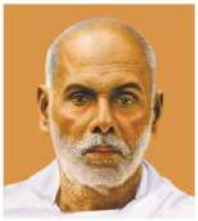
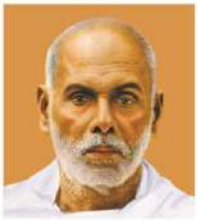
Sree Narayana Guru College
of Engineering & Technology
CHALAKKODE P.O., KOROM, PAYYANUR, KANNUR-670 307



SUPPORTING DOCUMENTS OF LAB

Sree Narayana Guru College
of Engineering & Technology
CHALAKKODE P.O., KOROM, PAYYANUR, KANNUR-670 307

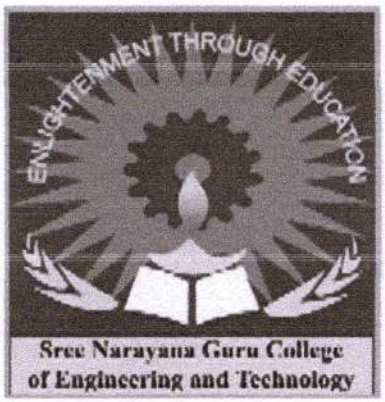


SUPPORTING DOCUMENTS OF LAB

1. LAB MANUAL

SREE NARAYANA GURU COLLEGE OF ENGINEERING & TECHNOLOGY, PAYYANUR

(AFFILIATED TO APJ ABDUL KALAM TECHNOLOGICAL UNIVERSITY)



COMPUTER AIDED DESIGN AND ANALYSIS LAB.

Department of Mechanical Engineering

Dr. LEENA A. V.
PRINCIPAL
SREE NARAYANA GURU COLLEGE OF
ENGINEERING & TECHNOLOGY, PAYYANUR
KANNUR

MEL332	COMPUTER AIDED DESIGN & ANALYSIS LAB	CATEGORY	L	T	P	CREDITS
		PCC	0	0	3	2

Preamble:

- To introduce students to the basics and standards of engineering design and analysis related to machine components.
- To make students familiarize with different solid modelling and analysis soft wares
- To convey the principles and requirements ofmodelling and analysis of machine elements.
- To introduce the preparation of part modelling and assembly modelling of machineries
- To introduce standard CAD packages to perform Finite Element Analysis of machine parts

Prerequisite:

EST 110 - Engineering Graphics
MEL 201 - Computer Aided Machine Drawing

Course Outcomes - At the end of the course students will be able to

CO1	Gain working knowledge in Computer Aided Design and modelling procedures.
CO2	Gain knowledge in creating solid machinery parts.
CO3	Gain knowledge in assembling machine elements.
CO4	Gain working knowledge in Finite Element Analysis.
CO5	Solve simple structural, heat and fluid flow problems using standard software

Mapping of course outcomes with program outcomes (Minimum requirements)

	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12
CO1	3	-	-	-	-	-	-	-	-	2	-	-
CO2	3	-	1	-	-	-	-	-	-	3	-	-
CO3	3	3	-	-	-	-	-	-	2	2	-	-
CO4	3	1	3	-	-	-	-	1	2	3	-	-
CO5	3	3	2	-	-	-	-	2	3	3	-	-

Mark Distribution

Total Marks	CIE Marks	ESE marks	ESE duration
150	75	75	2.5 hours

Continuous Internal Evaluation (CIE) Pattern:

Attendance	15 marks
Regular class work/Modelling and Analysis/Lab Record and Class Performance	30 marks
Continuous Assessment Test (minimum two tests)	30 marks

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Continuous Assessment test pattern

Bloom's Taxonomy	Continuous Assessment Tests	
	Test 1 - PART A MODELLING (marks)	Test 2 - PART B ANALYSIS (marks)
Remember	10	10
Understand	10	10
Apply	20	20
Analyse	15	15
Evaluate	20	20
Create	25	25

End semester examination pattern

End semester examination shall be conducted on modelling and analysis and based on complete syllabus. The following general guidelines should be maintained for the award of marks

Part A Assembly Modelling – 35 marks.

Part B Analysis – 30 marks.

Viva Voce – 10 marks.

Conduct of University Practical Examinations

The Principals of the concerned Engineering Colleges with the help of the Chairmen/Chairperson will conduct the practical examination with the approval from the University and bonafide work / laboratory record, hall ticket, identity card issued by college are mandatory for appearing practical University examinations. No practical examination should be conducted without the presence of an external examiner appointed by the University.

References Books:

1. Daryl Logan, A First course in Finite Element Method, Thomson Learning, 2007
2. David V Hutton, Fundamentals of Finite Element Analysis, Tata McGraw Hill,2003
3. Ibrahim Zeid, CAD/ CAM Theory and Practice, McGraw Hill, 2007
4. Mikell P. Groover and Emory W. Zimmer, CAD/ CAM – Computer aided design and manufacturing, Pearson Education,1987
5. T. R. Chandrupatla and A. D. Belagundu, Introduction to Finite Elements in Engineering, Pearson Education, 2012

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Experiment List (Minimum 12 exercises)

SL.NO	PART - A (Minimum 6 models)	COURSE OUTCOMES	HOURS
1	Creation of high end part models (minimum 2 models, Questions for examinations must not be taken from this portions)	CO1, CO2	6
2	Creating assembly models of Socket and spigot joint, Knuckle Joint, Rigid flange couplings, Bushed Pin flexible coupling, Plummer block, Single plate clutch and Cone friction clutch. Pipe joints, Screw jack, Tail stock etc. (minimum 4 models)	CO1, CO2, CO3	12
	PART – B (Minimum 6 problems)		
3	Structural analysis. (minimum 3 problems)	CO4, CO5	6
4	Thermal analysis. (minimum 2 problems)	CO4, CO5	3
5	Fluid flow analysis. (minimum 1 problem)	CO4, CO5	3

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Prepared By

JACOB THOMAS

Verified By

Athul Raj PP

HOD

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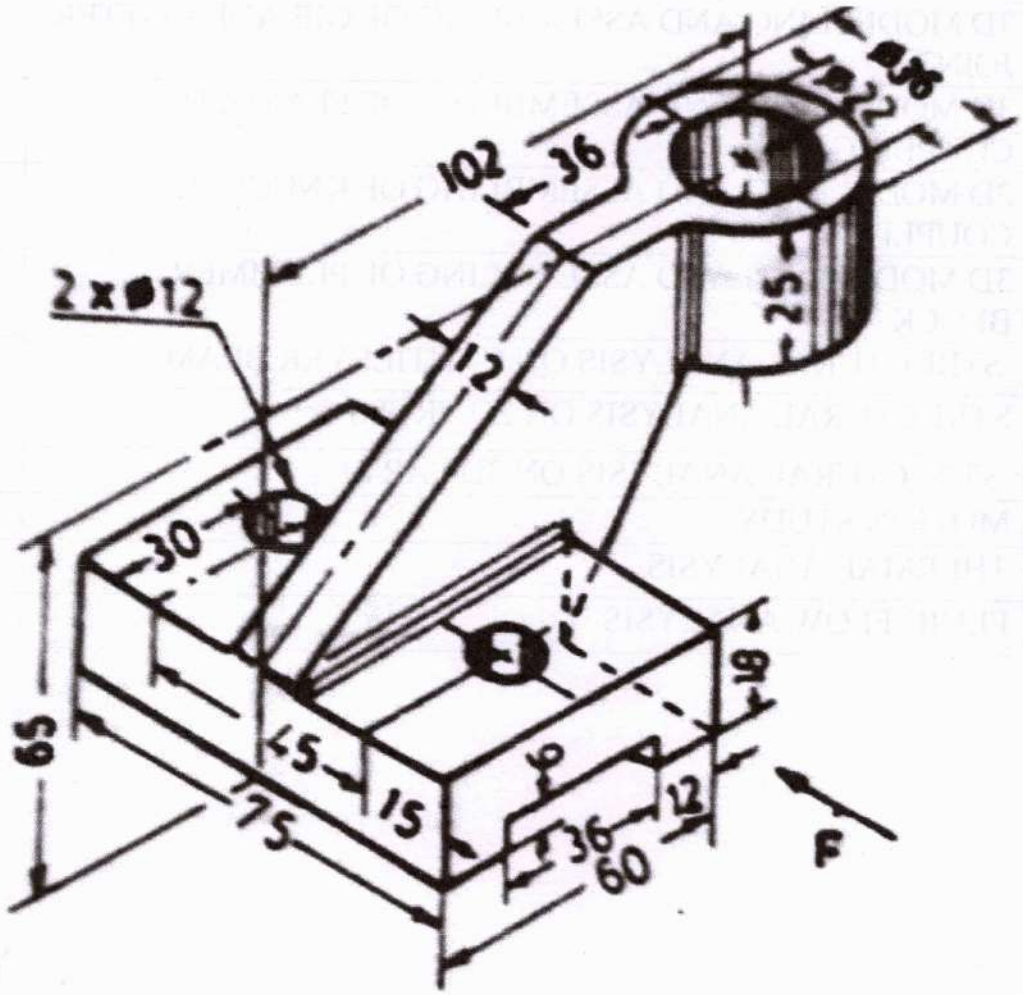

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EXPERIMENT No.1

ORTHOGRAPHIC VIEWS FROM 3D MODELS

Aim: To familiarize the 3D modelling of various design of objects and to generate 2D drawings for shop floor.

Equipments required;

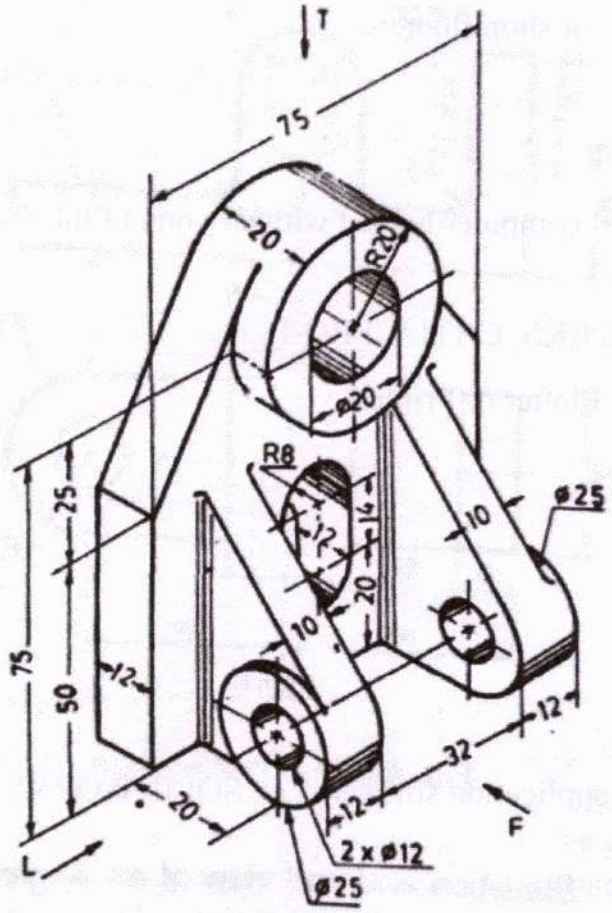
1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, CATIA, PRO-E

2. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS
2. Set unit "mmgs".
3. Select new part.
4. Select suitable 3D commands (Extrude, Revolve, Loft, and Sweep).
5. Select the required plan (Top, Front, and Side) and sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D sketch and Execute the selected 3D command.
7. Repeat the above steps from 4th until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (Fillet, Chamfer, Mirror, Pattern etc)
9. On completion of 3D modelling, save and close the file and open the new module 2D drafting and generate 2D drawing of the 3D model created previously.
10. 2D drafting file is an another file that also to be saved and closed.



