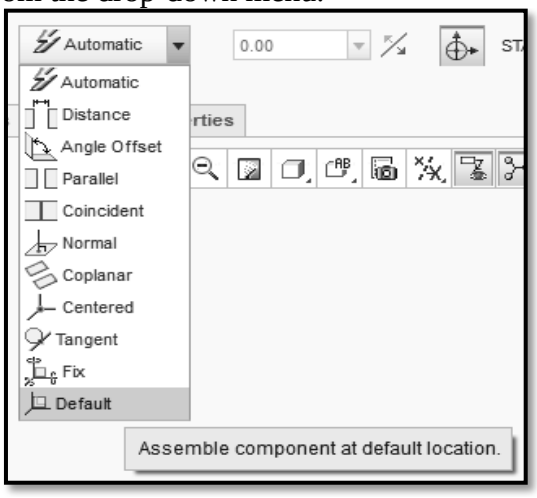
Drag the Screw Jack Body just to the left of the assembly coordinate system, and then click in the graphics area to place it. Later, when placing components, you will use the 3D Dragger to position the component close to its final destination.



Location of part before final placement.

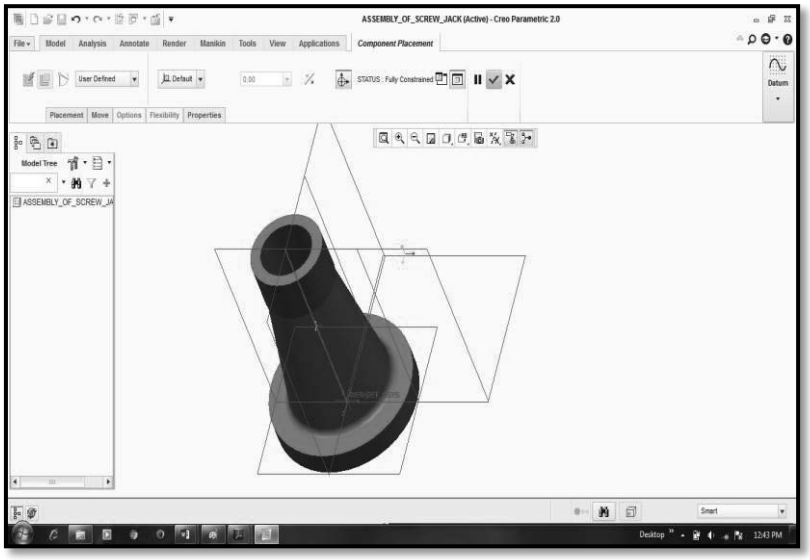
2. Adding assembly constraints:

- In the Assembly dashboard, click Automatic and select **Default** from the drop-down menu.



Default location.

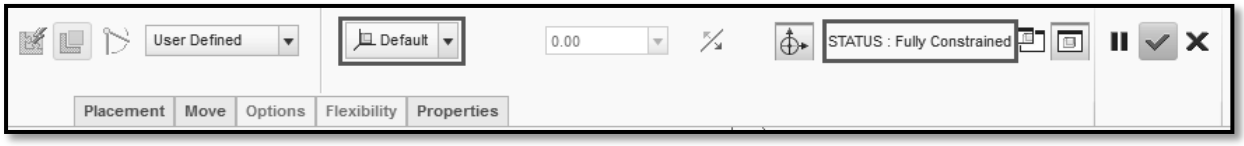
The Screw Jack model is now constrained to the default center of the graphics area, where the assembly coordinate system is located.



Screw Jack body constrained to default center.

Components change to a yellow-orangecolor after they have been fully constrained.

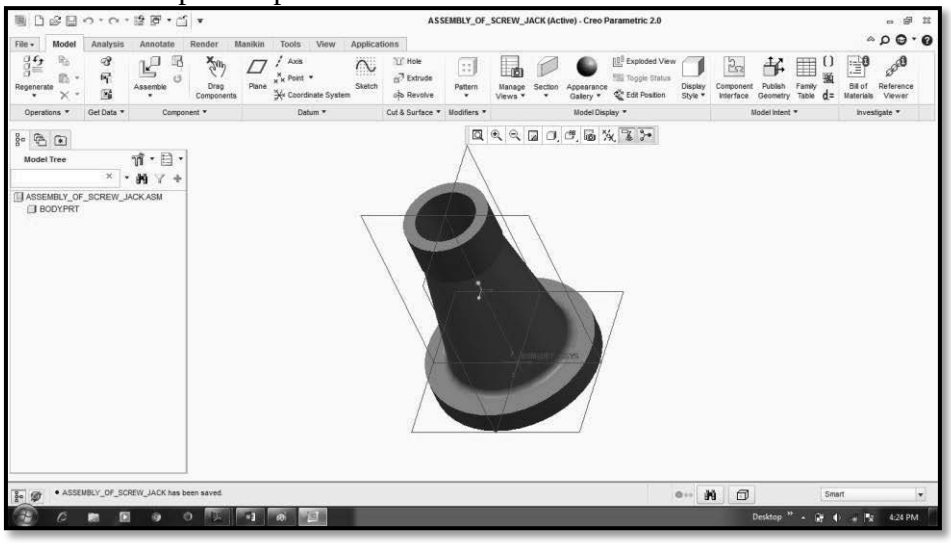
The Assembly dashboard shows the **Default** constraint type message confirms the part is **Fully Constrained.**



Default and Fully Constrained dashboard.

3. Complete the placement of the part:

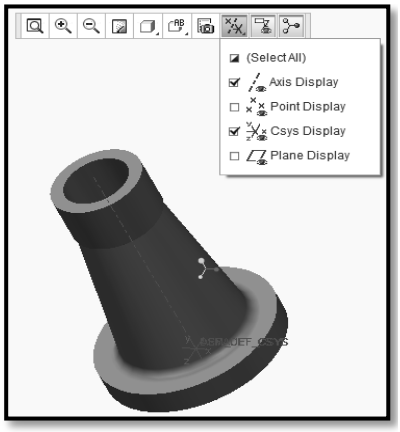
- In the Assembly dashboard, click **Complete Component**  to complete the component placement.



Placement of Base component (body).

4. Click **Save**  to save your work.

5. Change the display of datum features:
   - In the Graphics toolbar, disable the display of Point display and Plane display.
   - Default coordinate system and axis display should be displayed at the center.

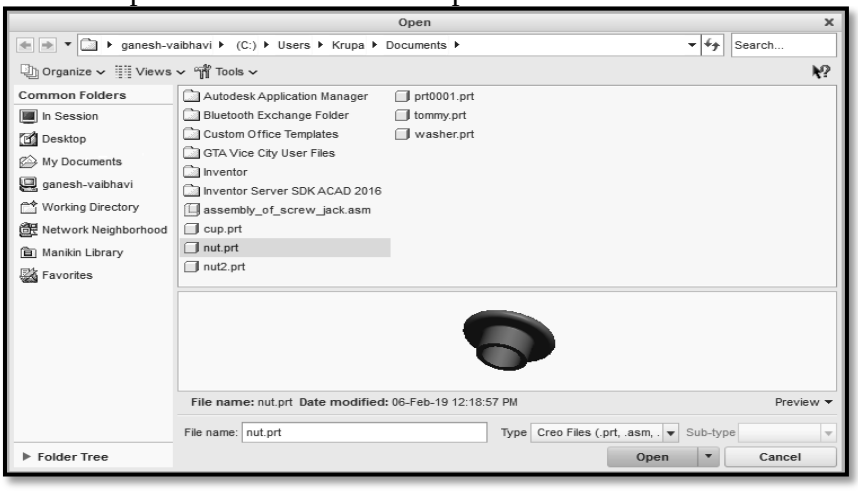


Datum features with Base component.

Step 3: Add the Second component (Nut) to the assembly

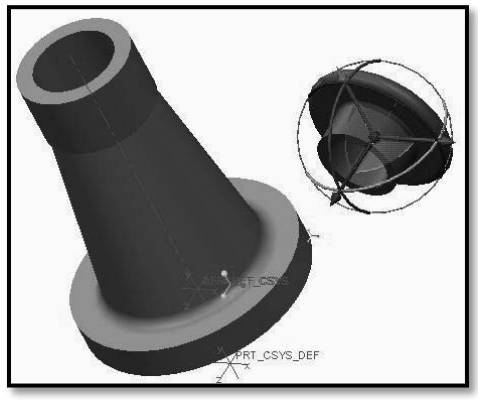
1. Selecting the Second part to assemble:

   - Click 'Assemble'  tool from Component group of Model tab.
   - In the lower-right corner of the Open dialog box, click to expand the Preview pane.
   - Browse the location where you have saved **Nut** model and select it.
   - Click Open to assemble this component.

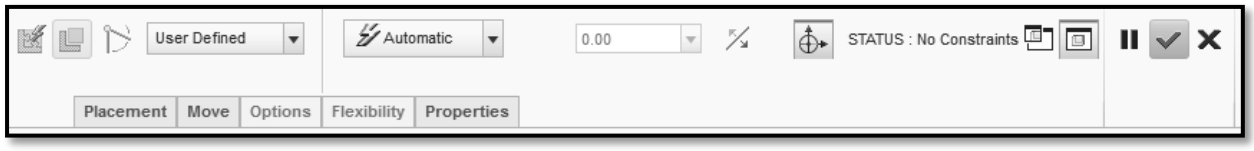


Calling of Second object i.e. Nut.

   - The part will be attached to the cursor and the Assembly dashboard will open.



Nut with base component (body).



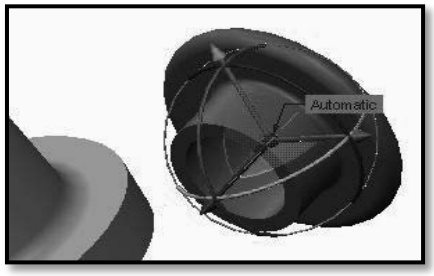
Automatic and No Constraints dashboard.

2. Locating the part temporarily, before final placement:
   - Drag the Nut to a position just to the right of the Body, and then click in the graphics area to place it.

3. Adding the first assembly constraint:
   - Move the cursor over the axis of Nut till it shows the cursor tip as shown atX1.

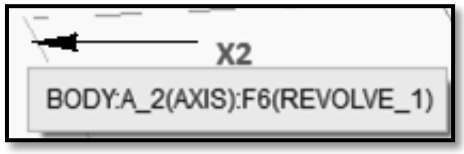
First Assembly Constraint i.e. Axis of Nut.

   - When the cursor shows cursor tip shown at X1, click to select the axis. Then it displays **Automatic**.



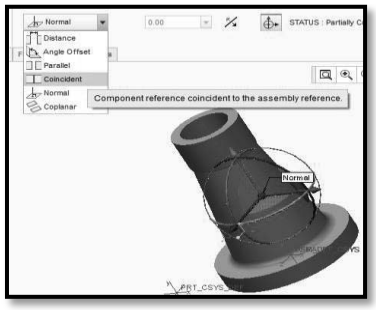
Selection of First Assembly Constraint i.e. Axis of Nut.

   - Move the cursor over the axis of the body till it shows the cursor tip as shown at X2. Then click to select the axis of body.



Second Assembly Constraint i.e. Axis of Body.

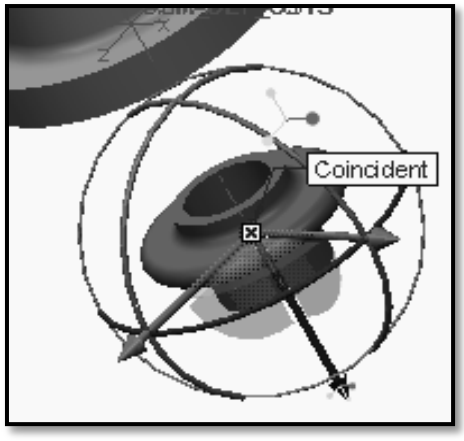
- Then select the **Coincident** constraint from the **Component Placement > Constraint dashboard.**



Application of Coincident Constraint.

- If required drag and click on **Flip Axis Constraint** from the **Placement** tab in order to flip the orientation of a constraint.

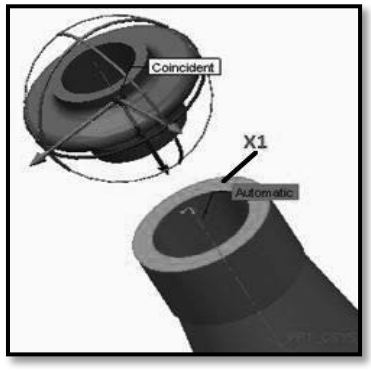
Before Flip Axis Constraint.



After Flip Axis Constraint.

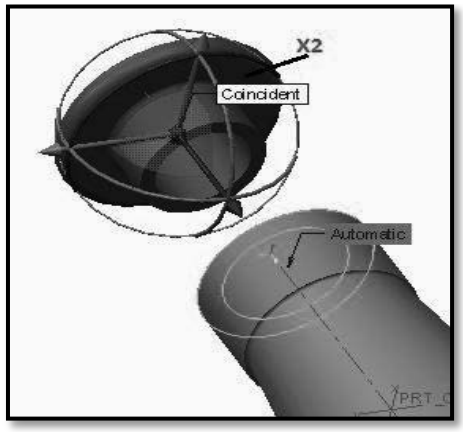
- Then drag the Nut towards upper side of the body.

4. Adding the second assembly constraint:
   - Press the middle-mouse button and drag to spin the model until you can see the flat surface of body shown as X1.
   - Click to select the flat surface of the Body(X1) that it closest to and facing the surface of Nut X2.



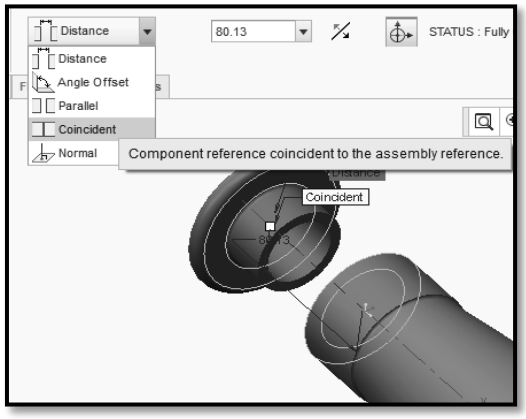
Second Assembly Constraint i.e. Surface of Body (X1).

- Press the middle-mouse button and drag to spin the model until you can see the flat surface shown as X2 of the Nut.



Selection of Second Assembly Constraint i.e. Surface of Nut (X2).

- Click to select the flat surface of Nut shown as X2.
- Then from Constrained dashboard select coincident constraint.



Selection of Coincident Constraint.

- Solid Modeling Parametric recognizes two flat surfaces facing each other and applies a Coincident constraint. These two selected surface are now coincident to each other.

Coincident both the parts.

- The Nut has changed to a **yellow-orange** color indicating that its position is fully constrained.
- The Assembly dashboard shows the Coincident constraint type was the last used and that the Nut is now Fully Constrained.

Coincident Assembly constraint with Fully Constrained dashboard.

- Click **Complete Component**  to complete the component placement.
- Then the Nut returns to its original gray color.

5. Reorienting and saving your work:
   - Press CTRL + D to reorient the model.
   - Click Save  to save your work.

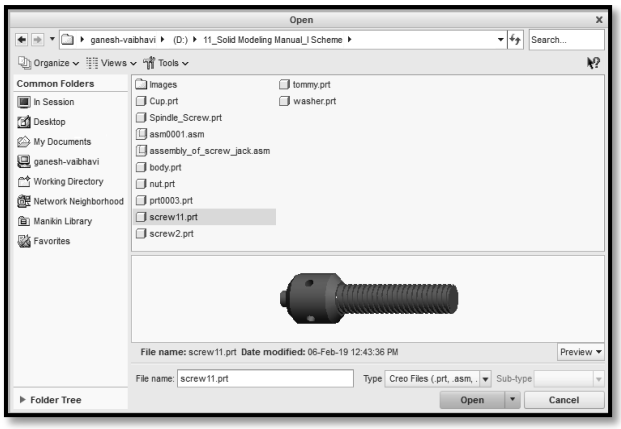
Reoriented body and nut assembly.

Step 3: Adding the Next component to the assembly (i.e. Screw Spindle)

1. Selecting the Next part to assemble:

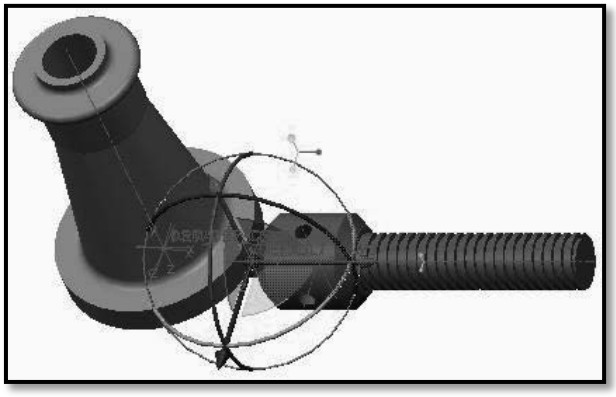
- Click ‘Assemble’ 

tool from Component group of Model tab.
- In the lower-right corner of the Open dialog box, click to expand the Preview pane.
- Browse the location where you have saved**Screw Spindle** model and select it.
- Click Open to assemble this component.



Calling of Next object i.e. Screw Spindle.

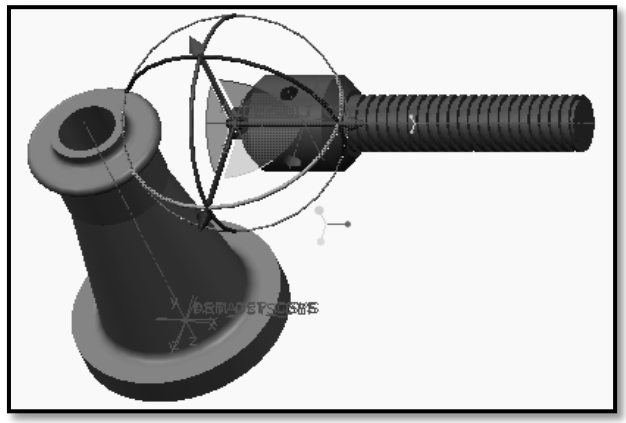
- The part will be attached to the cursor and the Assembly dashboard will open.



Screw Spindle with body and nut.

Automatic and No Constraints dashboard.

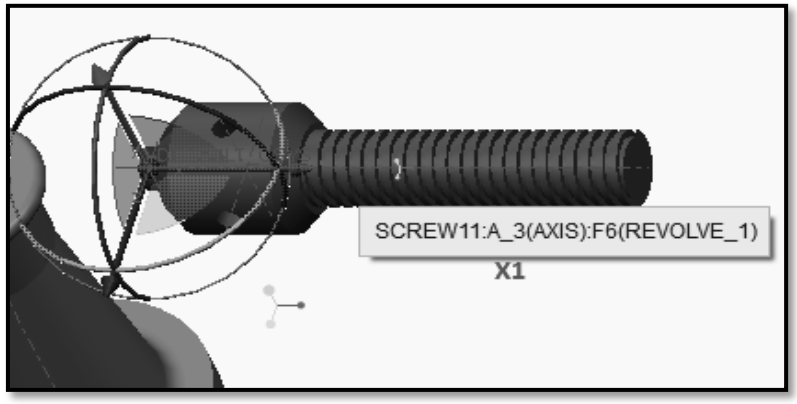
2. Locating the part temporarily, before final placement:
   - Drag the Screw Spindle to a position just to the right of the Assembly, and then click in the graphics area to place it.



Screw Spindle towards right side of assembly.

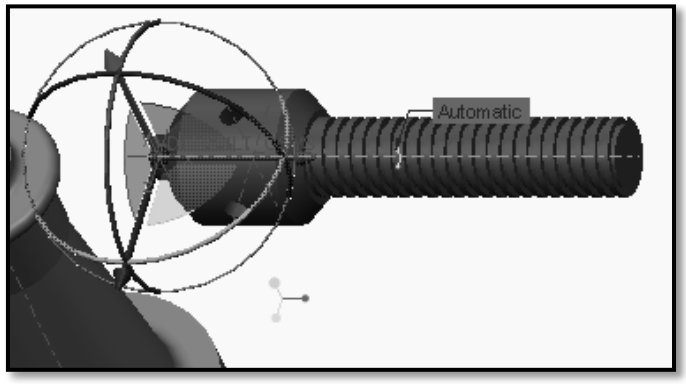
3. Adding the first assembly constraint:

- Move the cursor over the axis of Screw Spindle till it shows the cursor tip as shown at X1.



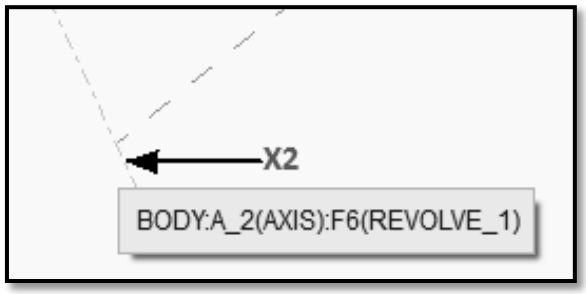
First Assembly Constraint i.e. Axis of Screw Spindle.

- When the cursor shows cursor tip shown at X1, click to select the axis. Then it displays **Automatic**.



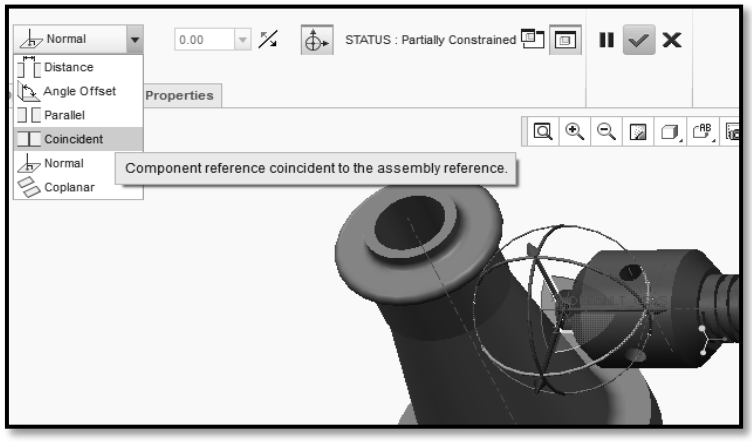
Selection of First Assembly Constraint i.e. Screw Spindle.

- Move the cursor over the axis of the body till it shows the cursor tip as shown at X2. Then click to select the axis of body.



Second Assembly Constraint i.e. Axis of Body.

- Then select the **Coincident** constraint from the **Component Placement** > **Constraint dashboard.**



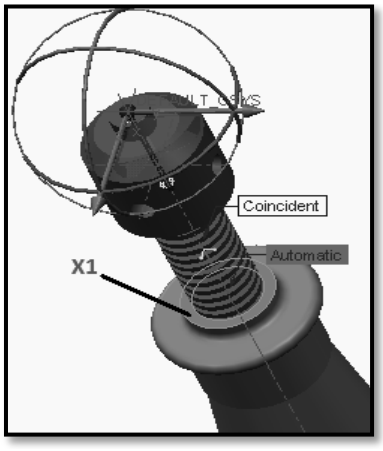
Before Application of Coincident Constraint.

After Application of Coincident Constraint.

- If required click on **Flip Axis Constraint** from the **Placement** tab in order to flip the orientation of a constraint.
- Then drag the Screw Spindle towards upper side of the Assembly.

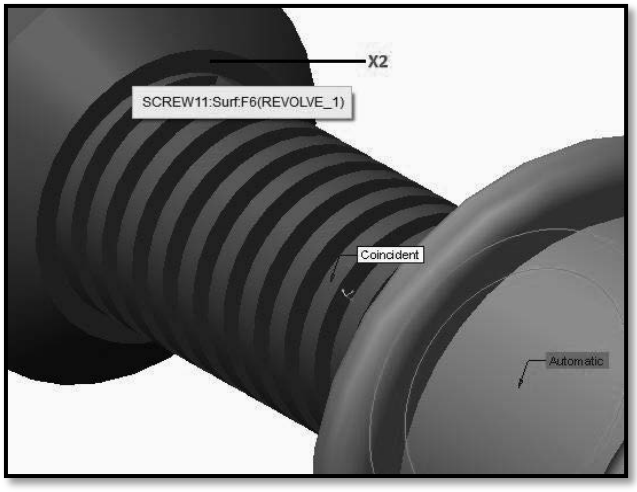
Dragging of Screw Spindle along the constrained axes.

4. Adding the second assembly constraint:
   - If required press the middle-mouse button and drag to spin the model until you can see the flat surface shown as X1 on the Nut.
   - Click to select the flat surface of the Nut(X1) that it closest to and facing the surface of Screw Spindle(X2).



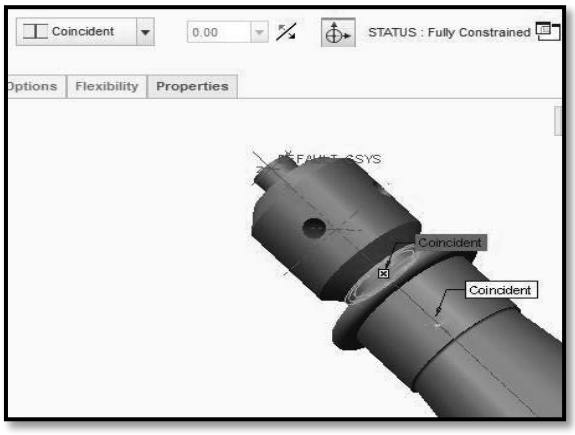
Second Assembly Constraint i.e. Surface of Nut (X1).

   - Press the middle-mouse button and drag to spin and zoom the model until you can see the flat surface shown as X2 of the Screw Spindle.



Selection of Second Assembly Constraint i.e. Surface of Screw Spindle (X2).

- Click to select the flat surface of Screw Spindle shown as X2.
- If required, then from Constrained dashboard select coincident constraint.



Coincident both the parts.

- Solid Modeling Parametric recognizes two flat surfaces facing each other and applies a Coincident constraint. These two selected surface are now coincident to each other.
- The Screw Spindle has changed to a **yellow-orange** color indicating that its position is fully constrained.
- The Assembly dashboard shows the Coincident constraint type was the last used and that the Screw Spindle is now Fully Constrained.

Coincident Assembly constraint with Fully Constrained dashboard.

- Click **Complete Component**  to complete the component placement.
- Then the Screw Spindle returns to its original gray color.

5. Reorienting and saving your work:
   - Press CTRL + D to reorient the model.
   - Click Save  to save your work.

Reoriented Assembly.

XI Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XII Actual Procedure Followed

.....
.....
.....
.....
.....

XIII Precautions Followed

.....
.....
.....
.....
.....

XIV Course Proficiency

.....
.....
.....
.....
.....

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Create models of individual components and Assemble any one of the following assembly – Bench vice ,Drill Jig.
2. List different assembly constraints.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XVI References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=xzteh5MFDs4
2. https://www.youtube.com/watch?v=Yp2SbrxhfNQ
3. https://www.youtube.com/watch?v=nbhQTShOS0o
4. https://www.youtube.com/watch?v=mFNZHaQYw60

XVII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

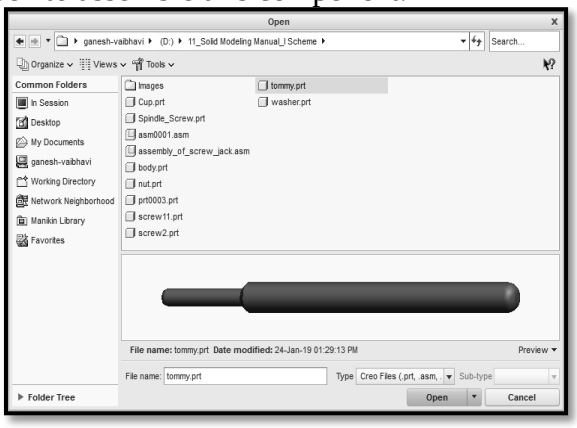
Practical No.10: Assemble and Print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials.

---------- Continued From Previous Practical –

Step 4: Adding the Next component to the assembly (i.e. Tommy bar)

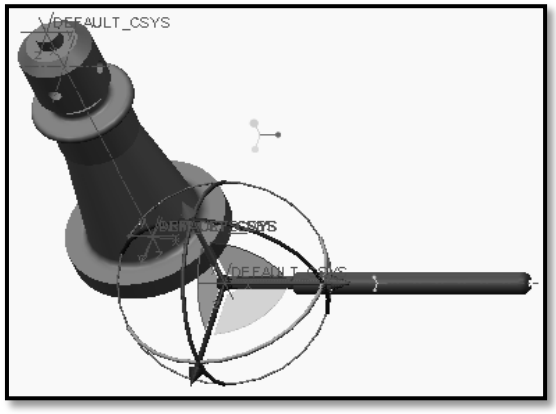
1. Selecting the Next part to assemble:

- Click ‘Assemble’ tool from Component group of Model tab.
- In the lower-right corner of the Open dialog box, click to expand the Preview pane.
- Browse the location where you have saved**Tommy bar** model and select it.
- Click Open to assemble this component.



Calling of Next object i.e. Tommy bar.

- The part will be attached to the cursor and the Assembly dashboard will open.

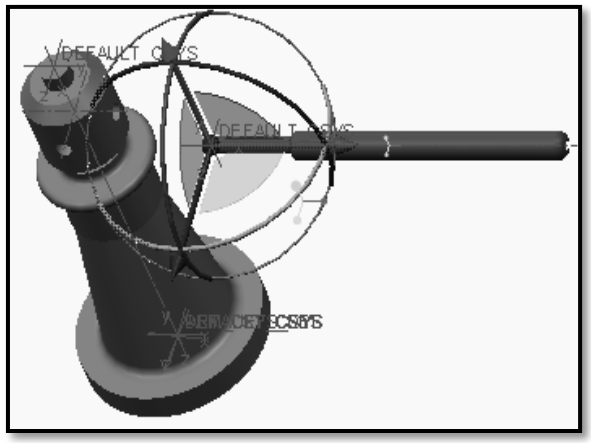


Tommy bar with Assembly.

Automatic and No Constraints dashboard.

2. Locating the part temporarily, before final placement:

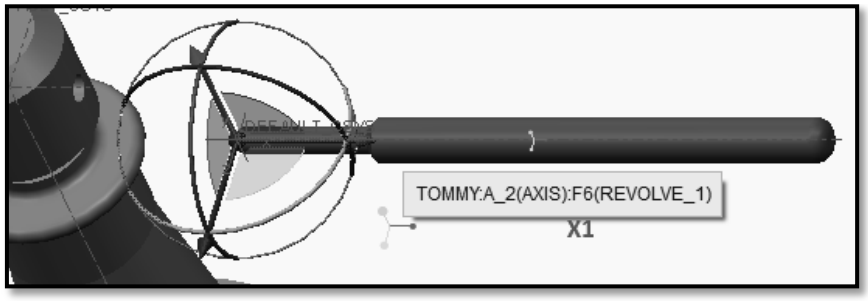
- Drag the Tommy bar to a position just to the right of the Assembly, and then click in the graphics area to place it.



