POORNIMA
COLLEGE OF ENGINEERING

ISI-6, RIICO Institutional Area, Sitapura, Jaipur-302022, Rajasthan

Phone/Fax: 0141-2770790-92, www.pce.poornima.org

FEA Lab Manual

(Lab Code:- 7ME4-21)

4th Year, 7th Semester

Department of Mechanical Engineering

Session: 2021-22

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

TABLE OF CONTENT

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

VISION & MISSION

INSTITUTE VISION & MISSION

VISION

- To create knowledge based society with scientific temper, team spirit and dignity of labor to face the global competitive challenges

MISSION

- To evolve and develop skill based systems for effective delivery of knowledge so as to equip young professionals with dedication & commitment to excellence in all spheres of life

DEPARTMENT VISION & MISSION

VISION

- To be recognized for quality education in the field of Mechanical Engineering and identified for its innovation & excellence

MISSION

- To provide education that transforms students through rigorous teaching and thought process to fulfill the needs of the society and industry
- To collaborate with leading industry partners and other academic & research institutes around the world to strengthen the education and research ecosystem.
- To prepare students with life-long learning for their career by fostering in them the ethical & technical capabilities pertinent to mechanical & allied engineering.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

RTU SYLLABUS AND MARKING SCHEME

<table>
<tr><td colspan="3">7ME4-21: FEA LAB</td></tr>
<tr><td colspan="2">Credit: NA</td><td>Max. Marks: 100 (IA:60, ETE:40)</td></tr>
<tr><td colspan="2">0L+0T+2P</td><td>End Term Exam: 3 Hours</td></tr>
<tr><td>S. No.</td><td colspan="2">NAME OF EXPERIMENT</td></tr>
<tr><td>1.</td><td colspan="2">Introduction to ANSYS Workbench</td></tr>
<tr><td>2.</td><td colspan="2">Structural analysis of a cantilever beam</td></tr>
<tr><td>3.</td><td colspan="2">Steady state thermal analysis of “car brake disc”</td></tr>
<tr><td>4.</td><td colspan="2">Modal Analysis of Cantilever beam</td></tr>
<tr><td>5.</td><td colspan="2">Comparison of Stress Concentrated Loads</td></tr>
<tr><td>6.</td><td colspan="2">Analysis of Axis-Symmetric Solids</td></tr>
<tr><td>7.</td><td colspan="2">Analysis Stepped Shaft in Axial Tension</td></tr>
<tr><td>8.</td><td colspan="2">Introduction to MATLAB</td></tr>
<tr><td>9.</td><td colspan="2">Modal Analysis of Cantilever beam by MATLAB</td></tr>
</table>

EVALUATION SCHEME

I+II Mid Term Examination			Attendance and performance			End Term Examination			Total Marks
Experiment	Viva	Total	Attendance	Performance	Total	Experiment	Viva	Total	
22	08	30	08	22	30	22	08	30	75

DISTRIBUTION OF MARKS FOR EACH EXPERIMENT

Attendance	Record	Performance	Total
2	3	5	10

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

LAB OUTCOME AND ITS MAPPING WITH PO & PSO

LAB OUTCOMES

After completion of this course, students will be able to –

7ME4-21.1	Discuss the basic features of an analysis package.
7ME4-21.2	Demonstrate the deflection of beams subjected to point, uniformly distributed and varying loads
7ME4-21.3	Use the modern tools to formulate and solve problems of bars, truss, beams, and plate to find stress with different loading conditions.
7ME4-21.4	Applying basic principle to solve and demonstrate 1D and 2D heat transfer with conduction and convection boundary conditions.

LO-PO-PSO MAPPING MATRIX OF COURSE

LO/PO/PSO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3
7ME4-21.1	3	-	-	-	-	-	-	-	-	-	-	2	-	3	2
7ME4-21.2	-	3	-	-	-	-	-	-	-	-	-	2	-	3	2
7ME4-21.3	-	-	-	-	3	-	-	-	-	-	-	2	-	3	2
7ME4-21.4	-	-	-	3	-	-	-	-	-	-	-	2	-	3	2

PROGRAM OUTCOMES (POs)

PO1	**Engineering knowledge :** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems
PO2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO3	**Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO5	**Modern tool usage:** Create, select, and apply appropriate techniques, resources, and

	modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO6	The engineer and society: Apply reasoning informed by 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 and norms of the engineering 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 with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO11	**Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and 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 (PSOs)

PSO1	Design, analyze and innovate solutions to technical issues in Thermal, Production and Design Engineering.
PSO2	Exhibit the knowledge and skills in the field of Mechanical & Allied engineering concepts.
PSO3	Apply the knowledge of skills in HVAC&R and Automobile engineering.

RUBRICS FOR LAB

Laboratory Evaluation Rubrics:

S. No.	Criteria	Sub Criteria and Marks Distribution			Outstanding (>90%)	Admirable (70-90%)	Average (40-69%)	Inadequate (<40%)
		Mid-Term	**End-Team**	**Continues Evaluation**				
A	**PERFORMANCE (PO1, PO8, PO9)**	Procedure Followed M.M. 50 = 3 M.M. 75 = 4 M.M. 100 = 6	Procedure Followed M.M. 50 = 3 M.M. 75 = 4 M.M. 100 = 6	Procedure Followed M.M. 50 = 1 M.M. 75 = 2 M.M. 100 = 2	• All possible system and Input/ Output variables are taken into account • Performance measures are properly defined • Experimental scenarios are very well defined	•Most of the system and Input/ Output variables are taken into account • Most of the Performance measures are properly defined • Experimental scenarios are defined correctly	• Some of the system and Input/ Output variables are taken into account • Some of the Performance measures are properly defined • Experimental scenarios are defined but not sufficient	•System and Input/ Output variables are not defined • Performance measures are not properly defined • Experimental scenarios not defined
		Individual/Team Work M.M. 50 = 3 M.M. 75 =4 M.M. 100 = 6	Individual/Team Work M.M. 50 = 3 M.M. 75 =4 M.M. 100 = 6	Individual/Team Work M.M. 50 = 1 M.M. 75 = 2 M.M. 100 = 2	•Coordination among the group members in performing the experiment was excellent	•Coordination among the group members in performing the experiment was good	•Coordination among the group members in performing the experiment was average	•Coordination among the group members in performing the experiment was very poor
		Precision in data collection M.M. 50 = 3 M.M. 75 = 4 M.M. 100 = 6	Precision in data collection M.M. 50 = 3 M.M. 75 = 4 M.M. 100 = 6	Precision in data collection M.M. 50 = 2 M.M. 75 = 2 M.M. 100 = 4	•Data collected is correct in size and from the experiment performed	•Data collected is appropriate in size and but not from proper sources.	•Data collected is not so appropriate in size and but from proper sources.	ta collected is neither appropriate in size and norfrom proper sources
B	**LAB RECORD/WRITTEN WORK (PO1, PO8, PO10)**	NA	NA	Timing of Evaluation of Experiment M.M. 50 = 3 M.M. 75 = 4 M.M. 100 = 6	• On the Same Date of Performance	• On the Next Turn from Performance	• Before Dead Line	n the Dead Line
		Data Analysis M.M. 50 = 3 M.M. 75 =5 M.M. 100 = 6	Data Analysis M.M. 50 = 3 M.M. 75 =5 M.M. 100 = 6	Data Analysis M.M. 50 = 2 M.M. 75 = 3 M.M. 100 = 4	•Data collected is exhaustively analyzed & appropriate features are selected	•Data collected is analyzed & but appropriate features are not selected	•Data collected is not analyzed properly. •Features selected are not appropriate	ta collected is not analyzed & the features are not selected