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EXPERIMENT No.2

ORTHOGRAPHIC VIEWS FROM 3D MODELS

Aim: To familiarize the 3D modelling of various design of objects and to generate 2D drawings for shop floor.

Equipments required;

3. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, CATIA, PRO-E

4. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS
2. Set unit "mmgs".
3. Select new part.
4. Select suitable 3D commands (Extrude, Revolve, Loft, and Sweep).
5. Select the required plan (Top, Front, and Side) and sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D sketch and Execute the selected 3D command.
7. Repeat the above steps from 4^{th} until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (Fillet, Chamfer, Mirror, Pattern etc)
9. On completion of 3D modelling, save and close the file and open the new module 2D drafting and generate 2D drawing of the 3D model created previously.
10. 2D drafting file is an another file that also to be saved and closed.

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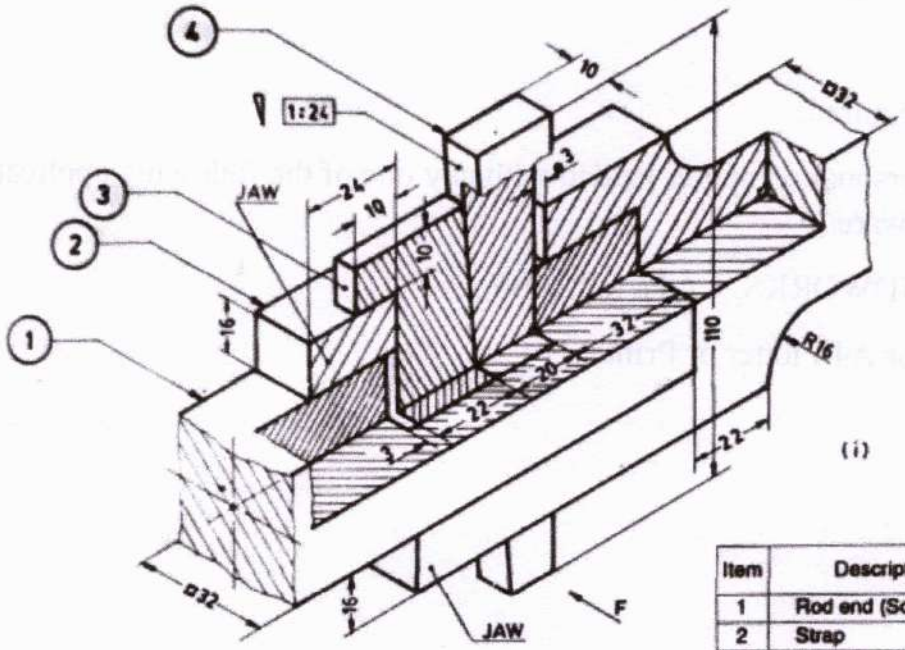


ITEM LIST

Item	Description	Qty.	Material
1	Rod end (Square)	1	M. S.
2	Strap	1	M. S.
3	Gib	1	M. S.
4	Cotter	1	M. S.

EXPERIMENT No.3

3D MODELLING AND ASSEMBLING OF GIB AND COTTER JOINT

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering functions.

Equipments required;

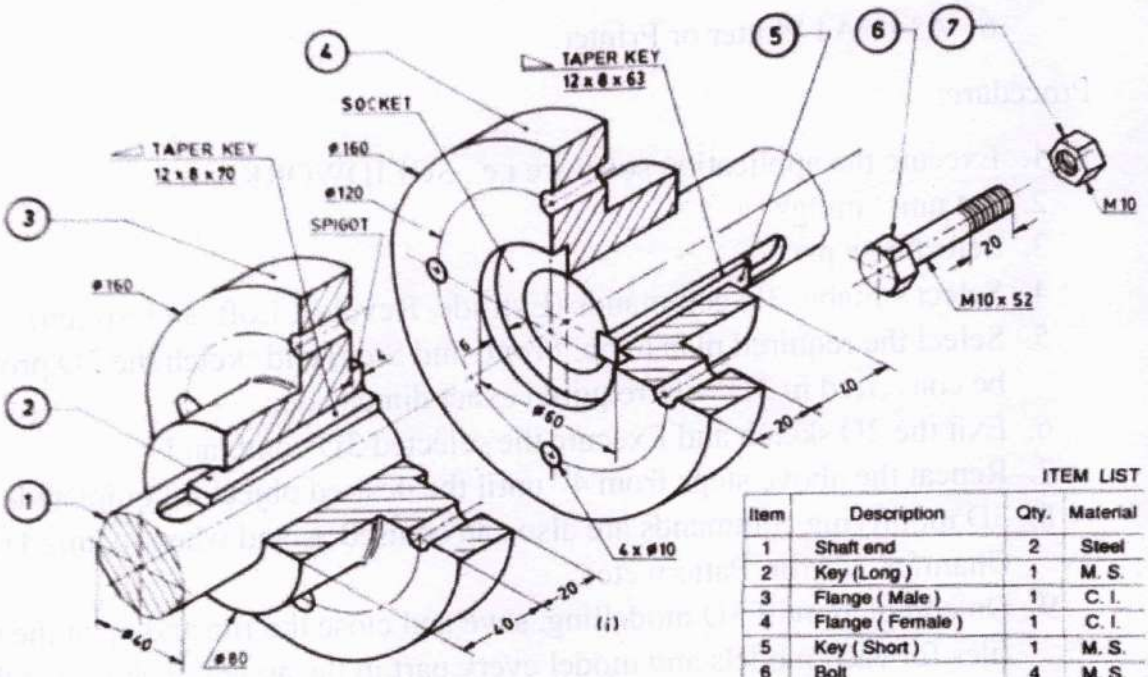
5. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, CATIA, PRO-E

6. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS
2. Set unit "mmgs".
3. Select new part.
4. Select suitable 3D commands (Extrude, Revolve, Loft, and Sweep).
5. Select the required plan (Top, Front, and Side) and sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D sketch and Execute the selected 3D command.
7. Repeat the above steps from 4^{th} until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (Fillet, Chamfer, Mirror, Pattern etc)
9. On completion of 3D modelling, save and close the file and open the new files for each models and model every part in the assembly and save them separately with their part name and close all files.
10. Open new module of assembly and open the base part for the assembly.
11. Insert all other parts as per the assembly sequence and make the assembly relationship with the base part or other parts inserted, using Mate command.
12. Save the assembly file with suitable name, and close.
13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.



ITEM LIST

Item	Description	Qty.	Material
1	Shaft end	2	Steel
2	Key (Long)	1	M. S.
3	Flange (Male)	1	C. I.
4	Flange (Female)	1	C. I.
5	Key (Short)	1	M. S.
6	Bolt	4	M. S.
7	Nut	4	M. S.

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EXPERIMENT No.4

3D MODELLING AND ASSEMBLING OF FLANGED COUPLING

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering functions.

Equipments required;

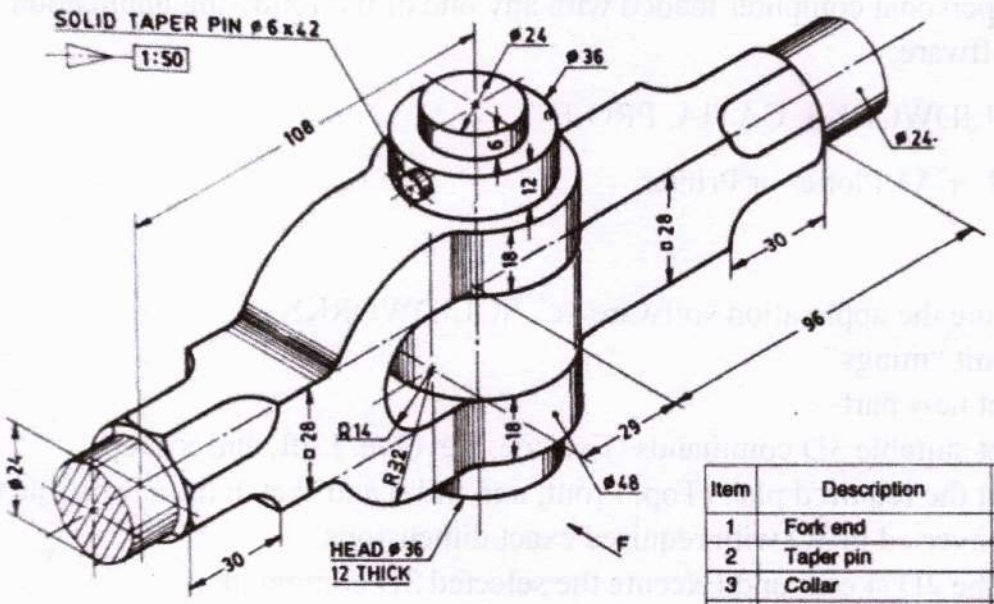
1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, CATIA, PRO-E

2. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS
2. Set unit "mmgs".
3. Select new part.
4. Select suitable 3D commands (Extrude, Revolve, Loft, and Sweep).
5. Select the required plan (Top, Front, and Side) and sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D sketch and Execute the selected 3D command.
7. Repeat the above steps from 4th until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (Fillet, Chamfer, Mirror, Pattern etc)
9. On completion of 3D modelling, save and close the file and open the new files for each models and model every part in the assembly and save them separately with their part name and close all files.
10. Open new module of assembly and open the base part for the assembly.
11. Insert all other parts as per the assembly sequence and make the assembly relationship with the base part or other parts inserted, using Mate command.
12. Save the assembly file with suitable name, and close.
13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.



ITEM LIST

Item	Description	Qty.	Material
1	Fork end	1	M. S.
2	Taper pin	1	M. S.
3	Collar	1	M. S.
4	Pin	1	M. S.
5	Eye end	1	M. S.

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EXPERIMENT No.5

3D MODELLING AND ASSEMBLING OF KNUCKLE COUPLING

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering functions.

