METHODIST
COLLEGE OF ENGINEERING & TECHNOLOGY
Estd: 2008
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

METHODIST

Estd: 2008

METHODIST

COLLEGE OF ENGINEERING & TECHNOLOGY

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

Abids, Hyderabad, Telangana, 500001

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
Estd: 2008

METHODIST

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 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

- To impart 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.
- To establish industry-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 global issues, 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 . Readandunderstandtheprocedurefrom Lab Manualashowtocarryoutanactivitythoroughly 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	PO
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	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	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	-	-

LIST OF EXPERIMENTS

INDEX

Experiment No	Experiment Name	Date	Page No	Marks				Remarks/ Signature
				P	R	V	T	

Experiment No	Experiment Name	Date	Page No	Marks				Remarks/ Signature
				P	R	V	T	

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:

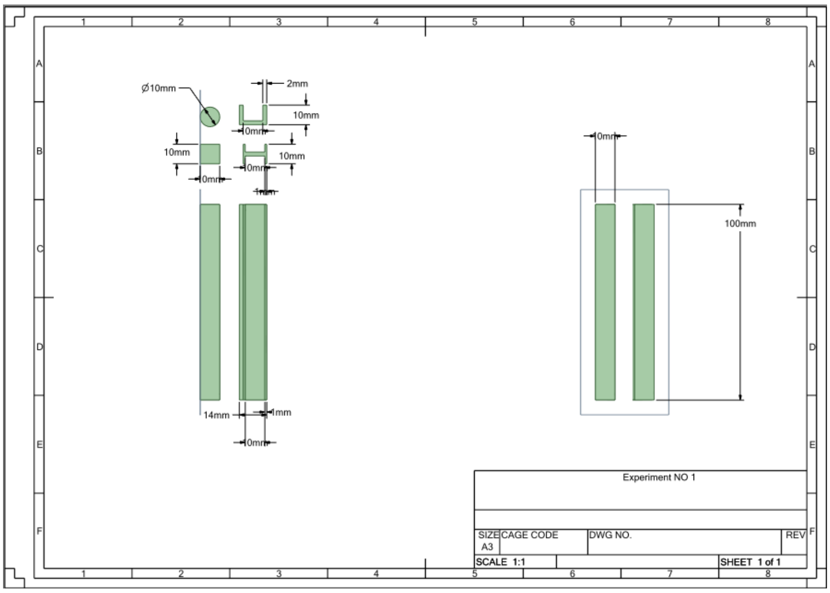
To Determine the Total Deflection and Von Misses Stress of various 3 Dimensional Beam.

SOFTWARE: ANSYS

THEORY:

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

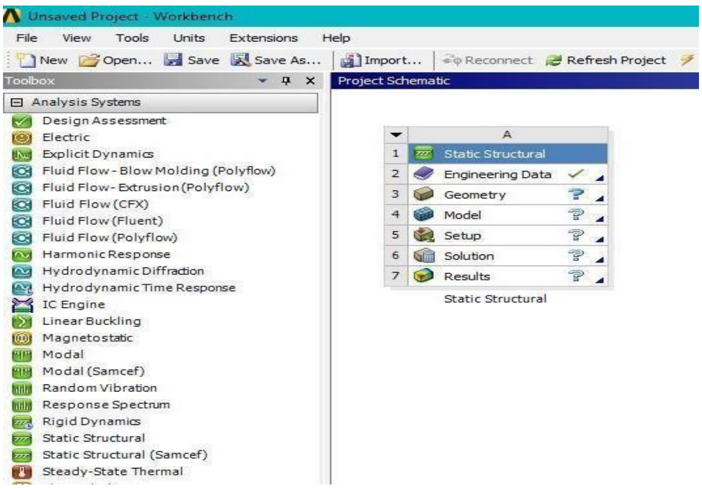
Calculate and Compare the deflection and Von Mess Stress.



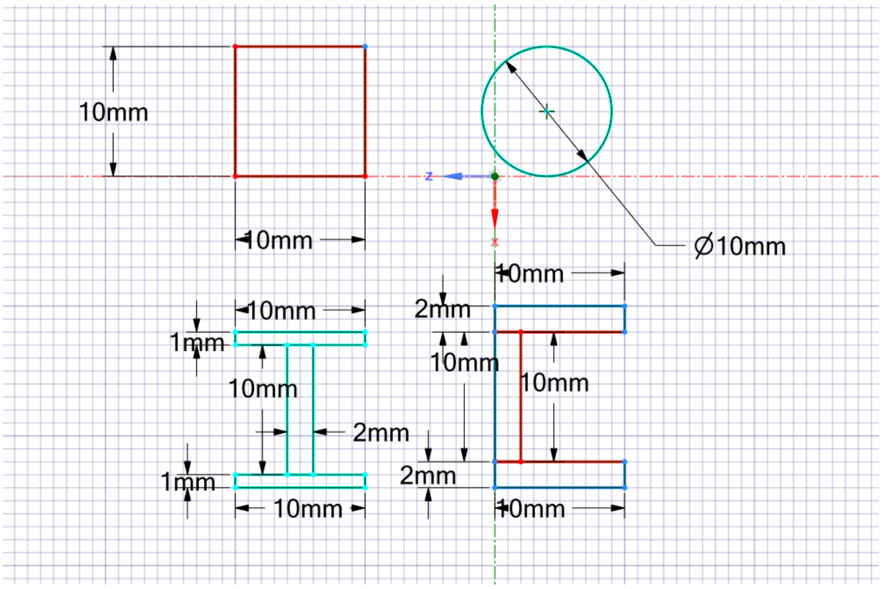
PROCEDURE:

Step I

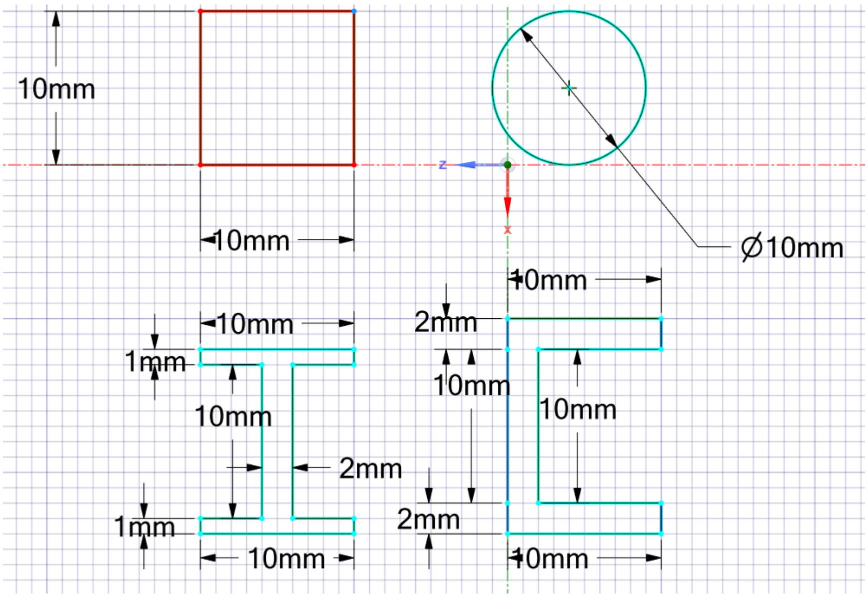
Create Geometry –Open Ansys Workbench –Static Structural



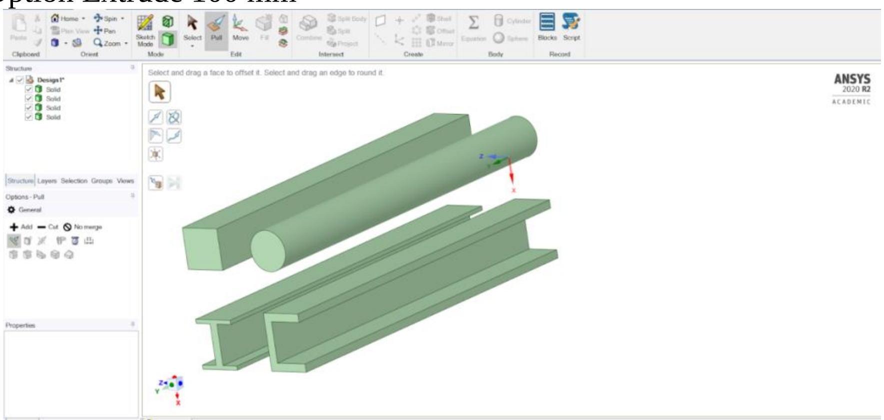
Create Select Rectangle Option Create Geometry



Trim Geometry

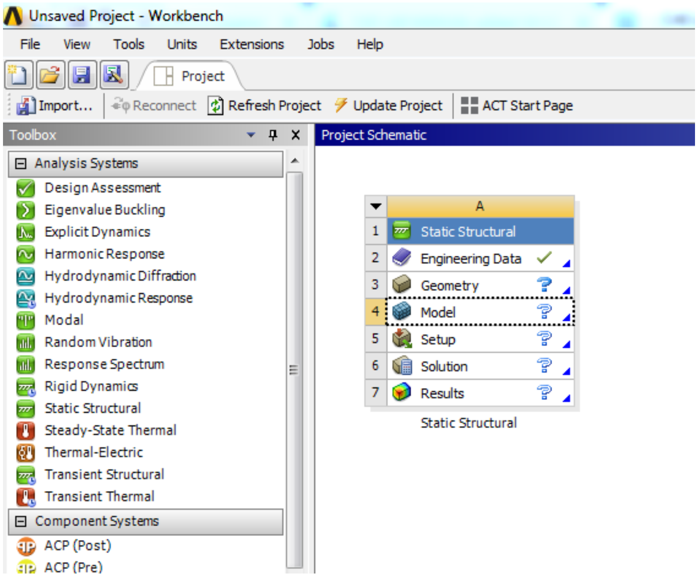


Select Pull Option Extrude 100 mm

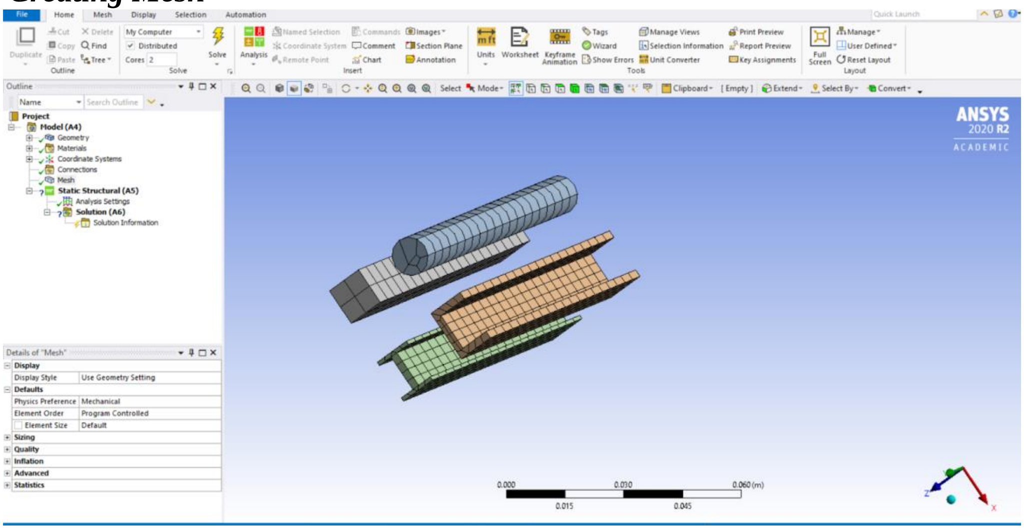


Step 2 -Updating Material Properties

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

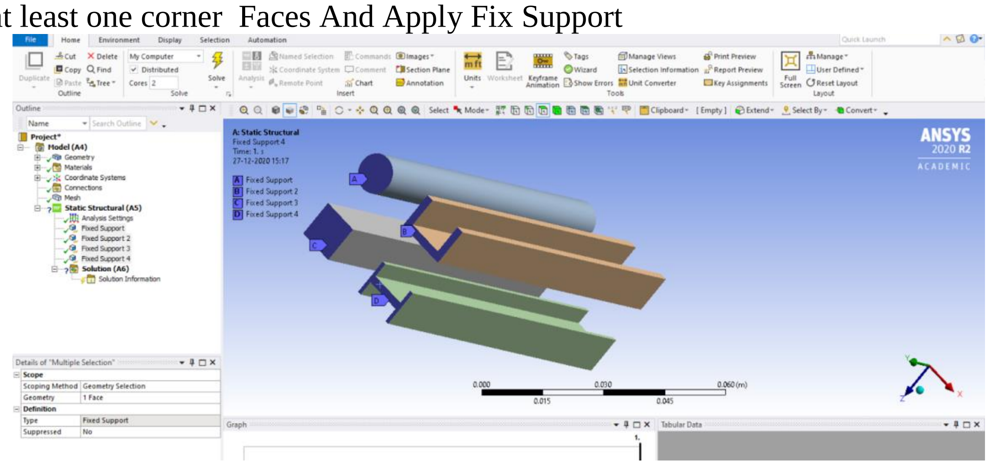


Step 3 -Creating Mesh

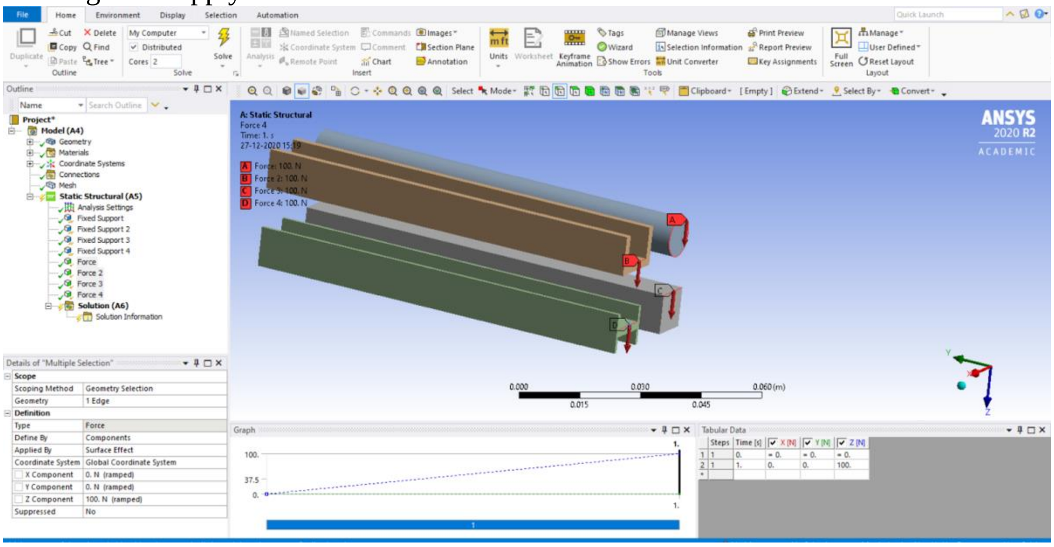


Step 4 -Creating Boundary Condition

Select at least one corner Faces And Apply Fix Support



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:

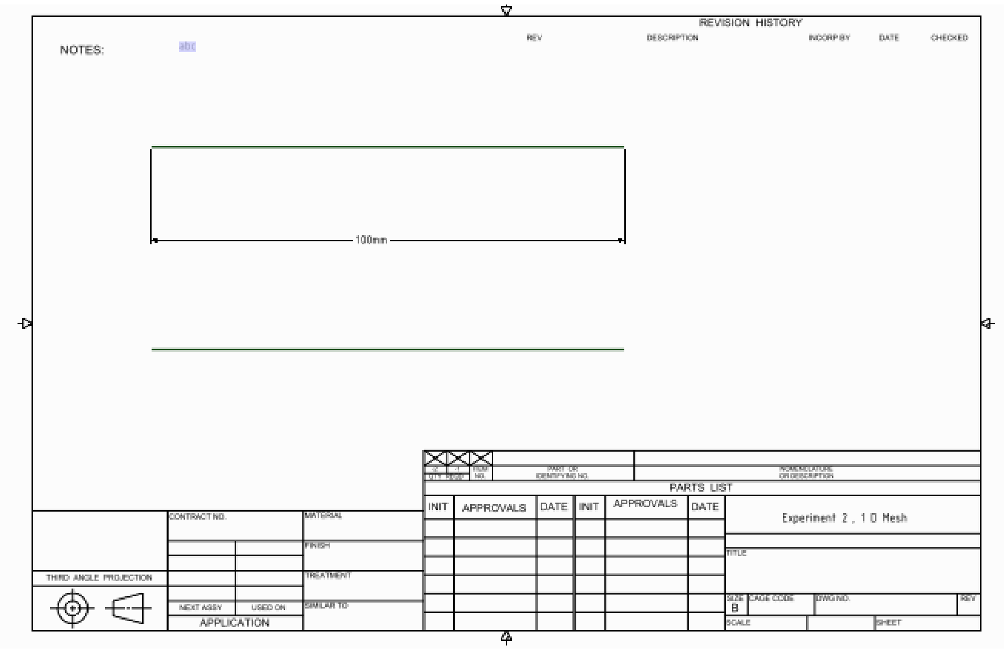
1D, 2D and 3D meshing with different element sizes for different CAD geometry

SOFTWARE: ANSYS

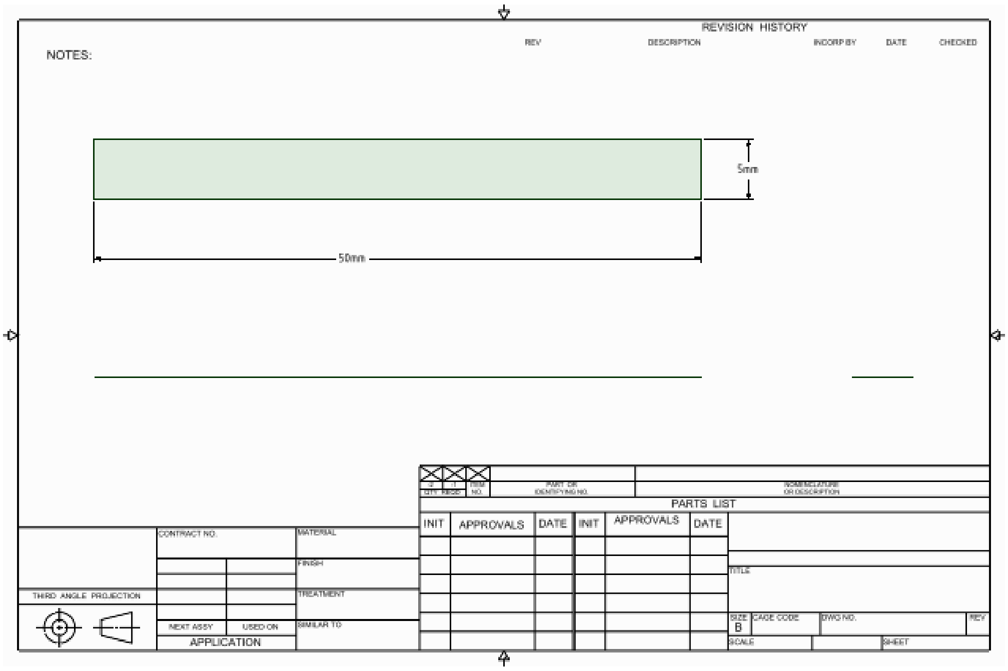
THEORY:

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

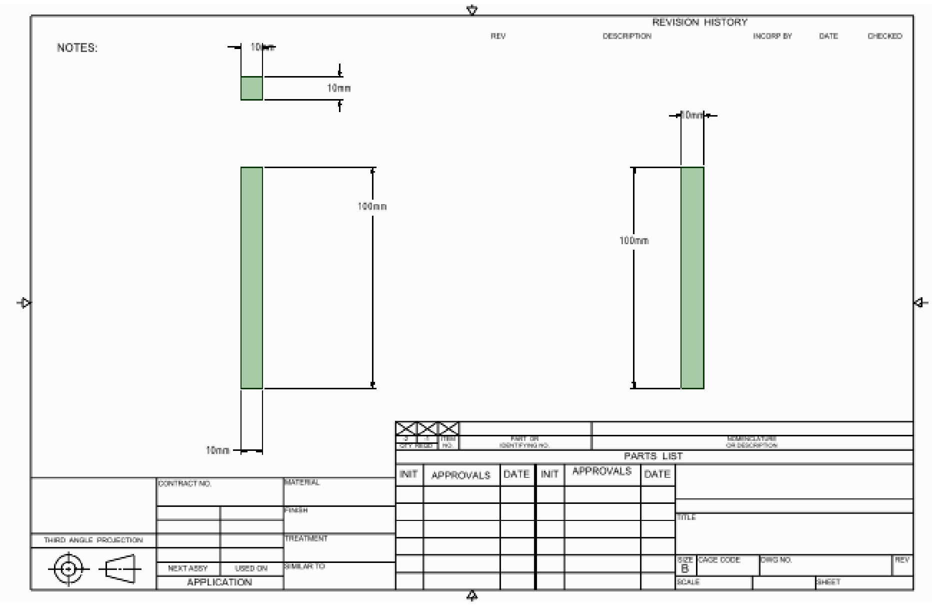
1D Mesh



2D Mesh



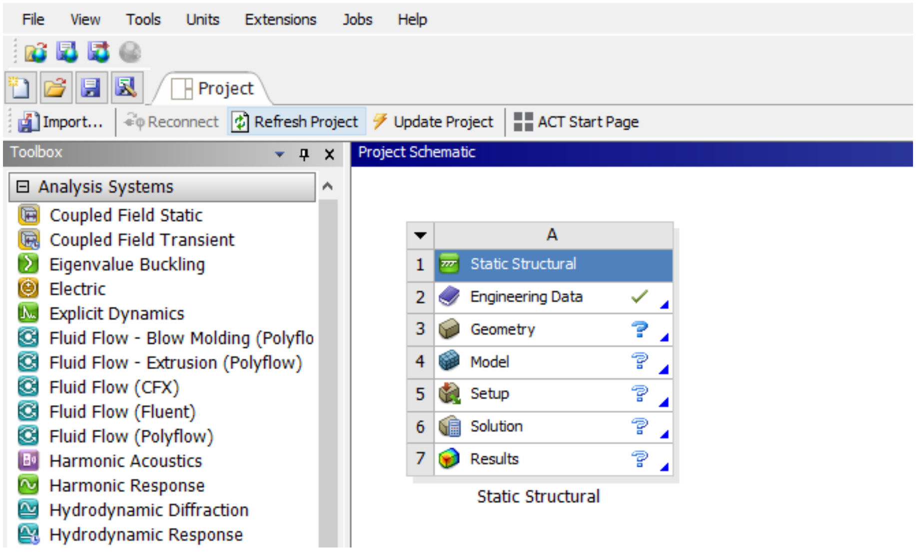
3D Mesh



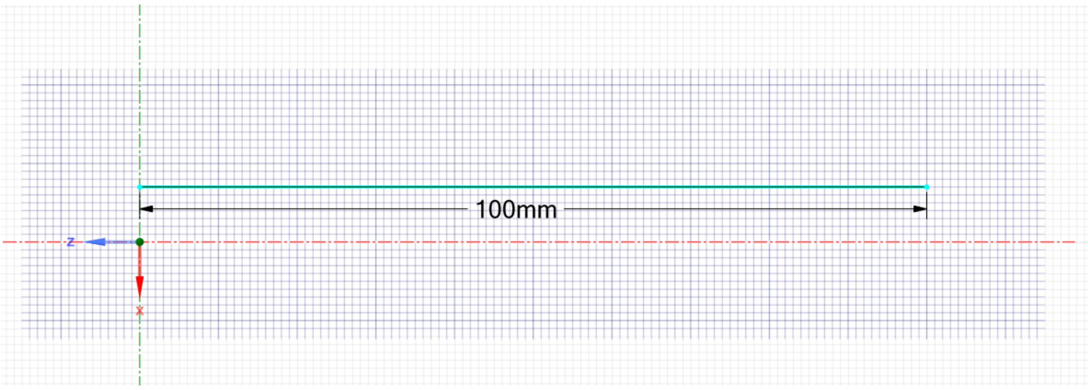
PROCEDURE:

Step 1 1D meshing

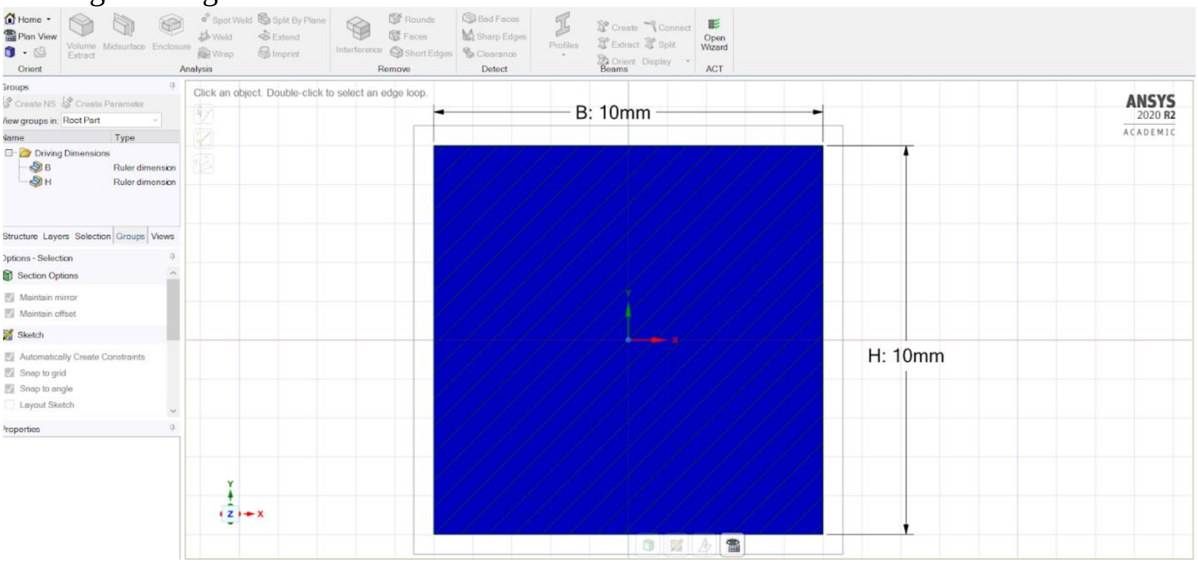
Click Static Structural Geometry



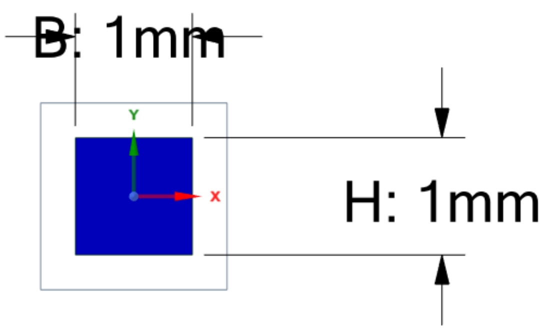
Dimensional Cantilever Beam Geometry



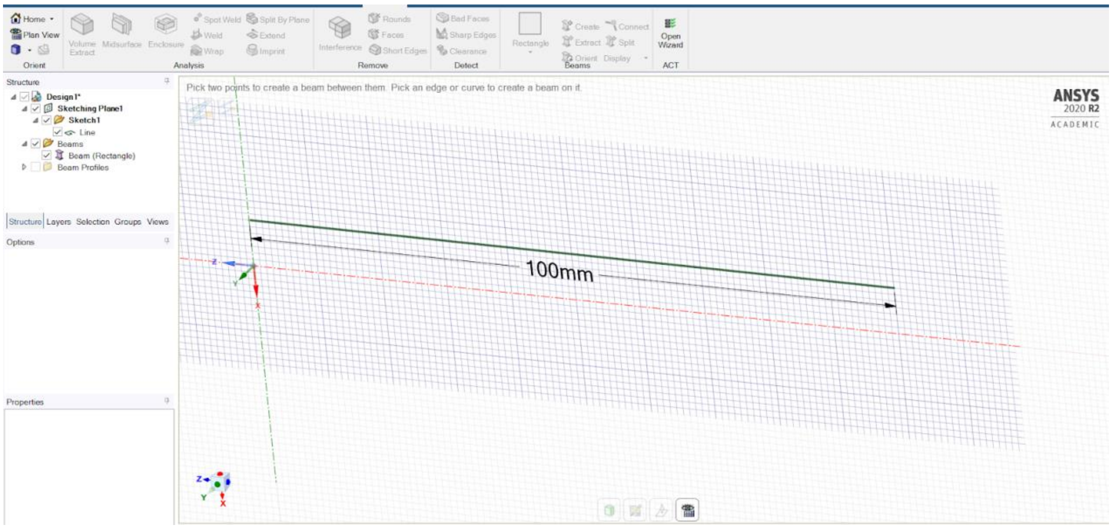
Creating Rectangular Beam Profile



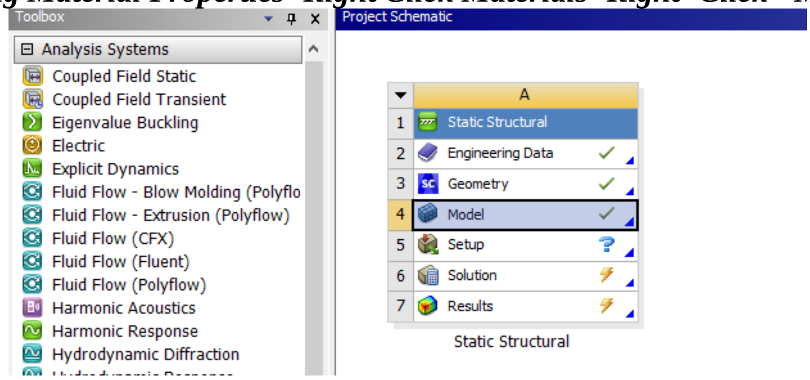
Reducing Rectangular Profile



Connect Line to beam



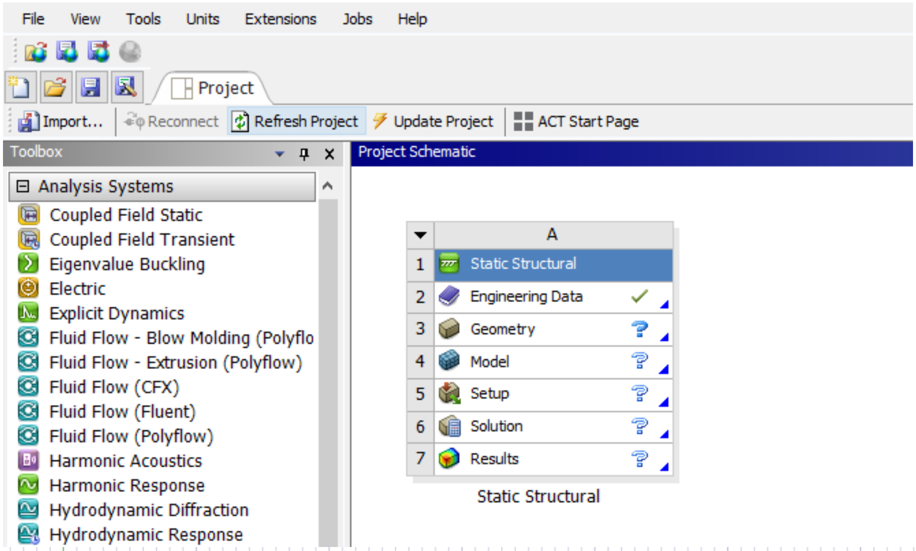
Step 2 :- Giving Material Properties -Right Click Materials -Right- Click - Model.

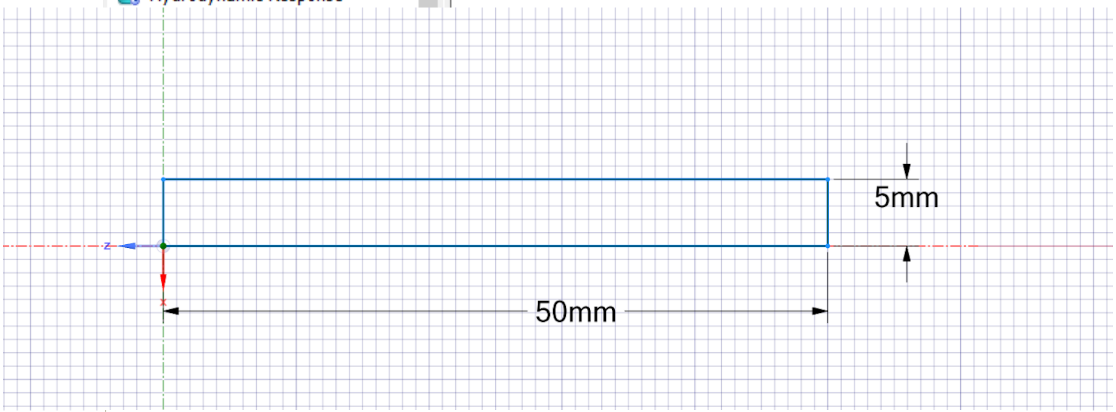


Step 3 :- Creating Mesh - Click Mesh - 1 Dimensional Meshing is done.

Step 2 2D meshing

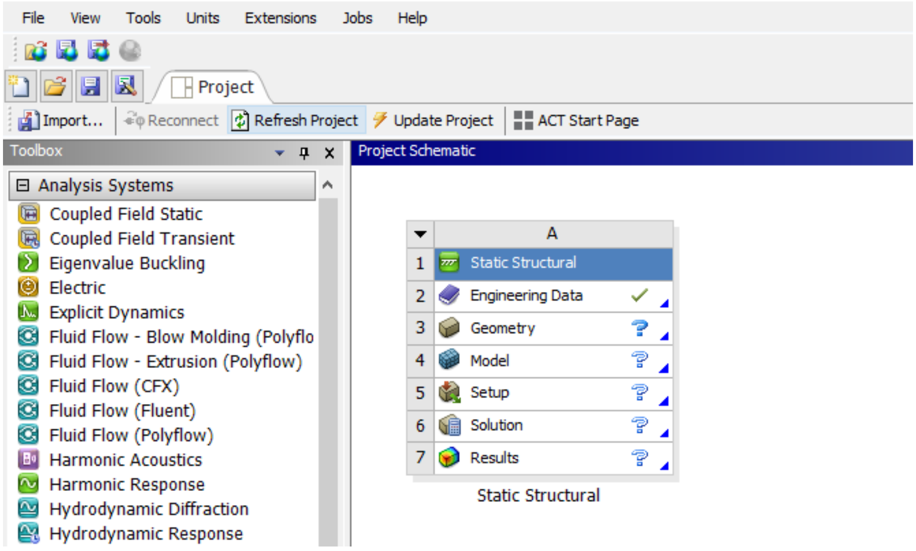
Click on static structural



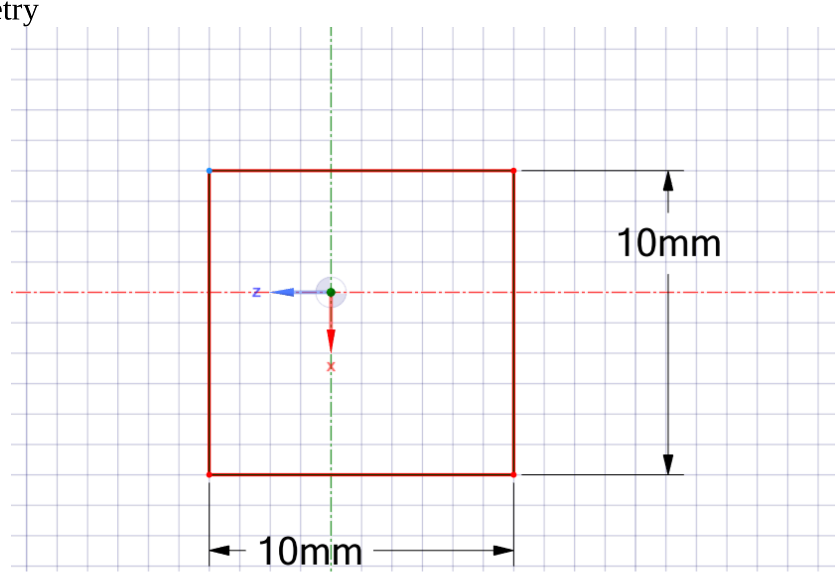


Right Click Update Geometry

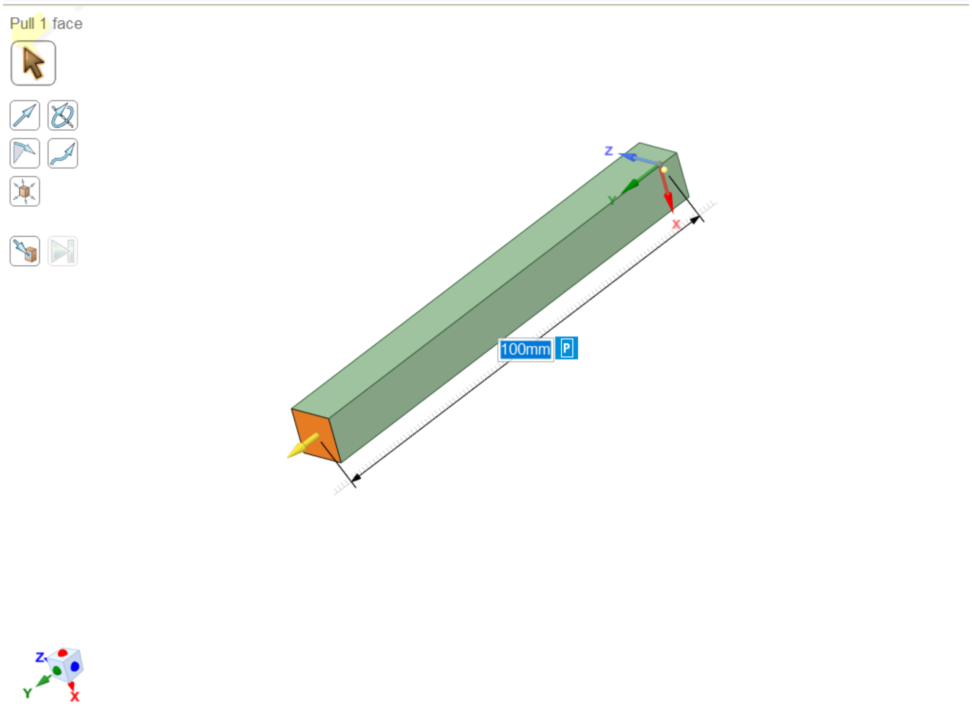
3D Mesh



Create Geometry



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

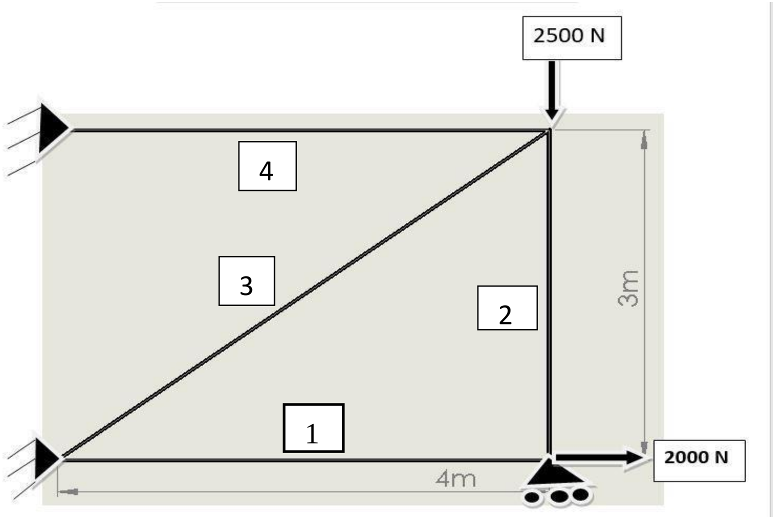
AIM:

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 m^2.