7/11
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

		Results and Discussion M.M. 50 = 3 M.M. 75 =5 M.M. 100 = 6	Results and Discussion M.M. 50 = 3 M.M. 75 =5 M.M. 100 = 6	Results and Discussion M.M. 50 = 2 M.M. 75 = 3 M.M. 100 = 4	• All results are very well presented with all variables • Well prepared neat diagrams/plots/ tables for all performance measured • Discussed critically behavior of the system with reference to performance measures • Very well discussed pros n cons of outcome	• All results presented but not all variables mentioned • Prepared diagrams /plots/ tables for all performance measured but not so neat • Discussed behavior of the system with reference to performance measures but not critical • Discussed pros n cons of outcome in brief	• Partial results are included • Prepared diagrams /plots/ tables partially for the performance measures • Behavior of the system with reference to performance measures has been superficially presented • Discussed pros n cons of outcome but not so relevant	• Results are included but not as per experimental scenarios • No proper diagrams /plots/ tables are prepared • Behavior of the system with reference to performance measures has not been presented • Did not discuss pros n cons of outcome
C	**VIVA (PO1, PO10)**	Way of presentation M.M. 50 = 2.5 M.M. 75 = 4 M.M. 100 = 5	Way of presentation M.M. 50 = 2.5 M.M. 75 = 4 M.M. 100 = 5	Way of presentation M.M. 50 = 2 M.M. 75 = 3 M.M. 100 = 4	• Presentation was very good	• Presentation was good	• Presentation was satisfactory	• Presentation was poor
		Concept Explanation M.M. 50 = 2.5 M.M. 75 = 4 M.M. 100 = 5	Concept Explanation M.M. 50 = 2.5 M.M. 75 = 4 M.M. 100 = 5	Concept Explanation M.M. 50 = 2 M.M. 75 = 3 M.M. 100 = 4	• Conceptual explanation was excellent	• Conceptual explanation was good	• Conceptual explanation was somewhat good	• Conceptual explanation was Poor
D	**ATTENDA NCE**	NA	NA	**A**ttendance M.M. 50 = 5 M.M. 75 =8 M.M. 100 =10	• Present more than 90% of lab sessions	• Present more than 75% of lab sessions	• Present more than 60% of lab sessions	• Present in less than 60% lab sessions

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

LAB CONDUCTION PLAN

Total number of Experiment – 09

Total number of turns required - 10

Number of turns required for:-

Experiment Number	Scheduled Week
Experiment -1	Week 1
Experiment -2	Week 2
Experiment -3	Week 3
Experiment -4	Week 4
Experiment -5	Week 5
I Mid Term	**Week 6**
Experiment -6	Week 7
Experiment-7	Week 8
Experiment-8	Week 9
Experiment-9	Week 10
II Mid Term	Week 11

DISTRIBUTION OF LAB HOURS

S. No.	Activity	Distribution of Lab Hours	
		Time (180 minute)	Time (120 minute)
1	Attendance	5	5
2	Explanation of Experiment & Logic	30	30
3	Performing the Experiment	60	30
4	File Checking	40	20
5	Viva/Quiz	30	20
6	Solving of Queries	15	15

GENERAL LAB INSTRUCTIONS

DO'S

1. Enter the lab on time and leave at proper time.
2. Feel that practical are essentials to lay the foundation for understanding the subject.
3. Have knowledge of the theoretical background of each experiment.
4. Handling every equipment carefully.
5. Consult your teacher, or your friend who had already done the experiment before entering the lab. This will help you to overcome difficulties while doing the experiments.
6. Turn off the machines before leaving the lab unless a member of lab staff has specifically told you not to do so.
7. Make as many observations/readings as possible. Large number of data will eliminate random errors and systematic errors.
8. Calculations must be done meticulously. For this, the knowledge of using calculators and mathematical tables is essential.
9. If you get wrong result others than the expected one, study your observation thoroughly and find out where you went wrong. Repeat the experiment until you get the correct observation, leading to the correct and expected result.
10. If you notice any problem with machine/ equipment/tool, then please report it to lab staff immediately. Do not attempt to fix the problem yourself.

DON'TS

1. Don't neglect the importance of practical.
2. Don't be lazy in making observation. Avoid copying someone else's observation.
3. There should not be any distraction. Don't play with your friend or the apparatus while doing the experiment.
4. Don't damage the equipment.
5. Don't touch the moving parts of the machine.
6. Don't play with electric instruments.
7. If you are going to be away from your machine for more than 10 or 15 minutes, switch off before leaving. This is for the security of your experiment and to ensure that others are able to use the lab resources while you are not.
8. No food or soft drink is allowed in the lab or near any of the equipment. Aside from the fact that it leaves a mess and attract pests. If you need to eat or drink, take a break and do so in the canteen.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

9. Do not work in a laboratory wearing loose hair, loose clothing or dangling jewelry.
10. Don’t wear rings, watches, bracelets or other jewelry that could get could get caught in moving machinery.
11. Do not eat food, drink beverages or chew gum in the laboratory.
12. Don’t bring any external material in the lab, except your lab record, copy and books.

LAB SPECIFIC SAFETY RULES

1. Always sit to in front of your specified system.
2. Always Login via students login.
3. Do not install anything on the system.
4. Do not delete anything from the system.
5. Take all your files to your pend rive or on your E-mail.

Experiment No.1- Introduction to ANSYS Workbench

Objective: - Introduction of GUI of the software in the areas of Mechanics of Solids, Vibration & Heat Transfer.

INTRODUCTION TO ANSYS

Introduction

Ansys enables you to do various types of finite element analysis such as Structural, Thermal, Fluid Flow, Electromagnetic, Coupled Field etc. You can create solid model, FE model and Postprocess the results using GUI mode as well as command prompt. You also have the ability to do parametric modeling for your design optimization study.

Concepts: Introduction to ANSYS

ANSYS is a powerful, finite element analysis tool used to create solid model, FE model and various types of analysis during new product development. ANSYS helps to carry out structural, thermal fluid flow, electromagnetic, couple field, NVH, Crash analysis.

ANSYS is mainly consists of three main steps

- Preprocessing
- Solution
- Post Processing

These concepts are discussed in details on the following slides

ANSYS Fundamentals

PreProcessing

Every Finite element analysis tool requires CAD geometry as input to create FE analysis. This process of creating or importing the solid model and FE model is called as preprocessing. In ANSYS solid model will be either directly created in ANSYS using solid modeling commands or it can be imported from other CAD software. FE model is nothing but is then created from input geometry to carry out required analysis.

Solution

Using ANSYS solution menu you can apply different loads and boundary conditions to your problem and define the suitable analysis type. ANSYS solves the required analysis using its solver.

Post Processing

Once the ANSYS solves the submitted analysis user can review the results using Post Processing tools. You can take the images, sections and also see and create animation of the results.

Capabilities

ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering. Partial listing of the capabilities:

- Structural
  - Linear
  - Nonlinear
    - Material, Geometric, Contact
  - Dynamics
    - Modal, Harmonic, Transient Dynamic, Spectrum, Random Vibration
    - Explicit Dynamics with ANSYS LS-DYNA
- Thermal
  - Steady State and Transient
- Fluid (CFD, Acoustics, and other fluid analyses)
- Low- and High-Frequency Electromagnetic
- Coupled Field

Application

Because of vast capabilities of ANSYS it is used in almost all industries for FE analysis. Following is the partial list of industries in which ANSYS is used,

- Aerospace
- Automotive
- Biomedical
- Bridges & Buildings
- Electronics & Appliances
- Heavy Equipment & Machinery
- MEMS - Micro Electromechanical Systems
- Sporting Goods

Finite Element Analysis

What is FEA?

Finite Element Analysis is a way to simulate loading conditions on a design and determine the design's response to those conditions. The design is modeled using discrete building blocks called elements.

Need of FEA

- To reduce the amount of prototype testing
  - Computer simulation allows multiple "what-if" scenarios to be tested quickly and effectively.
- To simulate designs that are not suitable for prototype testing
  - Example: Surgical implants, such as an artificial knee
- The bottom line:
  - Cost savings
  - Time savings… reduce time to market!
  - Create more reliable, better-quality designs

ANSYS Interface

There are two methods to use ANSYS; Graphical Interface and Command File Coding. The graphical user interface or GUI follows the conventions of Windows based programs. This method is probably the best approach for new users. The command approach is used by professional users. It has the advantage that an entire analysis can be described in a small text file, typically in less than 50 lines of commands. This approach enables easy model modifications and minimal file space requirements. In this lab manual we mainly use the GUI. However, in some cases both the GUI and command code are used to show that how command code could be easy to use.

• **What is *ANSYS Workbench*?**

– ***ANSYS Workbench*** is a new-generation solution from ANSYS that provides powerful methods for interacting with the ANSYS solver functionality. This environment provides a unique integration with CAD systems, and your design process, enabling the best CAE results.

• ***ANSYS Workbench* is comprised of five modules:**

– *Simulation* for performing structural and thermal analyses using the ANSYS solver

– *CFX-Mesh* for generating a CFX-Pre mesh for the CFX-5 solver

– *DesignModeler* for creating and modifying CAD geometry to prepare the solid model for use in *Simulation* or *CFX-Mesh*

– *DesignXplorer* and *DesignXplorer VT* for investigating the effect of variations input to the response of the system

– *FE Modeler* for translating a Nastran mesh for use in ANSYS

• **Analysis types available in *Simulation*:**†

– **Linear Stress:** Determines deflections, stresses, factors of safety, etc. based on standard strength of materials concepts under static loading.

– **Modal:** Determines natural frequencies of a system (free vibration), including the effects of loading on a pre-stressed structure.