Tommy bar towards right side of assembly.

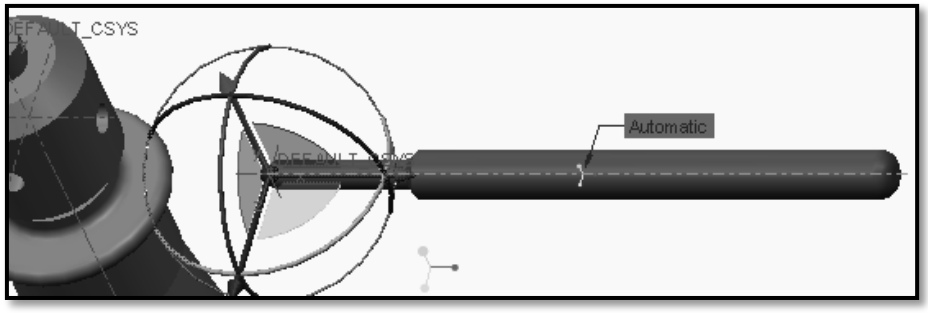
3. Adding the first assembly constraint:

- Move the cursor over the axis of Tommy bar till it shows the cursor tip as shown at X1.



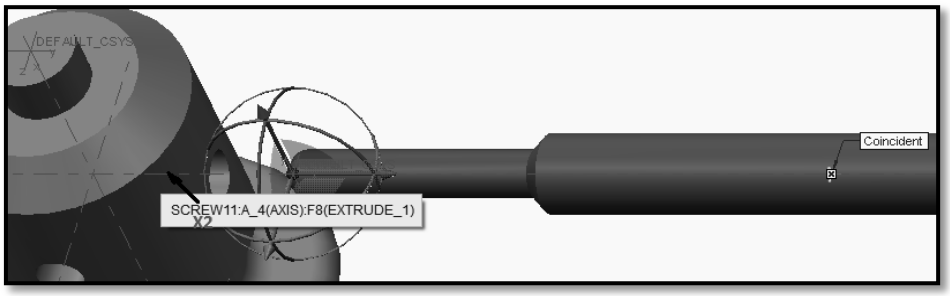
First Assembly Constraint (X1) i.e. Axis of Tommy bar.

- When the cursor shows cursor tip shown at X1, click to select the axis. Then it displays **Automatic**.



Selection of First Assembly Constraint (X1) i.e. Axis of Tommy bar.

- Move the cursor over the axis of the Screw Spindle till it shows the cursor tip as shown at X2. Then click to select the axis of Screw Spindle.

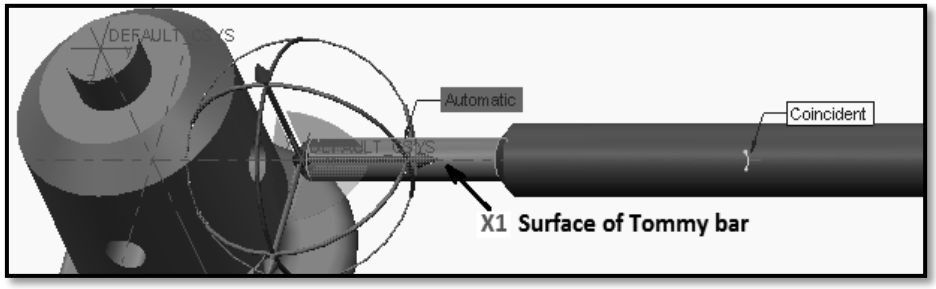


Second Assembly Constraint (X2) i.e. Axis of Screw Spindle.

- Then if required, select the **Coincident** constraint from the **Component Placement > Constraint dashboard.**
- If required click on **Flip Axis Constraint** from the **Placement** tab in order to flip the orientation of a constraint.
- Then if required, drag the Tommy bar towards the hole of screw spindle.

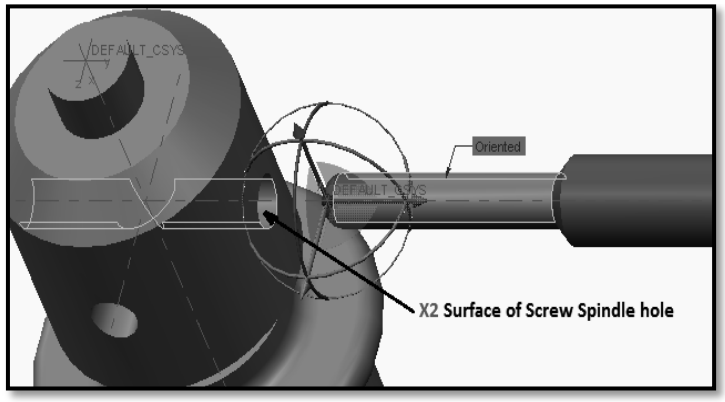
4. Adding the second assembly constraint:

- If required press the middle-mouse button and drag to spin the model until you can see the flat surface shown as X1 on the Tommy bar.
- Click to select the flat surface of the Tommy bar(X1) that it closest to and facing the surface of Screw Spindlehole (X2).



Second Assembly Constraint i.e. Surface of Tommy bar (X1).

- Press the middle-mouse button and drag to spin and zoom the model until you can see the flat surface shown as X2 of the Screw Spindle hole.

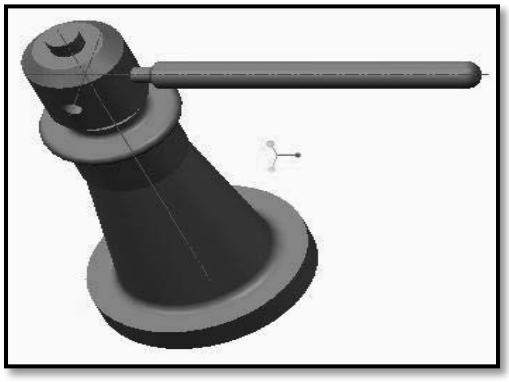


Selection of Second Assembly Constraint i.e. Surface of Screw Spindle hole (X2).

- Click to select the flat surface of Screw Spindle hole shown as X2.
- If required, then from Constrained dashboard select coincident constraint.

- Solid Modeling Parametric recognizes two cylindrical surfaces facing each other and applies a Coincident constraint. These two selected surface are now coincident to each other.
- Click **Complete Component** to complete the component placement.
- Then the Tommy bar returns to its original gray color.

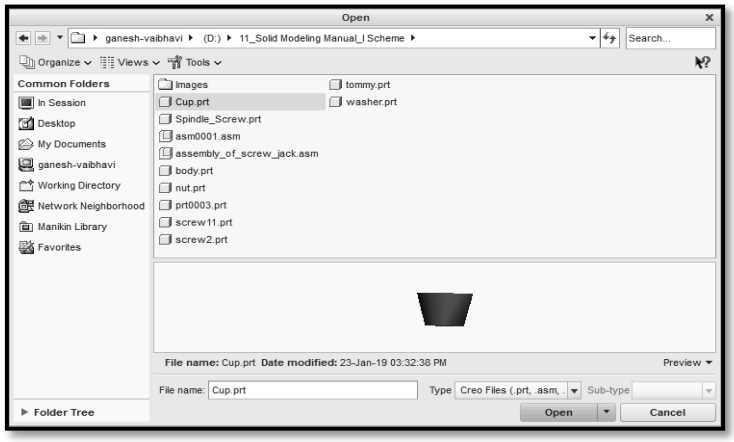
5. Reorienting and saving your work:
   - Press CTRL + D to reorient the model.
   - Click Save to save your work.



Reoriented Assembly.

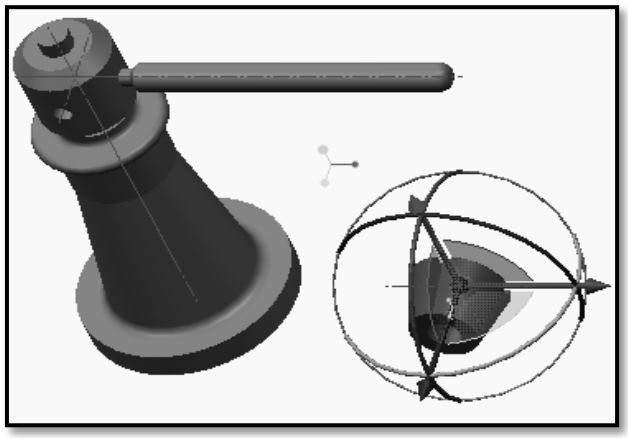
Step 5: Adding the Next component to the assembly (i.e. Cup)

1. Selecting the Next part to assemble:
   - Click 'Assemble' tool from Component group of Model tab.
   - In the lower-right corner of the Open dialog box, click to expand the Preview pane.
   - Browse the location where you have saved**Cup** model and select it.
   - Click Open to assemble this component.

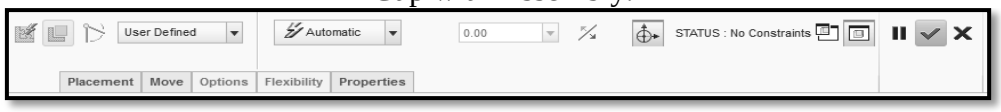


Calling of Next object i.e. Cup.

- The part will be attached to the cursor and the Assembly dashboard will open.

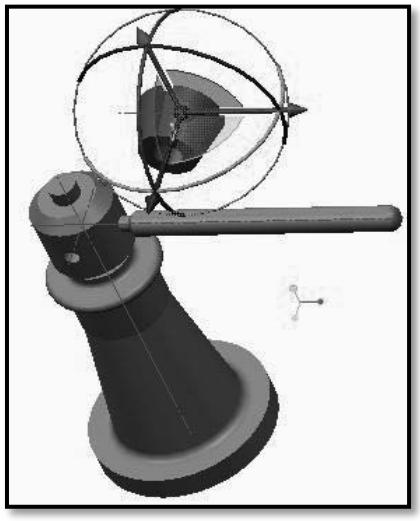


Cup with Assembly.



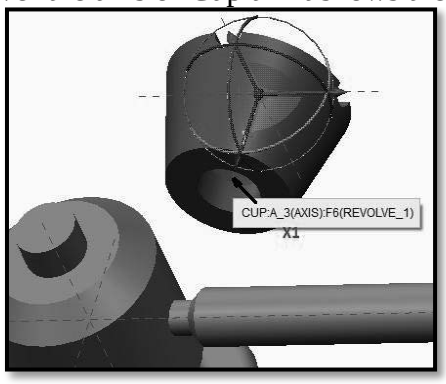
Automatic and No Constraints dashboard.

2. Locating the part temporarily, before final placement:
    - Drag the Cup to a position just to the top of the Assembly, and then click in the graphics area to place it.



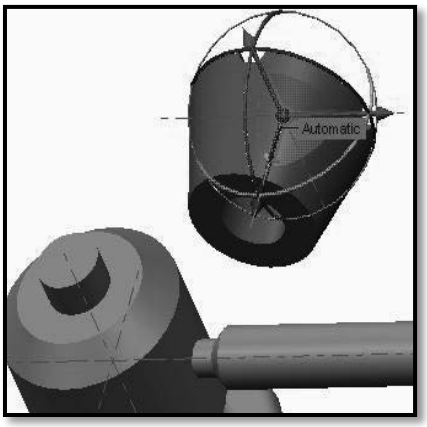
Cup towards Top side of assembly.

3. Adding the first assembly constraint:
    - Move the cursor over the axis of Cup till it shows the cursor tip as shown at X1.



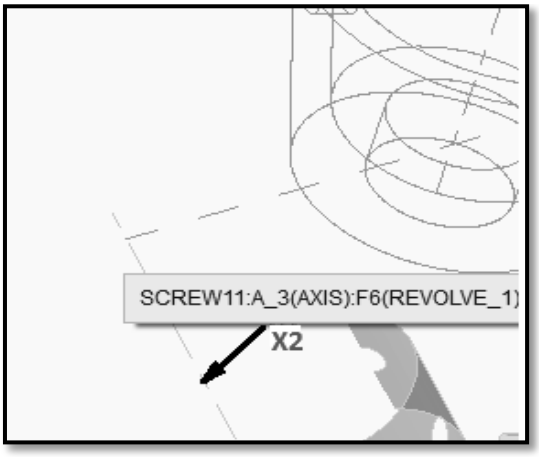
First Assembly Constraint (X1) i.e. Axis of Cup.

- Then the cursor shows cursor tip shown at X1, click to select the axis. Then it displays **Automatic**.



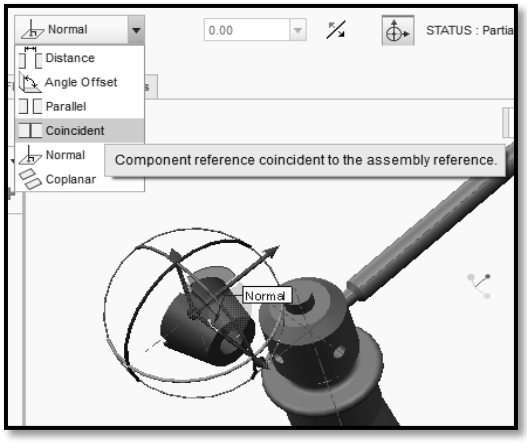
Selection of First Assembly Constraint (X1) i.e. Axis of Cup.

- Move the cursor over the axis of the Screw Spindle till it shows the cursor tip as shown at X2. Then click to select the axis of the Screw Spindle.

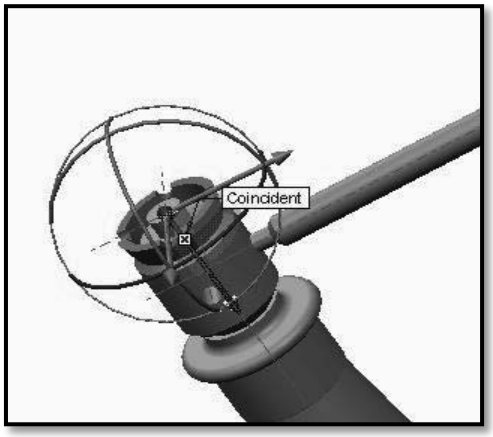


Second Assembly Constraint (X2) i.e. Axis of Screw Spindle.

- Then select the **Coincident** constraint from the **Component Placement > Constraint dashboard.**



Before Application of Coincident Constraint.

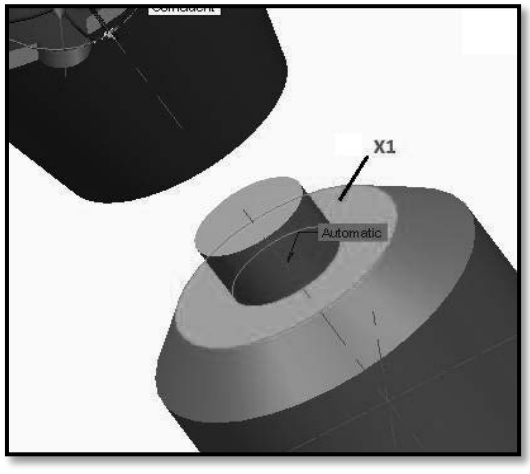


After Application of Coincident Constraint.

- If required click on **Flip Axis Constraint** from the **Placement** tab in order to flip the orientation of a constraint.
- Then drag the Cup towards upper side of the Assembly.

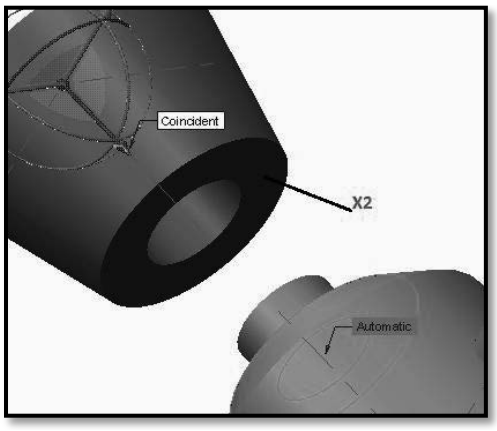
Dragging of Cup along the constrained axes.

4. Adding the second assembly constraint:
   - If required press the middle-mouse button and drag to spin the model until you can see the flat surface shown as X1 on the Screw Spindle.
   - Click to select the flat surface of the Screw Spindle (X1) that it closest to and facing the surface of Cup(X2).



Second Assembly Constraint i.e. Surface of Screw Spindle (X1).

- Press the middle-mouse button and drag to spin and zoom the model until you can see the flat surface shown as X2 of the Cup.



Selection of Second Assembly Constraint i.e. Surface of Cup (X2).

- Click to select the flat surface of Cup shown as X2.
- If required, then from Constrained dashboard select coincident constraint.

Coincident both the parts.

- Solid Modeling Parametric recognizes two flat surfaces facing each other and applies a Coincident constraint. These two selected surface are now coincident to each other.
- The Cup has changed to a **yellow-orange** color indicating that its position is fully constrained.
- The Assembly dashboard shows the Coincident constraint type was the last used and that the Cup is now Fully Constrained.

Coincident Assembly constraint with Fully Constrained dashboard.

- Click **Complete Component** to complete the component placement.
- Then the Cup returns to its original gray color.

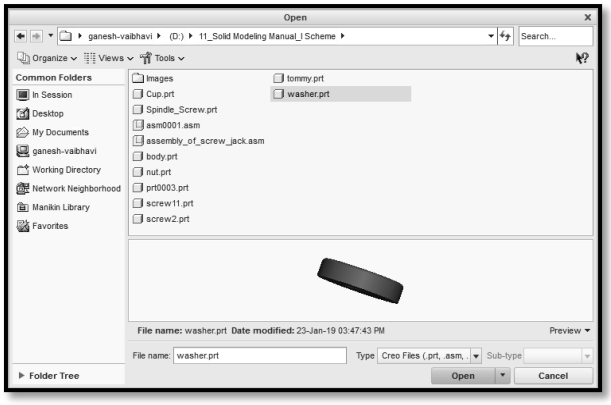
5. Reorienting and saving your work:
   - Press CTRL + D to reorient the model.
   - Click Save to save your work.

Reoriented Assembly.

Step 6: Adding the Next component to the assembly (i.e. Washer)

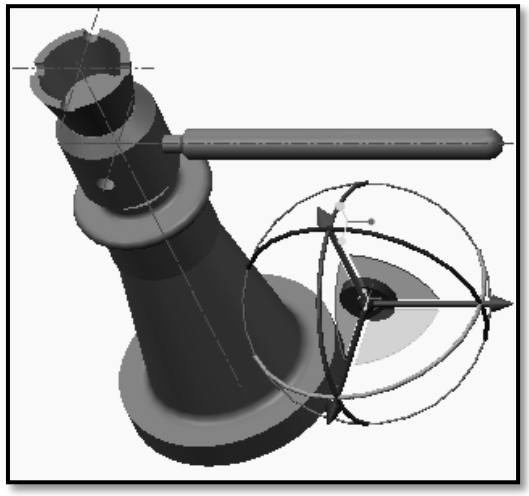
1. Selecting the Next part to assemble:

- Click 'Assemble' tool from Component group of Model tab.
- In the lower-right corner of the Open dialog box, click to expand the Preview pane.
- Browse the location where you have saved **Washer** model and select it.
- Click Open to assemble this component.



Calling of Next object i.e. Washer.

- The part will be attached to the cursor and the Assembly dashboard will open.

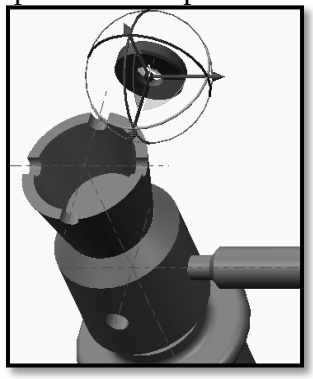


Washer with Assembly.

Automatic and No Constraints dashboard.

2. Locating the part temporarily, before final placement:

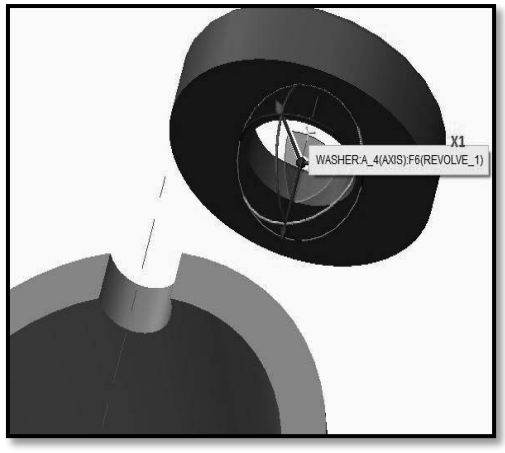
- Drag and zoom the Washer to a position just to the top of the Assembly, and then click in the graphics area to place it.



Washer towards Top side of assembly.

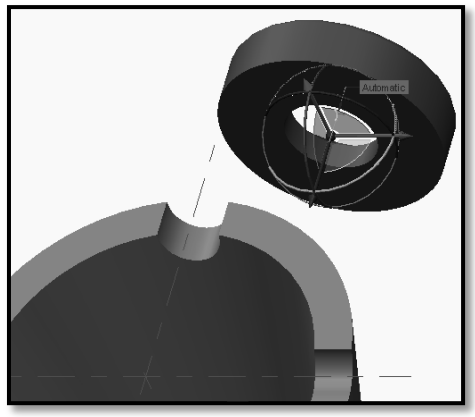
3. Adding the first assembly constraint:

- Zoom and move the cursor over the axis of Washer till it shows the cursor tip as shown at X1.



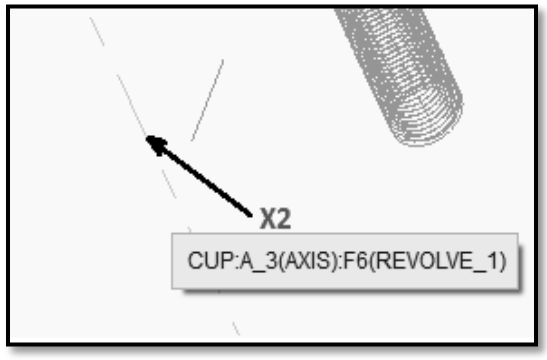
First Assembly Constraint (X1) i.e. Axis of Washer.

- When the cursor shows cursor tip shown at X1, click to select the axis. Then it displays **Automatic**.



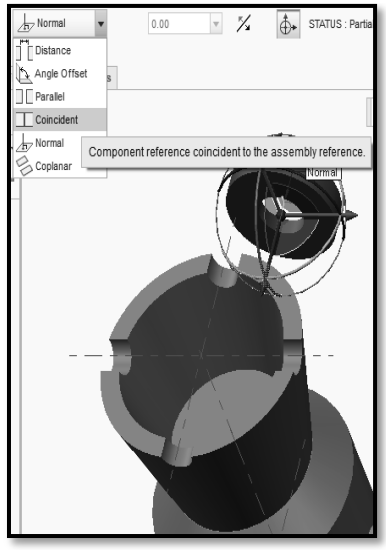
Selection of First Assembly Constraint (X1) i.e. Axis of Washer.

- Move the cursor over the axis of the Cup till it shows the cursor tip as shown at X2. Then click to select the axis of the Cup.

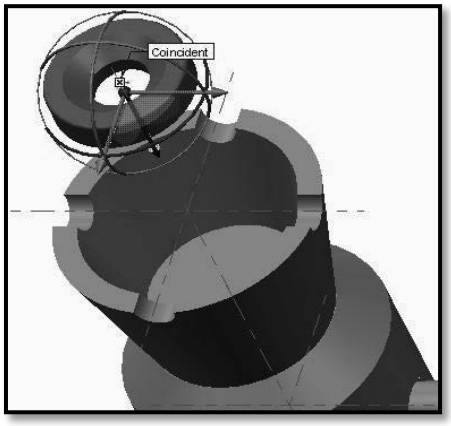


Second Assembly Constraint (X2) i.e. Axis of Cup.

- Then select the **Coincident** constraint from the **Component Placement > Constraint dashboard.**



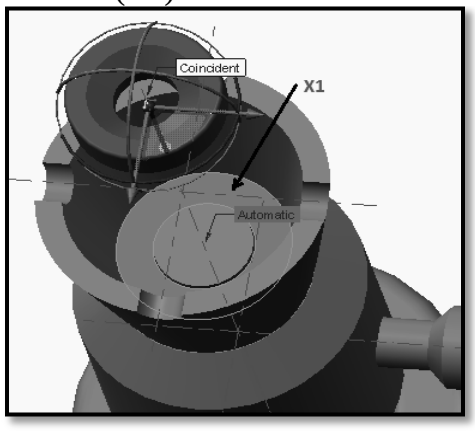
Before Application of Coincident Constraint.



After Application of Coincident Constraint.

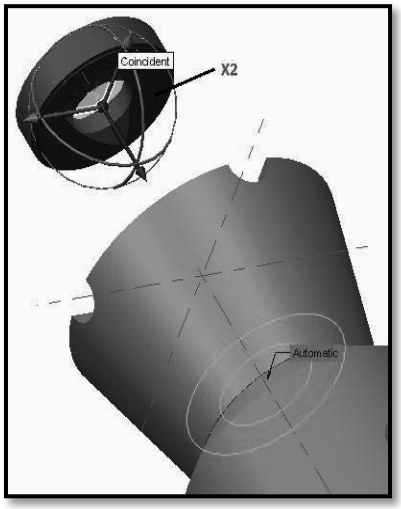
- If required click on **Flip Axis Constraint** from the **Placement** tab in order to flip the orientation of a constraint.
- Then drag the washer towards upper side of the Assembly.

4. Adding the second assembly constraint:
   - If required press the middle-mouse button and drag to spin the model until you can see the flat surface shown as X1 on the Cup base.
   - Click to select the flat surface of the Cup base (X1) that it closest to and facing the surface of Washer(X2).



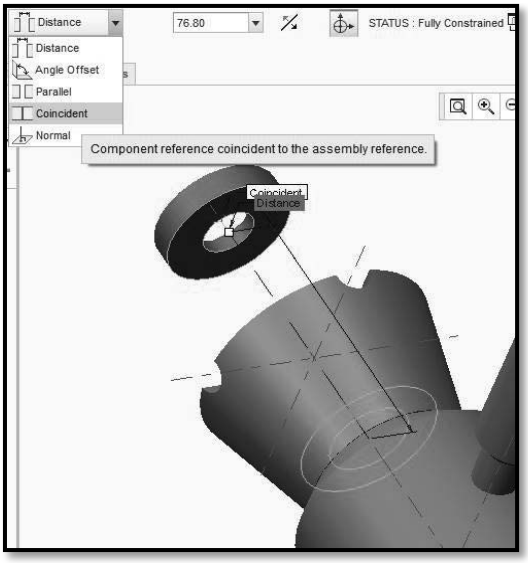
Second Assembly Constraint i.e. Surface of Washer (X1).

   - Press the middle-mouse button and drag to spin and zoom the model until you can see the flat surface shown as X2 of the Washer.

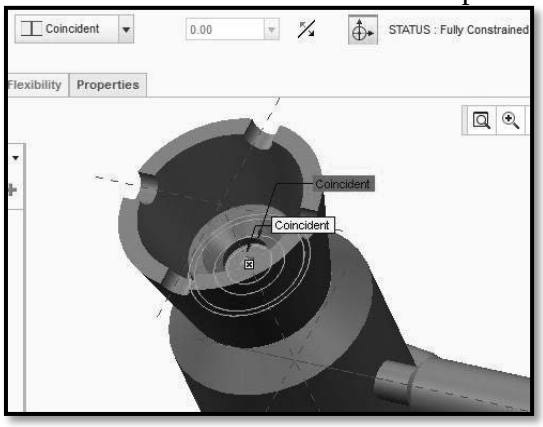


Selection of Second Assembly Constraint i.e. Surface of Washer (X2).

- Click to select the flat surface of Washer shown as X2.
- Then from Constrained dashboard select coincident constraint.



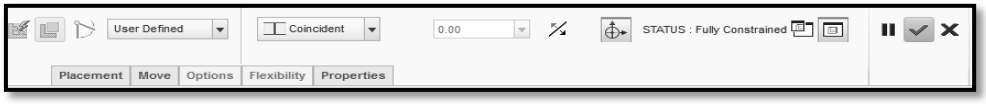
Coincident constraints of both the parts.



Coincident both the parts.

- Solid Modeling Parametric recognizes two flat surfaces facing each other and applies a Coincident constraint. These two selected surface are now coincident to each other.

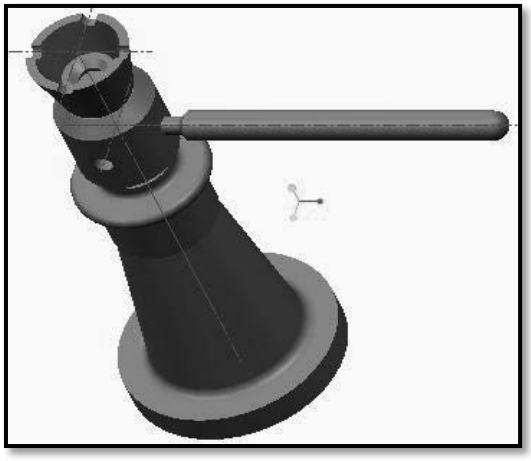
- The Washer has changed to a **yellow-orange** color indicating that its position is fully constrained.
- The Assembly dashboard shows the Coincident constraint type was the last used and that the Washer is now Fully Constrained.



Coincident Assembly constraint with Fully Constrained dashboard.

- Click **Complete Component**  to complete the component placement.
- Then the Washer returns to its original gray color.

5. Reorienting and saving your work:
   - Press CTRL + D to reorient the model.
   - Click Save  to save your work.

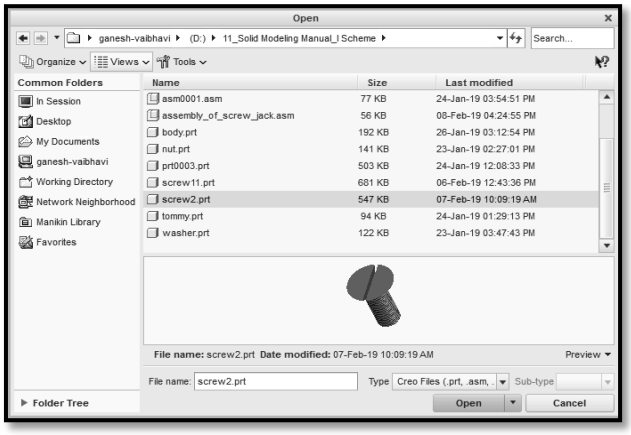


Reoriented Assembly.

