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 M	Ir. / Ms	
Roll No	of Fifth Semeste	er of Diploma in
		of Institute
(Code)	has completed the term w	vork satisfactorily
in course Solid Model	ing and Additive Manu	facturing(22053)
for the academic year	· 20to 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 outcomebase 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 broadbasedmechanicalengineering 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	CO	CO	CO	CO	CO
511100		a.	b.	c.	d.	e.	f.
1.	Prepare drawing template consisting of name plate boundary lines and projection symbol.	\checkmark	-	-	-	-	-
2.	Draw and print two simple 2D geometries using sketcher commands	\checkmark	-	-	-	-	-
3.	Draw and print two complex 2D geometries using sketcher commands	\checkmark	-	-	-	-	-
4.	Draw and print the given two simple 3-D drawings using 3D modeling commands	\checkmark	\checkmark	-	-	-	-
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)	V	\checkmark	\checkmark	-	-	-
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)	V			-	-	-
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)	V		\checkmark	\checkmark	-	-
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)	V	\checkmark		\checkmark	_	-

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

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

Sr. No	Practical Outcome	Page No.	Date of performance	Date of submi ssion	Assess ment marks(25)	Dated sign. of teacher	Remarks (if any)
1	Prepare drawing template consisting of name plate boundary lines and projection symbol. Draw and print two simple 2D	1					
2	geometries using sketcher commands	23					
3	Draw and print two complex 2D geometries using sketcher commands	40					
4	Draw and print the given two simple 3-D drawings using 3D modeling commands	59					
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)	90					
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)	98					
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)	106					
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)	119					
9	Assemble and print the orthographic views of the assembly developed in PrO 5 to 8 with bill of materials. (Problem 2)	147					

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 ₃ Size.	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 DIMENS	ONS ARE IN MM
NAME-	INSTITU	TENAME
ROLL NO-	DEPARTM	
5EM.& SEC		
	SCREW JACK ASSEMBLY	SHEET NO
SUB.DATE -	ASSEMBLY	

ALL DIMENSIONS ARE IN MM						
NAME-	INSTITUTE NAME					
ROLL NO-	DEPARTM	ENT NAME				
SEM.& SEC						
SUB.DATE-	SCREW JACK	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.

■ 1 2 3 7 • •				Creo Parame	etric 2.0		
File - Home							
New Open Open Last Select Working Erase No	t Model Syst	em Play					
Session Directory Displaye	d Display Colo	rs Trail File		Coloret Manteira Director			
Data	Settings *			Select Working Directory	1		X
		🔹 💌 🔻 🛄 < (D:)	 Solid Model 	ing Manual + WORKING_DIR_manua	il 🔻	* Search	
· · · ·		🖓 Organize 🗸 🏢 V	iews 🗸 👘 To	ols 🗸			₩?
Common Folders		Common Folders	<u>.</u> .				
In Session		Desktop					
3 Desktop		And Documents					
A My Documents		datirsir					
atirsir		Working Directory					
Morking Directory		Tavorites					
で Network Neighborhood							
B) Manikin Library							
Ex Favorites							
and a second sec							
			File name:	. Date modified: 26-Jan-19 11:05:2	25 AM		
			File name:		Туре	Directories	
		Folder Tree			OK	* Can	cel

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.

	Ne	W	Х
Type	Layout Sketch Part Assembly Manufacturing Drawing Format Report Diagram Notebook Markup	Sub-type	
Name Common No	PRACTCAL1	TEMPLETE_FORMAT	
Use de	efault template		
	ОК	Cancel	

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

	N	lew Format		×
Speci Us En	ify Templa se templa npty with npty	te te section		
Orien	ntation			
			2?	
P	ortrait	Landscape	Variable	
Size				-1
Standa	rd Size	С		-
In Inc	ches			
O Mi	llimeters			- 11
Width	22.00			
Height	17.00			
	ОК		Cancel	

- 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.

🖾 Edge 🔻 Offset Edge command from sketching tool bar. Select



2.

Ent Chain Select Ent. Chain from menu manager.

3. Select the A4 size sheet by pressing Ctrl button of key board. 4.



from'select' dialog box. 5. Click OK

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.

∗∰× Height and Width

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.

Height and Width	h X
Row	
Automatic height adjustmer	nt
Height in drawing units	0.312
Height in number of characters	1.0
Column	
Width in drawing units	1.375
Width in number of characters	10.0
Wrap text	
OK Can	Preview

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.

	-		
	1	I	

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.

Rows & Columns	Data	Format 🔻		Note Properties X
NAME-			2	Text Syle Copy from Style name Default Existing text. Character Font The Co Times O Efault Default Bight Default Underine Width factor Default Underine Width factor Default Underine Width factor Default Verical Middle Mircor Angle 000000 Break crosshatching Calor Margin Preview Rest
				OK Cancel

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

NAME-	INSTITU	UTE NAME
ROLL NO-	DEPARTM	IENT NAME
SEM.& SEC	LINES	
SUB.DATE-	&	SHEET NO
GRADE-	LETTERING	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 Styledefinition dialog box will appear on screen.

Mod	lify Line Style		
Copy Fro	m		
Style	none		
Drawing	Select Line		
Attributes			
Line Font	SOLIDFONT		
Width	0.500000		
Color			
Model Ed	ge Options		
Selection	ted edges only		
O All oc	curences in sheet		
Apply	Cancel R	eset	

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		
ROLL NO-	DEPARIMENT NAME		
SEM.& SEC	LINES		
SUB.DATE-	& LETTERING	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



3. Select

L. To createFIRST ANGLE symbol in template:

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

File - Lavout	Table	Annotate	Sketch	Review	Tools	
Remove All Jogs	Remove All Jogs 🖓 Draft Group				J=7	
- 🙀 Remove All Breaks	I≤ [®] Re	late to Object	Custor	m Symbol	Ĩ	
X Delete	@ Un	relate	Symbol From Palette			
Delete 🔻		Group	A Symbo	ol Gallery		
fine from Mer	nu N	Ianage	r.			

4. Specify symbol name as **FIRST_ANGLE** and accept selection.

		-		
Ē	Enter symbol name [QUIT]:			
	FIRST_ANGLE		\checkmark	X

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 Styledefinition dialog box will appear on screen.



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

Modify Line Style					
Copy From	Copy From				
Style	none	•			
Drawing	Select Line				
Attributes					
Line Font	CTRLFONT	•			
Width	SOLIDFONT				
	DOTFONT	Ξ			
Color	CTRLFONT				
	PHANTOMFONT	•			
Model Edg	e Options				
Select	ed edges only				
	curences in sheet				
Apply Cancel Reset					
_		_			

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 Applybutton.



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

NGLE (Active) - Creo Parametric 🛛 🗖 🕮	Menu Manag X
	Define SYMBOL EDIT Attributes Copy Drawing Copy Symbol Parameters Groups Note Rotate Done Quit

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

5. Tick the Free option and click on OK.

	Symbol Definition Attribute	es	×
General Var Text			OK
Allowed Placement Types	Symbol Instance Height	Attributes	Cancel
Free Pick Origin Free Free Fick Origin	Fixed Variable - Drawing Units e - Model Units no leader. Fixed Text Pelated	Fixed Text Angle Allow Elbow Allow geometry to mirror Allow text to mirror	
Normal to Entity Pick Origin	Select Text	Allow text to flip	
Left Leader Pick Origin			
Right Leader Pick Origin			
Radial Leader Pick Circle			
Name: FIRST_ANGLE			Text Symbol >>

New Select Point definition dialog box will open.

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

Se	elect Point X
x y y	·····
Absolute Coor X 0.00	Select a point using abs
	OK Cancel

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

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

SYM_EDIT_FIRST_ANGLE (Active) - Creo Parametric	1 23	Menu Manag X
t Analysis Info Applications Tools Window Help		Define •
x * X & & & # = • • • • • • • • • • • • • • • • • •		SYMBOL EDIT
		Attributes
ž	en e	Copy Drawing
2	ι=π ≪ /4	Copy Symbol
	<u> </u>	Parameters
1	4	Groups
	~ +	Note Rotate
	⊙ •	Done
· · · · · · · · · · · · · · · · · · ·	\mathcal{D}	Quit
	L.	

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

	• O • 1≙ ₽	• 🖆 🌾 •		PRACTCAL1_TEMPLET	TE_FORMAT (Active) - Creo Parametric 2.0	- 9
File + Layout	Table Annotate	Sketch Review T	fools View			•⊖ Q ∾
¦ [™] _X Remove All Jogs - [™] _R Remove All Breaks X Delete	Draft Group Draft Gro	Note Break	Attachment	A		
Delete *	Group	Annotations	Edt *	Format *		
ga 🔥 🖈 Drawin Sheet 1 of PRACT	ITRE	; · ;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;		KAME- RGLI NO- BEM A STO- UII DATTI- JEADE-	BITTUTE SAME DEPARTMENT SAME DEPARTMENT SAME LIDES LETTERED BEET NO- LETTERED	weinstand with GALLEY Define Redefine Define Write Symbol Dir Show Name Doce

- 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.