– **Heat Transfer:** Steady-state thermal analyses to solve for temperature field and heat flux. Temperature-dependent conductivity and convection allowed. Thermal-stress analysis supported as well.

– **Harmonic:** Determines structural response of system under sinusoidal excitation as a function of frequency.

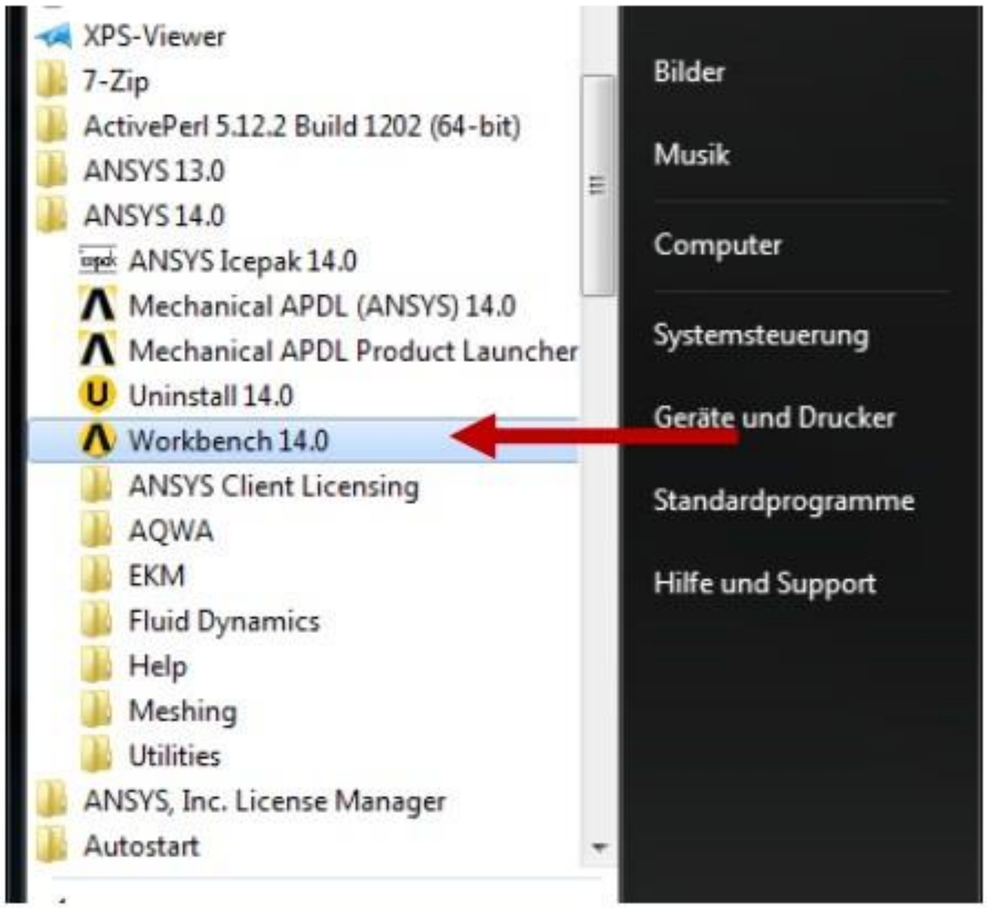
– **Linear Buckling:** Determines failure load or safety factor for buckling and its buckling mode shapes.

– **Shape Optimization:** Indicates areas of possible volume reduction based on load paths through the part using Topological Optimization technology.

– **Nonlinear Structural:** Calculates deflections and stresses of system under static loading, accounting for large deflection effects, plasticity, and contact nonlinearities.

START THE FE SOFTWARE ANSYS:

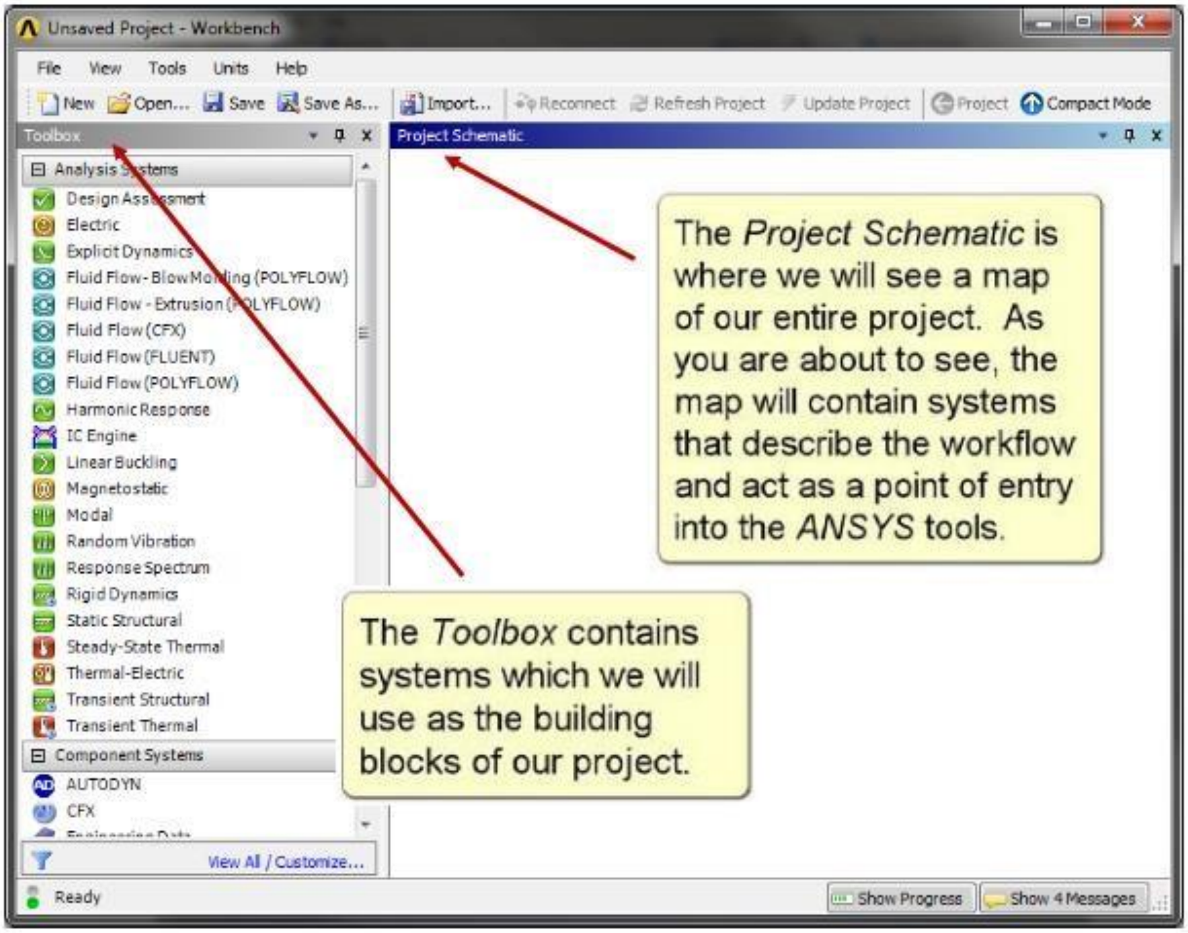
ANSYS Workbench: This is a brand new GUI with an emphasis on CAD connectivity, ease of use, and easy management of assembly contact. This GUI is covered in a separate guideline.
Launch Workbench in Windows via "Start Menu > Programs > ANSYS 14.5 > ANSYS Workbench"



Starting ANSYS Workbench

Preprocessor (Setting up the Model)