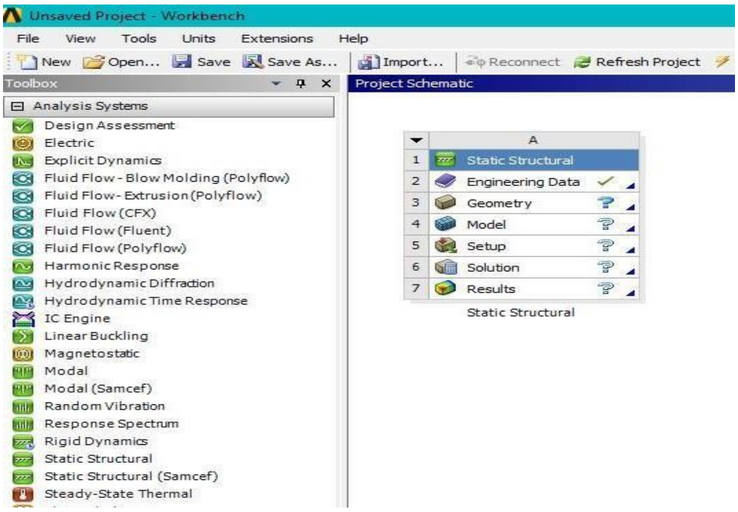


PROCEDURE:

Step 1: Workbench Toolbox

Toolbox - Analysis Systems - Static Structural

Open Static Structural dialog box will appear

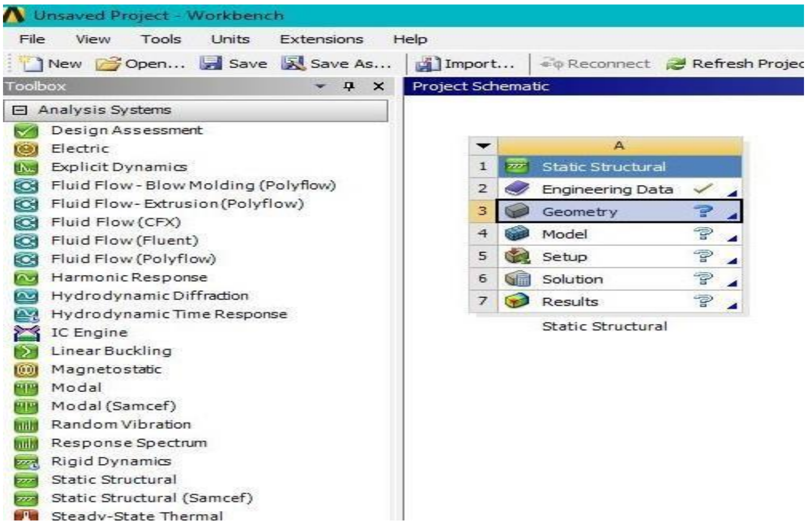


In that Engineering Data open by default Structural Steel will be there

Close Engineering Data and from dialog box open Geometry

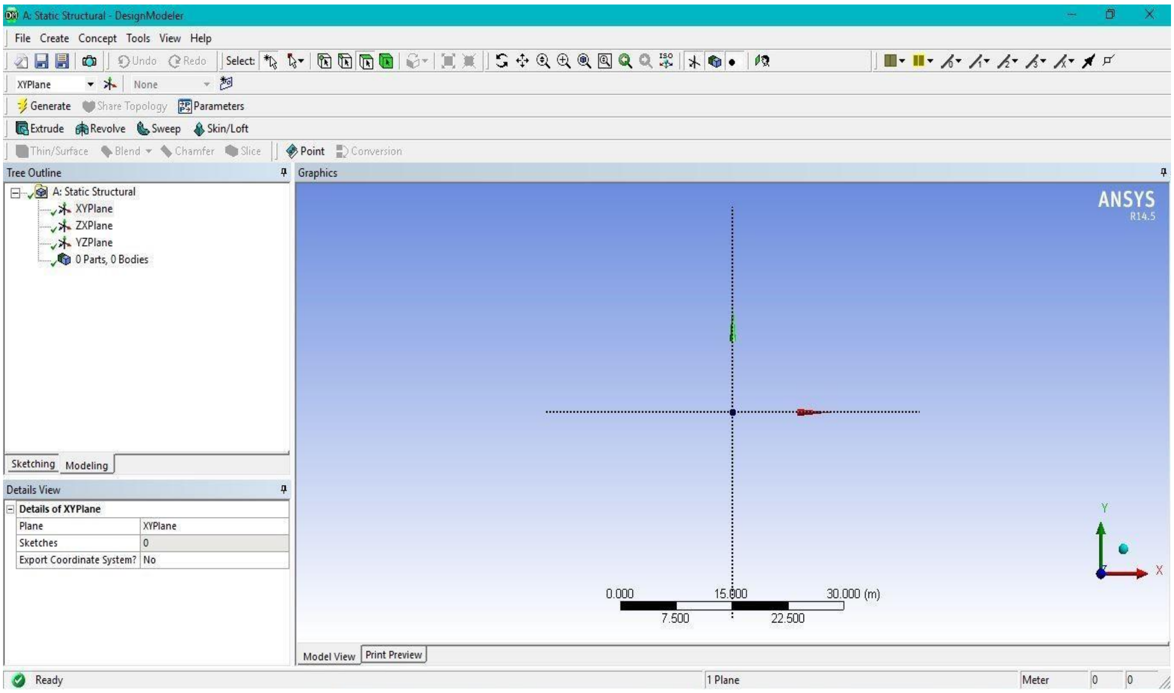
Step 2: Create A Geometry

From dialog box open Geometry



Select XY Plane and make it Look at Face

Set Units as Meter

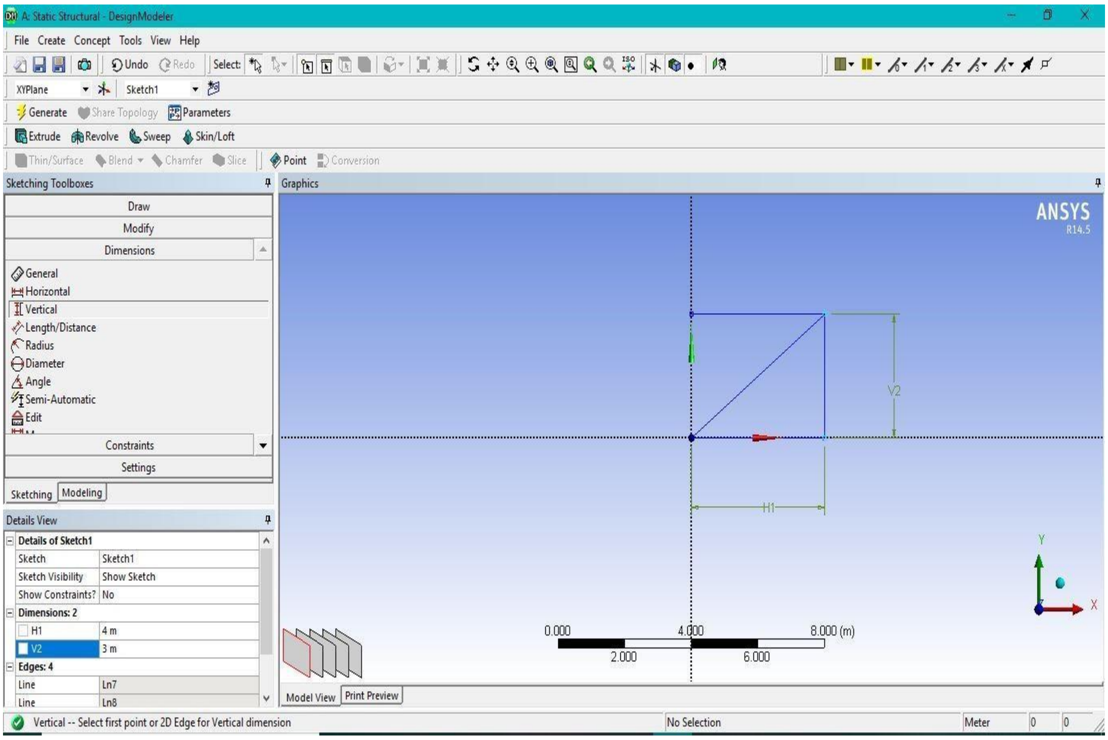


Sketching - Draw – Line

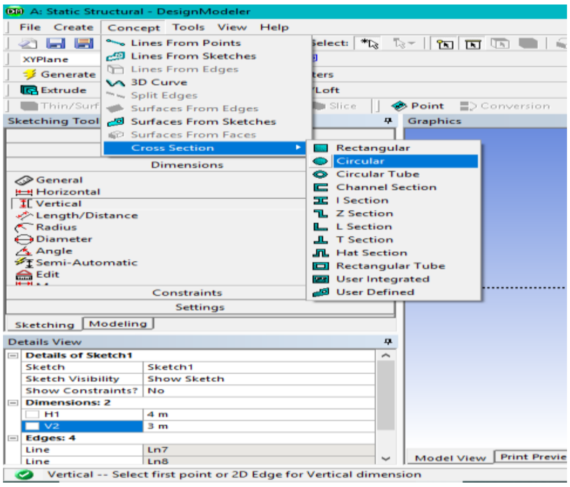
Draw the line on X axis by using Line tool bar complete the diagram

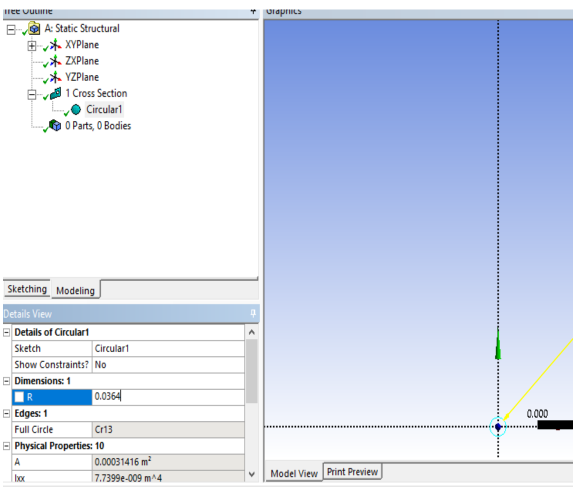
Sketching - Dimensions - Horizontal 4m

Vertical 3m

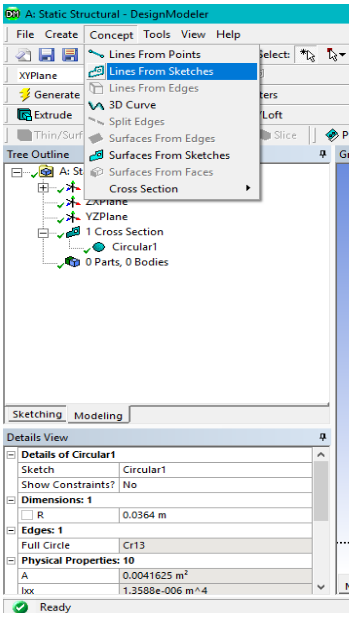


Go to Concept - Cross Section - Circular – R0.0364

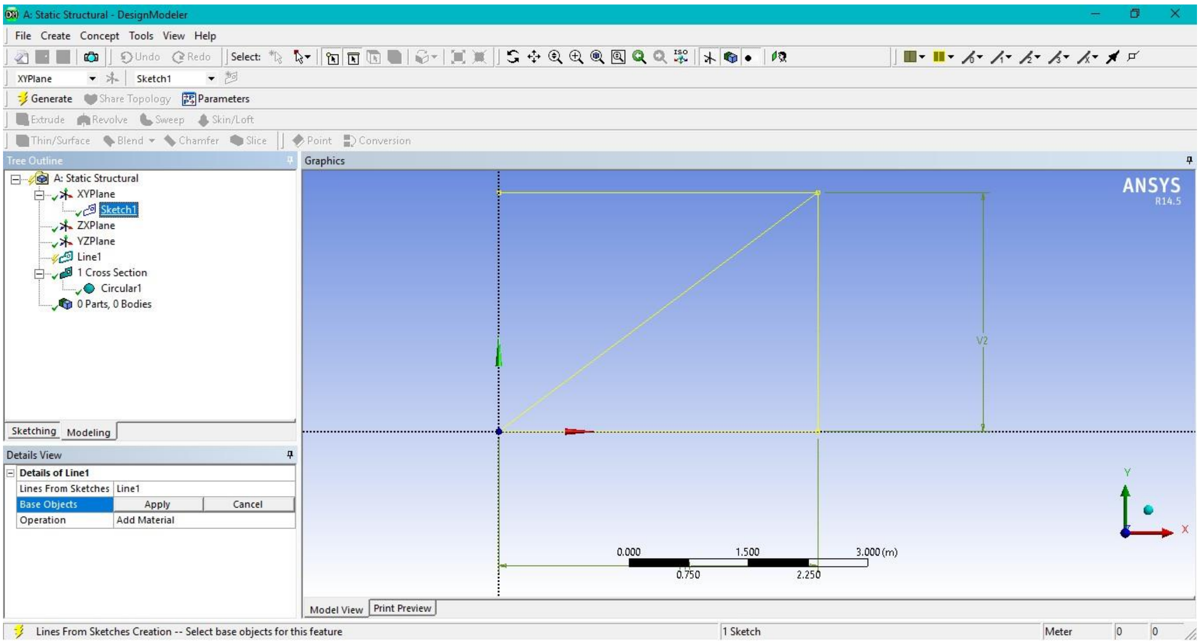




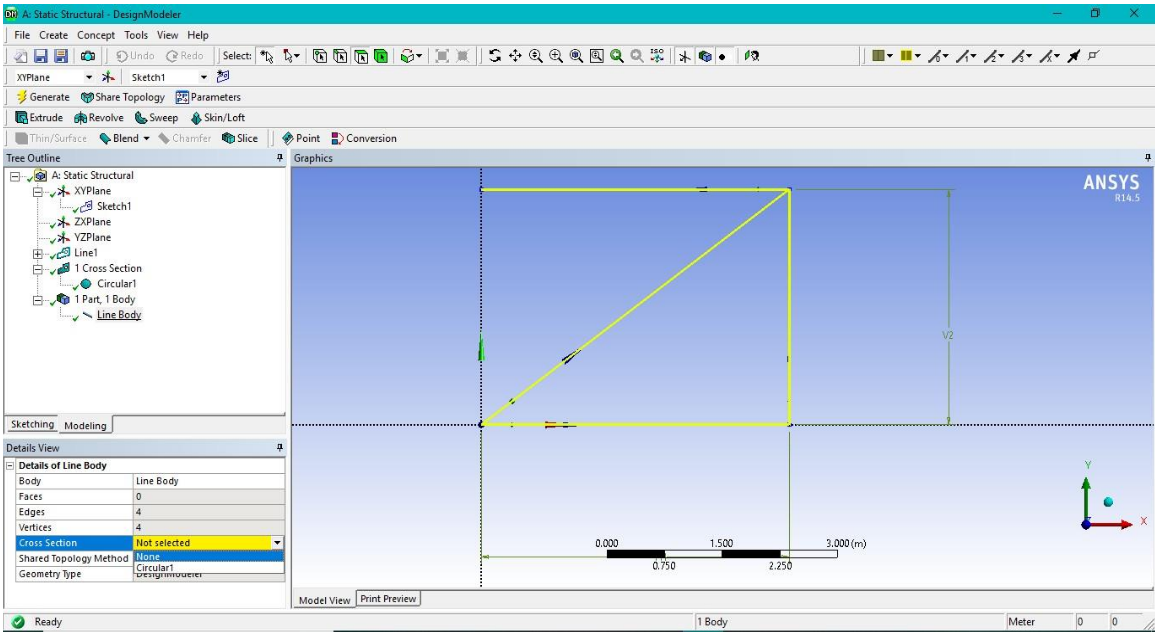
Go to Concept – Lines From Sketches



A: Static Structural – XY Plane – Sketch1 – Apply



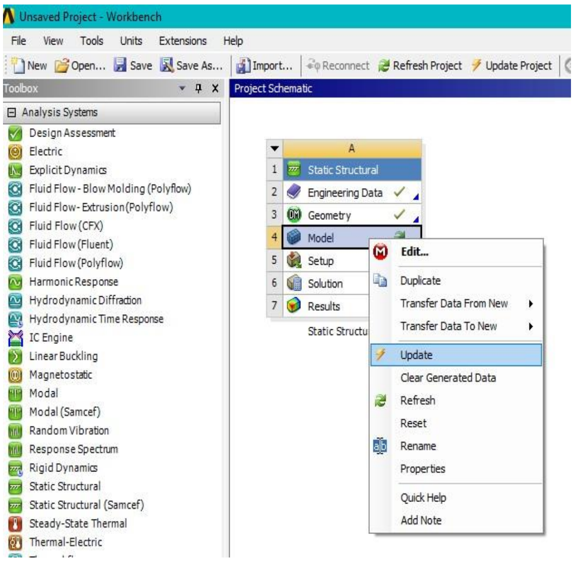
Line1 – Generate – 1 Part 1 Body – Line Body – Cross Section – Circular 1



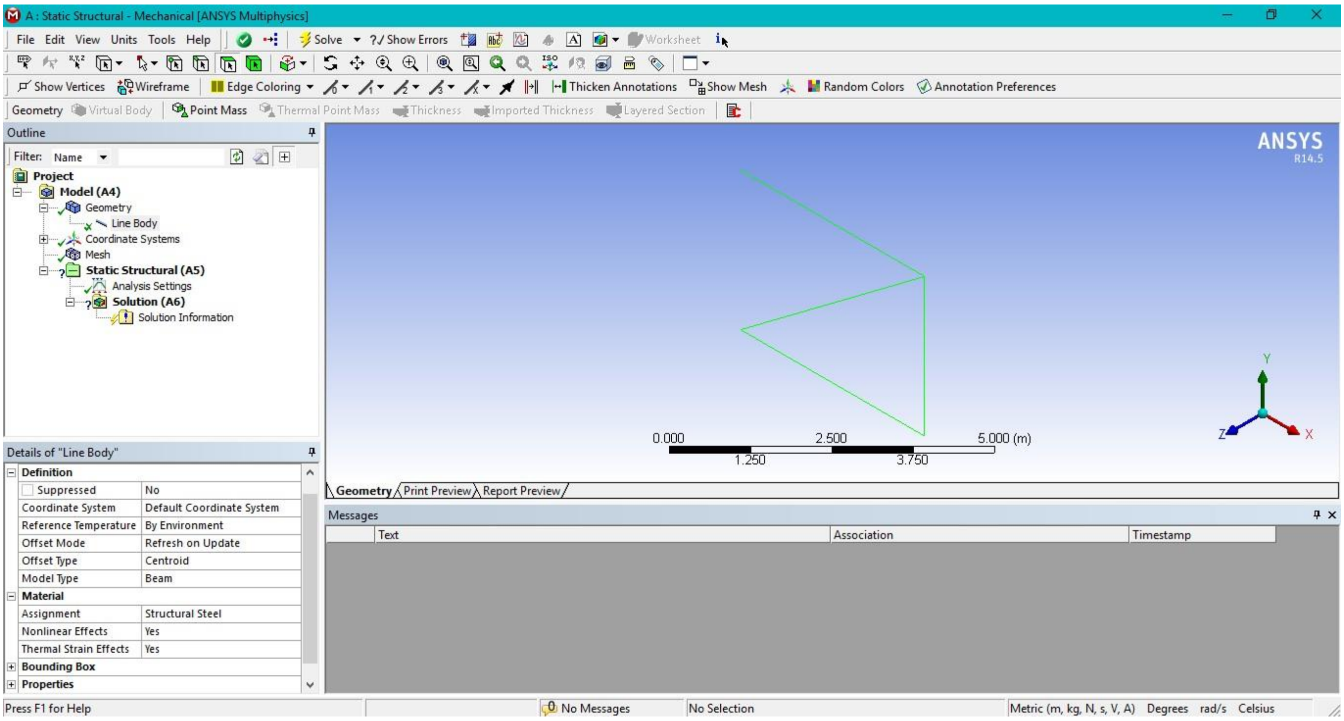
Step 3: Model

From dialog box Select

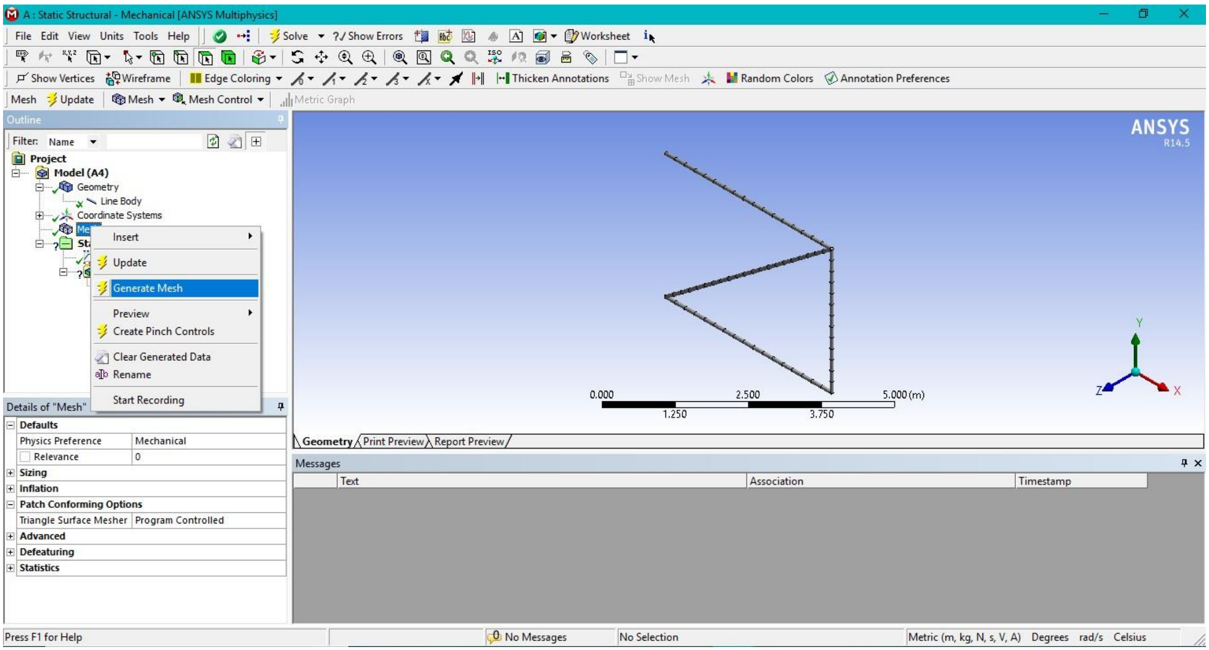
Model Model – Update



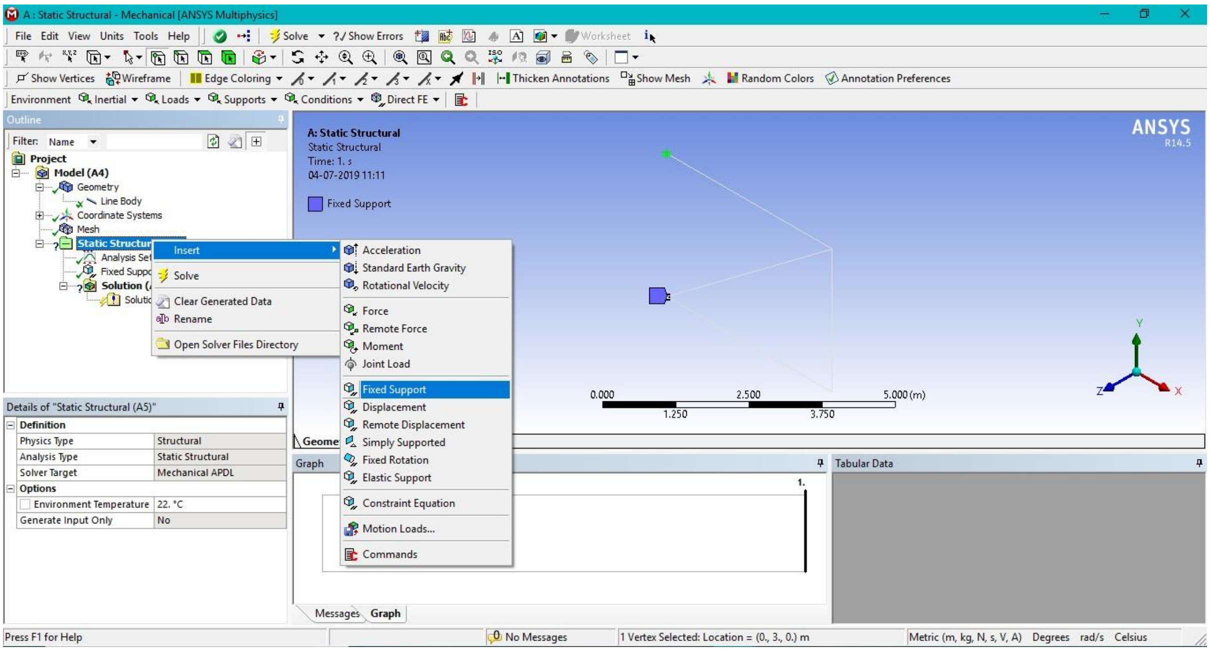
Open Model
Geometry – Line Body – Assignment – Structural Steel



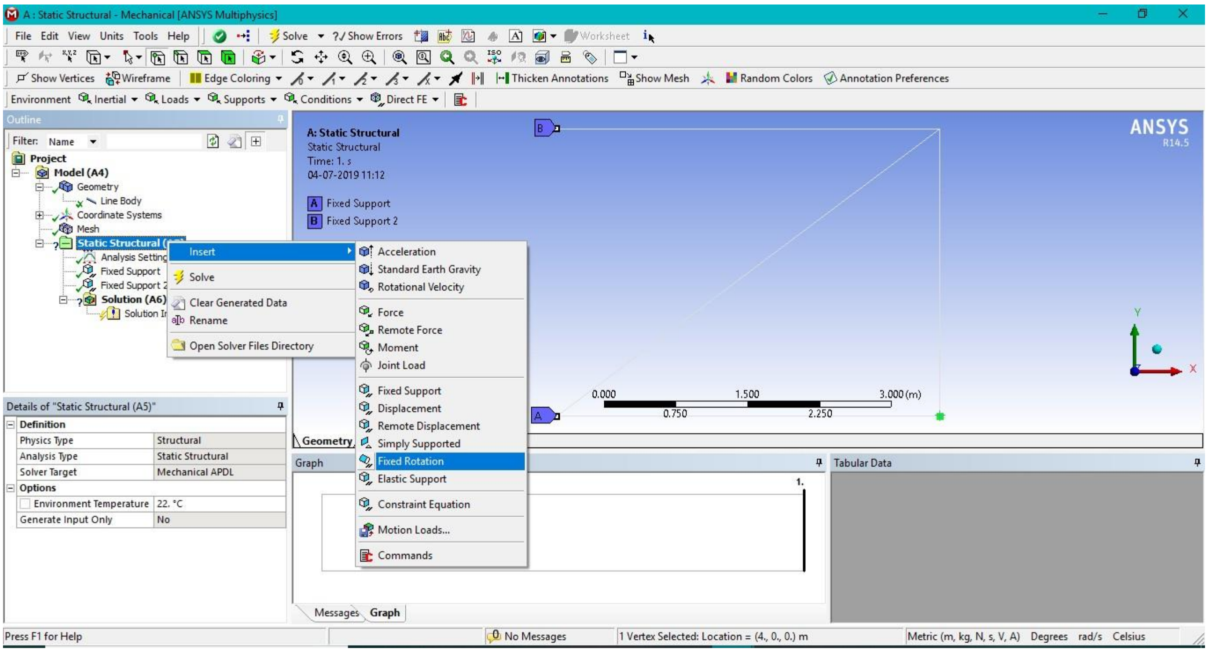
Mesh – Generate Mesh



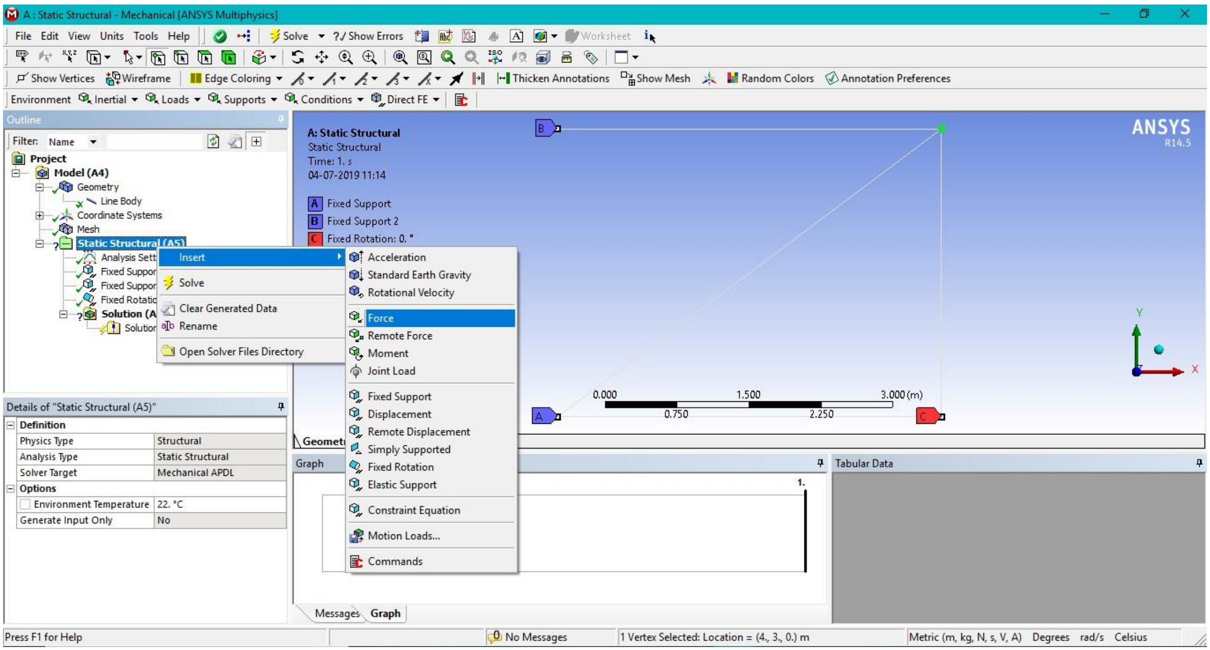
Select Vertex – First Point – Static Structural – Insert – Fixed Support
Second Point – Static Structural – Insert – Fixed Support

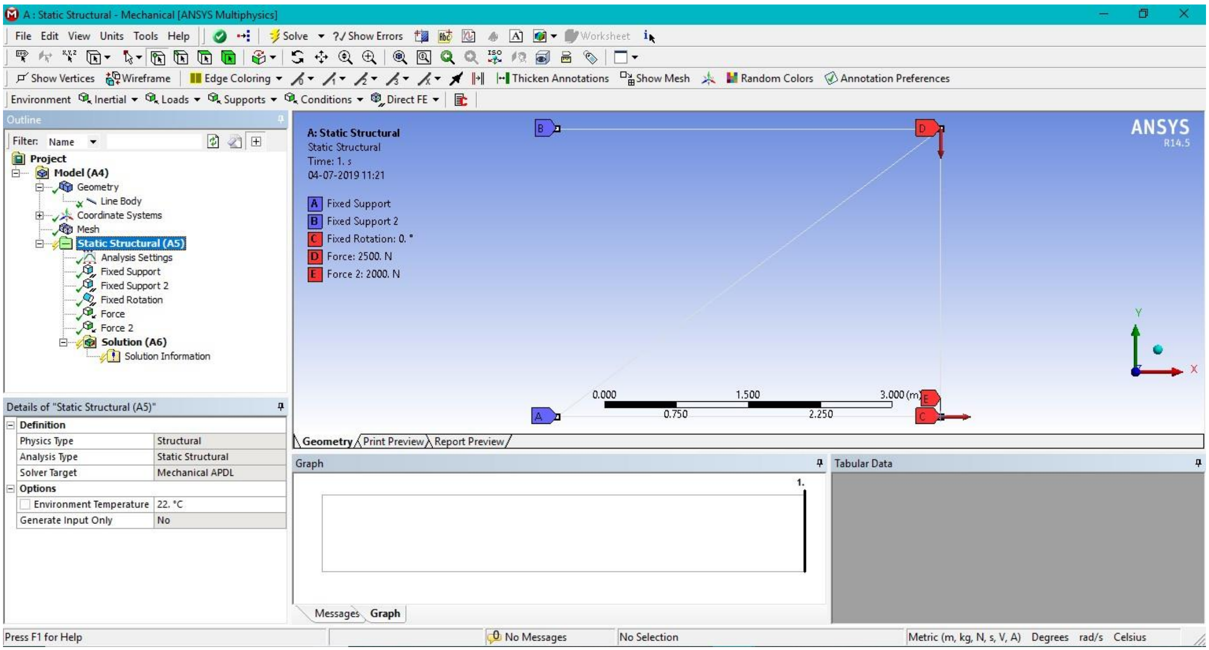


Third Point – Static Structural – Insert – Fixed Rotation

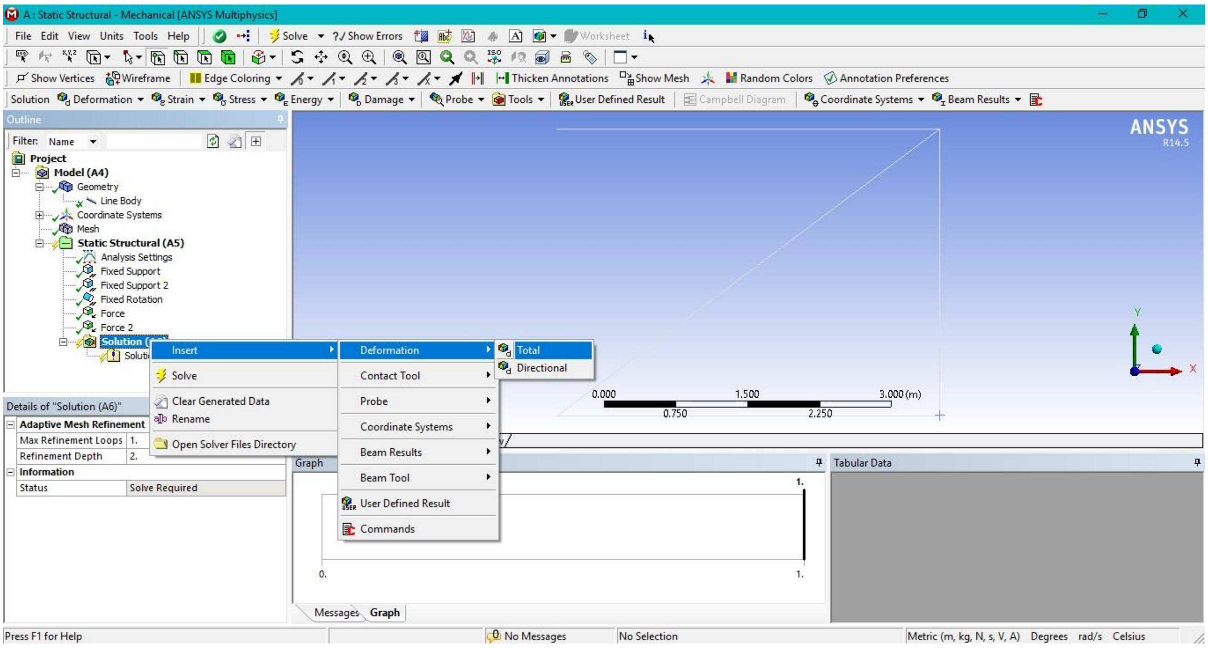


Fourth Point – Static Structural – Insert – Force
Components – Y Component – (-2500 N) -ve sign is for down ward direction
Third Point – Static Structural – Insert – Force
Components – X Component – (2000 N)

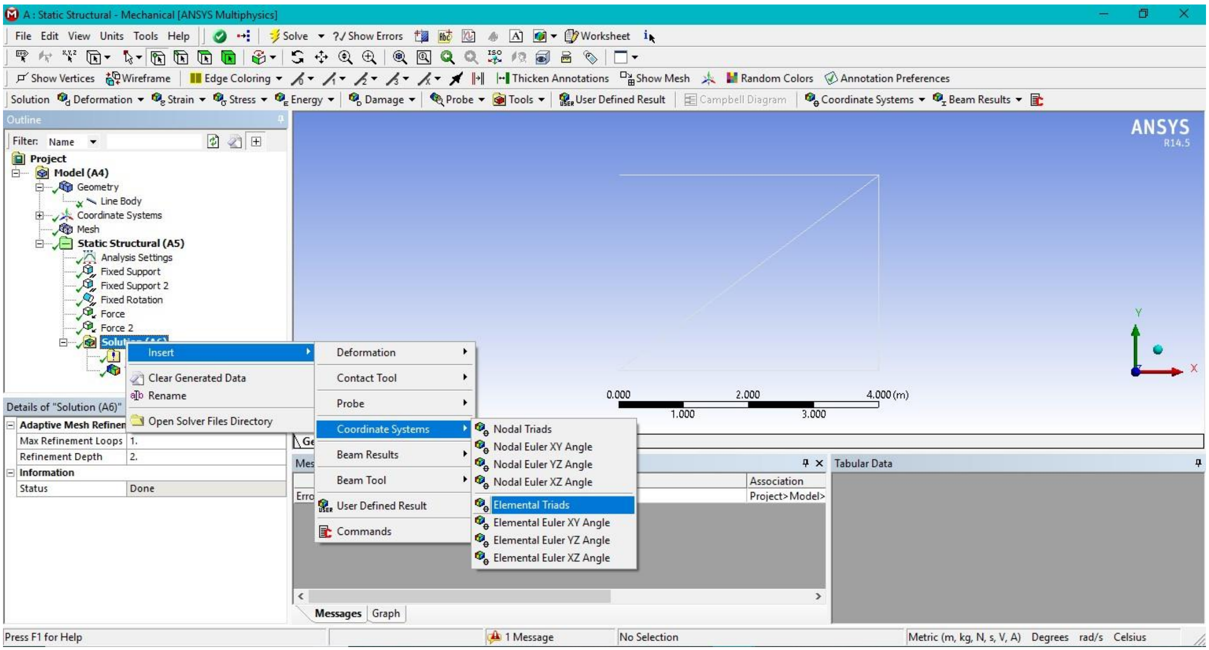




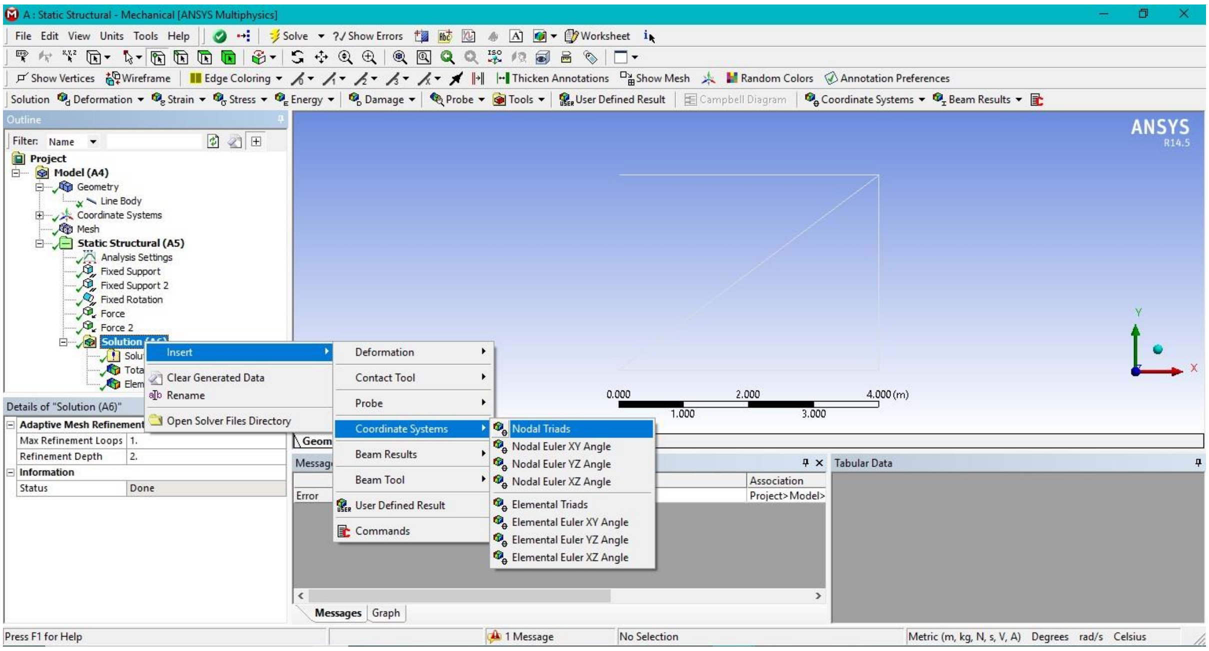
Solution – Insert – Deformation – Total – Solve



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?
- What is F.E.A ?
- Explain the process of F.E.A

EXPERIMENT - 04

STATIC ANALYSIS OF PLATES WITH A HOLE

AIM:

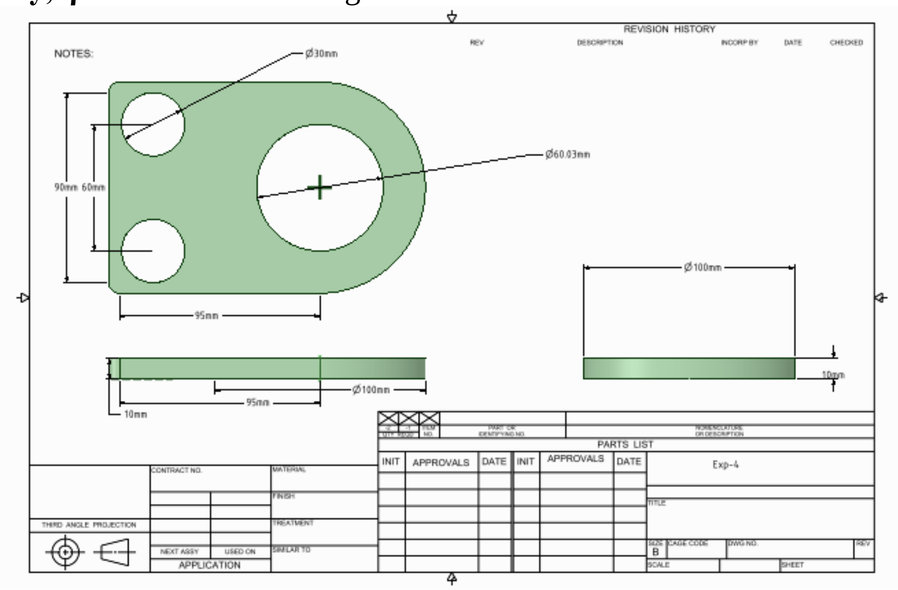
Static Analysis of plates with a hole to determine the deformation and Equivalent Stress.

