

METHODIST

Estd: 2008

COLLEGE OF ENGINEERING & TECHNOLOGY Approved by AICTE New Delhi | Affiliated to Osmania University, Hyderabad

Abids, Hyderabad, Telangana, 500001

DEPARTMENT OF MECHANICAL ENGINEERING

LABORATORY MANUAL

COMPUTER AIDED ENGINEERING LAB MANUAL

BE VI Semester

For the Students admitted in AICTE Scheme

Name:
Roll No:
Branch:SEM:
Academic Year:

Empower youth- Architects of Future World



VISION

To produce ethical, socially conscious and innovative professionals who would contribute to sustainable technological development of the society.

MISSION

To impart quality engineering education with latest technological developments and interdisciplinary skills to make students succeed in professional practice.

To encourage research culture among faculty and students by establishing state of art laboratories and exposing them to modern industrial and organizational practices.

To inculcate humane qualities like environmental consciousness, leadership, social values, professional ethics and engage in independent and lifelong learning for sustainable contribution to the society.



METHODIST COLLEGE OF ENGINEERING & TECHNOLOGY

Approved by AICTE New Delhi | Affiliated to Osmania University, Hyderabad

Estd: 2008

Abids, Hyderabad, Telangana, 500001

DEPARTMENT OF MECHANICAL ENGINEERING

LABORATORY MANUAL

COMPUTER AIDED ENGINEERING LABORATORY (PC692ME)

Prepared by

Dr. Md. Fakhruddin H.N., Associate Professor Mr. Kamal Kumar Ojha, Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING

VISION

To be a reputed centre of excellence in the field of mechanical engineering by synergizing innovative technologies and research for the progress of society.

MISSION

- Toimpart quality education by means of state-of-the-art infrastructure.
- To involve in trainings and activities on leadership qualities and social responsibilities.
- To inculcate the habit of life-long learning, practice professional ethics and service the society.
- Toestablishindustry-institute interaction for stake holder development.

DEPARTMENT OF MECHANICAL ENGINEERING

After 3-5 years of graduation, the graduates will be able to:

PEO1: Excel as engineers with technical skills, and work with complex engineering systems.

PEO2: Capable to be entrepreneurs, work on globalissues, and contribute to industry and society through service activities and/or professional organizations.

PEO3: Lead and engage diverse teams with effective communication and managerial skills.

PEO4: Develop commitment to pursue life-long learning in the chosen profession and/or progress towards an advanced degree

DEPARTMENT OF MECHANICAL ENGINEERING

PROGRAM OUTCOMES

Engineering Graduates will be able to:

Po1. Engineering knowledge: Apply the basic knowledge of mathematics, science and engineering fund a mentals along with the specialized knowledge of mechanical engineering to understand complex engineering problems.

PO2. Problem analysis: Identify, formulate, design and analyse complex mechanical engineering problems using knowledge of science and engineering.

Po3. Design/development of solutions: Develop solutions for complex engineering problems, design and develop system components or processes that meet the specified needs with appropriate consideration of the public health and safety, and the cultural, societal, and environmental considerations.

PO4. Conduct investigations of complex problems: Formulate engineering problems, conduct investigations and solve using research-based knowledge.

PO5. Modern tool usage: Use the modern engineering skills, techniques and tools that include IT tools necessary for mechanical engineering practice.

Po6. The engineer and society: Apply the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7. Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

PO8. Ethics: Apply ethical principles and commit to professional ethics and responsibilities during professional practice.

PO9. Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO10.Communication: Communicate effectively on complex engineering activities to various groups, ability to write effective reports and make effective presentations.

PO11. Project management and finance: Demonstrate and apply the knowledge to understand the management principles and financial aspects in multidisciplinary environments.

PO12. Life-long learning: Recognize the need for, and have the preparation and ability to engage in Independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES

Mechanical Engineering Graduates will be able to:

PSO1: Apply the knowledge of CAD/CAM/CAE tools to analyse, design and develop the products and processes related to Mechanical Engineering.

PSO 2: Solve problems related to mechanical systems by applying the principles of modern manufacturing technologies.

PSO 3: Exhibit the knowledge and skill relevant to HVAC and IC Engines.

CODE OF CONDUCT

- 1. Students should report to the concerned labs as per the time table schedule.
- 2. Students who turn up late to the labs will in no case be permitted to perform the experiment scheduled for the day.
- 3. After completion of the experiment, certification of the concerned staff in-charge in the observation book is necessary.
- 4. Staff member in-charge shall award marks based on continuous evaluation for each experiment out of maximum 15 marks and should be entered in the evaluation sheet/attendance register.
- 5. Students should bring a note book of about 100 pages and should enter the readings/observations into the note book while performing the experiment.
- 6. The record of observations along with the detailed experimental procedure of the experiment performed in the immediate last session should be submitted and certified by the staff member in-charge.
- 7. Not more than three students in a group are permitted to perform the experiment on a setup for conventional labs and one student in case of computer labs.
- 8. The components required pertaining to the experiment should be collected from stores in-charge after duly filling in the requisition form.
- 9. When the experiment is completed, students should disconnect the setup made by them, and should return all the components/instruments taken for the purpose.
- 10. Any damage of the equipment or burn-out of components will be viewed seriously either by putting penalty or by dismissing the total group of students from the lab for the semester/year.
- 11. Students should be present in the labs for the total scheduled duration.
- 12. Students are required to prepare thoroughly to perform the experiment before coming to Laboratory.

DO'S

- 1. Leave footwear & bag outside the laboratory at their designated place.
- 2. Enter the system number in the register & use the system alone.
- 3. Report any broken plugs, exposed electrical wires or any unsafe conditions to your lecturer/laboratory staff immediately.
- 4

ReadandunderstandtheprocedurefromLabManualashowtocarryoutanactivitythoroughl y before coming to the laboratory.

- 5. Always keep anti-virus in active mode
- 6. Students must carry their Identity Cards & Observation Notes in the Lab.
- 7. Enter or Leave the lab only with the permission of the lab in charge.
- 8. Turn off the respective system and arrange the chairs properly before leaving the laboratory.

DON'TS

- 1. Do not install, uninstall or alter any software on computer.
- 2. Do not touch electrical fittings nor connect or disconnect any plug or cable.
- 3. Do not plug in external drives like pen drive, external hard disk or mobile phone
- 4. Students are not allowed to work in the Lab without the presence of faculty or instructor.
- 5. Do not leave your place, misbehave or make noise while in the Lab.
- 6. Don't scatter around unwanted things while doing an experiment.
- 7. Do not eat or drink in the laboratory.

COURSE OBJECTIVES

The objective of this course is

1.	To introduce fundamentals of the analysis software, its features and applications.
2.	To learn the basic element types in Finite Element analysis.
3.	To know the concept of discretization of continuum, Loading conditions and analyse the structure using pre-processor and postprocessor conditions.

COURSE OUTCOMES

CO No.	Course Outcomes	РО
CO 1	Classify different types of truss and perform static analysis	1,2,3,5,8,9,10,12
CO 2	Classify different types of meshing	1,2,5,8,9,10,12
CO 3	Analyze stress and deformation in case of axi-symmetric loading .	1,2,3,5,8,9,10,12
CO 4	Perform static analysis on connecting rod with 3D elements	1,2,3,5,8,9,10,12
CO 5	Predict natural frequencies in case of critical loading condition.	1,2,3,5,8,9,10,12
CO 6	Simulate coupled analysis using static structural and steady state thermal	1,2,3,5,8,9,10,12

COURSE OUTCOMES VS POS MAPPING

SNO	P01	PO2	PO3	PO4	PO5	P06	P07	PO8	PO9	PO10	P011	P012	PSO1	PSO2	PSO3
PC692ME.1	3.0	3.0	3.0	3.0	1.0	-	-	1.0	1.0	1.0	-	1.0	3.0	-	-
PC692ME.2	3.0	3.0	-	3.0	1.0	-	-	1.0	1.0	1.0	-	1.0	3.0	-	-
PC692ME.3	3.0	3.0	3.0	3.0	1.0	-	-	1.0	1.0	1.0	-	1.0	3.0	-	-
PC692ME.4	3.0	3.0	3.0	3.0	1.0	-	-	1.0	1.0	1.0	-	1.0	3.0	-	-
PC692ME.5	3.0	3.0	3.0	3.0	1.0	-	-	1.0	1.0	1.0	-	1.0	3.0	-	-
PC692ME.6	3.0	3.0	3.0	3.0	1.0	_	-	1.0	1.0	1.0	-	1.0	3.0	-	-
Avg	3.0	3.0	3.0	3.0	1.0	-	-	1.0	1.0	1.0	-	1.0	3.0	-	-

Exp. No	Experiment Name	Page No.					
Introduction	to Ansys	01					
I.	2D & 3D beam analysis with different sections, different materials for different loads (forces and moments with different end supports.	06					
2.	ID, 2D and 3D meshing with different element sizes for different CAD geometry	12					
3.	Analysis of Plane Truss & Spatial Truss with various cross sections and materials to determine member forces, member strains & stresses, joint deflections under static, thermal and combined loading.						
4.	Static analysis of plates with a hole to determine the deformations, the Stresses to study the failure behavior and SCF.	33					
5.	Plane stress, plane strain and axi-symmetric loading on the in plane members with in plane landing to study the stresses and strains.	44					
6.	Static analysis of connecting rod with tetrahedron and brick elements	51					
7.	Static Analysis of flat and curved shell due to internal pressure and moments to estimate the strains, stresses and reactions forces and moments with different boundary conditions .	60					
8.	Modal analysis of beams, plates and shells for natural frequencies and mode shapes.	67					
9.	Steady state heat transfer Analysis Cross section of chimney and transient heat transfer analysis of solidification of castings.	73					
10.	Non linear analysis of cantilever beam with non-linear materials at tip moment and post Buckling analysis of shells for critical loads	79					
II.	Coupled field analysis	86					
12.	Buckling analysis of plates, shells and beams to estimate BF and modes.	94					
13.	Harmonic analysis of a Shaft subjected to periodic force and transient analysis of plate subjected to stepped and ramped loading with varying time .	101					
I4.	Explicit analysis of car with 100m/s	109					

LIST OF EXPERIMENTS

INDEX

Experiment	Experiment Name	Date	Page No		Ma	rks	Remarks/	
No		Date		Ρ	R	v	т	Signature

Experiment	Experiment Name	Data	Page No.		Ma	rks	Remarks/	
No		Date	i uge i io	Р	R	v	т	Signature

Introduction to Ansys

Description:

The ANSYS Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and/or Design Modeler. ANSYS Workbench is a software environment for performing structural, thermal, and electromagnetic analyses. The class focuses on geometry creation and optimization, attaching existing geometry, setting up the finite element model, solving, and reviewing results. The class will describe how to use the code as well as basic finite element simulation concepts and results interpretation.

Steps in Ansys Workbench:

- Workbench GUI
- Engineering Data
- Design Modeler
- Geometry
- Model
- Meshing
- Setup
- Solution
- Result

Workbench GUI

Introduction This document serves as a step-by-step guide for conducting a Finite Element Analysis (FEA) using ANSYS Workbench. It will cover the use of the simulation package through the graphical user interface (GUI). More advanced topics will also be briefly covered.

Aims and Objectives

The purpose of this document is to provide step-by-step instructions on how to use ANSYS Workbench through the GUI. Upon completion, the student should be able to:

- use symmetry conditions to simplify a typical engineering problem
- perform a finite element simulation of a typical engineering problem
- investigate the effects of certain variables that are changed

Launch Ansys Workbench:

Launching ANSYS Workbench the ANSYS installation has many packages included. For this tutorial, we will be using ANSYS Workbench.

Engineering Data:

A part's response is determined by the material properties assigned to the part.