Equipments required;

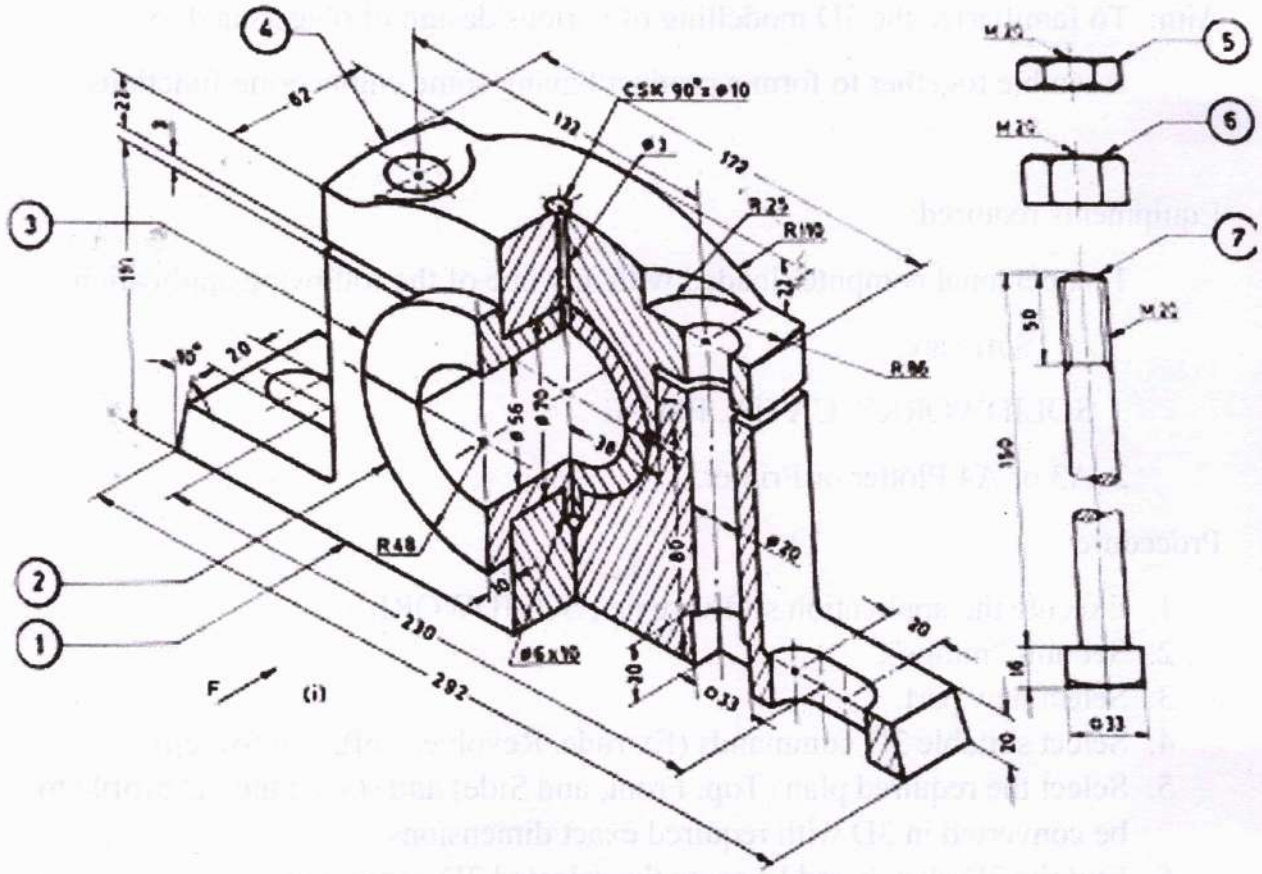
1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, CATIA, PRO-E

2. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS
2. Set unit "mmgs".
3. Select new part.
4. Select suitable 3D commands (Extrude, Revolve, Loft, and Sweep).
5. Select the required plan (Top, Front, and Side) and sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D sketch and Execute the selected 3D command.
7. Repeat the above steps from 4^{th} until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (Fillet, Chamfer, Mirror, Pattern etc)
9. On completion of 3D modelling, save and close the file and open the new files for each models and model every part in the assembly and save them separately with their part name and close all files.
10. Open new module of assembly and open the base part for the assembly.
11. Insert all other parts as per the assembly sequence and make the assembly relationship with the base part or other parts inserted, using Mate command.
12. Save the assembly file with suitable name, and close.
13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.



EXPERIMENT No.6

3D MODELLING AND ASSEMBLING OF PLUMMER BLOCK

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering functions.

Equipments required;

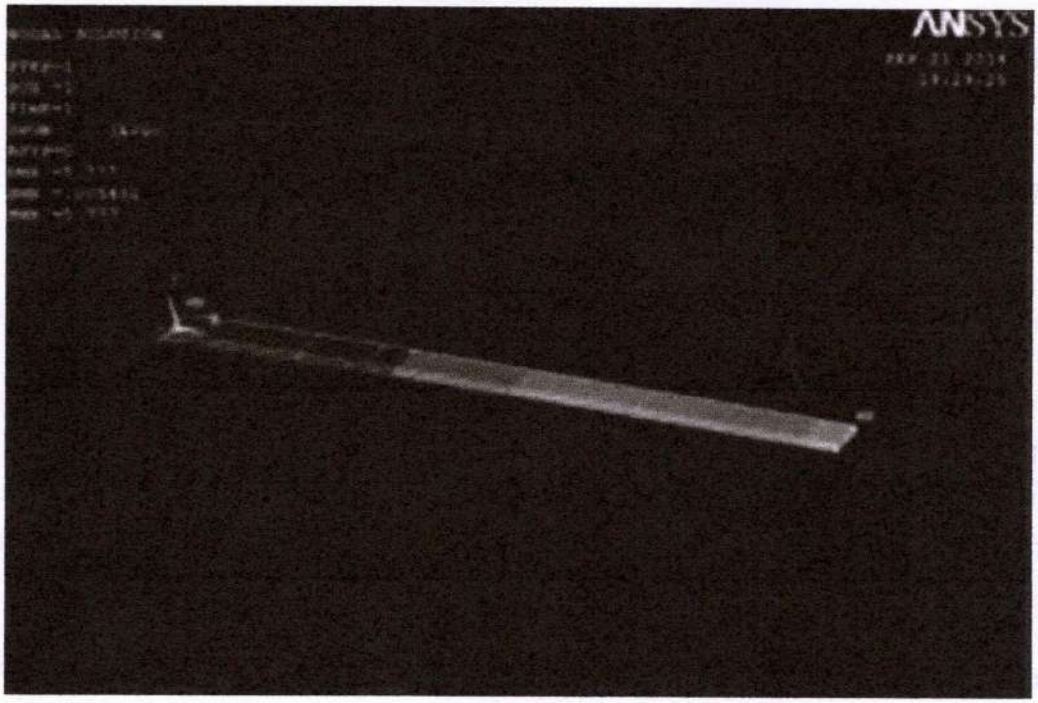
1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, CATIA, PRO-E

2. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS
2. Set unit "mmgs".
3. Select new part.
4. Select suitable 3D commands (Extrude, Revolve, Loft, and Sweep).
5. Select the required plan (Top, Front, and Side) and sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D sketch and Execute the selected 3D command.
7. Repeat the above steps from 4^{th} until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (Fillet, Chamfer, Mirror, Pattern etc)
9. On completion of 3D modelling, save and close the file and open the new files for each models and model every part in the assembly and save them separately with their part name and close all files.
10. Open new module of assembly and open the base part for the assembly.
11. Insert all other parts as per the assembly sequence and make the assembly relationship with the base part or other parts inserted, using Mate command.
12. Save the assembly file with suitable name, and close.
13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.



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EXPERIMENT No.7

STRUCTURAL ANALYSIS ON CANTILEVER BEAM

Aim: To familiarize the structural analysis on ANSYS Software

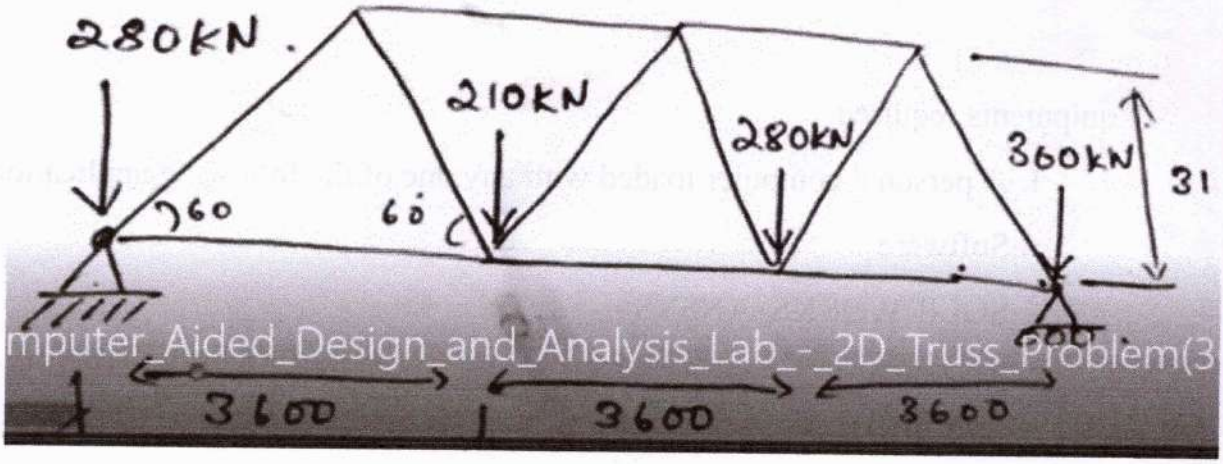
Equipments required;

1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, ANSYS
2. A4Printer.

Procedure:

1. Execute the application software i.e., ANSYS (Mechanical APDL).
2. Set Preference of study.
3. Model the 2D elements to be studied, as per the requirements.
4. Set the material type.
5. Assign the material properties.
6. Create the meshing of the modelled part.
7. Assign the boundary conditions, by applying degree of freedom and external loads at required positions.
8. Solve problem with given boundary conditions.
9. Open the result of deformation, stress and strain plot.
10. Generate the report.