		Cus	stom Drawing Sy	/mbol	1
General	Grouping	Variable Text			Preview
— Definiti Symbol n	ame 1ST_A	NGLE	¥	Browse	
Placem Type Next lea	ent Free der	A v	Arrowhead Auto	matic v	
Propert Height Proportie	0.40342	6	Origin Default Custom		
Angle Color	0.00000	0 +90			
					OK Cancel

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

				(Cust	tom Drawing	g Sy	/mbol					
General Definiti	Gro	ouping	Varia	able Tex	ct						Preview		
Symbol n	ame	1ST_A	NGLE				Ŧ	Brows	se				
		1ST_AN	IGLE	F				New	·				
Placem	ent			-									
Type Next lea	der			Speci symb	fies ols f	the drawing that are alre	g sy ady	/mbol in / associ	stance ated w	e to rith	insert. Th the file.	ie lis	t displays
													1.4.1
Proper	ies					Origin						·}	•••
Propert Height Proporti	ies (on	0.403426	3			Origin Default Custom	1					<u>}</u>	
Propert Height Proportio Angle	ies - ()n	0.403426 0.000000	5	+90		Origin Default Custom						· <u></u>	
Propert Height Proportio Angle Color	ies – On –	0.403426	3	+90		Origin Default Custom						·F	

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

▼ - → Break	Attachment	Image: Arrow Style Text Line Style Style Hyperlink	Custom Drawing Symbol General Grouping Variable Text Definition Symbol name FRST_ANGLE V Browse	Preview
			Plasmont Type Pree Next leader Answires Propertion Angle Colorr	
				OK Cancel
		NAME DETITUTE NAME SOLL NO. DEFARTMENT NAME SMARTSC. LINGS STEDATE & REMETING. JRADE LETTRIDAS DESCRETES		

4. Move the symbol at required position.

NAME- ROLL NO-	INSTITU' DEPARTMI	TE NAME ENT NAME	
SEM.& SEC	IINES		
SUB.DATE-	&	SHEET NO	
GRADE-	LETTERING	CHECKED BY-	

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

NAME-	INCTIT	ITE MAME
ROLL NO-	DEPARTI	MENT NAME
SEM.& SEC	LINES	⊕
SUB.DATE-	LETTERING	SHEET NO
	LEILERING	

N. To add note without leader:

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.

Note Properties

ALL DIMENSIONS ARE IN MM	\checkmark	X

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

Next
Previous
Pick From List
Cuţ
Copy
Erase
× <u>D</u> elete
Edit Attachment
Move to Sheet
Move Special
P <u>r</u> operties
Add Hyperlink

	Note P	roperties	х
Text Text S	tyle		
Style name	Default		•
Existing text	Select Text		
Character			
Font	Tr CG Times	•	Default
Height	4	Default Slant angle	0.000000
Thickness	1	Default Underlin	e
W Sets the Enter av	character thick value in drawing	g units.	
Horizontal D	Default 🔻	Line spacing 0.500000	🖌 Default
Vertical 1	op v	Mirror	
Angle 0	.000000	Break crosshatchin	g
Color		Margin 0.150000	v
	Preview	Reset	
		ОК	Cancel

 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.

Note Properties	×
Text Text Style	
LINES &	Open
LETTERING	Save
	Text Symbol
	Report Symbol
	Hyperlink

2. Type SCREW JACK ASSEMBY in Text option.

	Note Properties			х	
ſ	Text	Text Style			
	SCRE	WJACK		🕞 Open	ור
	ASSE	MBLY		Save	51
				Editor	

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

	Note Properties
ext Text S	tyle
Copy from	
Style name	Default
Existing text	Select Text
Character	
Font	The CG Times 💌 🗌 Default
Height	4.000000 Default Slant angle 0.000000
Thickness	1.000000 Default Underline
Width factor	Default Kerning
Note/Dimer	sion
Horizontal C	Center Line spacing 0.500000 Defau
Vertical M	Aiddle 🔻 🗌 Mirror
Angle 0	.000000 Break crosshatching
Color	Margin 0.150000 v
	Preview Reset
	OK Canc
	OK Canc

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

XIII Actual Procedure Followed

 	 	••••••
 	 	•••••
 	 	••••••

XIV Precautions Followed

• • • • • • • • • • • • • • • • • • • •	• • • • • • • • • • • • • • • • • • • •	• • • • • • • • • • • • • • • • • • • •	• • • • • • • • • • • • • • • • • • • •	• • • • • • • • • • • • • • • • • • • •
	• • • • • • • • • • • • • • • • • • • •			

XV Observations and Calculations

XVI Results

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XVII Interpretation of Results

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XVIII Course proficiency

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XIX Practical Related Questions

Note: Below given are few sample questions for reference. Teachers <u>must design</u> 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

- o https://www.youtube.com/watch?v=B96jGGkt-zk
- o https://www.youtube.com/watch?v=cWCI8K1ji80
- o https://www.youtube.com/watch?v=zJrea53xtG8

XXI Assessment Scheme

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

Names of Student Team Members

- 1.
- 2.
- 3.

Marks Obtained			Dated signature of Teacher
Process	Product	Total	
Related(10)	Related(15)	(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 of parametric 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.

				S2D0001 (Active) - Creo Parametric 2.0				- 6 %							
File •	Sketch	Analysis	Tools View												
Grid	File System	K Select	Centerline X Point Coordinate System	Construction Mode	✓ Line ▼ □ Rectangle ○ Circle ▼) Arc ▼ O Ellipse O Spline	► Fillet ▼ ▼ Chamfer ▼	D Offset D Thicken	Centerline ▼ X Point , ¹ → Coordinate System	→ Modify ph Mirror r Divide	SHE Delete Segment	∓ ↔ \ + ∕ = + & +	Normal #		
Setup *	Get Data	Operations *	Datum				Sketching				Editing	Constrain '	Dimension *	Inspect *	
80 P	*														

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	Grid	Define the grid settings.				

2 **Operations:**

	Operations Tool Bar					
Tool Name	Symbols	Use				
	4	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	Select one by one entity. For next entity selection press Ctrl and then click the entities.				
	t⊋ Chain	Select the chain of entities.				

All Geometry	Select all Geometry entities.
R All Ctrl+Alt+A	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	х	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.			
Constructi on Mode	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	α	Sketch a rectangle by selecting two opposite corners of the rectangle using the LMB.			
Slanted Rectangle	1	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			
Parallelogr am		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	O	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	0	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	0	Draw a circle through three points which are selected by pressing the LMB three times.			
3-Tangent Circle	Ø	Draw a circle tangent to three features which are selected by pressing the LMB three times.			
3- Point/Tang ent End Arc	2	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	ĥ	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	0	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	0	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	ζ	Draw a free-hand spline curve by selecting spline points using the LMB.
Circular Fillet	٢	Draw a circular fillet or round tangent to two features.
Circular Trim Fillet	r	Draw a circular fillet or round tangent to two features.
Elliptical Fillet	٢	Draw an elliptical fillet or round tangent to two features at the two points selected on the features using the LMB.
Elliptical Trim Fillet	ſ	Draw an elliptical fillet or round tangent to two features at the two points selected on the features using the LMB.
Chamfer Fillet	٢	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	\mathbf{k}	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	Q	Create duplicate geometry an offset distance from the selected geometry.
Thicken	œ	Create entities by offsetting an edge or a sketched entity ontwo sides.
Text	A	Create alpha characters and symbols on the sketch.
Text Along A Curve	A	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	\bigcirc	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	r.	Draw an infinitely long sketcher centerline tangent to two circles or arcs using the LMB.
Construction Point	X	Create a construction point using the LMB. This point is known only inside sketcher and is not visible outside sketcher.
Construction Coordinate System	4. ¹ 2	Create a construction coordinate system at the specified point by pressing the LMB.

	Editing Tool Bar					
Tool Name	Symbol	Use				
Modify	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	0:0	Mirror a selection of features about a specified centerline, thus there must be a centerline present in the sketch.				
Divide Entity at This Point	r A	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	4	Force two selected lines to be perpendicular to each other. After selecting the tool, select the two lines using the LMB.				
Tangent Constraint	X	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	1	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	ţ	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		Force two or more features to be equal size. After the tool is
Constraint		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	//	Force two lines to be parallel. After the tool is selected, select
Constraint		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	ţ	Add a strong dimension to the existing sketch.				
Perimeter	□II	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	↓ → REF	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	\checkmark	Accept the changes made in the sketcher and exit sketcher.								
Cancel	x	Cancel the changes made in sketcher and exit sketcher.								

Sketcher Graphics Tool Bar									
Refit	<u>a</u>	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	Q	Zoom out to gain a wider perspective on the geometry.
Repaint		Redraw the current view (refresh).
Sketcher display filter		Define sketcher display.
Select All	₩ ¥ (SelectAll)	To display all sketcher display.
Disp.Dims.	Se Hond Disp Dims	To display dimensions.
Disp.Constr.	⊠ [⊥] ∥ Disp Constr	To display constraints.
Disp.Grid.	Disp Grid	To display grid.
Disp.Verts	Disp Verts	To display vertices.

VIII. Experimental setup

IX. Resources Required

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

🗏 🗋 🖒 ど 🖾 🗠 ។ 🗠 ។ 🎁 ។ 🖆 🔻 🖆 🔹 PRACTICAL2_EXERCISE1 (Active) - Creo Parametric 2.0													
File •	Sketch	Analysis	Tools View										
Grid	File System	X Na Select	Centerline X Point Coordinate System	Construction Mode	∽ Line ▼ □ Rectangle ▼ ⓒ Circle ▼	⊖ Arc ▼ Q Ellipse ∿ Spline	Fillet • Chamfer	Offset Offset De Thicken Offset De Palette	Centerline ▼ X Point , ¹ → Coordinate System	Hodify % Delete Segment Mirror -⊢ Corner r≤ Divide ③ Rotate Resize	± ↔ // + × =	I↔I 📲 Normal 🛱	80
Setup *	Get Data	Operations \bullet	Datum			:	Sketching			Editing	Constrain 🔻	Dimension *	Inspect *
Commo Co	m Folders ession dop Documents sir king Director vork Neighbo kin Library orites	y rhood								2 0, 111,			

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.