- Depending on the application, material properties can be linear or nonlinear, as well as temperature-dependent.
- Linear material properties can be constant or temperature dependent, and isotropic or orthotropic.
- Non linear material properties are usually tabular data, such as plasticity data (stress-strain curves for different hardening laws), hyper elastic material data.
- To define temperature dependent material properties, you must input data to define a property versus temperature graph.
- Although you can define material properties separately for each analysis, you have the option of adding your materials to a material library by using the **Engineering Data** tab. This enables quick access to and re-use of material data in multiple analyses.
- For all orthotropic material properties, by default, the Global Coordinate System is used when you apply properties to apart in the Mechanical application. If desired, you can also apply a local coordinate system to the part.

To manage materials, right-click on the Engineering Data cell in the analysis system schematic and choose **Edit**.

Geometry:

All analysis systems and several component systems, including Geometry, Meshing, and Mechanical Model, begin with a geometry-definition step. You can define the geometry differently depending on the type of simulation you are running. In most cases, you will use the **Geometry** cell. Via the **Geometry** cell, you can:

- Create a geometry from scratch in Design Modeler
- Import an existing geometry:
 - From neutral formats like IGES, STEP, Para solid, ACIS
 - From CAD files on disk
 - From an active CAD session that is already running on the same machine

For Fluid Flow simulations, you can also start with an imported mesh or case file; see Basic Fluid Flow Analysis, Starting from an Imported Mesh for details.

Specifying Geometry via the Context Menu

- 1. Right-click the **Geometry** cell.
- 2. Choose **New Geometry** to launch Design Modeler and create a new model, or choose **Import Geometry** and browse to an existing CAD model.

Alternatively, you can also launch ANSYS Workbench directly from some CAD systems. When doing so, ANSYS Workbench starts with a Geometry system in place and the CAD file already attached.

After you have attached or imported your geometry, the state appears as Up to Date, and the icon indicates the type of file imported.

If you do not need to make any additional changes to your geometry, you can continue working through the analysis as described in the next sections.

If your geometry needs to be modified before continuing with your analysis, you can edit the geometry in Design Modeler. After modifying the geometry in Design Modeler or importing a Design Modeler file, the icon in the **Geometry** cell will change to a Design Modeler icon. For a file imported and then modified in Design Modeler, you can open the file in Design Modeler, and the Design Modeler model tree will indicate the original source of geometry.

After the geometry is defined, you can share it with other systems. See Data Sharing and Data Transfer for more information on sharing geometry systems.

Model / Mesh:

ANSYS Twin Builder is a powerful platform for **modelling**, simulating and analysing virtual systems prototypes. It enables product development teams to verify and optimize performance of their software-controlled, multi domain systems designs.

The **Model** cell in the Mechanical application analysis systems or the Mechanical Model component system is associated with the Model branch in the Mechanical application and affects the definition of the geometry, coordinate systems, connections and mesh branches of the model definition.

When linking two systems, you cannot create a share between the Model cells of two established systems. You can generate a second system that is linked at the **Model** cell of the first system, but you cannot add a share after the second system has been created. Likewise, you cannot delete a link between the **Model** cells of two systems.

The **Mesh** cell in Fluid Flow analysis systems or the Mesh component system is used to create a mesh using the Meshing application. It can also be used to import an existing mesh file.

Edit

Launches the appropriate Model or Mesh application (the Mechanical application, Meshing, and so on.)

Setup:

Use the **Setup** cell to launch the appropriate application for that system. You will define your loads, boundary conditions, and otherwise configure your analysis in the application. The data from the application will then be incorporated in the project in ANSYS Workbench, including connections between systems.

For the Mechanical application systems, you will see the following **Setup** options, in addition to the common options:

Edit

Launches the Mechanical application with the geometry loaded and with cells mapped to their respective tree locations in the Mechanical application.

For CFX systems, you will see the following Setup options, in addition to the common options:

Edit

Launches CFX-Pre.

Import Case

Imports an existing case file containing physics data, region and mesh information for your analysis.

For Fluent systems, you will see the following Setup options, in addition to the common options:

Edit

Launches ANSYS Fluent.

Import Case

Imports an existing Fluent case file.

Solution:

From the **Solution** cell, you can access the Solution branch of your application, and you can share solution data with other downstream systems (for instance, you can specify the solution from one analysis as input conditions to another analysis). If you have an analysis running as a remote process, you will see the Solution cell in a pending state until the remote process completes. See the discussion on Understanding Cell States, below.

For the Mechanical application systems, you will see the following Setup options, in addition to the common options described earlier:

Edit

Launches the Mechanical application open to the Solution branch.

Delete

Deletes the **Solution** and **Results** cell. Deleting the solution cell makes the system a setup-only system, meaning the system will generate only an input file. It will not solve

or post results. The Solution object and below are removed from the Mechanical application tree.

For CFX systems, you will see the following Solution options, in addition to the common options:

Edit

Launches CFX-Solver Manager.

Import Solution

Displays the most recent CFX-Solver Results files imported (if any) and enables you to browse for such files using the **Open** dialog box, where you can specify the CFX- Solver Results file to load. When the results file is loaded, the system will display only the **Solution** cell and the **Results** cell.

Display Monitors

Opens the ANSYS CFX-Solver Manager and shows the results of the previous run.

For Fluent systems, you will see the following Solution options, in addition to the common options:

Edit

Launches ANSYS Fluent.

Import Final Data

Enables you to select an existing Fluent data set (for example, one solved on an external cluster) into a **Solution** cell in a Fluent system and immediately start post-processing in CFD-Post, without the need to run the minimum of one more solver iteration. This option becomes available after importing case file into the **Setup** cell.

Results:

The **Results** cell indicates the availability and status of the analys is results (commonly referred to as post processing). From the **Results** cell, you cannot share data with any other system.

EXPERIMENT - 01

3D BEAM ANALYSIS WITH DIFFERENT CROSS SECTIONS, SAME MATERIALS FOR SAME LOADS

AIM:

To Determine the Total Deflection and Von Misses Stress of various 3 Dimensional Beam.

SOFTWARE: ANSYS

THEORY:

Consider For different Section as Shown in Figure having Young's Modulus Of Elasticity of 2000000 $N/_{mm^2}$

Calculate and Compare the deflection and Von Mess Stress.



PROCEDURE:

Step I

Create Geometry – Open Ansys Workbench – Static Structural

	New 🚰 Open 🔚 Save 🔣 Save As	Import		≪@ Reconnect 🙈	Refresh Project
oolb	юх 🝷 🕈 🗙	Project Sch	nema	tic	
	Analysis Systems				
2	Design Assessment	- the			21
Q	Electric	-		A	
A.	Explicit Dynamics	1	222	Static Structural	
C	Fluid Flow - Blow Molding (Polyflow)	2		Engineering Data	1
	Fluid Flow-Extrusion (Polyflow)	3	0	Geometry	2.
0	Fluid Flow (CFX)	4		Model	
G	Fluid Flow (Fluent)	-	1	Cables	-
8	Fluid Flow (Polyflow)	5	CE2	setup	5 1
\sim	Harmonic Response	6		Solution	P
	Hydrodynamic Diffraction	7	1	Results	? .
	Hydrodynamic Time Response			Static Structural	
	IC Engine				
2	Linear Buckling				
w	Magnetostatic				
	Modal .				
	Modal (Samcer)				
and a	Random vibration				
mill	Response Spectrum				
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	Rigio Dynamics				
~	Static Structural				
222	Static Structural (Samcer)				
	Steady-State Inermal				

#### Create Select Rectangle Option Create Geometry



Trim Geometry



#### Select Pull Option Extrude 100 mm



#### Step 2 - Updating Material Properties

Right Click Update Geometry-Update Model – By Default Ansys will Provide Steel Property



#### Step 3 -Creating Mesh



#### Step 4 -Creating Boundary Condition Select at least one corner Faces And Apply Fix Support



Methodist College of Engineering & Technology Department of Mechanical Engineering



Select An Edge and apply load of 100 N in Z direction

Step 5 -Solution -Total Deformation -Von Misses stress

- Total Deformation
- Equivalent Stress

## **RESULT & CONCLUSIONS:**

## **VIVA QUESTIONS:**

- Differentiate Between Beam vs Bar ?
- ➢ What is the physical Significance of Young's modulus of Elasticity ?
- ➤ What is the full form of ANSYS?
- ➤ What are the basic steps for Analysis ?
- Differentiate between Meshing and Nodes ?
- ➤ What do you mean by Boundary Conditions?
- > Define Factor of Safety ?
- ▶ How can you validate your design using F.E.A Results ?
- Define Modulus of Rigidity ?
- Solid vs Hollow Shaft which has more load bearing capacity ?

#### **EXPERIMENT - 02**

#### 1D, 2D AND 3D MESHING OF CANTILEVER BEAM

#### AIM:

1D, 2D and 3D meshing with different element sizes for different CAD geometry

#### **SOFTWARE:** ANSYS

## **THEORY:**

Calculate 1D, 2D and 3D Mesh For The Given Below Geometry 1D Mesh







#### 3D Mesh



#### **PROCEDURE:**

*Step 1 1D meshing* Click Static Structural Geometry



#### Dimensional Cantilever Beam Geometry



#### Creating Rectangular Beam Profile



#### **Reducing Rectangular Profile**



#### Connect Line to beam



Step 2 :- Giving Material Properties -Right Click Materials -Right- Click - Model.



Step 3 :- Creating Mesh - Click Mesh - 1 Dimensional Meshing is done.

*Step 2 2D meshing* Click on static structural

#### Methodist College of Engineering & Technology Department of Mechanical Engineering



Right Click Update Geometry

**3D Mesh** 

#### Methodist College of Engineering & Technology Department of Mechanical Engineering

File View Tools Units Extensions	Jo	bs H	elp					
🗋 📴 🛃 🕕 Project								
Import 🖗 Reconnect 🔮 Refresh Project 🥖 Update Project 📕 ACT Start Page								
Toolbox 🔻 🗸	×	Project	Sche	ematic				
Analysis Systems	^							
🕞 Coupled Field Static								
Coupled Field Transient			•	А				
Eigenvalue Buckling			1	🚾 Static Structural				
Electric			2	Engineering Data	1			
Explicit Dynamics			-	G Lighteening botto	· · · ·			
S Fluid Flow - Blow Molding (Polyflo			3	🥪 Geometry	? 🖌			
S Fluid Flow - Extrusion (Polyflow)			4	🍘 Model	2			
G Fluid Flow (CFX)			5	🖄 Setup	-			
G Fluid Flow (Fluent)			-	No	· 4			
Fluid Flow (Polyflow)			6	Solution	° 🖌			
Harmonic Acoustics			7	😥 Results	?			
Harmonic Response				Chabia Chrushural				
Hydrodynamic Diffraction				Stauc Structural				
Response								



3D Geometry





3D Mesh

## **RESULT & CONCLUSIONS:**

#### **VIVA QUESTIONS:**

- > Differentiate between , 1D, 2D , 3D Meshing ?
- > Define Nodes ?
- > Define Convergence ?
- > What are the various quality Creations for meshing ?
- > Is it possible to solve a problem without meshing ?
- > Enumerate geometry Clean up ?
- > Define Auto mesh ?
- Geometry clean-up, Mid-Surface ?
- When to use 3-D Elements, DOFs for Solid Elements, Quality Checks for 3-D Elements ?
- How to do Meshing Without Surface

#### **EXPERIMENT - 03**

#### ANALYSIS OF 2D TRUSSES

#### AIM:

Analysis of Plane Truss to determine member forces, stresses, deflections under static loading.

#### **SOFTWARE:** ANSYS

#### **THEORY:**

Consider the four-bar truss shown in figure. For the given data, find Stress in each element, Reaction forces, Nodal displacement. E = 210 GPa, A = 0.1 m².