EXPERIMENT No.8

STRUCTURAL ANALYSIS ON 2D TRUSS

Aim: To familiarize the structural analysis on ANSYS Software

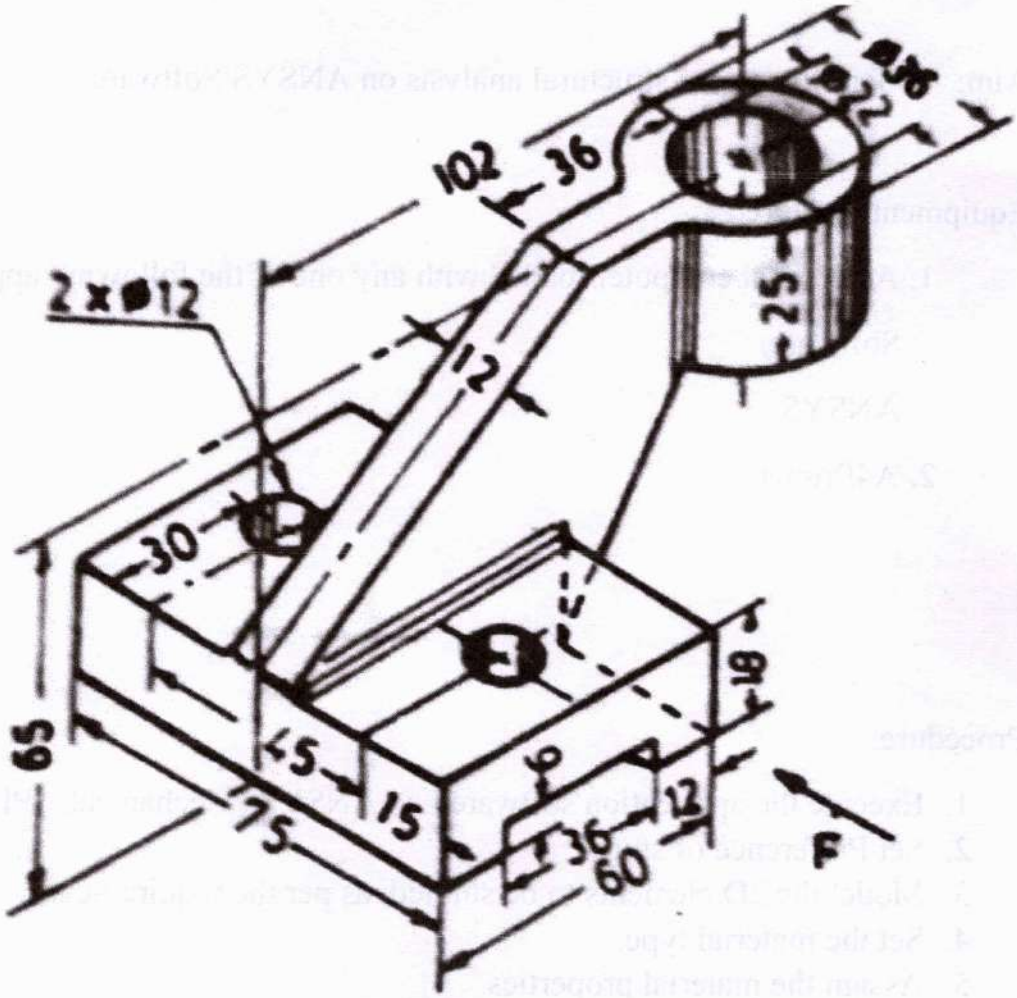
Equipments required;

1. A personal computer loaded with any one of the following application Software.

   ANSYS
2. A4Printer.

Procedure:

1. Execute the application software i.e., ANSYS (Mechanical APDL).
2. Set Preference of study.
3. Model the 2D elements to be studied, as per the requirements.
4. Set the material type.
5. Assign the material properties.
6. Create the meshing of the modelled part.
7. Assign the boundary conditions, by applying degree of freedom and external loads at required positions.
8. Solve problem with given boundary conditions.
9. Open the result of deformation, stress and strain plot.
10. Generate the report.



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EXPERIMENT No.9

STRUCTURAL ANALYSIS ON 3D PART

Aim: To familiarize the structural analysis on SOLIDWORKS Software

Equipments required;

1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS
2. A4Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS and add in solidworks simulation.
2. Create new study (static).
3. Insert the 3D model.
4. Set the material type.
5. Assign the material properties.
6. Assign the boundary conditions, by applying degree of freedom and external loads at required positions.
7. Create the meshing of the modelled part
8. Solve problem with given boundary conditions.
9. Open the result of deformation, stress and strain plot.
10. Generate the report.

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EXPERIMENT No.10

MOTION STUDY OF SCREW AND NUT

Aim: To familiarize the motion study on SOLIDWORKS Software

Equipments required;

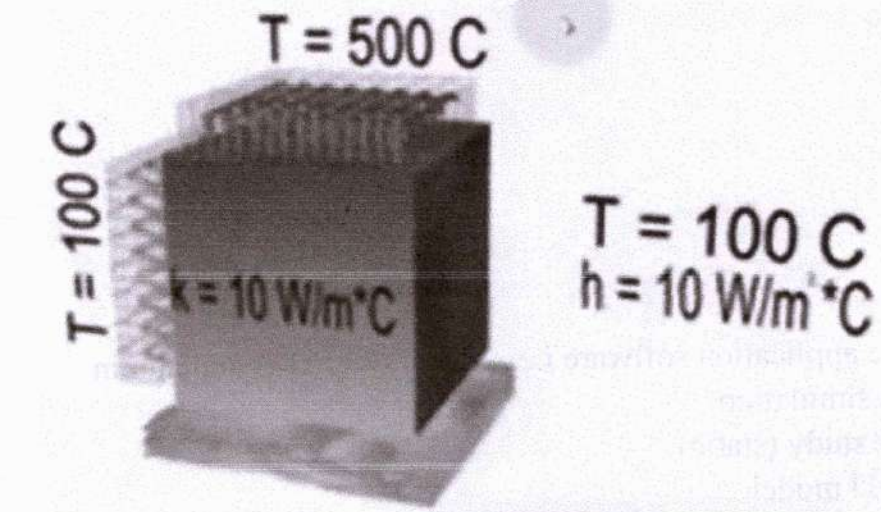
1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS
2. A4Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS and add in solidworks simulation.
2. Create new study (static).
3. Insert the 3D model.
4. Set the material type.
5. Assign the material properties.
6. Assign the boundary conditions, by applying degree of freedom.
7. Play the motion study.

ditions: Example is constrained as shown in the following figure (Note that the vertic



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EXPERIMENT No.11

THERMAL PROBLEM ON MIXED BOUNDARY ANALYSIS

Aim: To familiarize the structural analysis on ANSYS Software

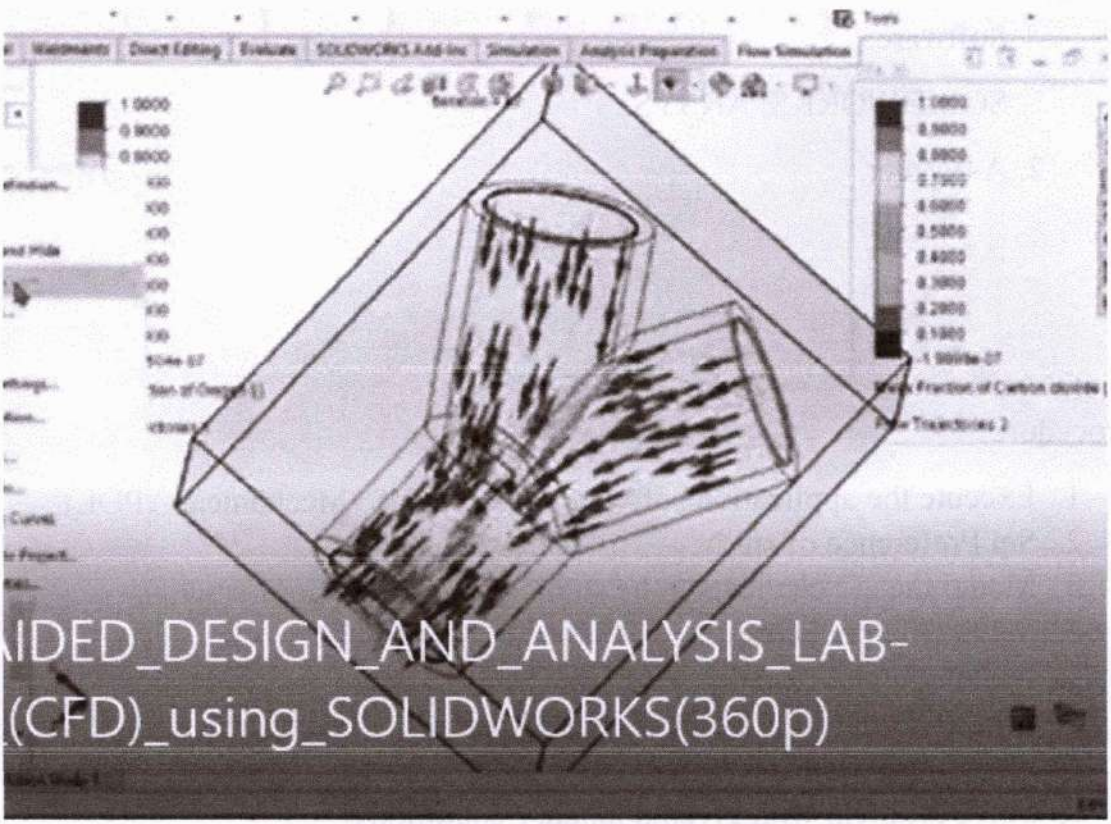
Equipments required;

1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS, ANSYS
2. A4Printer.

Procedure:

1. Execute the application software i.e., ANSYS (Mechanical APDL).
2. Set Preference of study.
3. Model the 2D elements to be studied, as per the requirements.
4. Set the material type.
5. Assign the material properties.
6. Create the meshing of the modelled part.
7. Assign the boundary conditions,
8. Solve problem with given boundary conditions.
9. Open the result of deformation, stress and strain plot.
10. Generate the report.



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EXPERIMENT No.12

FLUID FLOW ANALYSIS

Aim: To familiarize the Fluid flow analysis on SOLIDWORKS Software

Equipments required;

1. A personal computer loaded with any one of the following application Software.

   SOLIDWORKS
2. A4Printer.

Procedure:

1. Execute the application software i.e., SOLIDWORKS and add in CFD simulation.
2. Create new study (CFD).
3. Insert the 3D model.
4. Set the material type.
5. Assign the material properties.
6. Assign the boundary conditions, by applying degree of freedom.
7. Run the CFD.
8. View the various results.
9. Generate the report.

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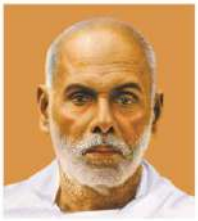

SUPPORTING DOCUMENTS OF LAB

2. LAB RECORD

CERTIFICATE

Date: 10 / 8 /2023

This is to certify that Mr./ Ms. ASHWIN JOHN

of Class S6 *Division* _____ *Roll No.* SNC20ME003

has satisfactorily completed the course experiments in practical

MEL-332 computer Aided Design and Analysis Lab

in the academic year of 2022 / 2023

in the Institution SNGCET

Teacher *Examiner* *Principal*

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Institution Rubber Stamp

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3	Orthographic views from 3D models	8	28/2/23	6/2/23	
4	3D modelling and Assembling of Gib and cotter joint	9-10	7/3/23	13/3/23	
5	3D modelling and Assembling of Flexible coupling	11-12	14/3/23	20/3/23	
6.	Experiment No:5 Knuckle Joint.	13-14	21/3/23	27/3/23	
7.	3D modelling and assembling of Plummer block	15-16	28/3/23	3/4/23	
8.	Structural Analysis on cantilever Beam	17	4/4/23	10/4/23	
9.	Structural Analysis on 2D Truss	18	11/4/23	17/4/23	
10	Structural analysis on 3D part	19	11/4/23	17/4/23	
11.	Motion study of Screw and Nut	20	18/4/23	19/6/23	
12	Thermal problem on mixed boundary Analysis	21	18/4/23	19/6/23	
13.	Fluid flow analysis.	22	20/6/23	21/6/23	

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SOLID WORKS

Solid work is a 3D mechanical CAD programme and is beloy developed by dessaut system.

Solid works is a mechanical design software to build programming models of parts and assemblies. It also includes various analysis and synthesis process. A solid is the computer type of geometric module used in CAD system. It contains all the mix framer and surface geometry to fully describe the edge and the face of mold.

Solid works includes many enhancements most in a CAD System. It contains all the enhancements most in delivered response to customer requist.

The top enhancements for solid works 2013 provides improvements to existing products and innovative new functionability.