SOFTWARE: ANSYS

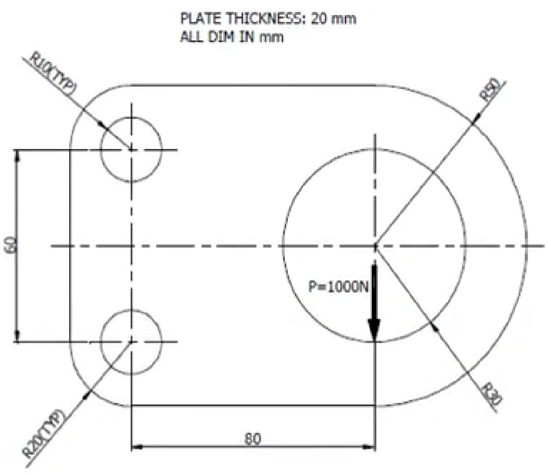
THEORY:

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 10^9 Pascal
- **Poisson's Ratio, γ** = 0.3
- **Density, ρ** = 7860 kg/m^3



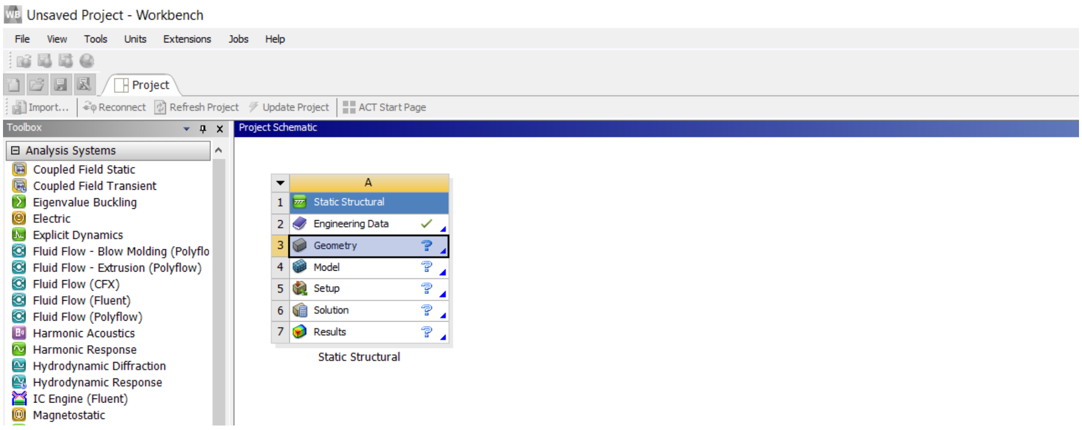
The forces Acting are shown below



PROCEDURE:

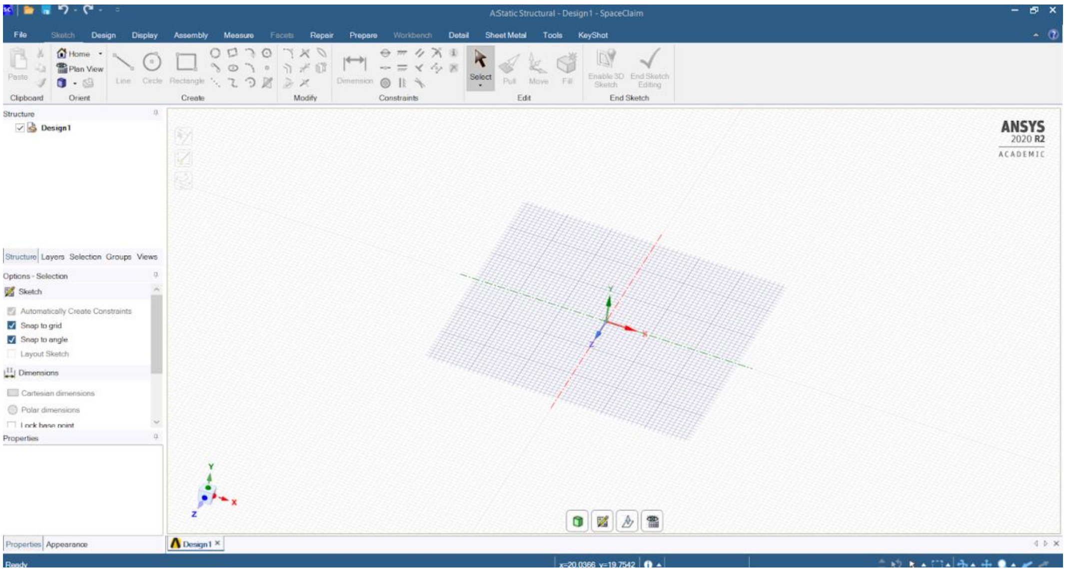
Step1: Workbench Tool box

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

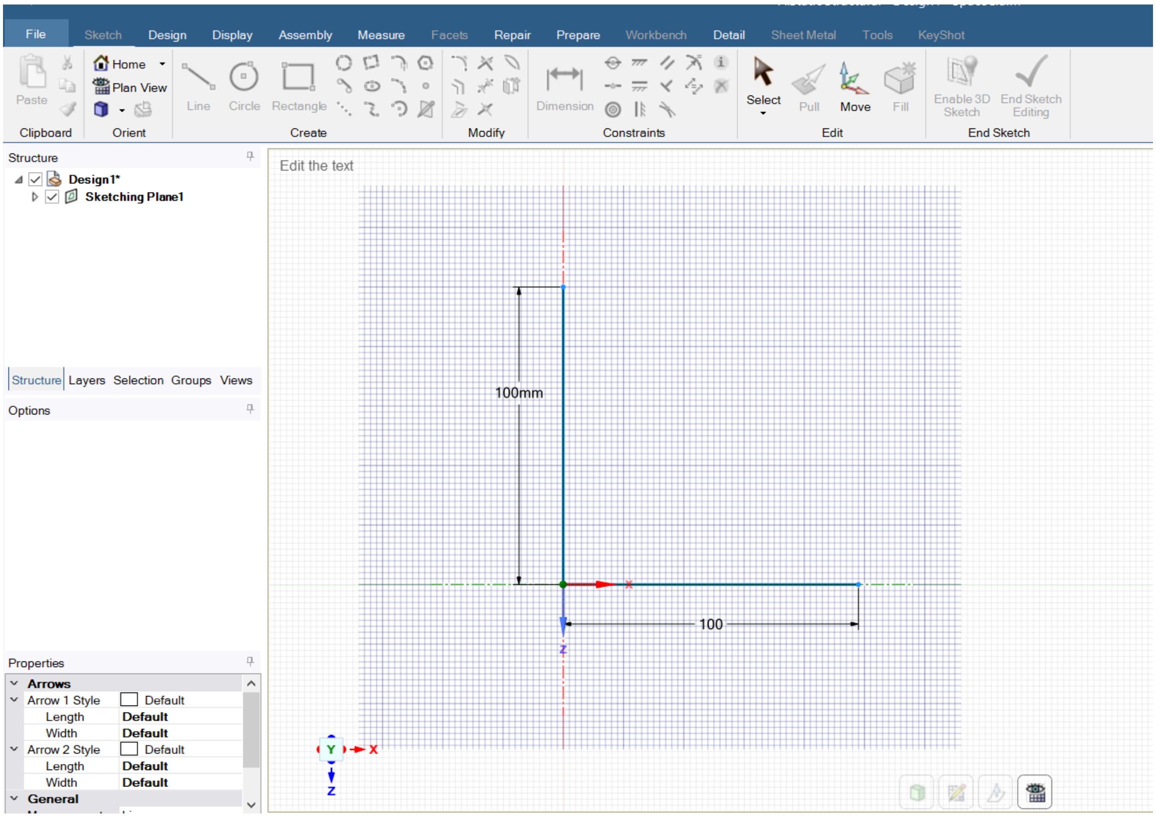


Step2-Creating Geometry

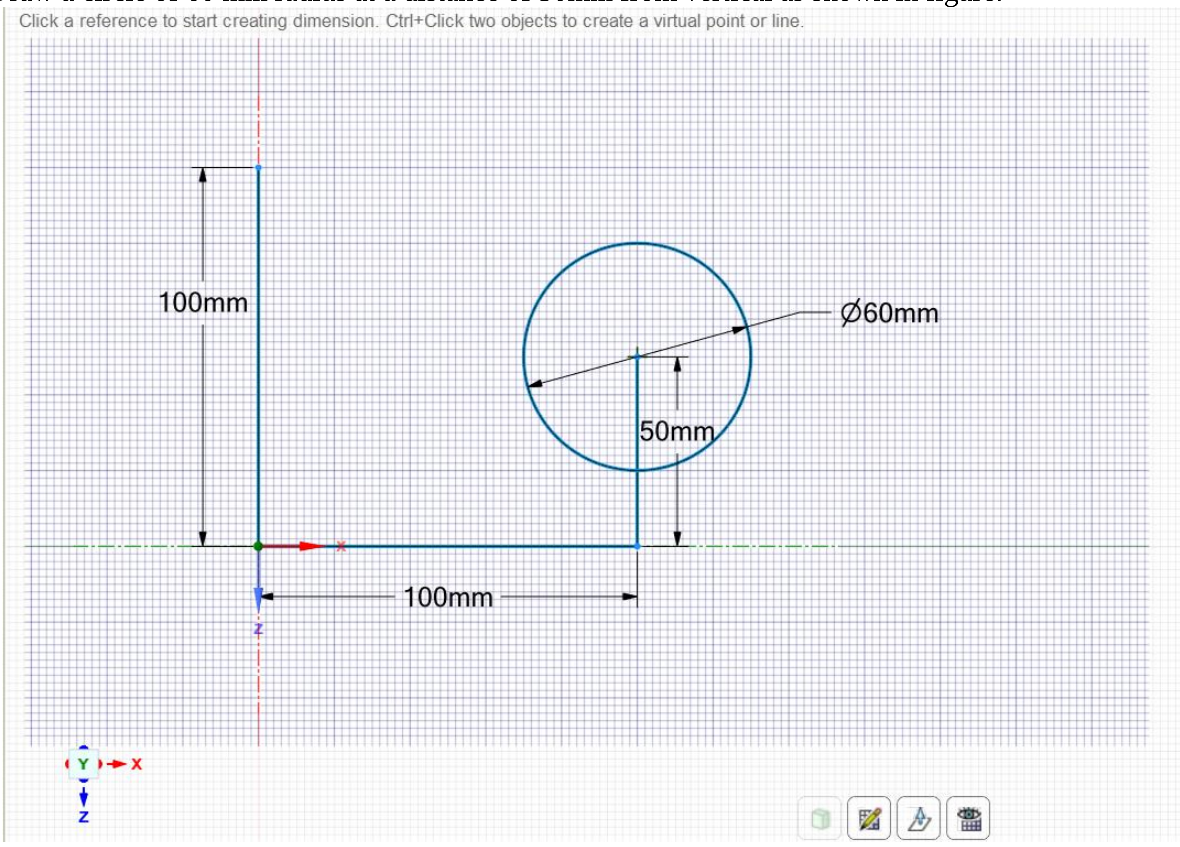
Click on geometry Dialogue box Space Claim option will open.



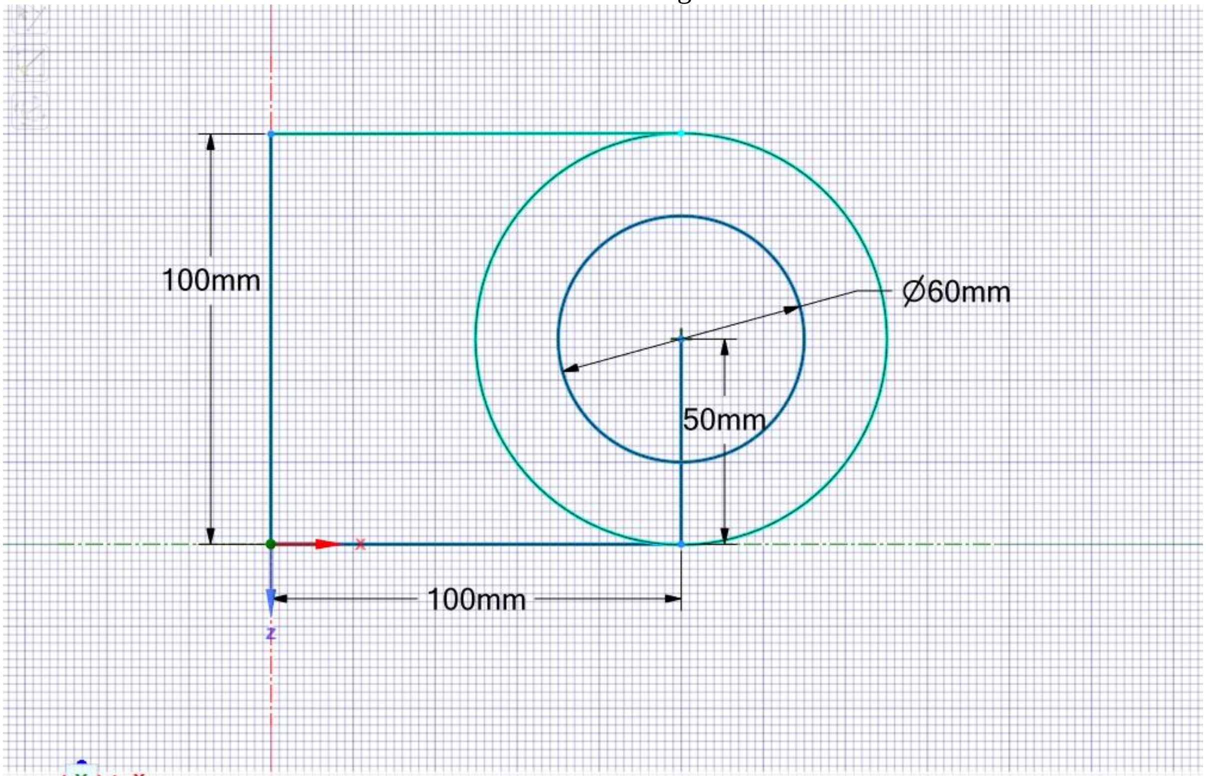
Select any Plane Using Pan View
Set Units as mm
Sketch Two Line perpendicular to each with 100mm dimension each as shown in figure.



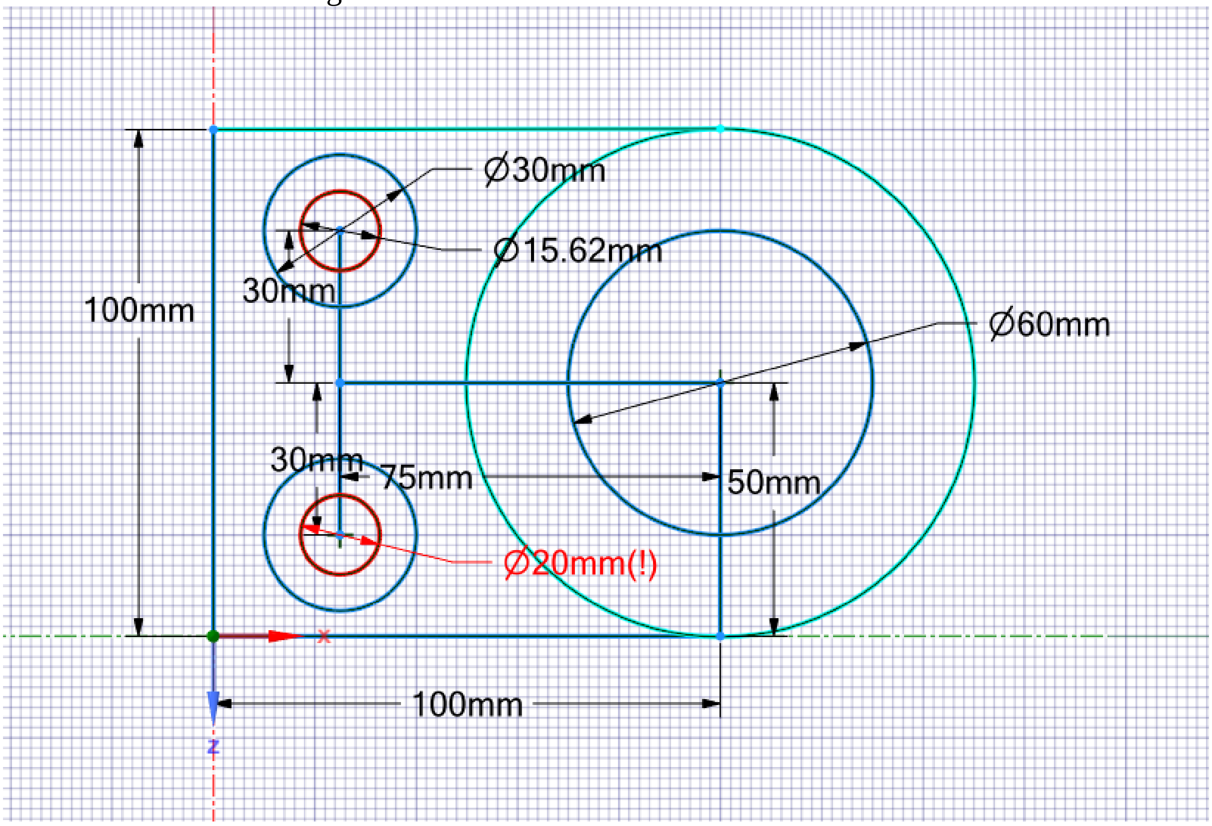
Draw a circle of 60 mm radius at a distance of 50mm from vertical as shown in figure.



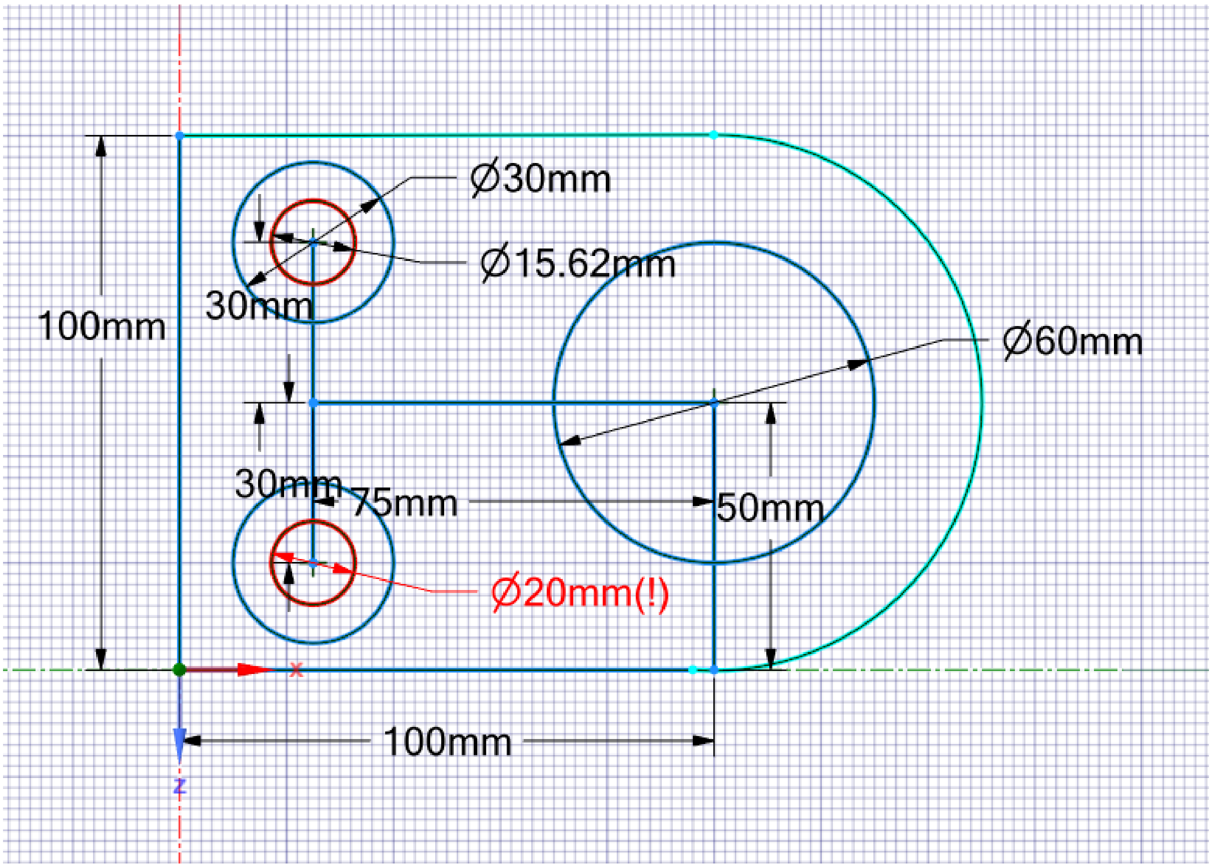
Draw another circle with 100 mm diameter as shown in figure.



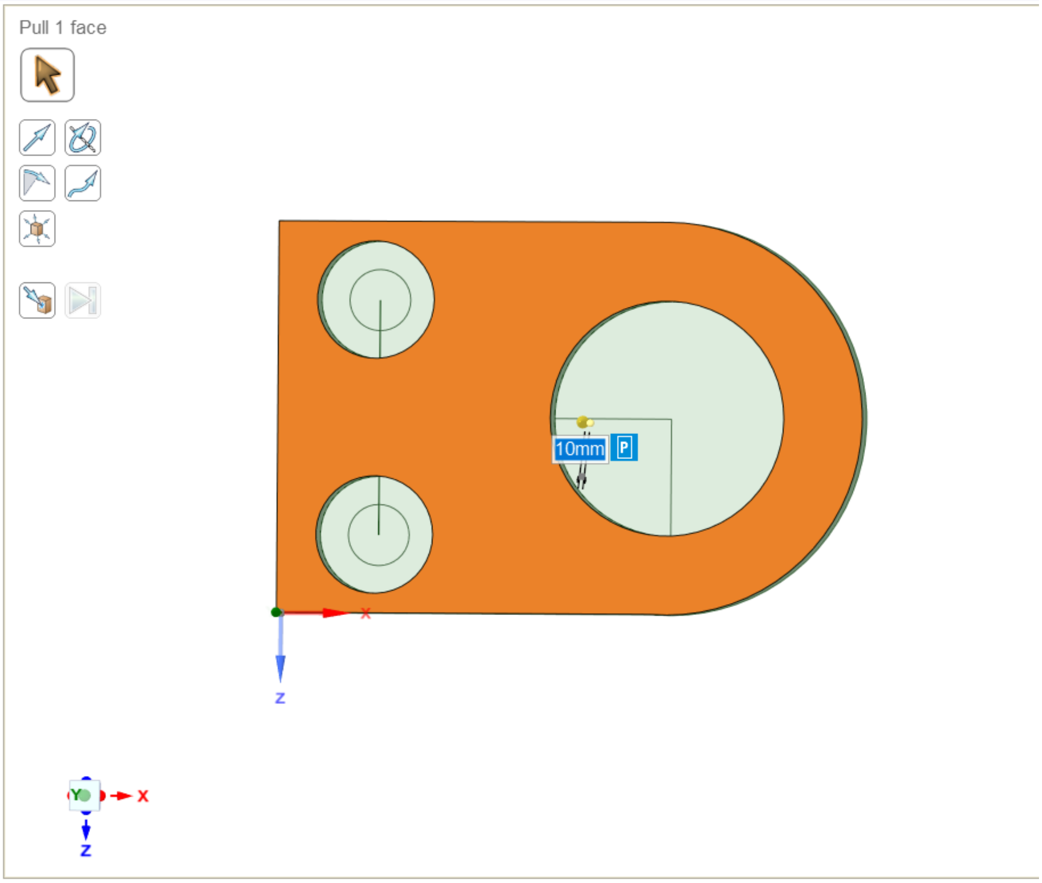
Draw another circle with given dimensions



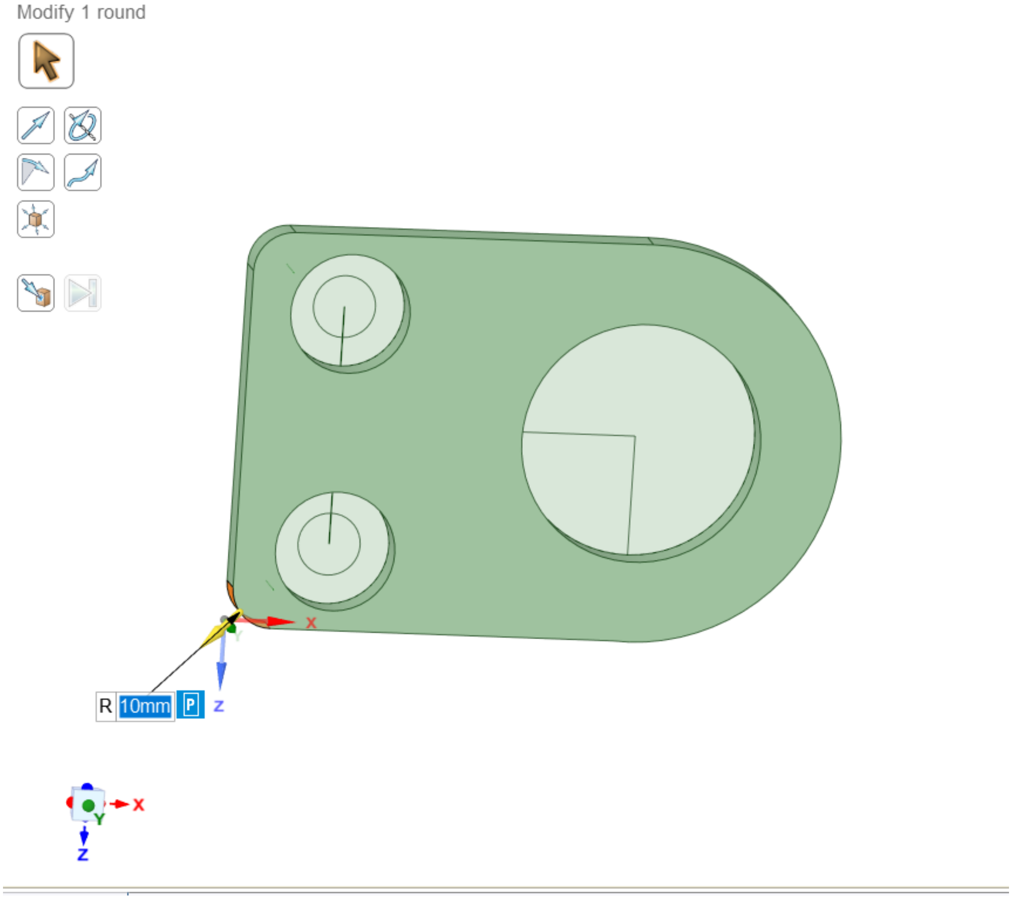
Trim


Ø30mm
Ø15.62mm
30mm
100mm
Ø60mm
30mm
75mm
50mm
Ø20mm(!)
x
100mm
z

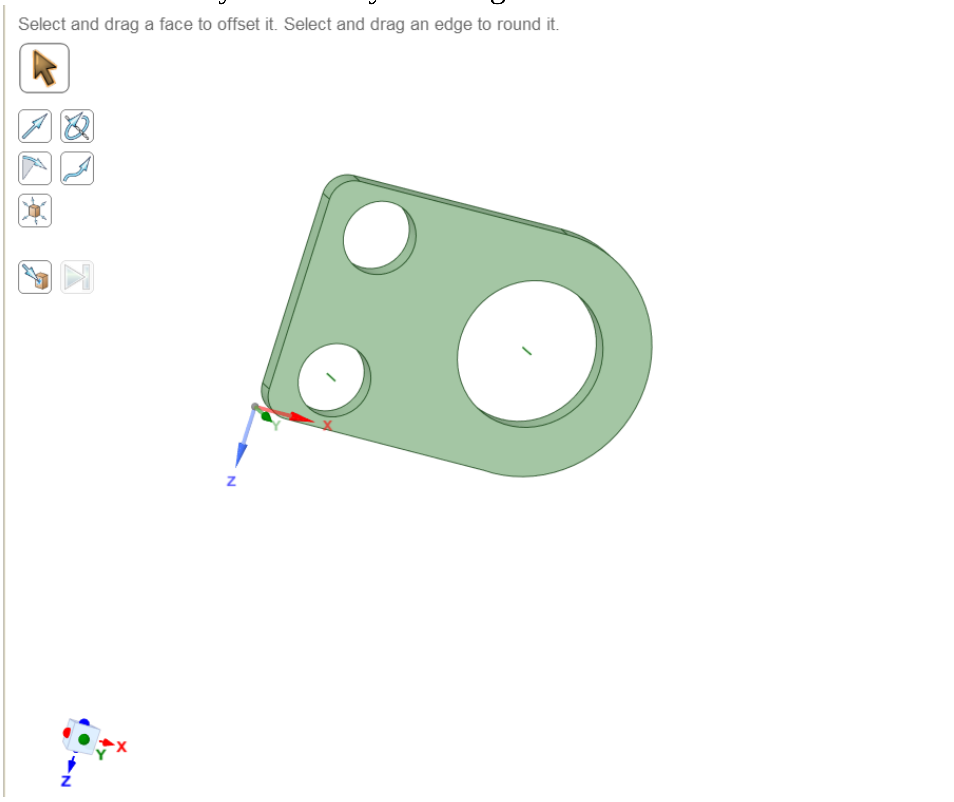
Pull


Pull 1 face
10mm
P
x
z
Y
x
z

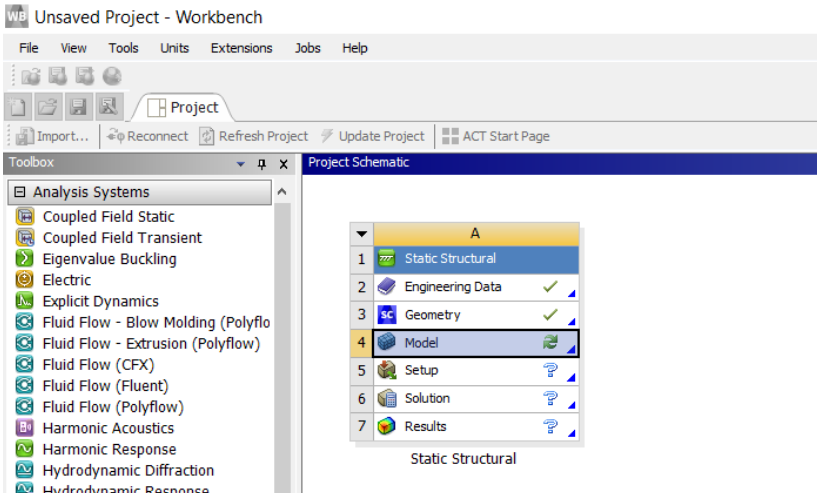
Giving Fillets

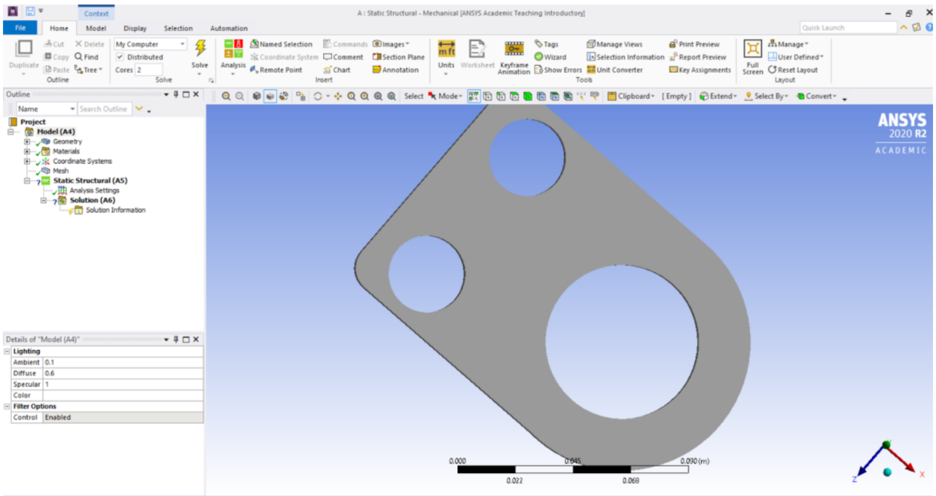


Deleting Unwanted Geometry and line by selecting Delete icon

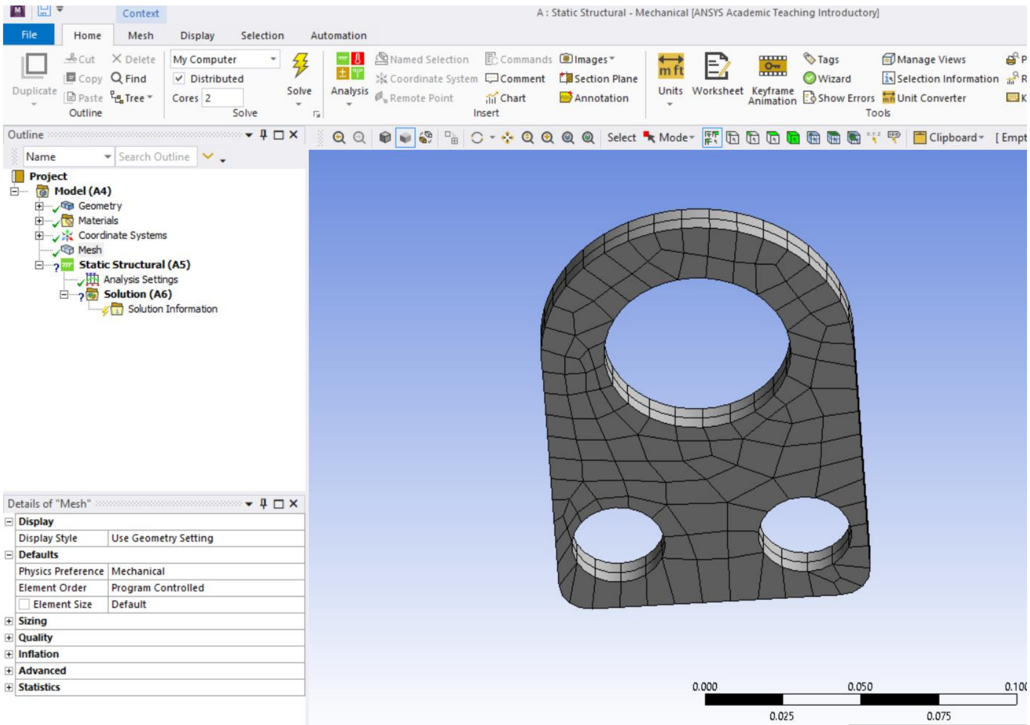


Click on Model Update it and wait for geometry to upload then double click it

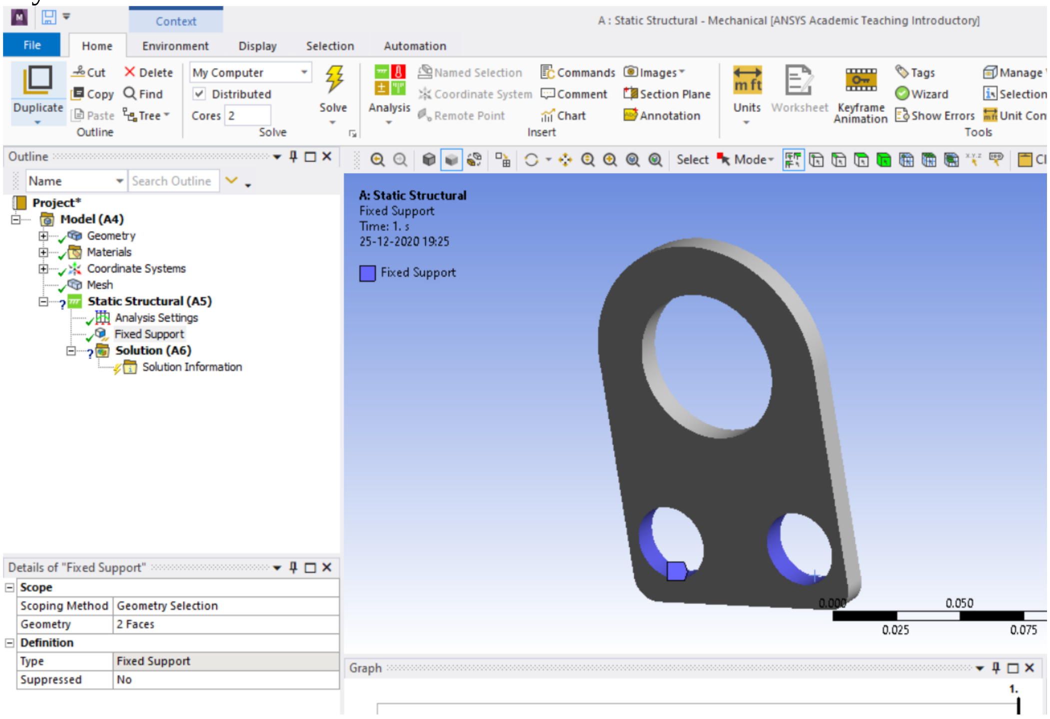




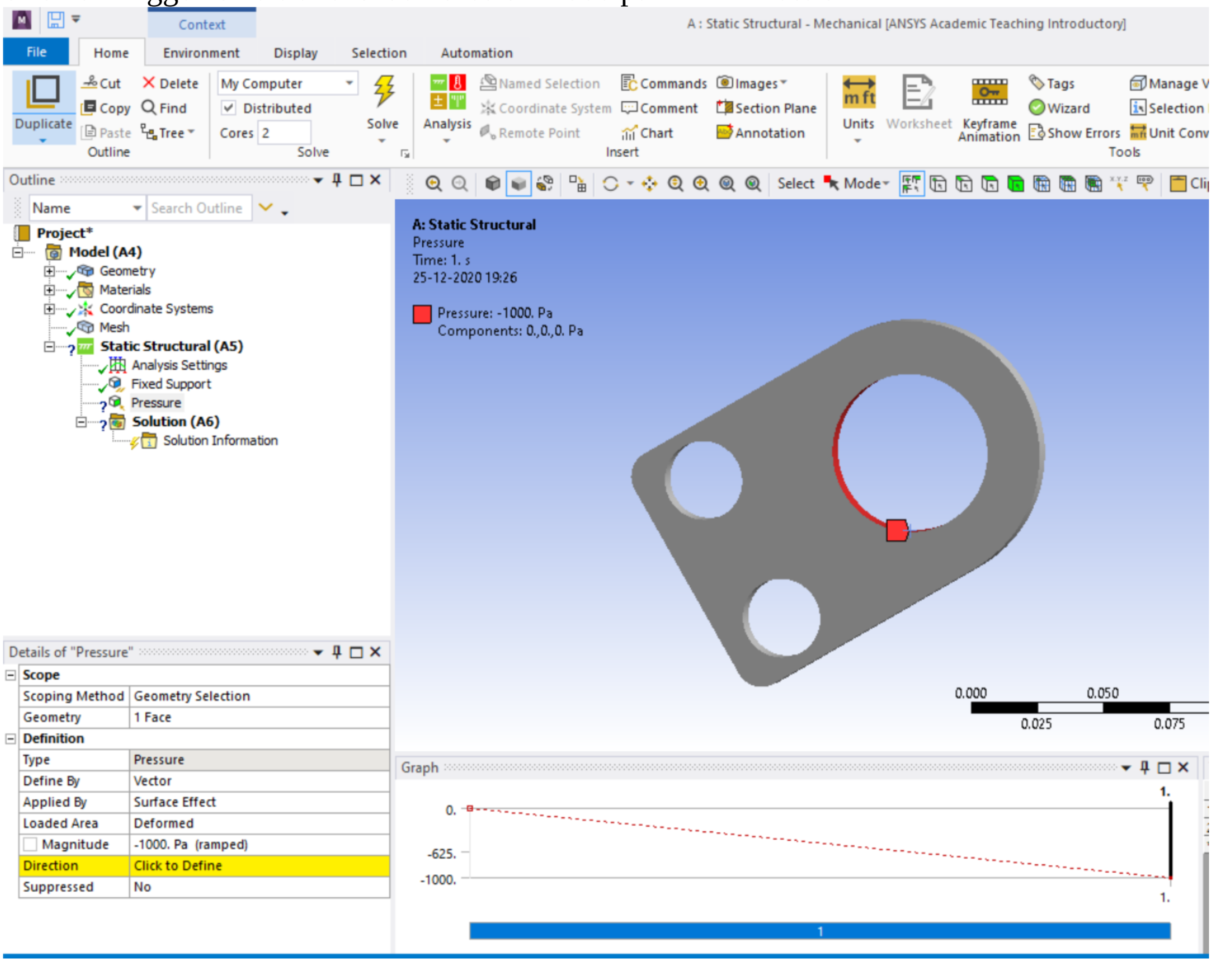
Click on Mesh



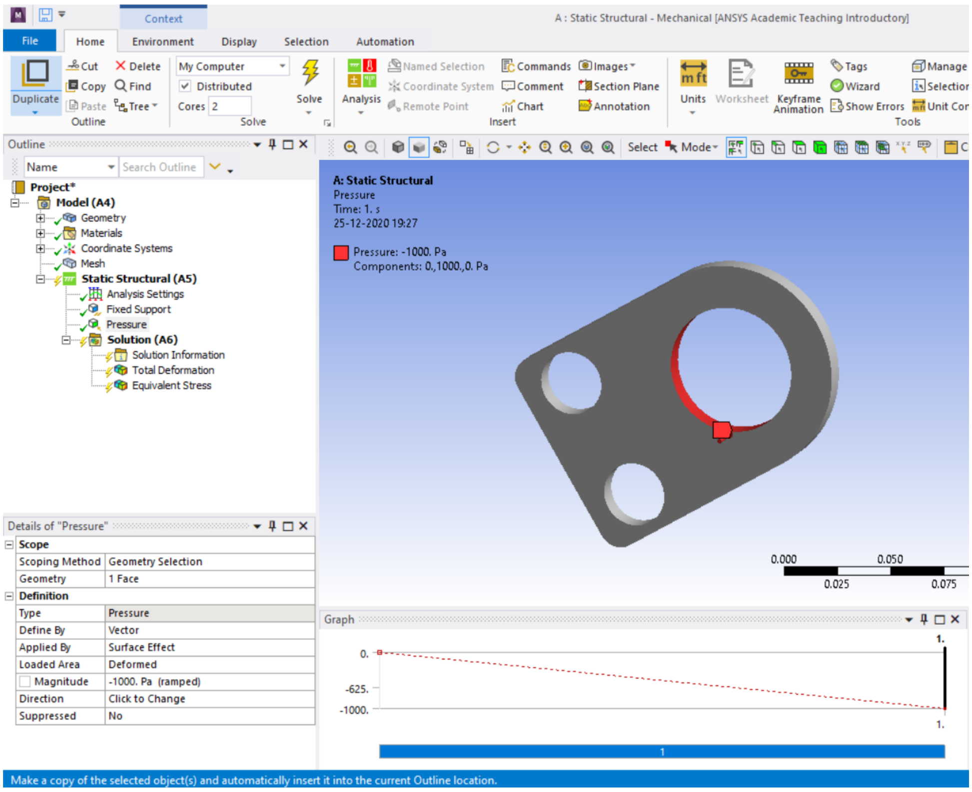
Boundary Condition :- Fix two small Holes