#### **PROCEDURE:**

#### Step 1: Workbench Toolbox

Toolbox - Analysis Systems - Static Structural Open Static Structural dialog box will appear

🐧 Unsaved Project - Workbench				
File View Tools Units Extensions	Help			
🎦 New 对 Open 🛃 Save 🔣 Save As	Import	Reconnect	🛃 Refresh Pi	roject 🥖
Toolbox 👻 🕈 🗙	Project Schem	atic		
Analysis Systems				
😸 Design Assessment				
(i) Electric	-	A		
Explicit Dynamics	1 777	Static Structural		
S Fluid Flow - Blow Molding (Polyflow)	2 🦪	Engineering Data	1	
S Fluid Flow-Extrusion (Polyflow)	3 🖌	Geometry	2	
G Fluid Flow (CFX)			-	
S Fluid Flow (Fluent)	7 9	Model	B 🔺	
C Fluid Flow (Polyflow)	5 🌑	l Setup	8	
Marmonic Response	6 📢	Solution	2	
W Hydrodynamic Diffraction	7 🗭	Results	9.	
Hydrodynamic Time Response				
IC Engine		Static Structural		
Dinear Buckling				
(iiii) Magnetostatic				
Modal Modal				
Modal (Samcef)				
Random Vibration				
Response Spectrum				
Rigid Dynamics				
5 Static Structural				
5 Static Structural (Samcef)				
Steady-State Thermal				

In that Engineering Data open by default Structural Steel will be there Close Engineering Data and from dialog box open Geometry

#### Step 2: Create A Geometry

From dialog box open Geometry

🚺 Unsaved Project - Workbench	
File View Tools Units Extensions	Help
New 🚰 Open 🛃 Save 🔣 Save As	👔 Import 🕫 Reconnect 对 Refresh Projec
Toolbox 👻 🕂 🗙	Project Schematic
Analysis Systems	
M Design Assessment	
Electric	▼ A
Explicit Dynamics	1 🚾 Static Structural
🔯 Fluid Flow - Blow Molding (Polyflow)	2 🥏 Engineering Data 🗸 🖌
Fluid Flow-Extrusion (Polyflow)	3 🥪 Geometry 📪 🖌
Eluid Flow (CFX)	4 🧼 Model 😨
Fluid Flow (Polyflow)	5 🎡 Setup 👕
Marmonic Response	6 Solution
Bydrodynamic Diffraction	7 Results
🙀 Hydrodynamic Time Response	
IC Engine	Static Structural
Linear Buckling	
(iii) Magnetostatic	
Modal	
Modal (Samcef)	
Random Vibration	
Response Spectrum	
Rigid Dynamics	
5 Static Structural	
5 Static Structural (Samcef)	
M Steady-State Thermal	

# Select XY Plane and make it Look at Face Set Units as Meter

🐼 A: Static Structural - DesignModeler		- 0 X
File Create Concept Tools View Help		
🖉 🚽 🗒 🚳 🔊 Undo @Redo Select: 🏠 🎝 🕇	3 13 17 18 19 19 19 19 19 19 19 19 19 19 19 19 19	h- h- h- h- # #
XYPlane - 🖈 None - 😕		
Generate 🖤 Share Topology 🔃 Parameters		
Extrude 🚓 Revolve 🐁 Sweep 🚯 Skin/Loft		
📕 Thin/Surface 💊 Blend 👻 💊 Chamfer 🐞 Slice 📗 🔗 Poin	Conversion	
Tree Outline 4 Grap	ics	đ
G → A Static Structural → X VPlane → X ZVPlane → X ZVPlane → \$ VZPlane → \$ 0 Parts, 0 Bodies		ANSYS RIA.5
Sketching Modeling		
Details View 4		
Details of XYPlane		Y
Plane XYPlane		+
Sketches 0 Evport Coordinate System? No		•
Ma	0.000 15.800 30.000 (m) 7.500 22.500	<b>↓</b> → ×
🖉 Ready	1 Plane	Meter 0 0

Sketching - Draw – Line Draw the line on X axis by using Line tool bar complete the diagram Sketching - Dimensions - Horizontal 4m Vertical 3m



Go to Concept - Cross Section - Circular - R0.0364

🕬 A: Static Structura	l - DesignModeler		
File Create Conc	ept Tools View Help	·	
) 🗠 🖬 🔚 🚬 H	nes From Points	ielect: *La Ta-	🖻 🖻 📼   🗧
XYPlane 🖾 Li	nes From Sketches	Ð	
🦪 🥩 Generate	nes From Edges	ters	
Extrude S	olit Edges	Loft	
Thin/Surf 🛹 S	urfaces From Edges	🖿 Slice 🔡 🛷 P	oint 📳 Conversion
Sketching Tool 🦽 St	urfaces From Sketches	7 Gr	aphics
i∰ St	urfaces From Faces		
C	ross Section	Rectangular	
	Dimensions	Circular Tube	
General		Channel Secti	on
I Vertical		I Section	
hength/Distance		L Z Section	
Radius		L Section	
Angle		L Hat Section	
Semi-Automatic		Bectangular T	ube
🚔 Edit		User Integrate	d
	Constraints	💋 User Defined	
	Settings		
Sketching Modeling	0		
Details View		4	
Details of Sketch1			
Sketch	Sketch1		
Sketch Visibility	Show Sketch		
Dimensions: 2			
- H1	4 m		
□ V2	3 m		
Edges: 4	107		
Line	Ln8	~ _ <u> </u>	Aodel View Print Previe
ee Outline	+ Orapi	its.	
A: Static Structural			
H			
XXPlane			
🗄 🖉 1 Cross Section			
Circular1			
📖 🔊 🖓 O Parts, O Bodies			
etching Modeling			
Modeling			
	<b></b>		
Details of Circular1	^		
Sketch Circular1			
Show Constraints? No			
Dimensions: 1			
0.0364			
Edeen 4			0.000
cuyes: i			U.000
Full Circle Cr13			v
Physical Properties: 10			
A 0.00031416 n	1 ⁴	ol View Print Preview	
lxx 7.7399e-009	m^4 V Mod	erview [finic freewew]	

#### Go to Concept - Lines From Sketches

😳 A: Static Structural - DesignModeler					
File Create	Concept Tools View Help				
2 🖬 🖪	🛰 Lines From Points	elect: 🐮 🖏 🕶			
XYPlane	🖽 Lines From Sketches	1			
	Eines From Edges	erc			
	VA 3D Curve				
Extrude	🐃 🐜 Split Edges	Loft			
Thin/Surf	Surfaces From Edges	🐚 Slice 📋 🧇 P			
Tree Outline	🧖 Surfaces From Sketches	4 Gi			
🖃 – 🏑 🏟 🗛 St	© Surfaces From Faces				
⊨⊸≁	Cross Section				
~*-	ZAPlane				
*	YZPlane				
	Circular1				
	Deats 0 Pedies				
	o Farts, o boares				
Sketching Ma	odeling				
		-			
Details View		+			
<ul> <li>Details of Circ</li> <li>Circletab</li> </ul>	cular1	^			
Sketch	Circular1				
Show Constra	aints?   No				
- Dimensions:	1				
Edger: 1	0.0304 III				
Eull Circle	Cr13				
Physical Pron	erties: 10				
	0.0041625 m ²				
Ixx	1.3588e-006 m^4	~ 1			
Ready					

A: Static Structural – XY Plane – Sketch1 – Apply


Line1 – Generate – 1 Part 1 Body – Line Body – Cross Section – Circular 1



#### Step 3: Model

From dialog box Select

Model Model – Update



#### Open Model Geometry – Line Body – Assignment – Structural Steel



#### Mesh – Generate Mesh



#### Select Vertex – First Point – Static Structural – Insert – Fixed Support Second Point – Static Structural – Insert – Fixed Support

🕥 A : Static Structural - Mecha	anical [ANSYS Multiphysics]							- 0 ×
File Edit View Units Too	ıls Help 🛛 🥑 📑 岁	Solve 🔻 🕯	🖌 Show Errors 🏥 👪 🖄	🚸 🖪 🞯 🕶 🌒 🗤 🕫	ksheet <b>i</b> n			
🖤 hr 🥸 📭 🗞 -	ित कि कि कि 💽 🗳 -	S &		12 10 0 8 8	<b>-</b> -			
F Show Vertices 🖓 Wirefr	ame 🛛 📕 Edge Coloring 👻	1-1	· h · h · h · # +	HThicken Annotatio	ns 🖧 Show Mesh 🍃	🙏 🕌 Random Colors 🛛 🐼 A	Innotation Preferences	
Environment 🔍 Inertial 💌 🕯	🔍 Loads 👻 🔍 Supports 👻	Conditio	ons 🔹 🔍 Direct FE 👻 📑					
Outline								ANCVO
Filter: Name 🔻	बि 🖉 🗄	A: Sta	tic Structural					AND TO R14.5
Project	- in the second	Time:	structural 1. s					
🖻 🙆 Model (A4)		04-07-	2019 11:11					
Line Body		Ein	red Sunnort					
E Coordinate Syste	ms		ica capport					
Mesh								
Analysis Se	Insert	•	Contraction					
Fixed Supp	🚽 Solve		Standard Earth Gravity					
Solution (	Clear Generated Data		Solutional Velocity					
	allo Rename		Q. Force					v
	Const Caluer Film Direct		🥰 Remote Force					· · · · · · · · · · · · · · · · · · ·
	Upen Solver Files Directo	ory	Moment					I
			içi Joint Load					<u> </u>
			Fixed Support	0.	000	2.500	5.000 (m)	z <b>z x</b>
Details of "Static Structural (A5	)" 7		Displacement		1.250	3.750		
Definition	Structural	Course	Green Constants					
Analysis Type	Static Structural	Kueome	<ul> <li>Fixed Rotation</li> </ul>					
Solver Target	Mechanical APDL	Graph	Flastic Support				bular Data	
Options			~	-		1.		
Environment Temperature	22. *C		Constraint Equation					
Generate Input Only	No	1	🔗 Motion Loads					
			Commands					
		Mess	ages Graph					
Press E1 for Help			<u> </u>	O No Merrager	1 Vertex Selectory I	ocation = (0, 3, 0) m	Metric (m. kg. N. e. V.	A) Degrees rad/s Celsius
rearrieriep				- no messages	r wenter Selected. L	location = (0, 3, 0.) III	meane (m, kg, N, S, V, J	A) Degrees nau/s Celsius

Third Point - Static Structural - Insert - Fixed Rotation

🔯 A : Static Structural - Mee	hanical [ANSYS Multiphysics]								3 <del></del>	ð ×
File Edit View Units T	ools Help 🛛 🥑 📑 💅	Solve 🔻 ?/	Show Errors 🏥 👪 🖉 🐠	\Lambda 👩 🗸 🌒 Work	sheet i					
🗑 ha 👯 💽 • 🖏 •	· 🖻 🖻 🖻 📽 •	S 🕂 🤅	l 🕀 🔍 🔍 🔍 🗶 🗶	12 🗃 🗃 🗞						
📙 🔎 Show Vertices  👸 Win	eframe   📕 Edge Coloring 👻	1-1-	1- 1- 1- × -	Thicken Annotation	s 🖞 Show Mesh	🔛 Random Colors	Annotation Pre	ferences		
Environment 🔍 Inertial 🕶	@ _k Loads ▼ @ _k Supports ▼	🔍 Condition	is 🔹 🔍 Direct FE 💌 👔							
Outline								140 V		ANSYS
Filter: Name 👻	Ø 🕢 🛨	A: Static Static Str	: Structural ructural							R14.5
Project  Model (A4)  Model (A4)  Corollars Systems  Corollars Systems  Model A4  Mode	/ stems	Time: 1. 04-07-20 A Fixed B Fixed	s 11911:12 d Support d Support 2							
Static Struct     Analysis     Fixed Sup     Fixed Sup     Fixed Sup     Fixed Sup	Setting Insert		<ul> <li>Control Acceleration</li> <li>Standard Earth Gravity</li> <li>Rotational Velocity</li> </ul>							
⊑ y⊴ souto	Ation II all Rename	ectory	<ul> <li>♀, Force</li> <li>♀, Remote Force</li> <li>♀, Moment</li> <li>♀ Joint Load</li> <li>♀ Fixed Support</li> </ul>	_	20	1570	1 000			× •
Details of "Static Structural (A	45)" <del>4</del>		Displacement	0.0	0.750	1.500	250			
Definition			🔍 Remote Displacement		unse					
Physics Type	Structural	Geometr	Y. 🛃 Simply Supported	_						
Analysis lype	Static Structural Mechanical APDI	Graph	Sixed Rotation				7 Tabular Data			<b></b>
<ul> <li>Options</li> </ul>	incontinuar ar de		🥰 Elastic Support			1.				
Environment Temperatu	ire 22. °C		🔍 Constraint Equation							
Generate Input Only	No	1	🔐 Motion Loads							
		Messag	Commands	_						
Press F1 for Help				No Messages	1 Vertex Selected: L	ocation = (4., 0., 0.) m		Metric (m, kg, N, s, V, A)	Degrees rad/s C	elsius //