MODELING METHODOLOGY

Parameters refers to conetrain where the value determines the shape of geometry of the model or assembly. Parameters are be either numeric parameters such as line length or circle diameter of geometri parameters, such as tangent petrallel horizontal or vertical etc.

Design Intent Intent is how the circles of the part wants to be respond to changes and updates, for example you would want the hole at the top of a a beverage can to stay at the top surface regardless of the height or size of the

Features refers to bountaliy blades re the Shapes and operations that conetract, the part shape based function typically.

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begin with 2D or 3D Sketch of shapes such as holes slots etc. operation based features are not Sketch based and include features such as fillets, chamfers etc!

Building a model in Solidworks usually start with a 2D Sketch. The consist of geometry Such as points, line are corners. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to specify the attributes Such as tangency, Parallelism, perpendicular. The parameteric nature at Solid works means that dimensions and relation drive the geometry, no way either grend.

In an assemble the analog to Sketch relations are mates just as Sketch relation define conditions Such as tangency, parallelism with respect Sketch geometry, assembly mate define equivalent relations with respect to the individual Solid works including additional advanced matine features Such as gear and cam following rutine.

finally, drawings can be created either from parts as resembles. views are automatically generated from Solid model and note, dimensions and tolerance can be easily added to the drawing as need. The drawing module includes

1. Drawing and elefaling
2. Assemblies
3. Motion Study
4. Parts and function
5. Roling
6. Short metal
7. Simulation.

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User Interface

Main elements at the Solid works user interface are

1. Member
2. command manager
3. Property manager
4. feature manager
5. configuration manager
6. Search assistant
7. Status bar
8. task Plane
9. Tool bar

Basic Concepts:

The Solid work model consist at 3D solid geometry in a part of assembly document. Drawing are created from model or by drafting view in a drawing document. We begin with a sketch. create a basic features and the add more features to our model. It is possible to eletino our design by addres: editing our decoding features. Associating between part, assemblies and drawings assume that changes made to one docket or file are automatically made to all other document and views

Assemblies:

In Solid works, we can create assemblies containing of many component which can be part or other assemblies called Sub assemblies. To create an assembly from parts.

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Signature:

1. ~~Go~~ click make assemble from part/assembly or file → assembly part Assembly open with the insert component property manager device
2. Click in the graphic area to add the parts to the assembly There are many ways to add the component the new or existing assembly
   a) Use the Insert ~~tool~~ component
   b) Drag from the 'file explore' tab in the task plan
   c) Drag from open documents windows
   d) Drag from design library in the task plane.

Mates:

Mates create geometric relationship between assembly components
eg: A coincident mate force two planes force to become coplanes

Types of mates

* Angle mates
* coincident mates
* Distance mate
* Lock mate
* Parallel and perpendicular

FLOXPRESS

Solid works floxpres in a fluid dynamics application that calculating ~~that~~ calculate how fluid ~~fluid~~ flow through operator assembly modules based on the calculated velocity fluid we can find problem areas in our device and ~~from~~ improve them before manufacturing

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Steps

1. Preparing the model for analysis enclose the inlet and outlet opening with tide
2. Start Solid works flowpress
   Click tools → Solid works flowpress
3. cheak geometry
4. Select ~~s~~thread
5. Selecting the flow inlet conditions
6. Selecting the flow outlet conditions
7. Solving the mode
   Click Solve
8. Viewing the results.

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Experiment Name / No.: _____ Camlin Page No. 7 Date 14/02/23

EXPERIMENT NO : 1

ORTHOGRAPHIC VIEWS FROM 3D MODELS

Aim: To familiarize the 3D modelling of various design of objects and to generate 2D drawings for shop floor.

Equipments Required:

1. A personal computer loaded with any one of the following application Software
   Solidworks, CATIA, PRO-E
2. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software ie SOLIDWORKS
2. Set unit "mmgs"
3. Select new part
4. Select Suitable 3D commands (Extrude, Revolve, Loft and Sweep).
5. Select the required Plan (Top, front, and side) and Sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D Sketch and execute the selected 3D command
7. Repeat the above steps from 4th until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (Fillet, chamfer, mirror, pattern etc)
9. On completion of modelling, save and close the file and open the new module 2D drafting and generate 2D drawing of the created previously
10. 2D drafting file is an another file saved and closed.

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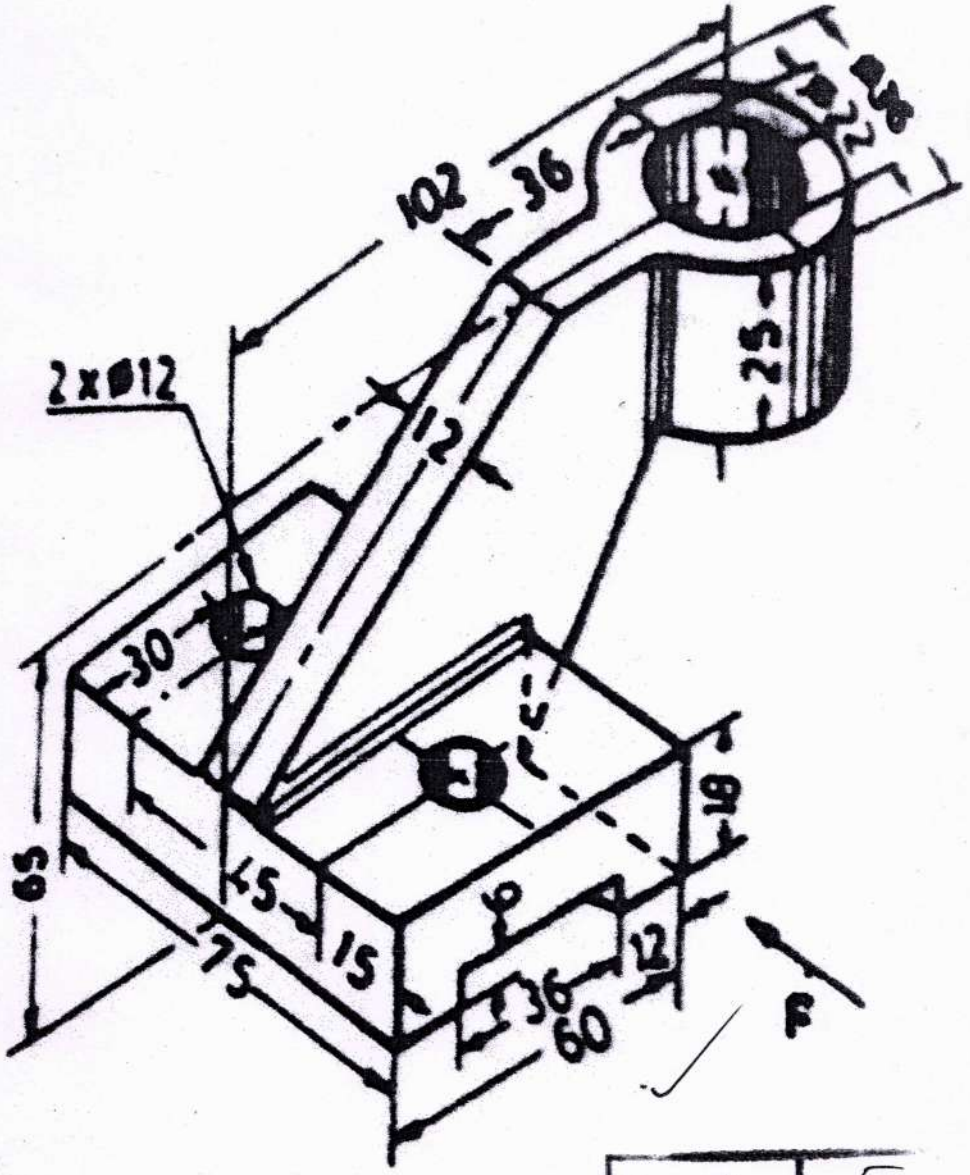

Procedure	5
Conduction	5
Result	5
Viva	10
Record	5

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Experiment Name / No.: _____ Camlin Page No. 8 Date 28 / 02 / 23

EXPERIMENT NO : 2

ORTHOGRAPHIC VIEWS FROM 3D MODELS

Aim :- To familiarize the 3D modelling of various design of objects and to generate 2D drawings for shop floor.

Equipments required;

1. A Personal computer loaded with any one of the following application Software.

   SOLIDWORKS, CATIA, PRO-E

2. A3 or A4 Plotter or Printer

PROCEDURE:

1. Execute the application Software ie SOLIDWORKS
2. Set unit "mmgs"
3. Select New Part
4. Select Suitable 3D command (Extrude, Revolve, Loft, and Sweep).
5. Select the required Plan (Top, front, and side) and sketch the 2D profile to be converted in 3D with required exact dimensions
6. Exit the 2D Sketch and Execute the selected 3D command
7. Repeat the above Step from 4th until the desired object is completed
8. 3D modifying commands are also can be used as and when required (fillet, chamfer, mirror, Pattern etc)
9. On completion of 3D modelling, save and close the file and open the new module 2D drafting and generate 2D drawing of the 3D model created previously
10. 2D drafting file is an another file that also to be saved and closed.

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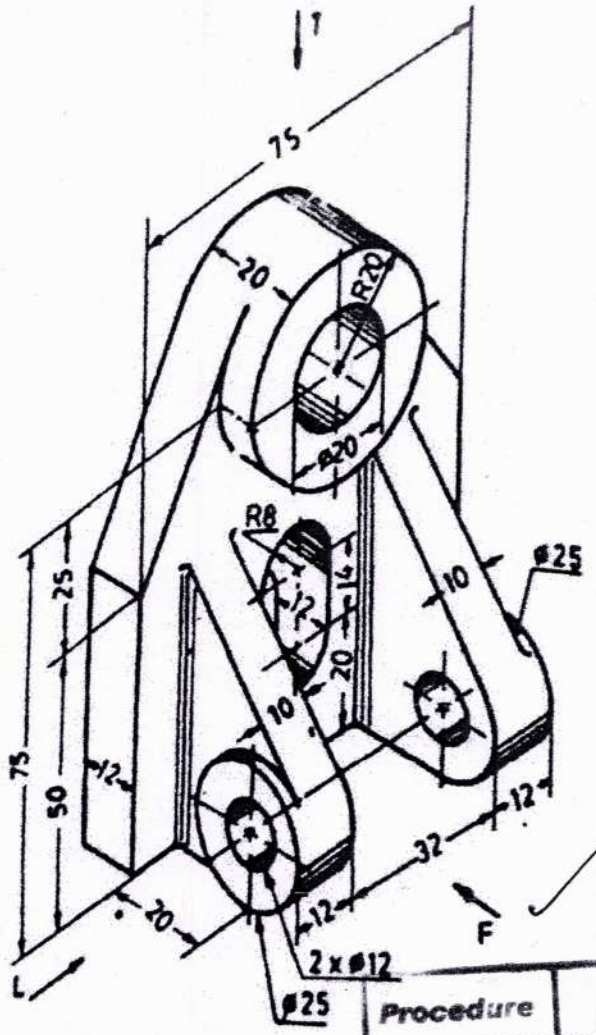