	Creo Parametric Options
Favorites	View and manage your favorite options.
System Colors Model Display	Display your favorite options here. In the Configuration Eddor, right click an option and choose Add to Eavorites from the shortcut menu.
Entity Display Selection	
Sketcher Assembly	
Sheetmetal	
Customize Ribbon Quick Access Toolbar Window Settings	
Licensing Configuration Editor	

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

	Creo Parametric Options	
Favorites	Set options for objects display, grid, style, and constraints.	
System Colors Model Display Entity Display Selection Sketcher Assembly	✓ Collinear ✓ Symmetric ✓ Midpoint ✓ Tangent	\$ * *
Data Exchange	Dimension and solver accuracy	
Sheetmetal	Number of decimal places for dimension:	2 *
Customize Ribbon	Relative accuracy for sketcher solving:	1.000000
Quick Access Toolbar Window Settings	Dimension behavior while dragging the section Lock modified dimensions	
Licensing	Lock user defined dimension	
Configuration Editor	Sketcher grid Show the grid Show the grid Grid snaht on ongooo Grid spacing type: Dynamic Spacing along X-axis: 1.000000 Spacing along Y-axis: 1.000000]

- 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.

m	Construction Mode	⊷ Line ▼ □ Rectangle ▼ ⊙ Circle ▼	O Arc ▼ O Ellipse ▼ O Spline	Fillet • Chamfer	Offset Offset Definition Definition	¦ Cei X Poi ,≯ Coi	nterline nt ordinat	• Syste	m	デ™ (約 Min r[#] Div	dify \$ ror - ide \$	⊭ Delete Segment - Corner) Rotate Resize	+ ≫ + + ∖ =	I⇔I 📲 Normal 🛄	
			S	iketching							E	diting	Constrain 🔻	Dimension *	Inspect 7
							1 0	Q		0.	12,				

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.

G	Grid Settings – 🗖 🗙									
Grid Type										
Cartesia	n									
O Polar										
Grid Spacing										
 Dynami 	c									
 Static 										
X Spacing	5.000000									
V Consiste	C 0000	_								
r Spacing	5.0000									
Grid Orientatio	n									
Osisis										
Origin	۲.									
Angle	0.000000									
	ок с	ancel								

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.

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									-
 	 	 		 	 			 	1.00
									_

- 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. ClickSave button which saves sketch by PACTICAL2_EXERCISE1with .sec file ext.
- G. Print the created sketch in MS Word by Print Screen Shot.





- 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

Circle Circle command from sketching tool bar.

using



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

30.00

8.00



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 .	Name of	Bi	road Specifications	Quantity	Remarks
No	Resource	Make	Details	Quantity	(If any)
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 <u>must design</u> 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=qFIev5cR1W4
- https://www.youtube.com/watch?v=lpH4ZUUD9N0
- https://www.youtube.com/watch?v=sVWsUS_7V6s&list=PLrOFa8sDv6jfVMc cV28fssFut0EG0NNb6

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	2004
	model tree.	2070
	Product Related (15 Marks)	60 %
3	Generation and printing of drawing views, tables,	2004
	etc. and their arrangement on different sheet size.	2070
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.

Ma	arks Obtained	Dated signature of Teacher	
Process	Product	Total	
Related(10)	Related(15)	(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
		(i5 or higher), RAM minimum 4 GB;	As per
1	Hardware: Personal	A3 / A4 size printer / plotter. Display-	batch
	computer.	wide Screen preferably.	size
		Windows XP/Windows 7/ Windows	As per
2	Operating system	8/Windows 10 or higher.	batch
		_	size
		Any parametric solid modeling	As per
3	Software	software.	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 explained in 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.

	Creo Parametric Options		х	ſ
Favorites Environment System Colors Model Display	Set options for objects display, grid, style, and constraints. Show weak dimensions Show entity D number on help text			
Entity Deplay Selection	Sketcher constraints assumptions		10	ŀ
Sketcher	Line up horizontally		11	l
Assembly Data Exchange	Line up vertically	1		2
Sheetmetal	Paralel	H		l
Customize Ribbon	Perpendicular Equal length	1 7		
Quick Access 100bar Window Settings	Equal radi	Υ.		
Licensing Configuration Editor	✓ Colinear ✓ Symmetric	*	1	
	Midpoint	`		l
	Tangent	9		ł
	Dimension and solver accuracy		08	l
	Number of decimal places for dimension:	2 4		l
	Relative accuracy for sketcher solving:	1.000000		l
	Dimension behavior while dragging the section		43	l
	Lock modified dimensions			
	Lock user defined dimension			
	Sketcher grid		410	ł
	Show the grid			l
	Snap to grid			
	Grid angle: 0.000000			l
	Grid type: Cartesian +			
			۳	
		Sketcher settings: Restore Defaults		
Export Configurations		OK Cance	AC Go	

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



- Centerline Centerlineoption to draw vertical and horizontal centerlines. 4 Use
- 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 51° using same procedure.
- 7 Use **Point** Point option to locate the center point of the circles.
 - O Circle ▼ ∿ Spline
- 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.





LineTangent

10 To draw tangent lines to the circles. Use 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 🔻

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 \emptyset 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.



bar. Click on the **Delete Segment** Delete Segment button and then choose unwanted lines one by one as shown in Figure. Draw $\phi 3 \ge 7$ on $\phi 30$ 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.





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

	Π		Creo Parametric Options
		Favorites	View and manage your favorite options.
	1	System Colors	Disnlav vour favorije options here
		Model Display Entity Display	In the Configuration Editor, right click an option and choose Add to Favorites from the shortcut menu.
		Selection	
		Assembly	
	1	Data Exchange Sheetmetal	
	I	Customize Ribbon	
Send +	I	Quick Access Toolbar	
Manage Session 🕨	I	Window Settings	
Help 🕨		Licensing Configuration Editor	
Options			

- 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.

Centerline 🔻

- 5. Use **Centerline**optionsto 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 40° 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.





14. Select **Arc racetrack** from the **Shapes** option. Drag it in drawing area as shown in Figure.



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



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

	ê 🛛	υ·Ο·	8 · 11 ·				PR	ACTICAL3_E	XERCISE2 (Active) - C	reo Parametric 2.0			
File +	Sketch	Analysis	Tools View										
Grid	File System	X N Select	Centerline X Point Coordinate System	Construction Mode	✓ Line ▼ ☐ Rectangle ⓒ Circle ▼) Arc ▼ Q Ellipse Q Spline 	Chamfer	D Offset D Thicken	Centerline ▼ X Point	Hodify St Delete Segment Mirror - ← Corner r≤ Divide ③ Rotate Resize	+ ✓ = + ≫ ₩	I⇔I Normal tt	
Setup *	Get Data	Operations *	Datum			1	Sketching			Editing	Constrain 🔻	Dimension •	Inspect
Commo	Folders									75.00			
in Se Desk My D	ssion op ocuments				Ű			н		-025 - 140 -025 -			

- 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 40° 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.
 - e
- 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.
- D Offset
- 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 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.

O Circle	•	∿ Sp
🖸 Cent	ter ar	nd Point

- 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 Ø1.625 and then select circle R 0. 625.
- 38. To tangent the circleR 0.625 and outer racetrack of R 0.875. As the 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 **under** 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.







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

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

3.	
II.	Actual Procedure Followed
TV	Procentions Followed
. 7	
v .	Course proficiency
VI.	Practical Related Questions
	Note: Below given are few sample questions for reference. Teachers <u>must design</u> 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

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

Names of Student Team Members

- 1.
- 2.
- 3.
- 4.

Ma	arks Obtained	Dated signature of Teacher	
Process	Product	Total	
Related(10)	Related(15)	(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
		(i5 or higher), RAM minimum 4 GB;	As per
1.	Hardware: Personal	A3 / A4 size printer / plotter. Display-	batch
	computer.	wide Screen preferably.	size
		Windows XP/Windows 7/ Windows	As per
2.	Operating system	8/Windows 10 or higher.	batch
		_	size
		Any parametric solid modeling	As per
3.	Software	software.	batch
			size
4.	Plotter	Plotter A_2 OR A_3 Size.	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** 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.

	х							
Type Image: State Sta	yout etch t sembly nufacturing swing mat port gram tebook rkup	Sub-type Solid Sheetmetal Bulk Harness						
Name Common Name	PRACTICAL4_E	KERCISE1						
Use default template								
OK		Cancel						

- 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.

New File Options	×
Template	
mmns_part_solid	Browse
Empty	
inlbs_harn_part	
inlbs_part_ecad	=
inlbs_part_solid	
mmns_harn_part	
mmns_part_solid	*
MODELED_BY DESCRPTION	
Copy associated drawings	
OK	ncel

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.





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.







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.





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.



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.





- 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.

47

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:



- 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

button of part modeling tool bar.





- 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:



- 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



- 0.1
- 5 Select 🖵 button of part modeling tool bar.



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.





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

New X								
Type Train Layout Sketch Sketch Image: Sketch Image: Sketch <td>Sub-type Sold Sheetmetal Bulk Hamess</td>	Sub-type Sold Sheetmetal Bulk Hamess							
Name PRACTICAL4_	Name PRACTICAL4_EXERCISE2 Common Name							
OK	Cancel							

- 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.

File - Mode	Analysis	Annotate Rende	r Tools	View Flexibl	le Modeling	Applications	Sketch										00.0
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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.





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.



- 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.





- 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 Grient 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.





- 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.
	Maddel Iree 11 PrACTICAL EXERCISE 2 PRT D RICHT D RICHT D RICHT D RICHT D RICHT Stethol 1 Stethol 2 Stethol 2 D Richt 2 Stethol 3 Stethol 3 Stethol 3 Stethol 3 Stethol 3 Stethol 3 Stethol 3 Stethol 4 D RICHT 4 Stethol 4 D RICHT 4 Stethol 4 D RICHT 4 D
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.