Click on Bigger Circle and Give Force of -1000 pa in –x Direction



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

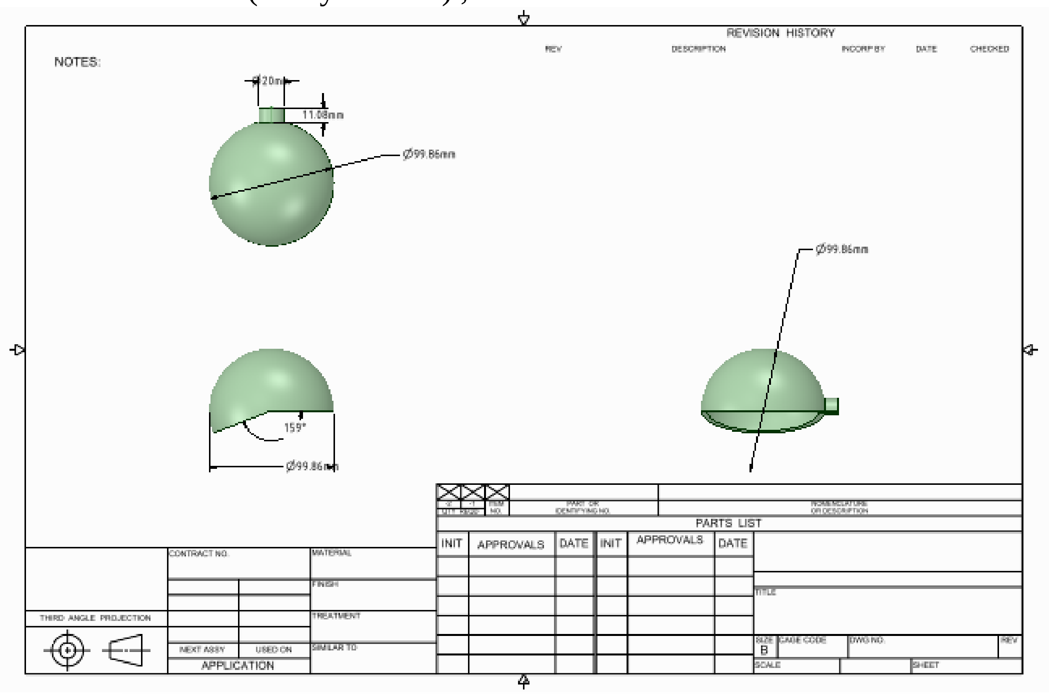
AIM:

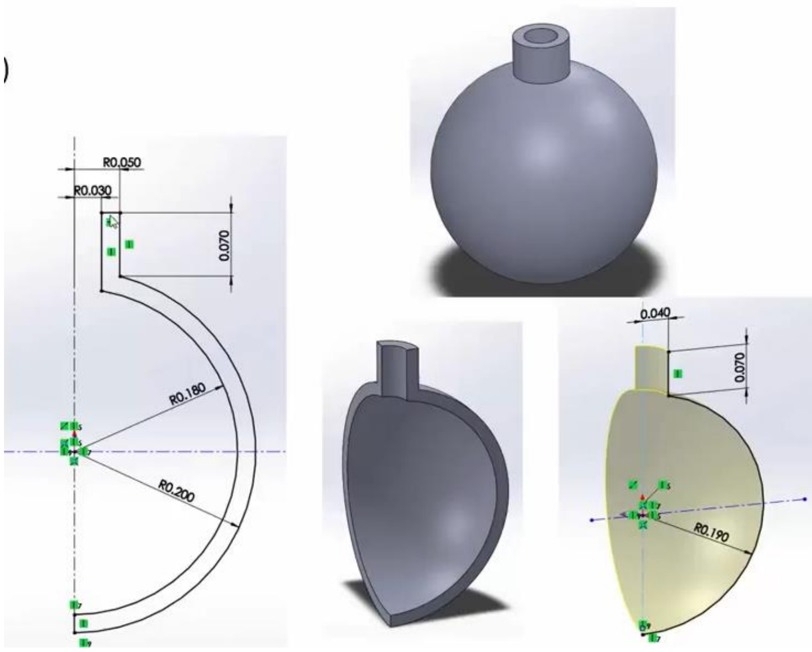
To calculate Stress in case of axi-symmetric body.

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

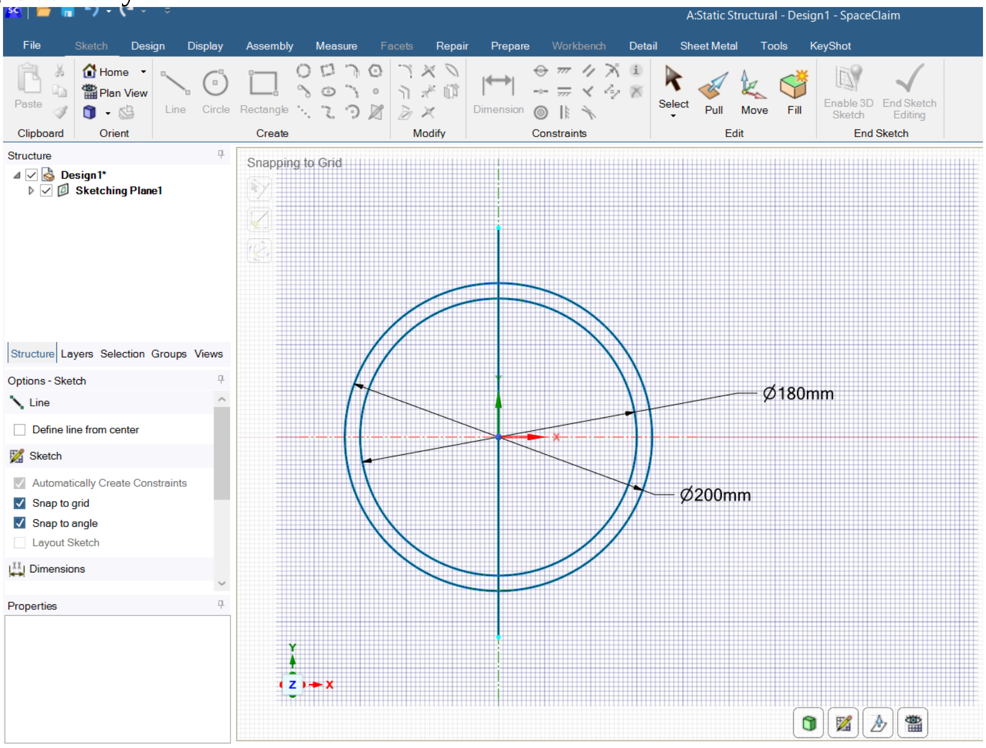




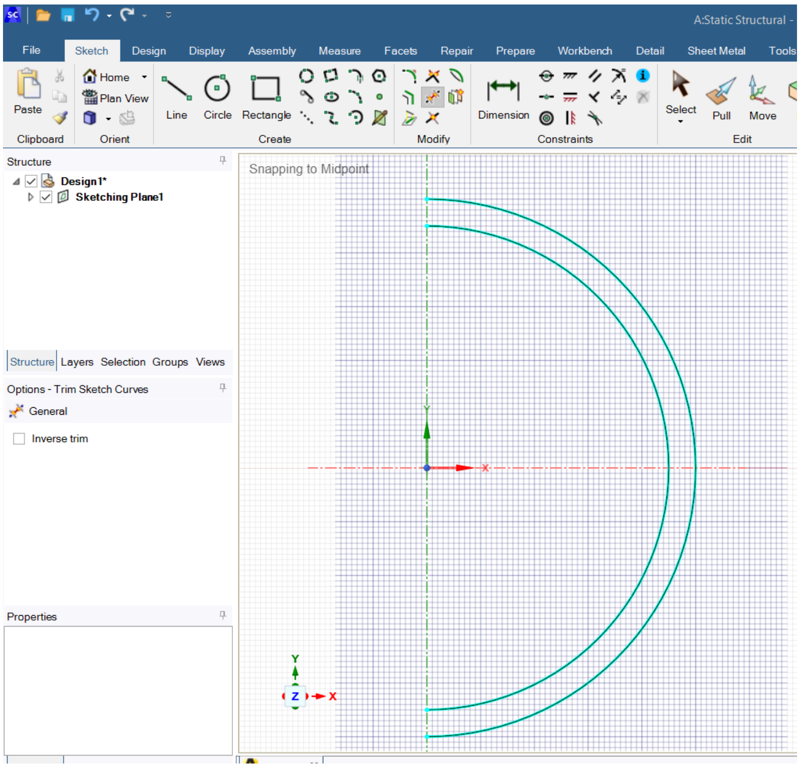
PROCEDURE:

Opening Space Claim

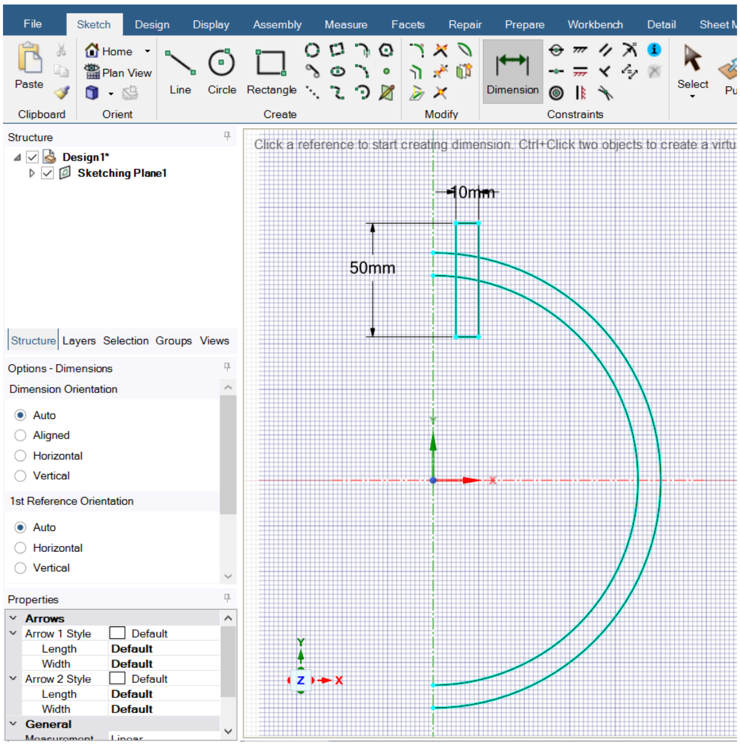
Drawing Geometry



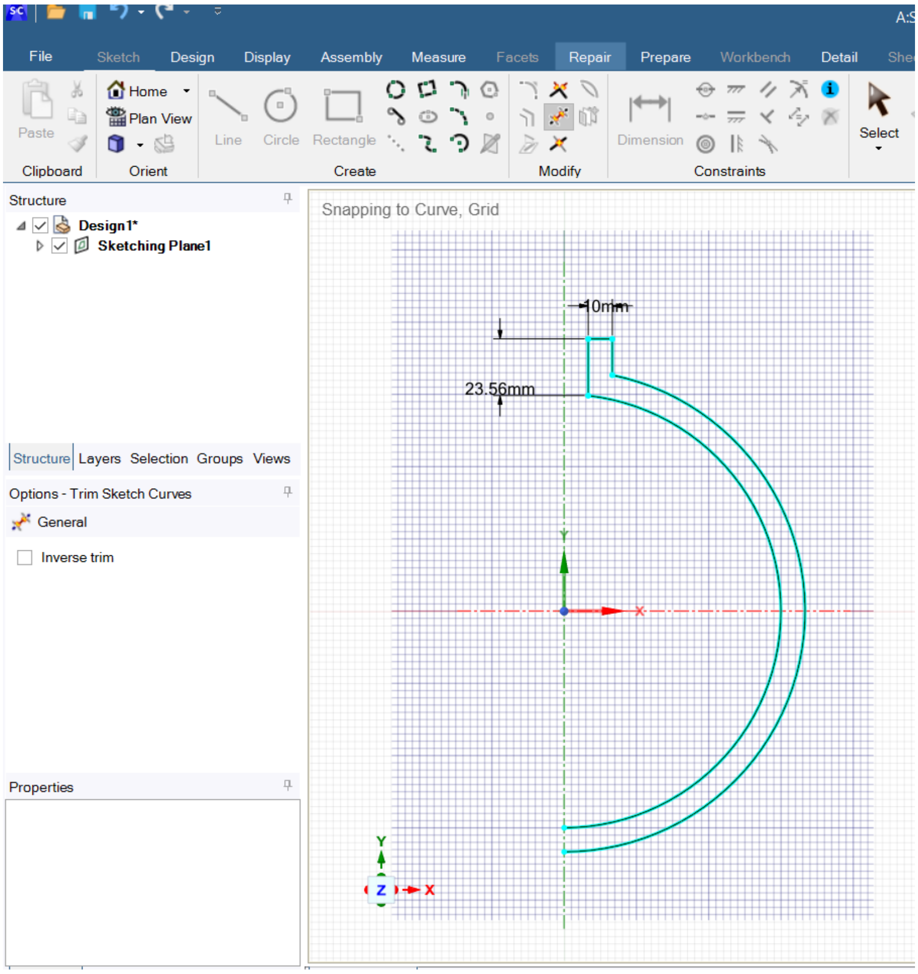
Trim



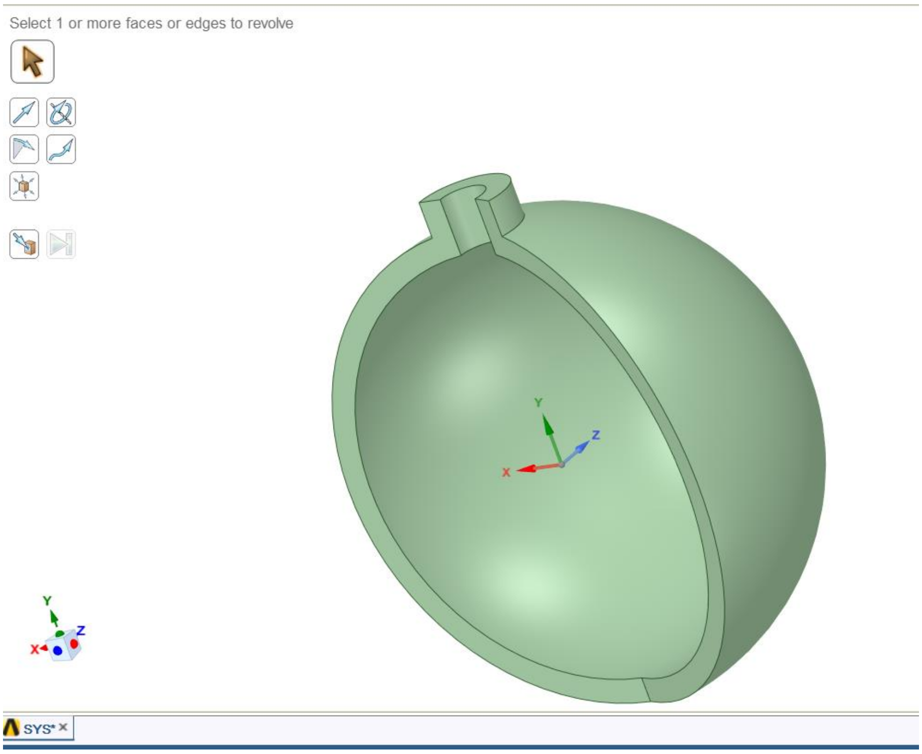
Draw rectangle



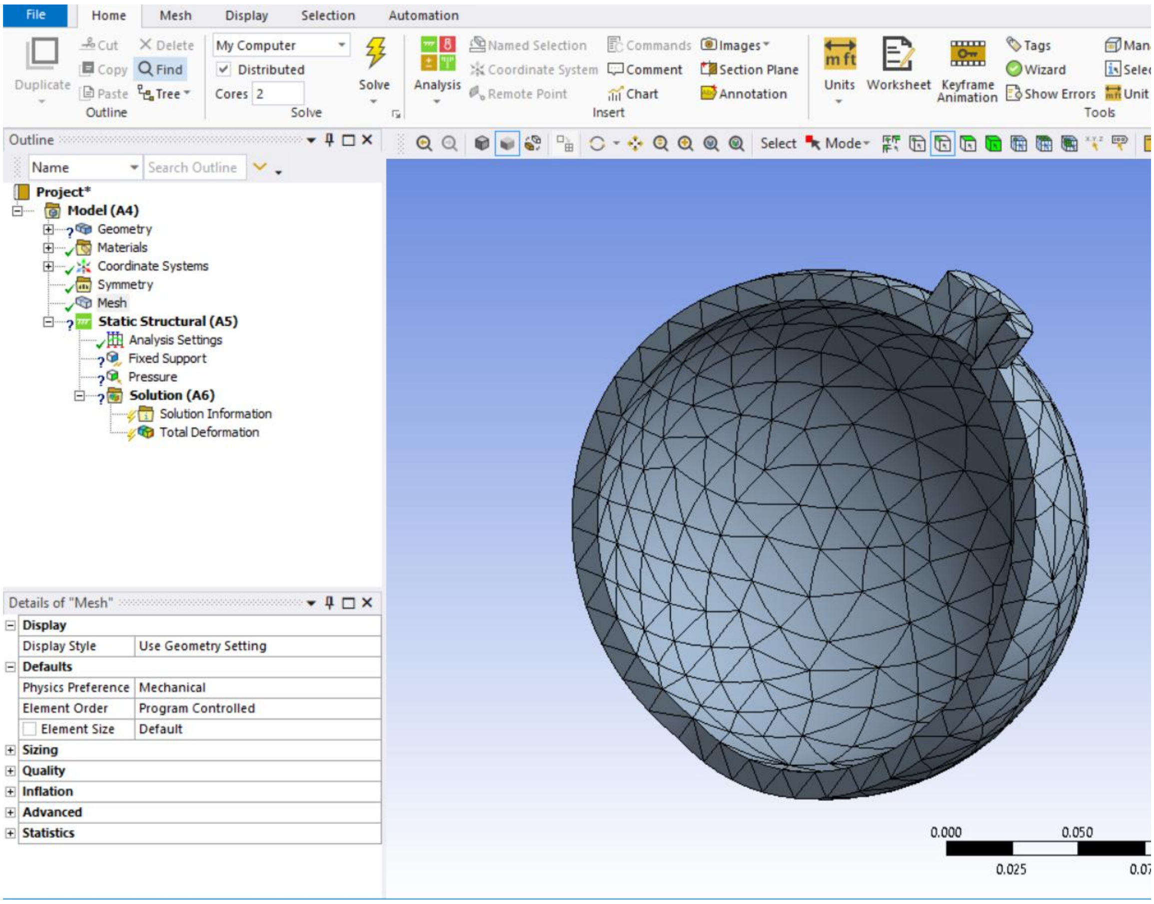
Trim



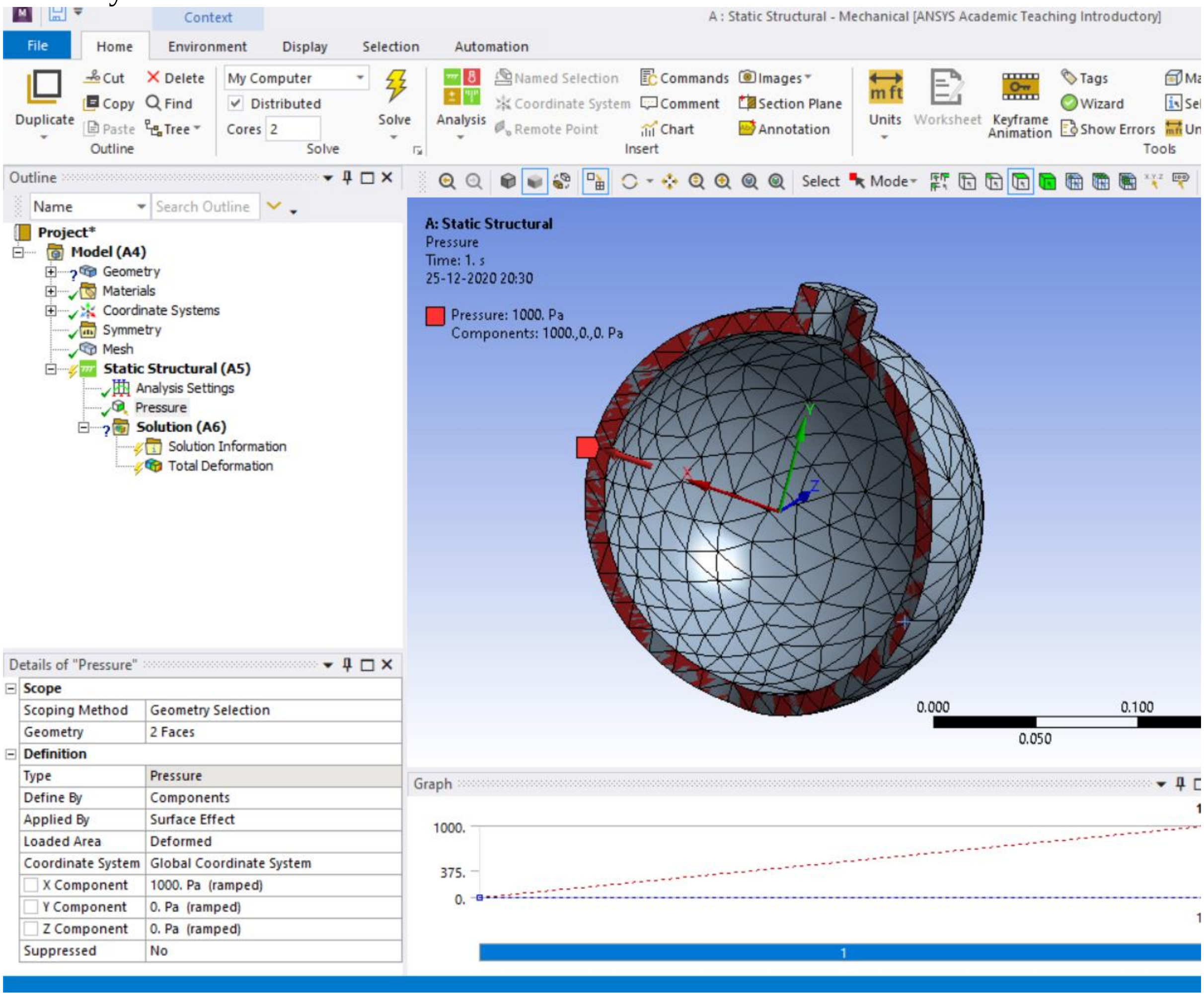
Pull-Revolve



Mesh



Boundary Condition



- 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

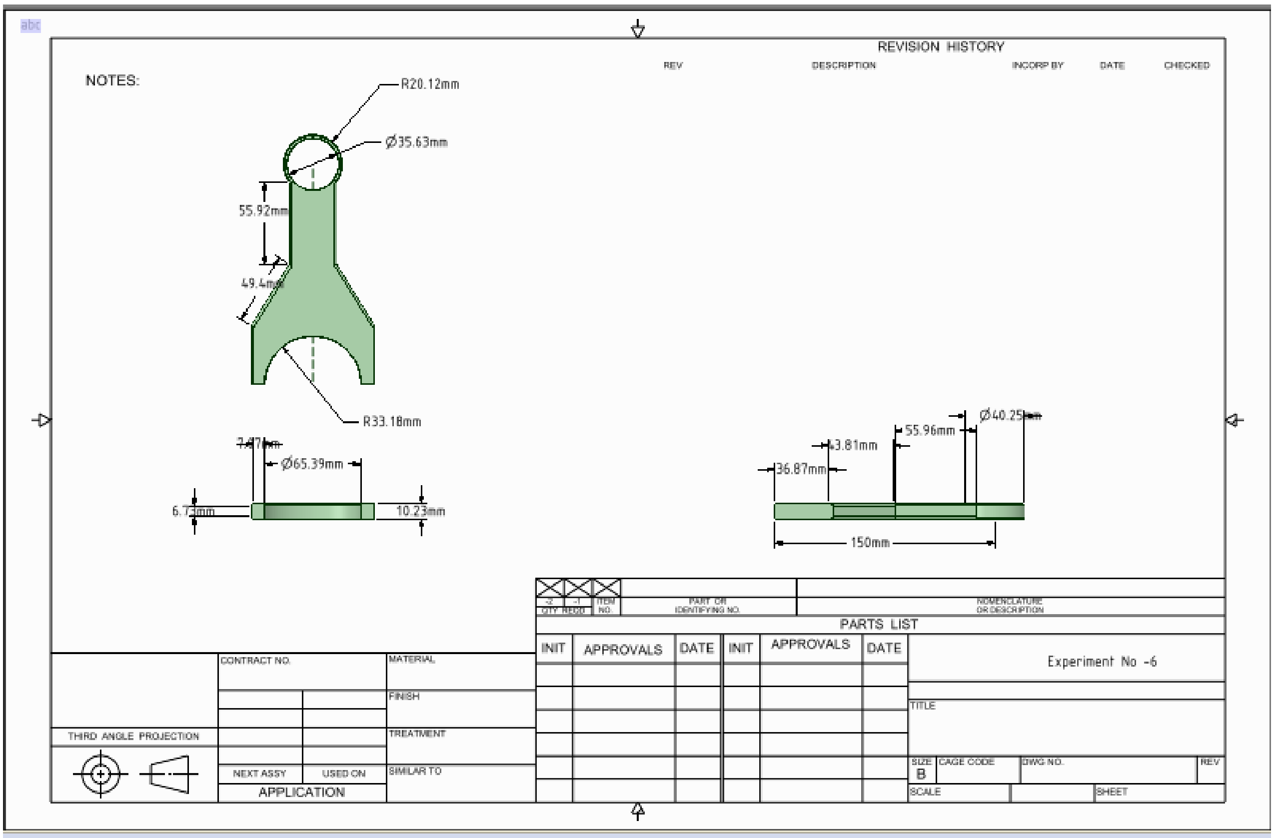
AIM:

To do Static analysis of connecting rod.

SOFTWARE: ANSYS

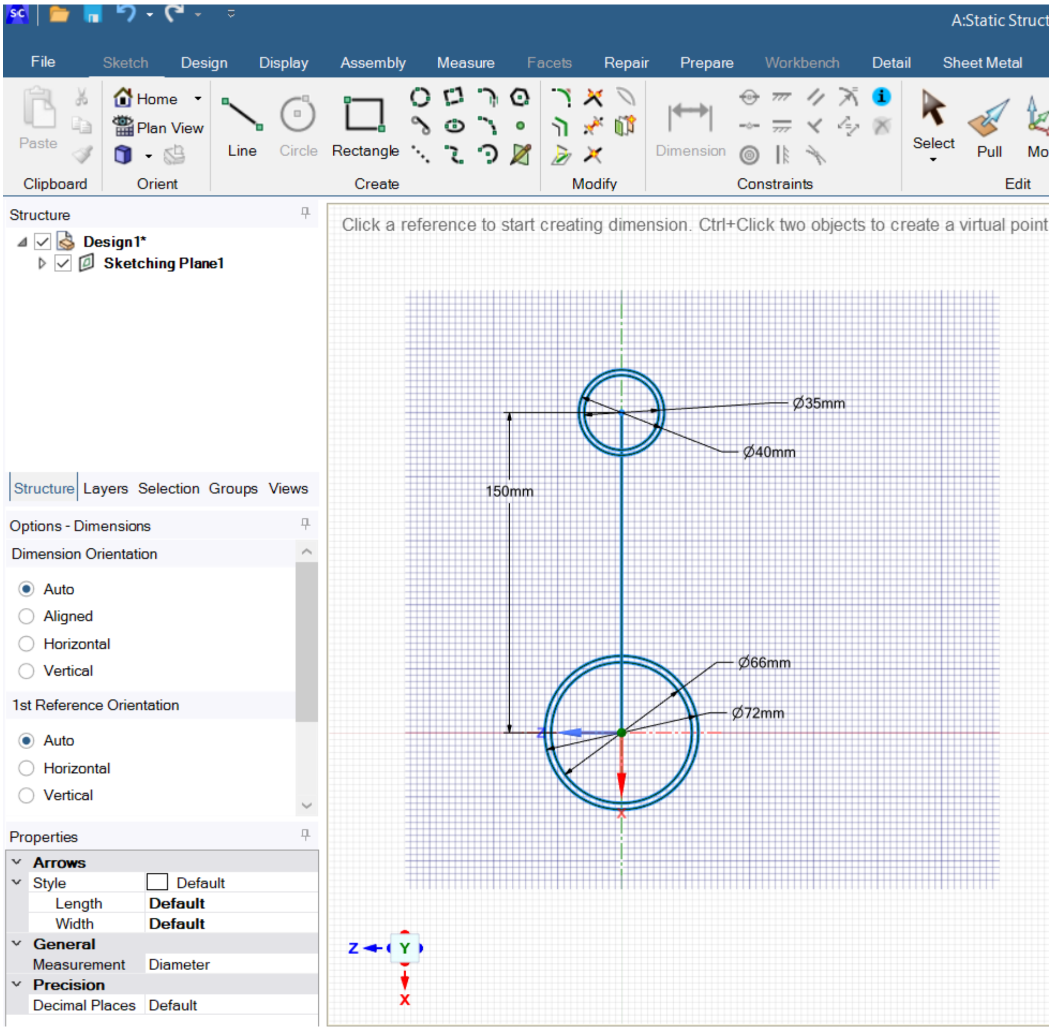
THEORY:

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

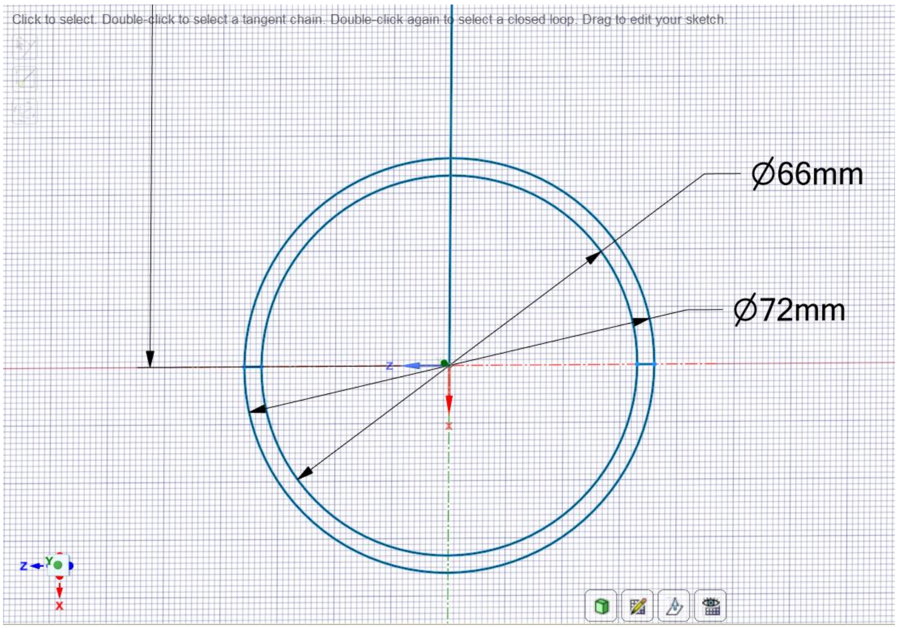


PROCEDURE:

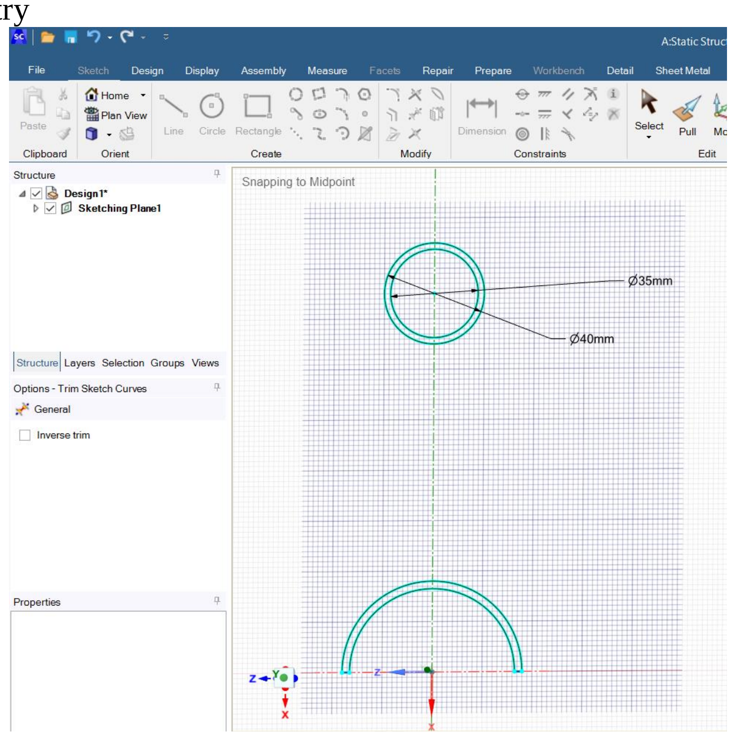
Opening Space Claim
Drawing Geometry


A:Static Struct
File
Sketch
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Paste
Home
Plan View
Line
Circle
Rectangle
Dimension
Select
Pull
Clipboard
Orient
Create
Modify
Constraints
Edit
Structure
Design1*
Sketching Plane1
Click a reference to start creating dimension. Ctrl+Click two objects to create a virtual point
Ø35mm
Ø40mm
150mm
Ø66mm
Ø72mm
Structure
Layers
Selection
Groups
Views
Options - Dimensions
Dimension Orientation
Auto
Aligned
Horizontal
Vertical
1st Reference Orientation
Auto
Horizontal
Vertical
Properties
Arrows
Style
Default
Length
Default
Width
Default
General
Measurement
Diameter
Precision
Decimal Places
Default

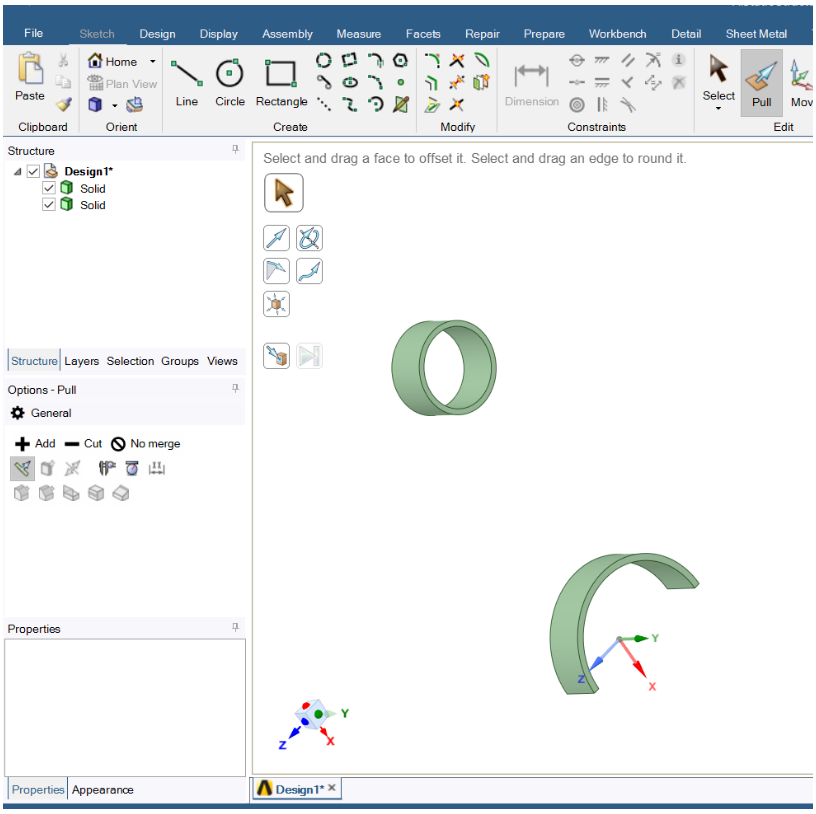
Draw Circle


Click to select. Double-click to select a tangent chain. Double-click again to select a closed loop. Drag to edit your sketch.
Ø66mm
Ø72mm