Procedure	5
Conduction	5
Result	5
Viva	9
Record	5

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Experiment Name / No.: _____ Camlin Page No. 9 Date 07 / 03 / 23

EXPERIMENT No 3

3D MODELLING AND ASSEMBLING OF GIB AND COTTER JOINT

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering functions

Equipments required:

1. A personal computer loaded with any one of the following application Software SOLIDWORKS, CATIA, PRO-E
2. A3 or A4 Plotter or Printer

Procedure:

1. Execute the application software ie SOLIDWORKS
2. Set unit 'mmgs'
3. Select new part
4. Select suitable 3D commands (Extrude, Revolve, Loft and Sweep).
5. Select the required plan (Top, front and side) and Sketch the 2D profile to be converted in 3D with required exact dimension
6. Exit the 2D Sketch and execute the selected 3D command
7. Repeat the above step from 4th until the desired object is completed.
8. 3D modifying commands are also can be used as and when required. (Fillet, chamfer, mirror, pattern etc)
9. On completion of 3D modelling, Save and close the file and open the new files for each model and model every part in the assembly and Save them Separately with their part name and close all file files.
10. Open new module of assembly and open the base part for the assembly
11. Insert all other parts as per the assembly, and open the base part

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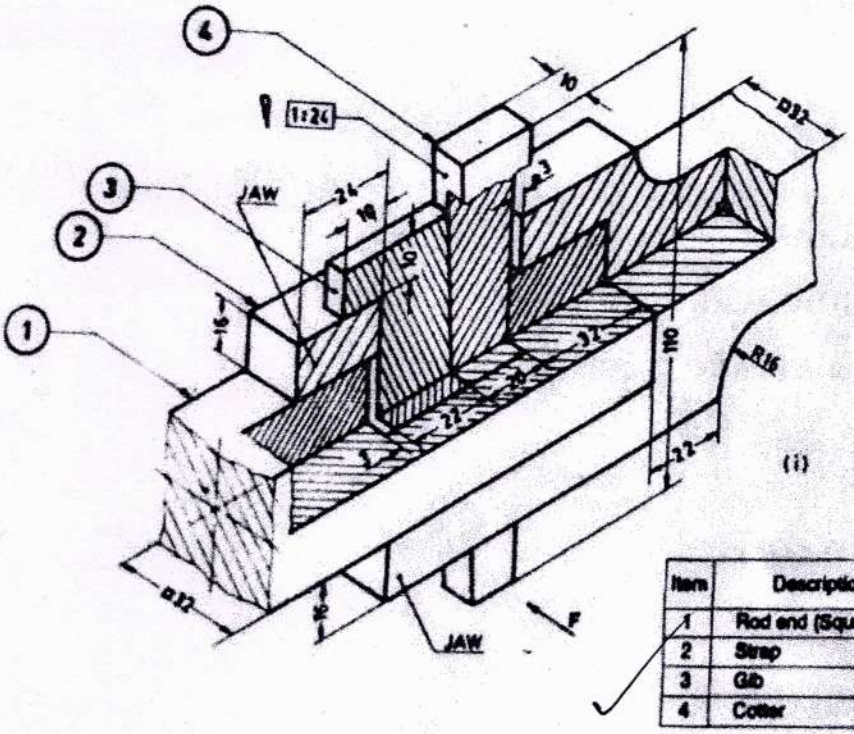

ITEM LIST

Item	Description	Qty.	Material
1	Rod end (Square)	1	M. S.
2	Strap	1	M. S.
3	Gib	1	M. S.
4	Cotter	1	M. S.

Procedure	5
Conduction	5
Result	5
Viva	9
Record	5

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for the assembly

11. Insert all other parts as per the assembly sequence and make the assembly relationship with the base part or other parts inserted using mate command.

12. Save the assembly file with suitable name, and close.

13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.

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Experiment Name / No.: _____ Camlin Page No. 11 Date 14/03/23

EXPERIMENT NO : 4

3D MODELLING AND ASSEMBLING OF FLANGED COUPLING

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering function.

Equipments required:

1. A personal computer loaded with any one of the following application Software

SOLIDWORKS, CATIA, PRO-E

Procedure

1. Execute the application Software ie SOLIDWORKS
2. Set unit "mmgs"
3. Select new part
4. Select Suitable 3D command (Extrude, revolve, Loft and Sweep)
5. Select the required plan (Top, front and side) and sketch the 2D Profile to be converted to 3D with required exact dimensions.
6. Exit the 2D sketch and Execute the selected 3D command
7. Repeate the above steps from 4th until the desired object is completed
8. 3D modifying command are also can be used as and when required (fillet, chamfer, mirror, pattern etc)
9. On completion of 3D modelling, save and close the file and open the new files for each models and model every part in the assembly and save them seperately with their part name and close all the files.
10. Open new module of assembly and open the base part for the assembly
11. Insert all other parts as per the assembly sequense and make the

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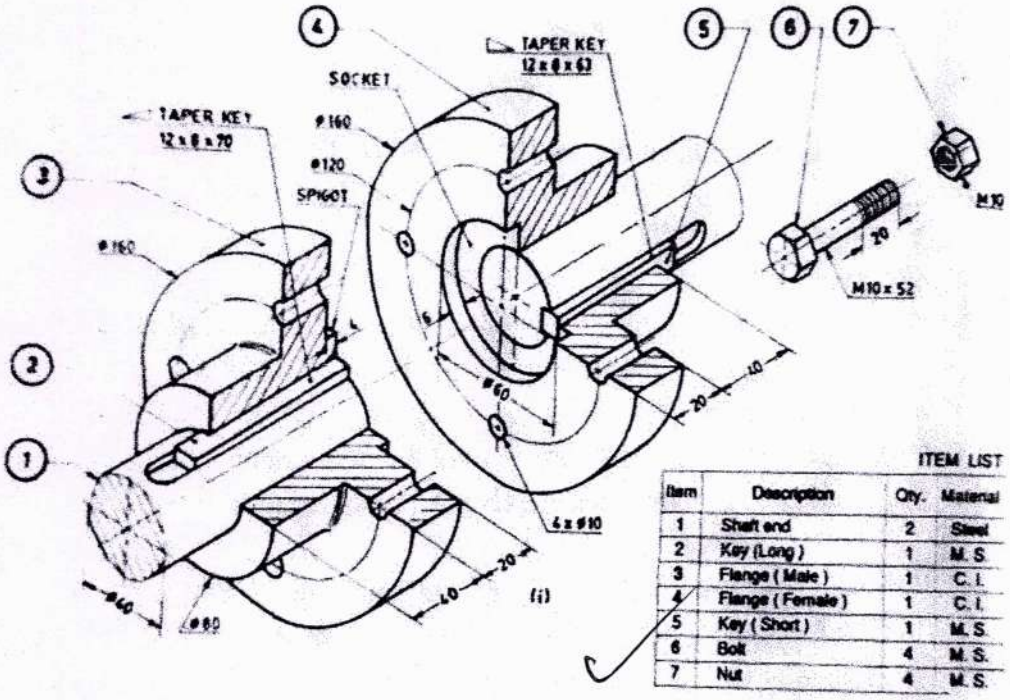

ITEM LIST

Item	Description	Qty.	Material
1	Shaft end	2	Steel
2	Key (Long)	1	M. S.
3	Flange (Male)	1	C. I.
4	Flange (Female)	1	C. I.
5	Key (Short)	1	M. S.
6	Bolt	4	M. S.
7	Nut	4	M. S.

Procedure	5
Conduction	5
Result	5
Viva	8.5
Record	5
	28½

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assembly relationship with the base part or other part inserted using mate command.

12. Save the assembly file with suitable name and close.

13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.

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Experiment Name / No.: _____ Camlin Page No. 13 Date 21/03/23

EXPERIMENT NO: 5

3D MODELLING AND ASSEMBLING OF KNUCKLE JOINT

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering functions.

Equipments required:

1. A personal computer loaded with any one of the following application softwares.
   SOLIDWORKS, CATIA, PRO-E
2. A3 or A4 Plotter or printer

Procedure:

1. Execute the application software ie SOLIDWORKS
2. Set unit "mmgs"
3. Select new part
4. Select suitable 3D command (Extrude, Revolve, Loft and Sweep)
5. Select the required plan (Top, front, and Side) and sketch the 2D profile to be converted in 3D with required exact dimentions
6. Exit the 2D sketch and Execute the selected 3D command
7. Repeat the above steps from 4th until the desired Object is completed.
8. 3D modifying commands are also can be used as and when required (fillet, chamfer, mirror, pattern etc)
9. On completion of 3D modelling Save and close the file and open the new files for each models and model every part in the assembly and Save them Separately with their part Name and close all files

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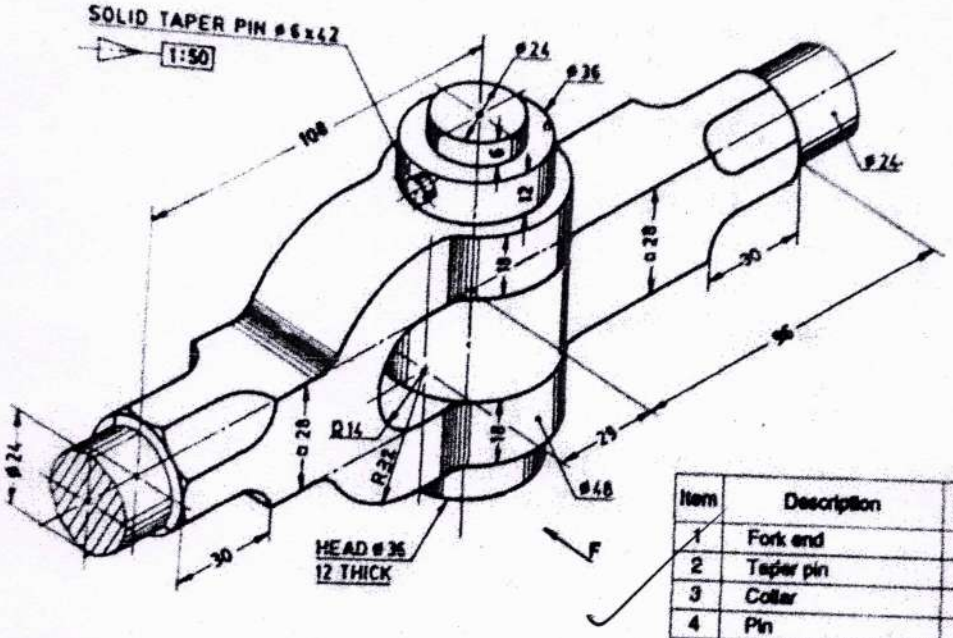

ITEM LIST

Item	Description	Qty.	Material
1	Fork end	1	M. S.
2	Taper pin	1	M. S.
3	Collar	1	M. S.
4	Pin	1	M. S.
5	Eye end	1	M. S.

Proced[illegible]	5
Conduction	5
Res[illegible]	5
Viva	8½
Record	5
	28½

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10. Open new module of assembly and open the base part for the assembly
11. Insert all other parts as per the assembly sequence and make the assembly relationship with the base part or other parts inserted, using mate command
12. Save the assembly file with suitable ~~name~~ name and close
13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.