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	File name: practical4_ex2_template.frm Type All Files (*)	-							
Folder Tree	Open 👻 Cancel								

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



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

CHECKED

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

	Drawing View X Categories View type View type View name View Sates View view_1 Socie Type Socie Commercial View Sates Select orientation View orientation Select orientation method Origin Asignment Model view names Default orientation Default orientation Timefric Default orientation Timefric BACK BACK BOTTOM View 10:00 FRONT View 10:00
SEM. & SEC SUB DATE- JRADE-	PRACICALA EZE ECISE PART PRINT HECKED BY-

1. SelectView Displayoption. Choose Hidden option.



H Dimension

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. <u>Click on</u> Annotate option from drawing tool bar.

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

XIII Actual Procedure Followed

XIV Precautions Followed

Choose

XV Course proficiency

XVI Practical Related Questions

Note: Below given are few sample questions for reference. Teachers <u>must design</u> 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]

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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

	Weightage		
	Process Related (10 Marks)		
1	Use of proper commands.	20%	
2	Completion of drawing with minimum size of	20%	
	model tree.		
	60 %		
3	3 Generation and printing of drawing views, tables,		
	etc. and their arrangement on different sheet size.		
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	Product	Total	
Related(10)	Related(15)	(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 .	Name of Resource	Suggested Broad Specification	Quantity	
No.				
		(i5 or higher), RAM minimum 4 GB;	As per	
1.	Hardware: Personal	A3 / A4 size printer / plotter. Display-	batch size	
	computer.	wide Screen preferably.		
		Windows XP/Windows 7/ Windows	As per	
2.	Operating system	8/Windows 10 or higher.	batch size	
		Any parametric solid modeling	As per	
3.	Software	software.	batch size	
4.	Plotter	Plotter A_2 OR A_3 Size.	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.



🔨 Line 🔻

Profile of Screw Jack Body without Fillet.

3. Create and Edit the dimensions by using $|\leftrightarrow|$

Normal from Dimension group.

+	Normal ↓	Feature Requirements	✓ З ок са			
ľ	Dimension *	Inspect *	Close			
	I↔I Normal Create dimensions that reference at least one sketched entity.					

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.

□ □ ↓ [InternalCL] 160.0 ▼ 1/2	
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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.

龙女	1.50	•		II	O KA	ා රං	<	X
		Drag radius	handle or enter value for constan	nt rad	ius round		_	,

Radius and Accept dashboard.



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

XI Resources Used

S.	Name of	ŀ	Broad Specifications	Quantity	Remarks
No.	Resource	Make	Details	Quantity	(If any)
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 <u>must design</u> 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]

	••••••	••••••		••••••		•••••
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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)			
1	Use of proper commands.	20%		
2	Completion of drawing with minimum size of	20%		
	model tree.			
	Product Related (15 Marks)			
3	Generation and printing of drawing views, tables,	20%		
	etc. and their arrangement on different sheet size.			
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.

M	arks 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
		(i5 or higher), RAM minimum 4 GB;	As per
1.	Hardware: Personal	A3 / A4 size printer / plotter. Display-	batch size
	computer.	wide Screen preferably.	
		Windows XP/Windows 7/ Windows	As per
2.	Operating system	8/Windows 10 or higher.	batch size
	Any parametric solid modeling		As per
3.	Software	software.	batch size
4.	Plotter	Plotter A ₂ OR A ₃ Size.	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.

Centerline

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 🔻



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.





(b)

Round command dashboard.







(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.	Name of	Broad Specifications		Quantity	Remarks
No.	Resource	Make Details			(If any)
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)				
1	Use of proper commands.	20%			
2	Completion of drawing with minimum size of	20%			
	model tree.				
	60%				
3	Generation and printing of drawing views, tables,	20%			
	etc. and their arrangement on different sheet size.				
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.

Ma	arks 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
		(i5 or higher), RAM minimum 4 GB; A3	As per
1.	Hardware: Personal	/ A4 size printer / plotter. Display-wide	batch size
	computer.	Screen preferably.	
		Windows XP/Windows 7/ Windows	As per
2.	Operating system	8/Windows 10 or higher.	batch size
		Any parametric solid modeling software.	As per
3.	Software		batch size
4.	Plotter	Plotter A ₂ OR A ₃ Size.	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.

Centerline

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.

Feature equirements	ок	X Cancel		
Inspect 🔻	Cl	ose		
	Save t)K he secti	on and exit.	

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.

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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.

viii)



(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.



Accept Dashboard.

feature tool and then closes the tool 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.
 - 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.

Extrude	c∲ T ♪ . Sveep	•	Pattern • Control Con	l Bi
onup		tt Cr	Helical Sweep reate a helical sweep).





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.	Name of	Bro	Broad Specifications		Remarks
No.	Resource	Make	Details	Quantity	(If any)
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 <u>must design</u> 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)				
1	Use of proper commands.	20%			
2	Completion of drawing with minimum size of	20%			
	model tree.	2070			
	Product Related (15 Marks)				
3	Generation and printing of drawing views, tables,	20%			
	etc. and their arrangement on different sheet size.	2070			
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.

Ma	arks 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 ₃ Size.	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.

Centerline

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.

∽ Line ▼

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.

Feature equirements	ок	X Cancel		
Inspect •	Close			
	✓ OK Save the section and exit.			

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.

	් Revolve)ن Hole	🔊 Draft	
	Sweep 🔻	🖓 Round 🔻	🔲 Shell	
Extrude	🖉 Swept Blend	🖒 Chamfer 🔻	👔 Rib 🦷	
Shapes 🔻		Engineering 🔻		
	Revolve Revolve a sketched section around a centerline. Create a revolved feature as a solid or surface, and add or remove material.			

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.

viii)

- 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.



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.

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



Construction of centerline

🔨 Line 🔻

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.

∧ Line 🔻



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.







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.



(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.

button.

- 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

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



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.

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.

🔨 Line 🤻



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



Round command dashboard.

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



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





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

XVII. Actual Procedure Followed

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XVIII. Precautions Followed

XIX. Practical Related Questions

Note: Below given are few sample questions for reference. Teachers <u>must design</u> 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	2004
	model tree.	2070
	Product Related (15 Marks)	60%
3	Generation and printing of drawing views, tables,	2004
	etc. and their arrangement on different sheet size.	2070
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.

M	arks Obtained	Dated signature of Teacher	
Process	Product	Total	
Related(10)	Related(15)	(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 ₃ Size.	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						
	Layout Sketch Part Assembly Manufacturing Drawing Format Report Diagram Notebook Markup	Sub-type © Design Interchange Verify Process Plan N C Model Mold Layout Ext. Simp.Rep. Configurable Module Configurable Product				
Name Common Na	Assembly_of	_Screw_Jack				
	ОК	Cancel				

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	×
Template	
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inlbs_asm_design	
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mmks_asm_design	
mmns_asm_design	
mmns_flat_harness	•
Parameters	
MODELED_BY	
DESCRIPTION	
Copy associated drawings	
OK	Cancel

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.

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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:

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

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🖌 Automatic	
Distance	rties
Angle Offset	
Parallel	< 2 0,0,0,10,74,12,2
Coincident	
Normal	
Coplanar	
- Centered	
🖓 Tangent	
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旦. Default	
Asser	nble component at default location.

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.**

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Placement	Move	Options	Flexibility	Properties				<u>`</u>		

Default and Fully Constrained dashboard.

- 3. Complete the placement of the part:
 - In the Assembly dashboard, click Complete Component
 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.

	Open		х
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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).

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 Placement Move Options	Flexibility Properties			

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.

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Placement Move	Options	Flexibility Properties				

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 savedScrew Spindle model and select it. •
- Click Open to assemble this component. •

		Open		x
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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.

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Placement Move Options	Flexibility Properties			

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.	Name of	Broad Specifications		Quantity	Remarks
No.	Resource	Make	Details		(If any)
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 <u>must design</u> 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								
	Process Related (10 Marks)								
1	Use of proper commands.	20%							
2	Completion of drawing with minimum size of	20%							
	model tree.								
	Product Related (15 Marks)	60 %							
3	Generation and printing of drawing views, tables,	20%							
	etc. and their arrangement on different sheet size.								
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.

Ma	arks 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.

e l	User Defined 🔻	Z Automatic -	0.00	ĸ	STATUS : No Constraints	II 🗸 X
	Placement Move Options	Flexibility Properties				

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' **Lool** 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.

	Open	х								
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Common Folders	<pre>images images imag</pre>									
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Calling of Next object i.e. Cup.

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



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:

•

- 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.

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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.

Open X												
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Folder Tree			Open Cancel									

Calling of Next object i.e. Screw.

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



User Defined	🖌 Automatic 🔻	0.00 💌 🏂	STATUS : No Constraints	II 🔽 X
Placement Move Options	Flexibility Properties			

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

XIX Actual Procedure Followed

•••••		•••••	•••••	•••••		•••••	•••••	•••••		••••	•••••	•••••	•••••	•••••	•••••	•••••	•••••	•••••	•••••	•••••	••••	•••••	•••••
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XX Precautions Followed

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XXI Course Proficiency

XXII Practical Related Questions

Note: Below given are few sample questions for reference. Teachers <u>must design</u> 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	20%
	model tree.	20%
	Product Related (15 Marks)	60%
3	Generation and printing of drawing views, tables,	20%
	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 ₃ Size.	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				
	Layout Sketch Part Assembly Manufacturing Drawing Format Report Diagram Notebook Markup	Sub-type			
Name Common Na	Assembly-of-	screw-jack			
	ок	Cancel			

New dialog box.