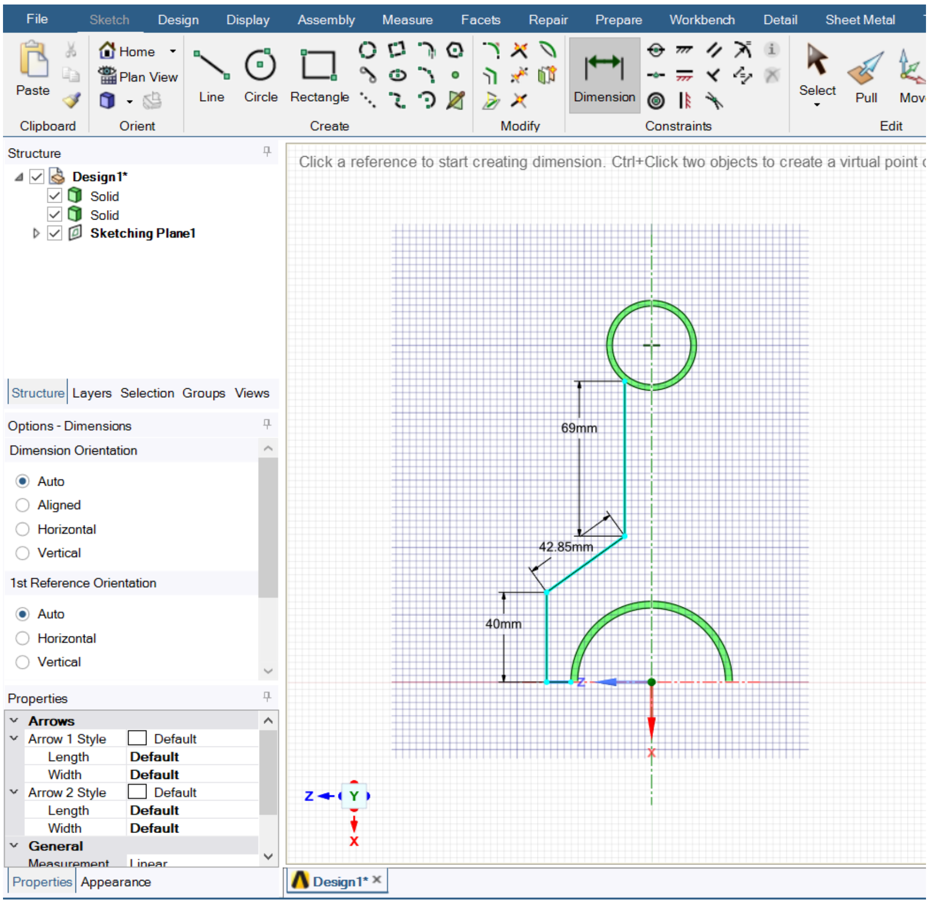
Trimming Geometry



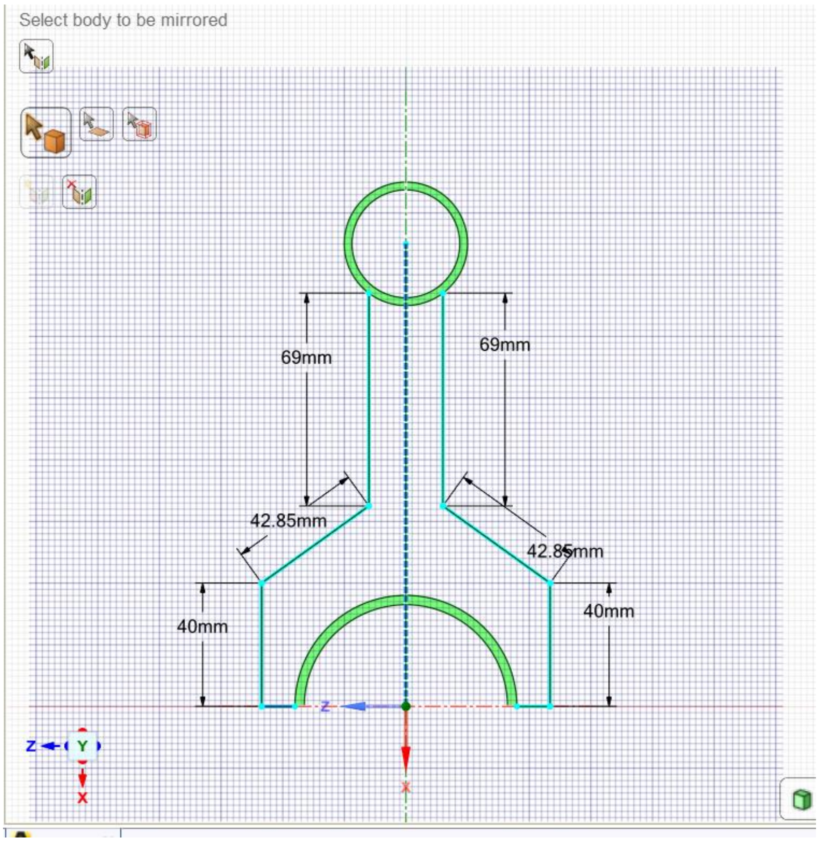
Pull



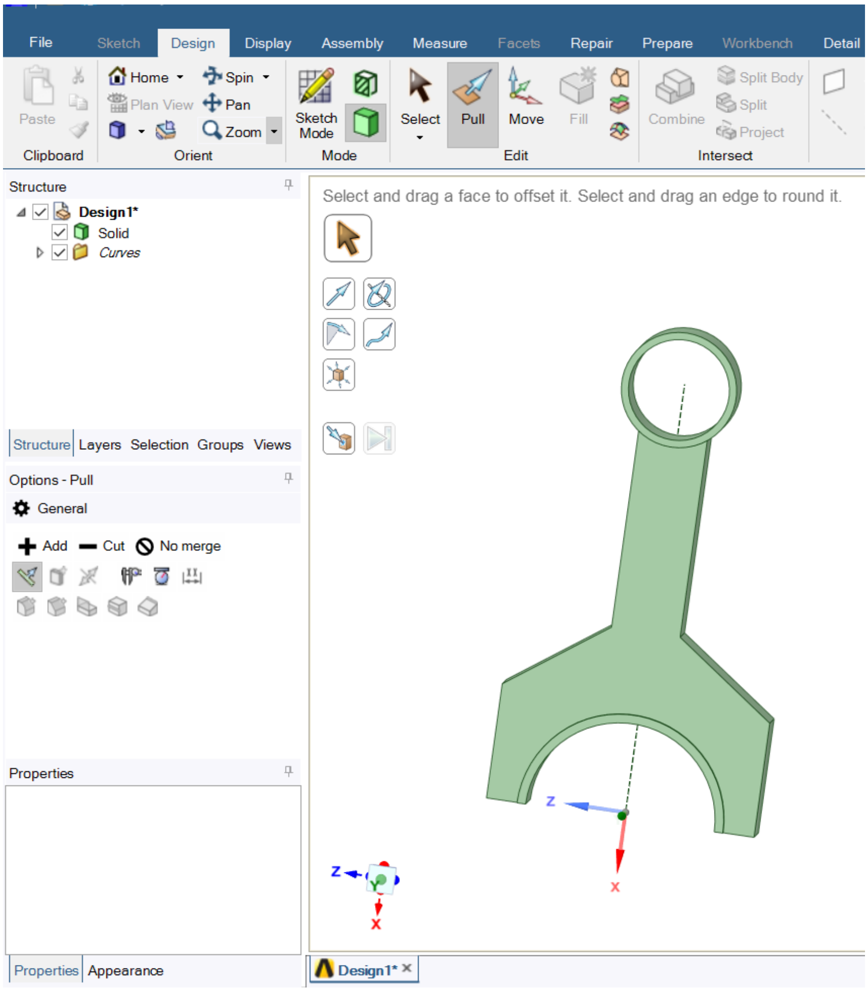
Draw 2D rod



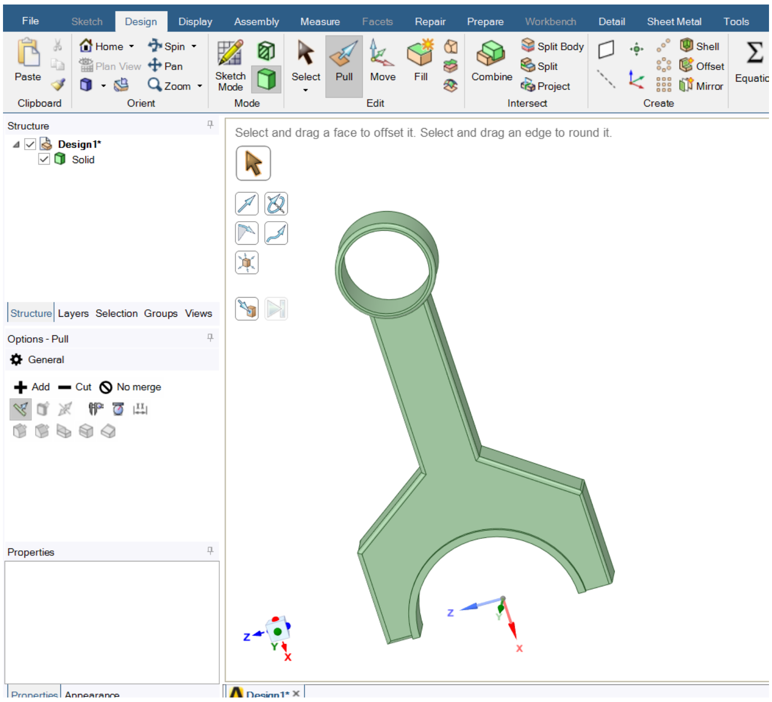
Mirror



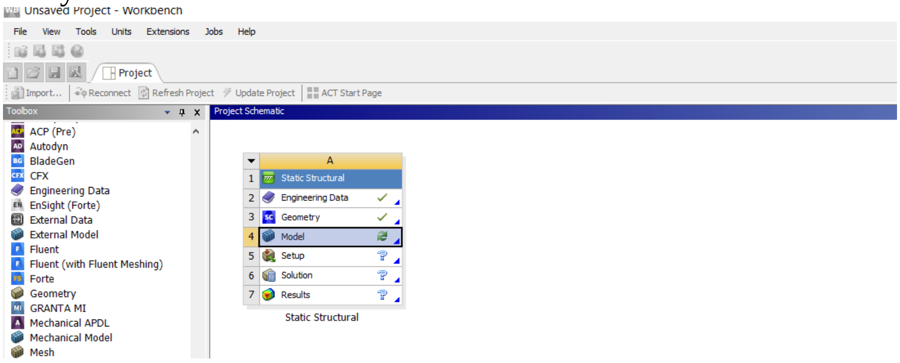
Pull



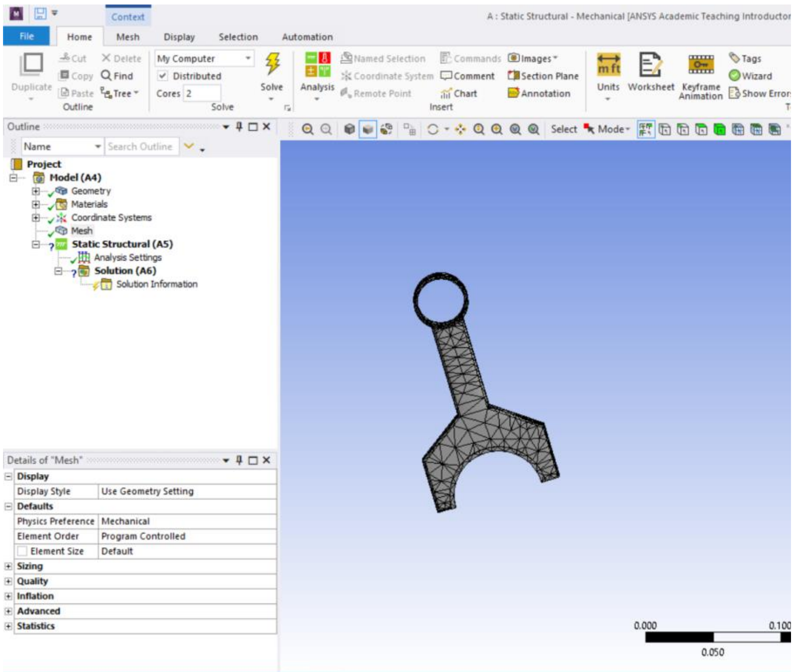
Giving Fillet



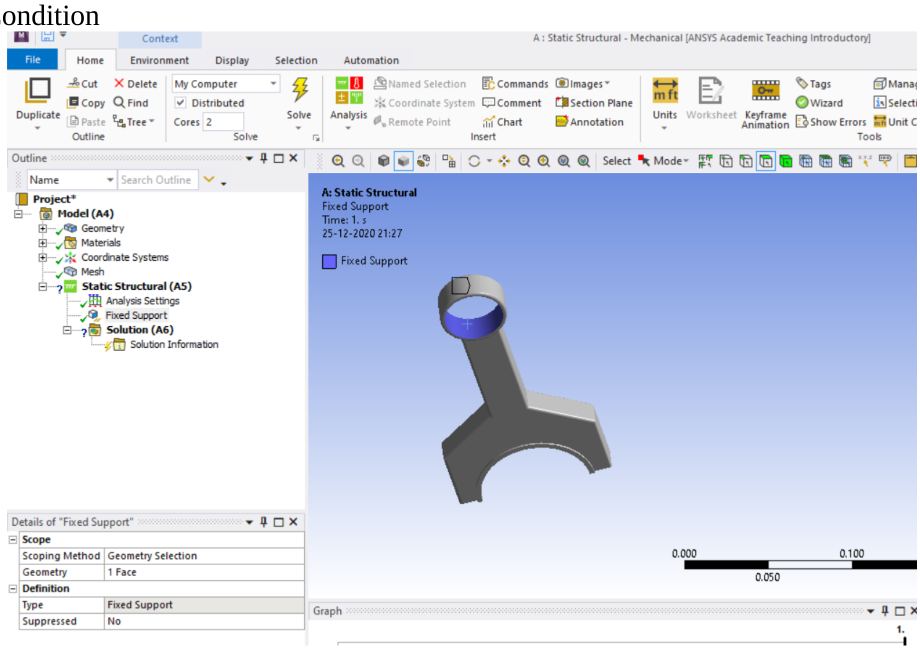
Update Geometry in Model



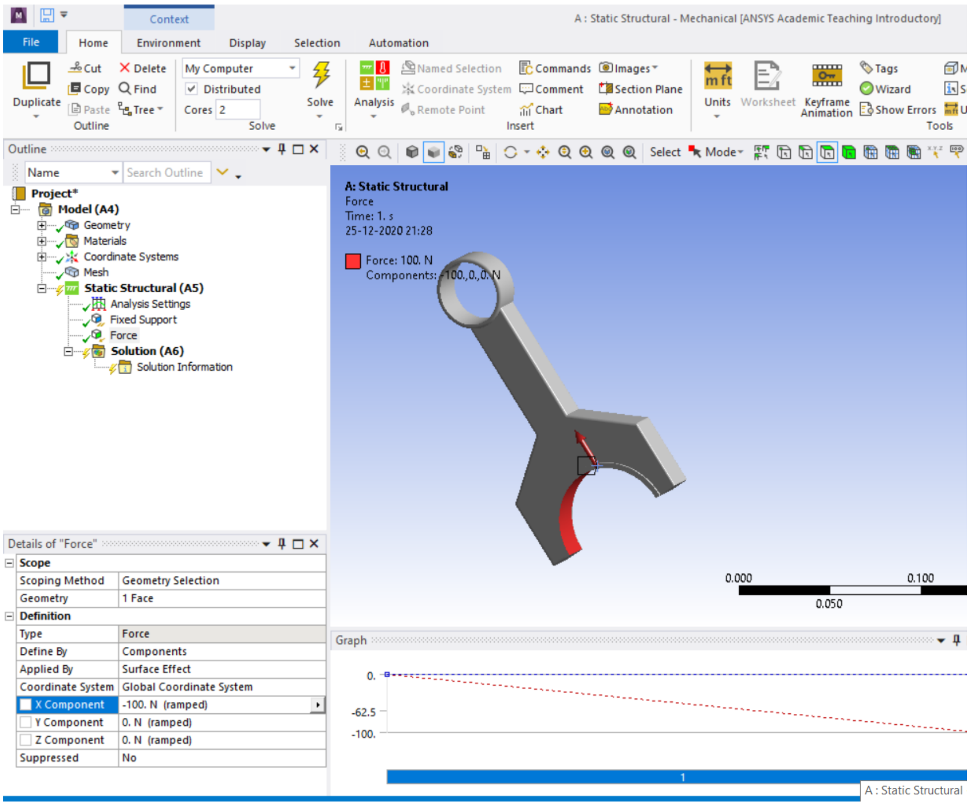
Mesh



Boundary Condition



Boundary Condition



- 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 ?
- How can FEA be used to optimize the design .

EXPERIMENT - 07

STATIC ANALYSIS OF CURVED SHELL DUE TO INTERNAL PRESSURE

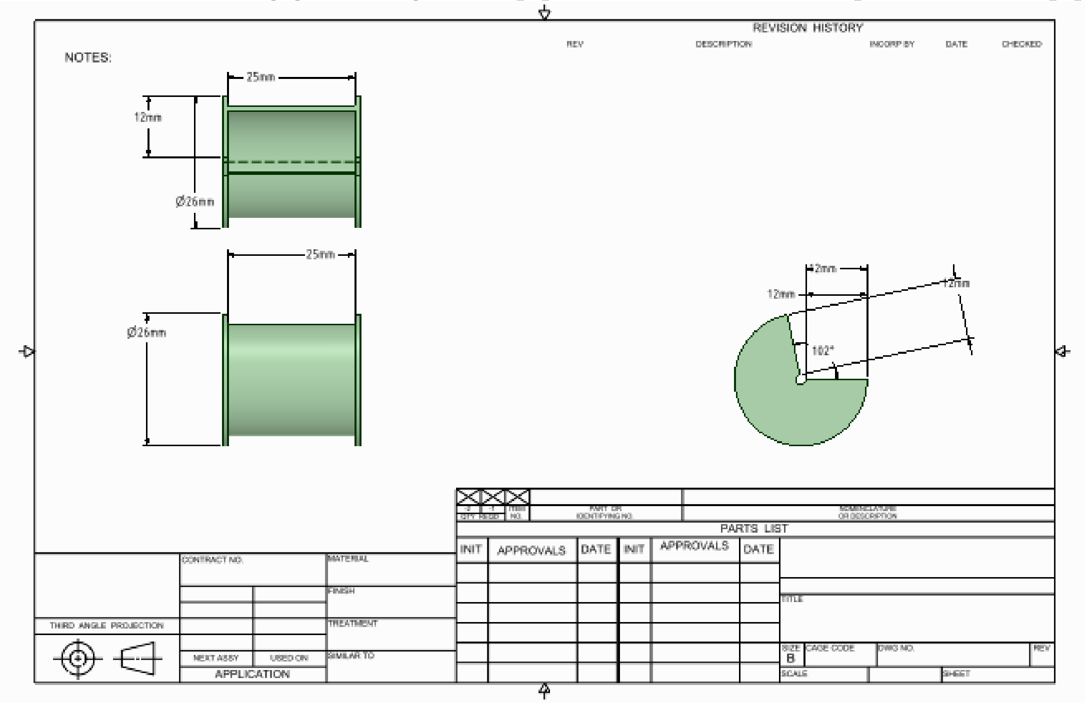
AIM:

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

SOFTWARE: ANSYS

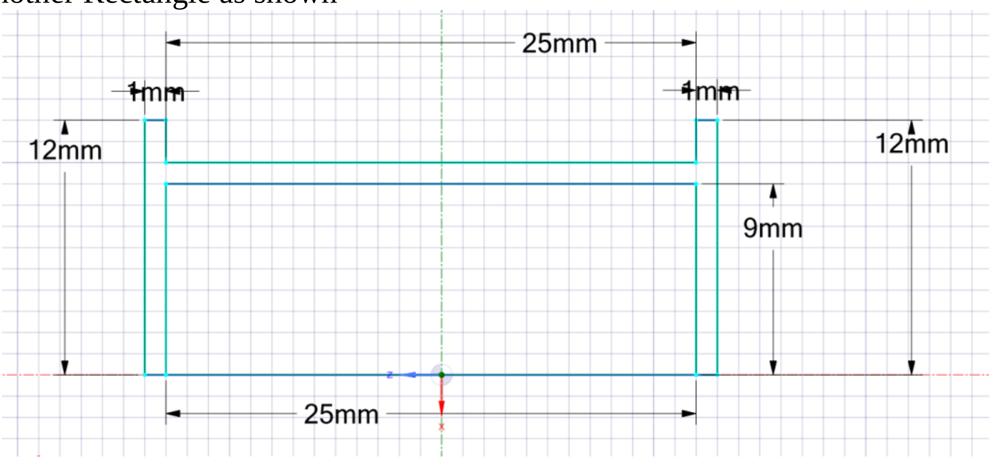
THEORY:

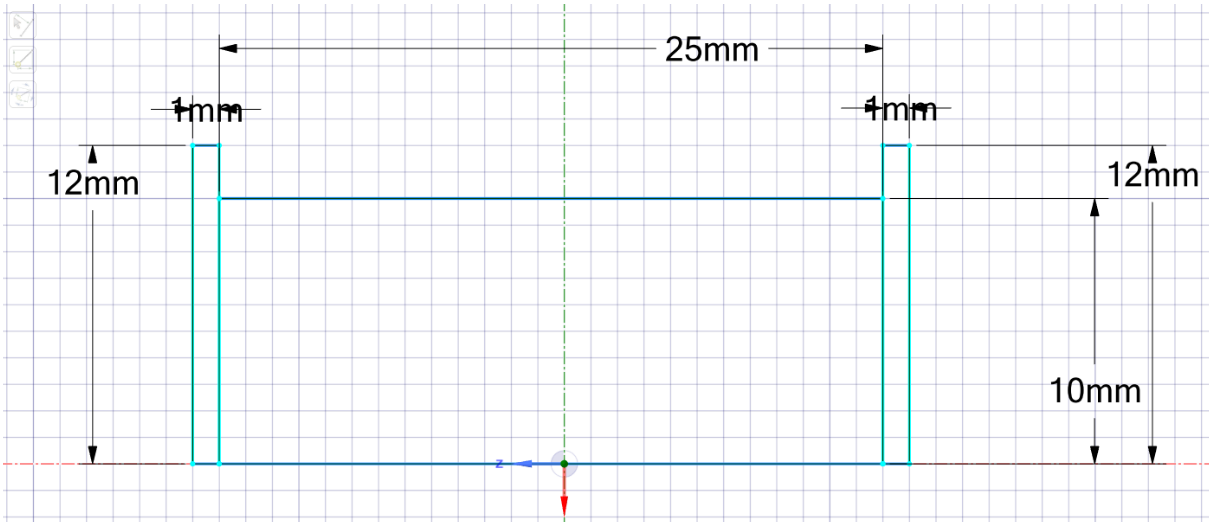
From the following given diagram of pipe calculate the internal pressure in the pipe.

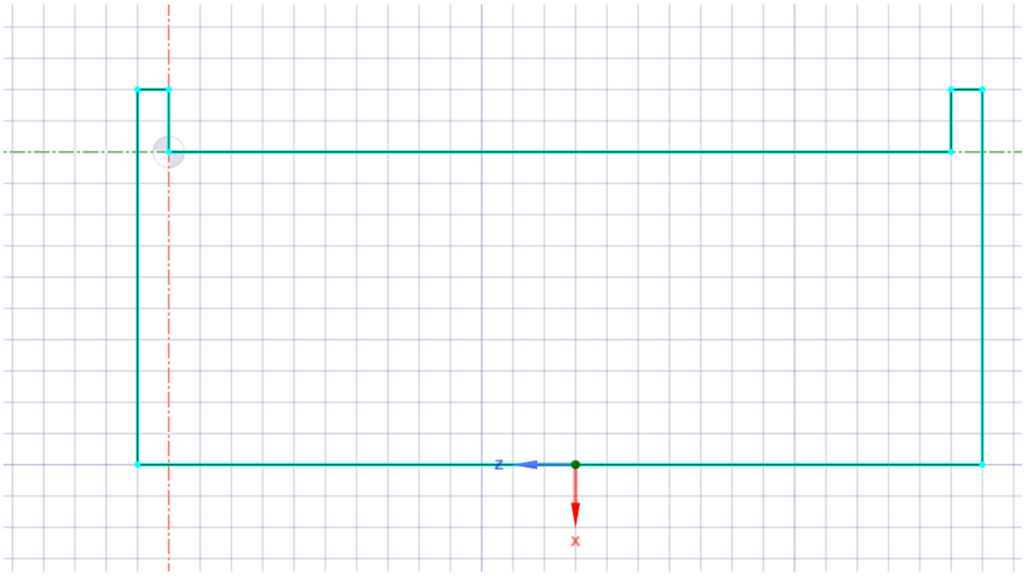


PROCEDURE:

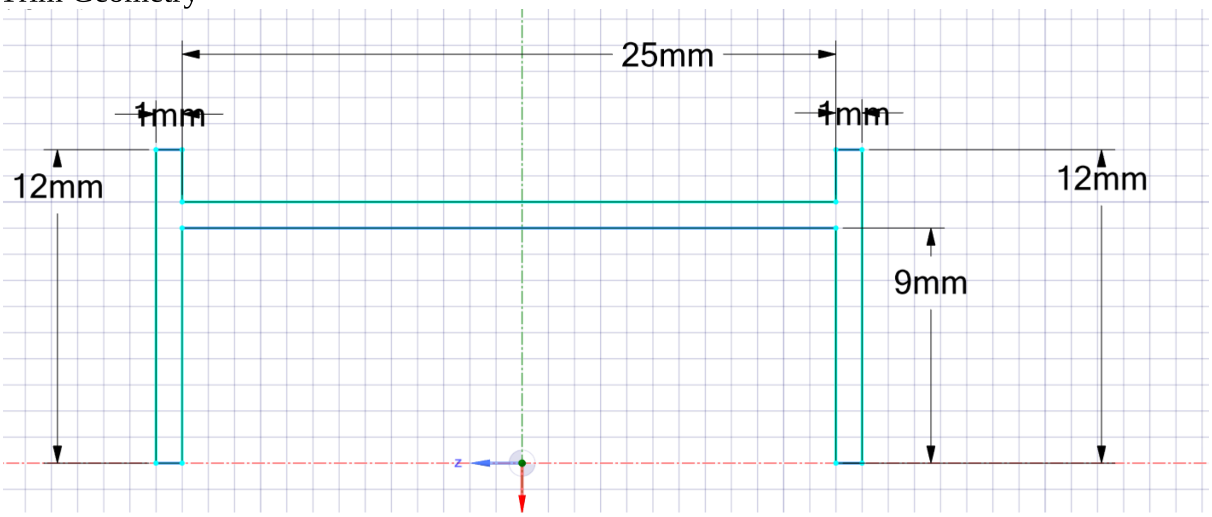
Creating Geometry
Trim Unwanted Lines :-
Draw Another Rectangle as shown



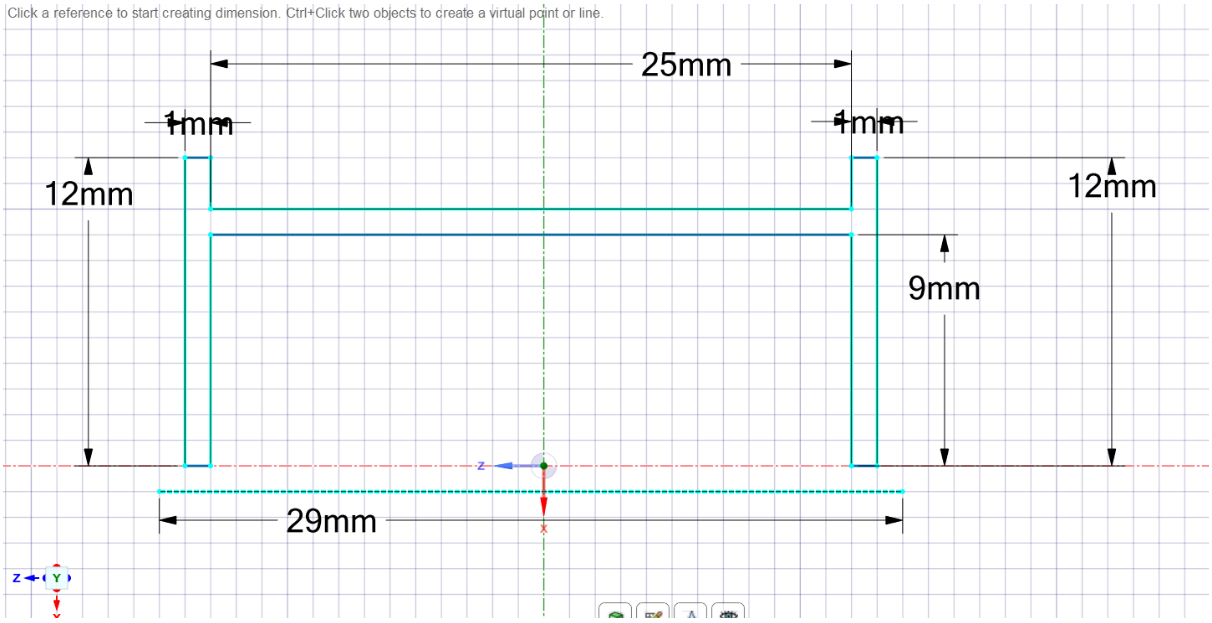




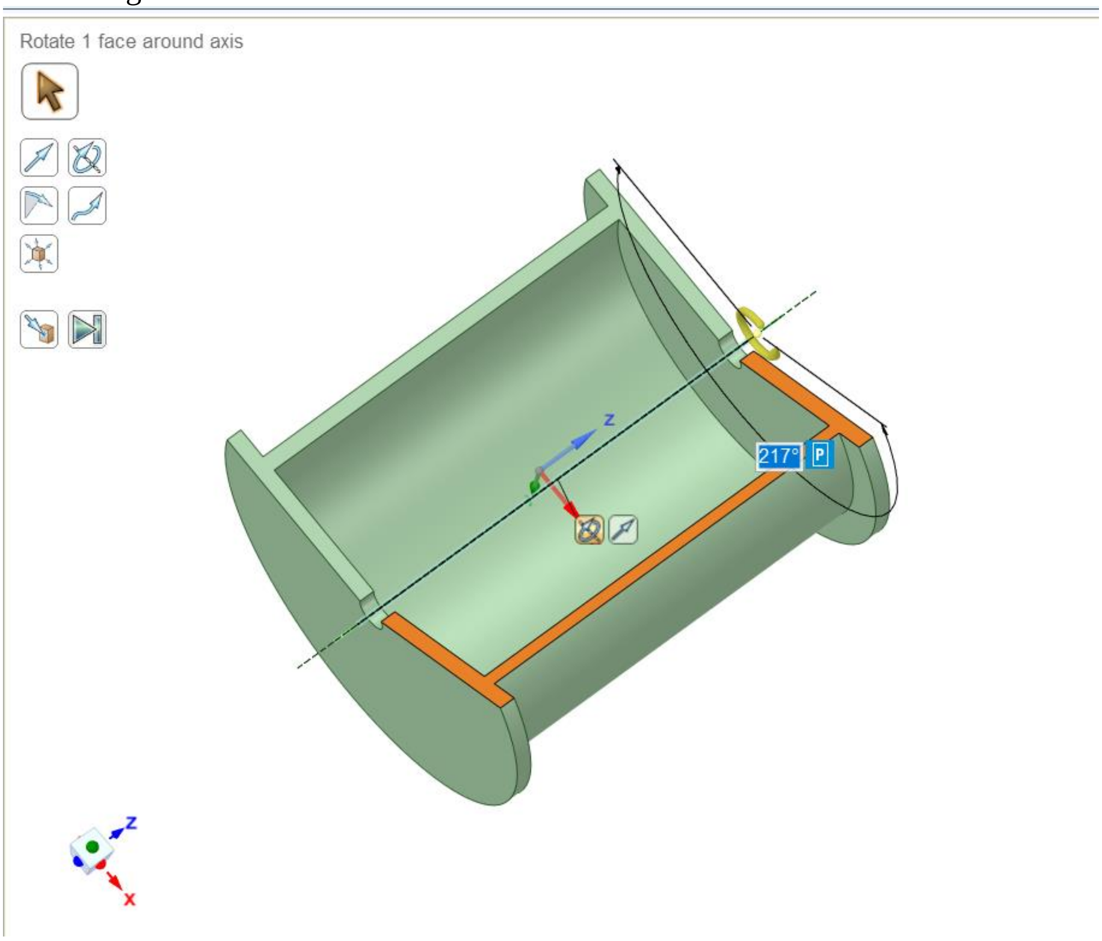
Trim Geometry



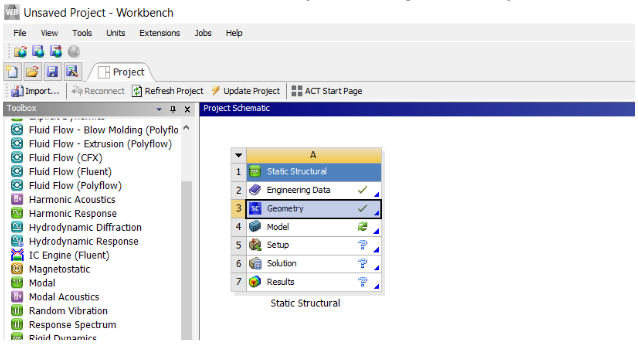
Create Construction Line



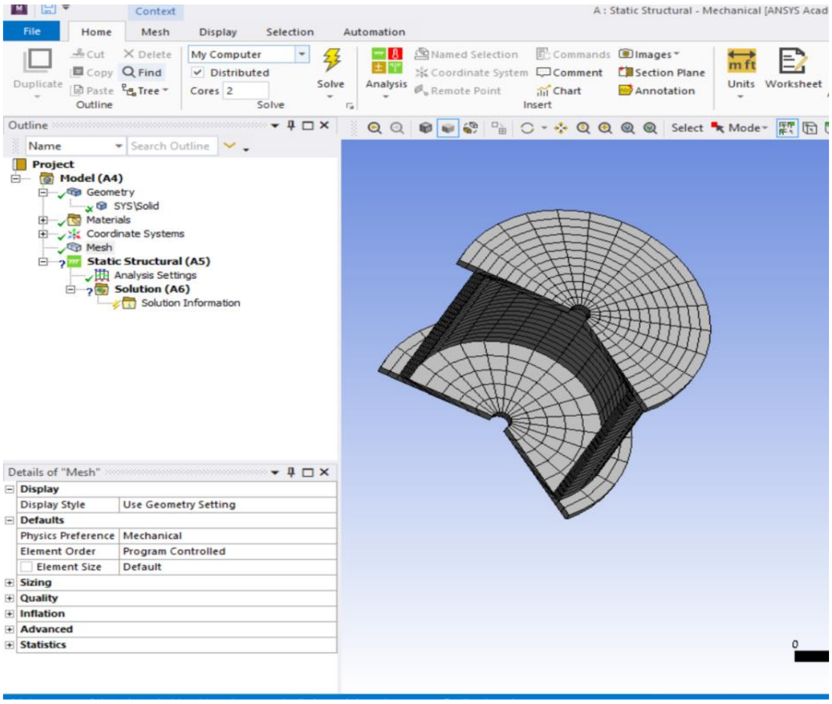
Revolve Along



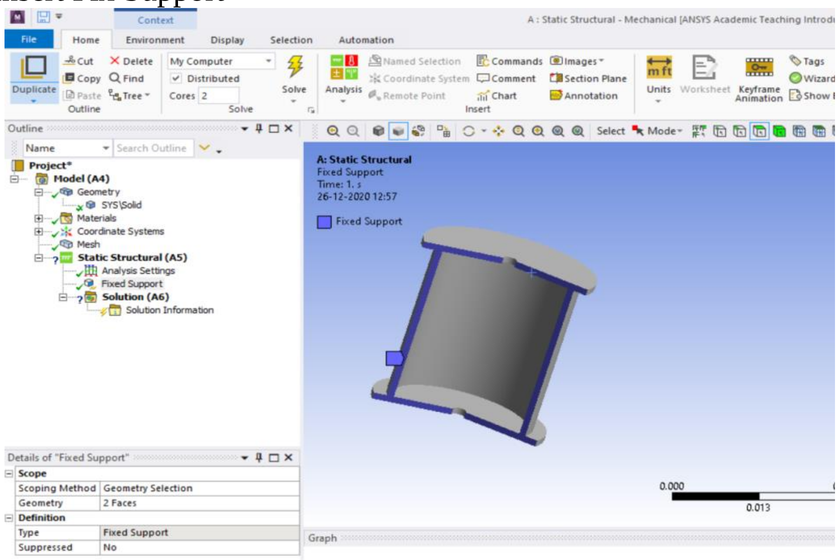
Right Click Model Update it will automatically Take geometry as Steel



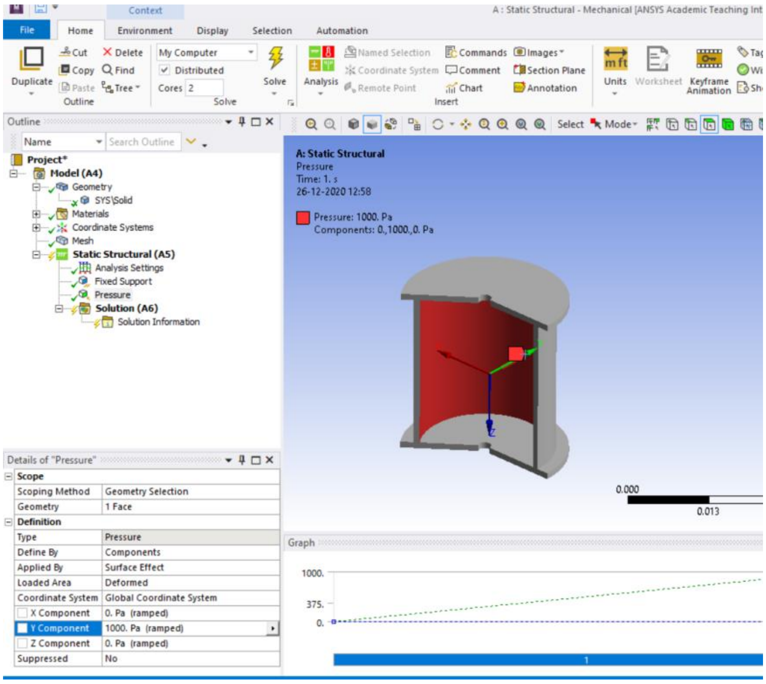
Right Click Mesh



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

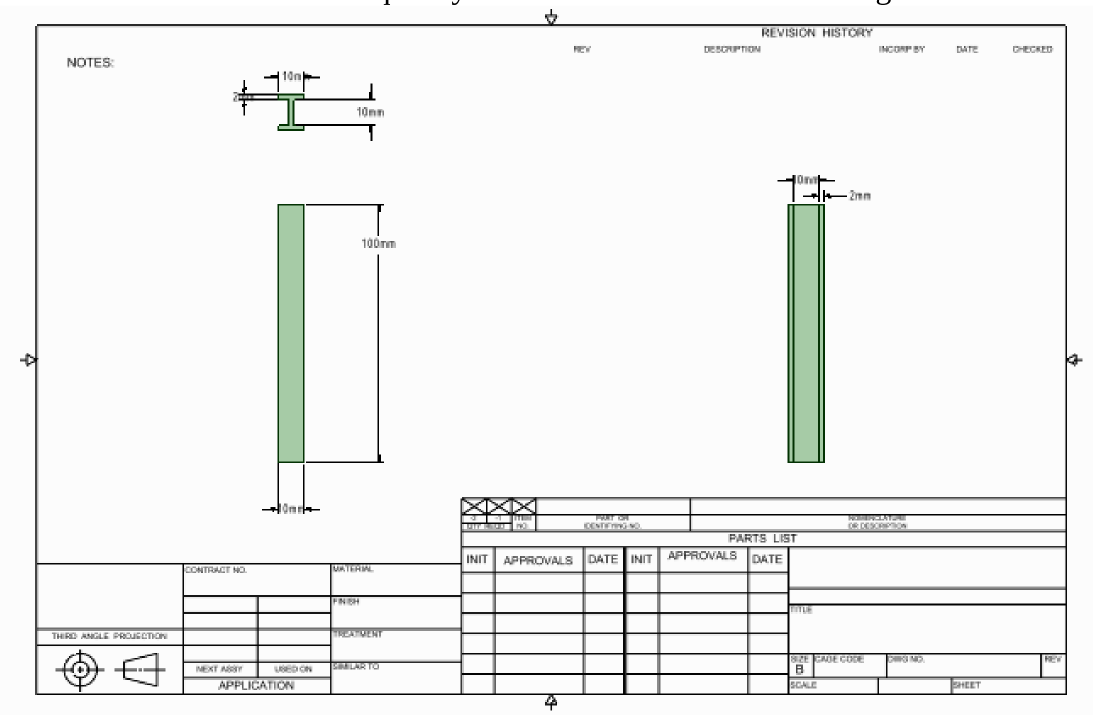
AIM:

To determine Natural frequency and modal analysis of beam.