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

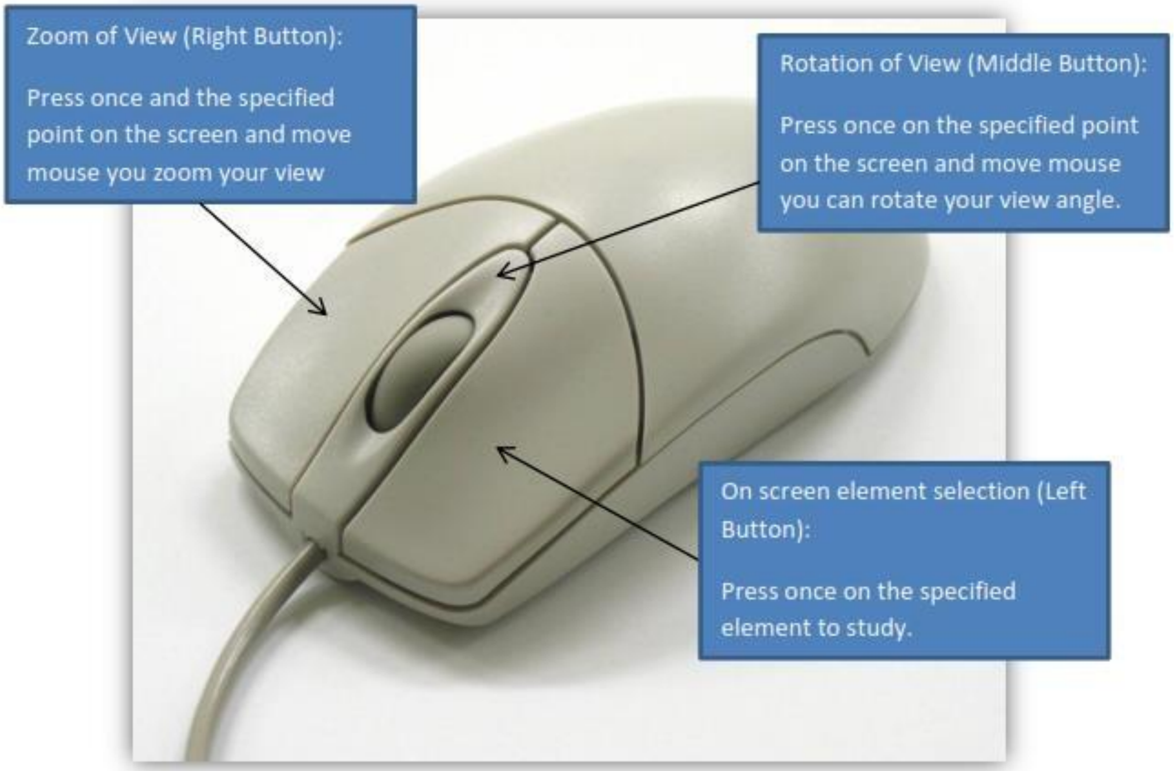


Interface of ANSYS Workbench

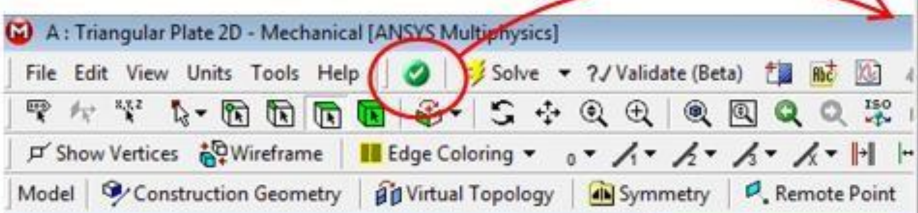
VIEW TOOL BAR

1. **Rotate** : For dynamic rotation of the object
2. **Pan**: To scroll the object around
3. **Zoom**: Zoom in or Zoom out the view by dragging LMB up or down respectively
4. **Box Zoom**: Zooms to the view enclosed by the box created by dragging the LMB
5. **Fit Model to Screen**: Zooms in to fit the full model in the screen
6. **Magnifier Window**: Zoomed in model will be displayed in a new window
7. **Previous View**: Works as Undo for the Displays
8. **Next View**: Works as Redo for the displays
9. **Set Iso View**: Set the model in Isometric view
10. **Display Plane**: Toggle to display the plane on or off
11. **Display Model**: Toggle to display the model on or off
12. **Display Points**: Toggle to display the points on or off
13. **Look at**: Orients the view normal to the selected Face or active Plane, or active Sketch

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

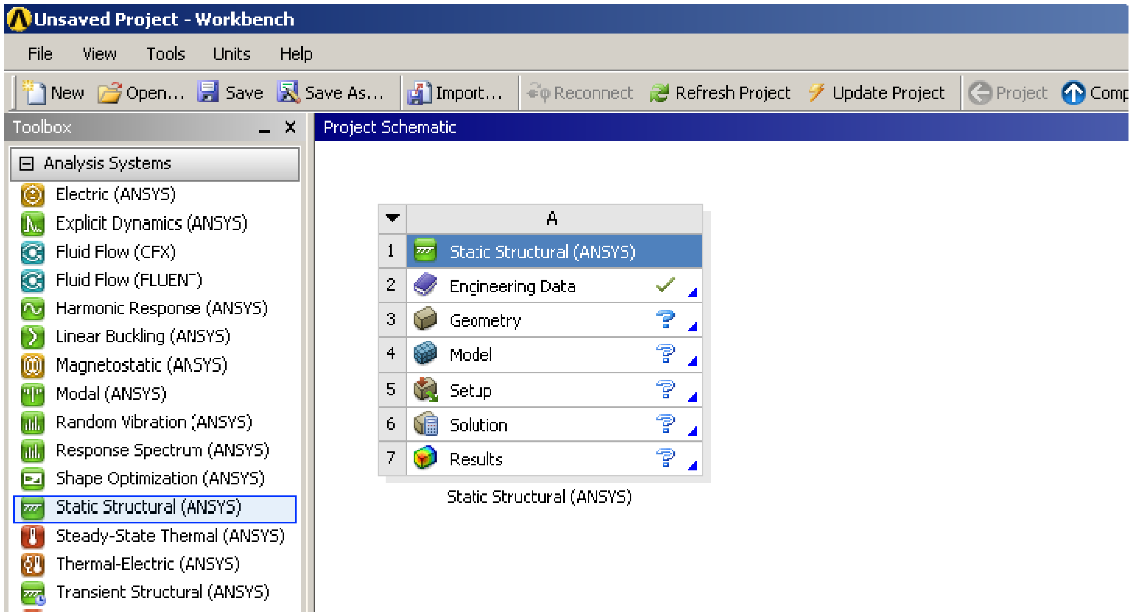


Mechanical Wizard



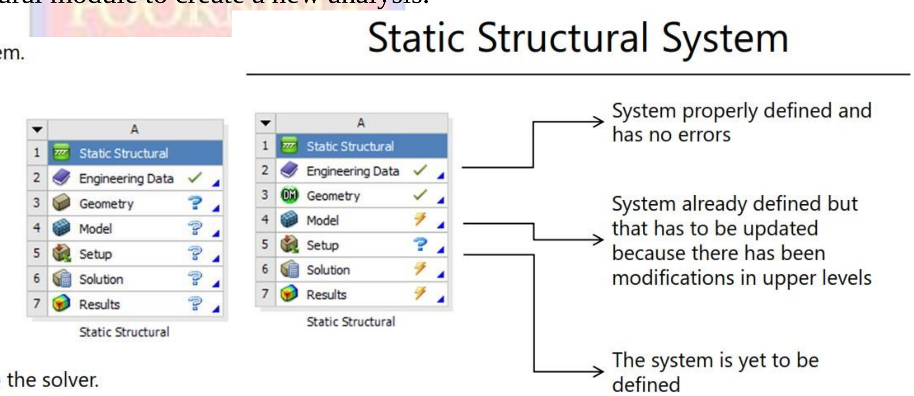
- The Mechanical Wizard provides a list of required steps and the status of them
- Green checkmark indicates the item is complete
- Green "i" shows an informational item
- A greyed symbol shows that the step cannot be performed yet
- A red question mark means that there is an incomplete item
- An "x" means that the item is not performed yet
- A lightning bolt means that the item is ready to be solved or updated

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



Drag-and-drop the Static Structural module to create a new analysis.

We will work with the Static Structural system.
It is composed by:
Engineering data:
Definition of **material properties**
Geometry:
Def. of the problem **geometry**
Model:
Def. of the **mesh**
Setup:
Def. of the **boundary conditions**
Results:
Contains the different results requied to the solver.



Engineering Data

Use the Engineering Data cell with Mechanical systems or the Engineering Data component system to define or access material models for use in an analysis. Double-click the Engineering Data cell, or right mouse click and choose **Edit** from the context menu to display the Engineering Data workspace to define material data.

Geometry

Use the Geometry cell to import, create, edit or update the geometry model used for analysis. Right mouse click to display the context menu to access these functions. The right mouse button options are context sensitive and will change as the state of your geometry changes. All geometry-specific options are described here; not all will be available at all times.

Model/Mesh

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. 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.

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.

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.