- In the New dialog box, click Drawing for the Type and type in Assembly-ofscrew-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.

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Drawing Environment with Template.

• From LAYOUT Tab select GENERAL from model views.

Drawing Models	General	Projection	⊡⇔ Re ⊜∘ Co ⊡ [®] Dr
		Model Views	·
	Create	eneral a general viev	v
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General view.

Select Combined State	×
Combined state names	
No Combined State	
DEFAULT ALL	
Do not prompt for Combined St	tate
OK Car	ncel

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.

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NAME-	Categories View type View name Scale Sca	
SEM & SEC. SUD. DATE- 3RADE-	Close Apply SC EE W JACK ASSEMBLY Checked BY- Close the dial	og.

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.

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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.

	Drawing View	×
	Categories View display options View Type Visible Area Scale Scale Scale Scale View States View States View States View States Origin Alignment Skeleton model display Hidde line removal for quits Skeleton model display Hide Skeleton model display	
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Application of hidden display style.

• Repeat the above three steps twice to convert the top view and side view as Display style Hidden, simultaneously.

NAME-	ALL DIMENSI	IONS ARE IN MM
SEM.& SEC SUB.DATE- GRADE-	SC REW JACK ASSEMBLY	SHEET NO CHECKED BY-

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.

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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.

	Show Model Annotations				
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¥_ =-					
OK Cancel Apply					

(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



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.

Family Table d=	Bill of Materials	Reference Viewer	
•	Invest	tigate 🔻	
	BOM	×	:
Sele S S S S S S S S S S S S S	ct Model op Level Subassembly ASSEMB ude Skeletons Inplaced Designated C nactive Desi	y LY_OF_SCF Dbjects ign Solutions Cancel	

Bill of materials dialog box.

Bom Report : ASSEMBLY_OF_SCREW_JACK										
Assemb	Assembly ASSEMBLY_OF_SCREW_JACK contains:									
Quantity		B er	Туре	film-	Name	ja-	Actions			
	1		Part		BODY		*		8	
	1		Part		NUT		<u>A</u> *	⊒ª	-	
	1		Part		SCREW11		4+		8	
	1		Part		TOMMY		<u>_</u> +	⊒∎	-	
	1		Part		CUP		*	⊒∎	-	
	1		Part		WASHER		4+	₽	-	
	1		Part		SCREW2		<u>*</u> *		8	
Summa	ry of part	s for ass	embly ASSE	MBLY_OF_S	CREW_JACK:					
Quantity		jin-	Туре	file-	Name	ju-	Actions			
	1		Part		BODY		4+	⊒∎	-	
	1		Part		NUT		4+	₽	8	
	1		Part		SCREW11		*	₽	8	
	1		Part		TOMMY		4+		8	
	1		Part		CUP		*		8	
	1		Part		WASHER		4+	⊒₿	-	
	1		Part		SCREW2		<u>*</u> *	⊒₿	8	

Bill of materials.

XI Resources Used

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

XII Actual Procedure Followed

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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]

_ . _

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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)					
1	Use of proper commands.	20%				
2	Completion of drawing with minimum size of	2004				
	model tree.	2070				
	Product Related (15 Marks)	60 %				
3	Generation and printing of drawing views, tables,	2004				
	etc. and their arrangement on different sheet size.	2070				
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.

M	arks Obtained	Dated signature of Teacher	
Process	Product	Total	
Related(10)	Related(15)	(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.

S. No.	Name of Resource	Suggested Broad Specification	Quantity
1		(i5 or higher), RAM minimum 4 GB;	As per
1.	Hardware: Personal	A3 / A4 size printer / plotter. Display-	batch
	computer.	wide Screen preferably.	size
		Windows XP/Windows 7/ Windows	As per
2.	Operating system	8/Windows 10 or higher.	batch
			size
		Any parametric solid modeling	As per
3.	Software	software.	batch
			size
4.	Plotter	Plotter A_2 OR A_3 Size.	1

VIII Resources Required

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 X					
★ ★ ▼ → ganesh-vaibhavi > (D:) > 11_Solid Modeling Manual_I Scheme > ▼ € € €					
D Organize v 💷 Views v 👘 Tools v					
Common Folders Im In Session Im Semanal Im Session Im Semanal Im Session Im S	Drawing Views Final_Models_N_Assembly Jinages Spindle_Screw.prt assm0001.asm assembly-of-scre-jack.asm body.drw body.drw body.trt body.trt body.trt drw0002.drw nut.prt practcal1_templete_format1.frm practcal1_templete_format1.frm	<pre>Int0001 prt prt0003.prt screw1.prt screw2.prt i tormy.prt i tormy.prt</pre>			
	File name: body11.prt Date modif	ied: 15-Feb-19 05:27:49 PM		Preview 🔺	
	File name: body11.prt	Type Cr	eo Files (.prt, .asm, . 🔻 Sub-ty	pe v	
▶ Folder Tree		Open	Open Subset	Cancel	

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 X
Default Model
assembly-of-scre-jack.asm Browse
Specify Template
O Use template
 Empty with format
○ Empty
Format
_templete_format11.frm v Browse
OK Cancel

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	х				
Combined state names					
No Combined State					
DEFAULT ALL					
Do not prompt for Combined Stat	е				
OK Canc	el				

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.

NAM ROLI SUB	Drawing View X Categories View type Visuble Area Scale Scale Type General View states View Display View orientation Origin Geometry references Alignment RONT Model view names Default orientation Back Default orientation Back BortroM EFT View
JRA	OK Close Apply

Drawing view.

- Select generated Front view, then click on Projection from Layout tab.
- To get Top view click below the Front view.

	Projection Bre Revolved	0 🖾	B
	ູ 🛱 Detailed 🛛 🖓 Copy and Align	[] 🛛	
General	Auxiliary 🖉 Drawing View	jo 🕀	
_	Model Views 🔻	Edit 🔻	Display 🔻
	Projection Create a projection view.	Q 🛛	0 , c ^B ,
	GYS.DEF		

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.

	Drawing View X
SCALE 0.014	Categories Scale and perspective options View Type Visible Area Scale Sections View States View Display Origin Alignment OK Close Apply

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.

t	Show Model Annotations	H→ Dimension ▼ = 12 Ordinate Dimension ▼ H→ Reference Dimension ▼
	हिल्ल Show I Show anno	Model Annotations tations from the model.

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.

to show all dimensions on the selected view. Click

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.

• We can place the dimensions by selecting Dimension – New References tool and picking individual entities.

^{▶—} Dimension ▼	A= Note
= ⁰ ₁₂ Ordinate Dimension ▼	³² ∕ Surface Finis
K→ Reference Dimension ▼	🐼 Symbol 🔻
H-H Dimension - New Re	eferences
Create dimensions using references.) new

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.

change
change

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.**

					/	
	These options control dimension tolerances					
	blank_zero_tolerance	no *	no	۲	Determines whether to blr	ank (not show) a plus or
	dim_tol_lead_trail_zeros	same_as_lead_trr	ail	۲	Controls display of leading	g and trailing zeros in dim
	dim_tol_trail_zero_max_places	same_as_dim_tol	6	۲	Sets the maximum number	r of decimal places trailin
	display_tol_by_1000	no *	no	۲	For non-angular dimensior	ns, tolerances will be dis
	symmetric_tol_display_standard	std_asme *	std_asme	۲	Controls how symmetric tr	olerances are displayed
	tol_display	yes	no	۲	Controls display of dimens	sion tolerances. You can
	tol_text_height_factor	STANDARD *	standard	۲	Sets default ratio betweer	n the tolerance text heigh
	tol_text_width_factor	STANDARD *	standard	٠	Sets default proportion be	stween the tolerance tex
_		(a)–Tolera	ance Displa	ay Opti	on.	
Option:		Ve	alue:			
tol_display	У	У	es			Add / Change
O Find		n	* 0,			Select or enter a value
~ · · · · ·		V	es			

(b)–Tolerance Display Option.

- To change the display mode of tolerance, double click on a dimension, following dialog box will be seen.
- Q 0 Dimension Propertie Properties Display Text Style 10.00+:01 ----Name ad56 Value and Displa Decimal Place Decimal O Fractiona 40 00 01 Nominal Value 25.00 Angular dimension units 185 00 Override Value (As Is) Tolerance Value Only R8.00^{+.01} 20.00* 8 20.00+.01 Ŧ 70.00 + 04 R25.00 Decimal Places +0.03 Decimal Places Default 2 ALL DIMENSIONS -0.03 NAME INSTITUTE DEPARTMEN OLL NO 42.22*.01 Default 1 EM.& SEC SCREW JACK SUB. DATE Move Text... Move. Edit Attach.. Text Symbol. Restore Values

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 X						
$-\Box$	Model Refs Datum Refs Tol Value Symbols Additional Text	ок				
$\cap \mathcal{M}$	Tolerance Value	Cancel				
	Verall Tolerance 0.001	New Gtol				
	Name gp1					
	Per Unit Tolerance	Copy From				
// 🕀	Value/Unit 0.001					
	Unit Length 0.001					
	Material Condition RFS(no symbol)	▼ Move				
STATUS: warning, this type of gtol should have at least one reference datum.						

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 entity of 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 entity of 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.

	Geometric Tolerance	x
	Model Refs Datum Refs Tol Value Symbols Additional Text	OK Cancel
	Select Model	New Gtol
20.00 ⁺ 01 P D D D D D D D D	Reference: Selected Placement: Placed	Copy From
SCALE 0 DIA'	Select Entity Place Gtol	Move

Application of Geometric Tolerance.