Step 7: Adding the Next component to the assembly (i.e. Screw)

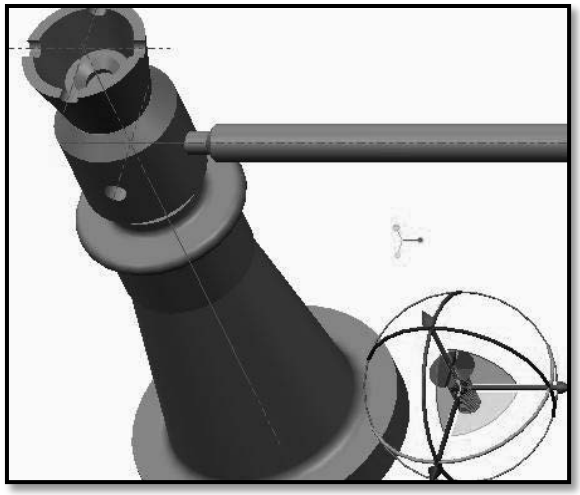
1. Selecting the Next part to assemble:

- Click ‘Assemble’ tool from Component group of Model tab.
- In the lower-right corner of the Open dialog box, click to expand the Preview pane.
- Browse the location where you have saved**Screw** model and select it.
- Click Open to assemble this component.



Calling of Next object i.e. Screw.

- The part will be attached to the cursor and the Assembly dashboard will open.

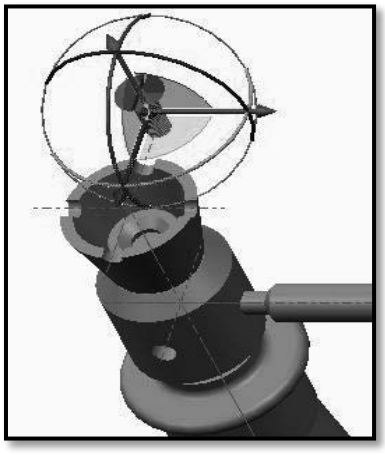


Screw with Assembly.

Automatic and No Constraints dashboard.

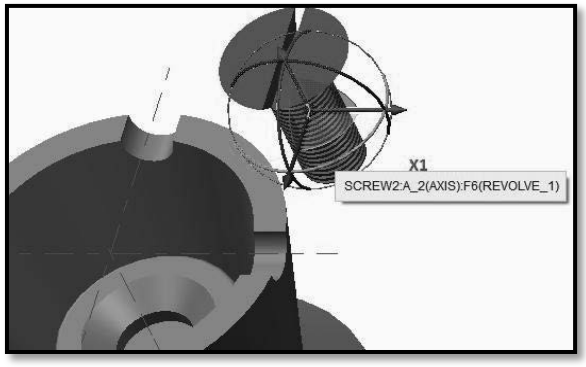
2. Locating the part temporarily, before final placement:

- Drag and zoom the Screw to a position just to the top of the Assembly, and then click in the graphics area to place it.



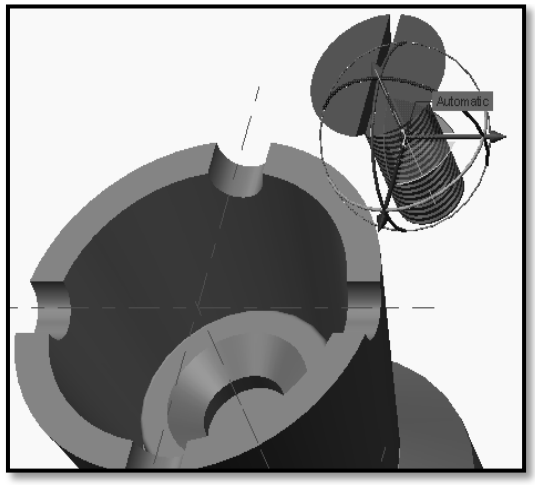
Screw towards Top side of assembly.

3. Adding the first assembly constraint:
   - Zoom and move the cursor over the axis of Screw till it shows the cursor tip as shown at X1.



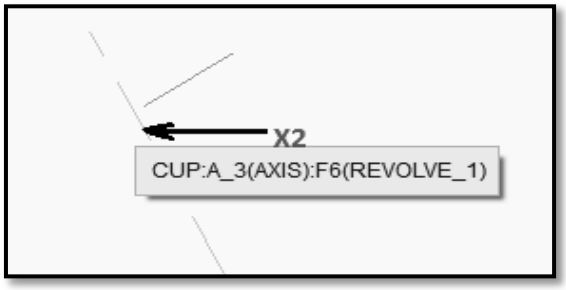
First Assembly Constraint (X1) i.e. Axis of Screw.

   - When the cursor shows cursor tip shown at X1, click to select the axis. Then it displays **Automatic**.



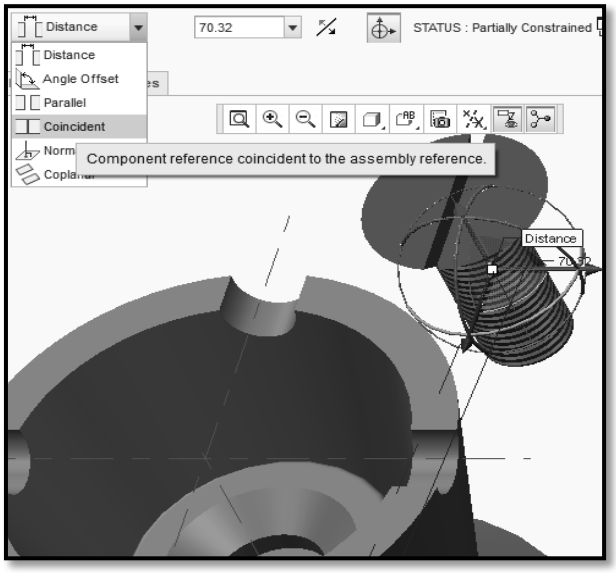
Selection of First Assembly Constraint (X1) i.e. Axis of Screw.

   - Move the cursor over the axis of the Cup / Washer till it shows the cursor tip as shown at X2. Then click to select the axis of the Cup / Washer.

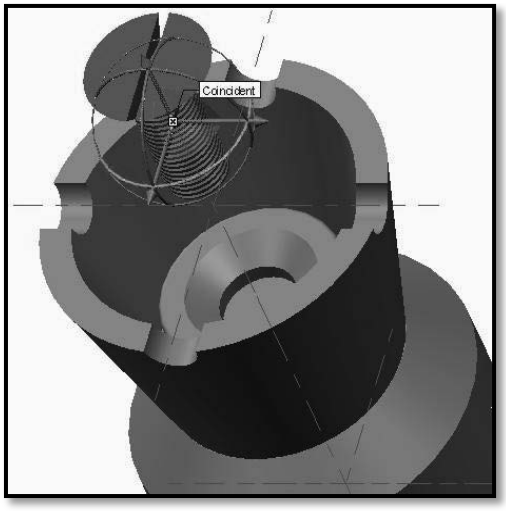


Second Assembly Constraint (X2) i.e. Axis of Cup.

   - Then select the **Coincident** constraint from the **Component Placement > Constraint dashboard.**



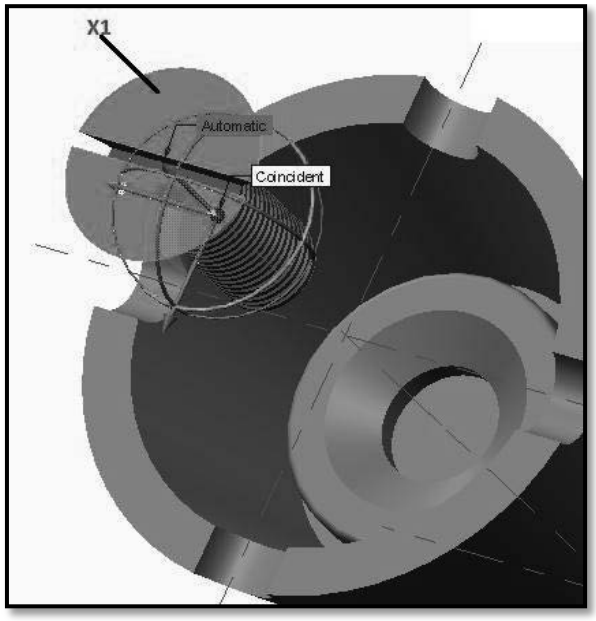
Before Application of Coincident Constraint.



After Application of Coincident Constraint.

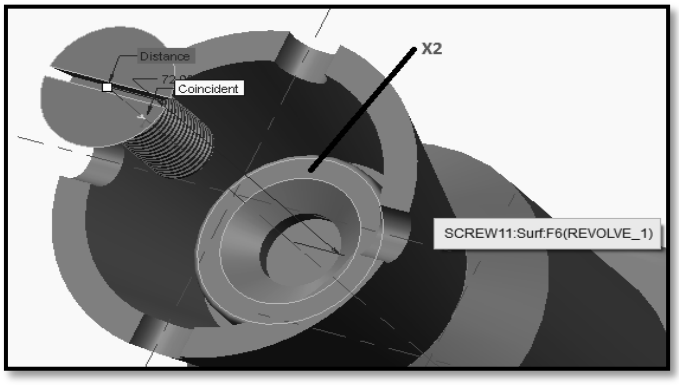
- If required click on **Flip Axis Constraint** from the **Placement** tab in order to flip the orientation of a constraint.
- Then drag the Screw towards upper side of the Assembly.

4. Adding the second assembly constraint:
   - If required press the middle-mouse button and drag to spin and zoom the model until you can see the head surface of screw shown as X1.
   - Click to select the head surface of the screw (X1) that it closest to and parallel to the surface of Washer(X2).



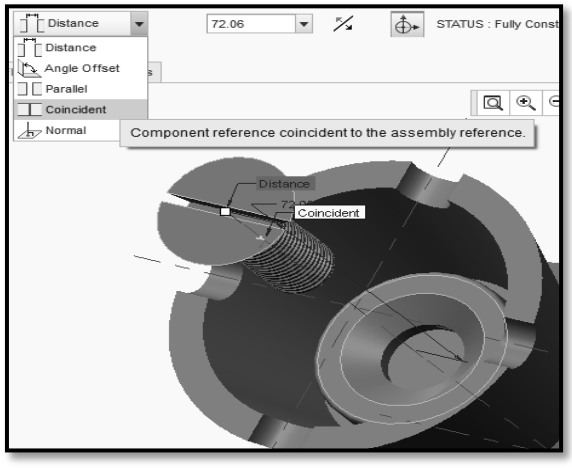
Second Assembly Constraint i.e. Surface of Screw (X1).

- Press the middle-mouse button and drag to spin and zoom the model until you can see the flat surface shown as X2 of the Washer.

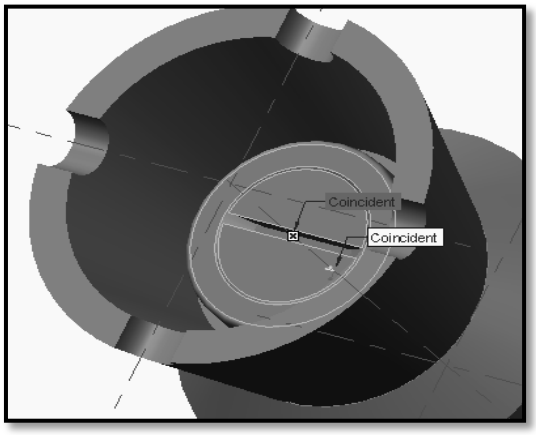


Selection of Second Assembly Constraint i.e. flatSurface of Washer (X2).

- Click to select the flat surface of Washer shown as X2.
- Then from Constrained dashboard select coincident constraint.

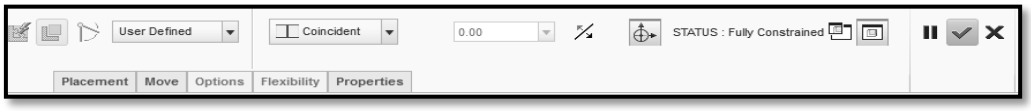


(a) – Coincident both the parts.



(b) – Coincident both the parts.

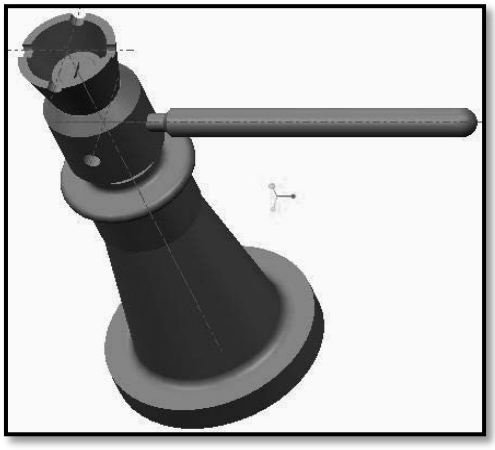
- Solid Modeling Parametric recognizes two flat surfaces facing parallel to each other and applies a Coincident constraint. These two selected surface are now coincident to each other.
- The Washer has changed to a **yellow-orange** color indicating that its position is fully constrained.
- The Assembly dashboard shows the Coincident constraint type was the last used and that the Screw is now Fully Constrained.



Coincident Assembly constraint with Fully Constrained dashboard.

- Click **Complete Component**  to complete the component placement.
- Then the Screw returns to its original gray color.

5. Reorienting and saving your work:
   - Press CTRL + D to reorient the model.
   - Click Save  to save your work.



Reoriented Assembly.

XVIII Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1					
2					
3					

XIX Actual Procedure Followed

.....

.....

.....

.....

.....

.....

XX Precautions Followed

.....

.....

.....

.....

.....

XXI Course Proficiency

.....

.....

.....

.....

.....

XXII Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Explain – fully constrained object.
2. Explain different constraint used in assembly with relative sketches.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XXIII Questions for Practice.

1. Create models of individual components of Tool Post and Assemble it.

XXIV References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=xzteh5MFDs4
2. https://www.youtube.com/watch?v=Yp2SbrxhfNQ

XXV Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.11: Assemble and Print the Orthographic Views of the Assembly Developed in Pro 5 to 8 With Bill of Materials.

I Practical Significance

Drawing views provide a means communication between the design engineers and production personnel. By studying this practical one can automatically create traditional 2D orthographic and detail views of either a 3D model or an assembly and one can easily prepare a Bill of Materials.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills for the industry identified competency –

- Opening an existing assembly / component.
- Starting a new drawing - paper size, template.
- Apply dimensions to the drawing views.
- Preparation of Bill of Material.

IV Relevant Course Outcome(s)

- Generate orthographic views of 3D solid assemblies using Drawing workbench of any parametric CAD software.
- Generate production drawing for given part models / assemblies.

V Practical Outcome

- Use any available parametric CAD modeling software to create and print orthographic views of assembly for any engineering products.

VI Relative Affective Domain-

- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Knowledge of drawing workbench of CAD modeling software.
- Types and Method of dimensions.
- Basic knowledge of reference projection views.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

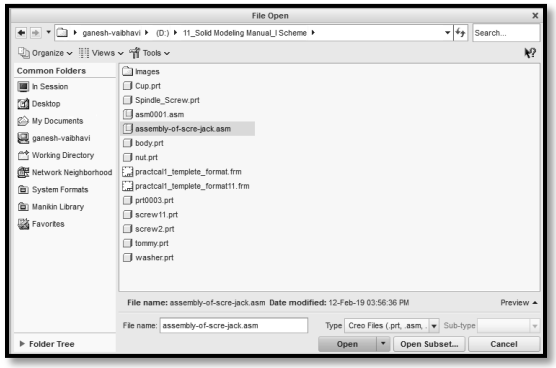
IX Precautions to be Followed

- Units used while designing the individual models should be properly selected.

X Procedure

Step 1: Set working directory and open assembly

1. Start Solid Modeling Parametric CAD software.
2. Set the working directory as explained in earlier practical's.
3. Opening the new assembly model of screw jack:
   - From the Quick Access toolbar click on **Open**tab. The file open dialog box opens.

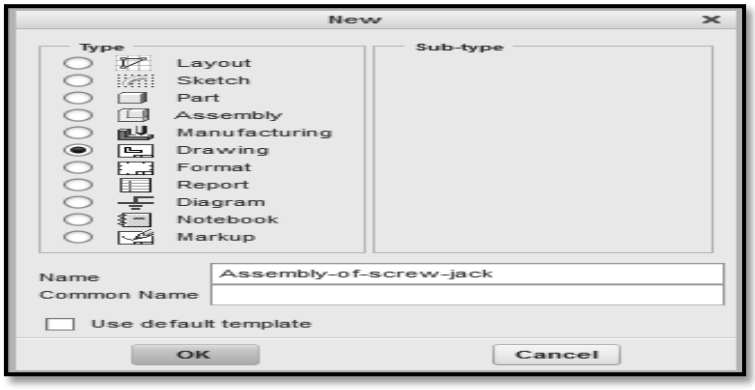


File open dialog box.

   - If necessary click on Working Directory in the left panel.
   - Select your assembly-of-screw-jack.asm file and click open.

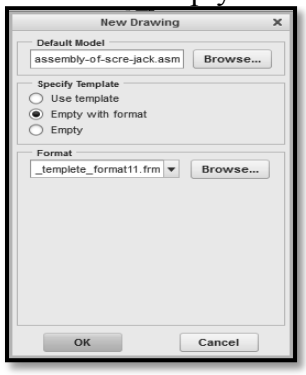
Step 2: New Engineering Drawing.

1. Starting a new drawing:
   - In the Quick Access toolbar, click on to start a new file.



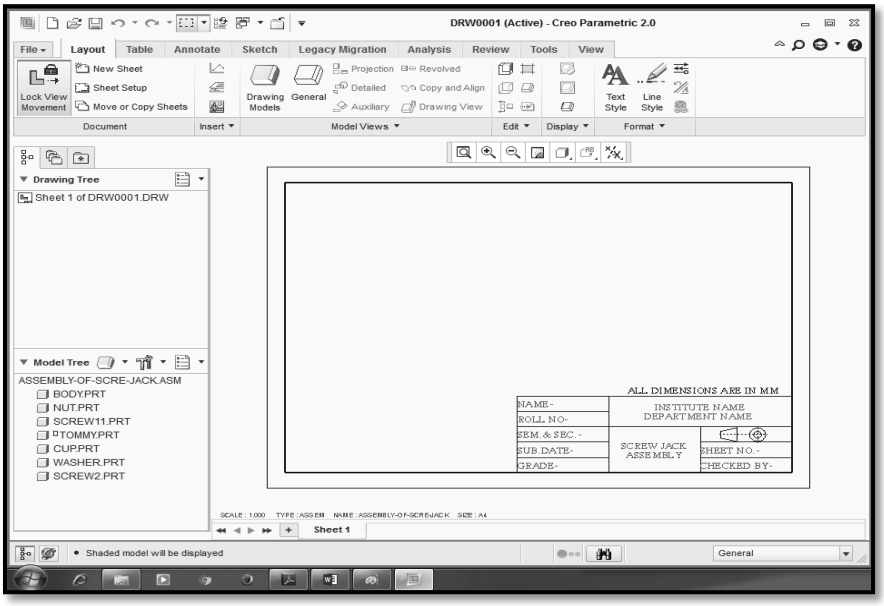
New dialog box.

- In the New dialog box, click Drawing for the Type and type in Assembly-of-screw-jack for the Name.
- Uncheck the Use default template option.
- Click on OK and New Drawing dialog box opens.
- A 'New Drawing' dialog box will pop up, browse default model as assembly file.
- From Specify Template select 'Empty with format' option.



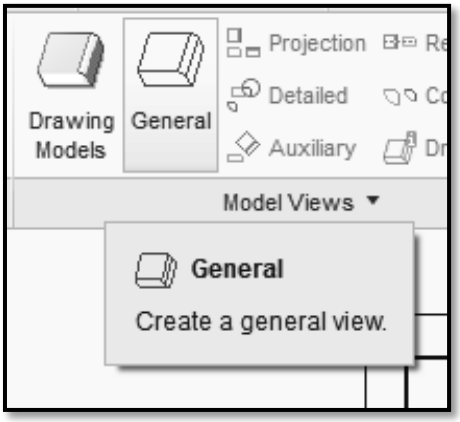
New drawing dialog box.

- Then from Format option browse and select the template as you need (or paper size you require).
- Click OK to create the drawing.
- Drawing environment with selected template will open.

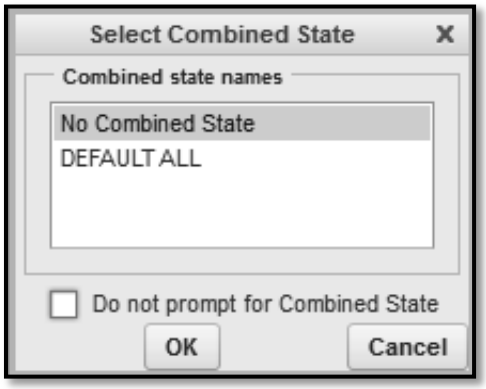


Drawing Environment with Template.

- From LAYOUT Tab select GENERAL from model views.



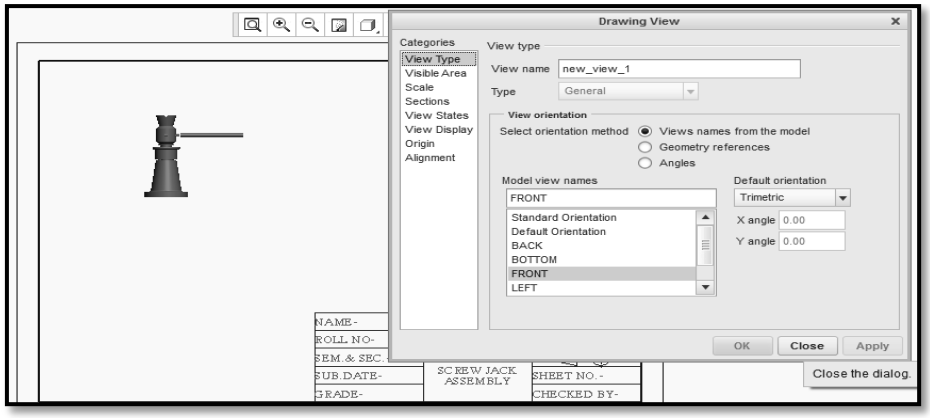
General view.



Select combined state dialog box.

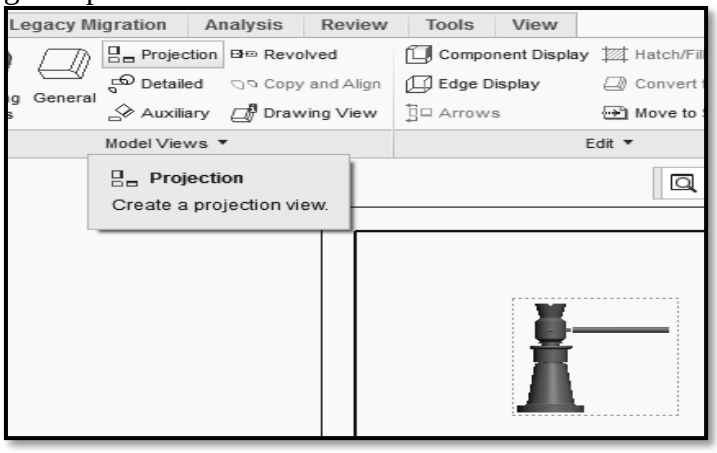
- Just click on OK button from Select Combined State dialog box.
- Click in the drawing area where you want to insert the General view.
- Then from Drawing View dialog box, select FRONT as a Model view name.

- Click on Apply and then click Close to close drawing view dialog box.



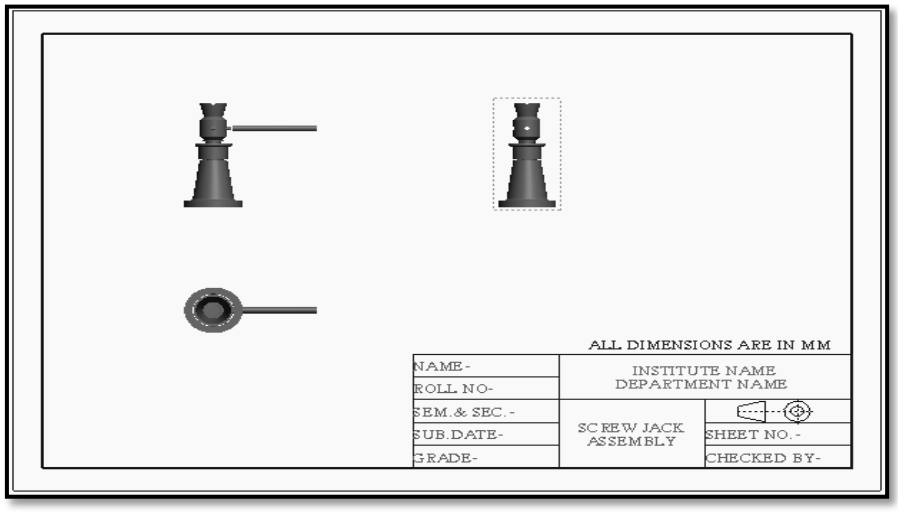
Drawing view.

- Select generated Front view, then click on Projection from Layout tab.
- To get Top view click below the Front view.



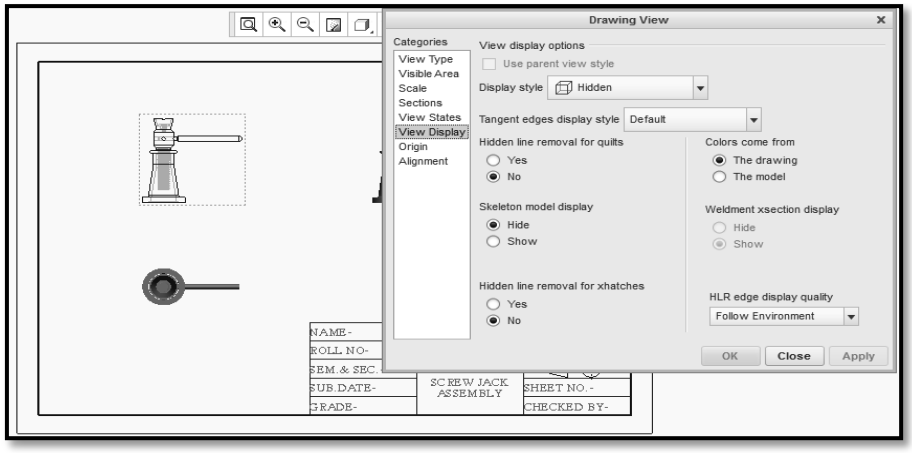
Projection view.

- Repeat above two steps to get Side view.



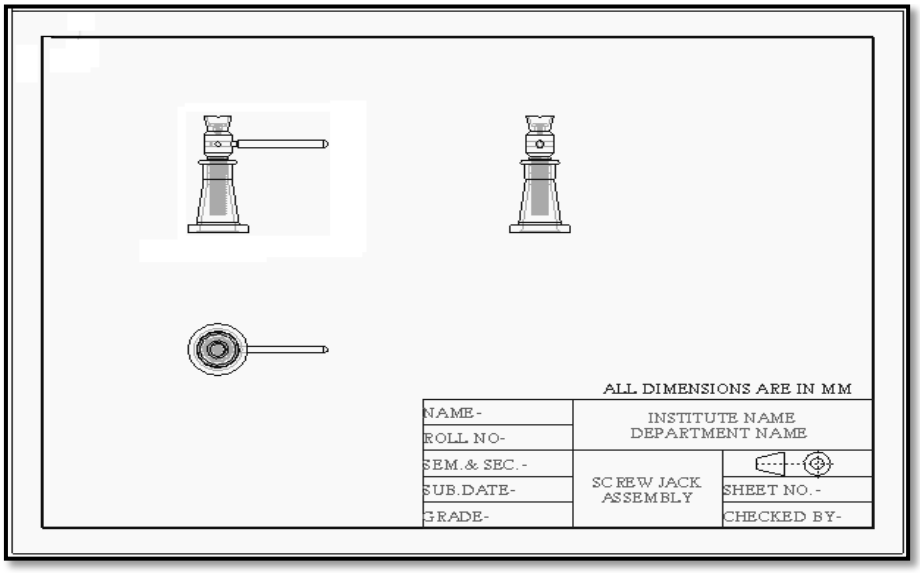
Drawing views before hidden display style.

- Double click on the generated Front view, we will get the drawing view dialog box again.
- From Categories select View Display and select Hidden Display style.
- Click on Apply and then Close.



Application of hidden display style.

- Repeat the above three steps twice to convert the top view and side view as Display style Hidden, simultaneously.



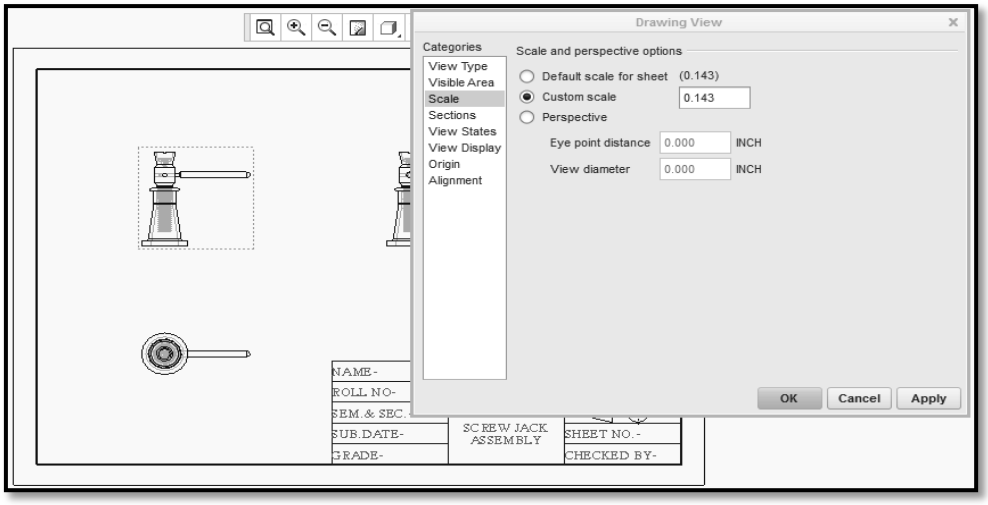
Drawing views after hidden display style.