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EXPERIMENT NO:6

3D MODELLING AND ASSEMBLING OF PLUMMER BLOCK

Aim: To familiarize the 3D modelling of various design of objects and to assemble together to form a product having some engineering functions.

Equipments required:

1. A personal computer loaded with any one of the following application software
   SOLIDWORKS, CATIA, PRO-E
2. A3 or A4 Plotter or Printer.

Procedure:

1. Execute the application software ie SOLIDWORKS
2. Set unit "mmgs"
3. Select new part
4. Select suitable 3D commands [extrude, revolve, Loft and sweep].
5. Select the required plan (Top, front and side) and sketch the 2D profile to be converted in 3D with required exact dimensions.
6. Exit the 2D sketch and execute the selected 3D command
7. Repeat the above steps from 4th until the desired object is completed.
8. 3D modifying commands are also can be used as and when required (fillet, chamfer, mirror, pattern etc)
9. On completion of 3D modelling, save and close the file and open the new files for each models and model every part in the assembly and save them seperately with their part name and close all files.

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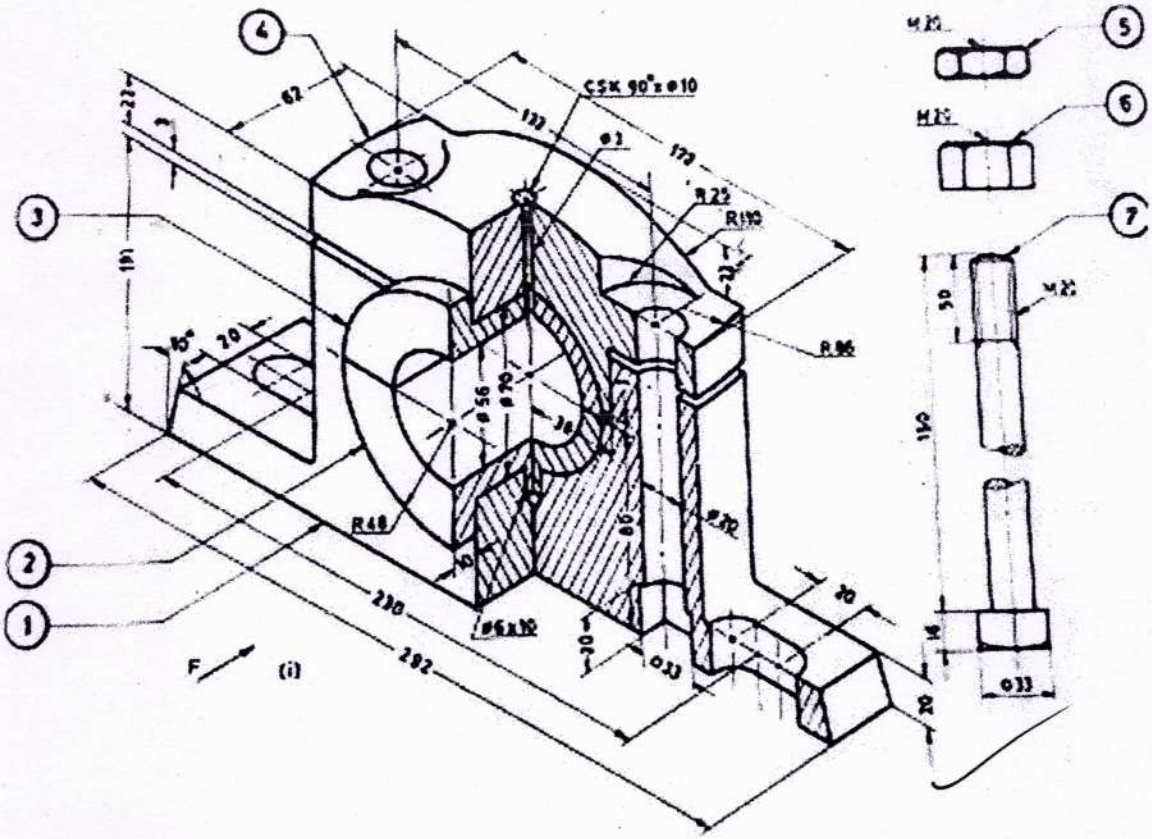

Procedure	5
Conduction	5
Result	5
Viva	8½
Record	5

28½

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10. Open new module of assembly and open the base part for the assembly
11. Insert all other parts as per the assembly sequence and make the assembly relationship with the base part or other parts inserted, using mate command
12. Save the assembly file with suitable name, and close
13. It is possible to generate 2D drawing of assembly whenever required with 2D drafting module.

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Experiment Name / No.: _____

Camlin | Page No. 17
Date 04/04/23

EXPERIMENT NO. 7

STRUCTURAL ANALYSIS ON CANTILEVER BEAM

Aim: To familiarize the structural analysis on ANSYS software

Equipment required:

1. A personal computer loaded with any one of the following application Software.
   SOLIDWORKS, ANSYS
2. A4 Printer

PROCEDURE;

1. Execute the application Software ie ANSYS (mechanical APDL)
2. Set preference of study
3. model the 2D elements to be studied, as per the requirements
4. Set the material type
5. Assign the material properties
6. Create the meshing of the modelled part.
7. Assign the boundary conditions, by applying degree of freedom and external loads at required position
8. Solve problem with given boundary conditions
9. Open the result of deformation, stress and strain plot
10. Generate the report.

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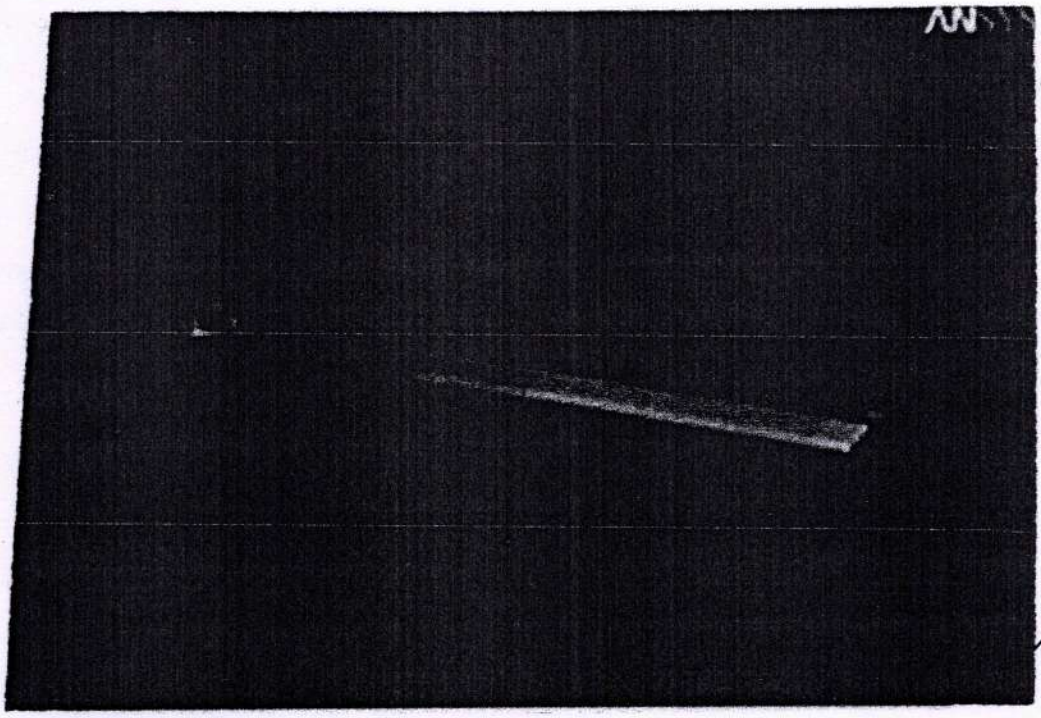

Procedure	5
Conduct	5
Result	5
va	9½
ord	5
	29½

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Experiment Name / No.: _____ Camlin Page No. 18 Date 11/04/23

EXPERIMENT NO. 8

STRUCTURAL ANALYSIS ON 2D TRUSS

Aim: To familiarize the structural analysis on ANSYS software.

Equipments required:

1. A personal computer loaded with any one of the following application Software
   ANSYS
2. A4 Printer.

Procedure:

1. Execute the application Software ie ANSYS (mechanical APDL)
2. Set preference of study
3. Model the 2D elements to be studied, as per the requirements.
4. Set the material type.
5. Assign the material properties.
6. Create the meshing of the modelled part.
7. Assign the boundary conditions, by applying degree of freedom and external loads at required positions
8. Solve problem with given boundary conditions
9. Open the result of deformation, stress and strain plot
10. Generate the report.

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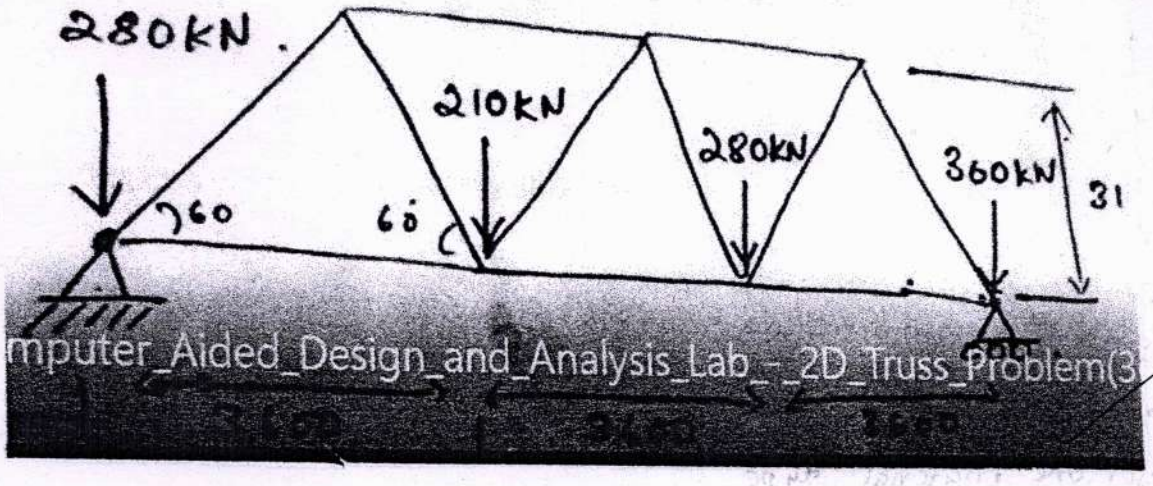

Procedure	5
Conduct	5
Result	5
va	9½
	5
	29½

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Experiment Name / No.: _____

Camlin / Page No. 19
Date 11 / 04 / 23

EXPERIMENT NO. 9

STRUCTURAL ANALYSIS ON 3D PART

Aim :

To familiarize the structural analysis on SOLIDWORKS Software

Equipment required

1. A personal computer loaded with any one of the following application Software
   SOLIDWORKS
2. A4 Printer

Procedure:

1. Execute the application software ie SOLIDWORKS and add in Solidworks Simulation
2. Create new study (Static)
3. Insert the 3D model
4. Set the material type
5. Assign the material properties
6. Assign the boundary conditions, by applying degree of freedom and external loads at required position
7. Create the meshing of the modelled part
8. Solve problem with given boundary conditions
9. Open the result of deformation, stress and strain plot
10. Generate the report.