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

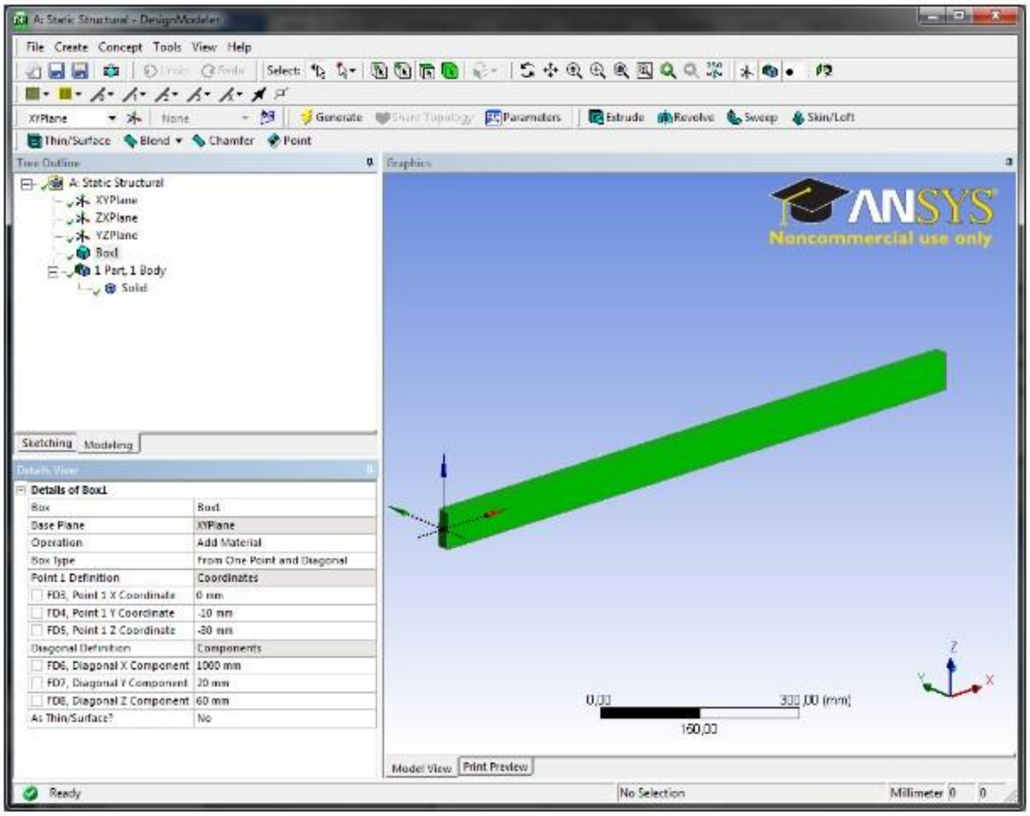
Design Modeler
If instead of importing the geometry from another code or from another problem defined in the same workbench we decide to create a new geometry, clicking on NEW GEOMETRY will launch the designing modeler.

Build the Geometry Using Design Modeler Module

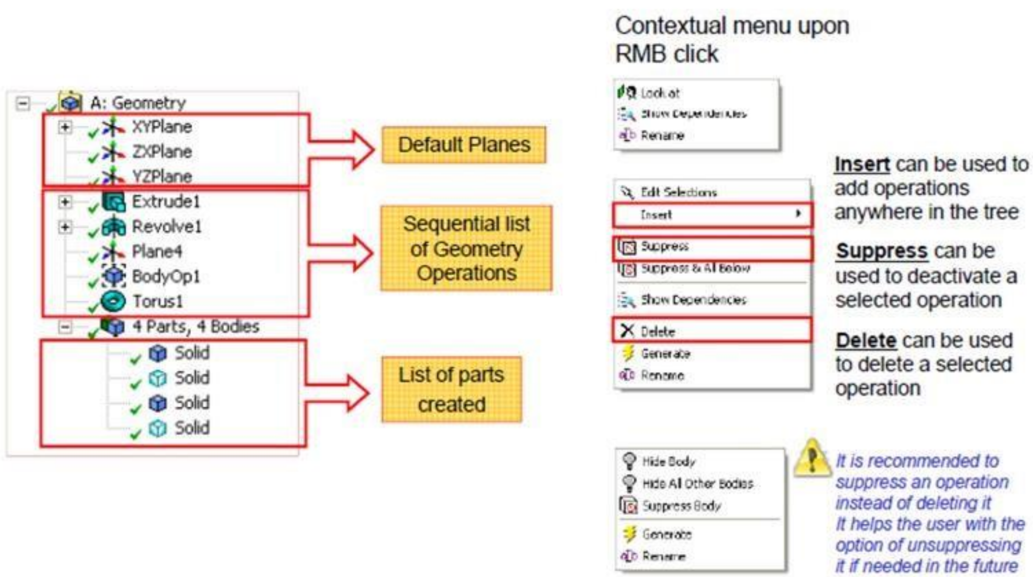
To begin; drag the **Static Structural Module** from the **Analysis Systems** toolbox and drop into the **Project Schematic** and double click on the **Geometry** sub module to open the **DesignModeler**. When asked, choose the desired unit system.

Choose **Create** → **Primitives** → **Box** from the main menu to create a beam by entering the desired dimensions in the **Details** box. Hit the **Generate** button to actually create the geometry.

When specifying the dimension of the beam, make sure that the origin of the coordinate system is located on the center of the beam's cross section. The beam axis must be oriented along the global x-axis.

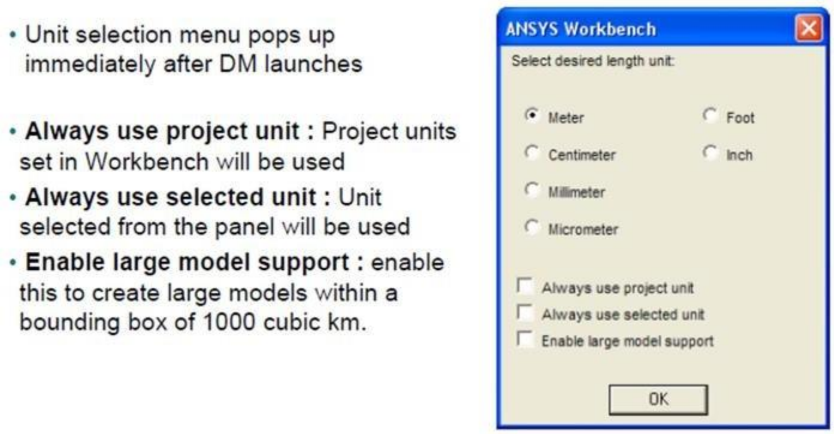


The beam on the Design Modeler



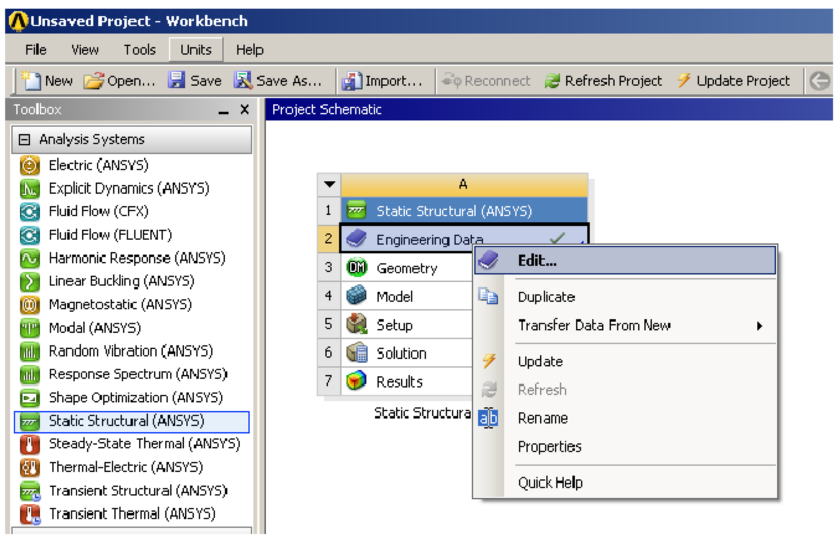
Note that ANSYS Workbench saves the model automatically; you can simply close **Design Modeler** now and continue with the next step.

Define Units:- ***Units Menu*** Use this menu to select the Units system you want to use.