2. Changing the drawing scale:

   Automatic obtained drawing views has a size comparatively smaller than that of the paper size. Therefore increase the scale to match the size of the model views to the paper size.

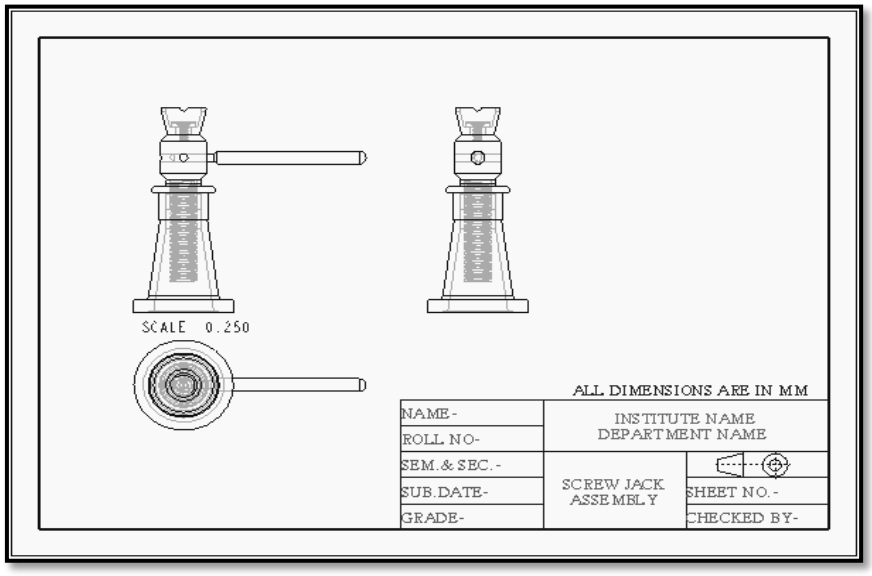
- Double click on General views (initially inserted) to get open Drawing View dialog box.
- From Categories select Scale, then choose Custom scale from Scale and perspective options.
- Change the Custom scale as required.

- Then click Apply and say OK.



Application of drawing scale.

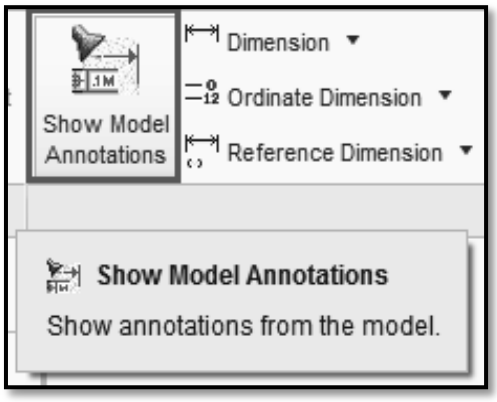
- After specifying the suitable scale, we will get the views as shown below.
- Drawing view displays the Scale factor below the view.



Scaled 3 drawing views.

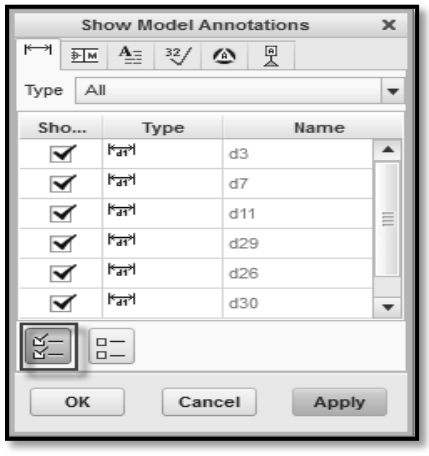
3. Showing dimensions:

- In the drawing ribbon make sure the **Annotate** tab is selected.
- In the graphics window, select the view you want to add dimensions to. The border of the sketch will turn green showing it is selected.
- In the Annotate ribbon, click on **Show Model Annotations.**



(a)– Model Annotation.

- Click on the view you want to add the dimensions.
- The Show Model Annotations dialog box will open listing all the dimensions that were used to create the 3D model.



(b)– Model Annotation.

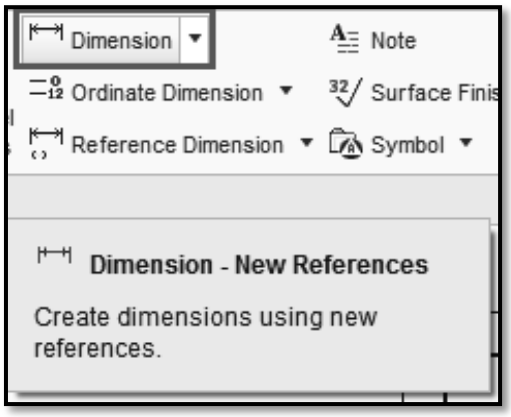
These can be checked/ticked individually to make them appear on the drawing in the select view. Near the bottom of the dialog box is a button to add all the dimensions

- Click  to show all dimensions on the selected view.

The dimensions will appear on the selected view but may or may not be placed properly. Then place it properly. Delete the dimensions which are not required.

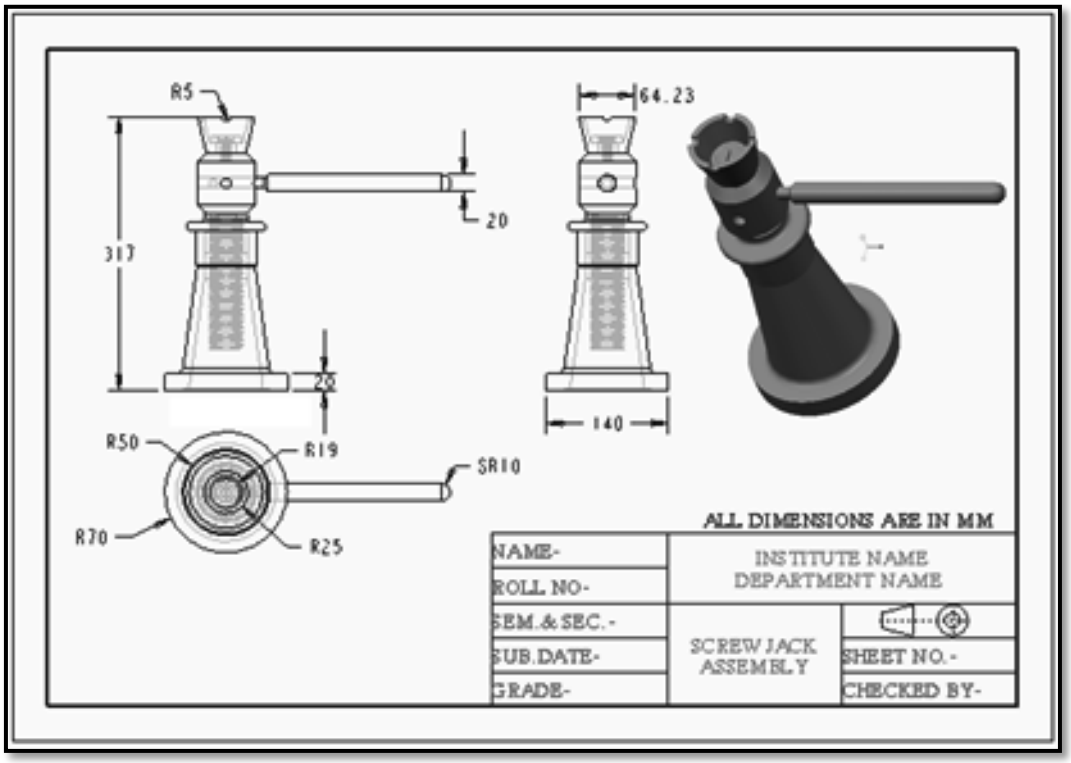
OR

- We can place the dimensions by selecting Dimension – New References tool and picking individual entities.



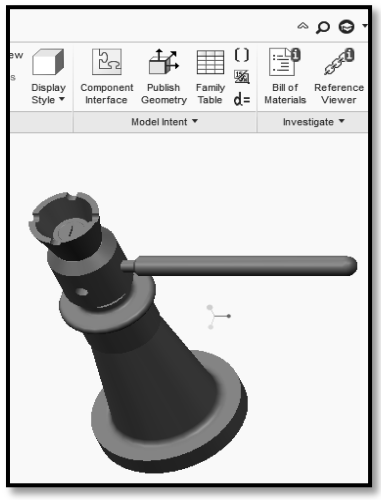
Dimension – New References

- Finally generate the Drawing views as shown below.



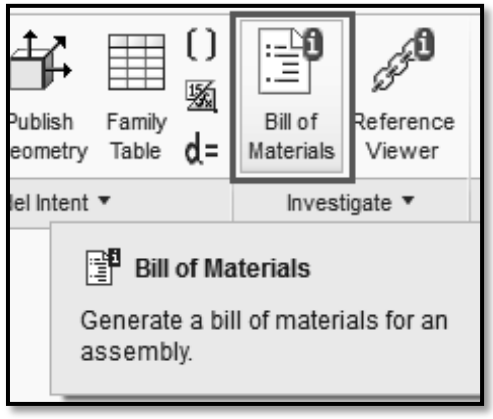
Assembly drawing views with all aspects.

4. The Bill of Material:
   The Bill of Material is a tabular representation of all the components of the assembly, along with the information associated with them.
   - Click on Open to browse and open the Assembly file (i.e. Assembly_of_Screw_Jack.asm).



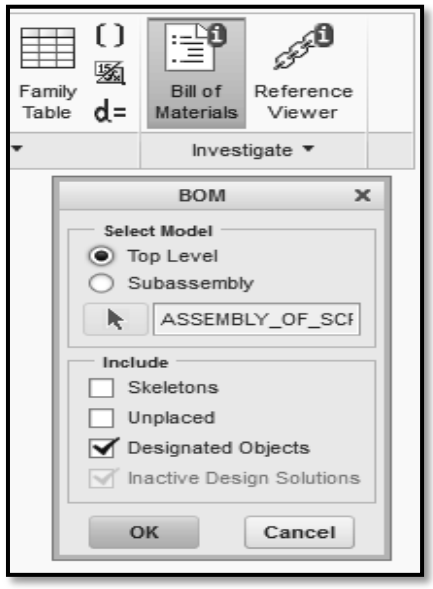
Assembly for Bill of material.

- Select and click 'Bill of Materials' tool from Investigate.

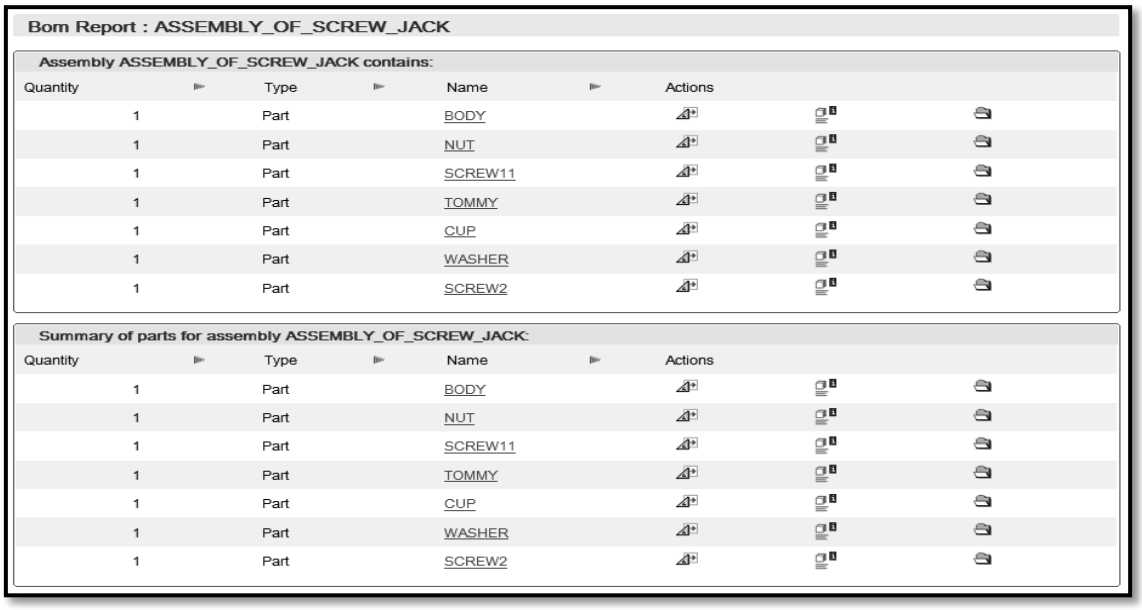


Bill of materials tool.

- BOM dialog box will display, from this dialog box keep selected Top level and check Designated Objects.
- Then click OK button.



Bill of materials dialog box.



Bom Report : ASSEMBLY_OF_SCREW_JACK

Assembly ASSEMBLY_OF_SCREW_JACK contains:

Quantity	Type	Name	Actions
1	Part	BODY	
1	Part	NUT	
1	Part	SCREW11	
1	Part	TOMMY	
1	Part	CUP	
1	Part	WASHER	
1	Part	SCREW2	

Summary of parts for assembly ASSEMBLY_OF_SCREW_JACK:

Quantity	Type	Name	Actions
1	Part	BODY	
1	Part	NUT	
1	Part	SCREW11	
1	Part	TOMMY	
1	Part	CUP	
1	Part	WASHER	
1	Part	SCREW2	

Bill of materials.

XI Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XII Actual Procedure Followed

.....

.....

.....

.....

.....

XIII Precautions Followed

.....

.....

.....

.....

.....

XIV Course Proficiency

.....

.....

.....

.....

.....

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Explain to generate BOM.
2. Assemble and Print the orthographic views of any one of the following assembly – Bench vice, Drill Jig, Tool Post.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

XVI References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=qlXXN872GqA&feature=youtu.be
2. https://www.youtube.com/watch?v=mBScLZ5yy58

XVII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.12: Draw and Print the production drawing of all individual components part models of assembly developed in PrO 5 to 8.

I Practical Significance

Drawing and documentation are essential for any product design, because they provide guidance in the manufacturing of mechanical devices. While creating part model we might have given various dimensions, geometric constraints, these details are used in drawing mode directly. Drawing views provide a means communication between the design engineers and production personnel. By studying this practical one can automatically create orthographic and detail views of either a 3D model or an assembly. Also here we can select multiple item types such as Dimension, Axis, Surface Finish and their options at a time.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills for the industry identified competency –

- Opening an existing component.
- Creating of General and projection views.
- Apply dimensions to the drawing views.

IV Relevant Course Outcome(s)

- Generate orthographic views of 3D solid models using Drawing workbench of any parametric CAD software.
- Generate production drawing for given part models.

V Practical Outcome

- Use any available parametric CAD modeling software to create and print orthographic views of individual models for any engineering products.

VI Relative Affective Domain-

- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Knowledge of drawing workbench of CAD modeling software.
- Types and Method of dimensions.
- Basic knowledge of reference projection views.
- Dimensional tolerances, tolerances methods and types.
- Geometrical &Dimensional Tolerances (GD&T).

In addition to Dimensional Tolerance, we must show geometric tolerances on a component for manufacturing. Dimensional tolerances control the size of a component whereas geometric tolerance controls the shape of the component. The various parameters shown by Geometric Tolerances are geometric conditions such as surface finish, perpendicularity, circularity etc.

Geometric Dimensioning and Tolerance (GD&T) is a symbolic language which communicates design intents. Geometric Dimensioning is a geometric characteristic the size of which is specified such as length, angle, location, or center distance. Geometric Tolerance is the total permissible variation in its size, which is equal to the difference between the limits of size, it states the maximum allowable variations of a form or its position from the perfect geometry implied on the drawing. Dimensional tolerance controls size of a part whereas Geometrical tolerance controls the shape of a part.

It is used to specify the size, shape, form, orientation, and location of features on a part. And it is basically a very good design tool. To apply GD&T one must have the parts functionality in an assembly.

VIII Resources Required

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter	Plotter A_2 OR A_3Size.	1

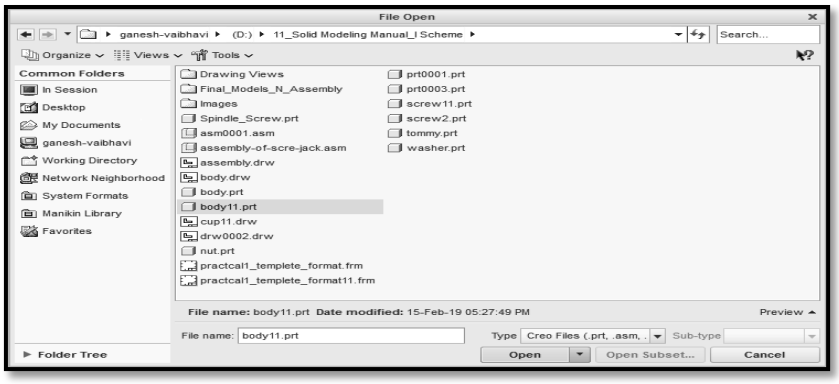
IX Precautions to be Followed

- Units used while designing the individual models should be properly selected.
- While taking drawing views, general view should take carefully.
- Care should be taken while applying dimensions, like placement of dimension and should not repeat the same dimension.

X Procedure

Step 1: Set working directory and open body part

1. Start Solid Modeling Parametric CAD software.
2. Set the working directory as explained in earlier practical's.
3. Opening the new part model body of screw jack:
   - From the Quick Access toolbar click on **Open** tab. The file open dialog box opens.

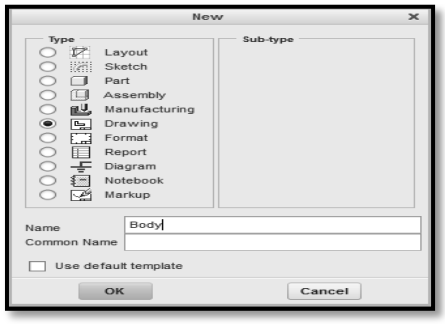


File open dialog box.

   - If necessary click on Working Directory in the left panel.
   - Select your part model body11.prt and click open.

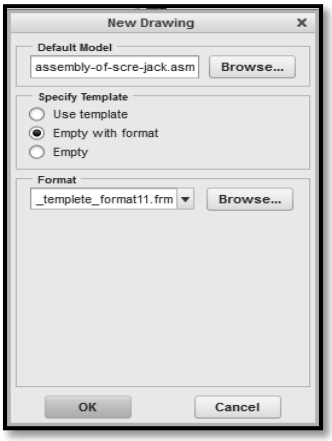
Step 2: New Engineering Drawing.

1. Starting a new drawing:
   - In the Quick Access toolbar, click on to start a new file.
   - In the New dialog box, click Drawing for the Type and enter as Body in Name field.
   - Uncheck the Use default template option.
   - Click on OK and New Drawing dialog box opens.



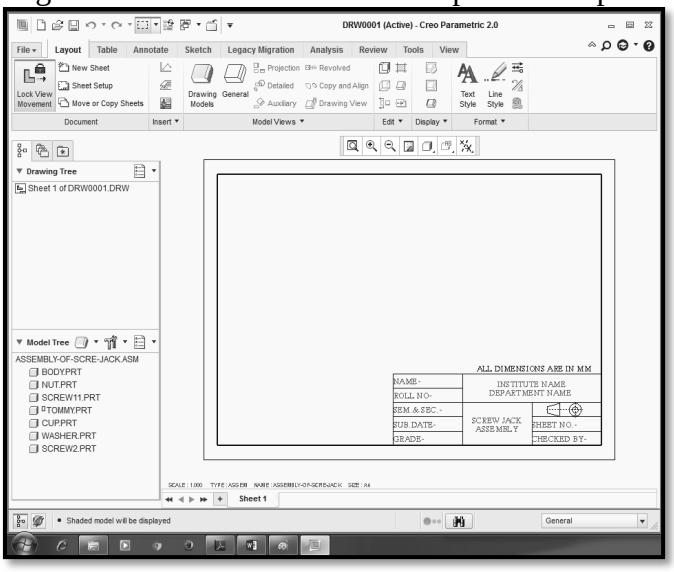
New dialog box.

   - A 'New Drawing' dialog box will pop up, browse default model as body file.
   - From Specify Template select 'Empty with format' option.



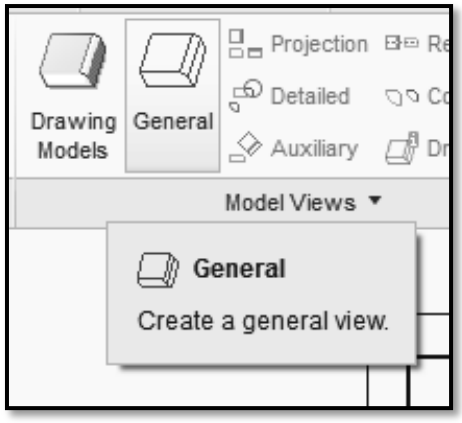
New drawing dialog box.

- Then from Format option browse and select the template as you need (or paper size you require).
- Click OK to create the drawing.
- Drawing environment with selected template will open.

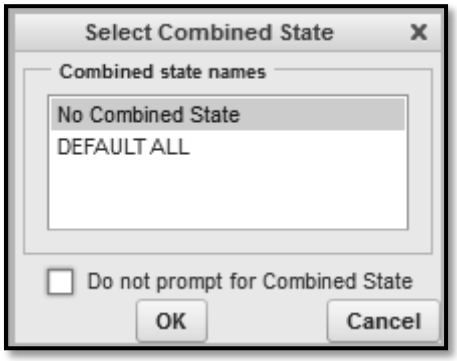


Drawing Environment with Template.

- From LAYOUT Tab select GENERAL from model views.

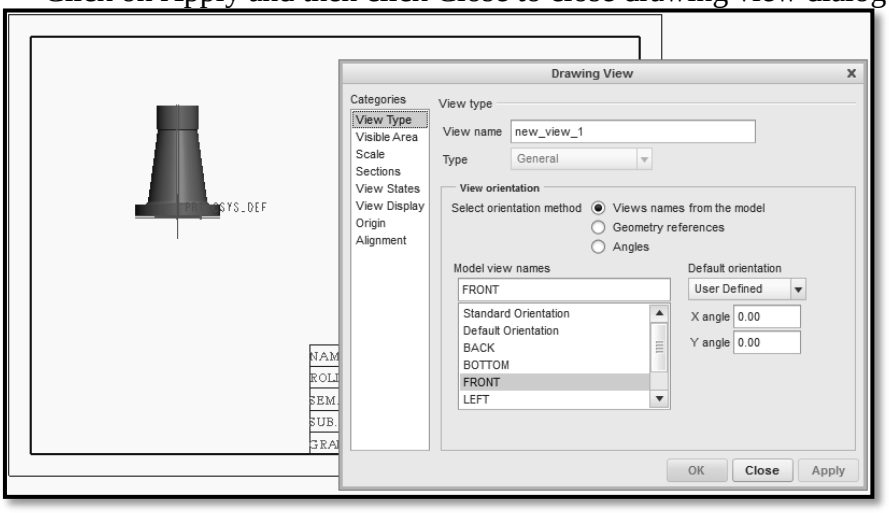


General view.



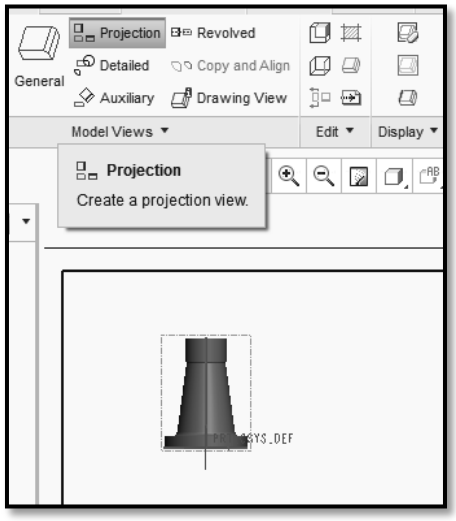
Select combined state dialog box.

- Just click on OK button from Select Combined State dialog box.
- Click in the drawing area where you want to insert the General view.
- Then from Drawing View dialog box, select FRONT as a Model view name.
- Click on Apply and then click Close to close drawing view dialog box.



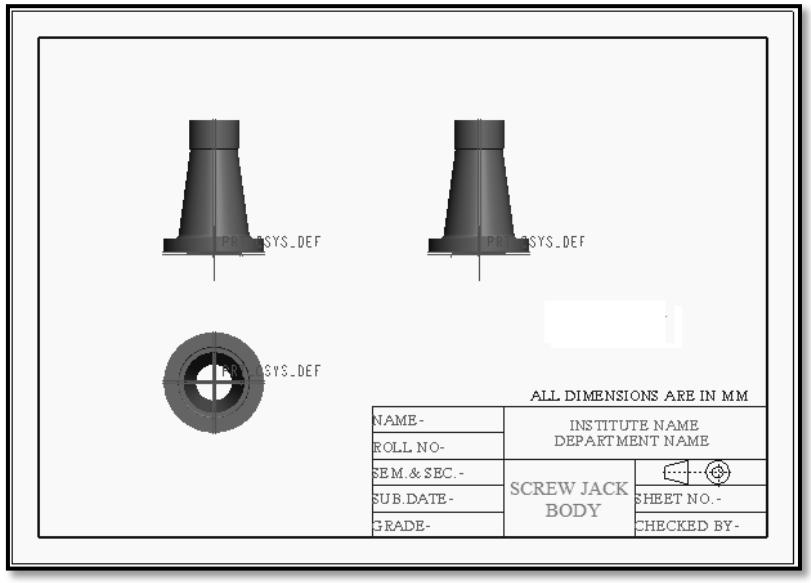
Drawing view.

- Select generated Front view, then click on Projection from Layout tab.
- To get Top view click below the Front view.



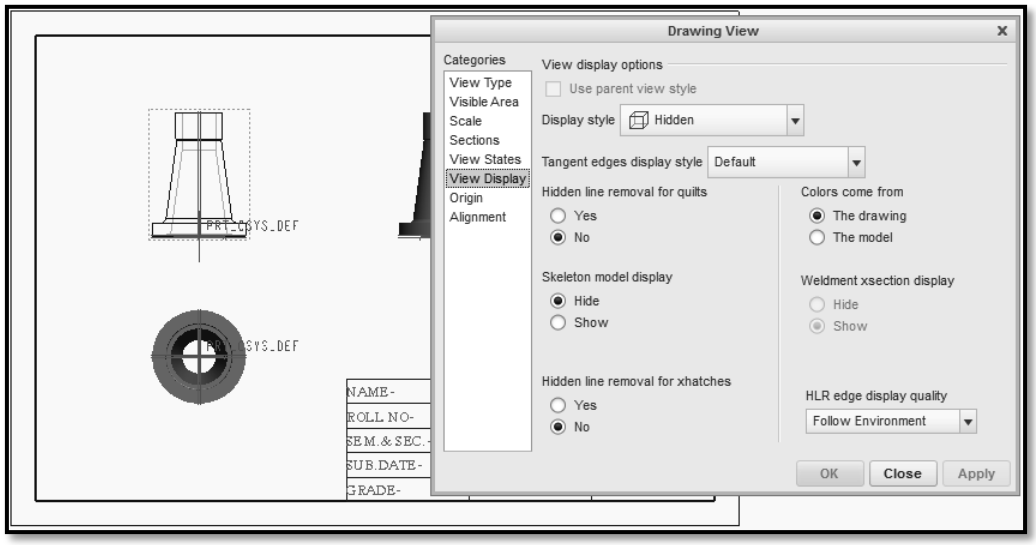
Projection view.

- Repeat above two steps to get Side view.



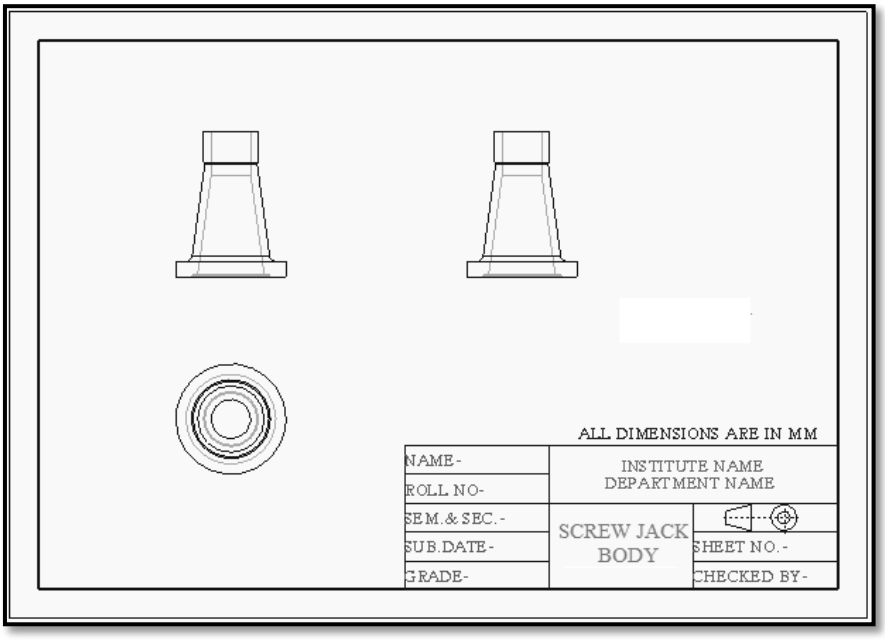
Drawing views before hidden display style.

- Double click on the generated Front view, we will get the drawing view dialog box again.
- From Categories select View Display and select Hidden Display style.
- Click on Apply and then Close.



Application of hidden display style.

- Repeat the above three steps twice to convert the top view and side view as Display style Hidden, simultaneously.

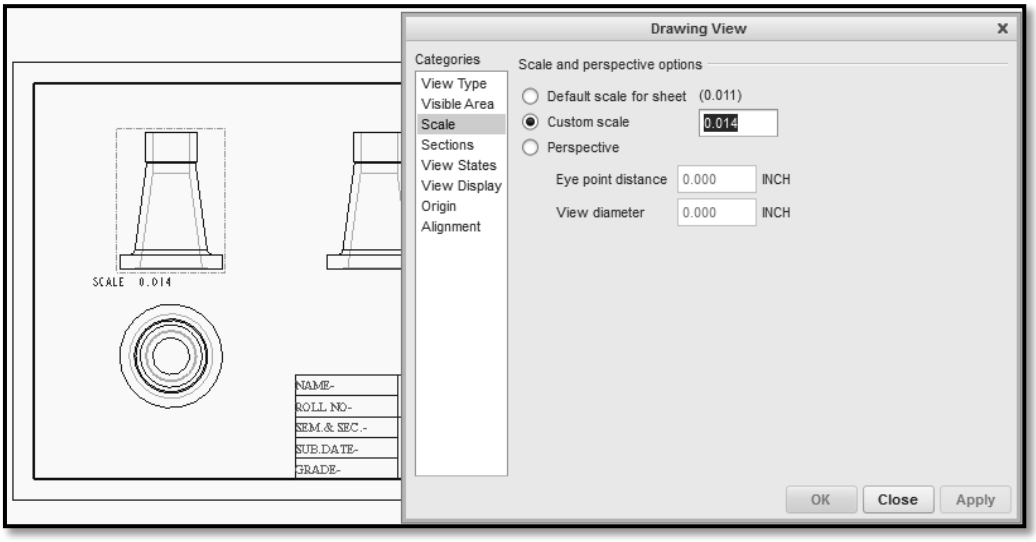


Drawing views after hidden display style.