- 7. Surface Finish Symbol :
 - Invoke Annotate>Surface finish^L menu mode.



from ribbon

 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.

Enter value for roughness_height			-
34	\checkmark	X	_



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.

-A-

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.



parallel button.

b) Select Datum Ref. tab option and select Primary A.

		G	eometric To	lerance		<u>//A</u>	×
-7	Model Refs	Datum Refs	Tol Value	Symbols	Additional Text		OK
	Datum Refere	nces				Unordered	Cancel
	Primary S	econdary Te	rtiary				New Gtol
$\cap \Box$	Basic	A		- 1	RFS(no symbol)	•	NOW OLDI
$\leq \perp$	Compound	none			RFS(no symbol)	T	Copy From
// 🕀		A					
$\tilde{\square}$							
-							Move
<u> </u>							
STATUS: incom	plete, please			ce entit	ty in "Model Refs".		
	. 0.0						
195 00	+.02 -						

(a)– Geometric Tolerance.

c) Select **Tol Value** option and enter the tolerance value = 0.003

	Geometric Tolerance X							
	Model Refs Datum Refs Tol	Model Refs Datum Refs Tol Value Symbols Additional Text						
	Tolerance Value		Cancel					
	Overall Tolerance	0.003	New Gtol					
	Name	9P Input the overall tolerance value						
	Per Unit Tolerance		Copy From					
// 0	Value/Unit	0.001						
a –	Unit Length	0.001						
	Material Condition	RFS(no symbol)	Move					
1 L								
STATUS: incomplete, please choose model and/or reference entity in "Model Refs".								

(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.

	Geometric Tolerance X							
	77	Model Ref	s Datum Refs	Tol Value	Symbols	Additional Text		OK
					-			Cancel
	19	Model BC	DY11.PRT				•	
\cap	\Box	Se	elect Model					New Gtol
			,					[
		Reference:	To Be Selected		Placemen	t: To Be Placed		Copy From
	⊕	Type	Axis		Type	Dimension	-	
0	-	1900	Axis		1900			
	11		Surface			Place Gtol		Move
	<u> </u>		Datum		1			
STATUS	3: incom	plete, pleas	Entity Select the type of reference entity lodel Refs".					
STATUS: incomplete, pleas Entry Select the type of reference entity Iddel Refs".								

(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.**

Occuratio	14 5165m // <u>A</u>	
Tolerance Symmetry L	Geometric Tolerance	×
Annotations *	Model Refs Datum Refs Tol Value Symbols Additional Text	ОК
		Cancel
	Model BODY11.PRT	
°a− -	Select Model	New Gtol
$3 \qquad 34 \qquad $	Reference: Selected Placement: To Be Placed	Copy From
	Type Surface Type Dimension	
	Dimension	Movo
	Dimension Elbow	MOVE
	CTATUS: incomplete places act placement in "Medel Defe" Note Elbow	
	With Leader	
	Tangent Leader	
	85.00 - 02	placement

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.	Name of		Broad Specifications	Quantity	Remarks		
No.	Resource	Make	Details	Quantity	(If any)		
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 <u>must design</u> 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]

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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						
	Process Related (10 Marks)						
1	Use of proper commands.	20%					
2	Completion of drawing with minimum size of	2004					
	model tree.	20%					
	Product Related (15 Marks)						
3	Generation and printing of drawing views, tables,	2004					
	etc. and their arrangement on different sheet size.	2070					
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.

Ma	arks 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	(i5 or higher), RAM minimum 4 GB; A3 / A4 size printer / plotter. Display-	As per batch
	computer.	wide Screen preferably.	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	1
		prototypingMachine.	

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.	Name of		Broad Specifications	Quantity	Remarks
No.	Resource	Make	Details	Quantity	(If any)
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 <u>must design</u> 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	20%
	model tree.	
	Product Related (15 Marks)	60%
3	Generation and printing of drawing views, tables,	20%
	etc. and their arrangement on different sheet size.	
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.

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

XII Actual Procedure Followed

•••••••••••••••••		••••••••••••••••••••••••••••••	•••••••••••••••••••
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XIII Precautions Followed

XIV Course Proficiency

XV Practical Related Questions

Note: Below given are few sample questions for reference. Teachers <u>must design</u> 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	2004	
	model tree.	2070	
	Product Related (15 Marks)		
3	Generation and printing of drawing views, tables,	2004	
	etc. and their arrangement on different sheet size.	2070	
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.

Ma	arks Obtained	Dated signature of Teacher	
Process	Product	Total	
Related(10)	Related(15)	(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 \Box 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 practical No.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 sicon 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 360° around the X axis using rotate \bigcirc 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.

Add Printer						
Printer	Tool Head Nozzle Diameter	Graphical LCD				
LubBot Mini LubBot Mini 2 LubBot TAZ 5 LubBot TAZ 6	SingleExtruder Aerostruder v1 DualExtruder v2 DualExtruder v2 DualExtruder v2 MOARstruder v2 MOARstruder H5 0.8 mm H5 + 1.2 mm SE 0.5 mm (Aero v2) SL 0.25 mm (Micro)	● Yes ○ No				

N. Machine setting:

1. Specify machine setting if any as per component size.

Machine Settings								
Printer	Hot End							
Printer Settings Printhead Settings			ngs					
X (Width)	280	mm		X min	20	mm		
Y (Depth)	280	mm		Ymin	10	mm		
Z (Height)	250	mm		X max	10	mm		
				V mov	10	mm		

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.

			Quality			-
			Layer Height	8 7	0.3	mm
	A REAL PROPERTY OF A REAL PROPER		Initial Layer Height	8 9	0.4	mm
		Mall Line Midth	Line Width	51	0.4	mm
			Wall Line Width		0.4	mm
		Width of a single wall line.	Outer Wall Line Width		0.4	mm

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.

		Width of the outermost wall line. By	Quality			-
		lowering this value, higher levels of	Layer Height	8 3	0.3	mm
	9	detail can be printed.	Initial Layer Height	8 7	0.4	mm
			Line Width	5 i	0.4	mm
		Affects	Wall Line Width		0.4	mm
		Wall Line Count	Outer Wall Line Width		0.4	mm

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.

	Inner Wall(s) Line Width	Quality			•
		Layer Height	8 2	0.3	mm
(Lin	Width of a single wall line for all wall lines except the outermost one	Initial Layer Height	8 7	0.4	mm
		Line Width	n 1	0.4	mm
	Affects	Wall Line Width		0.4	mm
	Martin Co	Outer Wall Line Width		0.4	mm
	Wall Line Count Cubic Subdivision Shell	Inner Wall(s) Line Width		0.4	mm

8. Top /bottom line width-0.4 mm width of single top/bottom line.

	A A A A A A A A A A A A A A A A A A A		Initial Layer Height	80	0.4	mm
			Line Width	51	0.4	mm
			Wall Line Width		0.4	mm
		Top/Bottom Line Width	Outer Wall Line Width		0.4	mm
		Width of a single top/bottom line.	Inner Wall(s) Line Width		0.4	mm
			Top/Bottom Line Width		0.4	mm

9. Infill line width -0.4mm

	Infill Line Width Width of a single infill line.	Outer Wall Line Width Inner Wall(s) Line Width	0.4	mm
10 Skint/Prim line width 0.4 mm	Affects	Top/Bottom Line Width	0.4	mm
10. Skirt/Brini inte widui-0.4 mini	Brim Line Count Prime Tower X Position	Skirt/Brim Line Width	0.4	mn

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. V	Vall line count	-3					
			by the wall thickness, this value is	🖄 Shell			•
		9	rounded to a whole number.	Wall Thickness		1.0	mm
				Wall Line Count	") i	3	
			Affects	Outer Wall Wipe Distance		0.25	mm
3. (Outer wall wipe	e distance-0.25 mm					
				🖄 Shell			-
	C C C C C C C C C C C C C C C C C C C	9		Wall Thickness		1.0	mm
				Wall Line Count	51	3	

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.

Outer Wall Wine Di

đ		Top Thickness The thickness of the top layers in the print. This value divided by the layer height defines the number of top layers.	Shell Wall Thickness Wall Line Count Outer Wall Wipe Distance Top/Bottom Thickness Top Thickness Top Layers	5 i 5 i	1.0 3 0.25 0.8 1.1 4	mm mm mm mm
5.	Top layers- 4					
B		Top Layers The number of top layers. When calculated by the top thickness, this value is rounded to a whole number. Affects	Shell Wall Thickness Wall Line Count Outer Wall Wipe Distance Top/Bottom Thickness Top Thickness Top Layers	5 / 5 /	1.0 3 0.25 0.8 1.1	mm mm mm mm
6.	Bottom layers-2	number.	Bottom Layers	5 i	2	

7. **Top /bottom pattern**- Lines, Bottom pattern initial layers-lines, Outer wall inset-0.05 mm

		Top/Bottom Pattern	Lines 🔻
Chr.		Bottom Pattern Initial Layer	Lines 💌
	Outer Wall Inset	Top/Bottom Line Directions	[]
	Inset applied to the path of the outer	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.