Fourth Point – Static Structural – Insert – Force

Components - Y Component - (-

#### 2500 N) -ve sign is for down ward direction

Third Point – Static Structural – Insert – Force

Components – X Component – (

2000 N)

🕅 A : Static Structural - Mecha	nical (ANSYS Multiphysics	]				- 0 X
File Edit View Units Too	ls Help	Solve 🔻 ?/Show Errors †	🚯 🐼 📣 🖪 🐼 – 💕 Worksheet	i,		
🕿 🗠 🥸 🕞 - 🏷 - 🗍	n n n n n .					
		1344444				
P Show vertices	ame	• /0 • /1 • /2 • /3 • /X	→ A Inicken Annotations  →	show Mesh 🔆 🔳 Random Co	olors Annotation Preferences	
Environment 🔍 Inertial 🔻 🕯	Loads 🔻 📽 Supports 🔹	Conditions • 🖤 Direct FE	▼   🖹			
			R n			ANSYS
Filter: Name 🔻	🗟 🕢 🕀	A: Static Structural Static Structural				R14.1
Project		Time: 1. s				
💮 🎯 Model (A4)		04-07-2019 11:14				
Geometry		The second				
Coordinate System	ns	A Fixed Support				
Mesh		Fixed Support 2				
Static Structur	al (A5)	Fixed Rotation: U.				
Analysis Set	t insert	D Acceleration				
- Fixed Suppo	r 誟 Solve	Detetionel Vel	Gravity			
Fixed Rotati	Clear Generated Data		Serv			
Solution (/	allo Rename	Sec. Force				Y
Solute		🗣 Remote Force				f
	Open Solver Files Dir	ectory 🚱 Moment				•
		🔷 Joint Load				
		Fixed Support	0,000	1.500	3.000 (m)	
ails of "Static Structural (A5)	•	4 Displacement		0.750	2 250	
Definition		Remote Displa	acement A	0.750		
Physics Type	Structural	Geometi Gimphy Sugar	rted			
Analysis Type	Static Structural	Graph Graph Fixed Potation			4 Tabular Data	
Solver Target	Mechanical APDL	G Flastic Suppor	+		1.	
Environment Tomporation	22 *C	- Liastic Suppor				
Generate Input Only	No.	🧐 Constraint Equ	Jation			
series are input only		R Motion Loads				
		Commands				
		Messages Graph				
Edit View Units Tools	Help 🔮 🕂 🏓	olve → 7/Show Errors 🛄 📗 S 💠 € € 🕀 🔍 🔍 🔇	[2]			
Show Vertices Wirefram	e 🛛 📕 Edge Coloring 🔻	10 - 11 - 12 - 13 - 1x -		ow Mesh 🤌 📕 Random Colo	rs 🛞 Annotation Preferences	
ironment 🍳 Inertial 💌 🔍 l	.oads 🔻 🔍 Supports 👻 🕻	Conditions 🔻 🌒 Direct FE 👻				
		a enanter a se				ANSVS
r: Name 🔻	बि 🗟 🕀	A: Static Structural	<u>e</u> a			R14.5
Project		Time: 1. s			/ +	
Model (A4)		04-07-2019 11:21				
Geometry						
Coordinate Systems		Fixed Support				
Mesh		Fixed Support 2				
Static Structural (	(A5)	Fixed Rotation: U.				
Analysis Setting	15	E Force 2: 2000 N				
Fixed Support	2	Porce 2: 2000, N				
Fixed Rotation						
Se Force						Y
Force 2						+
Solution I	nformation					•
			0.000	1.500	3.000 (m	
s of "Static Structural (A5)"	ą			0.750	2 250	
finition						
vsics Type St	ructural	Geometry Print Preview Re	port Preview/			
alysis Type St	atic Structural	Graph			7 Tabular Data	
ver larget M	echanical APDL			1.		
Environment Temperature	, 'C					
nerate input Only	0					
in the second se						
		Muna Cash				
		wiessages Graph				
s F1 for Help			No Messages No Sel	ection	Metric (m, kg, N, s, V, A) D	egrees rad/s Celsius

#### Solution-Insert-Deformation-Total-Solve

🙆 A : Static Structural - Mecha	anical [ANSYS Multiphysics]			- 0 ×
File Edit View Units Too	ols Help 🛛 🥑 💀 💈 Solve	🝷 ? / Show Errors 🏥 腿 🕼 🧄	🖌 🗃 🕶 🖤 Worksheet 🛛 🖡	
P 1 1 1 1 1 1	6 6 6 8 S · S ·	🔆 ତ୍ 🕀 🔍 🔍 🔍 🕲 🔍 💐	\$ 19. 📾 🖶 🗞   🗖 <del>-</del>	
」 戸 Show Vertices  🖓 Wirefr	rame   📕 Edge Coloring 👻 🔏 👻	1. k · k · k · *	🛏 Thicken Annotations 📲 Show Mesh 🏼 🙏 🕌 Random Colors 🔗	Annotation Preferences
Solution 🍕 Deformation 👻	💁 Strain 👻 💁 Stress 👻 💁 Energ	y 🕶 🤷 Damage 🕶 🔍 Probe 🕶 🧕	🗃 Tools 👻 🐘 User Defined Result 📔 🗄 Campbell Diagram 🛛 🍫 Coo	ordinate Systems 💌 💁 Beam Results 👻 👔
Outline				ΔΝςγς
Filter: Name 👻	🕼 🕢 🖽			R14.5
Project     Model (A4)     Mode	ms ral (AS) ttogs ort ort 2 bon			
Force 2				Ý †
Soluti	Insert	Deformation	Ca Total	•
	📁 Solve	Contact Tool	ag Directional	*> X
Details of "Solution (A6)"	🔄 Clear Generated Data	Probe +	0.000 1.500	3.000 (m)
Adaptive Mesh Refinement	allo Rename	Coordinate Systems	0.750 2.250	+
Max Refinement Loops 1.	🔁 Open Solver Files Directory	Ream Results	٧/	
Refinement Deptn 2.     Information	Grap	h Durintedució	<del>4</del> T	fabular Data 무
Status Solv	e Required	beam roor	<u>t.</u>	
		See User Defined Result		
		Commands	]	
		0.	1.	
		Messages Graph		
Press F1 for Help			No Messages No Selection	Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

Solution -- Insert -- Coordinate Systems -- Elemental Triads -- Solve



#### Solution – Insert – Coordinate Systems – Nodal Triads – Solve



# **RESULT& CONCLUSIONS:**

#### **VIVA QUESTIONS:**

- > What do you mean truss?
- ➢ Define Meshing ?
- Differentiate Beam Vs Bar?
- $\blacktriangleright$  What is F.E.A ?
- > Explain the process of F.E.A

## **EXPERIMENT - 04**

## STATIC ANALYSIS OF PLATES WITH A HOLE

## AIM:

Static Analysis of plates with a hole to determine the deformation and Equivalent Stress.

#### **SOFTWARE:** ANSYS

#### **THEORY:**

The objective of this experiment is to demonstrate the basic ANSYS Space Claim procedure to draw and perform simple analysis This problem is a simple dimensional structural problem of a simple bracket as shown in figure .The bracket is made of 10 mm thick steel plate .The material properties of steel are given below:

- Young's Modulus, E = 200 10⁹ Pascal
- **Poisson's Ratio**,  $\gamma = 0.3$
- **Density**,  $\rho$  = 7860 kg/m³



The forces Acting are shown below



## **PROCEDURE:**

Step1: Workbench Tool box Toolbox-Analysis Systems-Static Structural Click on Static Structural Analysis Click on Geometry Space Claim Response Dialogue Box Will Appear :-



#### Step2-Creating Geometry

Click on geometry Dialogue box Space Claim option will open.



Select any Plane Using Pan View

Set Units as mm

Sketch Two Line perpendicular to each with 100mm dimension each as shown in figure.

Methodist College of Engineering & Technology Department of Mechanical Engineering



Draw a circle of 60 mm radius at a distance of 50mm from vertical as shown in figure.



Methodist College of Engineering & Technology Department of Mechanical Engineering



Draw another circle with given dimensions



Methodist College of Engineering & Technology Department of Mechanical Engineering





#### Deleting Unwanted Geometry and line by selecting Delete icon



#### Click on Model Update it and wait for geometry to upload then double click it



#### Click on Mesh



lary Con	dition	:- Fix	x two	o sm	all I	Holes										
🔟 🔛 🖛	Conte	ext							A : Static	Structura	- Mechan	ical [ANS	YS Acad	emic Teach	ing Introducto	ry]
File Hom	e Environi	ment Dis	splav S	Selection	Auto	mation										
Duplicate	× Delete by Q Find te ⁹ ⊈Tree *	My Comput Distribut Cores 2	ter • uted Solve	Solve	Analysis	laned Sele 梁 Coordinate 《 Remote Poi	ction System nt Ins	ि Commar ः Commer र्ते Chart ert	nds 💿 li nt 🖽 S 📑 A	mages * Section Pla Annotatio	ne Uni	ft ts Wor	ksheet	Keyframe Animation	Nags       Wizard       Show Error	Mana Select s 🖬 Unit C Tools
Outline				X	<b>Q Q</b>	ې چې کې		- 🔅 Q	<u>.</u>	Sel	ct 🛰 Mo	de- 😰		R R R	🛱 🖬 🖷 '	Y.Z 💷 📩
Name	▼ Search Or	rtline 🗸 –						• •				11				
☐ 0 Model (U) ☐ √0 Ceo ☐ √0 Ceo ☐ √0 Mati ☐ √0 Mes ☐ √0 Mes ☐ 7 0 Mes ☐	A4) werty erials dridnate Systems the thic Structural () Analysis Sectif () Fixed Support () Fixed Support () Solution (A6	s (A5) ggs ; ; ; ; ; ; ; ; ;			Time: 1. s 25-12-202	2019:25 Support										
Details of "Fixed Si	upport"		····▼ # □	×												
Ecoping Mathod	d Coomator Sal	laction		_									0.0	00	0.05	n
scoping method	Geometry Sel	rection		_									0.00		3.05	
Constant														0	005	
Geometry	2 Faces													0.	025	0.07
Geometry Definition	2 Faces													0.	025	0.07
Geometry Definition Type	2 Faces Fixed Suppor	rt		G	raph ::::::										025	0.07: ▼ 및 □ ×



#### Insert Deformation and Von Mess Stress



- Total Deformation
- Equivalent Stress

# **RESULT & CONCLUSIONS:**

#### **VIVA QUESTIONS:**

- Define Stress Concentration Factor?
- ➤ What is the difference between surface and solid.?
- ➢ How to provide thickness of surface in ANSYS Workbench?
- ➤ What type of meshing is used for Surface Modeling ?
- What type of meshing is used for Solid Modelling ?
- ▶ What are the boundary condition for plate with a hole analysis problem ?
- Differentiate between Tetrahedron ,Quadrilateral ?
- Differentiate between structured Grid Vs Unstructured Gird?
- ➤ What do you mean by mesh Quality ?
- ➤ How to decide Type of meshing ?