3. Changing the drawing scale:

   Automatic obtained drawing views has a size comparatively smaller than that of the paper size. Therefore increase the scale to match the size of the model to the paper size.

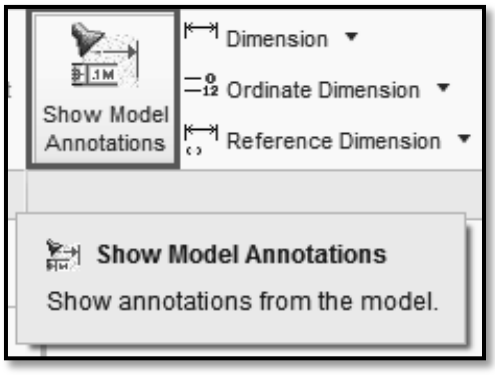
   - Double click on General views (initially inserted) to get open Drawing View dialog box.
   - From Categories select Scale, then choose Custom scale from Scale and perspective options.
   - Change the Custom scale as required.
   - Then click Apply and say OK.



Application of drawing scale.

- After specifying the suitable scale, we will get the views as shown below.
- Drawing view displays the Scale factor below the view.

4. Showing dimensions:
    - In the drawing ribbon make sure the **Annotate** tab is selected.
    - In the graphics window, select the view you want to add dimensions to. The border of the sketch will turn green showing it is selected.
    - In the Annotate ribbon, click on **Show Model Annotations.**



Model Annotation.

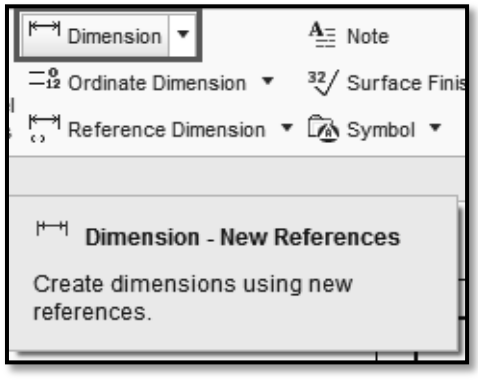
The **Show Model Annotations** dialog box will open listing all the dimensions that were used to create the 3D model.These can be checked/ticked individually to make them appear on the drawing in the select view. Near the bottom of the dialog box is a button to add all the dimensions.

- Click  to show all dimensions on the selected view.

The dimensions will appear on the selected view but may or may not be placed properly. Then place it properly. Delete the dimensions which are not required.

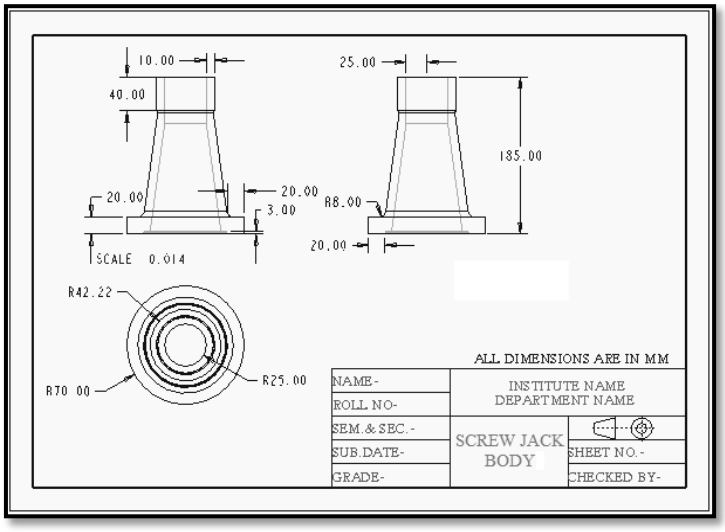
OR

- We can place the dimensions by selecting Dimension – New References tool and picking individual entities.



Dimension – New References

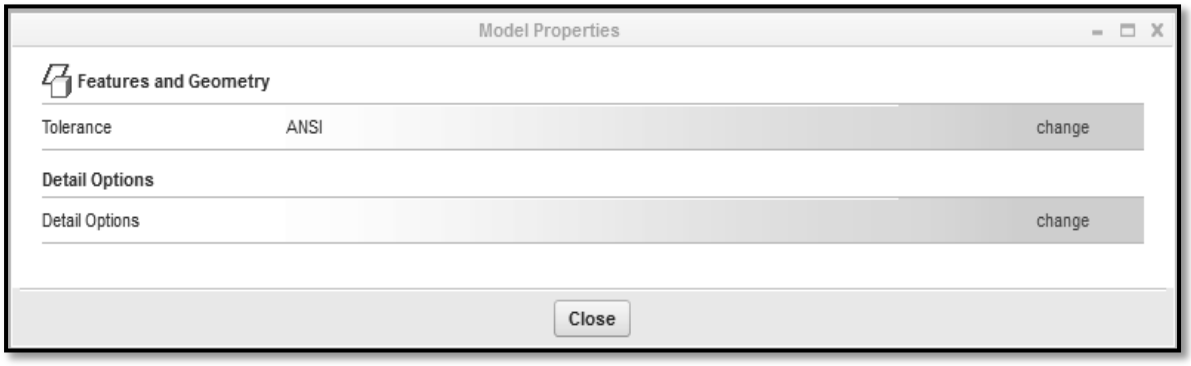
- Finally generate the Drawing views as shown below.



Body part drawing views.

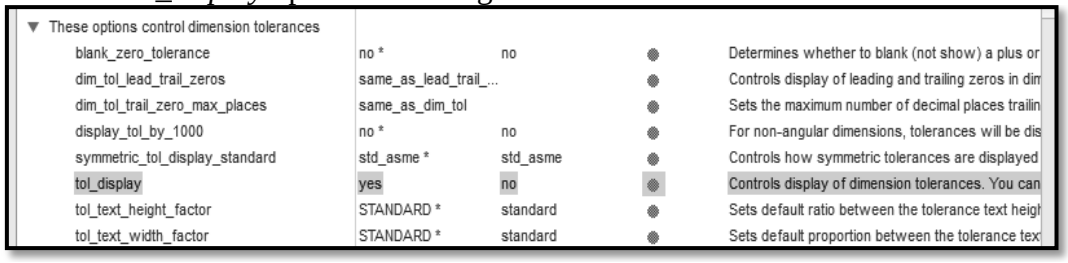
5. **Showing the Dimensional Tolerance :**

- By default dimensional tolerance are not shown in the drawing. To show dimensional tolerance on a drawing invoke **File>Prepare>Drawing Properties** from main menu bar. It display Model properties dialog box.

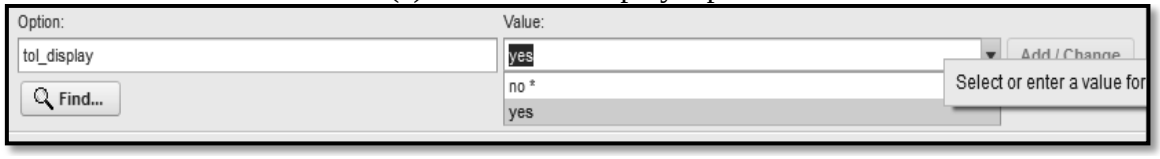


Model Properties.

- From this dialog box select change from Detail Options. Then it displays drawing configurations file window.
- Search *tol_display* option and change its default value **NO to YES.**

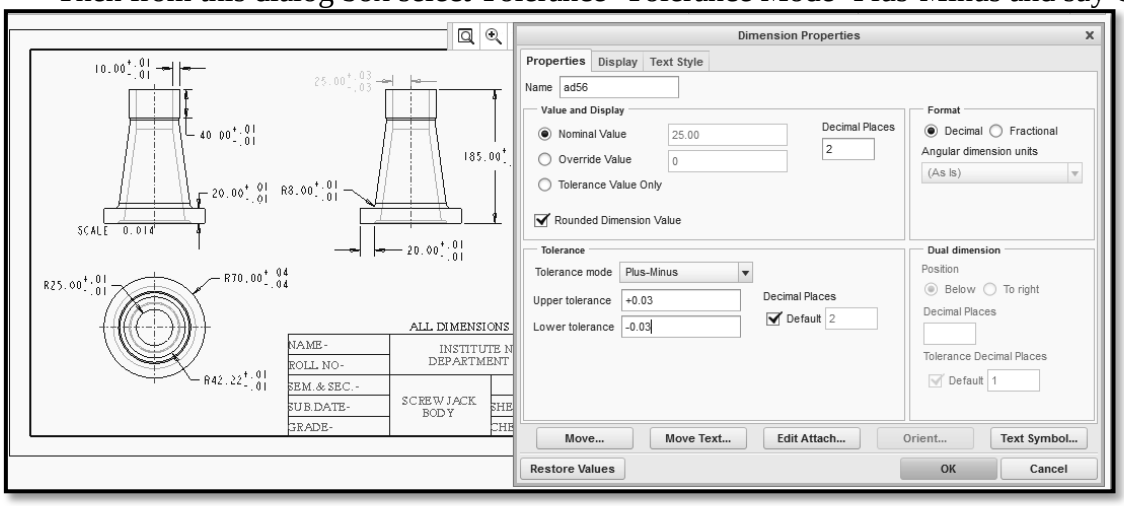


(a)–Tolerance Display Option.



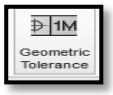
(b)–Tolerance Display Option.

- To change the display mode of tolerance, double click on a dimension, following dialog box will be seen.
- Then from this dialog box select Tolerance>Tolerance Mode>Plus-Minus and say OK.

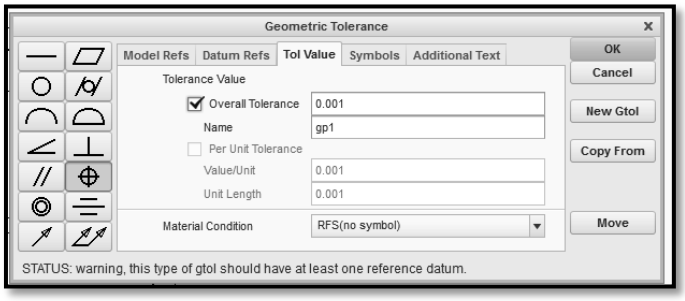


Three views with dimension properties dialog box.

6. **Geometric Dimensions & Tolerances (GD&T) :**

- To add geometric tolerances invoke Annotate>Geometric Tolerances 

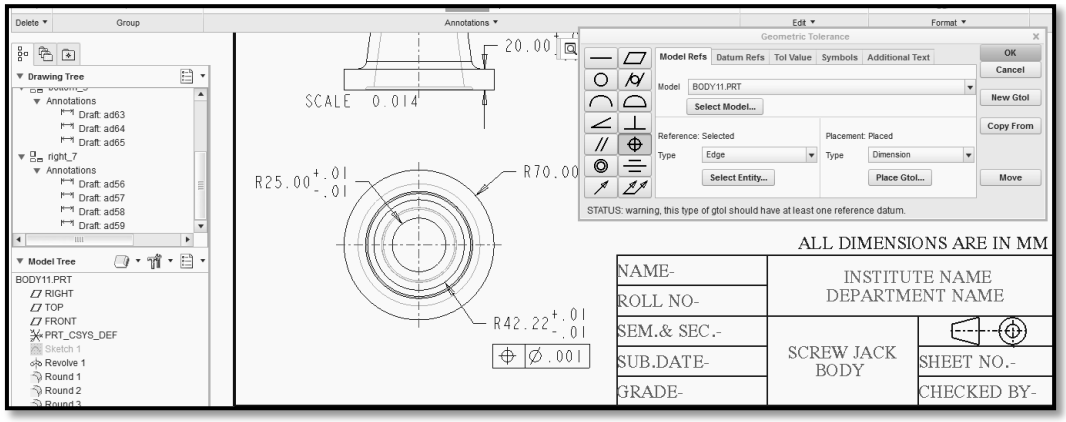
from ribbon menu which displays Geometric Tolerance doalog box as shown in Figure.



Geometric Tolerance dialog box.

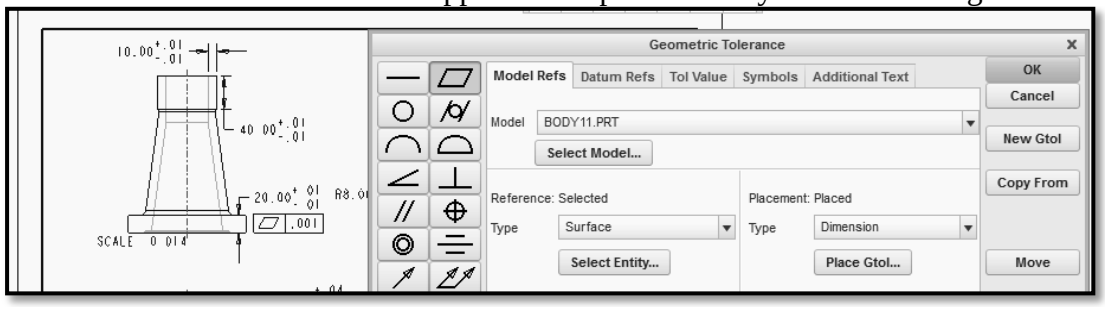
- Symbol  Tolerance is default selected.

- Tolerance quantity is mentioned in the **Tol Value** option as shown in Figure. Here tolerance value is 0.001.
- In **Model Ref. Tab** option, select **'Select Entity'** and clickon the entityof the drawing to which tolerance to be provide. Select **'Place Gtol'** tab and then select dimension from the part.



Selection of Geometric Tolerance.

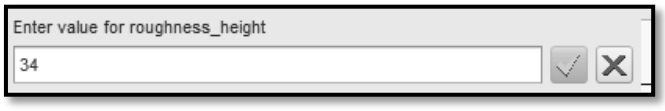
- To add geometric tolerance flatness. Select flatness symbol  .
- Flatness quantity is mentioned in the **Tol Value** option as shown in Figure. Here tolerance value is 0.001.
- In **Model Ref. Tab** option, select **'Select Entity'** and clickon the entityof the drawing to which flatness to be providing. Select '**Place Gtol'** tab and then select dimension from the part. Click OK button.
- Geometric tolerance flatness will appear on respective entity as shown in Figure.

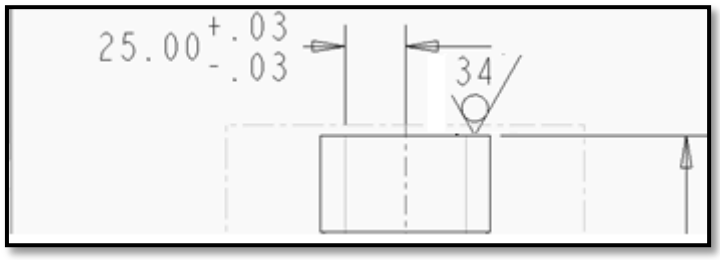


Application of Geometric Tolerance.

7. **Surface Finish Symbol :**

- Invoke **Annotate>Surface finish**  from ribbon menu mode.
- Click **Retrieve** option and open Generic/Unmachined or machined folder. Here we open **Unmachined** folder and open **standard2.sym.** Select **Normal** option, then select an entity or an edge or a dimension or a curve or a point on a surface from drawing views and then enter roughness value = 34. Then click Check option as shown in Figure.





Surface roughness value and symbol.

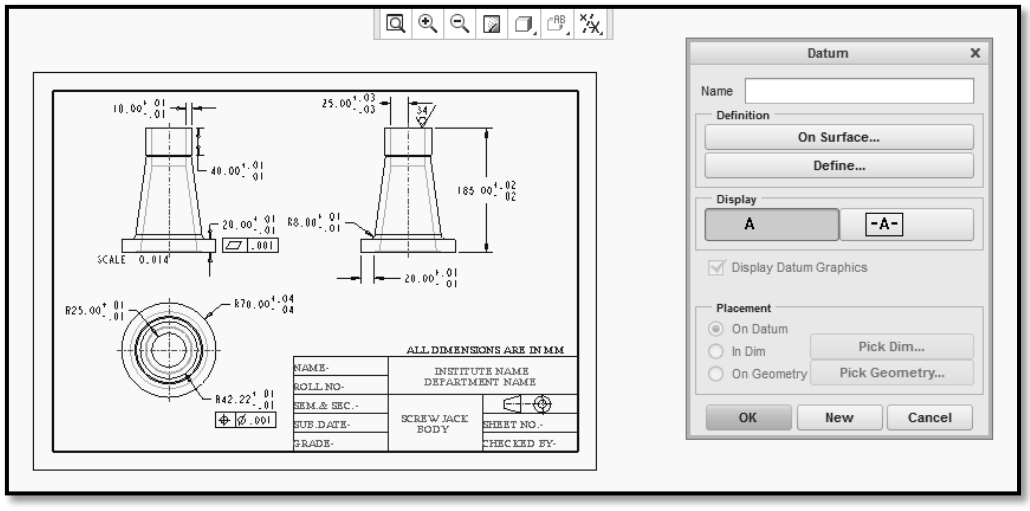
8. **Datum Reference Frames :**

We must define datum reference before applying geometric tolerance.

- **Create Reference Datum-**The reference must be created earlier to apply geometric tolerances.

a) Reference Datum's are created by clicking Annotate**>Model datum>Model Datum plane** 

from ribbon menu, a datum dialog box is displayed as shown in Figure.

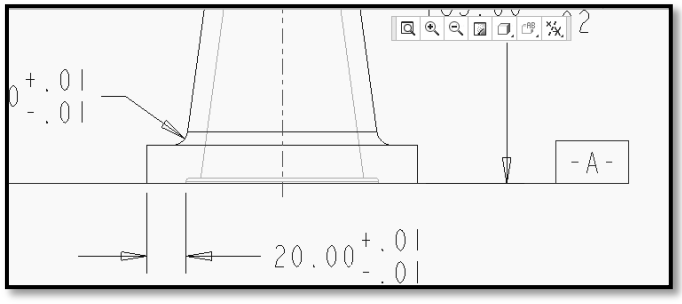


Selection of Datum Reference.

Enter the name of datum as **A** and click right button from the **Display** frame 

to enclose the datum inside a feature control frame.

b) Click Define, select Through> Plane option and select BASE plane of the screw jack body and click Done. Reference datum is created.



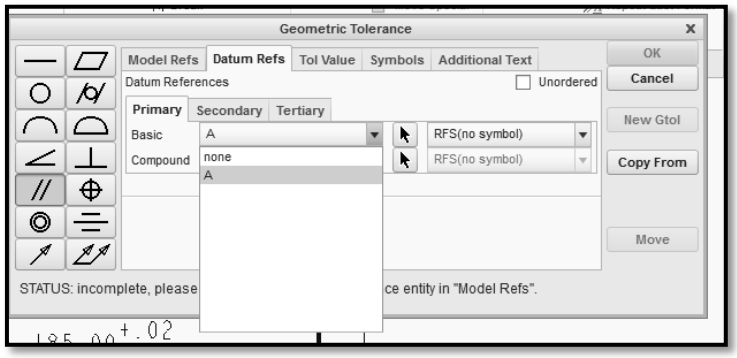
Application of Datum Reference.

9. **Show Geometric Tolerance :**

We have place reference datum either on views or on a selected dimensions. To add geometric tolerance invoke **Annotate> Geometric Tolerance** from ribbon menu.

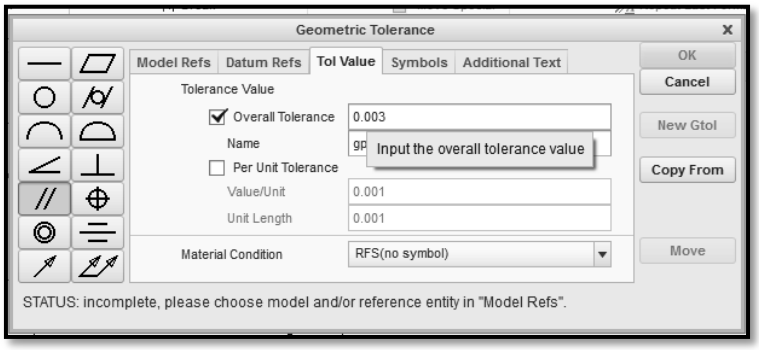
a) Click on  parallel button.

b) Select **Datum Ref.** tab option and select **Primary A.**



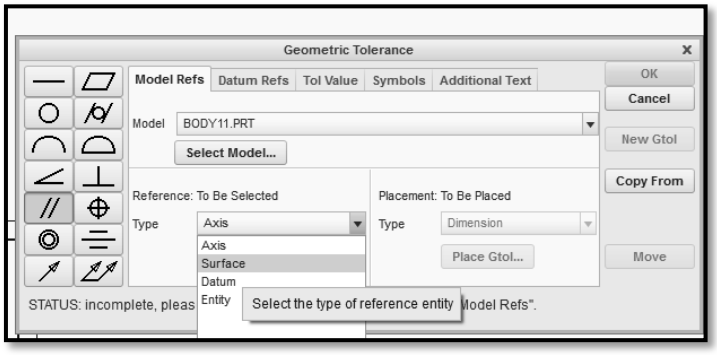
(a)– Geometric Tolerance.

c) Select **Tol Value** option and enter the tolerance value = 0.003



(b)– Geometric Tolerance.

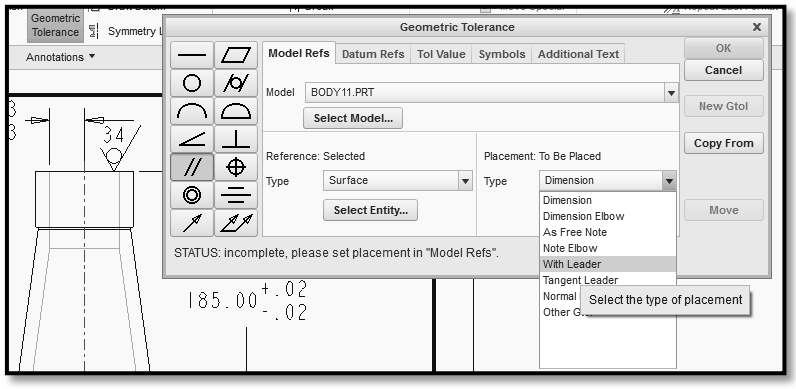
d) **Click Model Refs tab**. Select **surface** option as shown in Figure. Then select the surface which parallel to base surface of the screw jack body.



(a)– Selection of Surface reference.

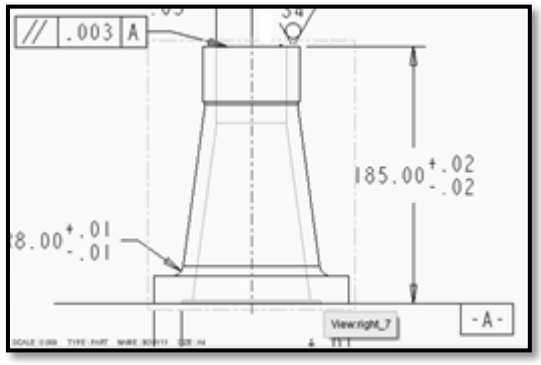
(b)– Selection of Surface reference.

e) Select Type of Placement as **With Leader** and clickon the same**Top plane of body sketch** placement. Click on **Done.**

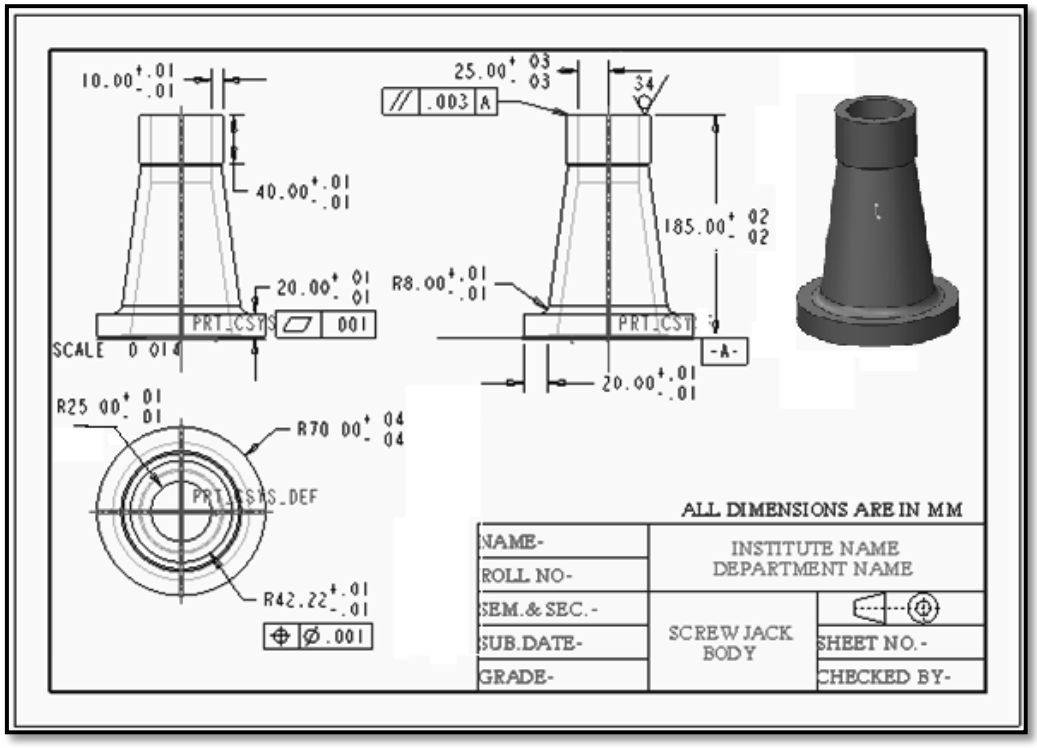


Geometric Tolerance.

f) Finally click OK button Geometric Tolerance is created as shown in Figure.



View with Tolerance.



Drawing views of body part with all aspects.

Continued in Next Practical – -----

XI Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XII Actual Procedure Followed

.....

.....

.....

.....

.....

.....

XIII Precautions Followed

.....

.....

.....

.....

.....

.....

XIV Course Proficiency

.....

.....

.....

.....

.....

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Explain the procedure for Projection views.
2. Create drawing views of all the individual components (models) of Tool Post and Print it.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XVI References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=qlXXN872GqA&feature=youtu.be
2. https://www.youtube.com/watch?v=mBScLZ5yy58

XVII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.13: Draw and Print the production drawing of all individual components part models of assembly developed in PrO 5 to 8.
---------- Continued From Previous Practical.

I Practical Significance

Drawing and documentation are essential for any product design, because they provide guidance in the manufacturing of mechanical devices. While creating part model we might have given various dimensions, geometric constraints, these details are used in drawing mode directly. Drawing views provide a means communication between the design engineers and production personnel. By studying this practical one can automatically create orthographic and detail views of either a 3D model or an assembly. Also here we can select multiple item types such as Dimension, Axis, Surface Finish and their options at a time.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills for the industry identified competency –

- Automation – borders, title blocks, views.
- Creating of General and projection views.
- Apply dimensions to the drawing views.

IV Relevant Course Outcome(s)

- Generate orthographic views of 3D solid models using Drawing workbench of any parametric CAD software.
- Generate production drawing for given part models.

V Practical Outcome

- Use any available parametric CAD modeling software to create and print orthographic views of individual models for any engineering products.

VI Relative Affective Domain-

- Working in team work.
- Practice good housekeeping.
- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Minimum Theoretical Background

- Knowledge of drawing workbench of CAD modeling software.
- Types and Method of dimensions.
- Basic knowledge of reference projection views.
- Geometrical &Dimensional Tolerances (GD&T).

VIII Resources Required

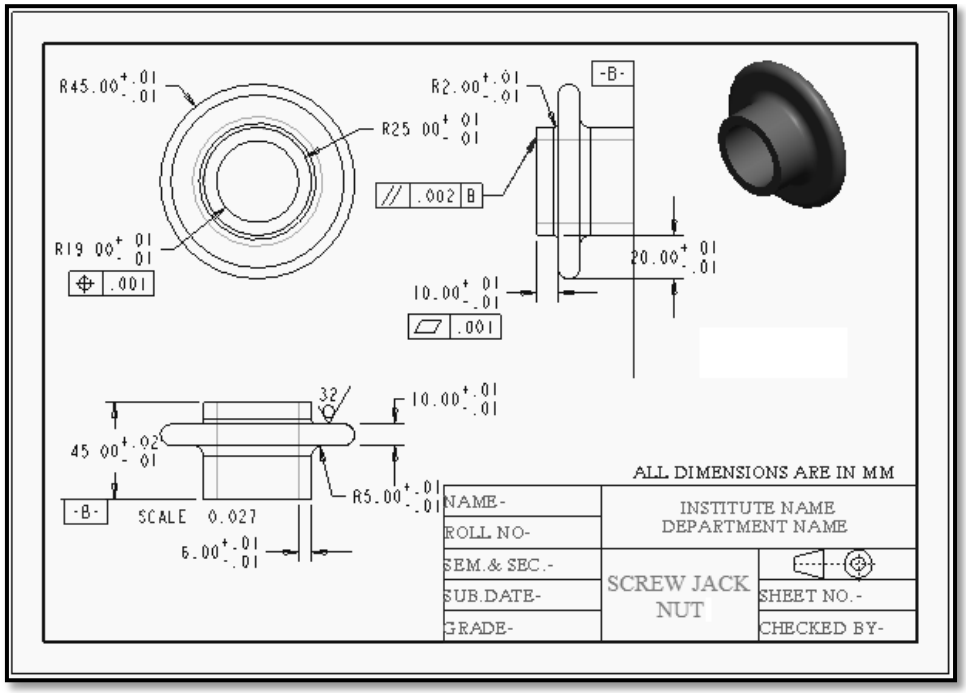
S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter and 3D printer	3D printer / Rapid prototypingMachine.	1

IX Precautions to be Followed

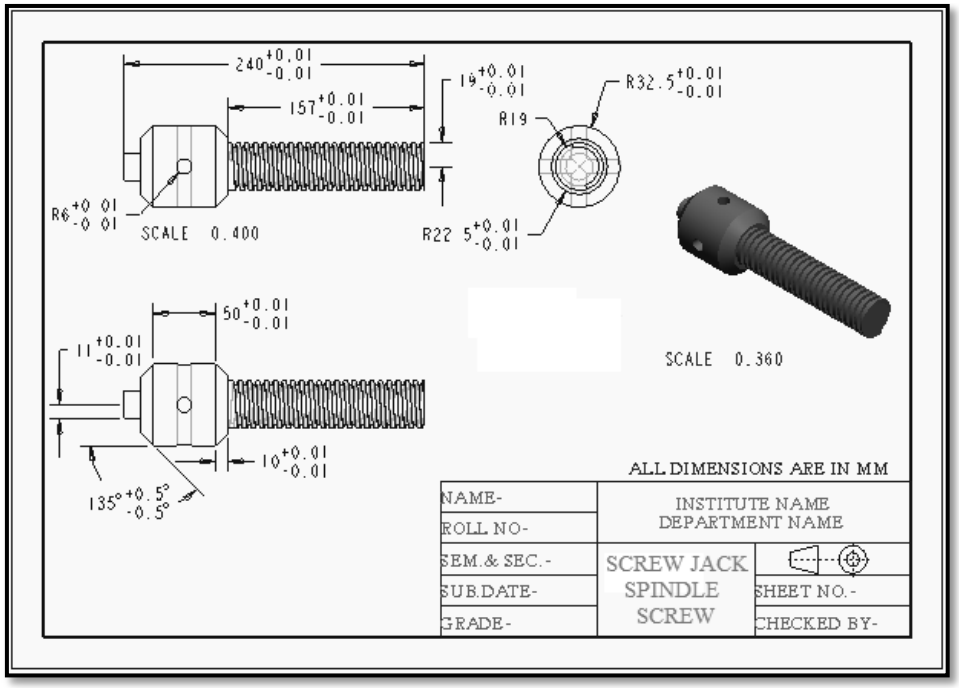
- Units used while designing the individual models should be properly selected.
- While taking drawing views, general view should take carefully.
- Care should be taken while applying dimensions, like placement of dimension and should not repeat the same dimension.