		Infil Domiter	🕅 Infill	-
			Infill Density	30 %
<u>Cir</u>	C Participation of the second se	Adjusts the density of infill of the print	Infill Line Distance	1.3333 mm
		princ.	Infill Pattern	Lines 💌
			Infill Line Directions	[]

2. Infill line distance-1.33mm, Infill pattern-Lines

		🔯 Infill		•
	Infill Line Distance	Infill Density	ຳ 30	%
C.C.		Infill Line Distance	1.3333	mm
	Distance between the printed infill	Infill Pattern	Lines	*
	infill density and the infill line width	Infill Line Directions	[]	

3. Infill Overlap-0.04 mm

Infill Overlap	Infill Line Distance	1.3333	mm
The amount of overlap between the	Infill Pattern	Lines	•
infill and the walls. A slight overlap	Infill Line Directions	[]	
allows the walls to connect firmly to	Infill Overlap Percentage	10	%
the infill.	Infill Overlap	0.04	mm

4. Skin overlap percentage- 5%

	and the innermost wall.	Skin Overlap Percentage	5	%
5. Skin overlap-0.02 mm				
	Affected Dy	Skin Overlap	0.02	mm
C Infill mine distance 0.1 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.

0.1

Infill Wipe Distance



- 7. Gradual infill steps-0
- 8. Minimum infill area- 0 mm^2

<u>an</u>	9		Gradual Infill Steps	0	
		Minimum Infill Area	Infill Before Walls Minimum Infill Area	2	mm ²
		Don't generate areas of infill smaller than this (use skin instead).	Skin Expand Distance	1.2	mm

Gradual Infill Steps

9. Skin expand distance-1.2 mm

and makes the walls on neighboring	Gradual Innii Steps	0
layers adhere better to the skin.	Infill Before Walls	
Lower values save amount of material	Minimum Infill Area	0 mm ²
used.	Skin Expand Distance	1.2 mm

- 10. Minimum skin angle for expansion-90°
- 11. Minimum skin width for expansion-00

	Maximum Skin Angle for Expansion	Gradual Infill Steps	0	
	Top and/or bottom surfaces of your	Infill Before Walls		
	object with an angle larger than this	Minimum Infill Area	0	mm ²
	setting, won't have their top/bottom	Skin Expand Distance	1.2	mm
	skin expanded. This avoids expanding	Maximum Skin Angle for Expansion	90	0

R. Process Parameter Options: Material setting

1. Default printing temperature-240° ^C Material Default Printing Temperatur Default Printing Temperature Printing Temperature 2.Printing temperature-240°C Material The temperature used for printing Set at 0 to pre-heat the printer Default Printing Temperature Printing Temperature 3.Probe temperature-170° ^C Material Default Printing Temperature Printing Temperature Probe Temperature 4.Soften temperature-170° ^C Probe Temperature often Temperature Soften Temperature Wipe Temperature Printing Temperature Initial La 5.Wipe temperature-170°^C Wipe Temperature 6.Printing temperature initial layer-245° ^C Printing Temperature Initial Laver Initial Printing Temperature

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 240° ^temperatures.

240

240 240

240

240 170

170

170

245

170

245

230

°C

1. Final printing temperature-225 ° c

Final Printing Temperature	Final Frinking Temperature		225	°C
	Build Plate Temperature	8	110	°C
The temperature to which to already start cooling down just before the end	Part Removal Temperature	8	50	°C
of printing.				

2. Build plate temperature-110° C

Build Plate Temperature	Final Printing Temperature		225	°C
	Build Plate Temperature	8	110	°C
he temperature used for the heated	Part Removal Temperature	Ø	50	°C
emperature will not be adjusted.	Keep Heating	8	~	

3. Part removal temperature- 50° ^C

Part Removal Temperature	Build Place remperature Part Removal Temperature	e D	50	°C
The ideal bed temperature to remove the model after printing.	Part Removal Temperature Keep Heating	8	~	
	Duild Dista Tamparatura Initial Lauran	0	110	0C I

- 4. Keep heating-Tick
- 5. Build plate temperature initial layer-110°^C

	Keep Heating	8	~
Build Plate Temperature Initial Layer	Build Plate Temperature Initial Layer	8	110 ℃
The temperature used for the heated	Diameter	າ	2.25 mm
build plate at the first layer.			

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.

Diameter	baild hate temperature miclor cayer	0 110	0
	Diameter	ວ 2.25	mm
Adjusts the diameter of the filament			
used. Match this value with the			
diameter of the used filament.	Ready to Save to File		

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.

	Initial Layer Flow Rate Changes the flow rate for the first layer only. Used to compensate for filament die swell.	Flow Initial Layer Flow Rate Enable Retraction Retract at Layer Change	8	100 100	%
9. Initial layer flow rate- 100%					_

	Initial Layer Flow Rate Changes the flow rate for the first layer only. Used to compensate for filament die swell.	Initial Layer Flow Rate Enable Retraction Retract at Layer Change	 ₽ 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.



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.

	Drint Sugar	③ Speed		+		
	Print Speed	Print Speed	ຳ 100	mm/s		
		The speed at which printing happens.	Infill Speed	40	mm/s	
				Wall Speed	50.0	mm/s

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.

Outer Wall Speed	Outer Wall Speed 🌼	45	mm/s
The speed at which the outermost	Inner Wall Speed	35	mm/s
walls are printed. Printing the outer wall at a lower speed improves the	Top/Bottom Speed	30	mm/s
final skin quality. However, having a	Travel Speed	175	mm/s
large difference between the inner	Initial Layer Speed	15	mm/s
wall speed and the outer wall speed will affect quality in a negative way.			

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.

			Ø Speed	¥
Cicle			Print Speed っ	100 mm/s
			Infill Speed 🤊 4	55 mm/s
			Wall Speed	50.0 mm/s
		Inner Wall Sneed	Outer Wall Speed 🤊 🕴	45 mm/s
	TR I FEE		Inner Wall Speed 🤊 i	100 mm/s

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.

THE PERSON AND THE PE	The speed at which travel moves are made	Inner Wall Speed	") i	100	mm/s
	more.	Top/Bottom Speed	5 i	45	mm/s
	Affacto	Travel Speed		175	mm/s

8 Initial layer speed-15 mm/s

	1				
		Initial Layer Speed	Inner Wall Speed	51	100 mm/s
		The speed for the initial layer. A lower	Top/Bottom Speed	" i C	45 mm/s
		value is advised to improve adhesion	Travel Speed		175 mm/s
MA		to the build plate.	Initial Layer Speed		15 mm/s

9 Initial layer print speed-15 mm/s



10 Initial layer travel speed- 26.26 mm/s

	Initial Layer Travel Speed	Inner Wall Speed	5 i	100	mm/s
	The speed of travel moves in the	Top/Bottom Speed	") i	45	mm/s
	initial layer. A lower value is advised to	Travel Speed		175	mm/s
TO DESCRIPTION OF THE OWNER	prevent pulling previously printed	Initial Layer Speed		15	mm/s
	parts away from the build plate. The value of this setting can automatically	Initial Layer Print Speed		15	mm/s
	be calculated from the ratio between	Initial Layer Travel Speed		26.25	mm/s
	the Travel Speed and the Print Speed.	Skirt/Brim Speed	θ	15	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.

				_	
	Skirt/Brim Speed	Skirt/Brim Speed	8	15	mm/s
	The speed at which the skirt and brim	Maximum Z Speed		0	mm/s
	are printed. Normally this is done at	Number of Slower Layers	8	2	
	the initial layer speed, but sometimes	Equalize Filament Flow			
TNV I I I I I I I I I I I I I I I I I I I	you might want to print the skirt or	e 11 a 11 a 18 a 1		H	
	brim at a different speed.				

12 Maximum Z speed-0 mm/s

Maximum 7 Sneed	okiro primopeco	0	15	
Maximum 2 opecu	Maximum Z Speed		1	mm/
The maximum speed with which the	Number of Clause Laures	- E	-	
build plate is moved. Setting this to	Number of Slower Layers	° L	2	
zero causes the print to use the	Equalize Filament Flow			
firmware defaults for the maximum z		_ F	٦	
speed.				
speed.				_

13 Number of slower layers-2

Numbe	er of Slower Layers	Maximum Z	Speed	0 0	mm/s
The firs than th	st few layers are printed slower ne rest of the model, to get	Equalize File	ament Flow		
better a improv prints.	adhesion to the build plate and ve the overall success rate of The speed is gradually	Ready to Save	to File Activate Windov	/S	
TA increas	sed over these layers.	00b 29min	Go to PC settings to a	ctivat /	Windows

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.	Name of]	Broad Specifications		Remarks
No.	Resource	Make	Details	Quantity	(If any)
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 <u>must design</u> 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					
	Process Related (10 Marks)					
1	Selecting relevant material, process and set up	30%				
	parameters.					
2	Slicing the solid model and transferring the file to	20%				
	the printer.					
3	3 Printing the components.					
	Product Related (15 Marks)	30%				
4	Safety unloading the manufactured component	10%				
	from the printer/machine.					
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.

M	arks 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
		(i5 or higher), RAM minimum 4 GB;	As per
1.	Hardware: Personal	A3 / A4 size printer / plotter. Display-	batch
	computer.	wide Screen preferably.	size
3.	Software	Any parametric solid modeling	As per
		software.	batch
			size
4.	3D printer	3D printer / Rapid prototyping	1
		Machine.	