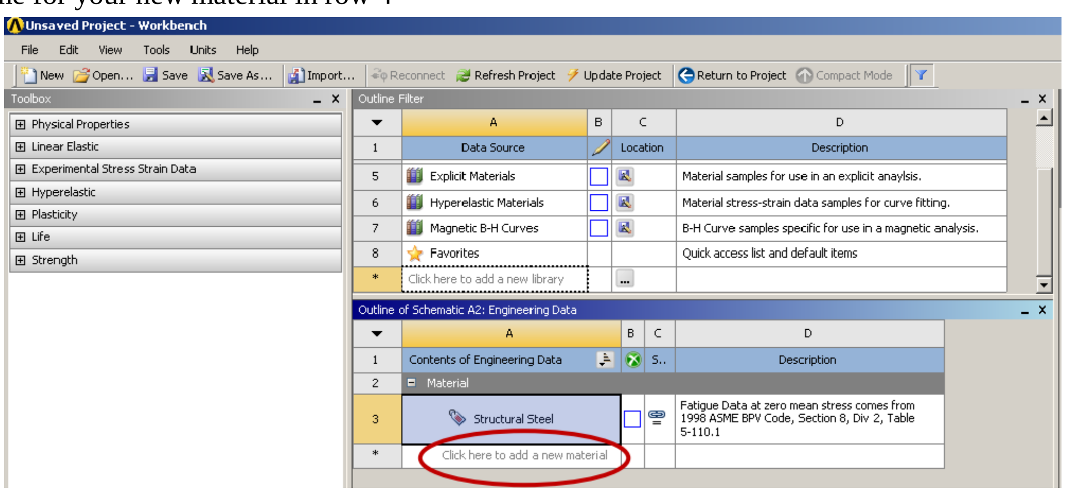
Material Properties

1. Define Material Properties Materials define the mechanical behavior of the FE model. We will use a simple linear- elastic, isotropic material model. In the project schematic, right click on **Engineering Data** to open a context menu and choose **Edit…**



Edit engineering data

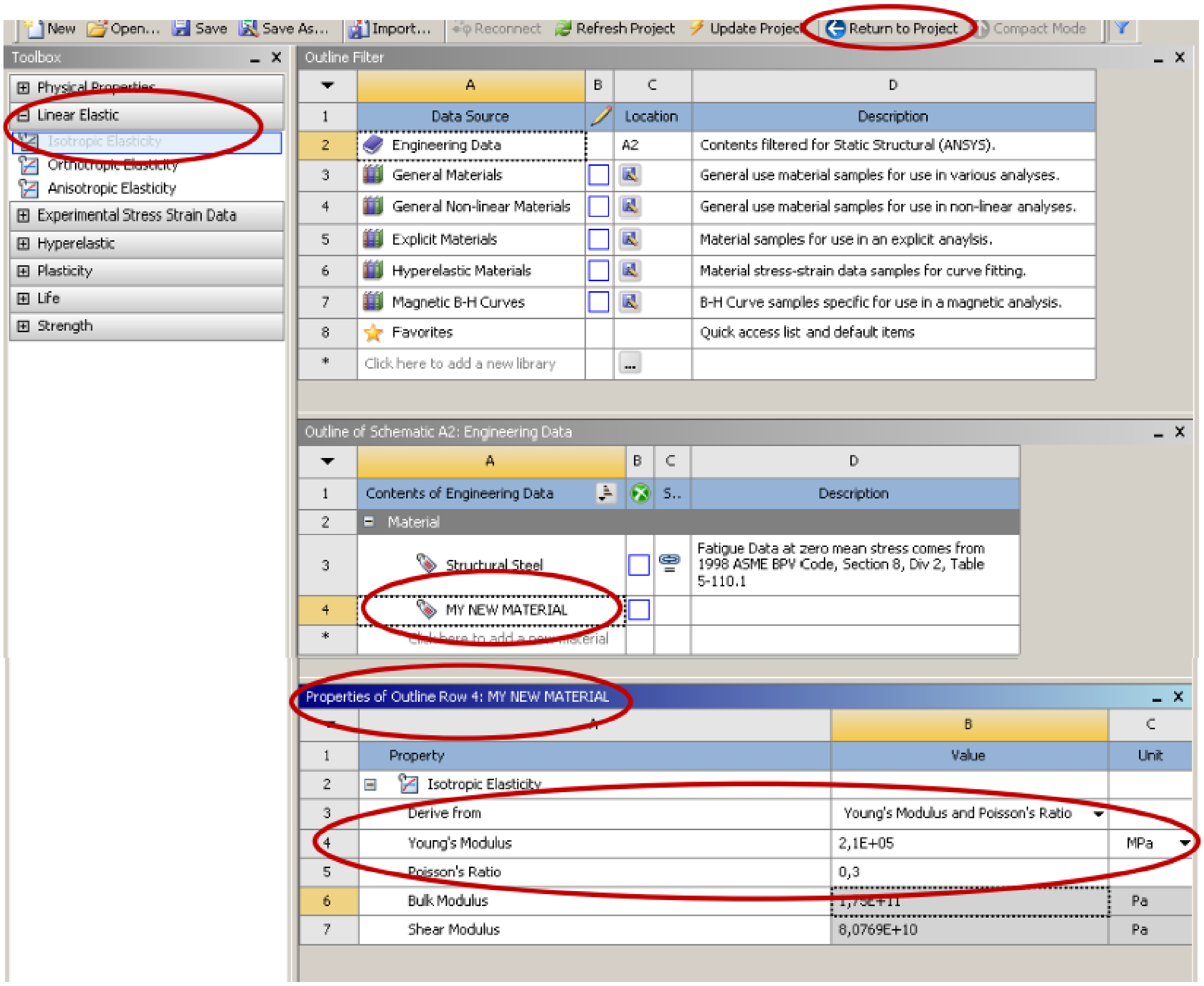
Enter a name for your new material in row 4



Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

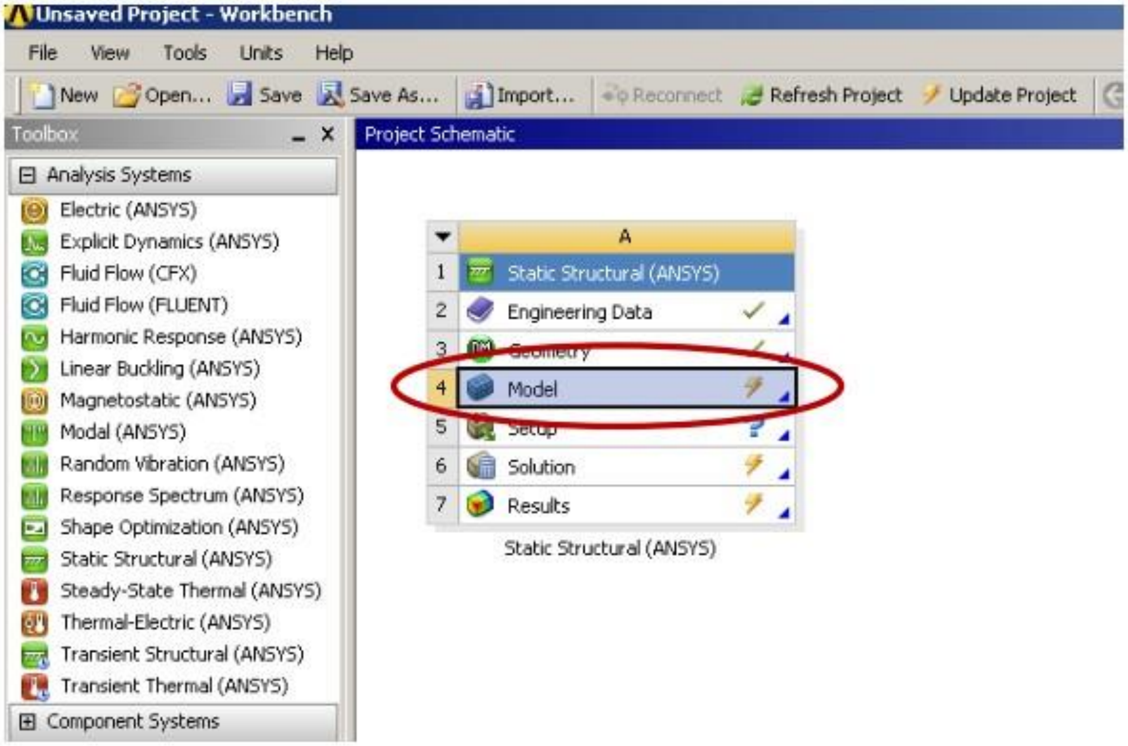
Naming the new material

Choose the simplest available material model by dragging the item **Isotropic Elasticity** from the **Toolbox** and dropping it onto the row with your newly defined material. **Isotropic Elasticity** requires certain material parameters: **Young's Modulus** and **Poisson's Ratio.**
After you entering the appropriate parameters do not forget to push **Return to Project** button and to update the project.



Meshing To mesh the solid body select double click the **Model** sub-module in the Project schematic to open the **Mechanical** module.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



Starting the meshing and analysis module

Right click on **Mesh** in the **Structure Tree** and select a meshing method Mesh (right click) Insert Mapped Mesh

At this point focus on the **Details** box and select the geometry (all 6 faces) and do not forget to update the project; then click the **Generate** button.