X Procedure

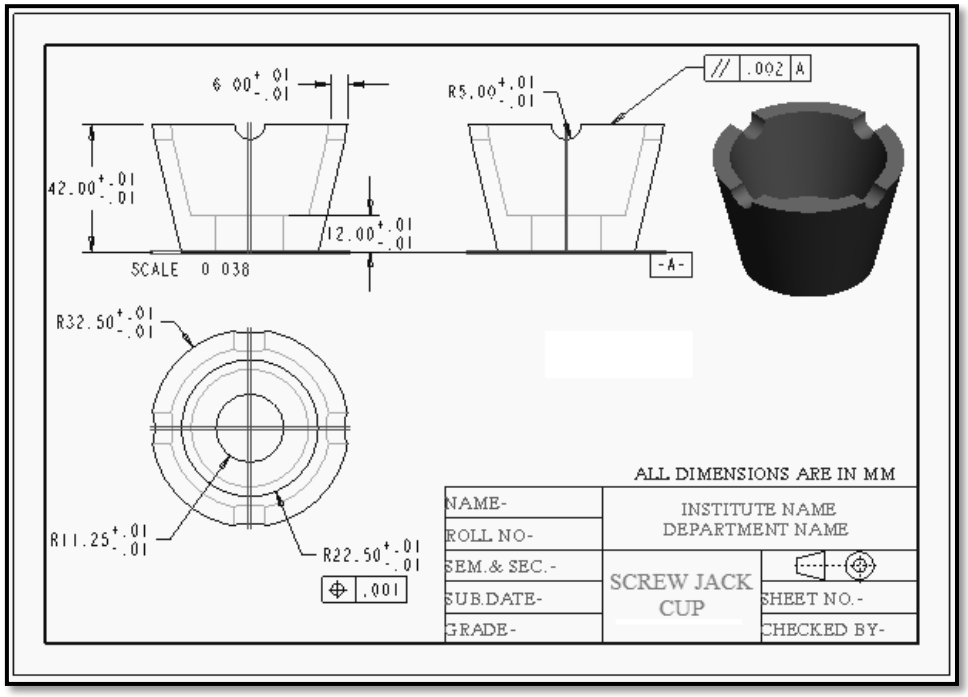
To get the Drawing views of Nut, Spindle Screw & Cup components (models) of Screw Jack individually, follow same steps explained in Practical No.-12.



Nut part drawing views with all aspects.



Screw Spindle part drawing views with all aspects.



Cup part drawing views with all aspects.

XI Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XII Actual Procedure Followed

.....

.....

.....

.....

XIII Precautions Followed

.....

.....

.....

.....

XIV Course Proficiency

.....

.....

.....

.....

.....

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Different template options available in drawing.
2. Create drawing views of all the individual components (models) of Bench vice and Print it.

[Space for Answer]

.....

.....

.....

.....

.....

XVI References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=qlXXN872GqA&feature=youtu.be
2. https://www.youtube.com/watch?v=mBScLZ5yy58

XVII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.14: Draw and Print the production drawing of all individual components part models of assembly developed in PrO 5 to 8.

---------- Continued From Previous Practical –

I Practical Significance

Drawing and documentation are essential for any product design, because they provide guidance in the manufacturing of mechanical devices. While creating part model we might have given various dimensions, geometric constraints, these details are used in drawing mode directly. Drawing views provide a means communication between the design engineers and production personnel. By studying this practical one can automatically create orthographic and detail views of either a 3D model or an assembly. Also here we can select multiple item types such as Dimension, Axis, Surface Finish and their options at a time.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills for the industry identified competency –

1. Opening an existing component.
2. Starting a new drawing - paper size, template.
3. Automation – borders, title blocks, views.
4. Creating of General and projection views.
5. Apply dimensions to the drawing views.

IV Relevant Course Outcome(s)

- Generate orthographic views of 3D solid models using Drawing workbench of any parametric CAD software.
- Generate production drawing for given part models.

V Practical Outcome

- Use any available parametric CAD modeling software to create and print orthographic views of individual models for any engineering products.

VI Relative Affective Domain-

- Working in team work.
- Practice good housekeeping.
- Demonstrate working as a leader/a team member.

VII Minimum Theoretical Background

- Knowledge of drawing workbench of CAD modeling software.
- Types and Method of dimensions.
- Basic knowledge of reference projection views.
- Geometrical and Dimensional Tolerances (GD&T).

VIII Resources Required

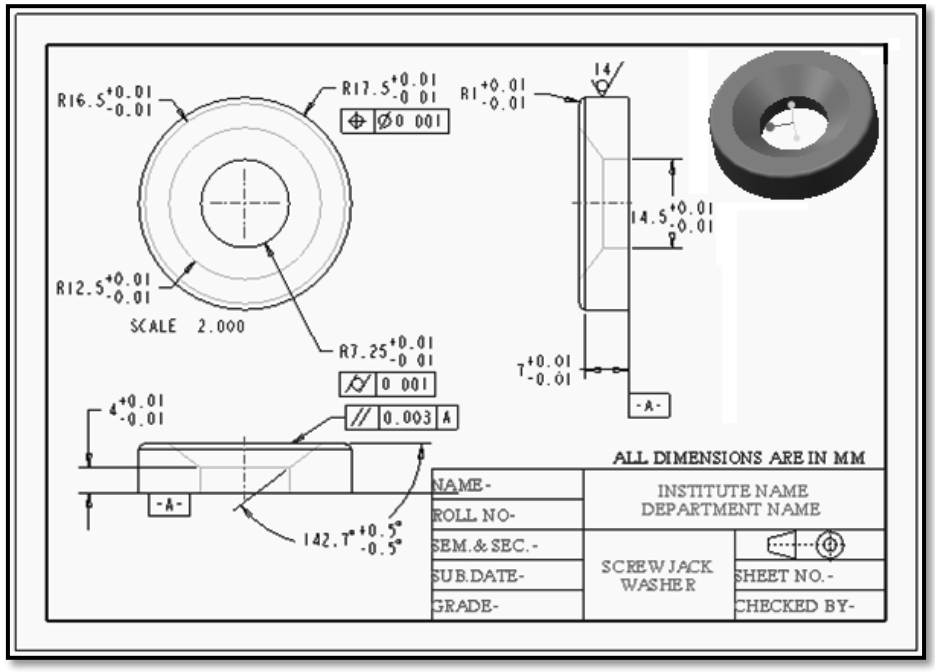
S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	Plotter and 3D printer	3D printer / Rapid prototyping Machine.	1

IX Precautions to be Followed

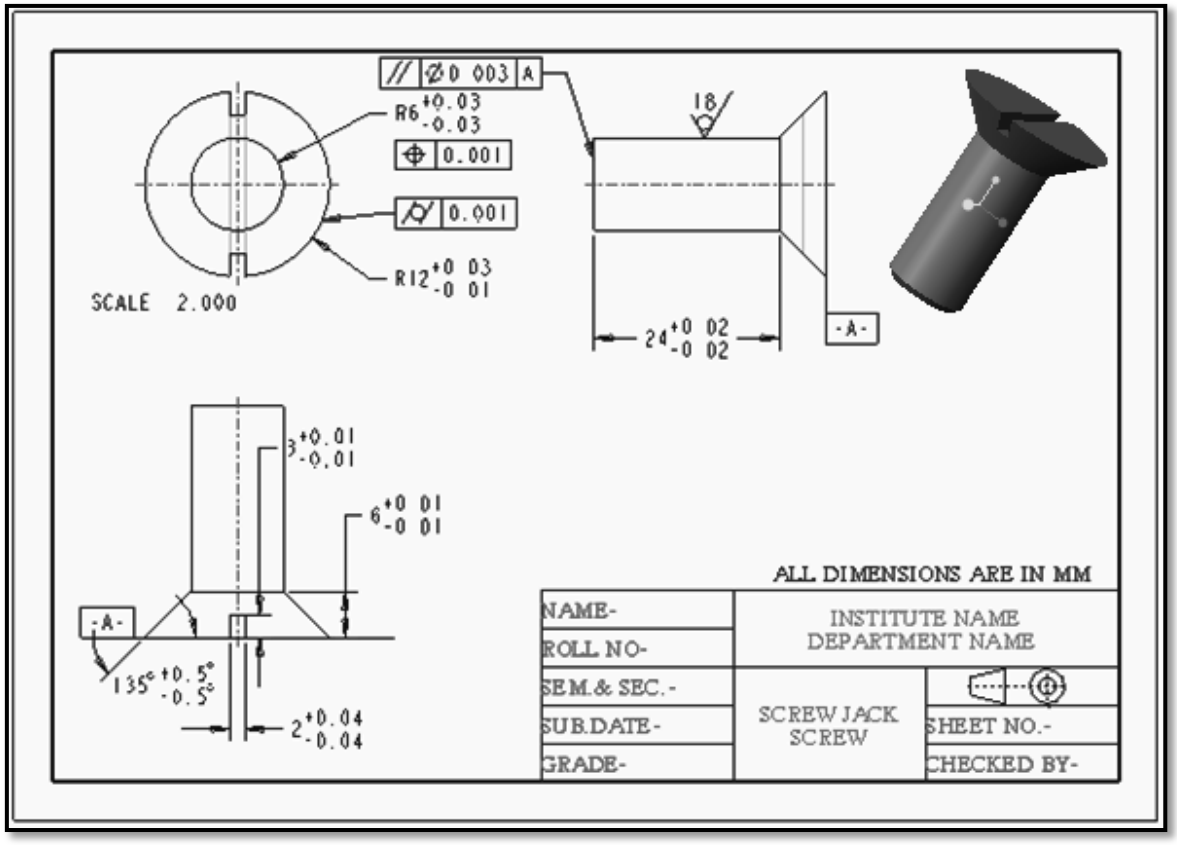
1. Units used while designing the individual models should be properly selected.
2. While taking drawing views, general view should take carefully.

X Procedure

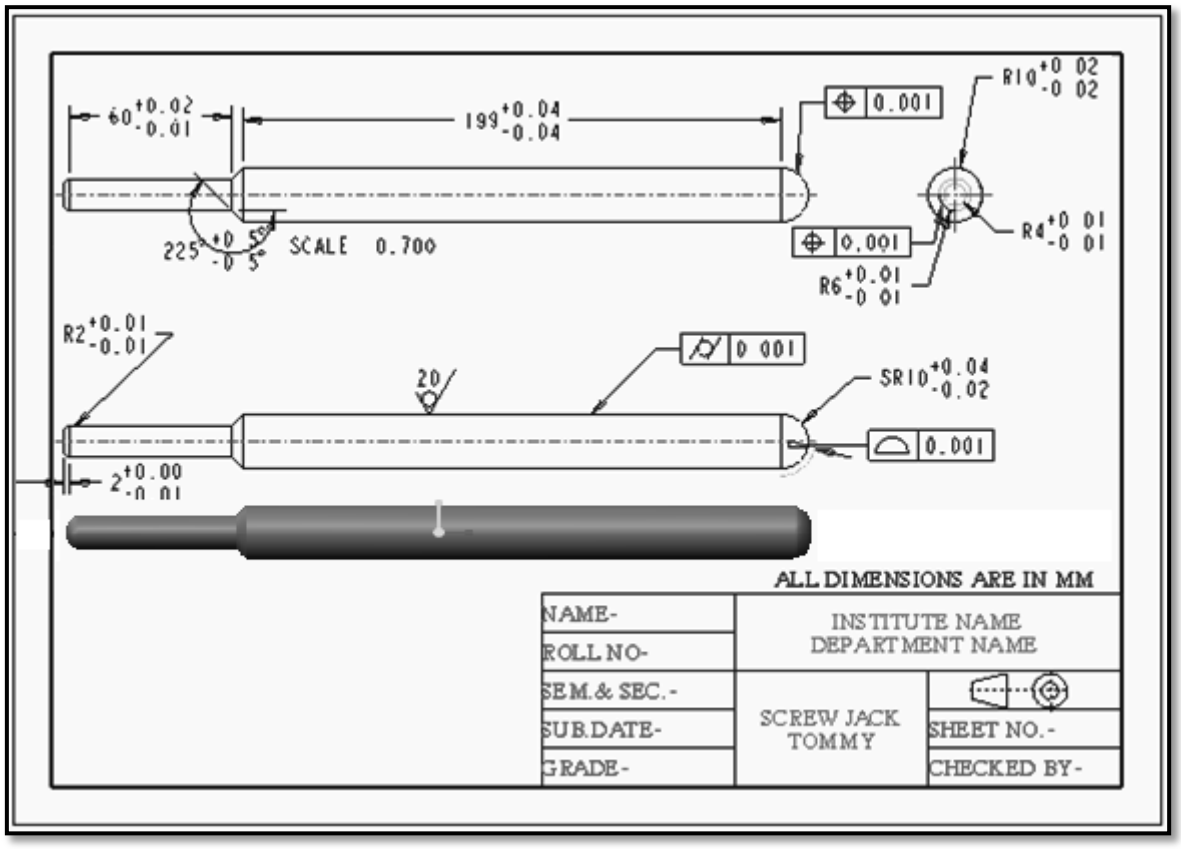
To get the Drawing views of Washer, Screw & Tommy Bar components (models) of Screw Jack individually, follow same steps explained in Practical No.-12.



Washer part drawing views with all aspects.



Screw part drawing views with all aspects.



Tommy part drawing views with all aspects.

XI Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XII Actual Procedure Followed

.....

.....

.....

.....

.....

XIII Precautions Followed

.....

.....

.....

.....

.....

XIV Course Proficiency

.....

.....

.....

.....

.....

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Explain the procedure of dimensioning on drawing views.
2. Create drawing views of all the individual components (models) of Drill Jig and Print it.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XVI References / Suggestions for Further Reading

1. https://www.youtube.com/watch?v=qlXXN872GqA&feature=youtu.be
2. https://www.youtube.com/watch?v=mBScLZ5yy58

XVII Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**40**%
1	Use of proper commands.	20%
2	Completion of drawing with minimum size of model tree.	20%
Product Related (15 Marks)		**60**%
3	Generation and printing of drawing views, tables, etc. and their arrangement on different sheet size.	20%
4	Able to answer oral questions.	20%
5	Completion of work in time.	20%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.15: Print One Simple Component Using 3D Printer/Rapid Prototyping Machine.

I. Practical Significance

3D Printing technology could revolutionize and re-shape the world. Advances in 3D printing technology can significantly change and improve the way we manufacture products and produce goods worldwide.3D Printing can revolutionize the learning experience by helping students interact with the subject matter. Affordable 3D printers in institute may be used for a variety of applications which can aid students in finding their field of interest easier and faster. Currently there are different types of educational projects in order to attract students to the various fields by giving them the opportunity to create and fabricate their own designs using 3D printing technology. The ability to develop and present ideas is one of the most important needs in the student and society. The education system plays an important role in aiding people achieve their full potential and human development. Regarding this 3D printing can enable the creation of complex geometries which are very difficult, expensive, or impossible to be manufactured using conventional production methods.

II. Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III. Competency and Skills

This practical is expected to develop the following skills for the industry identified competency ***'Create component model in CAD software and produce it using 3D Printer.***

IV. Relevant Course Outcome(s)

- Print component using 3D Printer/Rapid prototyping machine.

V. Practical Outcome

- Design and create component model by CAD software and manufacturing it by 3D printer.

VI. Relative Affective Domain-

- Working in team work.
- Practice good housekeeping.
- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII. Minimum Theoretical Background

- Basic knowledge of computer handling.
- Basic knowledge CAD software.
- Basic knowledge of plastic material properties

Introduction to 3D Printing:

A method of manufacturing known as 'Additive manufacturing', due to the fact that instead of removing material to create a part, the process adds material in successive patterns to create the desired shape.

Main areas of use:

- Prototyping Specialized parts – aerospace, military, biomedical engineering, dental
- Hobbies and home use Future applications– medical (body parts), buildings and cars

3D printer-

Software Overview-

To operate your desktop 3D printer you will need to install a few software packages onto your PC. You will need a 3D printer host, and .STL to .GCODE generator, and optional CAD or 3D modeling software.

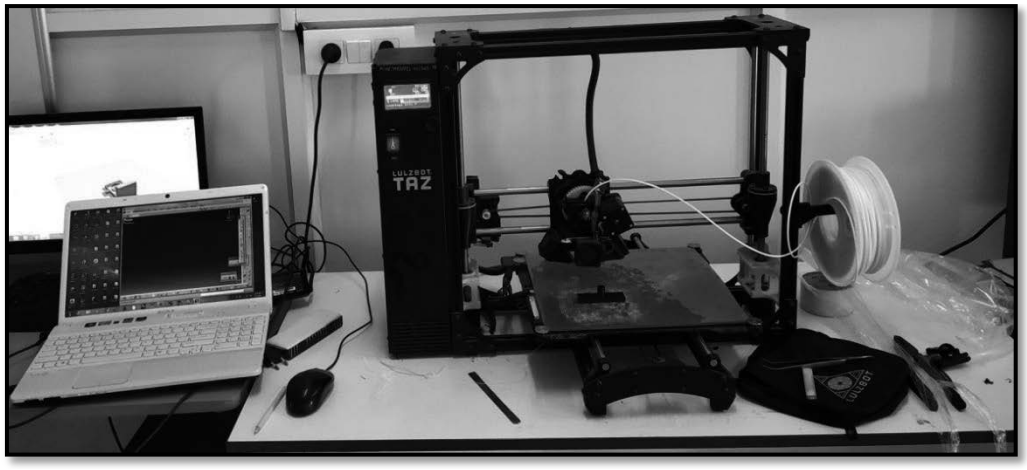
Printer Hosts-

Printer Host software is used to control the 3D printer. The program not only allows you to manually move the printer along all the axes, but set temperatures manually, send commands, and receives feedback/error messages from the onboard electronics.

CAD and 3D Modeling Software-

Other common CAD and 3D modeling software are also capable of exporting the required .STL files.

VIII. Experimental setup



IX. Resources Required

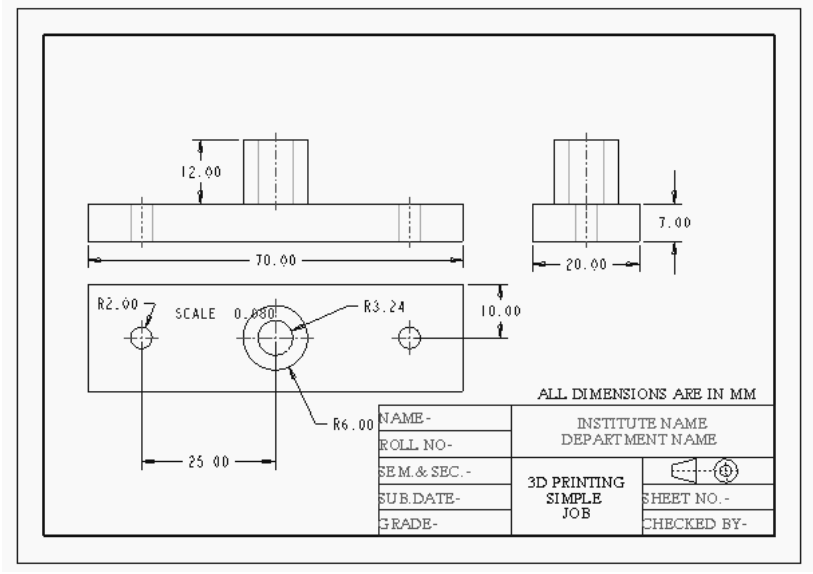
Sr. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
2.	Operating system	Windows XP/Windows 7/ Windows 8/Windows 10 or higher.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	3D printer	3D printer / Rapid prototyping Machine.	1

X. Precautions to be Followed

1. The part needs to be a solid, that is, not just a surface; it needs to have a real volume.
2. Be sure to calibrate the 3D printer before using it, it is essential to ensure that the part sticks properly to the build plate. If it does not, at some point the part may come loose and ruin the entire print job.
3. Parts with overhanging features will need supports to be printed properly. This should be taken into account since after the model needs to be cleaned by removing the supports. This may not be an issue unless the part is very delicate, since it might break.

XI. Procedure-

Following two views of the component drawing are given to manufacture by 3D printer.



The following steps are required to manufacture component by 3D printer:

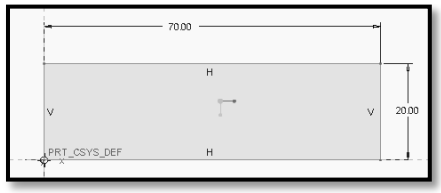
A. Starting CAD software: As explain in practical No.01.

B. Setting the Working Directory: As explain in practical No.01.

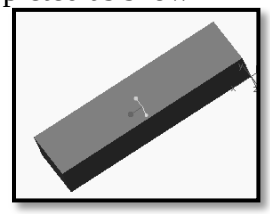
C. Starting a New Object File:As explain in practicalNo.04.

D. Selecting the Sketching Plane for the Base Feature: Asexplainimpractical No.4.

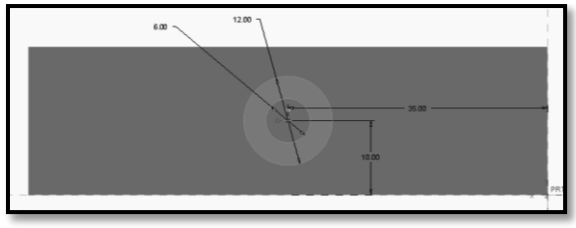
E. Creating and Dimensioning the Sketch for the Base Feature: As explain in practical No.04



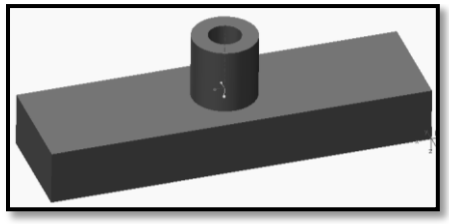
The first extruded feature is completed as shown in Figure.



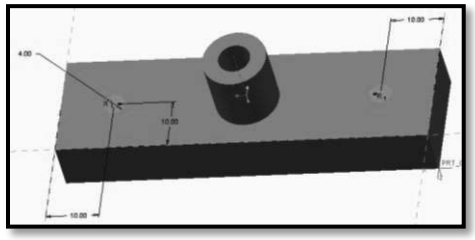
F. Creating and Dimensioning the Sketch for the second Feature: As explain in practical No.04



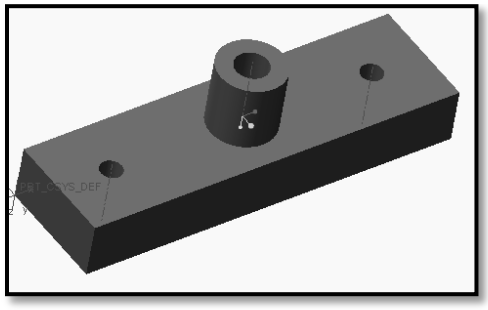
The second extruded feature is completed and as shown in Figure.



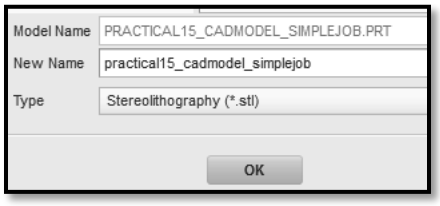
G. Creating and Dimensioning the Sketch for the third Feature: As explained in practical No.04



Finally CAD model is shown in Figure. Choose the **Save** button from the **File** toolbar and save the model.

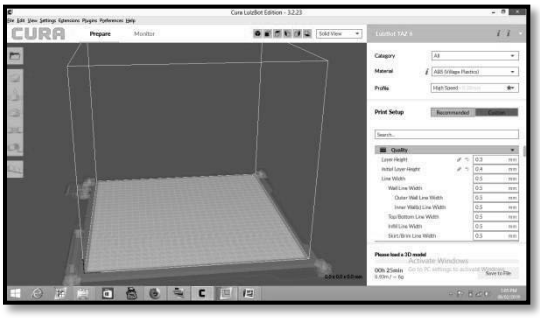


H. The important step is that you have to Save As above CAD Model with .stl file extension.



I. Start 3D print software:

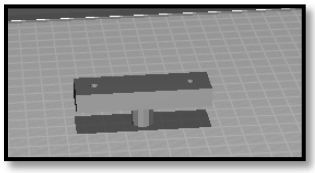
1. Start 3D printer software by double-clicking on the  icon on the desktop of your computer. (Here Additive manufacturing Edition software used for demo).
2. After setting up Additive manufacturing for the first time, you will be shown the main interface screen.



J. Load Model File:

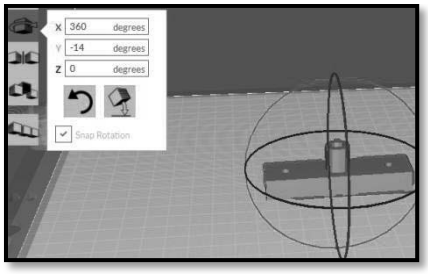
1.Select the model you would like to print. Select **Open File**  > Load Model. Once the file has been loaded, you will see a 3D rendering of your object on the build platform.

2. Select  **PRACTICAL15_CADMODEL_SIMPLEJOB.stl** file from working directory as shown in Figure.
3. CAD Model, which was saved with **.stl**file extension, will appear on the screen as shown in Figure. Select the model to see the various options.

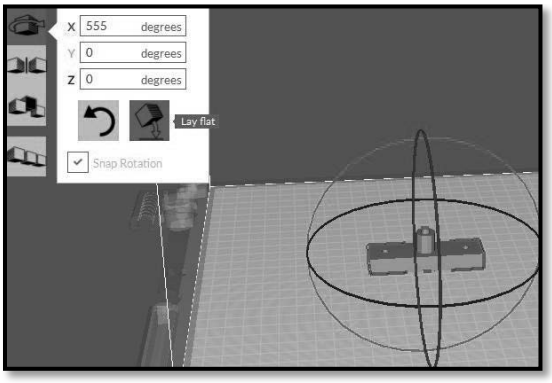


K. Model Orientation:

1. **Rotate-**The Rotate button will give you the ability to orient your model in along all three axes. Once you click the rotate button, three circles will around your model. The red circle will allow you to rotate around the X axis. The Blue circle will rotate around the Z axis. The Green circle will rotate around the Y axis.
2. Rotate model by in 360º around the X axis using rotate button.



3. **Lay Flat-**
   The Lay Flat button will ensure that the flat portion of your print is securely attached to the bed. It is highly recommended to use this option after rotating your model in the Z direction, as it will help prevent adhesion issues during the print.



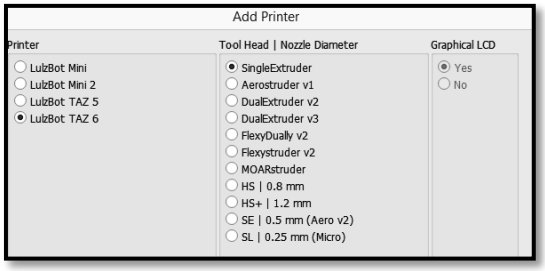
L. Material Type Selection:

1. Choose **Material** drop down menu and select ABS (Village Plastic) as shown in Figure.

M. Adding 3D printer first time:

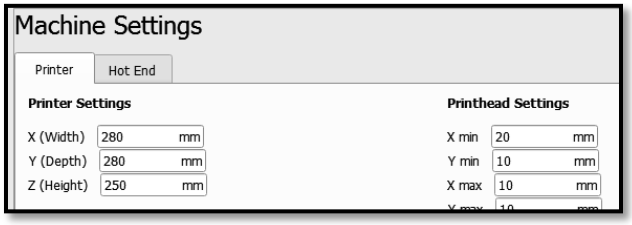
1. Select 

Add Printer option as shown in Figure. The add printer new window will display on the screen.
2. Select the Lulzbot **TAZ 6** printer.



N. Machine setting:

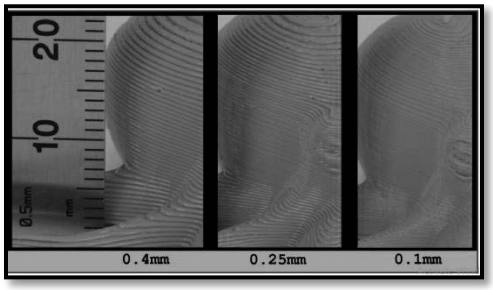
1. Specify machine setting if any as per component size.