## **EXPERIMENT - 05**

## STRESS AND DEFORMATION OF AXI-SYMMETRIC MODEL

#### <u>AIM:</u>

To calculate Stress in case of axi-symmetric body.

## **SOFTWARE:** ANSYS

#### **THEORY:**

- A. Create a 2D axi-symmetric model
- B. Create a 3D sold model (1/4 symmetric)
- C. Create a 3D shell (1/4 symmetric), assume E = 207





#### **PROCEDURE:**

Opening Space Claim Drawing Geometry



#### Trim



#### Draw rectangle



#### Trim







- Total Deformation
- Equivalent Stress

# **RESULT & CONCLUSIONS:**

#### **VIVA QUESTIONS:**

- > Define Axi-symmetric Element?
- > What are different types of loading in case of axi-symmetric loading ?
- > Differentiate between 2D, 3D Analysis?
- > Differentiate between Stress And Pressure ?
- > Differentiate between symmetric and Axi-symmetric?

## **EXPERIMENT - 06**

## STATIC ANALYSIS OF CONNECTING ROD

## AIM:

To do Static analysis of connecting rod.

#### **SOFTWARE:** ANSYS

#### **THEORY:**

To determine the deformation and Equivalent Stress in case of connecting Rod



## **PROCEDURE:**

Opening Space Claim Drawing Geometry

Methodist College of Engineering & Technology Department of Mechanical Engineering

🚾 📄 🖷 🤊 • ሮ - 🗢							A:Static Struct
File Sketch Design Display	Assembly Me	easure Fa	cets Repair	r Prepare		Detail	Sheet Metal
Image: A state of the stat	Rectangle	2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2	``. × ヽ ``. × 応 ♪ × Modify	Dimension	‴ ⁄ ≯ ‴ ≺ ⁄₂ ∥	i j Se	lect Pull Mo Edit
Structure 7-	Click a referen	nce to start o	reating dime	nsion Ctrl+Clic	k two objects	s to creat	e a virtual noint
<ul> <li>✓ Boesign 1*</li> <li>▷ ✓ Ø Sketching Plane1</li> </ul>					— Ø35mm		
Structure Layers Selection Groups Views		150mm					
Options - Dimensions							
Dimension Orientation							
Auto     Aligned     Horizontal     Vertical				∞Ø66	imm		
1st Reference Orientation				Ø721	nm		
<ul> <li>Auto</li> <li>Horizontal</li> <li>Vertical</li> </ul>		<u>.</u>					
Properties	Z ← (Ŷ)						

#### Draw Circle



Methodist College of Engineering & Technology Department of Mechanical Engineering





Pull





Mirror



#### Pull



**Giving Fillet** 



#### Update Geometry in Model

Mesh

nport   «ˈo̞ Reconnect 🖉	] Refresh Project 🚽	Update Project     ACT     ACT	Start Page						
×	÷ ņ x Pro	Ject Schematic							
ACP (PFE) Autodyn	^								
BladeGen		▼ A							
FX .		1 Static Structur	al						
Engineering Data		2 Regineering D	1a						
EnSight (Forte)		2 S Engriceing de							
xternal Data		3 Geometry	× 4						
xternal Model		4 Model	2						
luent (with Fluent Meshir	ing)	5 🎡 Setup	? 🖌						
Forte		6 🍿 Solution	? 🖌						
Geometry		7 🥪 Results	2						
GRANTA MI		Static Struc	-						
lechanical APDL		State Struc	ural						
Aechanical Model									
resn									
	Context				A:S	cacic structurdi - M	conanicăl	minuto Académic	. reaching introduct
File Home	Mesh Di	splay Selection	Automation						
L _& Cut	X Delete My	Computer - 5		Named Selection	E Commands	Images *	$\leftrightarrow$	E #	Tags
Copy	Q Find	Distributed 7	± 🛛 🔺	Coordinate System	Comment	Section Plane	mft		Wizard
Duplicate Daste	E Tree - Con	s 2 Solve	Analysis a	Remote Point	ili Chart	Annotation	Units	Worksheet Keyf	frame Show Err
Outline		Solve	G	In	sert				
Outline		▼ ₽ 🗆 X	000		0 0	@ @ Select	Mode-		
Name	· Search Outline	~	~~~			a a sector			
E Coord									
	ic Structural (A5) Analysis Settings Solution (A6)	nation		ς					
	ic Structural (AS) Analysis Settings Solution (A6) Construction Inform	nation		ς	2				
Details of "Mesh"	ic Structural (A5) Analysis Settings Solution (A6) Construction Solution Inform	nation • ậ 🗆 X		ς	2				
Details of "Mesh"	ic Structural (AS) Analysis Settings Solution (A6)	nation → 및 _ X			2				
Details of "Mesh"	ic Structural (AS) Analysis Settings Solution (A6) Construction Solution Information Solution Information Use Geometry Se	nation → 및 _ × tting			2				
Details of "Mesh"	ic Structural (AS) Analysis Settings Solution (A6) Comparison	v ₽ □ × ting							
Details of "Mesh" Display Style Display Style Details Perference Figure Order	k Structural (AS) Analysis Settings Solution (AS) Columnian (AS) Columnian (AS) Use Geometry Se Mechanical Engram Control	v ₽ □ × ting							
Details of "Mesh" Details of "Mesh" Display Display Style Defaults Prysitis Preference Element Order Element Order	k Structural (AS) Anayas Settings Solution (A6) Solution (A6) Solution Inform Use Geometry Set Mechanical Program Control Profault	+ ↓ □ × tting ed							
Details of "Mesh" Details of "Mesh" Display Display Style Display Style Details Peternee Element Size	k Structural (AS) Anayas Settingo Solution (A6) F Solution (A6) F Solution Infor	+ ₽ □ × ting ed							
Details of "Mesh" Details of "Mesh" Display Display Style Defaults Privics Perference Element Order Element Size String Calify	ic Structural (AS) Analysis Structural (AS) Solution (AG) Solution (AG) Solution Inform Mechanical Program Control Default	+ ↓ □ × tling ed							
Details of "Mesh" Details of "Mesh" Display Style Display Style Details of reference Element Size Size Quality Quality Quality Quality Display Style	ic Structural (AS) Analysis Settings Solution (A6) Solution Inform Use Geometry Set Mechanical Program Control Default	+ ↓ □ × tting ed							
Details of "Mesh" Details of "Mesh" Display Display Style Display Style Detaults Physics Perference Element Size Straing Quality Inflation Hardwared	k Structural (AS) Anayas Settings Solution (A6) Solution (A6) Construction (A6) Solution Inferi Use Geometry Se Mechanical Program Control Default	+ ₽ □ × ting ed							
Details of "Mesh" Display Details of "Mesh" Display Display Style Detaults Physics Preference Element Size String Quality Inflation Advanced Statistics	ic Structural (AS) Analysis Settings Solution (AG) Solution (AG) Control (AG) Solution Inferr Use Geometry Se Mechanical Program Control Default	+ ↓ □ × tting ed			2			0.000	.01
Details of "Mesh" Details of "Mesh" Display Display Style Display Style Details Person Element Size Statistics	ic Structural (A5) Anayas Settings Solution (A6) F Solution (A6) F Solution Information Solution Information Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solution Solut	+ ₽ □ ×						0.000	

#### **Boundary Condition**



Methodist College of Engineering & Technology Department of Mechanical Engineering

	antion	L													
	Ŧ	Context						A : 5	Static Struct	tural - Me	echanical	ANSYS Aca	demic Teacl	hing Introduct	ory]
File	Home	Environmen	t Display	Selection	Auto	mation									
Duplicate	Cut Copy Copy Paste Outline	× Delete My Q Find ✓ ₽Tree ▼ Co	Computer Distributed res 2 Solve	Solve	Analysis	≌Named Sele ¥ Coordinate € Remote Poin	ction 💽 Co System 대교 Co nt ㎡ Ch Insert	ommands omment hart	Images Images Images Images Images Images Images Images	Plane ation	<mark>₩ ft</mark> Units	E2 Worksheet	Keyframe Animation	ॐ Tags ⊘Wizard Show Erro	ors i Tool
Outline 🔅				₽□×	0	ê 😜 🛞	°alo-∢	. 🕘 🕢	0	Select	K Mode	া য		<b>m m m</b>	X.Y.Z
Name		Search Outlin	· · ·		-		• •	~~~~	~ ~		•	100			`
	Model (A4) Geome Materia Coordir Mesh Static A Coordir Mesh Static Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir Coordir	try als <b>Structural (AS</b> nalysis Settings ixed Support orce <b>olution (A6)</b>	•) rmation		Force Time: 1. s 25-12-202 Force Comp	10 21:28 : 100. N poments: 100.,0.	y6,								
Details of	"Force"		Ŧ	₽ □ ×											
Details of □ Scope	"Force"			₽ □ ×											
Details of Scope Scoping	"Force"	Geometry Selec	tion	₽ □ ×							0.0	00		0.1	00
Details of Scope Scoping Geomet	"Force" g Method try	Geometry Selec 1 Face	tion	₽ □ ×							0.0	00	0.050	0.1	00
Details of Scope Geomet Definitio	"Force" g Method try on	Geometry Selec 1 Face	• tion	₽ □ ×				A			0.0 1	00	0.050	0.1	00
Details of Scope Scoping Geomet Definiti Type	"Force" g Method try on	Geometry Select 1 Face Force	Tion	₽ □ ×	raph						0.0	00	0.050	0.1	00
Details of Scope Scoping Geomet Definit Type Define E	"Force" g Method try on By	Geometry Selec 1 Face Force Components	tion	<b>₽ □ ×</b>	raph						0.0 1	00	0.050	0.1	00
Details of Scoping Geomet Definiti Type Define E Applied	"Force" g Method try on By By	Geometry Select 1 Face Force Components Surface Effect	tion	<b># —</b> ×	raph						0.0 I	00	0.050	0.1	00
Details of Scope Scoping Geomet Definitis Type Definitis Coordin	"Force" g Method try on By By By hate System	Geometry Select 1 Face Force Components Surface Effect Global Coordin	₹ tion	<b>4 -</b> ×	raph 09-						0.0 I	00	0.050	01	00
Details of Scope Scoping Geomet Definit Type Define E Applied Coordin	"Force" g Method try on By I By ate System mponent	Geometry Select 1 Face Force Components Surface Effect Global Coordin 100. N (rampe	• tion hate System d)	₽ □ × 	raph 0, -e- -62.5 -						0.0	00	0.050	0.1	00
Details of Scoping Geomet Definiti Type Define E Applied Coordin Score	"Force" g Method try on By By By tate System <b>mponent</b>	Geometry Selec 1 Face Force Components Surface Effect Global Coordir -100. N (ramped) 0. N (ramped)	tion hate System d)	<b>₽ □ ×</b> G	-62.5 -						0.0	00	0.050	0.1	00
Details of Scope Scoping Geomet Definitis Type Define E Applied Coordin V Cor V Cor	"Force" g Method try on By By By By ate System mponent mponent	Geometry Select 1 Face Force Components Surface Effect Global Coordir -100. N (ramped) 0. N (ramped)	₹ tion sate System d)	ф — ×	-62.5 - -100.						0.0	00	0.050	0.1	00

- Total Deformation
- Equivalent stress

# **RESULT & CONCLUSIONS:**

## **VIVA QUESTIONS:**

- ➢ Differentiate Between Solid Meshing and Shell Meshing ?
- Explain the boundary condition in case of connecting rod ?
- ➤ What are different types of forces acting in connecting rod ?
- ➢ Define factor of safety ?
- $\blacktriangleright$  How can FEA be used to optimize the design .