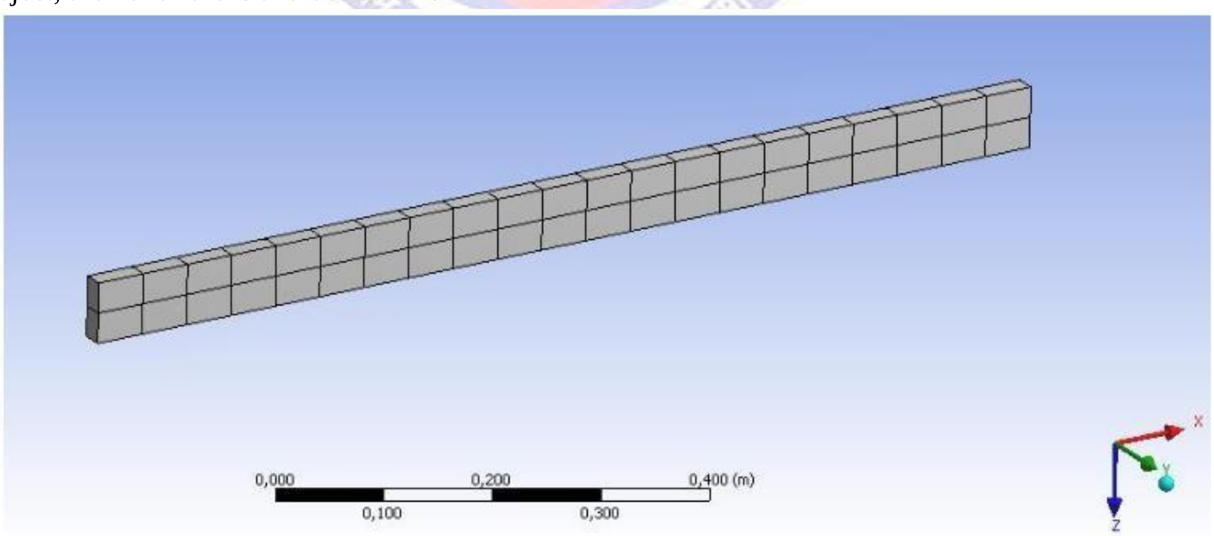
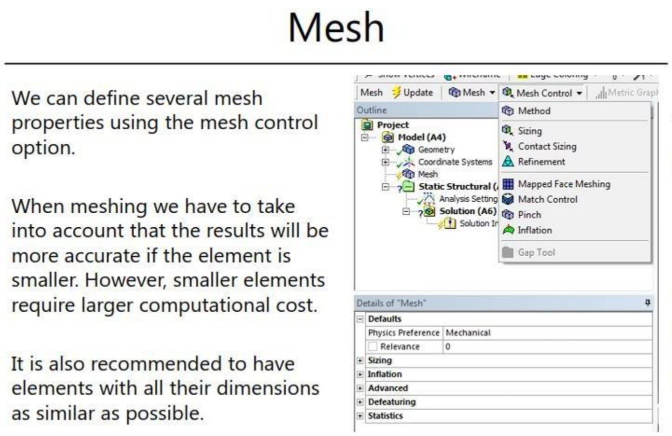
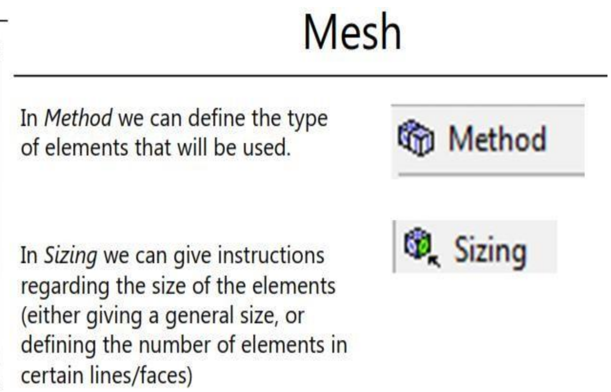


Figure 13: Meshed body

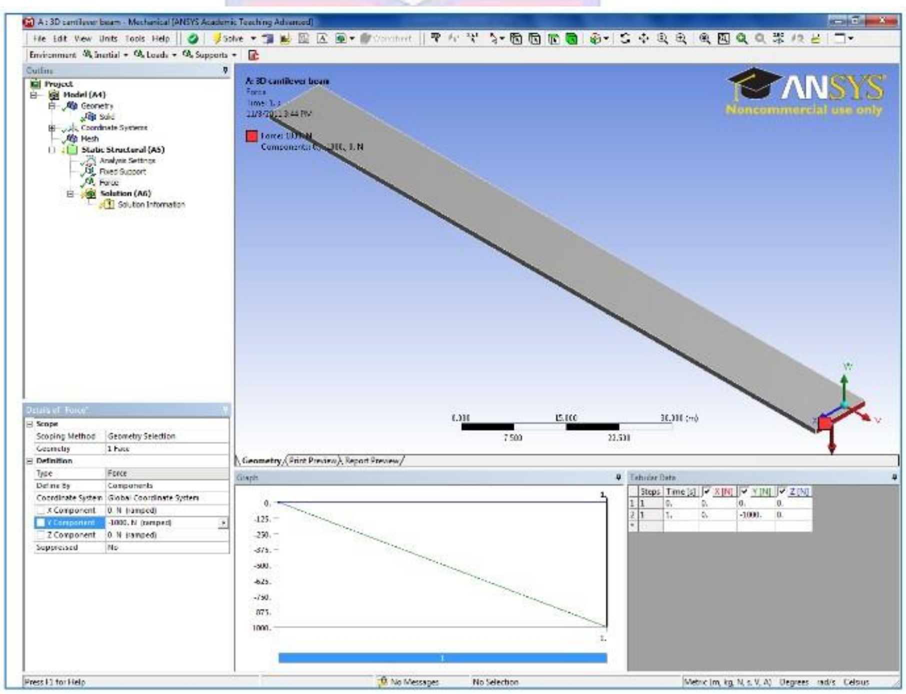
Meshed body

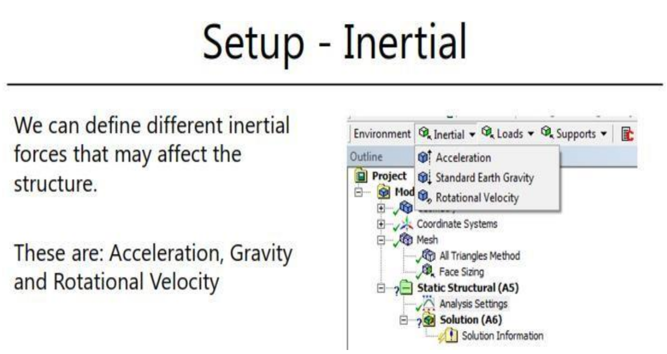




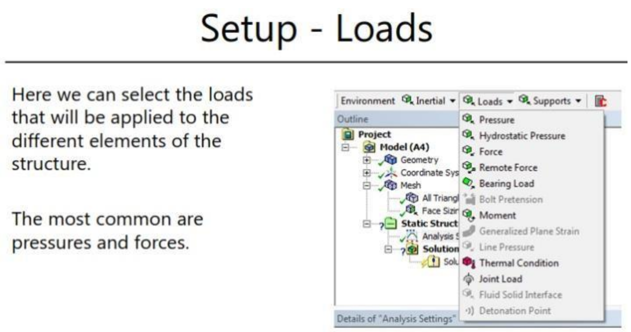
Applying Loads and Boundary Conditions

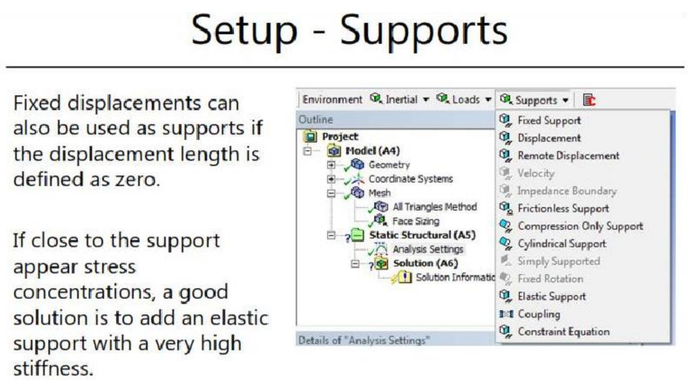
Boundary Conditions To setup the necessary support: *Structure Tree* □ *Static-Mechanic (right click)* □ *Insert* □ *Fixed Support* **Select the Face** you want to assign as a fixed support. In the same way apply the load.