SOFTWARE: ANSYS

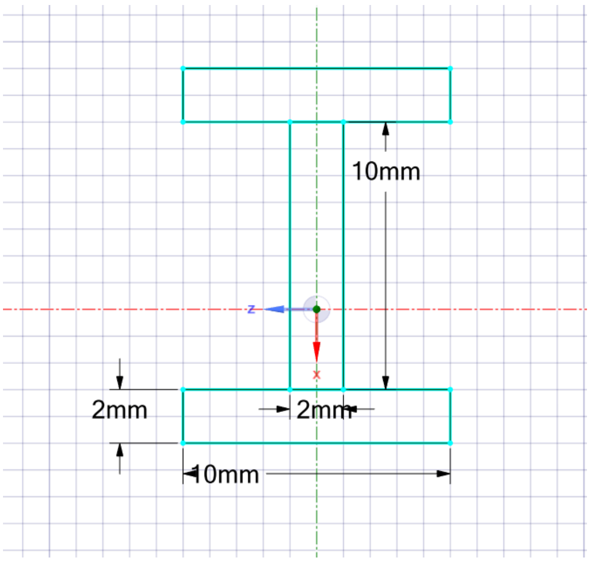
THEORY:

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

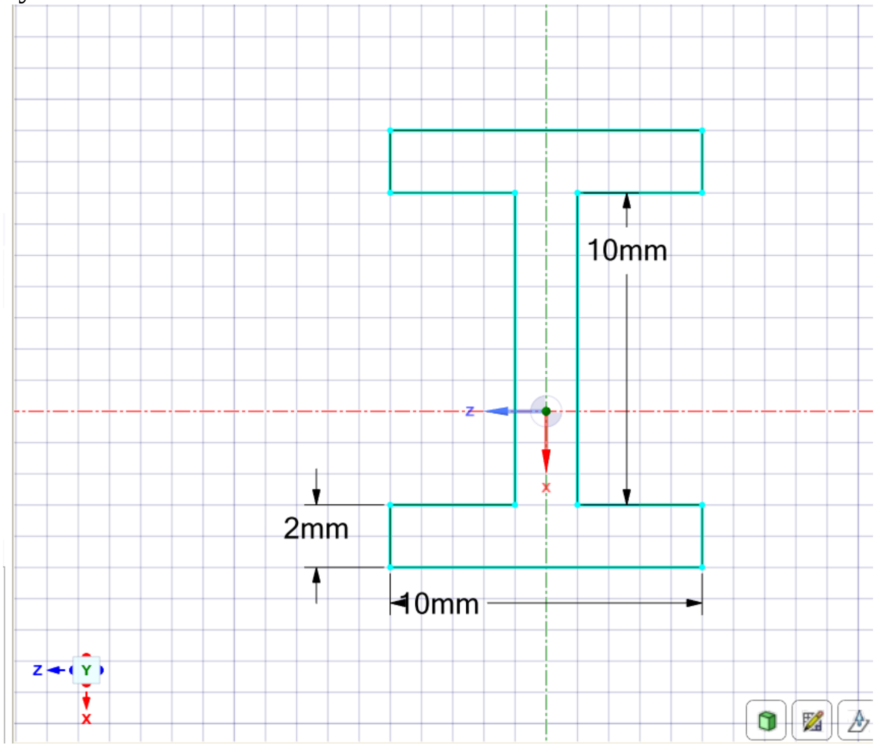


PROCEDURE:

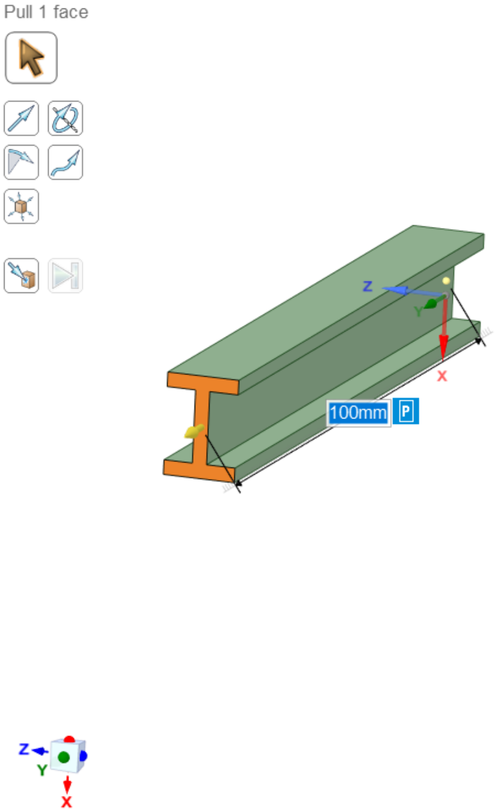
Create Geometry



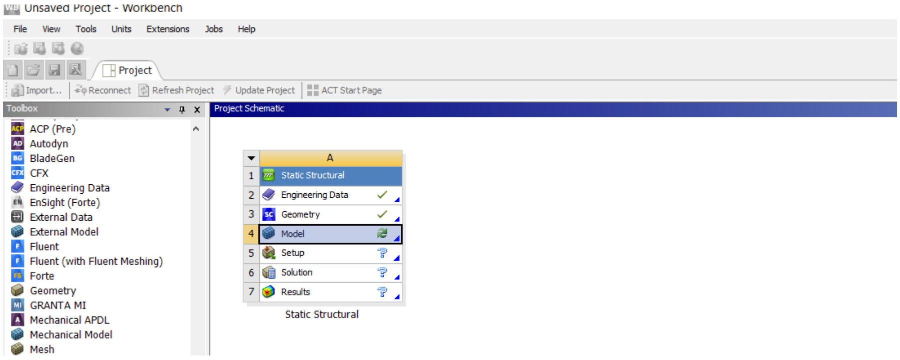
Trim Geometry



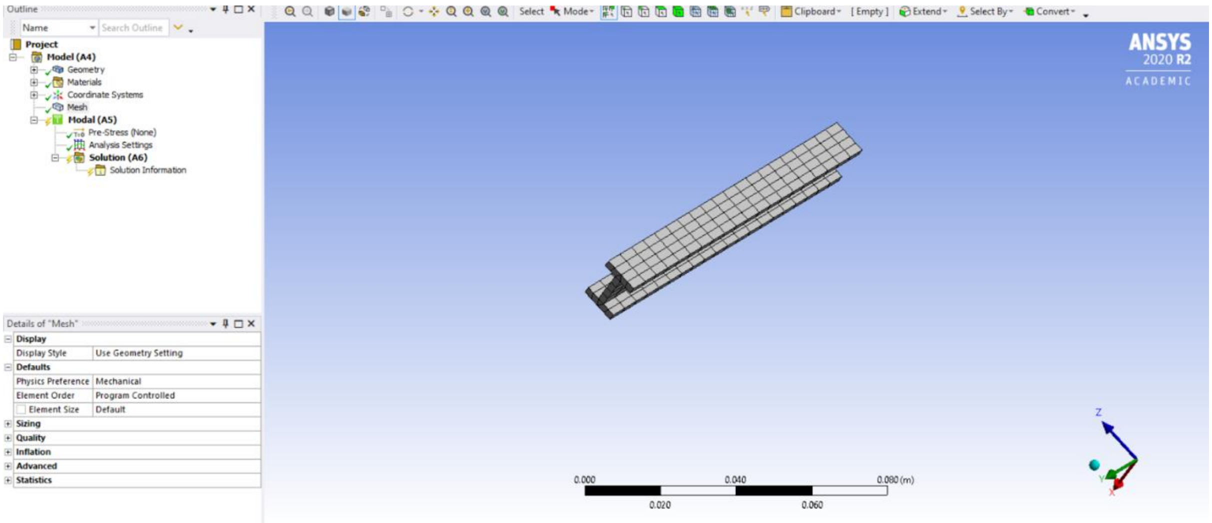
Extrude up to 100mm



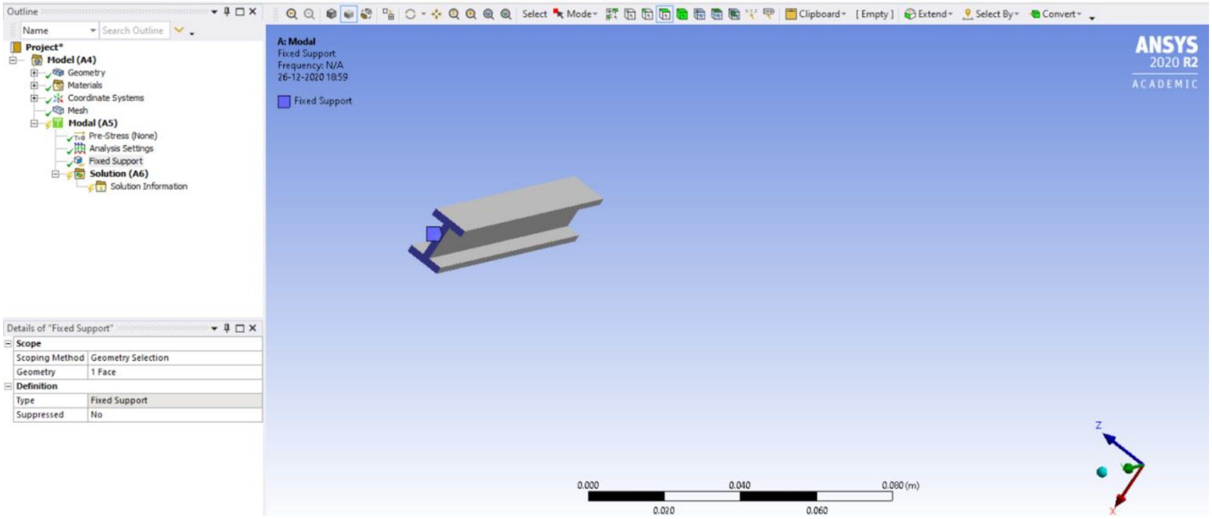
Update Model – Default Structural Steel



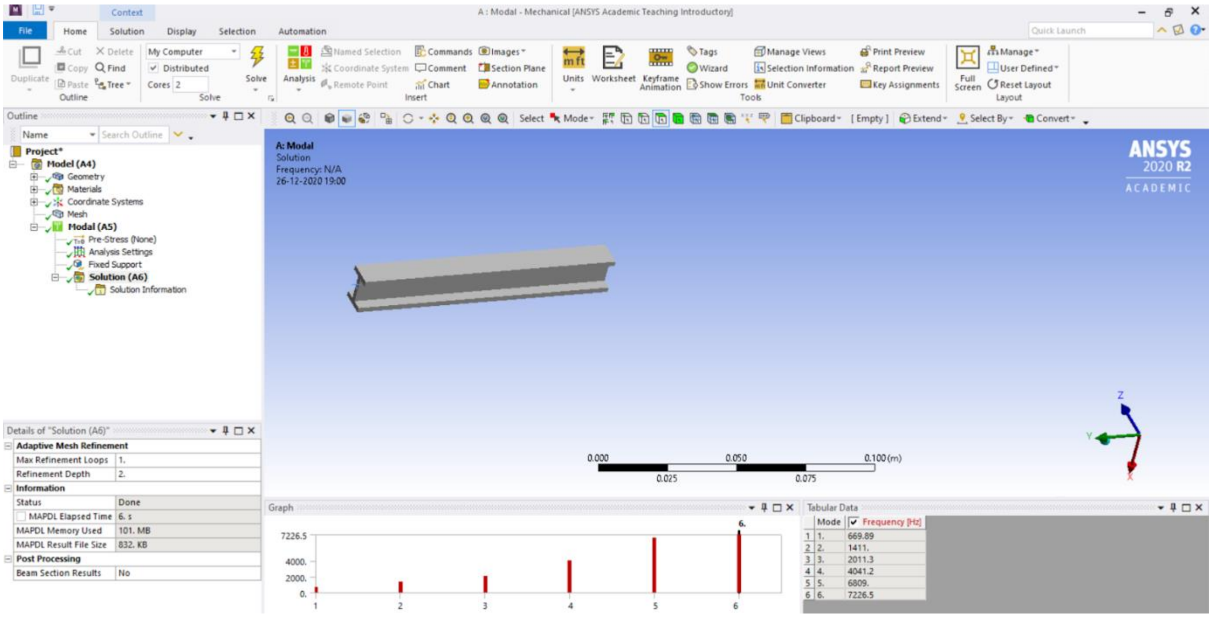
Mesh



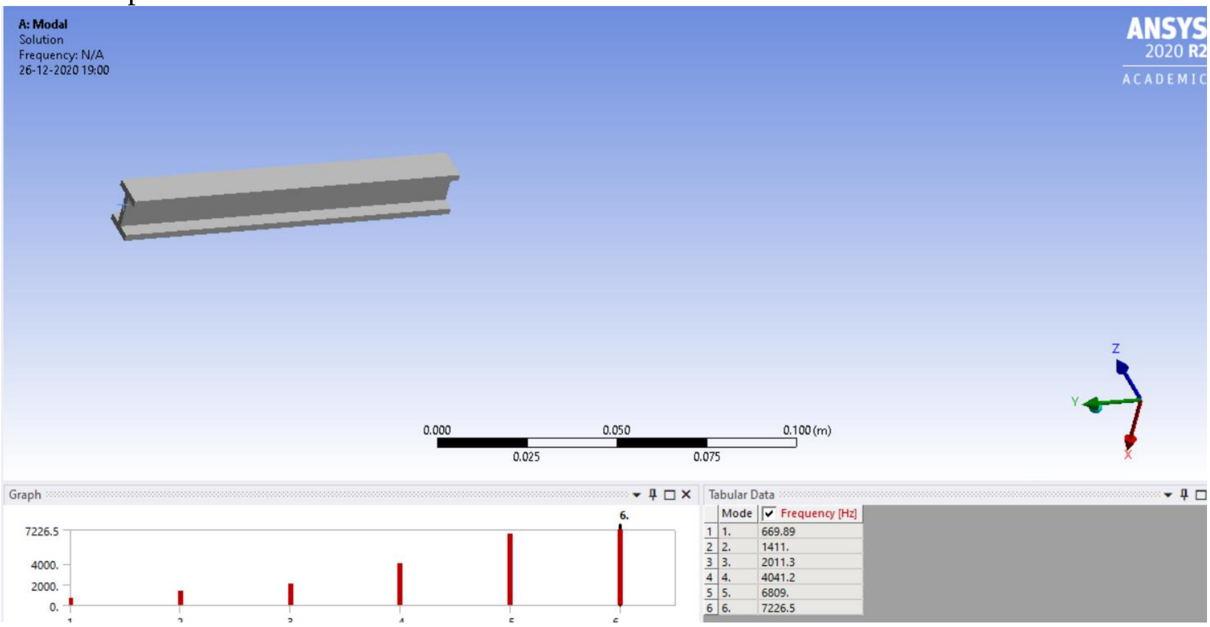
Insert Fix Support



Insert Solution



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

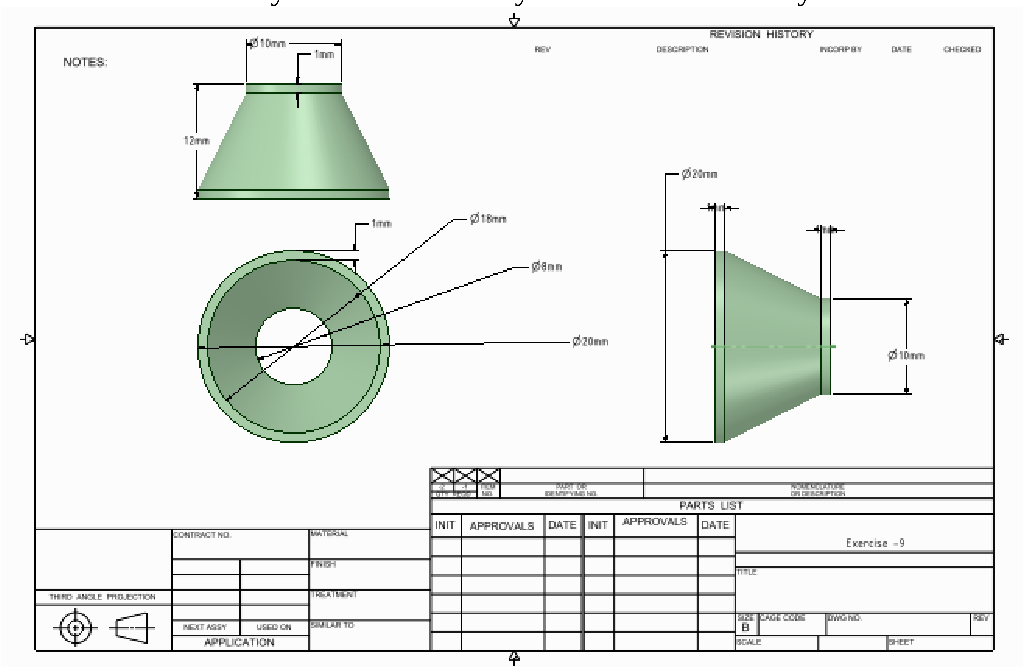
AIM:

Steady state heat transfer Analysis Cross section of chimney.

SOFTWARE: ANSYS

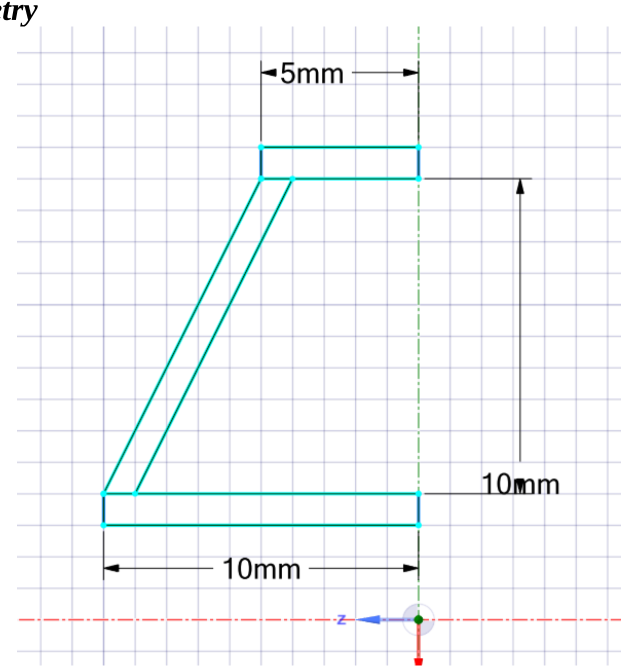
THEORY:

Determine The steady heat transfer analysis in case of chimney

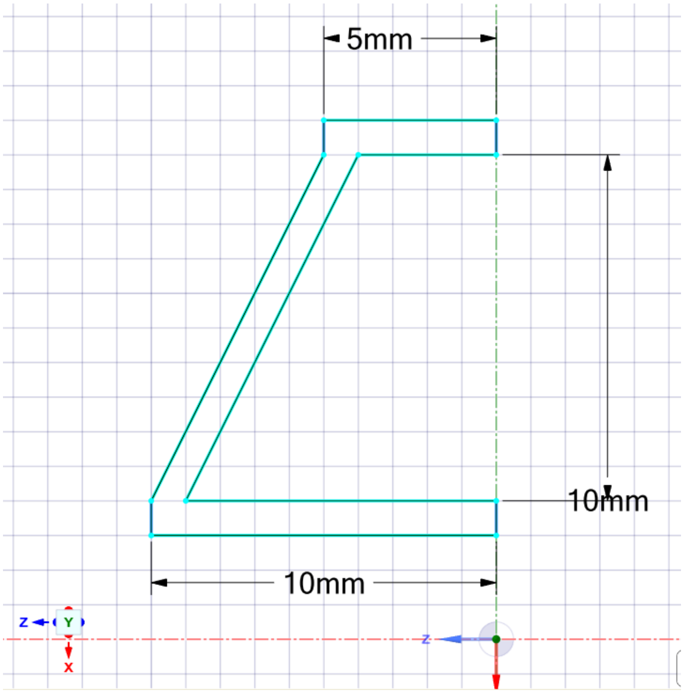


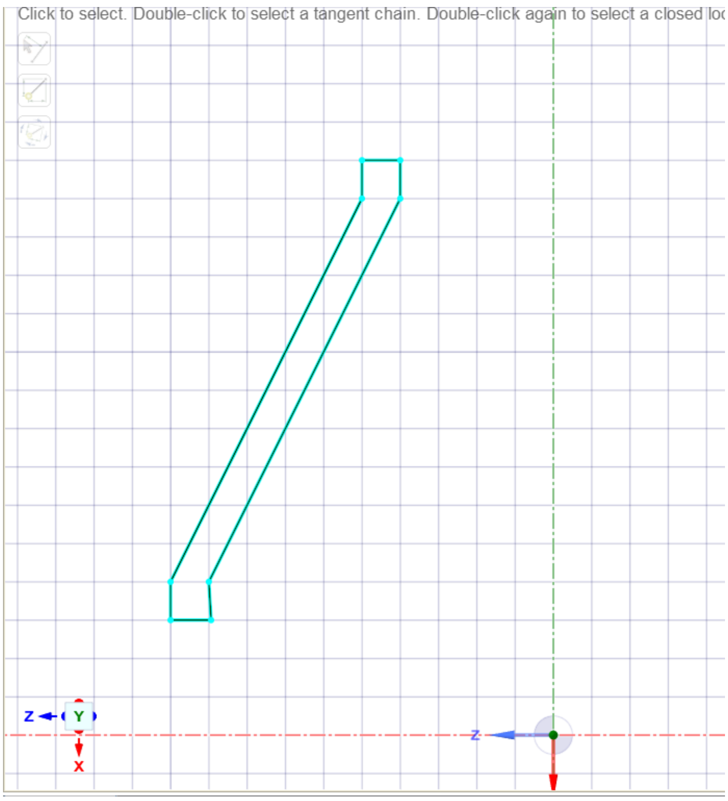
PROCEDURE:

Step -1 Create Geometry

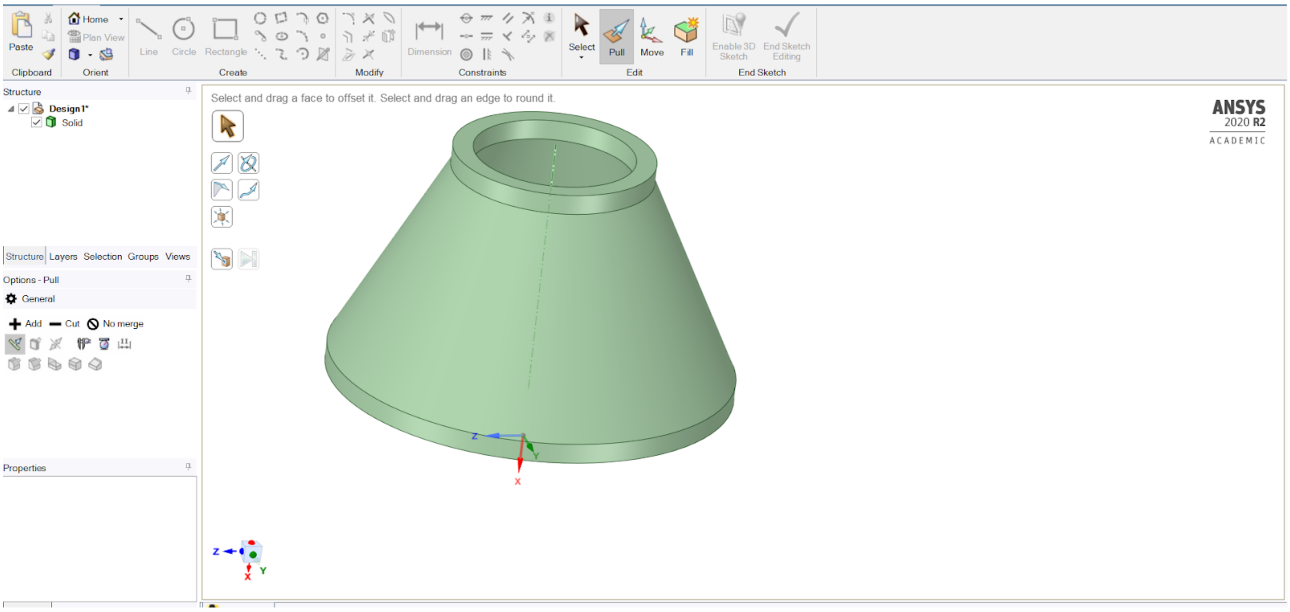


Trim Geometry

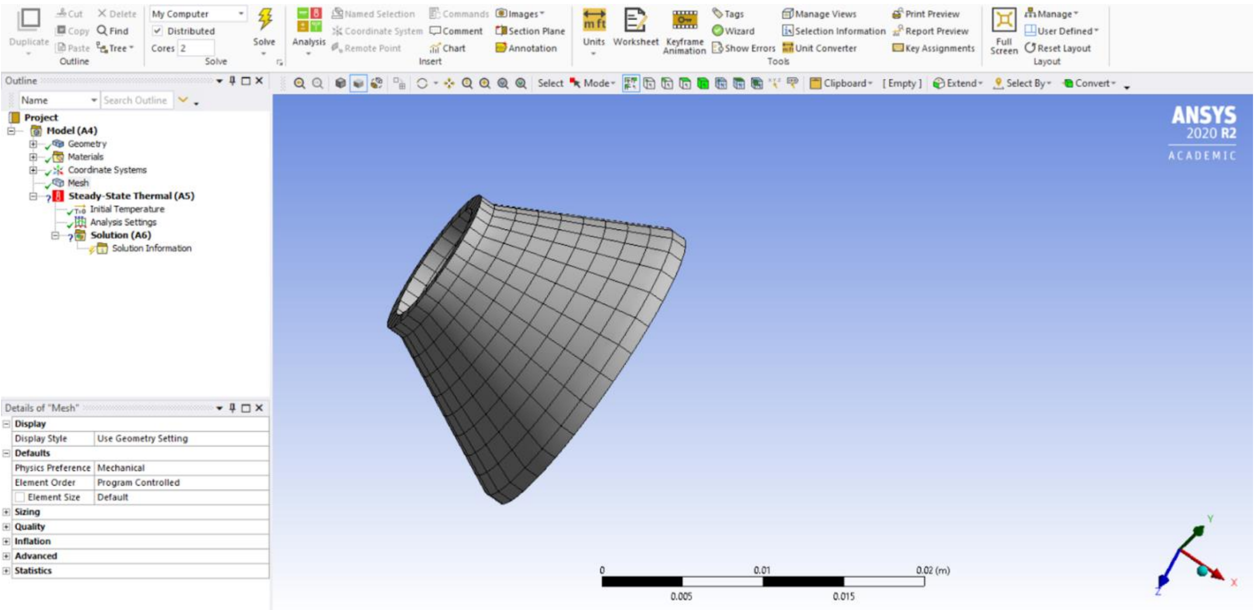




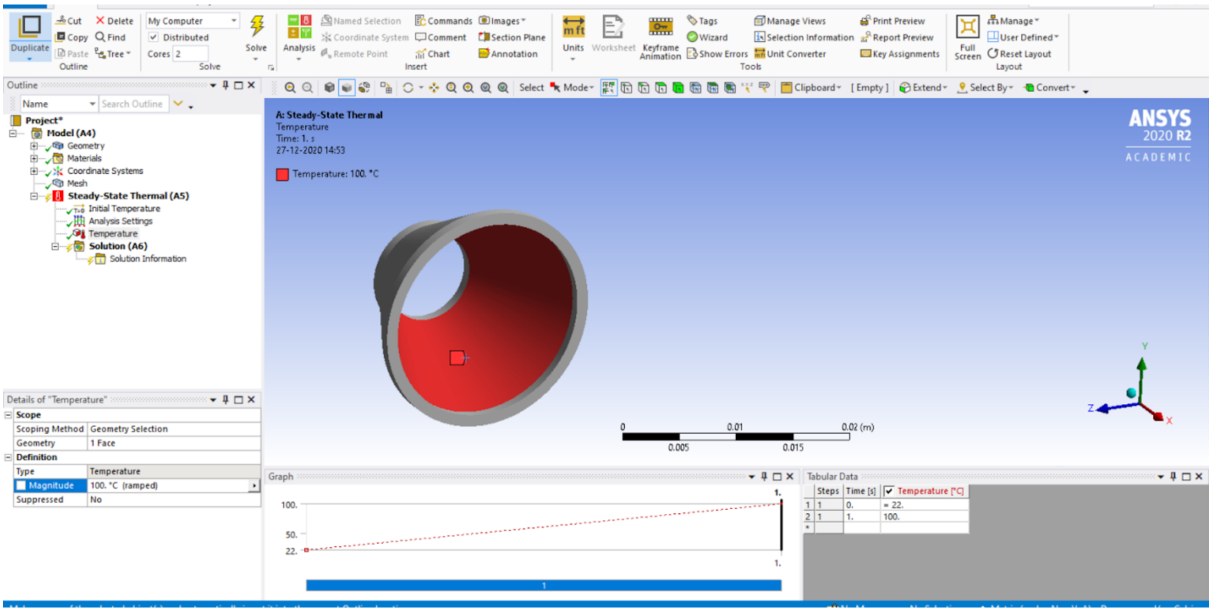
Revolve Pull



Update Mesh



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

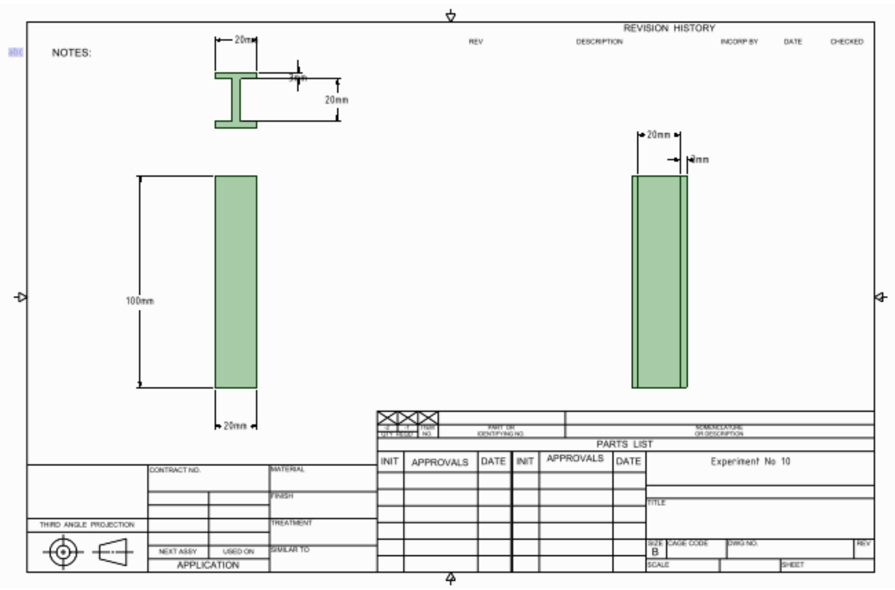
AIM:

To determine deformation in case of cantilever beam with nonlinear material

SOFTWARE: ANSYS

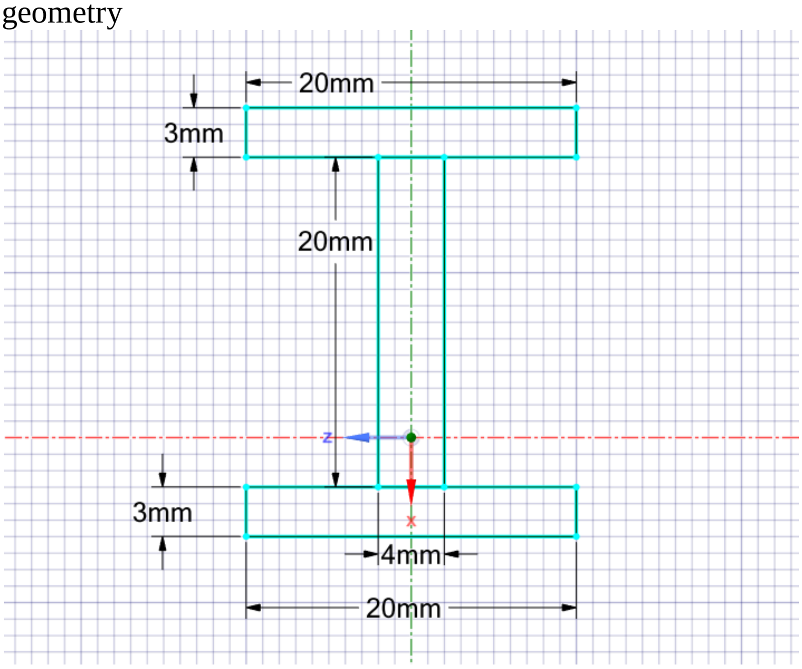
THEORY:

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

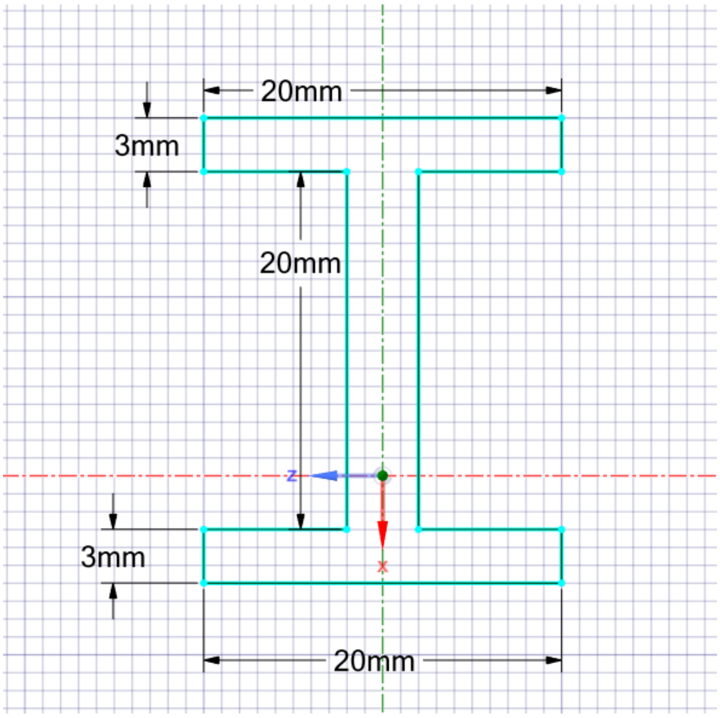


PROCEDURE:

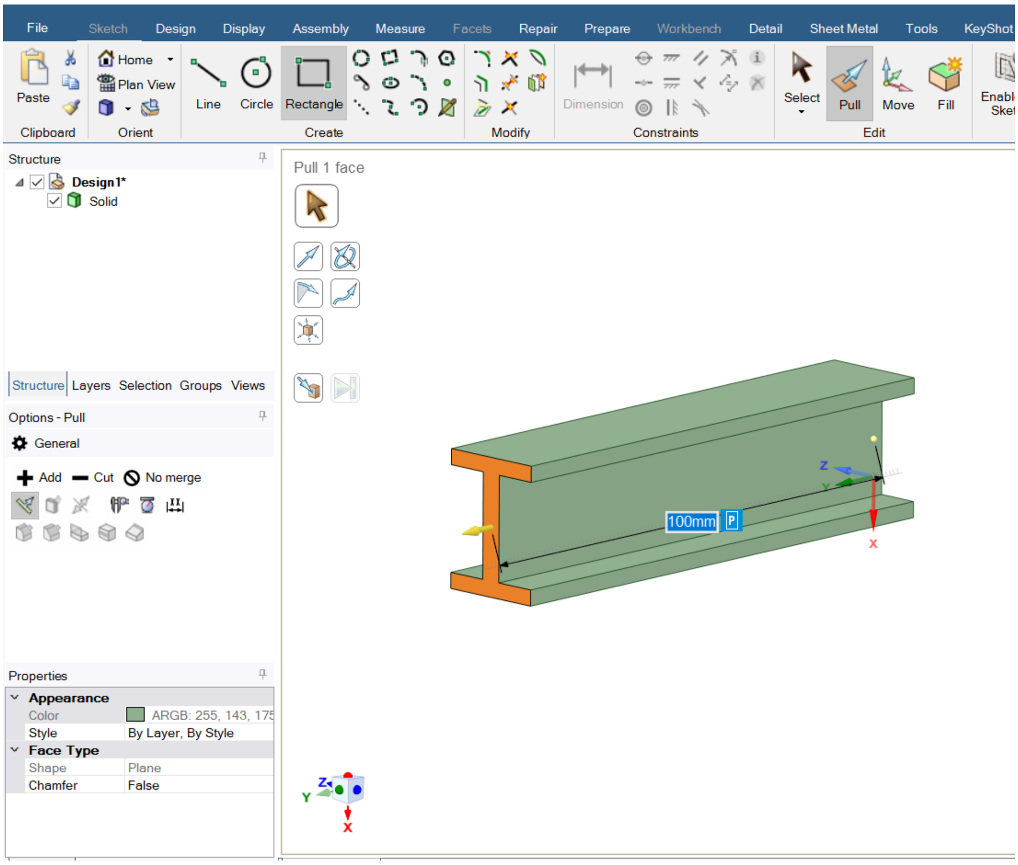
Step 1 – Create geometry



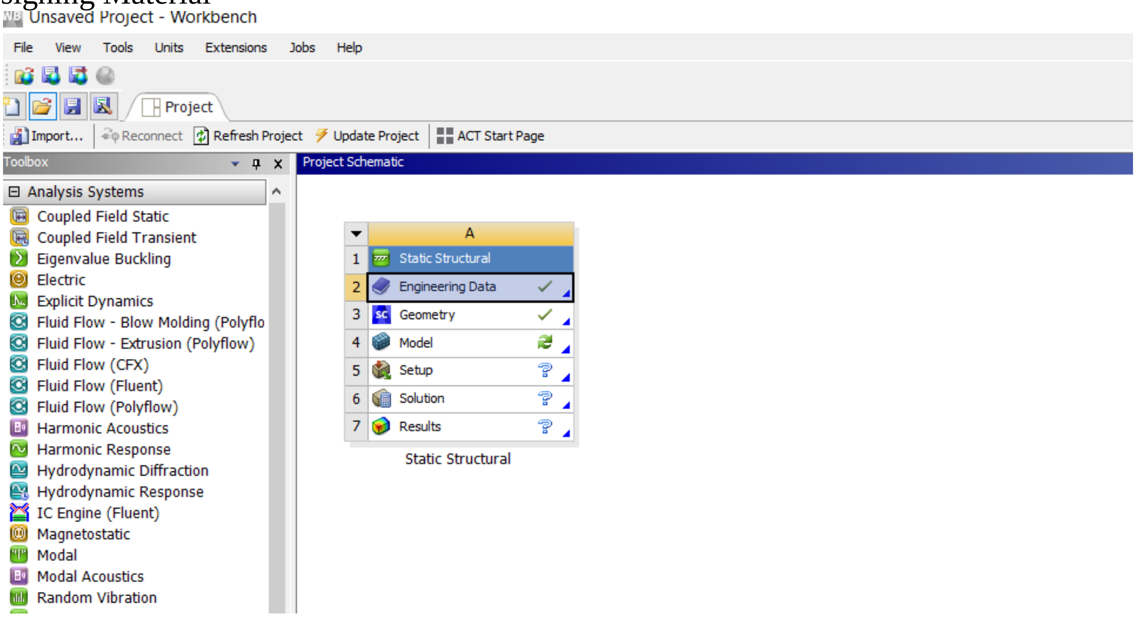
Trimming Geometry



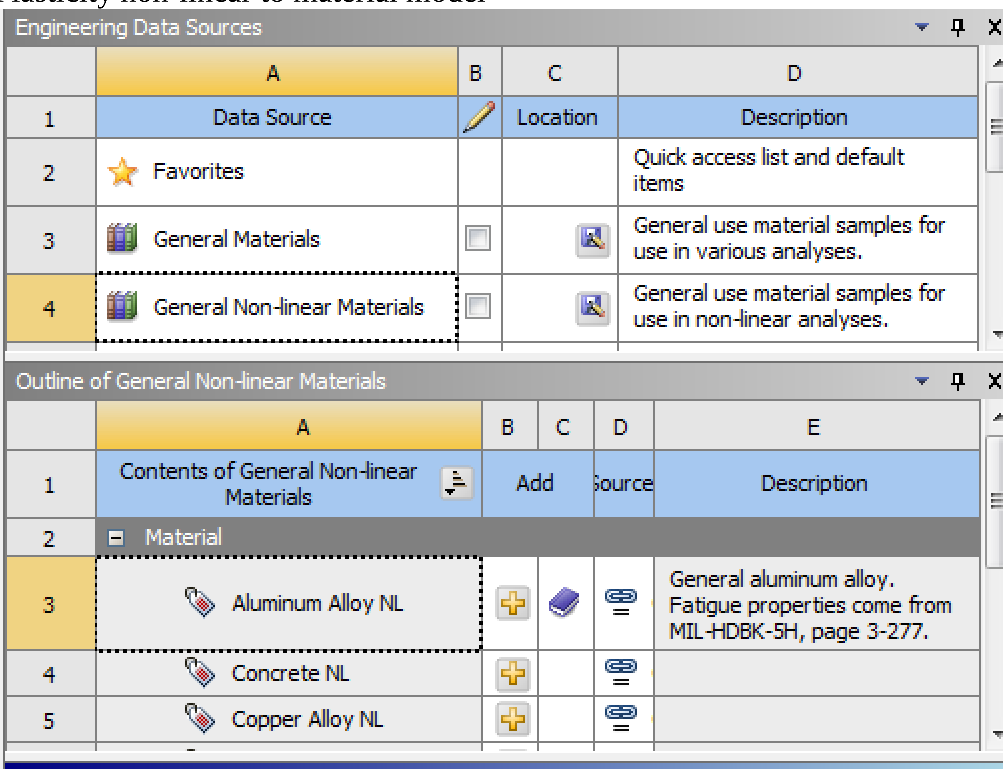
Extruding Geometry 100 mm



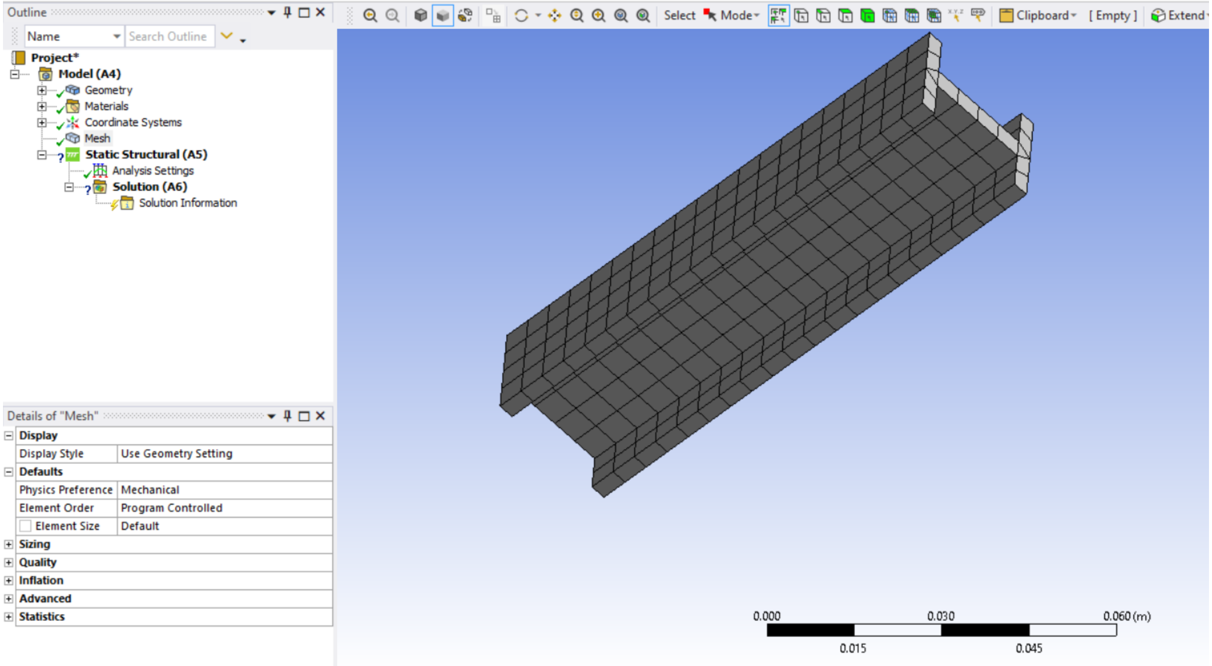
Assigning Material



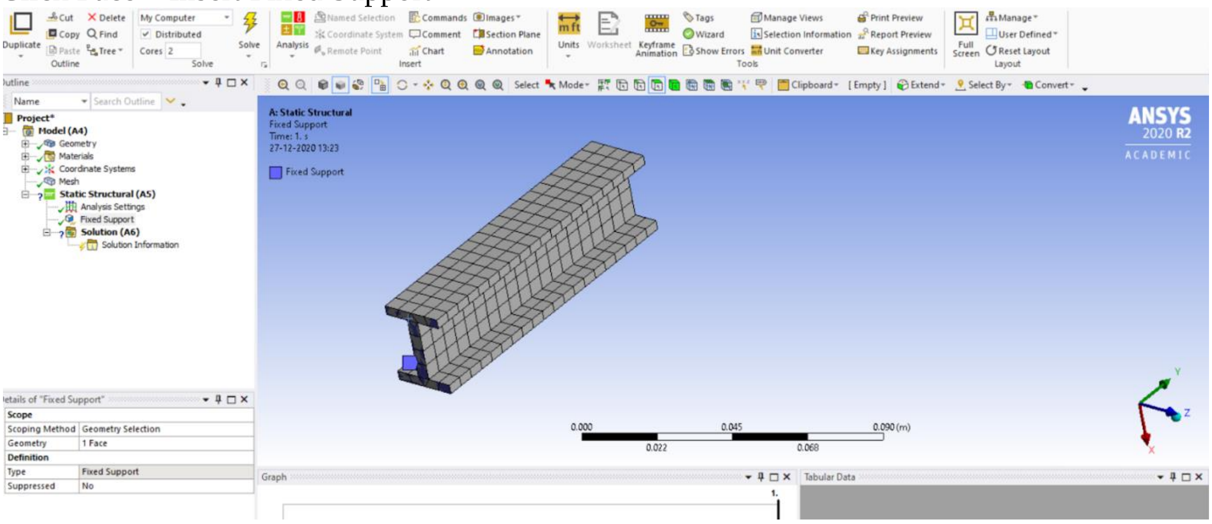
Assign Plasticity non-linear to material model