#### **EXPERIMENT - 07**

## STATIC ANALYSIS OF CURVED SHELL DUE TO INTERNAL PRESSURE

## AIM:

To find out deformation and equivalent stress of a curved shell due to internal pressure.

## **SOFTWARE:** ANSYS

#### **THEORY:**

From the following given diagram of pipe calculate the internal pressure in the pipe.



## **PROCEDURE:**

Creating Geometry Trim Unwanted Lines :-Draw Another Rectangle as shown






### Create Construction Line

### **Revolve Along**



Right Click Model Update it will automatically Take geometry as Steel



#### **Right Click Mesh**



### Select Faces Insert Fix Support



Inserting Pressure in Y Direction of 1000 Pa



- Equivalent Stress
- Total Deformation

# **RESULT & CONCLUSIONS:**

## **VIVA QUESTIONS:**

- ➢ What are different types of stress in case of Pipe
- ➤ What is the difference between Longitudinal stress and lateral stress ?
- > What type of meshing is used in the analysis of shell element
- ➢ What do you mean by stiffness matrix
- Define shape function

## **EXPERIMENT - 08**

## MODAL ANALYSIS OF BEAM TO CALCULATE NATURAL FREQUENCY OF BEAM

## AIM:

To determine Natural frequency and modal analysis of beam.

### **SOFTWARE:** ANSYS

### **THEORY:**

Determine the Natural Frequency in case of Beam as shown in figure.



## **PROCEDURE:**

Create Geometry





### Methodist College of Engineering & Technology Department of Mechanical Engineering

### Extrude up to 100mm





## Update Model – Default Structural Steel

ACP (Pre) ^ / / / / / / / / / / / / / / / / / /	A         1       Image: State Observation         2       Propresmy Data         3       Scaemety         4       Model         5       Scaemety         6       Solution         7       Results         Static Structural
• # □ ×   Q Q @ ●	은 ° 🔐 〇 수 Q, Q, Q, Q, Select 🍡 Mode~ 武 臣 한 한 한 🐌 한 한 환 🐨 🤎 💆 Clipboard~ [Empty] 😜 Extend* 👤 Select By- 🖜 Conv



#### Insert Fix Support

Mesh







### Mode Shapes Generated



### Right Click Evaluate All Results

# **RESULT & CONCLUSIONS:**

## **VIVA QUESTIONS:**

- > Define Natural Frequencies ?
- > What is the significance of modal analysis ?
- > What is the Significance of Poisson's Ratio ?
- > Define Rigid Body ?
- > How To Determine the Element size in case of analysis

# **EXPERIMENT - 09**

# STEADY STATE THERMAL ANALYSIS OF CHIMNEY

### AIM:

Steady state heat transfer Analysis Cross section of chimney.

## **SOFTWARE:** ANSYS

## **THEORY:**

Determine The steady heat transfer analysis in case of chimney



# **PROCEDURE:**



Methodist College of Engineering & Technology Department of Mechanical Engineering



### Revolve Pull



## Update Mesh

Copy Q, Find     Duplicate     Duplica	Image: Steedion       Image: Commands       Image: Tags       Tags       Manage Views       Image: The Preview         Image: Steedion       Image: Commands       Image: Tags       Image: T	
Name - Search Outline -	Sort of the set of the	
Project     ☐ Project     ☐ 16del (A4)     ☐4® Geometry     ⊕6® Geometry		ANSYS 2020 R2 ACADEMIC
-vria Intial Remperature -viii Analysis Settings Solution (A6) F Solution Information		
Details of "Mesh" 🔹 🖣 🗆 🗙		
Display		
Display Style Use Geometry Setting		
Defaults		
Physics Preference Mechanical		
Element Order Program Controlled		
Element Size Default	· · · · · · · · · · · · · · · · · · ·	
Sizing		Y
Quality		
Inflation		
Advanced		
Statistics	0.005 0.015	×

### Select face Apply Temperature



- Temperature
- Heat Flux

# **RESULT & CONCLUSIONS:**

# **VIVA QUESTIONS:**

- > Differentiate between Steady state temperature vs Transient temperature >
- > Define heat Flux ?
- > What is the significance of the coefficient of thermal expansion ?
- > Define heat Conduction ?
- > Define Heat Convection ?

## EXPERIMENT - 10 NON-LINEAR ANALYSIS OF CANTILEVER BEAM WITH NON-LINEAR MATERIAL

### AIM:

To determine deformation in case of cantilever beam with nonlinear material

### **SOFTWARE:** ANSYS

### **THEORY:**

From the given dimension below assign non linear material and find out the TOTAL DEFORMATION.



## **PROCEDURE:**



## Trimming Geometry



### Extruding Geometry 100 mm



# Assigning Material

onsaved roject workbenen									
File View Tools Units Extensions Jobs Help									
🗋 💕 🛃 🔣 🕞 Project									
🚮 Import 🛛 🏹 Refresh Project 👙	Update Project ACT Start Page								
Toolbox 🗸 🕈 🖌 Pro	ect Schematic								
Analysis Systems									
🕞 Coupled Field Static									
🙀 Coupled Field Transient	▼ A								
Eigenvalue Buckling	1 🚾 Static Structural								
electric	2 🦪 Engineering Data 🗸								
Explicit Dynamics	3 SG Geometry								
Fluid Flow - Blow Molding (Polyflo									
Fluid Flow - Extrusion (Polyflow)	4 We Model								
Fluid Flow (CFX)	5 🙀 Setup 👕 🧧								
S Fluid Flow (Fluent)	6 🕼 Solution 🔗								
Harmonic Acoustics	7 😭 Results 🧇								
Harmonic Response									
Hvdrodynamic Diffraction	Taminin Response Static Structural								
Response									
IC Engine (Fluent)									
🔘 Magnetostatic									
😳 Modal									
Modal Acoustics									
Mandom Vibration									

Assign Plasticity non-linear to material model

Engineer	ngineering Data Sources 🔹 🝷 📮								
	А	в		С		D			
1	Data Source	1	Lo	catior	1	Description			
2	🚖 Favorites		Quick access list and d items			uick access list and default ems	L		
3	🗱 General Materials				Ge us	eneral use material samples for se in various analyses.			
4	🏭 General Non-linear Materials			R		eneral use material samples for e in non-linear analyses.			
Outline o	Outline of General Non-linear Materials								
	А		в	С	D	E	4		
1	Contents of General Non-linear Materials	3	Ac	ld	Source	Description	=		
2	<ul> <li>Material</li> </ul>								
3	📎 Aluminum Alloy NL		÷	۲	8	General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.			
4	🏷 Concrete NL		÷		8				
5	🏷 Copper Alloy NL		÷		8				
	-								



#### Click Face – Insert Fixed Support



### Apply Force 100 N , X direction



- Total Deformation
- Equivalent Stress

# **RESULT & CONCLUSIONS:**

# **VIVA QUESTIONS:**

- > Differentiate between linear and non linear Material ?
- > Define Yield Point ?
- Define Ultimate tensile strength?
- ➢ Define Plasticity ?
- ➢ Define Elasticity ?

## **EXPERIMENT - 11**

# **COUPLED FIELD ANALYSIS**

# <u>AIM:</u>

To determine total deformation and temperature distribution using coupled field analysis

### **SOFTWARE:** ANSYS

### **THEORY:**

Given below the disc brake as shown in figure , determine the total deformation and temperature Evolved.







Extrude :- 5mm



### Creating Construction line up to 15mm



Creating a hole of 5mm at a distance 15 mm





# Creating Circular Pattern of Hole



### Updating the model in the geometry



#### Double Click Model





Methodist College of Engineering & Technology Department of Mechanical Engineering

Insert Fix Support	ton Automation Quick	aunch 📉 🔽 😈
Cutire     Copy Q Find     Copy Q Find     Copy Q Find     Copy Q Find     Cores 2     Cores 2     Solve	Image: Section         Commands         Demands         Full         Demands         Full         Demands         Lup out         Lup out <thlup out<="" th="">         &lt;</thlup>	
Outline 👻 🖡 🗖 🗙	🔍 Q. 📦 📦 🚱 😘 🔿 * 🔆 Q. Q. Q. Q. Select 🍡 Moder 📰 ਇ ਇ 🛅 🝓 📾 🐄 🖤 🧮 Clipboard * [Empty] 😜 Extend * 🙎 Select By * 🖜 Conv	ert- 🖕
Project     Image: Section Dubline       Image: Project     Image: Section Dubline </th <th>A Stati Structural Free Support 2712-2001218 Free Support</th> <th>ANSYS 2020 R2 ACADEMIC</th>	A Stati Structural Free Support 2712-2001218 Free Support	ANSYS 2020 R2 ACADEMIC
Details of "Fixed Support" 👻 🖣 🗆 🗙		
Scope		
Scoping Method Geometry Selection	0.000 0.020 0.040 (m)	
Geometry 1 Face	0.010 0.030	Z* 🖕 * X
Definition		
Type Fixed Support	Graph Tabular Data	- 1 m x
Suppressed No	1,	T U I

## Select all the hole faces and apply Force 0f $100\ N$

Outline	<b>→</b> ‡ □ ×	- 🔍 🔍 📾 💭 🕾 🔿 🔹 🔿 🚳 🎯 🕲 Select 🕏 Moder 📅 🕞 🕞 📾 🕞 📾 📾 📾 🐨 🖤 🧖 Clipboard - I Empty I 🖉 Extend - 📍 Select By - 🖷 Convert	
Name	Search Outline 🖌 🗸		
Name         Project*                Project*               Project*                 -//- @ Groome               Project*                 -//- @ Groome               -//- @ Groome                 -//- @ Groome                -//- @ Groome                 -//- @ Groome               -// @ Groome                 -//- @ Groome               -/	Search Outline  yr  s s s s s s s s s s s s s s s s s s	A: Static Structural Force t Time: Li 3 27/1-200 12:00 Components: 100, 0, 0, N	ANSYS 2020 R2 ACADEMIC
Details of "Force"	<b>-</b> ↓ □ ×		×
Scope			
Scoping Method	Geometry Selection	0.000 0.025 0.050 (m)	
Geometry	7 Faces	0.013 0.038	•
<ul> <li>Definition</li> </ul>	-		Z
Туре	Force	Graph 👻 🖣 🗖 🗙 Tabular Data	<b>→</b> ‡ 🗆 ×
Define By	components	1. Steps Time [3] V X [N] V X [N] V Z [N]	
Applied By	Surface Effect	100. 1 1 0. = 0. = 0. = 0.	
Coordinate System	100 N (ramped)	2 1 1. 100. 0. 0.	
	0. N (ramped)	37.5 -	
7 Component	0. N (ramped)	0	
Suppressed	No	1.	

- Total Deformation
- Equivalent Stress

### Couple Field Analysis

onsaved Project - Workbench										
File View Tools Units Extensions	Jobs	s Help								
💕 😼 🖾 🚳										
🚹 💕 🛃 🔣 🕞 Project										
📓 Import 🖗 Reconnect 😰 Refresh Project 🍠 Update Project 🚦 #ACT Start Page										
Foolbox 🗸 🖬 🗙	Pr	roject Sche	ematic							
Analysis Systems										
📴 Coupled Field Static				-			_		_	
🕞 Coupled Field Transient		-		Α			•		В	
😢 Eigenvalue Buckling		1	👼 Static Stru				1		Steady-State Thermal	
electric		2	Engineering	g Data 🗸		-	2	1	Model	2,
Explicit Dynamics		3	Geometry	1	-		3		Setun	-
Fluid Flow - Blow Molding (Polyflo			A N L L	¥	4	/	-			• 4
S Fluid Flow - Extrusion (Polyflow)		4	Model	7	4		4		Solution	° 4
Fluid Flow (CFX)		5	🎇 Setup	\$			5	1	Results	? 🖌
Eluid Flow (Polyflow)		6	Golution	\$					Steady-State Thermal	
Harmonic Acoustics		7	🔗 Results	\$					,	
Harmonic Response					-					
Hydrodynamic Diffraction			static St	ructural						

- 8 × Envi Automation
Automation Cut X Delete My Comp Copy Q Find Distrii Paste Tree Cores 2 Outline Solue • 4 🗆 × B: Steady-State Th ANSYS 2020 R2 lel (B3) Time: 1. s 27-12-2020 12:30 Materials Coordinate Temperature: 100. *C on (85) • 4 🗆 × Method Geometry Selection 0.040 (m) 1 Body Exterior Faces Only ply Te • 4 🗆 × 
 Steps
 Time [s]
 Image: Temperature ["C]

 1
 1
 0.
 = 22.