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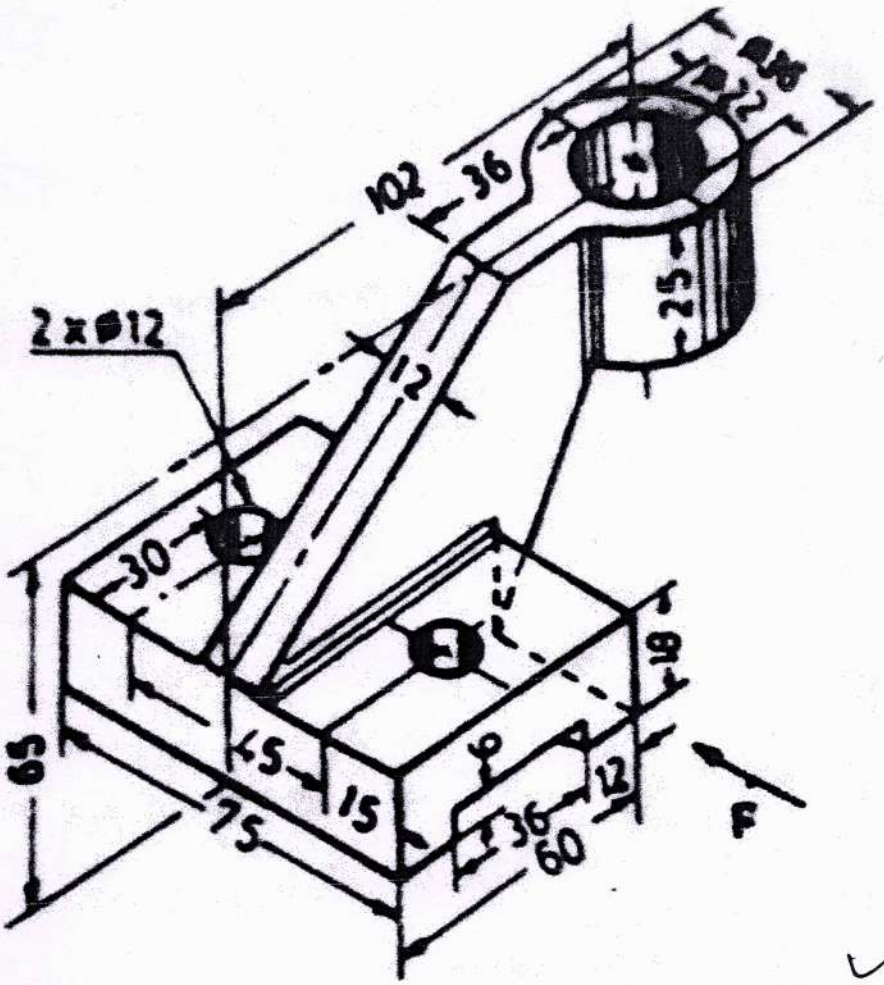

Procedure	5
Conduct	5
Result	5
	9½
	5
	29½

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EXPERIMENT NO.10

MOTION STUDY OF SCREW AND NUT

Aim: To familiarize the motion Study on SOLIDWORKS ~~Solid edge Works soft~~ Software

Equipment required;

1. A Personal computer loaded with any one of the following application Software
   SOLIDWORKS
2. A4 Printer.

Procedure:

1. Execute the application Software ie SOLIDWORKS and add in Solidwork simulation.
2. Create new Study (Static)
3. Insert the 3D model
4. Set the material type
5. Assign the material properties
6. Assign the boundary conditions by applying degree of freedom
7. Play the motion Study

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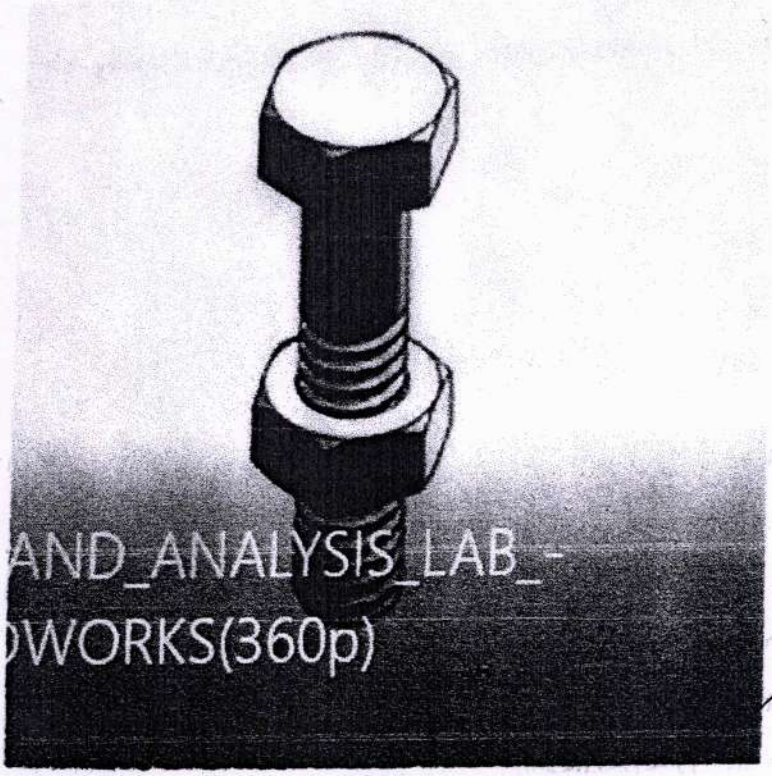

Procedure	5
Conduction	5
Result	5
Viva	9
Record	5

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PAYYANUR, KANNUR

Experiment Name / No.: _____ Camlin Page No. 21 Date 18/04/23

EXPERIMENT No. 11

THERMAL PROBLEM ON MIXED BOUNDARY ANALYSIS

Aim : To familiarize the structural analysis on ANSYS software

Equipments required

1. A personal computer loaded with any one of the following application Software.
   SOLIDWORKS, ANSYS
2. A4 Printer.

PROCEDURE :

1. Execute the application Software ie ANSYS (mechanical APDL)
2. set Preference of Study
3. Model the 2D elements to be studied, as per the requirements
4. Set the material type.
5. Assign the material properties
6. Create the meshing of the modelled part.
7. Assign the boundary conditions.
8. Solve problem with given boundary conditions.
9. Open the result of deformation, Stress and Strain plot.
10. Generate the report.

Teacher's Signature: _____

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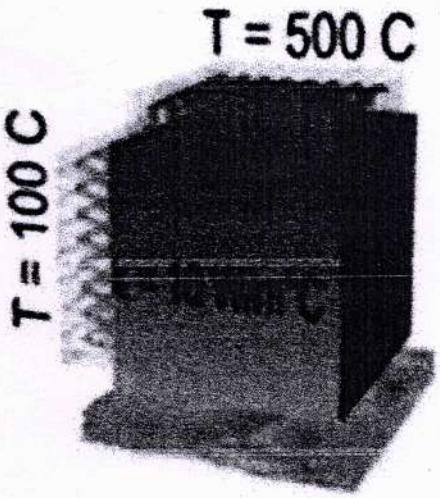
shown. Example is constructed as shown in the following figure (Note that the metal



Procedure	5
Conduction	5
Result	5
Viva	8½
Record	5
	28½

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EXPERIMENT NO : 12
FLUID FLOW ANALYSIS

Aim :

To familiarize the fluid flow analysis of solidworks software

Equipment required:

1. A personal computer loaded with any one of the following application software
   SOLIDWORKS
2. A4 printer

Procedure:

1. Execute the application software ie SOLIDWORKS and add in CFD Simulation.
2. Create new study (CFD)
3. Insert the 3D model
4. Set the material type
5. Assign the material properties
6. Assign the boundary conditions, by applying degree of freedom
7. Run the CFD
8. View the various results
9. Generate the report.

Teacher's Signature: _____

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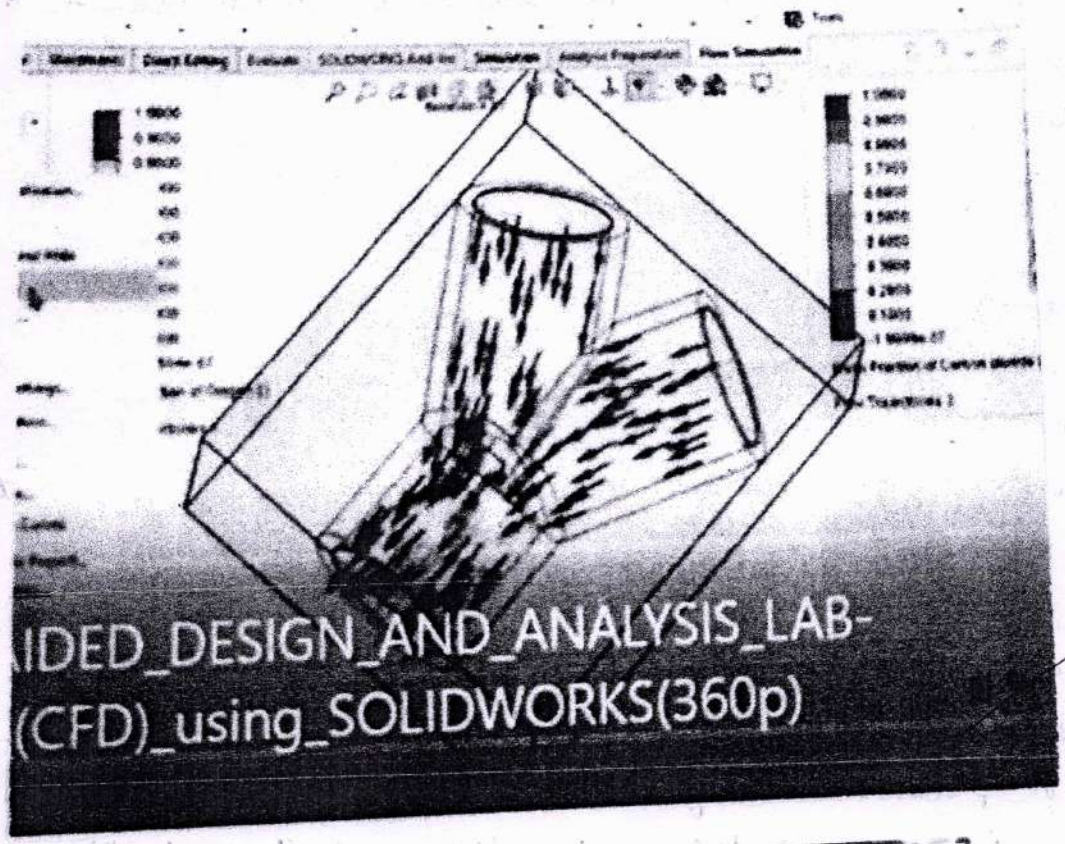

Procedure	5
Conduction	5
Result	5
Viva	9½
Record	5
	29½

Leena