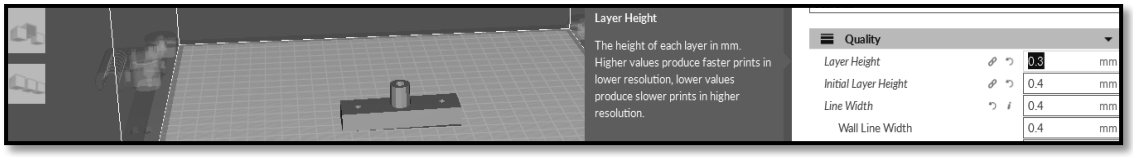


O. Process Parameter Options: Quality

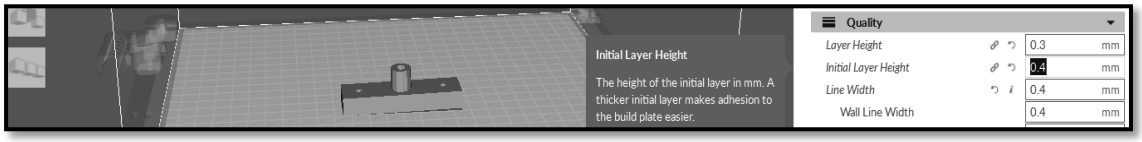
1. **Layer Height-0.3mm**

The thickness of each printed layer is known as the Layer Height. The smaller the layer height, the smoother curves will appear. Larger layer heights are better for bridging and overhang. Figure shows differences in Layer Height



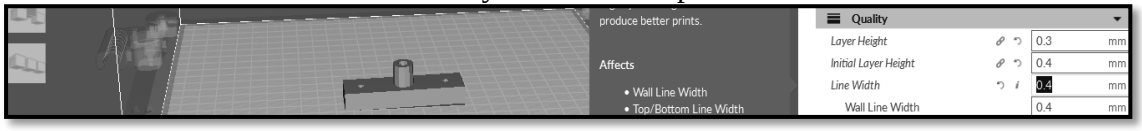


2. **Initial layer height-** The height of the initial layer in mm. A thickener initial layer makes adhesion to the build plate easier. Enter 0.4mm initial layer height in our job.

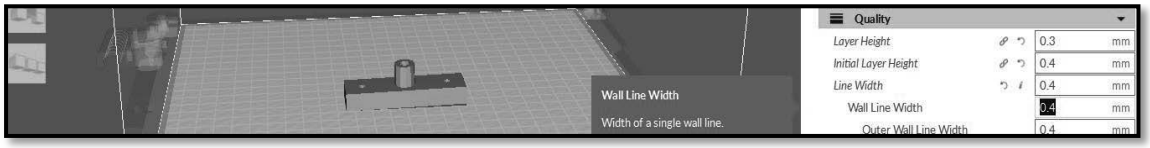


3. **Base Line Width-0.4 mm**

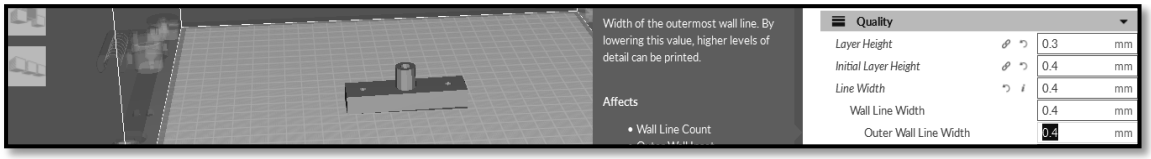
This will define how wide your "support" material is for the raft. This setting will determine how well the surface layers of the raft print.



4. **Wall line width-**This is width of a single wall line and value will be 0.4 mm is enough in our case.

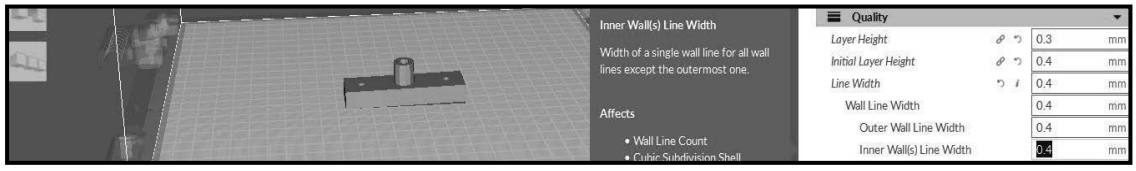


5. **Outer wall line thickness-**Width of outermost wall line by lowering the value. Higher level of value will print detail. Again 0.4 mm is sufficient for our case.

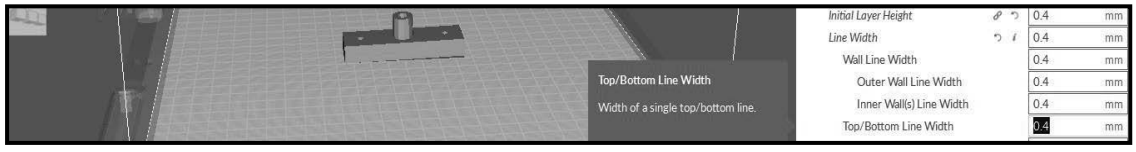


6. **Outer Wall Line Width-**Width of outermost wall line. By lowering this value, higher level of detail can be printed. Specify 0.4 mm value in our case.

7. **Inner walls line width-0.4 mm.**It is width of a single wall line for all wall lines except the outermost one. Enter 0.4 mm value for current job.



8. **Top /bottom line width-0.4 mm** width of single top/bottom line.



9. **Infill line width -0.4mm**

10. **Skirt/Brim line width-0.4 mm**

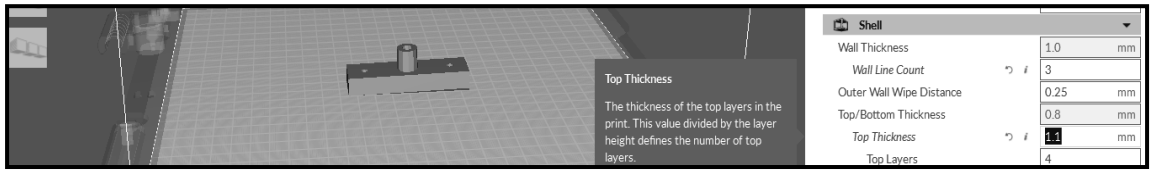
P. Process Parameter Options: Shell Setting

1. **Shell Wall Thickness-** This defines the number of vertical walls that comprise the outside of your model. We recommend keeping this set to multiples of your nozzle width. Your 3D printer is equipped with a 1 mm nozzle.
2. **Wall line count**-3

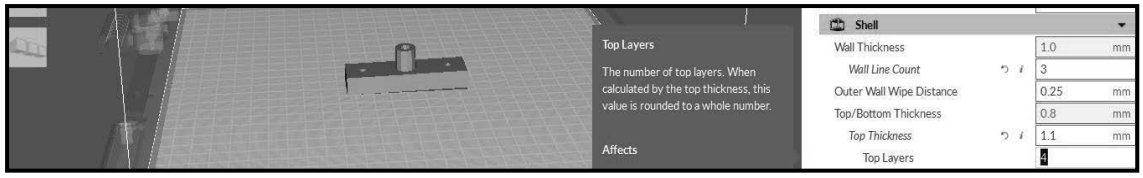
3. **Outer wall wipe distance-**0.25 mm

4. **Bottom/Top Thickness (mm)-**

Also known as Surface Layers- this will determine how thick the top and bottom layers are. A larger number here will create a thicker top and bottom which can be helpful for strength, bridging, and quality purposes. We recommend keeping this number as a multiple of your layer height. For current job mention the value as 0.8 mm.



5. **Top layers**- 4

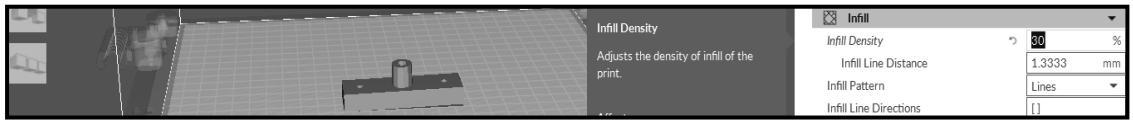


6. **Bottom layers**-2

7. **Top /bottom pattern**- Lines, Bottom pattern initial layers-lines, Outer wall inset- 0.05 mm

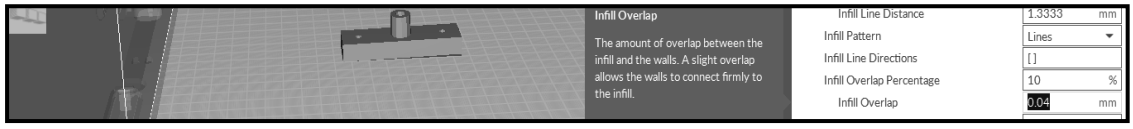
Q. Process Parameter Options: Infill Setting-

1. **Infill Density**- This number is expressed as a percentage. 0% will give a completely hollow print, while 100% will give you a completely solid object. We have found that 20% to 40% fill density is functional for most prints. For current job use **30 %** infill density.

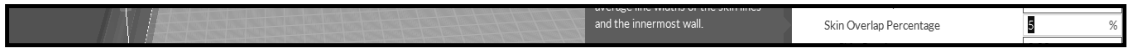


2. Infill line distance-1.33mm, Infill pattern-Lines

3. Infill Overlap-0.04 mm



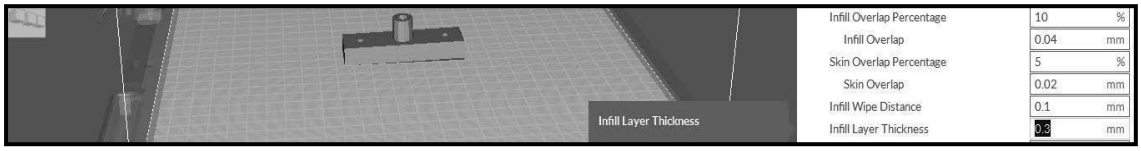
4. Skin overlap percentage- 5%



5. Skin overlap-0.02 mm

6. Infill wipe distance-0.1 mm

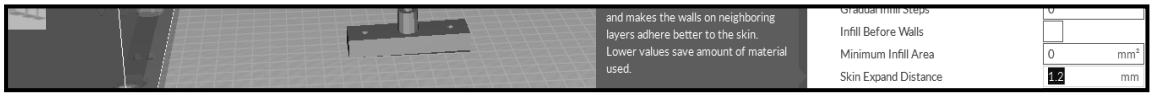
8. Infill Layer Thickness-This will control how thick your first printed layer height is printed onto the heated bed. Having a larger initial layer height will help prevent your part from popping off the plate.



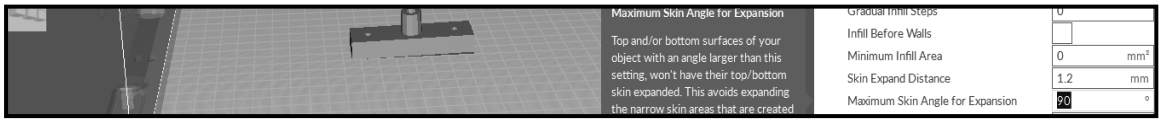
7. Gradual infill steps-0

8. Minimum infill area-0 mm^2

9. Skin expand distance-1.2 mm



10. Minimum skin angle for expansion-90º
11. Minimum skin width for expansion-00

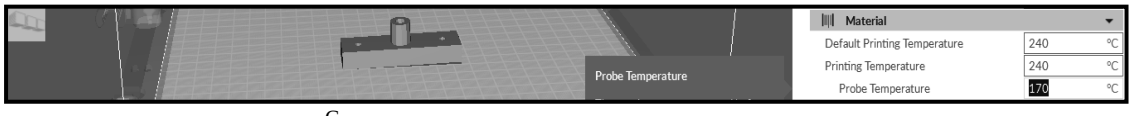


R. Process Parameter Options: Material setting

1. Default printing temperature-240º C

2.Printing temperature-240ºC

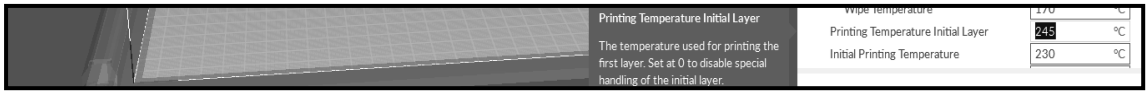
3.Probe temperature-170º C



4.Soften temperature-170º C

5.Wipe temperature-170ºC

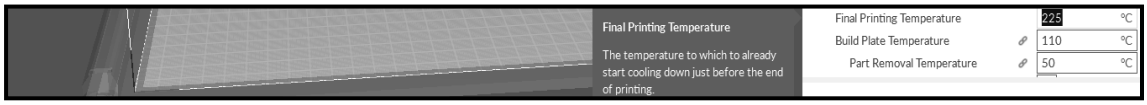
6.Printing temperature initial layer-245º C



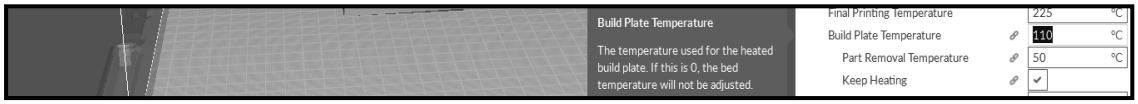
S. Process Parameter Options: Printing Temperature

When using different filament materials you'll need to update the desired hot end and heated bed temperature. Any temperatures specified here will be used to automatically set both the hot end and heated bed. Your print will not begin until these temperatures are met. The current job needs 240º temperatures.

1. Final printing temperature-225 $^{o\ C}$



2. Build plate temperature-110$^{o\ C}$



3. Part removal temperature- 50$^{o\ C}$

4. Keep heating-Tick
5. Build plate temperature initial layer-110^{oC}

6. Filament Diameter-

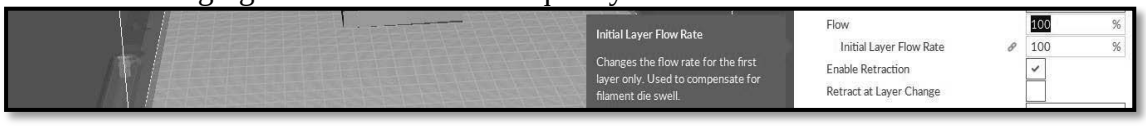
   The filament diameter setting is one of the more important settings. Make sure that you update this value periodically with your average filament diameter. While your filament may be referred to as 3mm, it is more likely going to be near 2.9mm +/- 0.1mm. You will want this to be an accurate average, as it will allow your printer to correctly calculate how much filament it is pulling into the hot end. For current case use value 2.25mm diameter.

7. Enable Retraction-Tick

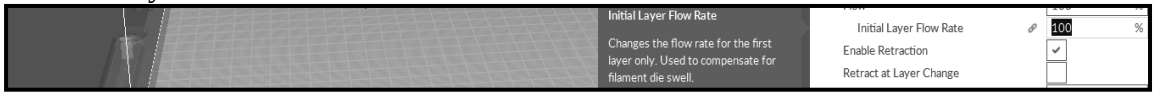
   Retraction tells your printer to pull filament out of the hot end upon travel moves. Travel moves are when your print head moves from one area of the print, to another without laying down filament. We recommend keeping this on for all filament types, and adjusting the retraction length and speed for the specific filament. For current job **Tick** the enable retraction option.

8. Filament Flow -100%

   This controls how much filament your printer is extruding in relation to speed. This setting is mainly used to adjust for filament density variations. Leave this value at 100% as changing it can lead to surface quality issues.



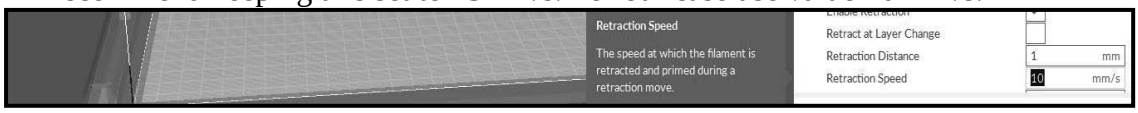
9. Initial layer flow rate- 100%



10. Retraction Distance-1 mm

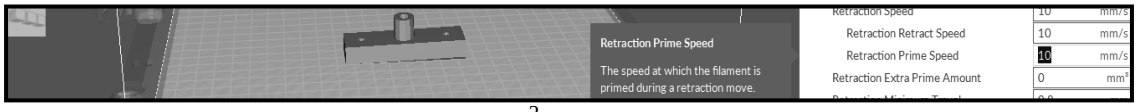
    Retraction Distance determines how much filament is pulled out of your hot end on travel moves and when changing direction. You will want to adjust this depending on temperature settings and filament type. Higher thermal retaining filaments such as PLA behave better with a longer retraction distance. We have found anywhere from 1mm to 3mm is a good starting range. For our case use 1 mm.

11. Retraction Speed-10 mm/s
    Retraction Speed determines the speed at which your filament is reversed out of the hot end for travel moves and when changing direction during printing. We recommend keeping this set to 25mm/s. For our case use value 10 mm/s.



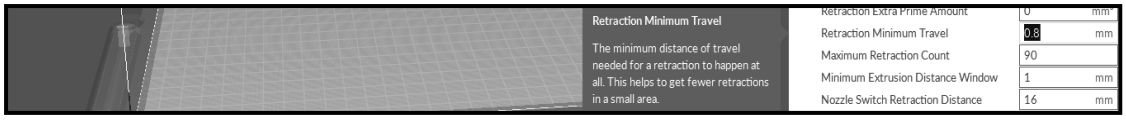
12. Retraction retract speed- 10 mm/s

13. Retraction prime speed-10 mm/s

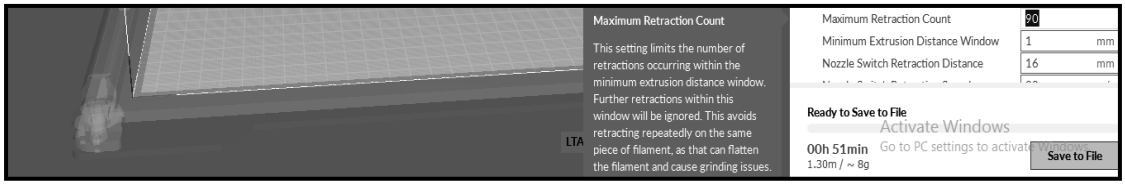


14. Retraction extra prime amount-0 mm^2

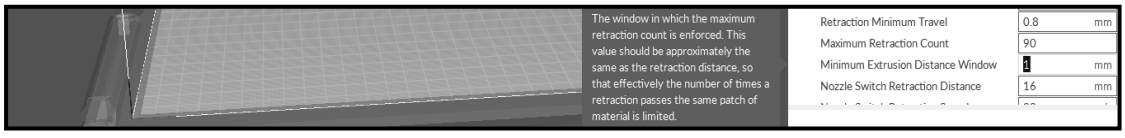
15. Retraction minimum travel-0.8 mm



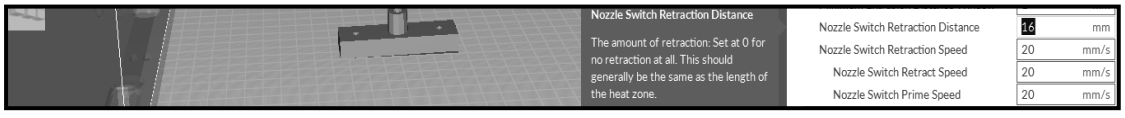
16. Maximum retraction count-90



17. Minimum extrusion distance window-1 mm



18. Nozzle switch retraction distance-16 mm



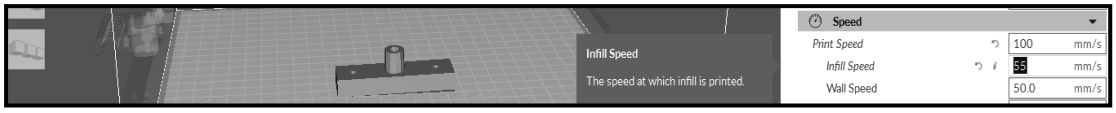
T. Process Parameter Options: Speed

1 Print speed- 100 mm/s

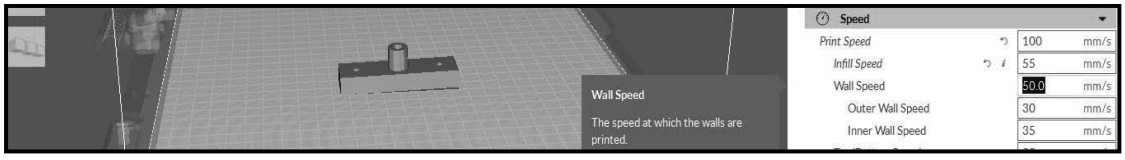
Your overall printing speed can be adjusted here. If no other speeds are determined in the later sections your printer will automatically default to this speed. This speed will be different, depending on what type of filament you are using.

2 Infill Speed-55 mm/s

This is how fast your print head speed will be while laying down the interior portion of your model. Faster speeds are usually tolerable here, as none of the infill will be visible from the outside of your object. If you go too fast compared to your inner and outer shells, you can have adhesion issues or globs of filament left behind from the print head. For current job give infill speed as 55 mm/s.

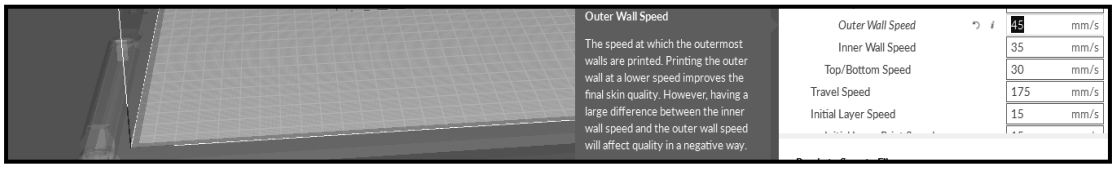


3 Wall speed-50 mm/s



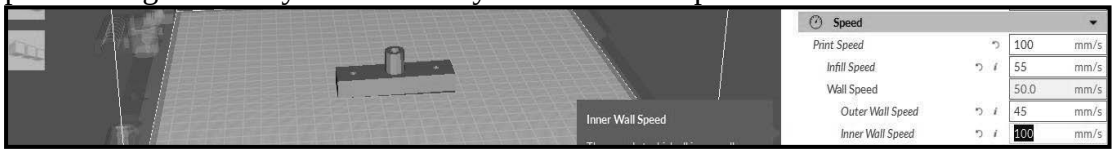
4 Outer Shell Speed-45 mm/s

This will be the outermost surface of the model. This is the most important setting, as it controls the speed of your print head on the visible layers. As a general rule of thumb, the slower you go the better looking print you will get.



5 Inner Shell Speed-100 mm/s

This affects vertical walls that are in between the outer shell and infill. This will not be visible but will help support the outer shell and the infill. We recommend keeping this speed setting between your infill and your outer shell speed.

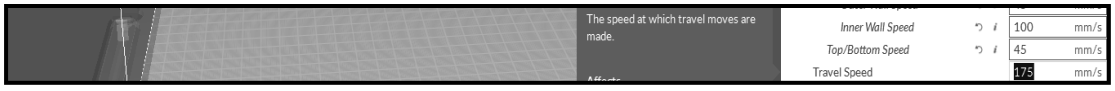


6 Bottom Layer Speed-45 mm/s

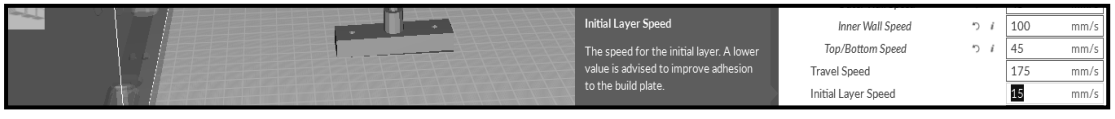
This will control your initial layer speed. In general, a slower initial layerspeed will help with first layer adhesion. In our case speed required 45mm/s

7 Travel Speed-175 mm/s

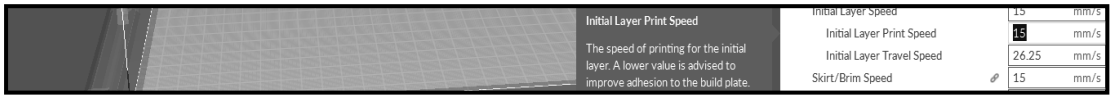
This setting will determine how fast your print head moves while not extruding filament. A normal travel speed of 125 - 150mm/s is recommended.



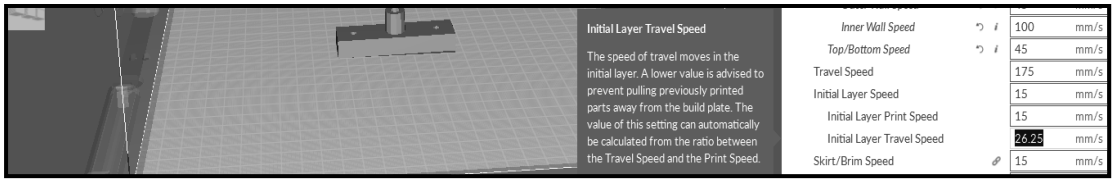
8 Initial layer speed-15 mm/s



9 Initial layer print speed-15 mm/s

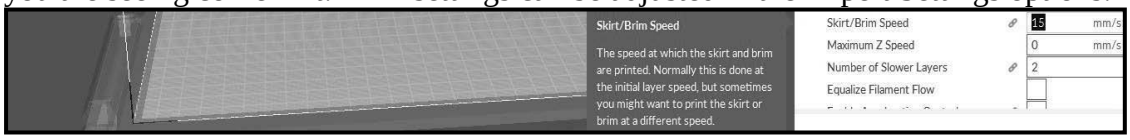


10 Initial layer travel speed- 26.26 mm/s

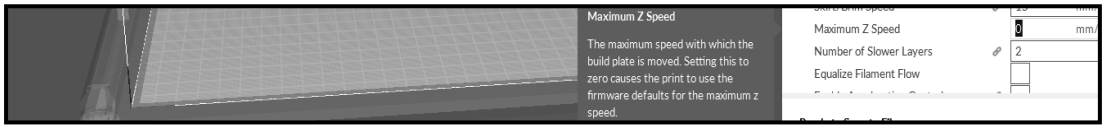


11 Brim speed-15 mm/s

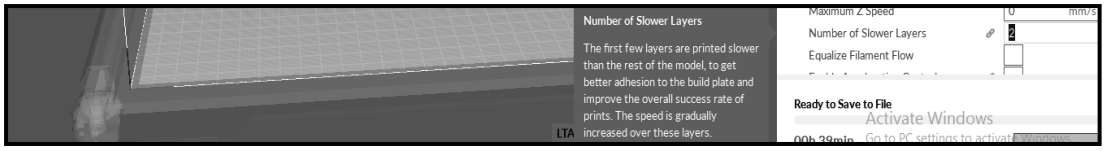
Brim will create a single layer of filament, contacting and surrounding your model. This will increase the surface area of the part contacting the build platform thereby preventing it from popping off the heated bed. Brim will also help in situations where you are seeing corner lift. Brim settings can be adjusted in the Expert Settings options.



12 Maximum Z speed-0 mm/s



13 Number of slower layers-2



U. Process Parameter Options: Support Type

1. Generate support-Tick
2. Support placement-Everywhere

Some models will require support material in order to print properly. This will usually occur when an object has an angle in relation to the build plate between 0 to 45 degrees. It is highly recommended to orient your object so that it minimizes or eliminates the need for support.This prints support material between the heated bed and object as well as between the object and itself. The green example is Support Everywhere.

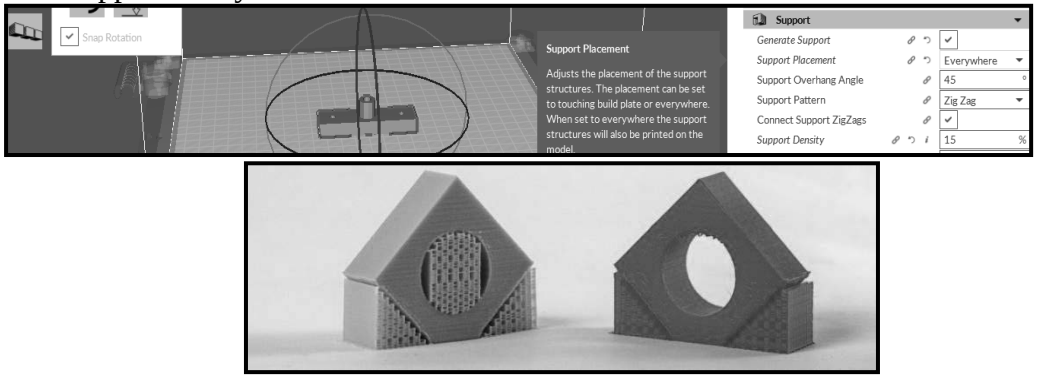


Figure- Support Types

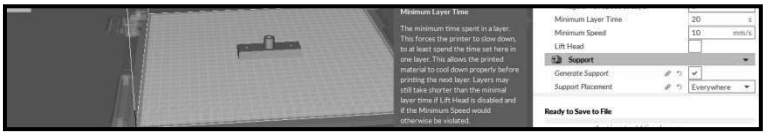
V. Process Parameter Options: Cooling

1. Enable cooling fan-Tick
2. Fan speed-100 %

This section will define how your extruder cooling fan will operate during the print. Your fan will not start until it has reached 25% or higher for speed settings. If your print speeds are slowed down due to minimal layer time, the fan will run between minimum and maximum speed based upon how much the layer is slowed down.

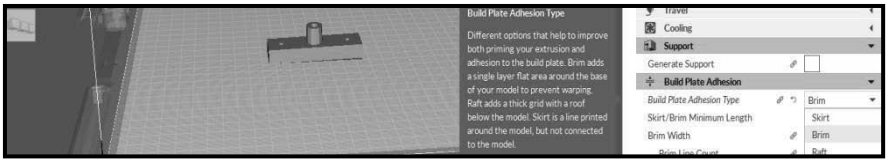
3 Minimal Layer Time-20 s

This will determine a minimum amount of time your printer will spend laying down each layer.