 2
 1
 1.
 100.
 50 22

#### Select Body Insert Temperature

• Temperature – Total Heat Flux

# **RESULT & CONCLUSIONS:**

### **VIVA QUESTIONS:**

- ➤ What do you mean by coupled analysis
- > Define the boundary condition in case of disc brake?
- ➢ Differentiate between steady state and transient heat thermal analysis ?
- $\blacktriangleright$  What is the coefficient of friction between the brake pad and brake disc?
- ➤ Which material is used to make disc brake ?

## **EXPERIMENT - 12**

## **BUCKLING ANALYSIS OF BEAM**

### AIM:

To Study the deflection of a buckling.

• Eigenvalue buckling analysis

### SOFTWARE: ANSYS

### **THEORY:**

Buckling loads are critical loads where certain types of structures become unstable. Each load has an associated buckled mode shape; this is the shape that the structure assumes in a buckled condition. There are two primary means to perform a buckling analysis:

### • Eigenvalue

Eigenvalue buckling analysis predicts the theoretical buckling strength of an ideal elastic structure. It computes the structural eigenvalues for the given system loading and constraints. This is known as classical Euler buckling analysis. Buckling loads for several configurations are readily available from tabulated solutions. However, in real-life, structural imperfections and nonlinearities prevent most real-world structures from reaching their eigenvalue predicted buckling strength; i.e. it over-predicts the expected buckling loads. This method is not recommended for accurate, real-world buckling prediction analysis.



A steel beam with a 10 mm X 10 mm cross section, rigidly constrained at the bottom. The required load to cause buckling, applied at the top-center of the beam, will be calculated.

## **PROCEDURE:**

### **Eigenvalue Buckling Analysis**

### Preprocessing: Defining the Problem

1. Open preprocessor menu /PREP7

### 2. Give example a Title

Utility Menu > File > Change Title ... /title, Eigen-Value Buckling Analysis

### 3. Define Keypoints

Preprocessor > Modeling > Create > Keypoints > In Active CS ... K,#,X,Y

We are going to define 2 Keypoints for this beam as given in the following table:

Keypoints	Coordinates (x,y)
1	(0,0)
2	(0,100)

### 4. Create Lines

 $\label{eq:cond} Preprocessor > Modeling > Create > Lines > Lines > In Active Coord L,1,2$ 

Create a line joining Keypoints 1 and 2

### 5. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use the BEAM3 (Beam 2D elastic) element. This element has 3 degrees of freedom (translation along the X and Y axes, and rotation about the Z axis).

### 6. Define Real Constants

Preprocessor > Real Constants... > Add...

In the 'Real Constants for BEAM3' window, enter the following geometric properties:

- i. Cross-sectional area AREA: 100
- ii. Area moment of inertia IZZ: 833.333
- iii. Total Beam Height HEIGHT: 10

This defines a beam with a height of 10 mm and a width of 10 mm.

### 7. Define Element Material Properties

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for steel:

- i. Young's modulus EX: 200000
- ii. Poisson's Ratio PRXY: 0.3

### 8. Define Mesh Size

Preprocessor > Meshing > Size Cntrls > Manual Size > Lines > All Lines... For this example we will specify an element edge length of 10 mm (10 element divisions along the line).

### 9. Mesh the frame

Preprocessor > Meshing > Mesh > Lines > click 'Pick All' LMESH,ALL

### Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

Solution > Analysis Type > New Analysis > Static ANTYPE,0

### 2. Activate prestress effects

To perform an eigenvalue buckling analysis, prestress effects must be activated.

- You must first ensure that you are looking at the unabridged **solution menu** so that you can select **Analysis Options** in the **Analysis Type** submenu. The last option in the solution menu will either be 'Unabridged menu' (which means you are currently looking at the abridged version) or 'Abriged Menu' (which means you are looking at the unabridged menu). If you are looking at the abridged version.
- Select Solution > Analysis Type > Analysis Options
- In the following window, change the [SSTIF][PSTRES] item to 'Prestress ON', which ensures the stress stiffness matrix is calculated. This is required in eigenvalue buckling analysis.

A Static or Steady-State Analysis	×
Nonlinear Options	
[NLGEOM] Large deform effects	☐ off
[NROPT] Newton-Raphson option	Program chosen
Adaptive descent	ON If necessary
Linear Options	
[LUMPM] Use lumped mass approx?	I No
[EQSLV] Equation solver	Program Chosen
Tolerance/Level -	
- valid for all except Frontal and Sparse Solvers	
Multiplier -	0
- valid only for Precondition CG	
[PRECISION] Single Precision -	☐ off
- valid only for Precondition CG	
[MSAVE] Memory Save -	☐ off
- valid only for Precondition CG	
[PIVCHECK] Pivots Check	⊡ On
- valid only for Frontal, Sparse and PCG Solvers	
[SSTIF][PSTRES]	
Stress stiffness or prestress	Prestress ON
Note: If NLGEOM,ON then set SSTIF,ON.	
[TOFFST] Temperature difference-	0
- between absolute zero and zero of active temp scale	
OK	Cancel Help

#### 3. Apply Constraints

Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Fix Keypoint 1 (i.e. all DOF constrained).

#### 4. Apply Loads

Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

The eignenvalue solver uses a unit force to determine the necessary buckling load. Applying a load other than 1 will scale the answer by a factor of the load. Apply a vertical (FY) point load of -1 N to the top of the beam (keypoint 2).

The applied loads and constraints should now appear as shown in the figure below.
BLENENTS		ANSIG
	×	
igenvalue Buckling Analysis	<u>.</u> Т	

5. Solve the System

Solution > Solve > Current LS SOLVE

#### 6. Exit the Solution processor

Close the solution menu and click **FINISH** at the bottom of the Main Menu. FINISH

Normally at this point you enter the postprocessing phase. However, with a buckling analysis you must re-enter the solution phase and specify the buckling analysis. Be sure to close the solution menu and re-enter it or the buckling analysis may not function properly.

7. Define Analysis Type

Solution > Analysis Type > New Analysis > Eigen Buckling ANTYPE,1

## 8. Specify Buckling Analysis Options

## • Select Solution > Analysis Type > Analysis Options

Complete the window which appears, as shown below. Select 'Block Lanczos' as an extraction method and extract 1 mode. The 'Block Lanczos' method is used for large symmetric eigenvalue problems and uses the sparse matrix solver. The 'Subspace' method could also be used, however it tends to converge slower as it is a more robust solver. In more complex analyses the Block Lanczos method may not be adequate and the Subspace method would have to be used.

Figenvalue Buckling Options	×
[BUCOPT] Buckling Analysis Options	
Method Mode extraction method	
	C Subspace
	Block Lanczos
NMODE No. of modes to extract	1
SHIFT Shift pt for eigenvalue	0
LDMULTE Load multiplier limit	0
- valid only for Block Lanczos	
OK Cance	el Help

9.	Solve the System					
	Solution	>	Solve	>	Current	LS

SOLVE

## 10. Exit the Solution processor

Close the solution menu and click **FINISH** at the bottom of the Main Menu. FINISH

Again it is necessary to exit and re-enter the solution phase. This time, however, is for an expansion pass. An expansion pass is necessary if you want to review the buckled mode shape(s).

## 11. Expand the solution

- Select Solution > Analysis Type > Expansion Pass... and ensure that it is on. You may have to select the 'Unabridged Menu' again to make this option visible.
- Select Solution > Load Step Opts > ExpansionPass > Single Expand > Expand Modes ...
- $\circ$   $\;$  Complete the following window as shown to expand the first mode

Expand Modes			×
[MXPAND] Expand Modes			
NMODE No. of modes to expand		1	
FREQB,FREQE Frequency range		0	0
Elcalc Calculate elem results?		∏ No	
SIGNIF Significant Threshold			
-only valid for SPRS and DDAM		0.001	
ок	Cancel		Help

#### 12. Solve the System

Solution > Solve > Current LS SOLVE

## Postprocessing: Viewing the Results

1. View the Buckling Load

To display the minimum load required to buckle the beam select **General Postproc** > **List Results** > **Detailed Summary**. The value listed under 'TIME/FREQ' is the load (41,123), which is in Newtons for this example. If more than one mode was selected in the steps above, the corresponding loads would be listed here as well. /POST1

SET,LIST

## 2. Display the Mode Shape

- Select General Postproc > Read Results > Last Set to bring up the data for the last mode calculated.
- Select General Postproc > Plot Results > Deformed Shape

# **RESULT & CONCLUSIONS:**

## **VIVA QUESTIONS:**

- ➢ Define Buckling .
- > State the Boundary Condition for buckling .
- > Differentiate Between Linear and non linear buckling .
- Define Shell Element.
- Define Solid Element

# EXPERIMENT - 13 HARMONIC ANALYSIS OF CANTILEVER BEAM

## AIM:

To Study the stress and deflection on the Cantilever Beam by using cyclic load

## SOFTWARE: ANSYS

## **THEORY:**

Conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1- 100 Hz. Modulus of elasticity = 200 GPa, Poisson's ratio = 0.3, Density = 7800 Kg/m³.



## **PROCEDURE:**

#### Step 1: Workbench Toolbox

Toolbox - Analysis Systems – Harmonic Response Open Harmonic Response dialog box will appear



In that Engineering Data open by default Structural Steel will be there Close Engineering Data and from dialog box open Geometry

## Step 2: Create A Geometry

From dialog box open Geometry

WB	Unsaved	Project -	Workbe	nch								
File	e View	Tools	Units	Extensions	Jo	obs ⊦	Help					
۳١	💕 🔒		- Proj	ect								
	Import	∉φ Rec	onnect	🚯 Refresh P	ojec	t 🍠 U	pda	te Pro	oject	ACT Start	Page	
Tool	box			▼ џ	×	Project	: Scł	emat	ic			
	Analysis S	ystems			^							
	Design A	ssessmen	t									
5	Eigenval	ue Bucklin	g				•			Α		
Ø	Electric						1	$\sim$	Harm	onic Response		
	Explicit D	ynamics					2		Engin	eering Data	× .	
œ	Fluid Flo	w (CFX)					3		Coom	otru	2	
Ø	Fluid Flo	w (Fluent)					3		Geom	ieuy	<b>1</b>	
B٩	Harmoni	c Acoustic	s				4	۲	Mode		7 🖌	
$\sim$	Harmoni	cRespons	e				5		Setup	)	? 🖌	
	IC Engin	e (Fluent)					6		Soluti	ion	?	
	IC Engin	e (Forte)					7		Resul	ts	- P .	
00	Magneto	static						~				
" <b>!</b> "	Modal								Harm	onickesponse		
Bø	Modal Ac	coustics										

Select XY Plane - Sketch and make it Look at Face Set Units as Meter



Sketching - Draw – Rectangle Sketching - Dimensions - Horizontal 0.1m Vertical 0.1m