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.stlfile 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 90° around the X axis using rotate \bigcirc 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



8. Infill line width- 0.4 mm

			Affects Infill Line Distance 	Infill Line Width	0.4	mm
	9.	Skirt/brim line width-0.4 mm				
	4		Skirt/Brim Line Width Width of a single skirt or brim line.	Infill Line Width Skirt/Brim Line Width 마카 Sholl	0.4	mm mm
K.	Proc	ess Parameter Options: Shell Settin	g:			
	1	Wall thickness-10 mm		-		_
			Wall Thickness	Wall Thickness	1.0	mm
	2	Wall line count-3				
			Affects	Wall Line Count Outer Wall Wipe Distance	⊃ i 0.25	mm
	3	Outer wall wipe distance-0.25 mm				
			Outer Wall Wipe Distance Distance of a travel move inserted	Outer Wall Wipe Distance	0.25	mm
	4	Top/Bottom thickness- 0.8 mm				
			Top/Bottom Thickness The thickness of the top/bottom layers in the print. This value divided by the layer height defines the number of top/bottom layers.	Outer Wall Wipe Distance Top/Bottom Thickness Top Thickness Top Layers Bottom Thickness	0.25 0.8 0.1 1.1 3 0.1 1.1	mm mm mm
	5	Top thickness-1.1mm				
			print. This value divided by the layer height defines the number of top layers. Affects	Top Bottom Entekness Top Thickness Top Layers Bottom Thickness	0.8 0.8 3 0 i 1.1 0 i 0	mm mm mm
	6	Top layers-03				
			Affects • Minimum Skin Width for Expansion	Top Thickness Top Layers Bottom Thickness		mm
	7	Bottom thickness-1.1 mm		00300 10 8300 10 FB0		
			Bottom Thickness The thickness of the bottom layers in	Top Layers Bottom Thickness		
	8	Bottom layers-03				
			Bottom Layers The number of bottom layers. When	Bottom Thickness Bottom Layers	5/1.1 5/8	mm
	9	Outer wall inset-0.05 mm				
			Outer Wall Inset	lop/Bottom Line Directions Outer Wall Inset	0.05	mm
	10	Horizontal expansion-00 mm				
			Horizontal Expansion Amount of offset applied to all	Horizontal Expansion	0 Sharpost	mm
	11	Extra skin wall count-01				
			Extra Skin Wall Count	Extra Skin Wall Count	1	

L. Process Parameter Options: Infill Setting:





M. Process Parameter Options: Material setting

1 Default printing temperature-250° ^C

		Default Printing Temperature	IIII Material		•
			Default Printing Temperature	ຳ 250	°C
2 H	Printing temperature-250° ^C				
		The temperature used for printing.	Printing Temperature	っ i 250	°C
3 I	Probe temperature-170 ° ^C				
		Probe leniperature	Probe Temperature	170	°C
		probing.	Soften Temperature Wipe Temperature	5 170 170	°C
4 \$	Soften temperature-170 ° ^C				
		Somen remperature The nextle temperature used before	Soften Temperature	っ 170	0
5 \	Wipe temperature-170 ° ^C				
		Wipe Temperature	Soften Temperature	ຳ 170	°C
		The nozzle temperature used before	Wipe Temperature	170	°C
		nozzle wiping.	Printing Temperature Initial Layer	255	2°
6	Drinting temperature initial layer 25	C00 C	initiai Printing, lemberature	240	°C
0	Printing temperature initial layer-25	0			
		Printing Temperature Initial Layer The temperature used for printing the	Printing Temperature Initial Layer Initial Printing Temperature	<i>つ i</i> 250 240	℃ ℃
7	Initial printing temperature-250° ^C				
		micual Princing lemperature	Initial Printing Temperature	う / 250	٥C
8	Final printing temperature-250° ^C				
		Final Printing lemperature	Final Printing Temperature	<i>う i</i> 250	°C
9	Build plate temperature-110° ^C				
		The temperature used for the heated	Build Plate Temperature	<i>∂</i> 110	°C
10	Part removal temperature-50° ^C				
			Part Removal Temperature	8 50	°C
11	Build plate temperature initial laye	er-110 ° ^C			
		Duille France reimperature militar Layer	Build Plate Temperature Initial Lay	er 🖉 110	°C
12	Diameter-2.25 mm				
		Diameter	Diameter	⊃ 2.25	mm
13	Flow percentage-100%				

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Row	Flow	100	9
14 Initial layer flow rate-100%			
Initial Layer Flow Rate	Initial Layer Flow Rate	<i>8</i> 100	%
15 Retraction distance-1mm			
Actual Million State	Retraction Distance	1	mm
16 Retraction speed-10 mm/s			
Retraction Speed	Retraction Speed	10	mm/s
17 Retraction retract speed-10 mm/s			
	Retraction Retract Speed	10	mm/s
18 Retraction prime speed-10 mm/s			
	Retraction Prime Speed	10	mm/s
19 Retraction extra prime amount-00 mm ²			
	Retraction Extra Prime Amount	0	mm³
20 Retraction minimum travel-0.8 mm			
The minimum distance of travel	Retraction Minimum Travel	0.8	mm
21 Maximum retraction count-90			
Mexanium Retraction Count	Maximum Retraction Count	90	
22 Minimum extrusion distance window-1 mm			
	Minimum Extrusion Distance Window	1	mm
23 Nozzel switch retraction distance-16 mm			
	Nozzle Switch Retraction Distance	16	mm
24 Nozzle switch retraction speed-20 mm/s			
Nozzle Switch Retraction Speed	Nozzle Switch Retraction Speed	20	mm/s
25 Nozzle switch retract speed-20 mm/s			
Nozzle Switch Retract Speed	Nozzle Switch Retract Speed	20	mm/s
26 Nozzle switch prime speed-20 mm/s			
	Nozzle Switch Prime Speed	20	mm/s

N. Process Parameter Options: Speed

1 Print speed-150 mm/s

e la		Drint Speed	Ø Speed		•
		Finit apeeu	Print Speed	つ 150	mm/s
2	Infill speed-150 mm/s				
			Infill Speed	ウ <i>i</i> 150	mm/s
3	Wall speed-75 mm/s				
		The speed at which the walls are	Wall Speed	75.0 n	nm/s
4	$O_{\rm ref}$ = $11 = 150 = 1/2$				

- 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



6 Click Save to file will start the 3D printing.

	p=====8.		
			Activate Windows
	LIAZ6_practical15_cadmodel_simplejob	00h 31min	Go to PC settings to activate Windows.
	70.0 x 20.0 x 22.0 mm	1.33m / ~ 8g	Save to File



7 Remove component carefully from printer table.

XII Resources Used

S.	Name of	Broad Specifications		Quantity	Remarks
No.	Resource	Make	Details		(If any)
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 <u>must design</u> 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)				
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.

Ma	arks Obtained	Dated signature of Teacher	
Process	Product	Total	
Related(10)	Related(15)	(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			
Sec	ond Semester:				
1	Bussiness Communication Using Computers	22009			
2	Computer Peripherals & Hardware Maintenace	22013			
3	Applied Science (Chemistry)	22014			
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 Electronics	22212			
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			
Thi	rd Semester:				
1	Applied Multimedia Techniques	22024			
2	Advanced Serveying	22301			
3 4	Mechanics of Structures	22302			
5	Building Construction	22304			
6	Concrete Technology	22305			
7	Strength Of Materials	22306			
8	Automobile Engines	22308			
9 10	Automobile Transmission System	22309			
11	Technology Of Inorganic Chemicals	22313			
12	Object Oriented Programming Using C++	22316			
13	Data Structure Using 'C'	22317			
14	Computer Graphics	22318			
15	Database Management System	22319			
10	Digital Techniques Principlos Of Database	22320			
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	Electrical Circuits & Networks	22329			
25	Electronic Measurments & Instrumentation	22333			
26	Principles Of Electronics Communication	22334			
27	Thermal Engineering	22337			
28	Engineering Matrology	22342			
29 30	Theory Of Machines	22343			
50		22044			
FOL	intri Semester:				
1	Hydraulics	22401			
2	Chemical Process Instrumentation & Control	22404			
4	Fluid Flow Operation	22407			
5	Technology Of Organic Chemicals	22410			
6	Java Programming	22412			
7	GUI Application Development Using VB.net	22034			
8	wicroprocessor	22415			
9 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	Dasic I UWEI EIECUUTICS	22421			

16 17 18 19	Digital Communication Systems Mechanical Engineering Measurments Fluid Mechanics and Machinery Fundamentals Of Mechatronics	22428 22443 22445 22048		
Fifth Semester:				
1 2 3 4 5 6 7 8 9 10 11 23 14 15 16	Design of Steel and RCC Structures Public Health Engineering Heat Transfer Operation Environmental Technology Operating Systems Advanced Java Programming Software Testing Control Systems and PLC's Embedded Systems Mobile and Wireless Communication Industrial Machines Switchgear and Protection Energy Conservation and Audit Power Engineering and Refrigeration Solid Modeling and Additive Manufacturing Guidelines & Assessment Manual for Micro Projects & Industrial Training	22502 22504 22510 22511 22516 22517 22518 22531 22533 22523 22523 22524 22525 22562 22053 22057		
<u>Sixt</u>	th Semester:			
$\begin{array}{c}1\\2\\3\\4\\5\\6\\7\\8\\9\\10\\11\\2\\13\\14\\15\\16\\17\\8\\9\\0\\21\\22\\3\\24\\5\\26\\27\\28\\29\end{array}$	Solid Modeling Highway Engineering Contracts & Accounts Design of R.C.C. Structures Industrial Fluid Power Design of Machine Elements Automotive Electrical and Electronic Systems Vehicle Systems Maintenance Software Testing Advanced Java Programming Mobile Computing System Programing Testing & Maintenance of Electrical Equipments Power Electronics Illumination Engineering Power System Operation & Control Environmental Technology Mass Transfer Operation Advanced Communication System Mobile Communication Embedded System Process Control System Industrial Automation Industrial Drives Video Engineering Optical Fiber & Mobile Communication Therapeutic Equipment Intensive Care Equipment Medical Imaging Equipment	17063 17602 17603 17604 17608 17610 17617 17618 17625 17632 17632 17637 17638 17637 17638 17643 17643 17648 17648 17665 17658 17663 17668 17663 17668 17669 17671 17672 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
<u>Se</u>	cond Year:	
1	Pharmaceutics - II	0811
2	Pharmaceutical Chemistry - II	0812

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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