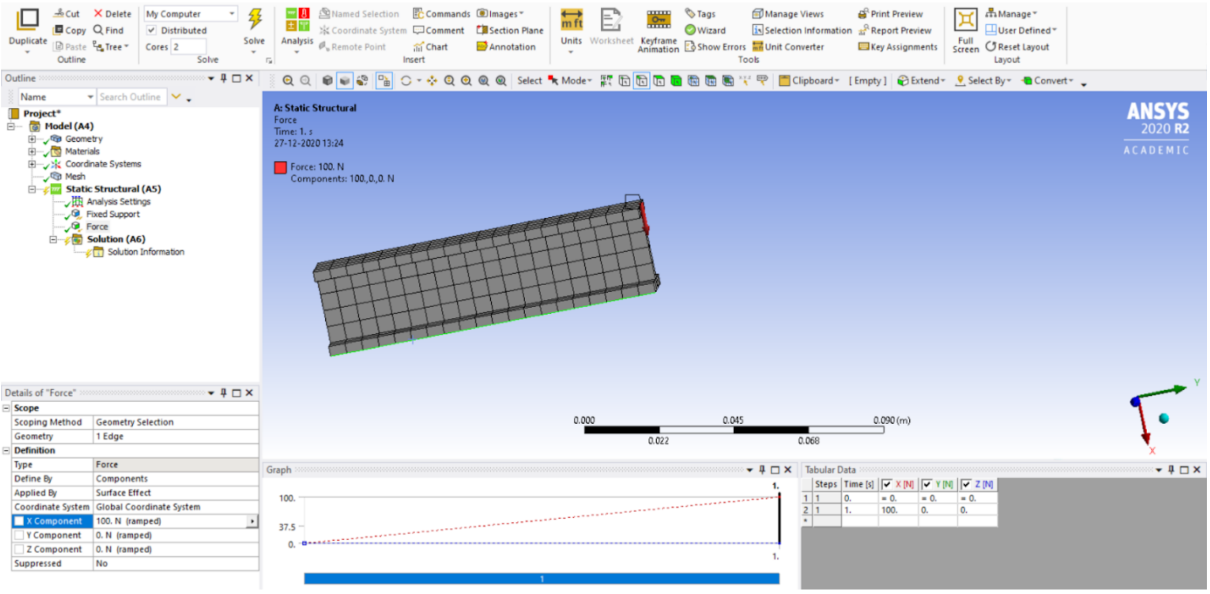
Mesh



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

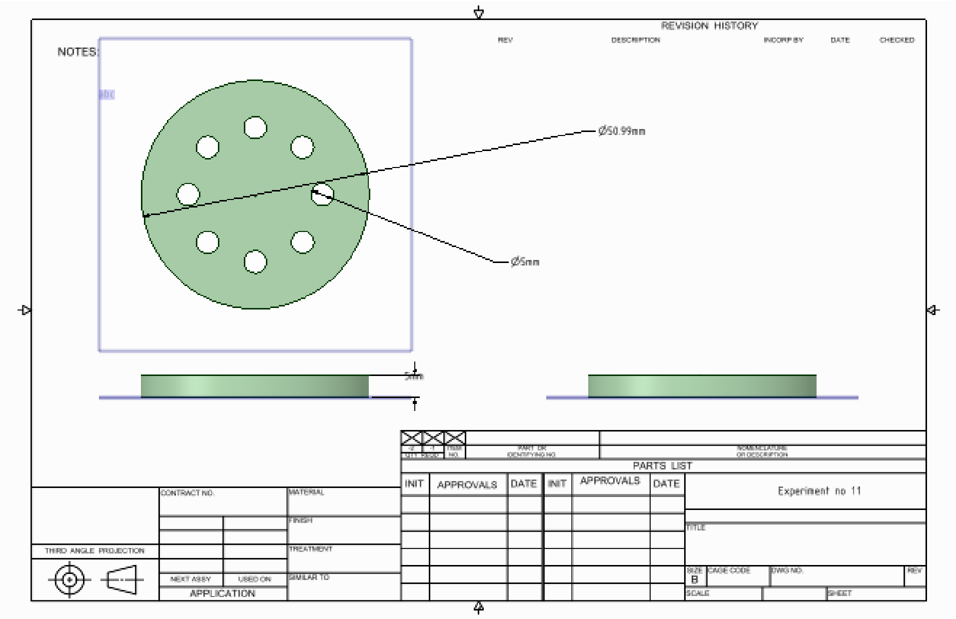
AIM:

To determine total deformation and temperature distribution using coupled field analysis

SOFTWARE: ANSYS

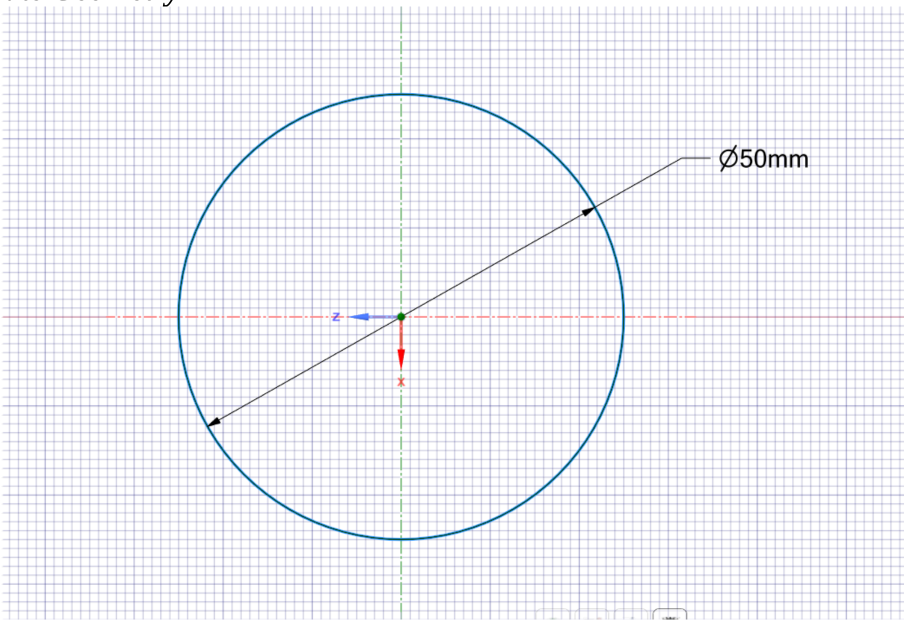
THEORY:

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

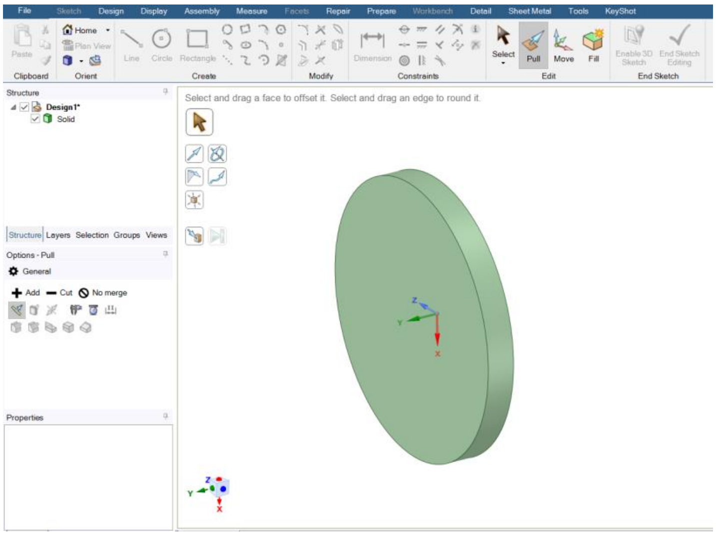


PROCEDURE:

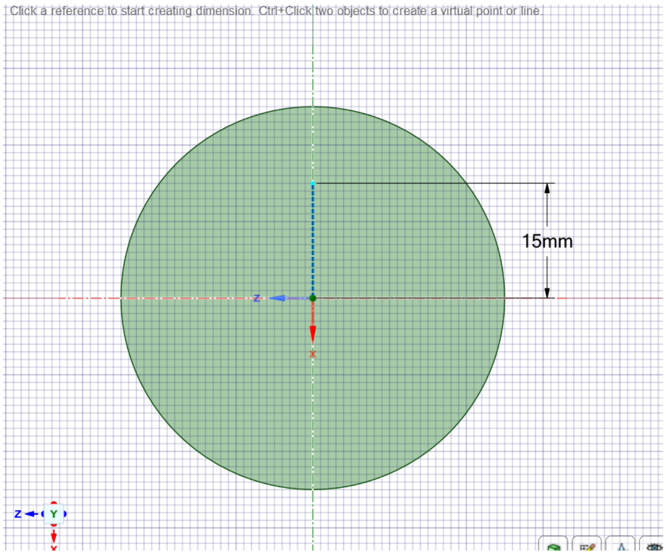
Step 1 - Create Geometry



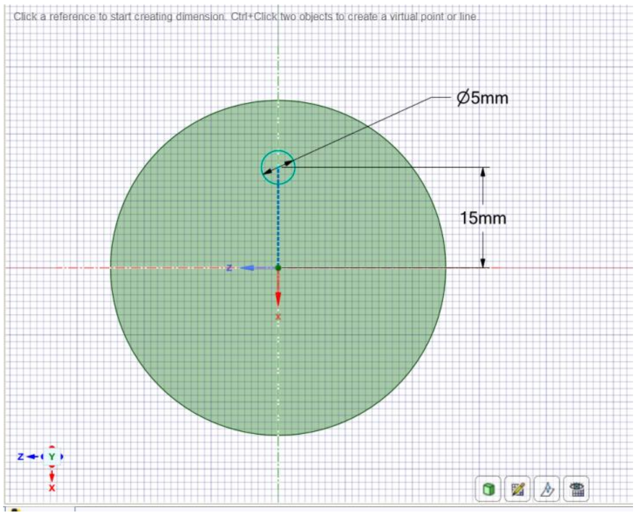
Extrude :- 5mm



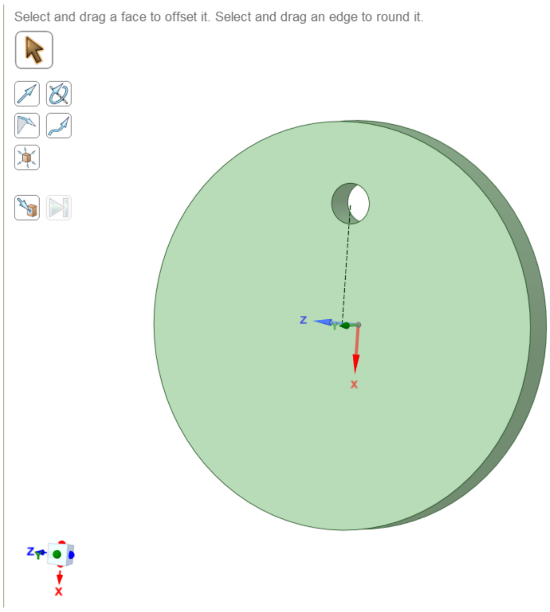
Creating Construction line up to 15mm



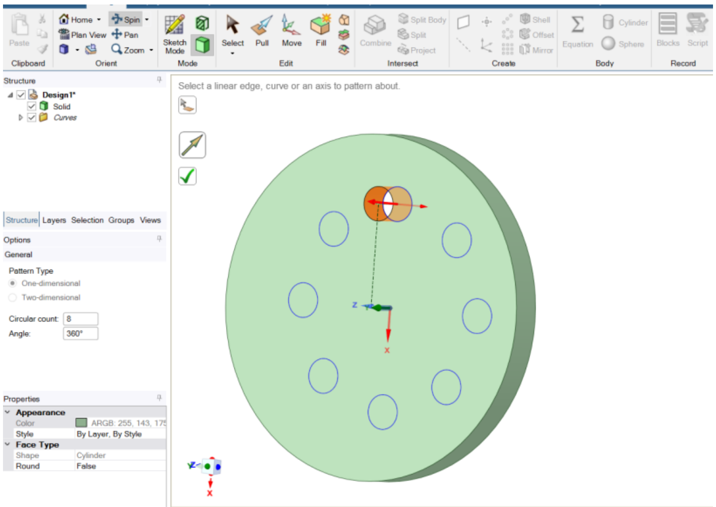
Creating a hole of 5mm at a distance 15 mm

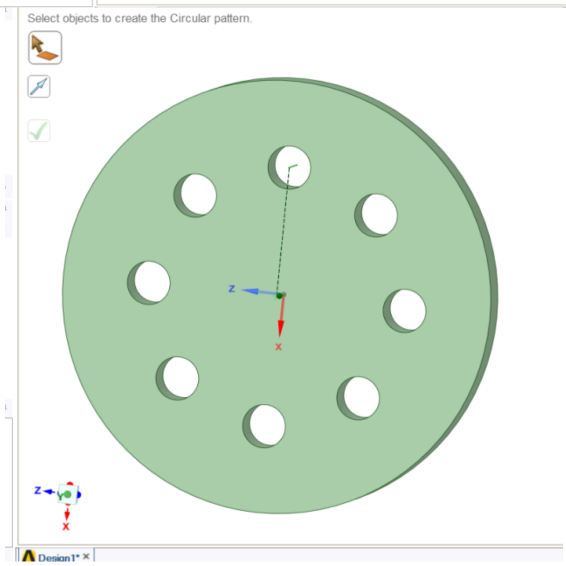


Creating a hole

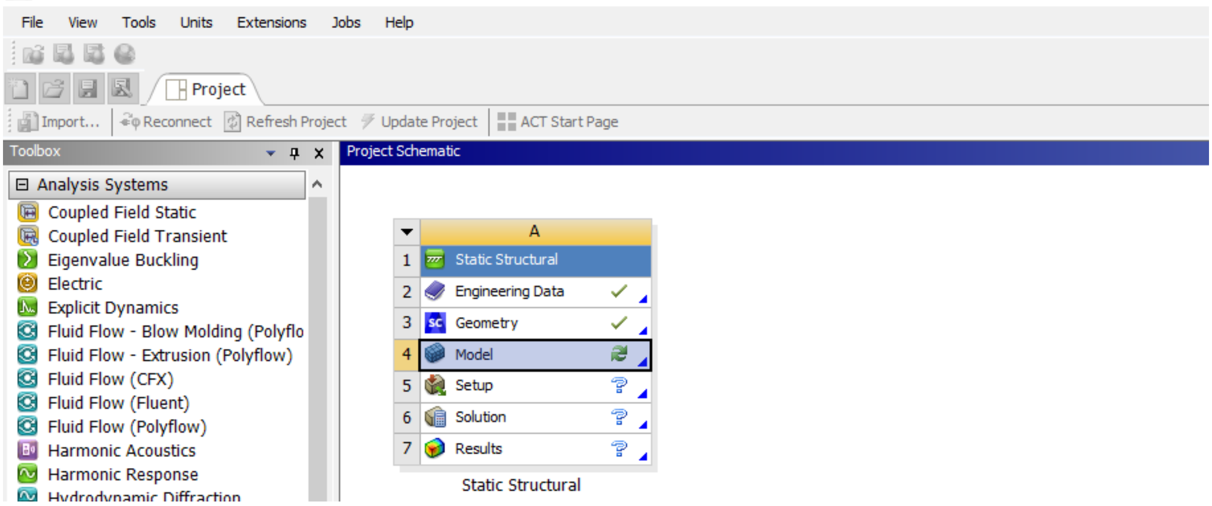


Creating Circular Pattern of Hole

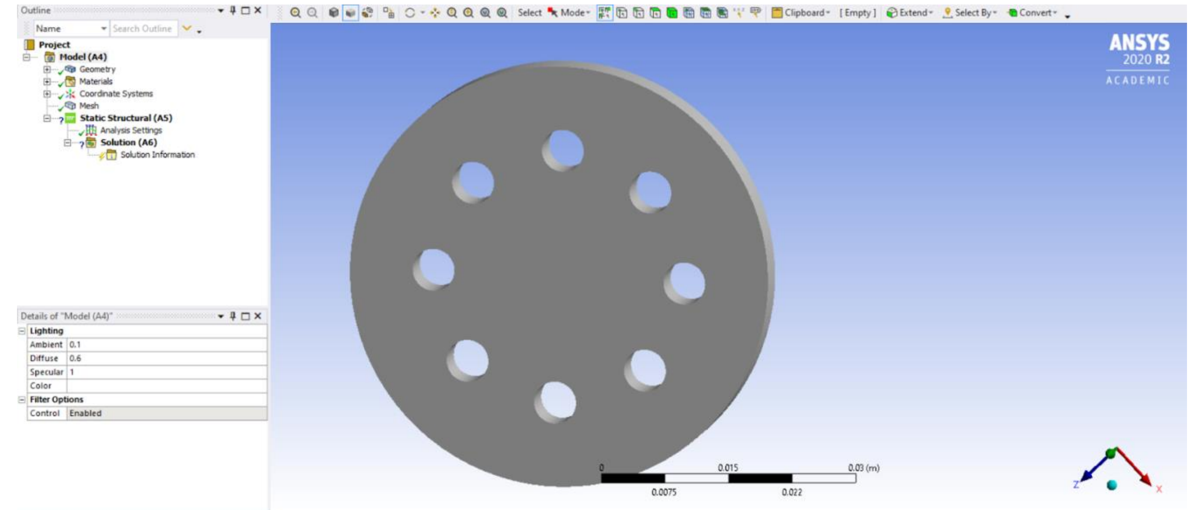




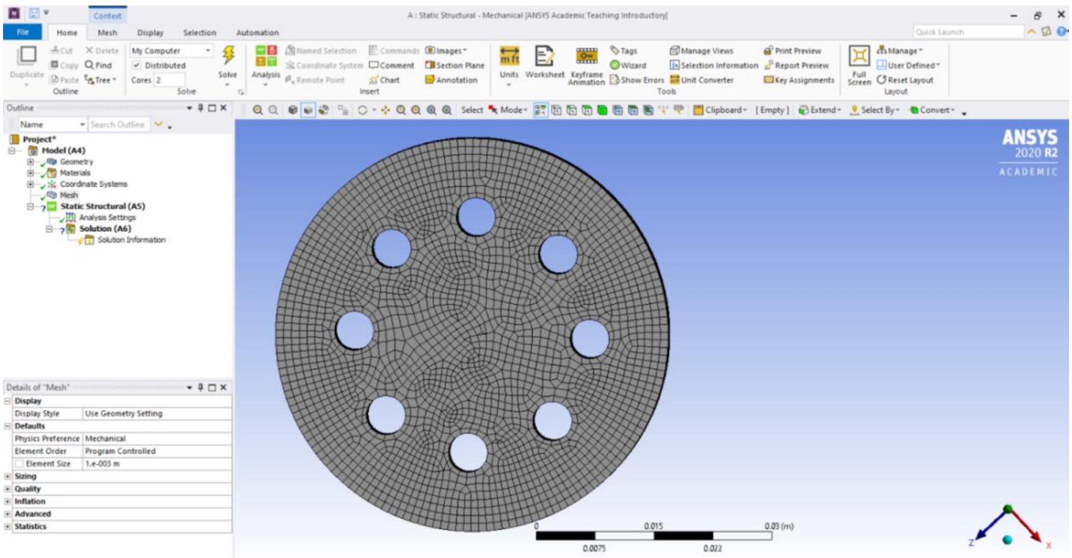
Updating the model in the geometry



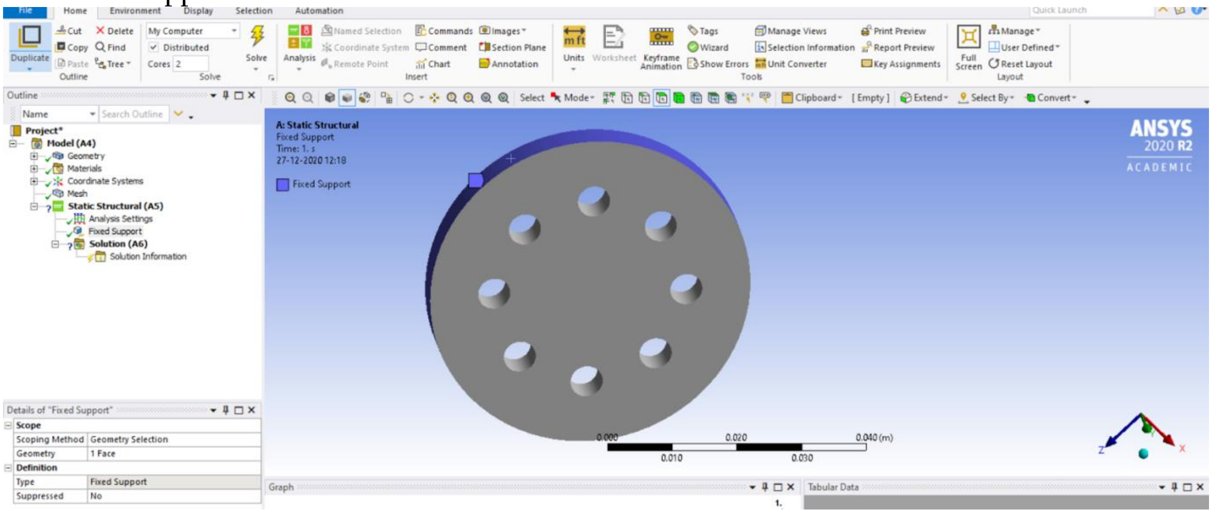
Double Click Model



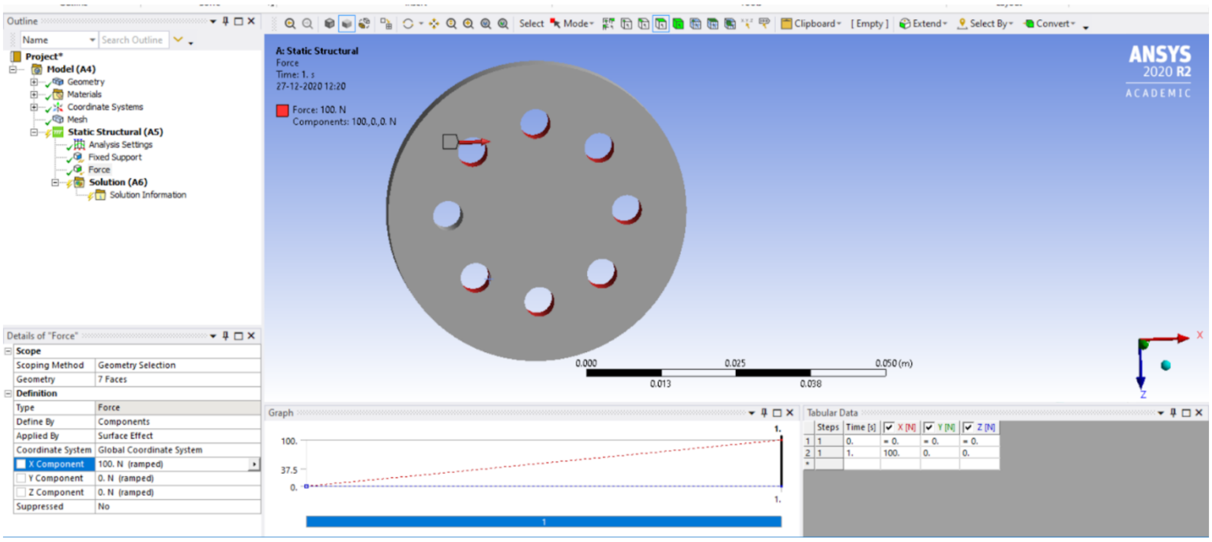
Mesh



Insert Fix Support

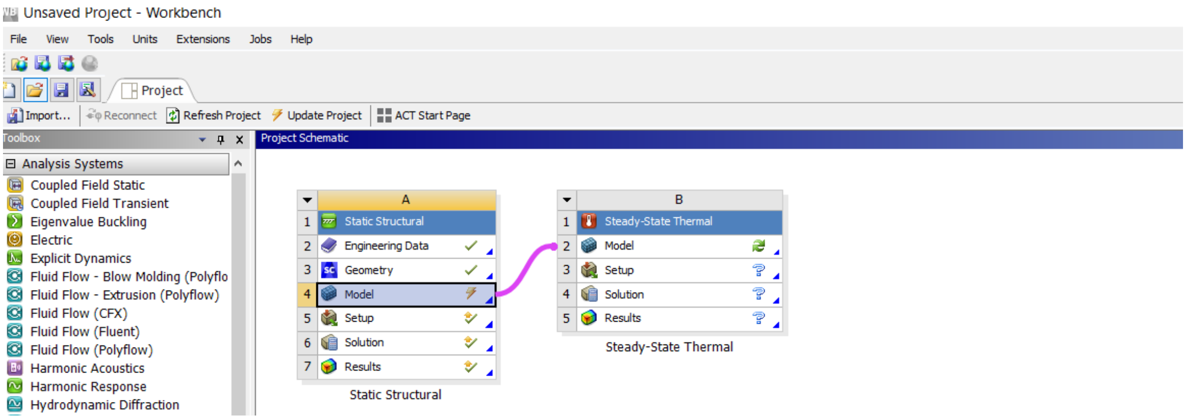


Select all the hole faces and apply Force 0f 100 N

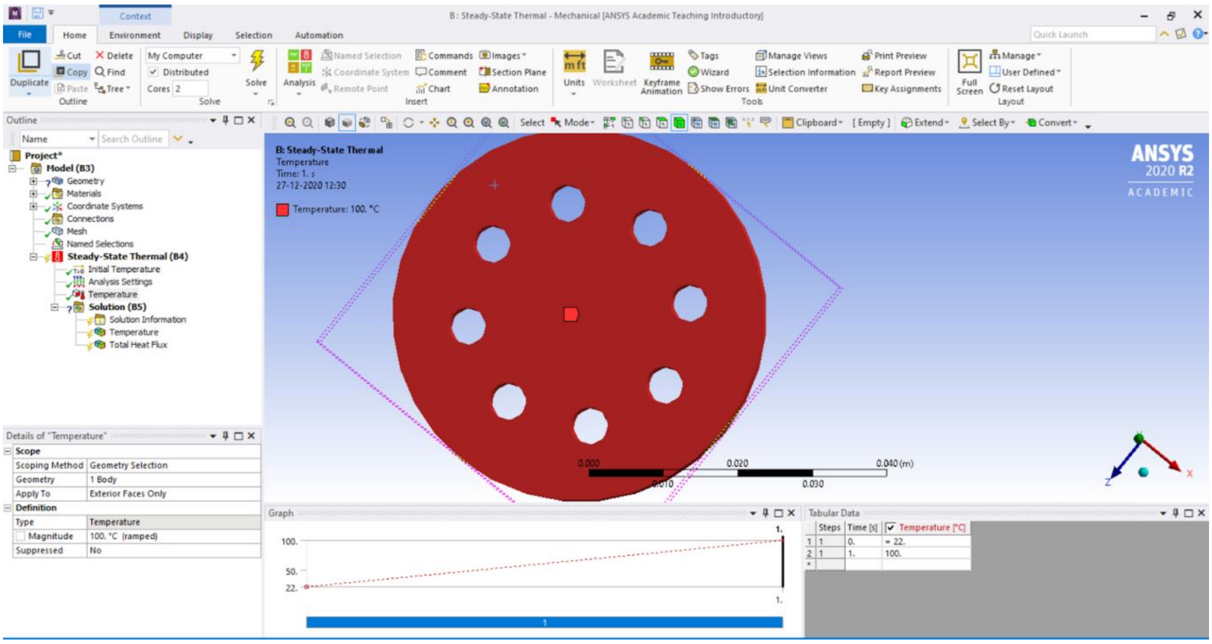


- Total Deformation
- Equivalent Stress

Couple Field Analysis



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 ?
- 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.

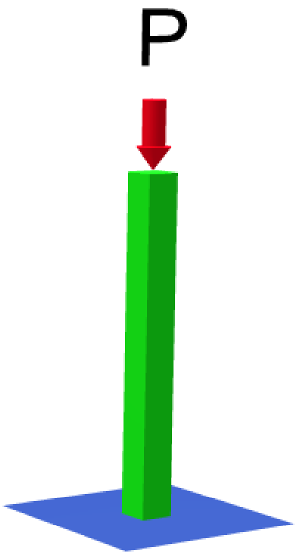
- Eigenvalue buckling analysis

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:

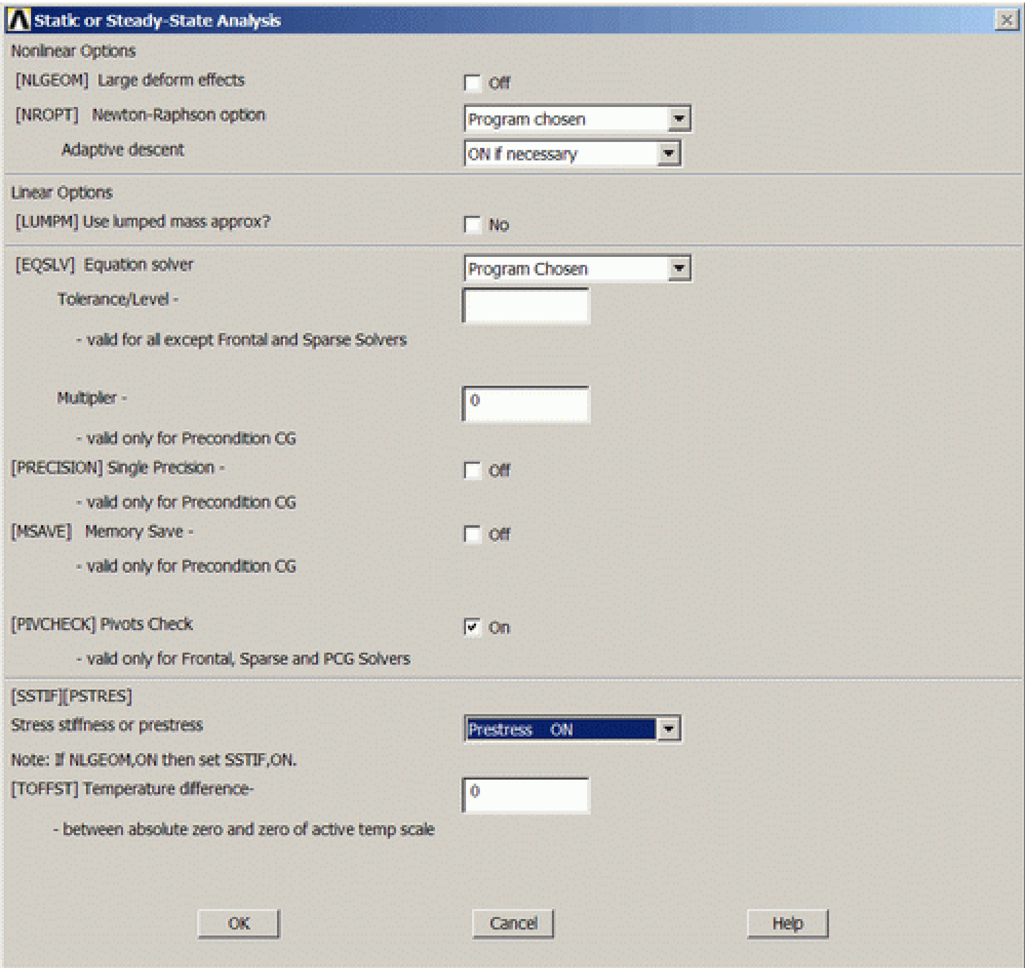
Keypoints	Coordinates (x,y)
1	(0,0)
2	(0,100)

4. ***Create Lines***
   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 menu, select the unabridged 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.



3. ***Apply Constraints***

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

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

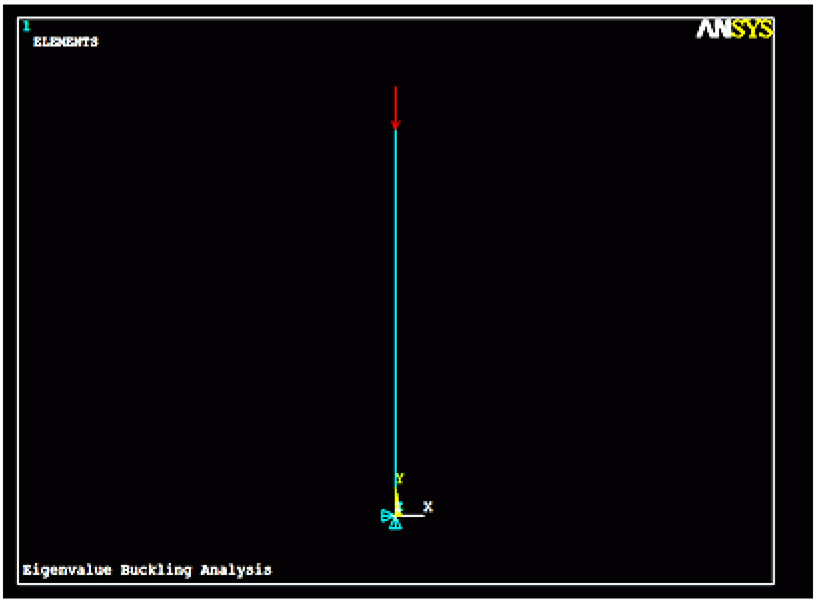
4. ***Apply Loads***

   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).

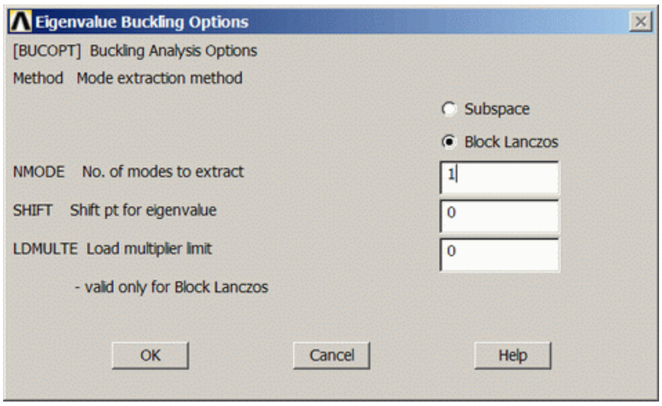
The applied loads and constraints should now appear as shown in the figure below.



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.



9. ***Solve the System***

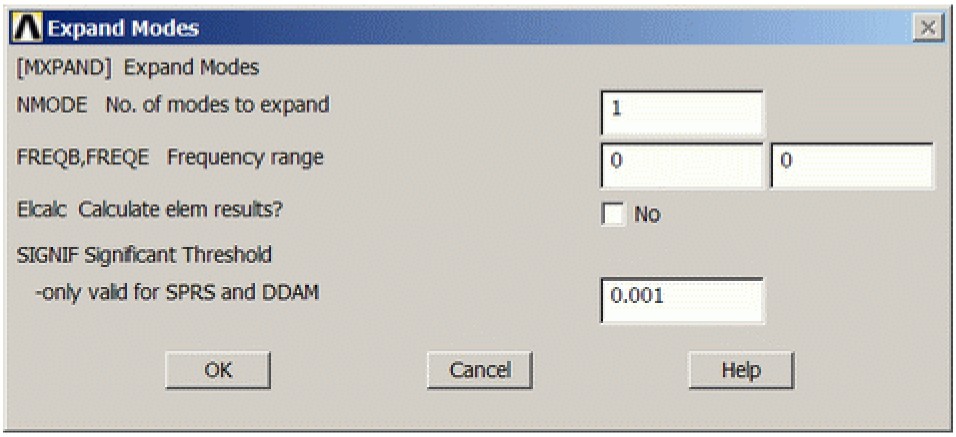
    Solution > Solve > Current LS
    SOLVE

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 ...**
    - Complete the following window as shown to expand the first mode



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

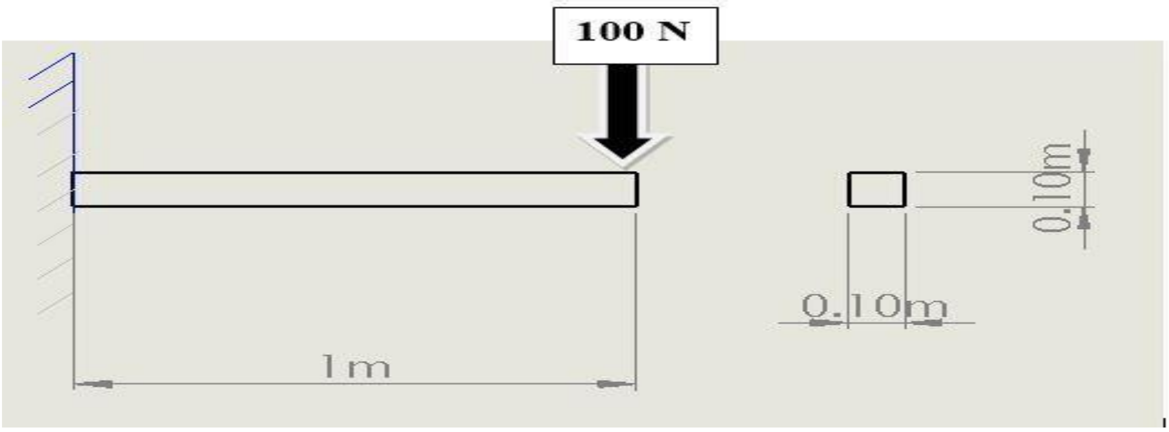
AIM:

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/m^3.