A: Harmonic Response - DesignModeler									
File Create Concept Tools Units View Hel	P								
🛛 🛃 🔚 📫 🗍 Đ Undo @ Redo 🗍 Sel	lect to 1	1 1 1	10-13×	()S 🔆 Q (	e e e e e	2, 📽 🕼 🔺 🌑	• 12		· /· * ¤
XYPlane 🔹 🛧 Sketch1 🔹 🖄									
Generate 🖤 Share Topology 🔀 Parameter	rs								
Extrude Revolve & Sweep & Skin/Lu	oft								
Dim Surface Deland - Chamfer	Stice 11	Boint Baco							
Distance design Distance Til	Land MIDE	Point moto	April A Comercia	heart	N. C. J. D. LL.		dence V	The Manager And	
BladeEditor: Mimport BGD Sal Load BGD	Load NDF	FlowPath	Blade Splitter	Vistal FExport	ExportPoints	StageFluidZone	SectorCut 16	ThroatArea FCAD Import • SPreference	185
] 逐还名言(迅速	•		• 191 st	/ m @					
Sketching Toolboxes	4	Graphics							ą
Draw						1			ANCWC
Modify									ANSIS
Dimensions									2019 R2
⊘ General						1			ACADEMIC
Horizontal						1			
I Vertical						1			
Length/Distance						1			
Radius									
& Angle									
(An and a second	100								
Constraints	•					I			
Settings									
Sketching Modeling									
Details View	4								Y
Details of Sketch1	^								+
Sketch Sketch1		-							•
Sketch Visibility Show Sketch									× 🔾
Show Constraints? No					0.000	0.100		200 (m)	
Dimensions: 2		Rece			0.000	0.64			
H1 0.1 m						0.050	0.150		
0.1 m			4			1			
e coges: +	~	Model View	Print Preview						
Vertical Select first point or 2D Edge for Ver	tical dimen	sion				No Sele	ction	Meter Degree	0 0 /

#### Extrude-Apply-1m-Generate



#### Step 3: Model

From dialog box Select Model Model – Update



Open Model Geometry – Solid – Assignment – Structural Steel



Mesh – Generate Mesh







Select Face - Face - Harmonic Response - Insert - Fixed Support

File	Home	Environ	ment D	isplay	Selection	Auton	ation									Quick Laune	ch ^ E
aplicate	L Cut Copy D Paste Outline	X Delete Q find ⁹ G Tree *	My Compu Distrib Cores 2	ter uted Solve	* Sohre	Analysis	월 Named 놨 Coordin Ø ₉ Remote	Selection ate Systen Point	Commands Comment Chart Insert	Images *     Section Plane     Annotation	mft Units	E2 Worksheet	Keyframe Animation	Nags Wizard Show Errors	Manage Views Selection Information Unit Converter pols	Print Preview Report Preview	Full Screen CReset Layou Layout
tine	110013		₽□×	Q	2	· ·	0 - *	QQ	Q Q Select	K Mode* 🛅	6 6		1. B	Clipboard *	[Empty] 🚱 Extend =	🧕 Select By = 🐞 🕲	Convert - 🖕
	odel (A4) Geome Materia Coordia Mesh United P 小市 P 小市 P 小市 S	) ebry als inate System onic Respo he-Stress,Mc inalysis Setti Solution (An Solution	da ins da ins 191 \$1 Sol )) In Du & Cle	ert ve plicate ar Genera	ted Data	*	a Acceler A Pressu A Force A Remot	ation e Force									2019 F
			A Ret	ame		F2 6	Rotatin	nt Ig Force									Y
			- or	up All Sir	Files Diserte		Fixed S	upport									T
			- Op	en sower	rnes Directo	ety	Displa	Fixed Su	pport		-	1					~
ails of "H Arfinition Hysics Typ	farmonic pe	Respons •	₽ □ ×	5			Remot	9	Insert a Fixed S to prevent a se entity from mo	upport boundary lected geometric wing or deformin	condition or mesh 3-	0.300		0.450	0.600 (m)		z <b>e N</b>
nalysis Ty	pe		tarmonic	Message	5		Elastic	① Pres	s F1 for help.								-
ptions Enviror	get nment Ter	nperature	12. °C		let		Constr	a <del>nt Equat</del> inds	non			Association	1		lime	stamp	
	Innut Onl	ly I	V0														

	onment D	isplay Sele	ection A	tomation								ch AB
Duplicate	My Compo Distrit Cores 2	iter * : iuted Si Solve	olve Anal	Named Sel Coordinat Sis & Remote Po	ection Command e System Comment lint A Chart Insert	i	mft Units W	orksheet Keyfra	Nizard Wizard Non Show Error T	Manage Views Selection Information s I Unit Converter ools	Print Preview 2 ¹⁰ Report Preview Key Assignments	Full Screen CReset Layout
utline	- # 🗆 ×	QQ		B 0 - + 0	Q Q Q Select	Node- 🗊 🖻			P 🛅 Clipboard -	[Empty] 🖗 Extend *	🤶 Select By 🐐 🔞 C	Convert* 🗸
Name     Search       Project*     Image: Constraint of the search       Image: Constraint of the search     Image: Constraint of the search       Image: Constraint of the search     Image: Constraint of the search       Image: Constraint of the search     Image: Constraint of the search	outline 🗣	A: Har monic Harmonic Re Frequency: 1 07-07-2019 1	e Response esponse 1. Hz 12:46 pport									ANSY 2019 R ACADEMI
E 7 Harmonic Res	Insert		• 9	Acceleration								
	S Solve		9	Force								
- Rixed Supp	-		6	Pamote Force								
Solution	U Duplica	ie .		Mamore Force	0							
SUT 2010	🤌 Clear Gr	enerated Data		Woment Fore	Insert a Force loa	ithat						
	ab Rename		F2		distributes a force	vector						Y
	Group A	All Similar Childr	ren	Rotating	K across one or mo	re topologies.		<i></i>				
	- On a f	abore Eiler Direct	9	Fixed Su								
	O Open s	siver riles Direct	soly G	Displace	Press F1 for help.							
to Barris Manager			9	Remote Displa	cement	0.000		0.300		0.600 (m)		z# 🍾
Definition	* 4 L A		0	Elastic Suppor			0.150	0.300	0.450			
Physics Type	Structural		9	Constraint Fou	lation		0.150		0.450			
Analysis Type	Harmonic	Messages		, constraint equ								•
	Mechanic	Text	E	Commands			Ac	sociation		Time	estamo	
Solver Target												
Solver Target Options												
Solver Target Options Environment Temperature	22. °C											

## Select Edge – Edge – Harmonic Response – Insert – Force

Components – Y Component – (- 100 N) -ve sign is for down ward direction

Contraction of the second s					annovin, respons	er - meenwitten presi		inc reacting	moducio					-
file Home	Environment	Display Select	on Auto	mation										~ [] (
uplicate Copy Outline	× Delete My Co Q, Find ✓ Dis tg_Tree + Cores	nputer = 2 tributed 2 Solve =	Analysis	Amed Selection	Commands Comment Comment Chart Insert	Images * <th>the second secon</th> <th>Worksheet</th> <th>Keyframe Animation</th> <th>© Tags ⊘Wizard ⊡ Show Errors To</th> <th>Manage Views Selection Information Information Information Information</th> <th>Print Preview Preport Preview Key Assignments</th> <th>Full Screen CF</th> <th>Manage * Iser Defined * Ieset Layout Iyout</th>	the second secon	Worksheet	Keyframe Animation	© Tags ⊘Wizard ⊡ Show Errors To	Manage Views Selection Information Information Information Information	Print Preview Preport Preview Key Assignments	Full Screen CF	Manage * Iser Defined * Ieset Layout Iyout
line	• 4 D >	000	• • •	0 - * Q Q	Q Q Select	💺 Mode= 🕞 💽			9 VF 👎	Clipboard +	[Empty] 🔀 Extend+	🤶 Select By 🐐 🕲 C	onvert* 🖕	
Project* Model (A4) Southern Second Southern Second S	) try ais nate Systems onic Response (AS re-Stress/Model (Mon nalysis Settings ixed Support orce olution (A6) Solution Informat	A: Har monic B Force Frequency: 1. H 07-07-2019 12: Force: (Rei Componer Componer	esponse 12 17 1) 100., (Imag 15: (Real) 0., 15: (Imag) 0.	) 0. N 100. 0. N 0. ,0. N							1		А 	NSYS 2019 R2 A D E M 1 C
		5											-	
tails of "Force"	- 1 🗆	2			I	0.00	0.50	0.30	2		0.600 (m)		z	×
nils of "Force" rope	→ 및 □ ) Geometry Select	>			I		0.150	0.30	,	0.450	0.600 (m)		2	×
nils of "Force" cope coping Method cometry	← 및 □ ) Geometry Select	Mescanes			l		0.150	0.30	1	0.450	0.600 (m)		2	×.
nils of "Force" cope coping Method eometry efinition	← 및 □ > Geometry Select 1 Edge	Messages			l		0.150	0.30	2	0.450	0.600 (m)	dama	2	×
nils of "Force" rope coping Method eometry efinition pe	← 및 □ ) Geometry Select 1 Edge Force	Messages				- C.O.	0.150	0.300 Association	) n	0.450	0.600 (m)	stamp	2	×
ails of "Force" cope coping Method eometry efinition ipe efine By	- 4 🗆 7 Geometry Select 1 Edge Force Components	Messages					0.150	0.30	) n	0.450	0.600 (m)	stamp	2	•
ails of "Force" cope coping Method eometry efinition ipe efine By oordinate System	Geometry Select 1 Edge Force Components Global Coordin	Messages					0.150	0.30	) n	0.450	0.600 (m)	stamp	2.8	•
ails of "Force" coping Method eometry efinition pe etine By oordinate System X Component	Geometry Select 1 Edge Force Components Global Coordin 0. N	Messages					0.150	0.30	) n	0.450	0.600 (m)	stamp	2	• •
ails of "Force" coping Method econetry befinition ppe befine By coordinate System X Component Component	Q      Geometry Select      Edge  Force Components Global Coordin 0. N  -100. N  -	Messages Text		000000000000000000000000000000000000000			0.150	0.30	) n	0.450	0.500 (m)	stamp	2	• •
ails of "Force" copie coping Method ieometry refinition pe letine By ordinate System X Component X Component Z Component	Geometry Select 1 Edge Force Components Global Coordin 0. N -100. N •	Messages Text					0.150	0.30	) n	0.450	0.600 (m)	stamp	2	•••

Solution-Insert-Deformation-Total-Solve



Solution-Insert-Stress-Normal-Solve



Solution-Insert-Stress-Equivalent-Solve



# **RESULT & CONCLUSIONS:**

## **VIVA QUESTIONS:**

- > Define Harmonic Analysis ?
- > What are differences between boundary value problem and initial value problem?
- ➢ How do you define two-dimensional elements?
- ➤ What is meant by plane stress analysis?
- > Write a displacement function equation for CST element?

## **EXPERIMENT - 14**

## EXPLICIT ANALYSIS OF CAR WITH 100M/S

## AIM:

To determine the deformation in case of crash of a car with bumper at 100m/s

## SOFTWARE: ANSYS

## **THEORY:**

Determine the total deformation in case of car moving with velocity of 100m/s



## **PROCEDURE:**



Methodist College of Engineering & Technology Department of Mechanical Engineering





## Putting Boundary Condition Fixing The Faces of the Wall

• 4 🗆 ×

Details of "Mesh"

Advanced

Details of Mech*
Display
Display Style
Use Geometry Setting
Display Style
Defaults
Element Order
Element Order
Element Size
Default
Station
Guadaty
Endeanced
Element



0.060 (m)



- Total Deformation
- Equivalent Stress

# **RESULT & CONCLUSIONS:**

## **VIVA QUESTIONS:**

- > Differentiate between implicit and explicit analysis ?
- ➢ Define time step ?
- > Differentiate between preprocessor vs postprocessor ?
- > Define Hourglass?
- What are the different post processor available for explicit analysis in ANSYS?