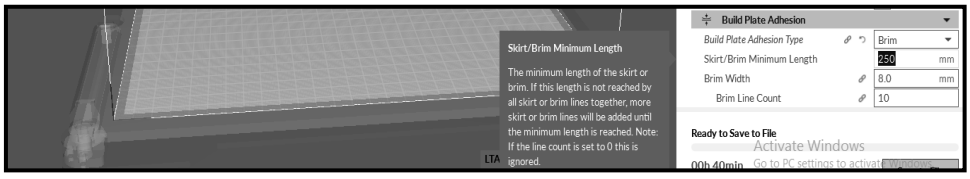


W. Process Parameter Options: Built plate adhesion setting

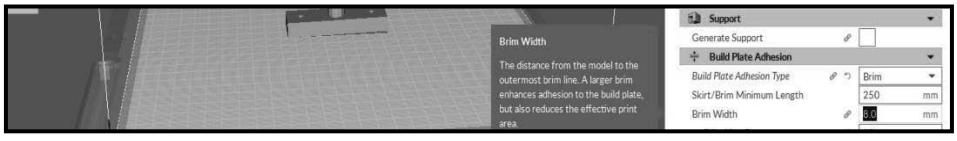
1 Build plate adhesion type- Brim



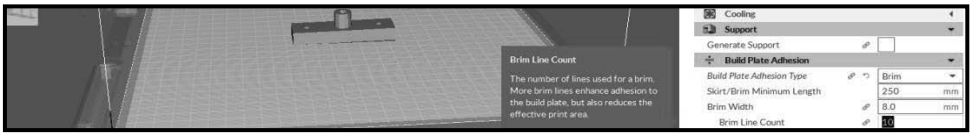
4 Start/ brim minimum length-250 mm



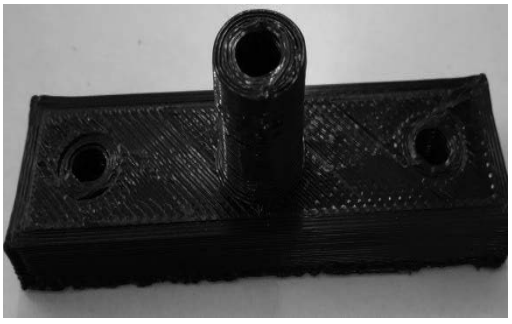
5 Brim width-80 mm



6 Brim line count-10



7 Click **Save to file** will start the 3D printing.



XII. Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					

XIII. Actual Procedure Followed

.....

.....

.....

.....

XIV. Precautions Followed

.....

.....

.....

.....

XV. Conclusions

.....

.....

.....

.....

XVI. Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Define 'Additive Manufacturing'
2. Explain the working principle of 3D Printing Technology.

[Space for Answer]

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

XVII. References / Suggestions for Further Reading

- https://www.youtube.com/watch?v=e0rYO5YI7kA
- https://www.youtube.com/watch?v=HVgPM1ojyLw

XVIII. Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**70%**
1	Selecting relevant material, process and set up parameters.	30%
2	Slicing the solid model and transferring the file to the printer.	20%
3	Printing the components.	20%
Product Related (15 Marks)		**30%**
4	Safety unloading the manufactured component from the printer/machine.	10%
5	Answer to sample questions.	10%
6	Submission of digital drawing file/plot in time.	10%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

Practical No.16: Print One Complex Component Using 3D Printer/Rapid Prototyping Machine.

I Practical Significance

3D Printing technology could revolutionize and re-shape the world. Advances in 3D printing technology can significantly change and improve the way we manufacture products and produce goods worldwide.3D Printing can revolutionize the learning experience by helping students interact with the subject matter. Affordable 3D printers in institute may be used for a variety of applications which can aid students in finding their field of interest easier and faster. Currently there are different types of educational projects in order to attract students to the various fields by giving them the opportunity to create and fabricate their own designs using 3D printing technology. The ability to develop and present ideas is one of the most important needs in the student and society. The education system plays an important role in aiding people achieve their full potential and human development. Regarding this 3D printing can enable the creation of complex geometries which are very difficult, expensive, or impossible to be manufactured using conventional production methods.

II Relevant Program Outcomes (POs)

PO2-Discipline knowledge: Apply Mechanical engineering knowledge to solve broad-based mechanical engineering related problems.

PO3-Experiments and practice: Plan to perform experiments and practices to use the results to solve broad-based Mechanical engineering problems.

PO4-Engineering tools: Apply relevant Mechanical technologies and tools with an understanding of the limitations.

PO10-Life-long learning: Engage in independent and life-long learning activities in the context of technological changes also in the Mechanical engineering and allied industry.

III Competency and Skills

This practical is expected to develop the following skills for the industry identified competency ***'Create component model in CAD software and produce it using 3D Printer.***

IV Relevant Course Outcome(s)

- Print component using 3D Printer/Rapid prototyping machine.

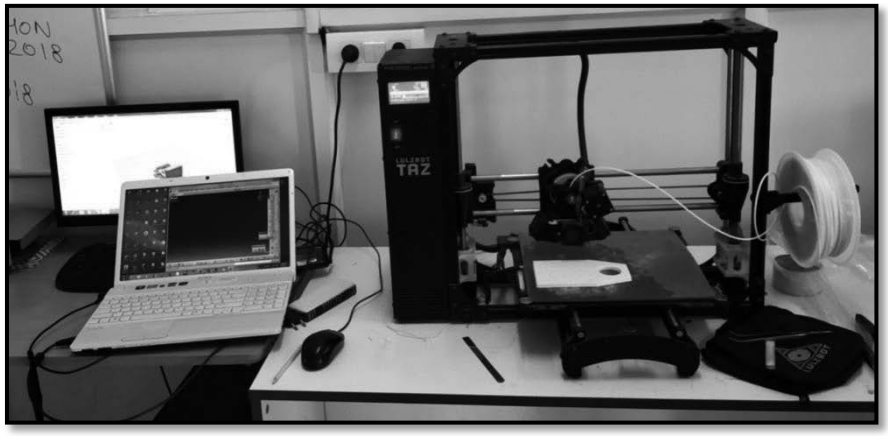
V Practical Outcome

- Design and create component model by CAD software and manufacturing it by 3D printer.

VI Relative Affective Domain-

- Working in team work.
- Practice good housekeeping.
- Demonstrate working as a leader/a team member.
- Follow ethical Practices.

VII Experimental set up:



VIII Minimum Theoretical Background

- Basic knowledge of computer handling.
- Basic knowledge CAD software.
- Basic knowledge of plastic material properties
- Basic knowledge of 3D printing technology.

IX Resources Required

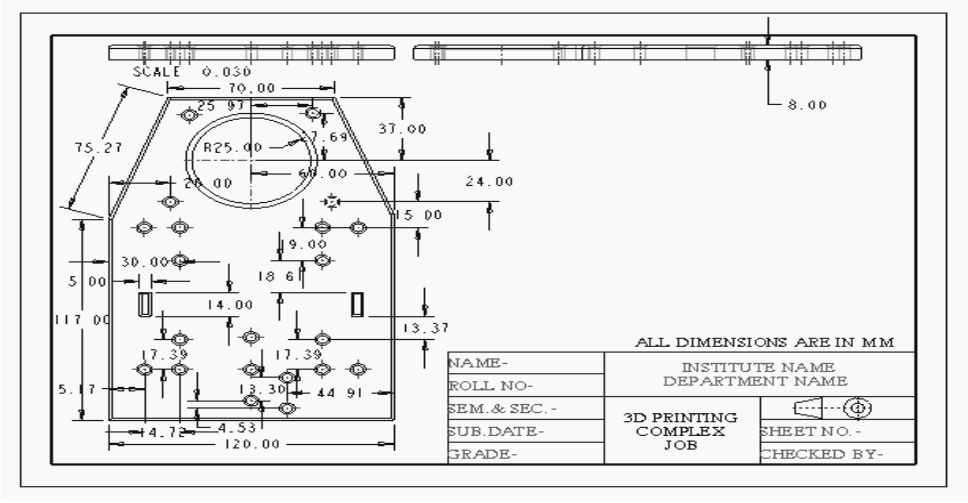
S. No.	Name of Resource	Suggested Broad Specification	Quantity
1.	Hardware: Personal computer.	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-wide Screen preferably.	As per batch size
3.	Software	Any parametric solid modeling software.	As per batch size
4.	3D printer	3D printer / Rapid prototyping Machine.	1

X Precautions to be Followed

As explained in previous practical No.01.

XI Procedure

Following views of the component drawing are given to manufacture by 3D printer.



The **following steps are required to manufacture component by 3D printer:**

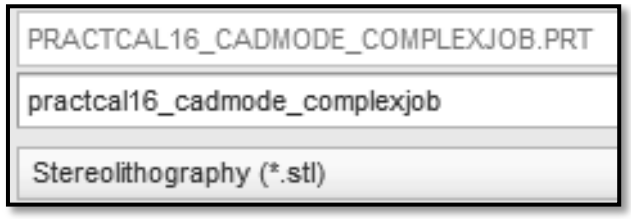
A. Starting solid modeling CAD software.-As explained in practical No. 01

B. Setting the Working Directory: As explained in practical No. 01

C. Starting a New Object File: As explain in practicalNo.04.

D. **To create CAD Model: As explained in practical No.04, 15.**CAD model is created and shown in Figure.

E. You have to **Save As** above CAD model with**.stl**file extension.



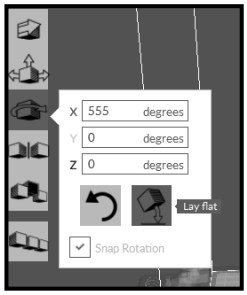
F. Start 3D Printer software: As explained in practical No.15

G. Load Model File: As explained in practical No.15

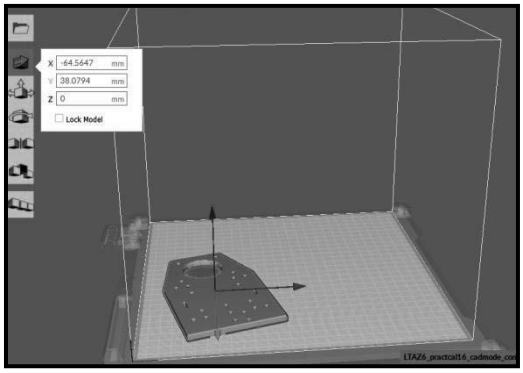
CAD Model, which was saved with **.stl** file extension, will appear on the screen as shown in Figure. Click the model for orientation.

H. Model Orientation: As explained in practical No.15

1. Rotate model by in 90º around the X axis using rotate button.
2. **Lay Flat-**The Lay Flat button will ensure that the flat portion of your print is securely attached to the bed. It is highly recommended to use this option after rotating your model in the Z direction, as it will help prevent adhesion issues during the print.



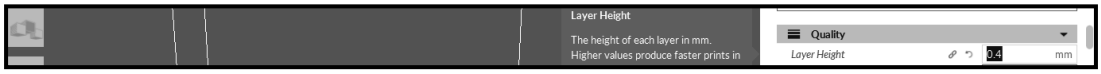
3. Resting the job on the table maintaining X= -64.56, Y=38.07 and Z=0, but in your case mostly locate the job at the center position of the printer table.



I. Material Type Selection: As explained in practical No.15

J. Process Parameter Options: Quality

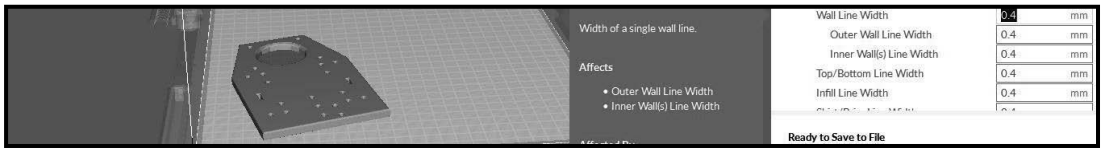
1. **Layer Height-0.4 mm**



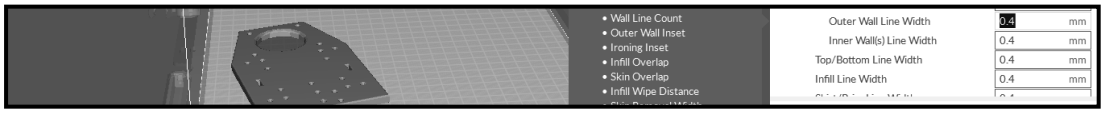
2. **Initial layer height-0.5 mm**

3 **Line Width-0.4 mm**

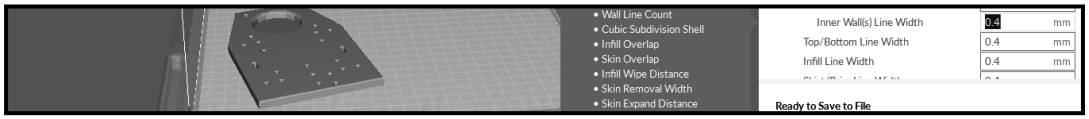
4 Wall line width-0.4 mm



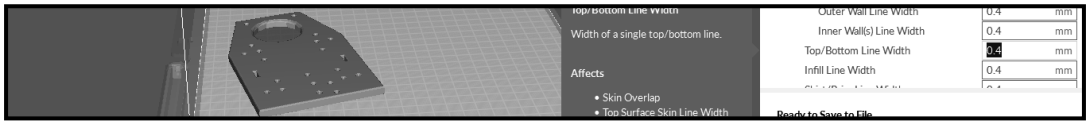
5. Outer wall line width-0.4 mm



6. Inner wall line width-0.4 mm



7. Top/Bottom line width-0.4 mm



8. Infill line width- 0.4 mm

9. Skirt/brim line width-0.4 mm

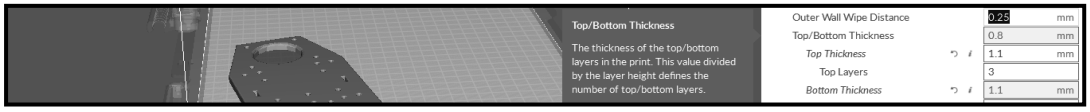
K. Process Parameter Options: Shell Setting:

1 Wall thickness-10 mm

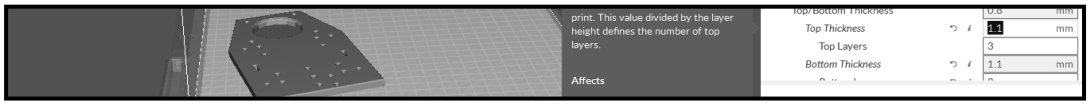
2 Wall line count-3

3 Outer wall wipe distance-0.25 mm

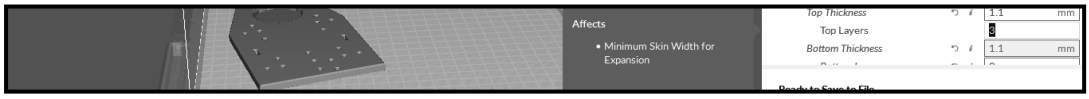
4 Top/Bottom thickness- 0.8 mm



5 Top thickness-1.1mm



6 Top layers-03



7 Bottom thickness-1.1 mm

8 Bottom layers-03

9 Outer wall inset-0.05 mm

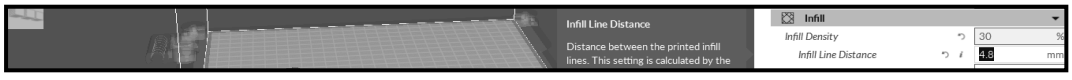
10 Horizontal expansion-00 mm

11 Extra skin wall count-01

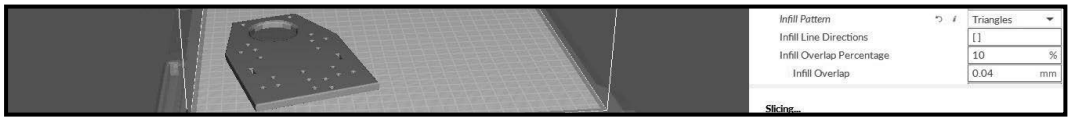
L. Process Parameter Options:Infill Setting:

1 Infill density- 30%

2 Infill line distance-4.8 mm



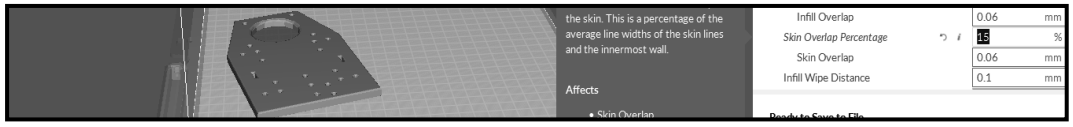
3 Infill pattern- Triangle



4 Infill overlap percentage-15 %

5 Infill overlap-0.06 mm

6 Skin overlap percentage-15 %



7 Skin overlap- 0.06 mm

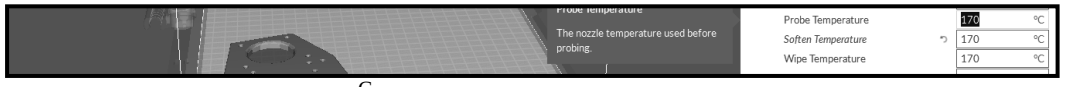
8 Infill wipe distance-0.1 mm

M. Process Parameter Options:Material setting

1 Default printing temperature-250º C

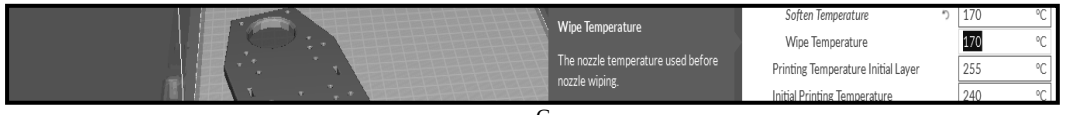
2 Printing temperature-250º C

3 Probe temperature-170 º C

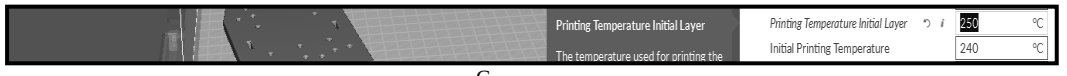


4 Soften temperature-170 º C

5 Wipe temperature-170 º C



6 Printing temperature initial layer-250º C

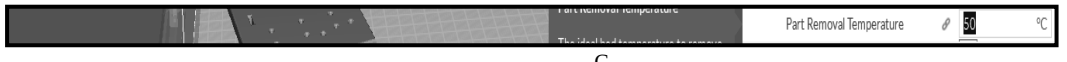


7 Initial printing temperature-250º C

8 Final printing temperature-250º C

9 Build plate temperature-110º C

10 Part removal temperature-50º C

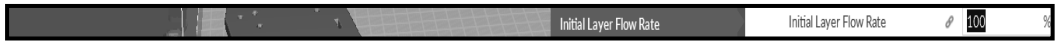


11 Build plate temperature initial layer-110 º C

12 Diameter-2.25 mm

13 Flow percentage-100%

14 Initial layer flow rate-100%

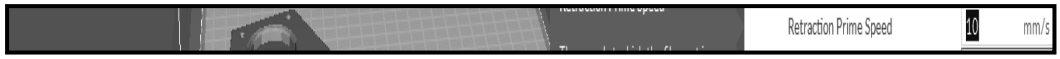


15 Retraction distance-1mm

16 Retraction speed-10 mm/s

17 Retraction retract speed-10 mm/s

18 Retraction prime speed-10 mm/s



19 Retraction extra prime amount-00 mm^2

20 Retraction minimum travel-0.8 mm

21 Maximum retraction count-90

22 Minimum extrusion distance window-1 mm

23 Nozzel switch retraction distance-16 mm

24 Nozzle switch retraction speed-20 mm/s

25 Nozzle switch retract speed-20 mm/s

26 Nozzle switch prime speed-20 mm/s

N. **Process Parameter Options: Speed**

1 Print speed-150 mm/s

2 Infill speed-150 mm/s

3 Wall speed-75 mm/s

4 Outer wall speed-50 mm/s

5 Inner wall speed-70 mm/s

6 Top/Bottom speed-70 mm/s

7 Travel speed-180 mm/s

8 Initial layer speed-15 mm/s

O. Process Parameter Options: Support

1 Generate support-Tick

2 Support stair step height-0.3 mm

3 Support stair step maximum width-5 mm

4 Tower roof angle-65º

P. Process Parameter Options: Built plate adhesion setting

1 Build plate adhesion type-Brim

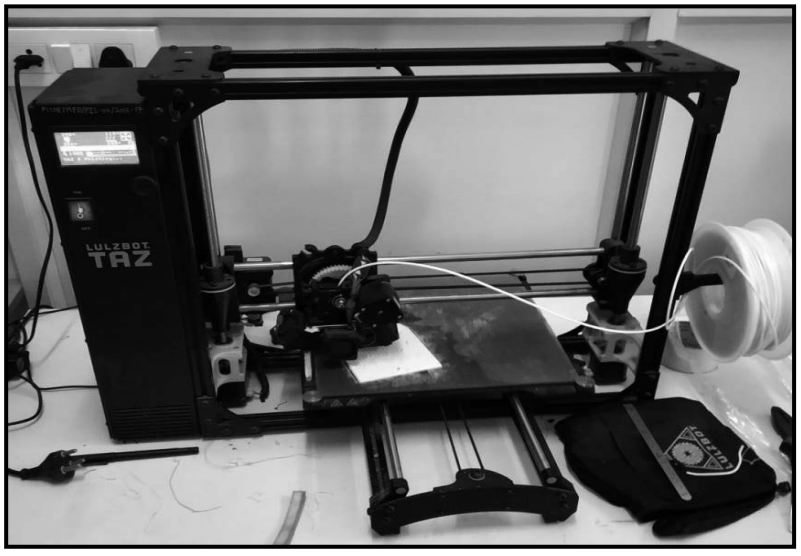
2 Skirt/Brim maximum length-200 mm

3 Brim width-1 mm

4 Brim line count-10

5 Save file in working directory.

6 Click **Save to file** will start the 3D printing.



7 Remove component carefully from printer table.

XII Resources Used

S. No.	Name of Resource	Broad Specifications		Quantity	Remarks (If any)
		Make	Details		
1.					
2.					
3.					

XIII Actual Procedure Followed

.....
.....
.....
.....

XIV Precautions Followed

.....
.....
.....
.....

XV Conclusions

.....
.....
.....
.....

XVI Practical Related Questions

Note: Below given are few sample questions for reference. Teachers must design more such questions as to ensure the achievement of identified CO.

1. Explain working principle of SLA – Stereo lithography.
2. Write major benefits of 3D printing manufacturing technology.

[Space for Answer]

.....
.....
.....
.....
.....
.....
.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

.....

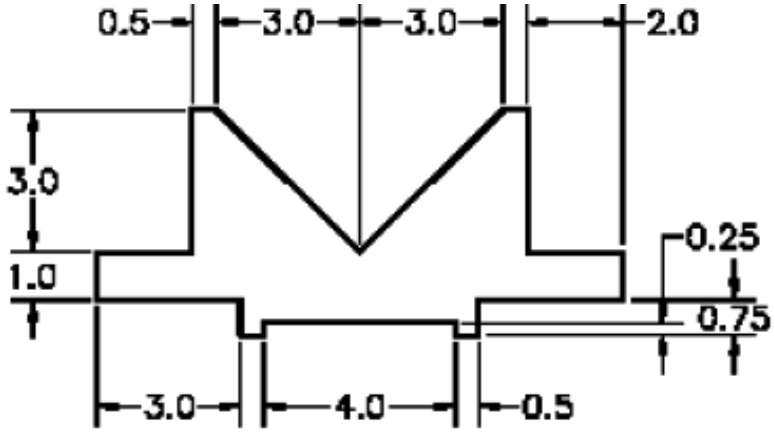
.....

.....

.....

.....

XVII To manufacture 3D components by 3D printer for practice:



XVIII References / Suggestions for Further Reading

- file:///C:/Users/Administrator/Downloads/cura-lulzbot_3.2.32.dmg
- https://www.youtube.com/watch?v=e0rYO5YI7kA
- https://www.youtube.com/watch?v=HVgPM1ojyLw
- https://www.youtube.com/watch?v=89BQz6X3ntc
- https://www.youtube.com/watch?v=LALTD5-TAkU
- https://www.lulzbot.com/learn?type_1=tutorial
- http://download.lulzbot.com/Software/cura-lulzbot/mac/cura-ulzbot_3.2.32.dmg

XIX Assessment Scheme

Performance Indicators		Weightage
Process Related (10 Marks)		**70%**
1	Selecting relevant material, process and set up parameters.	30%
2	Slicing the solid model and transferring the file to the printer.	20%
3	Printing the components.	20%
Product Related (15 Marks)		**30%**
4	Safety unloading the manufactured component from the printer/machine.	10%
5	Answer to sample questions.	10%
6	Submission of digital drawing file/plot in time.	10%
Total (25 Marks)		**100 %**

Names of Student Team Members

1. ……..
2. …….
3. ……

Marks Obtained			Dated signature of Teacher
Process Related(10)	**Product Related(15)**	**Total (25)**	

List Of Laboratory Manuals Developed by MSBTE

First Semester:

1	Fundamentals of ICT	22001
2	English	22101
3	English Work Book	22101
4	Basic Science (Chemistry)	22102
5	Basic Science (Physics)	22102

Second Semester:

1	Bussiness Communication Using Computers	22009
2	Computer Peripherals & Hardware Maintenace	22013
3	Web Page Design with HTML	22014
4	Applied Science (Chemistry)	22202
5	Applied Science (Physics)	22202
6	Applied Machines	22203
7	Basic Surveying	22205
8	Applied Science (Chemistry)	22211
9	Applied Science (Physics)	22211
10	Fundamental of Electrical Engineering	22212
11	Elements of Electronics	22213
12	Elements of Electrical Engineering	22215
13	Basic Electronics	22216
14	'C' programming Language	22218
15	Basic Electronics	22225
16	Programming in "C"	22226
17	Fundamentals of Chemical Engineering	22231

Third Semester:

1	Applied Multimedia Techniques	22024
2	Advanced Serveying	22301
3	Highway Engineering	22302
4	Mechanics of Structures	22303
5	Building Construction	22304
6	Concrete Technology	22305
7	Strength Of Materials	22306
8	Automobile Engines	22308
9	Automobile Transmission System	22309
10	Mechanical Operations	22313
11	Technology Of Inorganic Chemicals	22314
12	Object Oriented Programming Using C++	22316
13	Data Structure Using 'C'	22317
14	Computer Graphics	22318
15	Database Management System	22319
16	Digital Techniques	22320
17	Principles Of Database	22321
18	Digital Techniques & Microprocessor	22323
19	Electrical Circuits	22324
20	Electrical & Electronic Measurment	22325
21	Fundamental Of Power Electronics	22326
22	Electrical Materials & Wiring Practice	22328
23	Applied Electronics	22329
24	Electrical Circuits & Networks	22330
25	Electronic Measurments & Instrumentation	22333
26	Principles Of Electronics Communication	22334
27	Thermal Engineering	22337
28	Engineering Matrology	22342
29	Mechanical Engineering Materials	22343
30	Theory Of Machines	22344

Fourth Semester:

1	Hydraulics	22401
2	Geo Technical Engineering	22404
3	Chemical Process Instrumentation & Control	22407
4	Fluid Flow Operation	22409
5	Technology Of Organic Chemicals	22410
6	Java Programming	22412
7	GUI Application Development Using VB.net	22034
8	Microprocessor	22415
9	Database Managment	22416
10	Electric Motors And Transformers	22418
11	Industrial Measurements	22420
12	Digital Electronics And Microcontroller Applications	22421
13	Linear Integrated Circuits	22423
14	Microcontroller & Applications	22426
15	Basic Power Electronics	22427
16	Digital Communication Systems	22428
17	Mechanical Engineering Measurments	22443
18	Fluid Mechanics and Machinery	22445
19	Fundamentals Of Mechatronics	22048

Fifth Semester:

1	Design of Steel and RCC Structures	22502
2	Public Health Engineering	22504
3	Heat Transfer Operation	22510
4	Environmental Technology	22511
5	Operating Systems	22516
6	Advanced Java Programming	22517
7	Software Testing	22518
8	Control Systems and PLC's	22531
9	Embedded Systems	22532
10	Mobile and Wireless Communication	22533
11	Industrial Machines	22523
12	Switchgear and Protection	22524
13	Energy Conservation and Audit	22525
14	Power Engineering and Refrigeration	22562
15	Solid Modeling and Additive Manufacturing	22053
16	Guidelines & Assessment Manual for Micro Projects & Industrial Training	22057

Sixth Semester:

1	Solid Modeling	17063
2	Highway Engineering	17602
3	Contracts & Accounts	17603
4	Design of R.C.C. Structures	17604
5	Industrial Fluid Power	17608
6	Design of Machine Elements	17610
7	Automotive Electrical and Electronic Systems	17617
8	Vehicle Systems Maintenance	17618
9	Software Testing	17624
10	Advanced Java Programming	17625
11	Mobile Computing	17632
12	System Programing	17634
13	Testing & Maintenance of Electrical Equipments	17637
14	Power Electronics	17638
15	Illumination Engineering	17639
16	Power System Operation & Control	17643
17	Environmental Technology	17646
18	Mass Transfer Operation	17648
19	Advanced Communication System	17656
20	Mobile Communication	17657
21	Embedded System	17658
22	Process Control System	17663
23	Industrial Automation	17664
24	Industrial Drives	17667
25	Video Engineering	17668
26	Optical Fiber & Mobile Communication	17669
27	Therapeutic Equipment	17671
28	Intensive Care Equipment	17672
29	Medical Imaging Equipment	17673

Pharmacy Lab Manual

First Year:

1	Pharmaceutics - I	0805
2	Pharmaceutical Chemistry - I	0806
3	Pharmacognosy	0807
4	Biochemistry and Clinical Pathology	0808
5	Human Anatomy and Physiology	0809

Second Year:

1	Pharmaceutics - II	0811
2	Pharmaceutical Chemistry - II	0812
3	Pharmacology & Toxicology	0813
4	Hospital and Clinical Pharmacy	0816

HEAD OFFICE

Secretary,
Maharashtra State Board of Technical Education
49, Kherwadi, Bandra (East), Mumbai - 400 051
Maharashtra (INDIA)
Tel: (022)26471255 (5 -lines)
Fax: 022 - 26473980
Email: -secretary@msbte.com
Web -www.msbte.org.in

REGIONAL OFFICES:

MUMBAI

Deputy Secretary (T),
Mumbai Sub-region,
2nd Floor, Govt. Polytechnic Building,
49, Kherwadi, Bandra (East)
Mumbai - 400 051
Phone: 022-26473253 / 54
Fax: 022-26478795
Email: rbtemumbai@msbte.com

PUNE

Deputy Secretary (T),
M.S. Board of Technical Education,
Regional Office,
412-E, Bahirat Patil Chowk,
Shivaji Nagar, Pune
Phone: 020-25656994 / 25660319
Fax: 020-25656994
Email: rbtepn@msbte.com

NAGPUR

Deputy Secretary (T),
M.S. Board of Technical Education
Regional Office,
Mangalwari Bazar, Sadar, Nagpur - 440 001
Phone: 0712-2564836 / 2562223
Fax: 0712-2560350
Email: rbteng@msbte.com

AURANGABAD

Deputy Secretary (T),
M.S. Board of Technical Education,
Regional Office,
Osmanpura, Aurangabad -431 001.
Phone: 0240-2334025 / 2331273
Fax: 0240-2349669
Email: rbteau@msbte.com