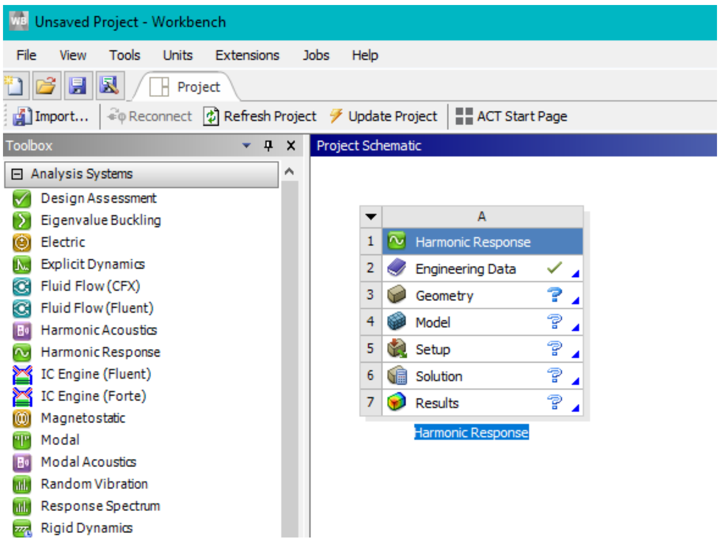


PROCEDURE:

Step 1: Workbench Toolbox

Toolbox - Analysis Systems – Harmonic Response

Open Harmonic Response dialog box will appear

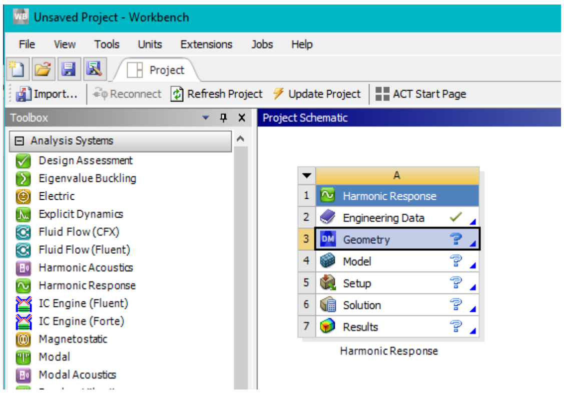


In that Engineering Data open by default Structural Steel will be there

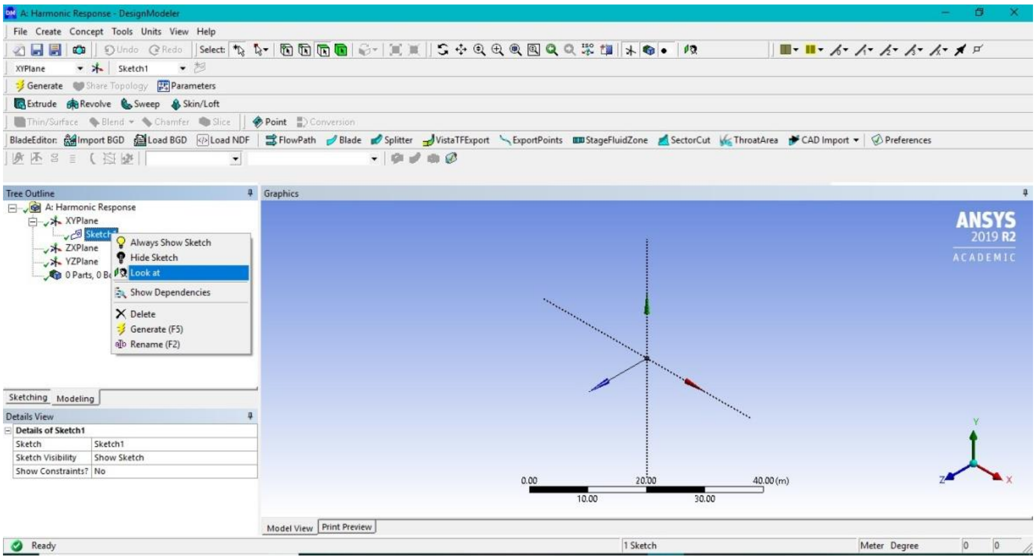
Close Engineering Data and from dialog box open Geometry

Step 2: Create A Geometry

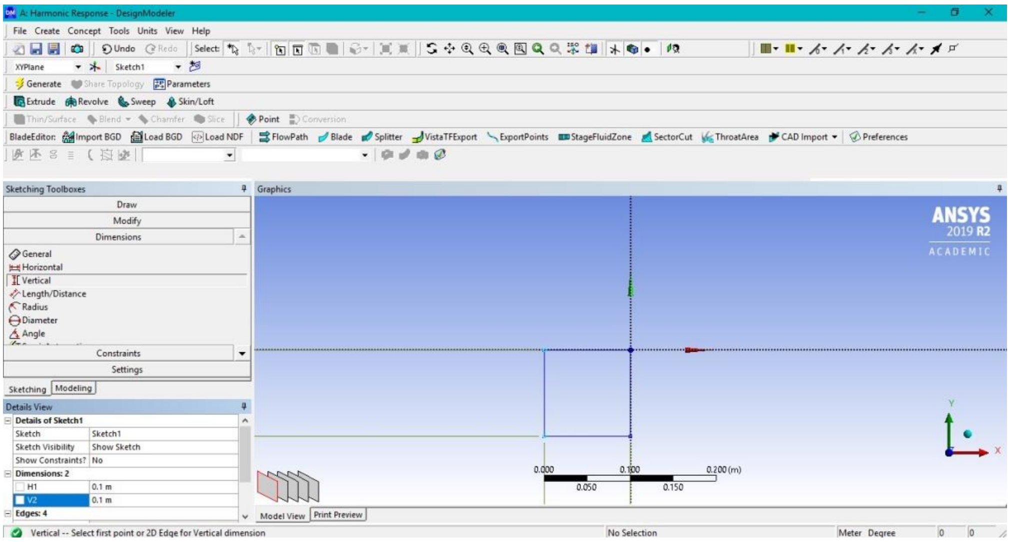
From dialog box open Geometry



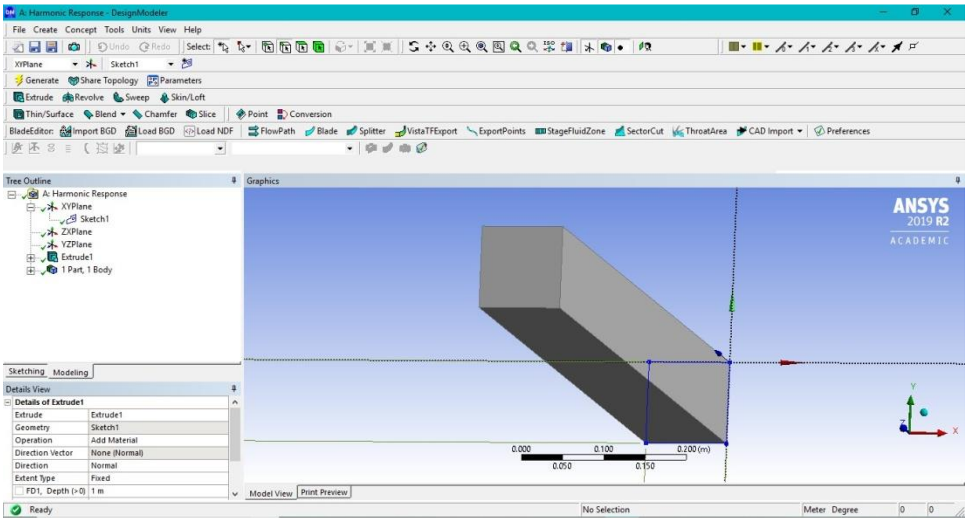
Select XY Plane - Sketch and make it Look at Face
Set Units as Meter



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



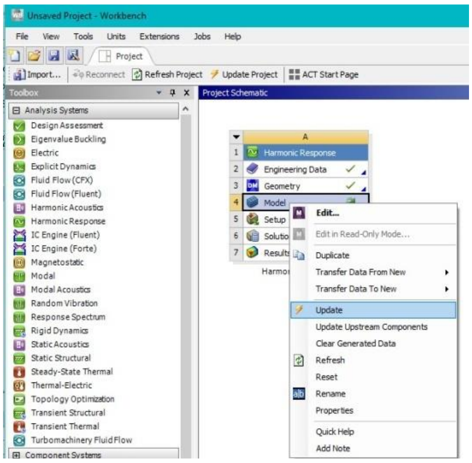
Extrude – Apply – 1m – Generate



Step 3: Model

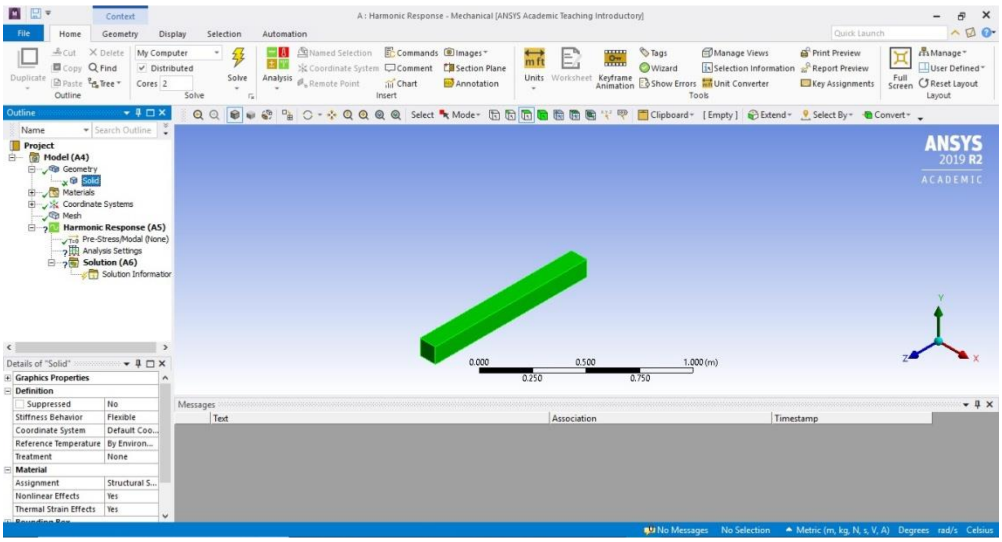
From dialog box Select Model

Model – Update

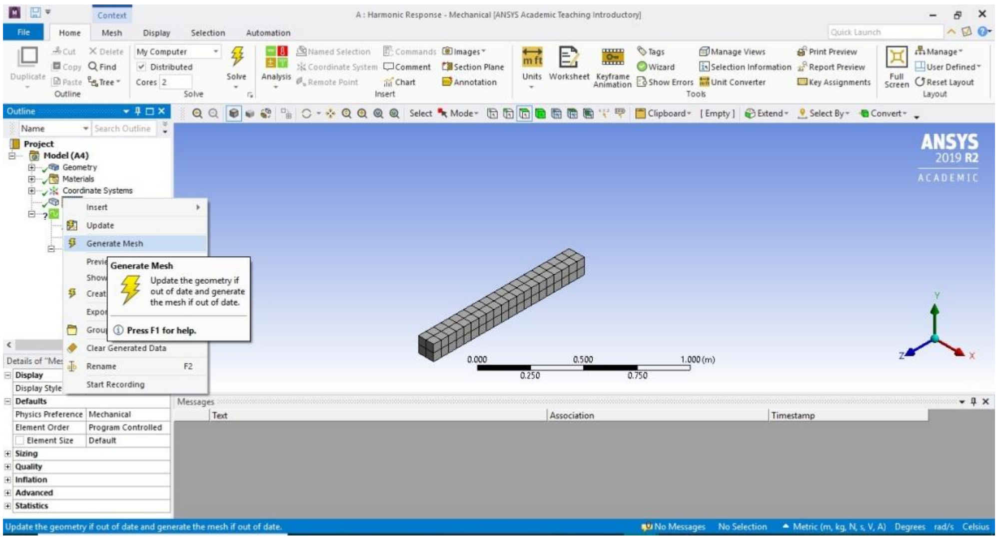


Open Model

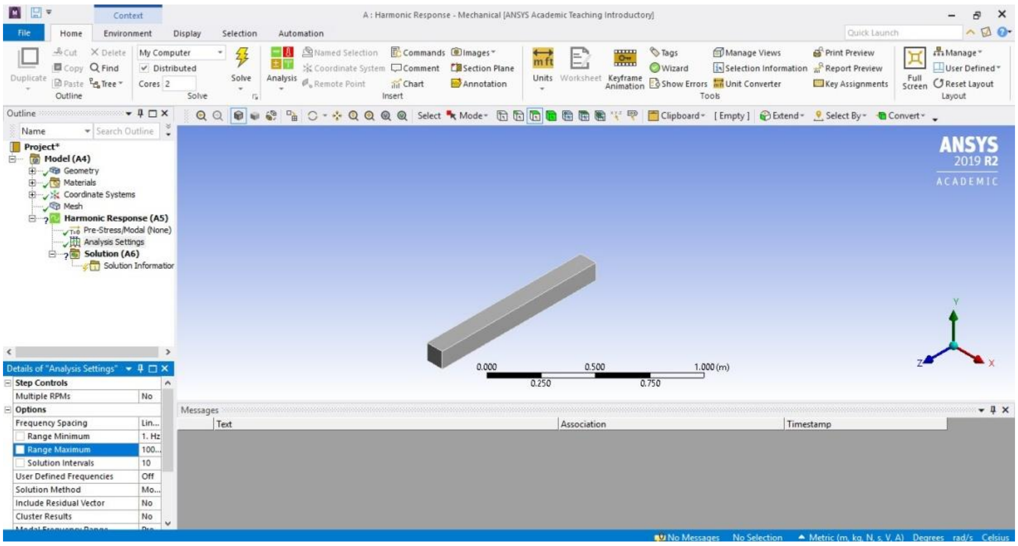
Geometry – Solid – Assignment – Structural Steel



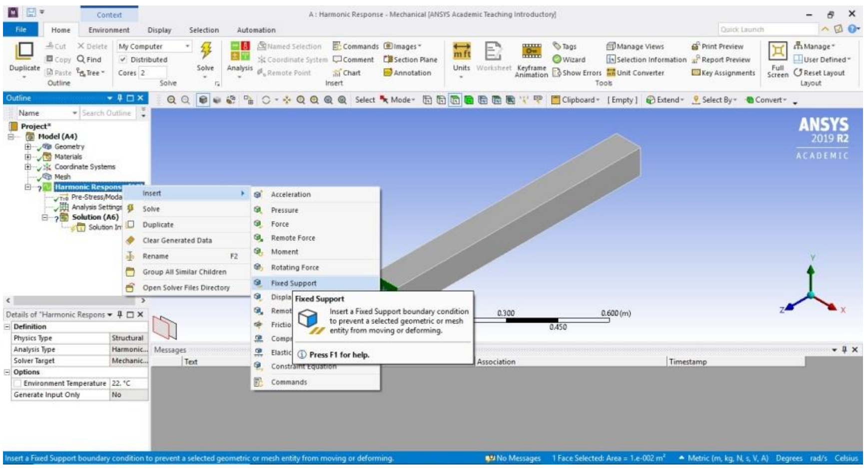
Mesh – Generate Mesh



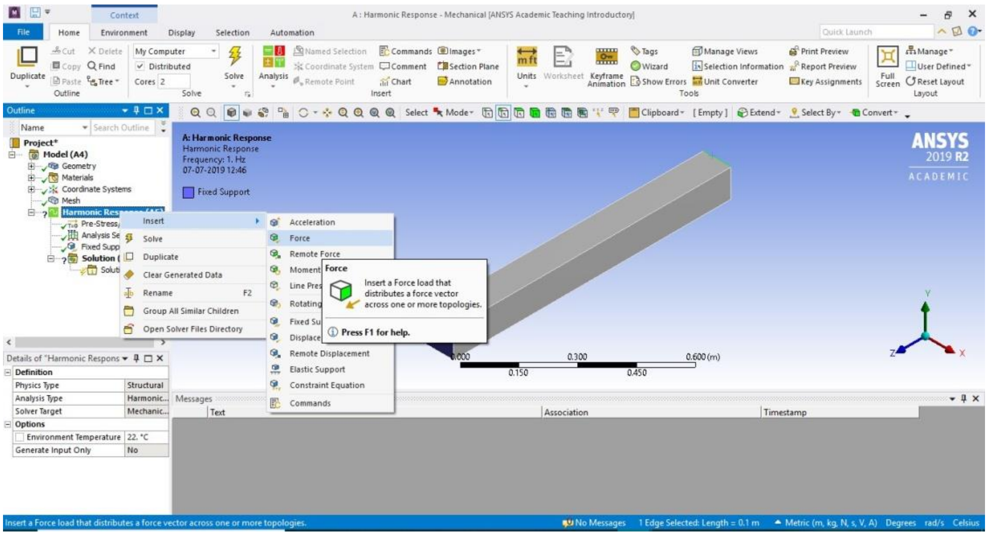
Harmonic Response – Analysis Settings
Range Minimum – 1 Hz
Range Maximum – 100 Hz



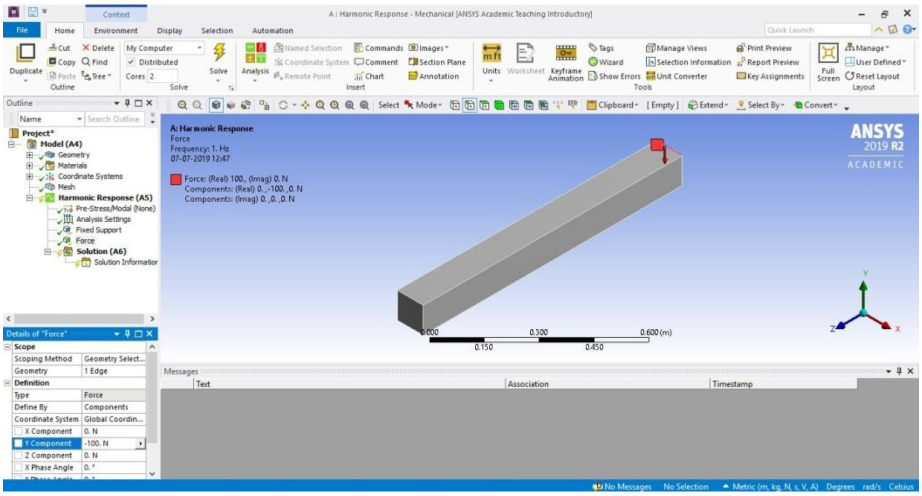
Select Face – Face – Harmonic Response – Insert – Fixed Support



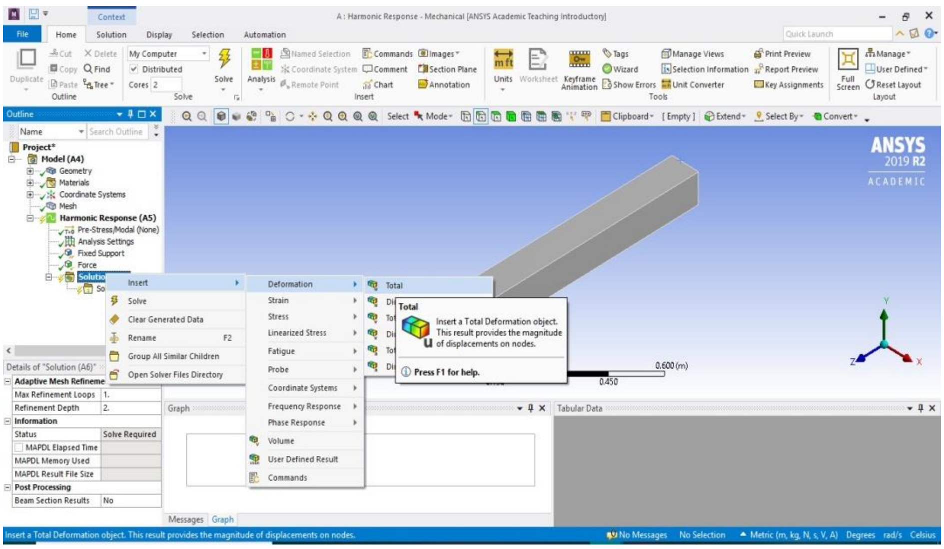
Select Edge – Edge – Harmonic Response – Insert – Force



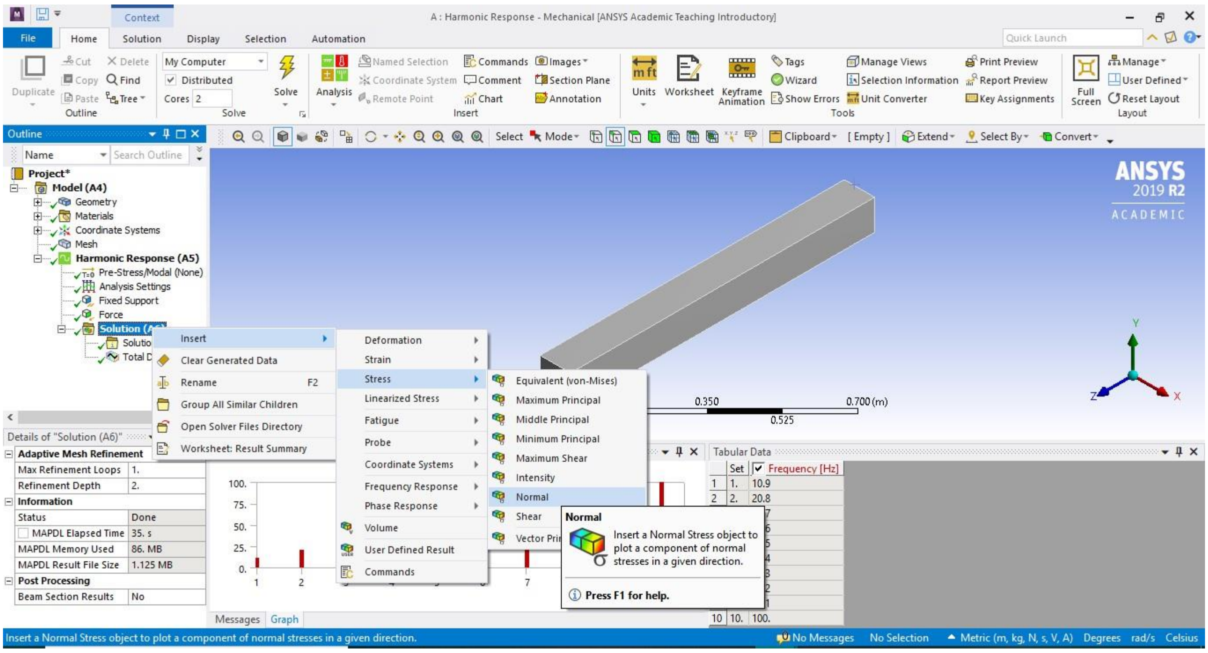
Components – Y Component – (- 100 N)
-ve sign is for down ward direction



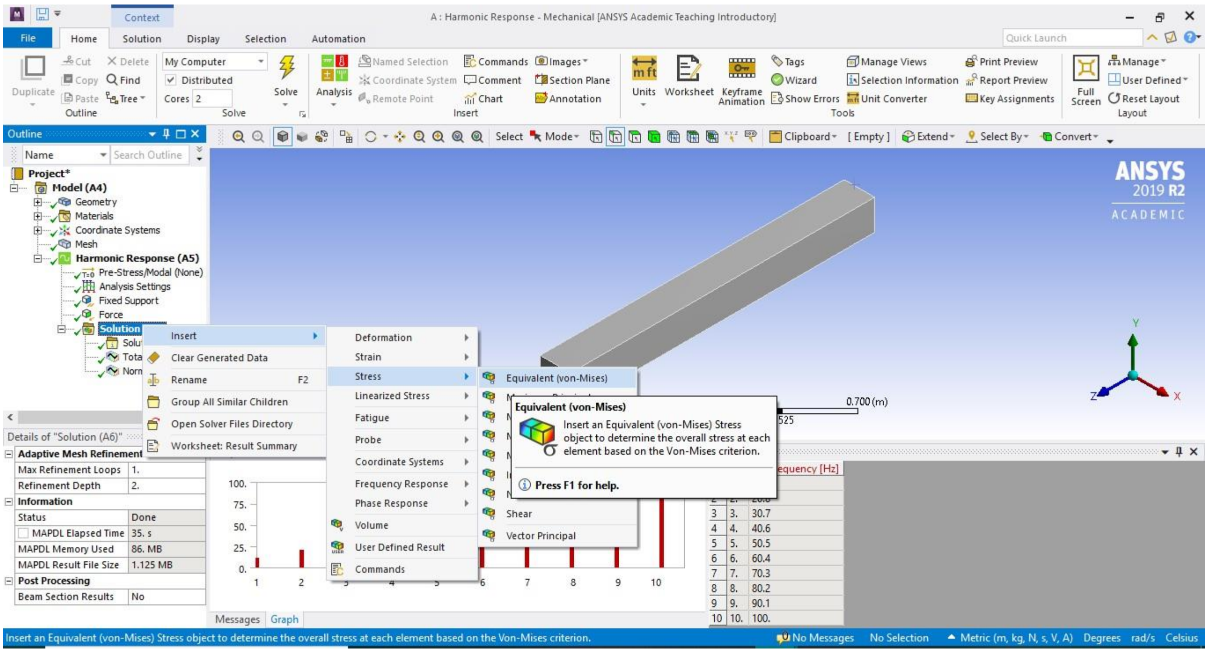
Solution – Insert – Deformation – Total – Solve



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

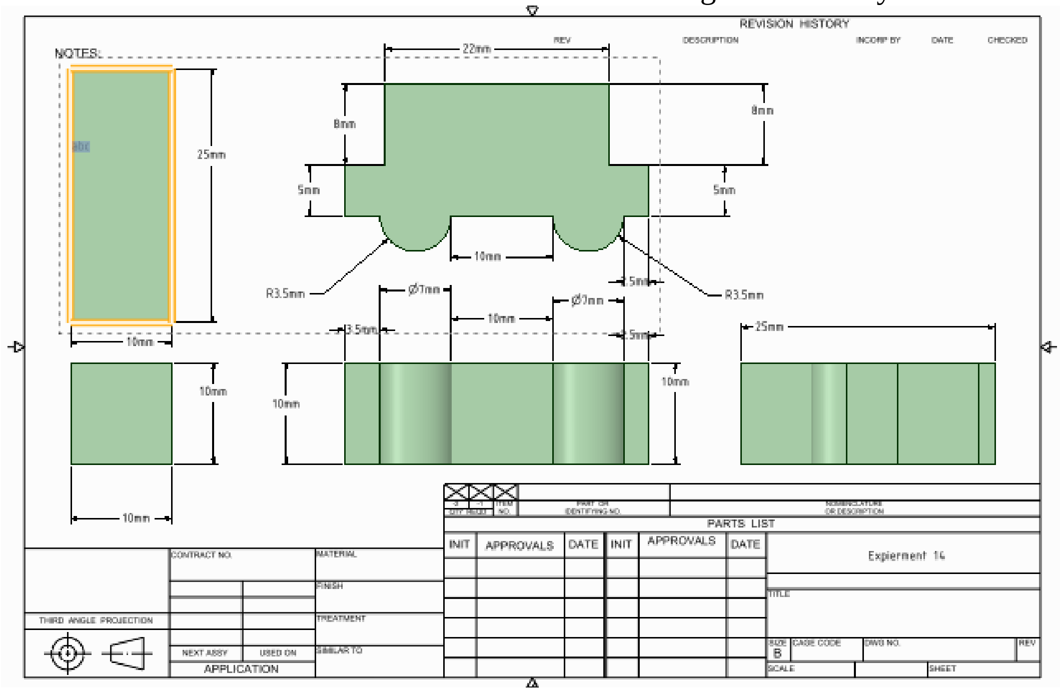
AIM:

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

SOFTWARE: ANSYS

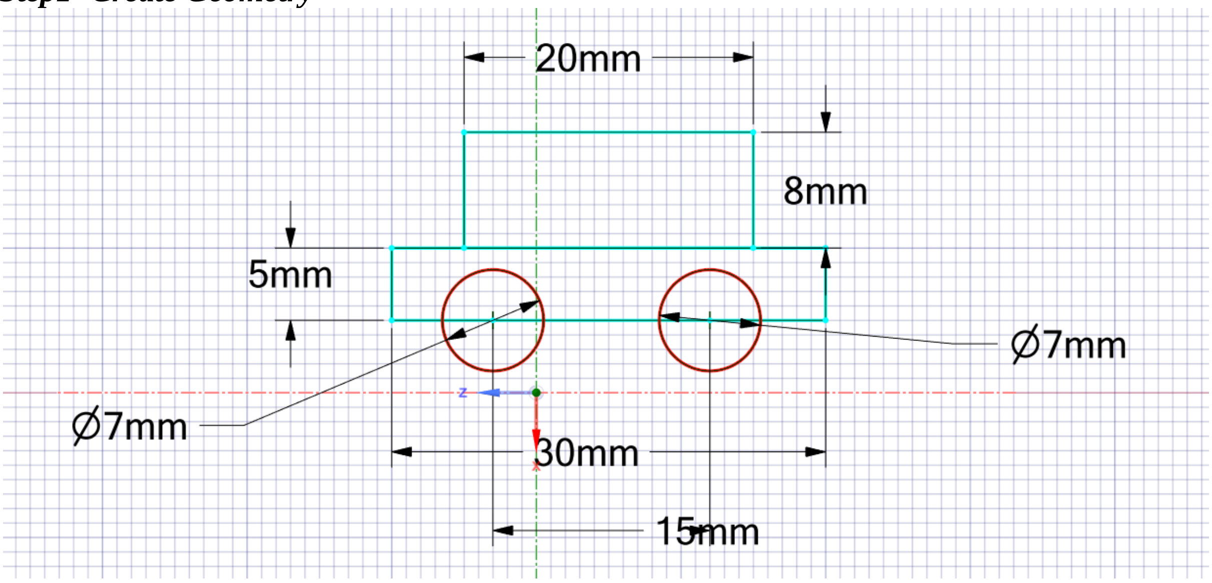
THEORY:

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

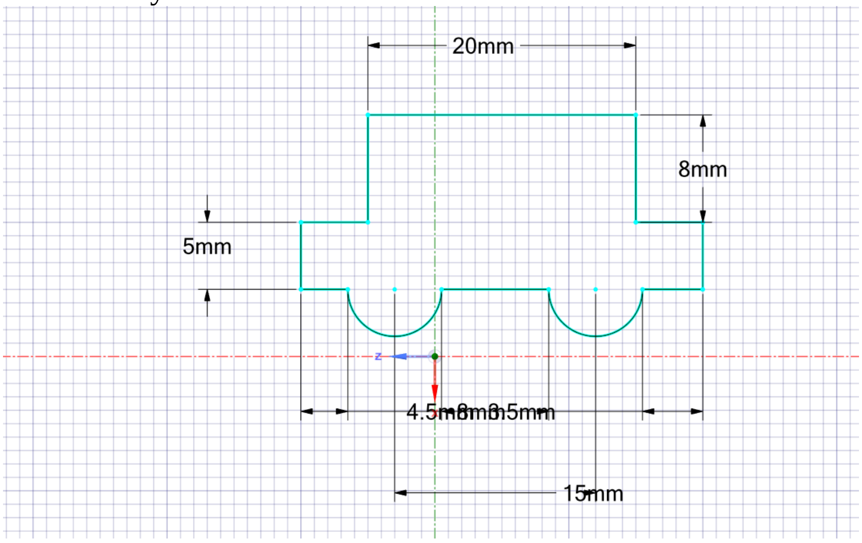


PROCEDURE:

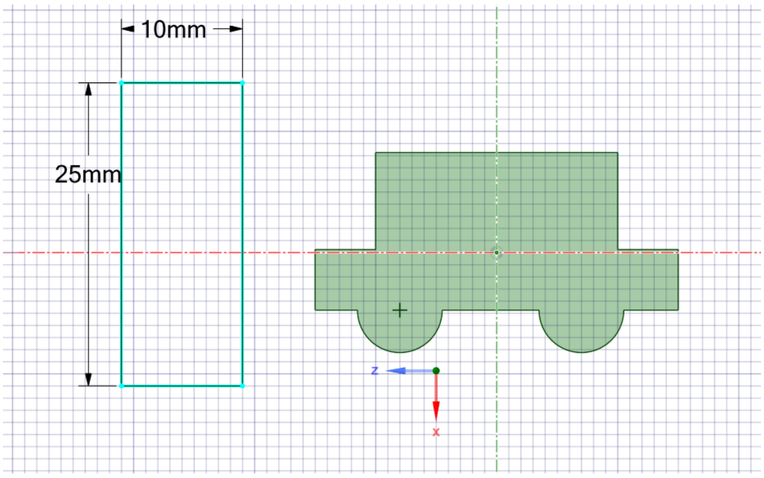
Step1- Create Geometry



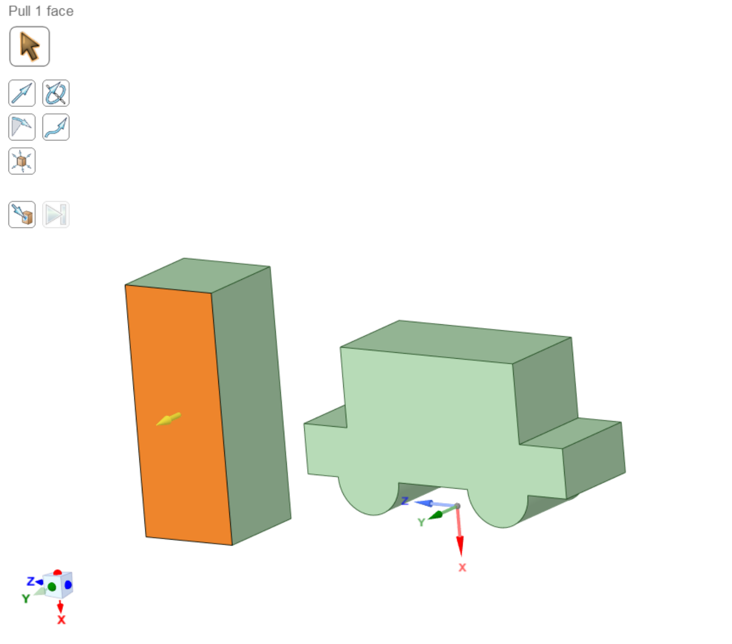
Trim Unwanted Geometry



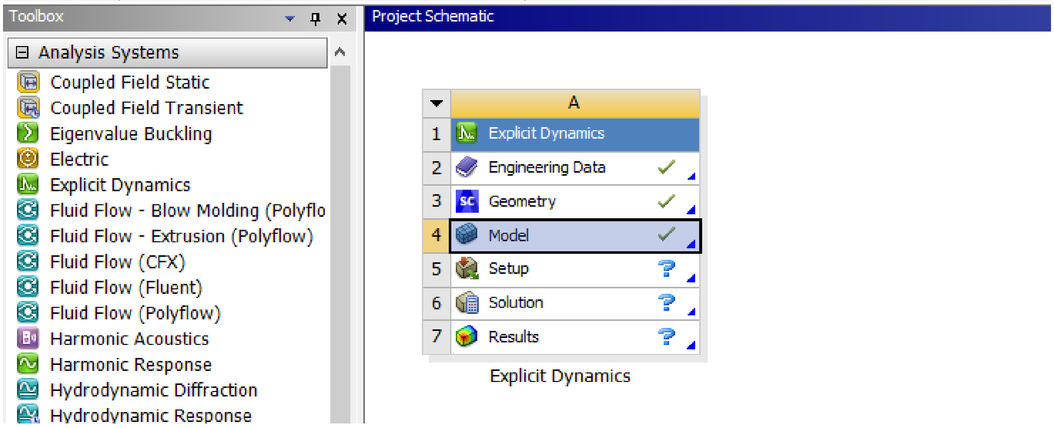
Click Sketch Mode to Create wall



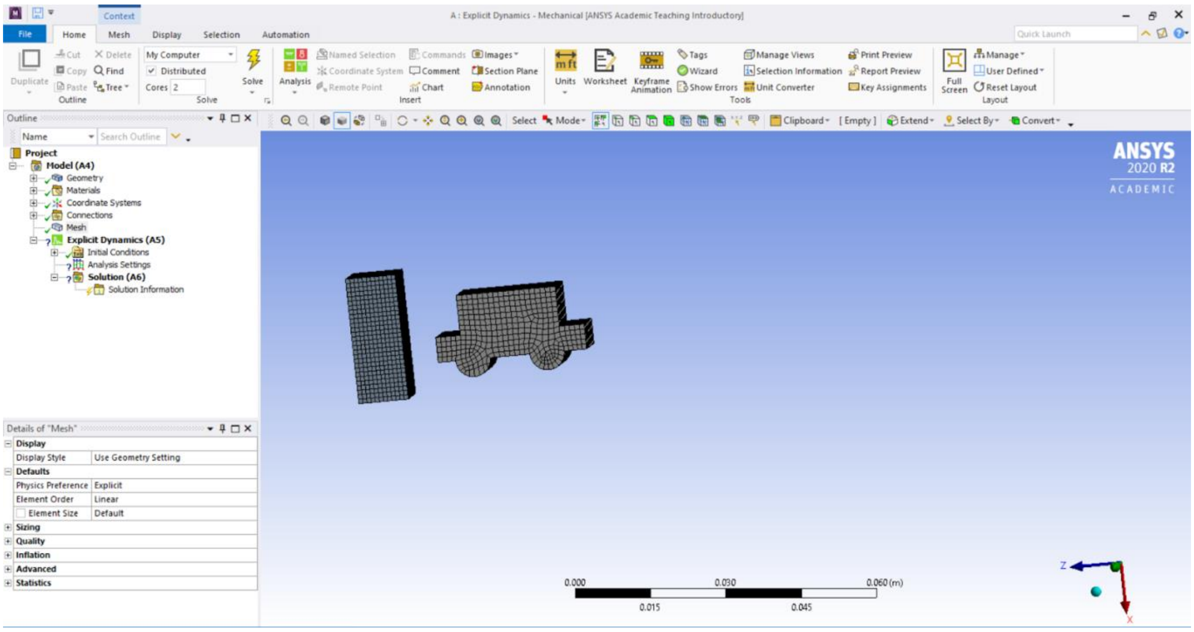
Extrude The Geometry



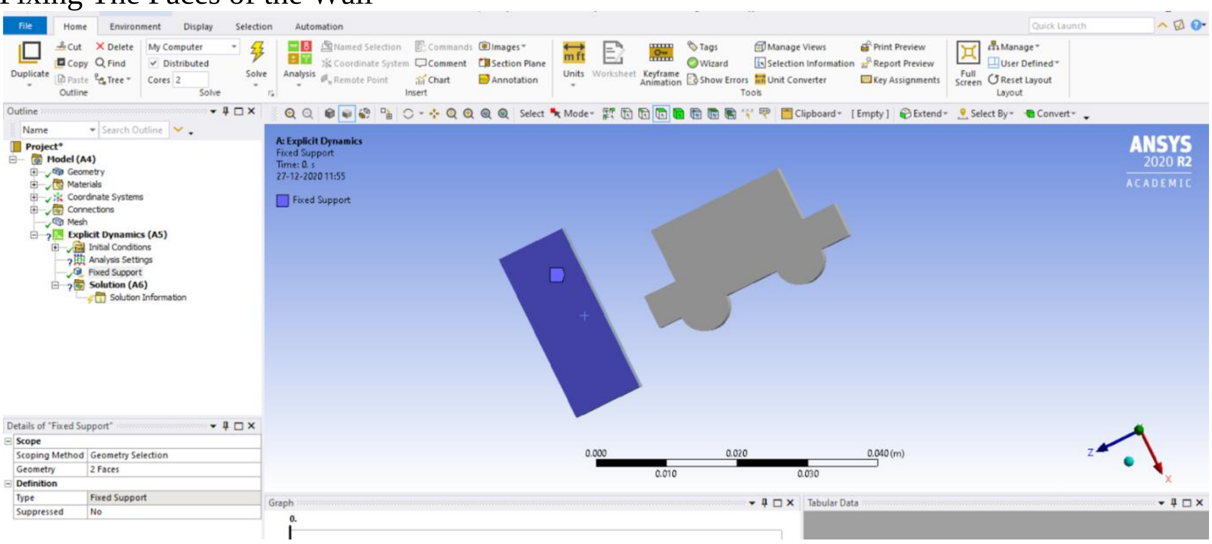
Right Click On Model to Update The model by default Steel is



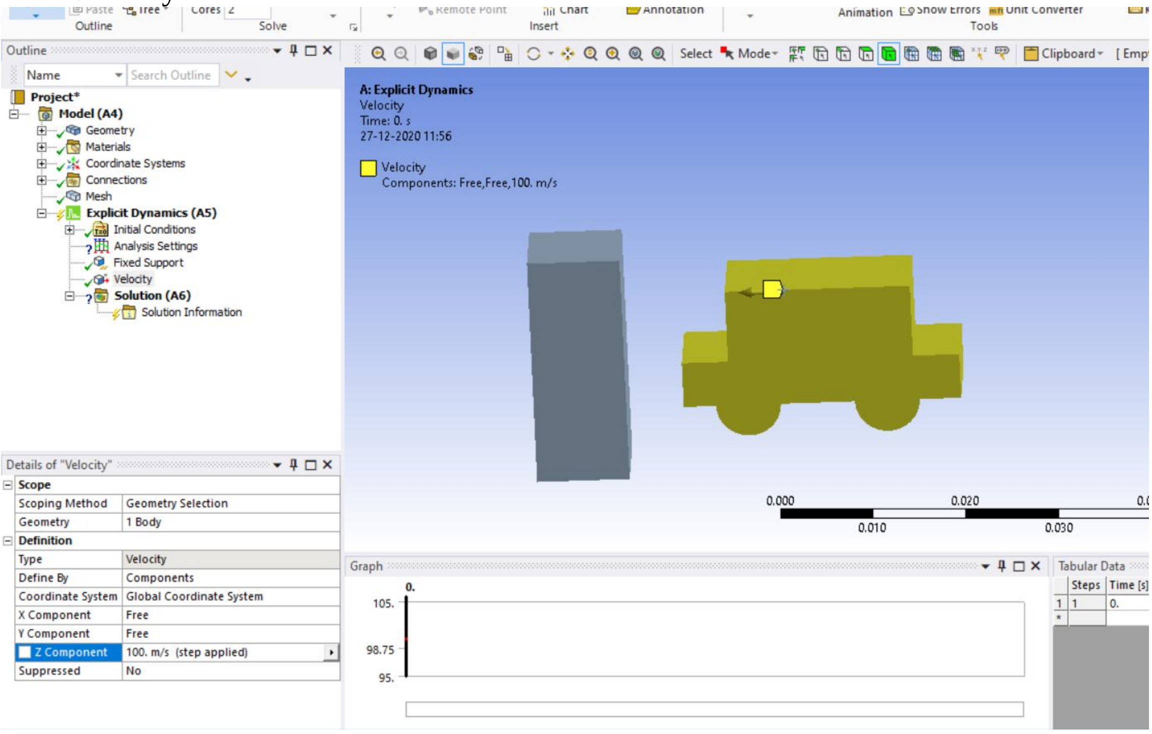
Mesh



Putting Boundary Condition
Fixing The Faces of the Wall



Give Velocity in Z Direction



- 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?

Estd: 2008

METHODIST

COLLEGE OF ENGINEERING & TECHNOLOGY

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

Abids, Hyderabad, Telangana, 500001