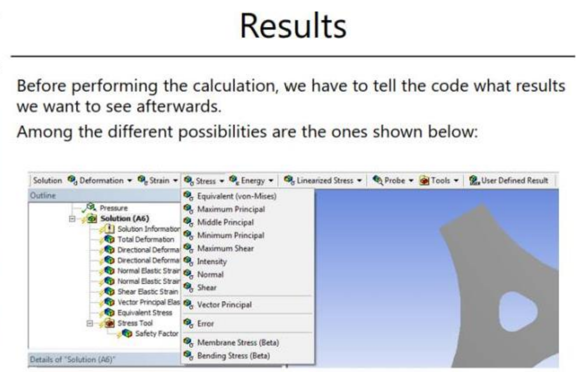




Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

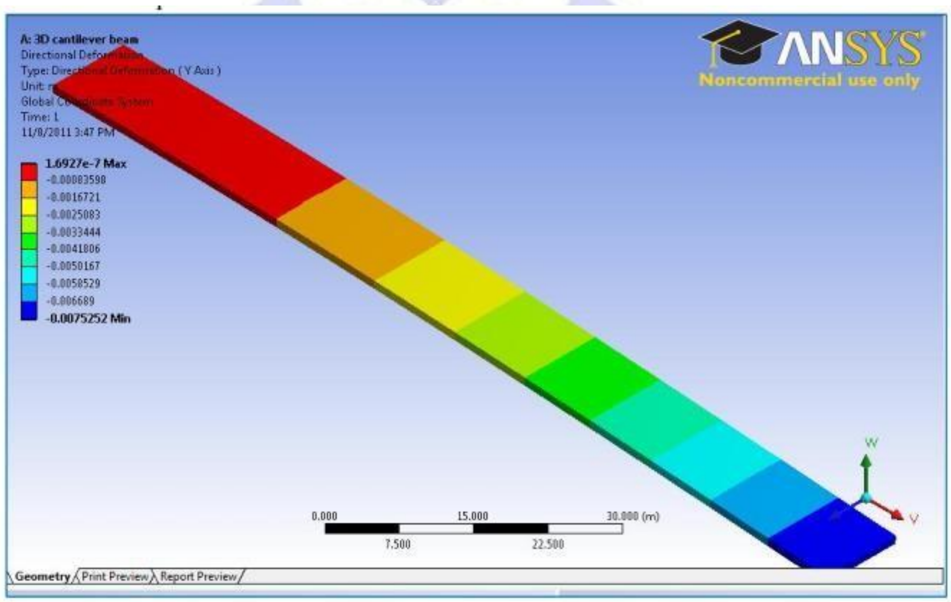






Solving
To solve the problem add some solving tools to the **Solution** node which is part of the **Structure tree.**
Solution (right click) Total Deformation
Solution (right click) Total Strain
Solution (right click) Total Stress

Click **Solve**



Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Results Animation

An **Animation** toolbar is available to view results from Simulation or from DesignXplorer. Access the toolbar from Simulation or from DesignXplorer. The toolbar is presented below along with descriptions of each animation control.

	Play: Initiates a new animation.
(same toolbar location as **Play**)	**Pause**: Pauses an existing animation. Choosing **Play** after **Pause** does *not* generate new animation frames. When the animation is paused, as you move the cursor across the **Timeline** controller, the cursor's appearance changes to a double horizontal arrow when you hover over the current frame indicator. With the cursor in this state, you can drag the frame indicator to define a new current frame. The result graphic will update accordingly.
	Stop: Halts a result animation. Choosing **Play** after **Stop** generates new animation frames.
	Distributed: For static simulations, frames display linearly interpolated results. Frame 1 represents the initial state of the model and the final frame represents the final results calculated by the solver. For sequenced and transient simulations, the frames in **Distributed** mode are distributed over a time range selected in the **Timeline** controller.[1]
	Result Sets: (available only for sequenced and transient simulations) Frames represent the actual result sets that were generated by the solver.[1]

ANSYS Files

- There are various types of files used by Workbench, each differentiated by the file extension:

– *.wbdb*: Workbench Project database, which keeps track of all of the different types of Workbench databases in a "Project"

– *.dsdb*: Simulation database, which has all the information necessary to perform a structural or thermal analysis in Simulation

– *.agdb*: DesignModeler database, containing geometry data for use with Simulation or CFX-Mesh

– *.dxdb*: DesignXplorer/DesignXplorer VT database, which investigate relationships between input and output parameters

– *.cmdb*: CFX-Mesh database, containing a mesh prepared for import to CFX-Pre and to be solved with CFX-5

– *.fedb*: FE Modeler database, which has mesh information from a Nastran or Simulation model, used to convert to ANSYS

Viva Questions

1. What do you mean by GUI?
2. What are the File extensions used in ANSYS?
3. What are different modes of analysis possible in ANSYS?
4. What is meant by finite element?
5. What are the steps involved in FEA?
6. What is meant by finite element analysis?
7. Mention some application of FEA
8. What are the packages available for FEA
9. Difference between FEA and FEM?
10. What is the current ANSYS VERSION?

Experiment No.2 Structural Analysis of a Cantilever Beam

The objective of this problem is to demonstrate a simple ANSYS Workbench problem with a textbook solution: finding Von Mises' stresses and total deflection throughout the beam. This is a simple, single load step, structural analysis of a cantilever beam. The left side of the cantilever beam is fixed while there is a distributed load of 20N/m.

Given Data:-

L =110m Length of beam

b =10m Cross Section Base

h =1 m Cross Section Height

w=20N/m Distributed Load

E=70GPa Young's Modulus of Aluminum at Room Temperature

μ =0.33 Poisson's Ratio of Aluminum

(1) Von Mises Stress

Assuming plane stress, the Von Mises Equivalent Stress can be expressed as:

$$\sigma' = (\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}} \qquad \text{(i)}$$

Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

$$(\tau_{xy} = 0,\ \sigma_y = 0) \qquad \text{(ii)}$$

Using these simplifications, the Von Mises Equivalent Stress from equation (i) reduces to:

$$\sigma' = \sigma_x \qquad \text{(iii)}$$

Bending Stress is given by:

$$\sigma_x = \frac{P(x-L)c}{I} \qquad \text{(iv)}$$

Where $I = \frac{1}{12}bh^3$and $c = \frac{h}{2}$. From statics, we can derive:

$$\sigma_x = \frac{6P(x-L)}{bh^2} = 66kPa$$

(2) <u>Beam Deflection</u>

$$\frac{d^2y}{dx^2} = \frac{M(x)}{EI}$$

Using Shigley's Mechanical Engineering Design, the beam deflection is:

$$\delta(x) = \frac{Px^2(x-3L)}{6EI}$$

With Maximum Deflection at:

$$\boldsymbol{\delta = \frac{PL^3}{3EI} = 7.61\text{mm}}$$

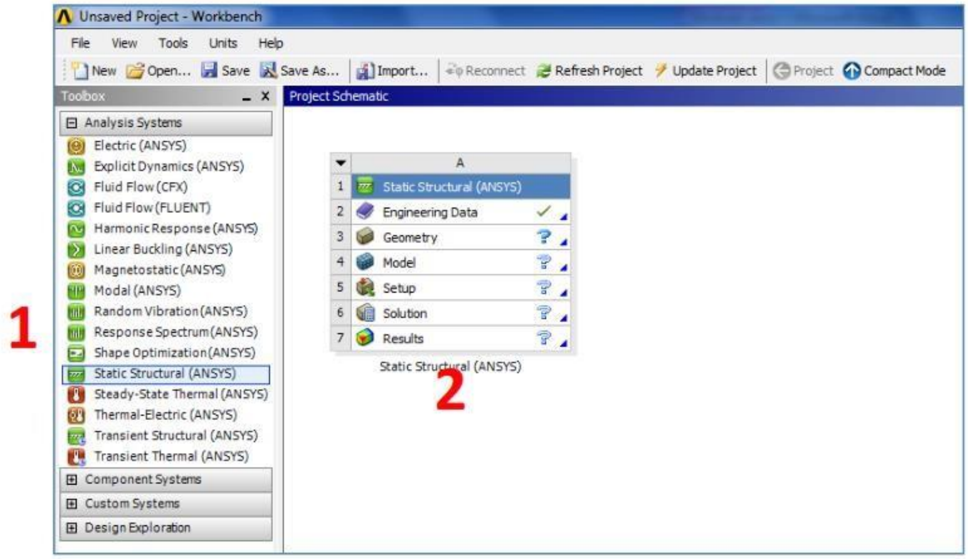
<u>Workbench Analysis System</u>

<u>Opening Workbench</u>

1. On your Windows 7 Desktop click the **Start** button.
2. Under **Search Programs and Files** type "*ANSYS*"
3. Click on **ANSYS Workbench** to start workbench. This step may take time.

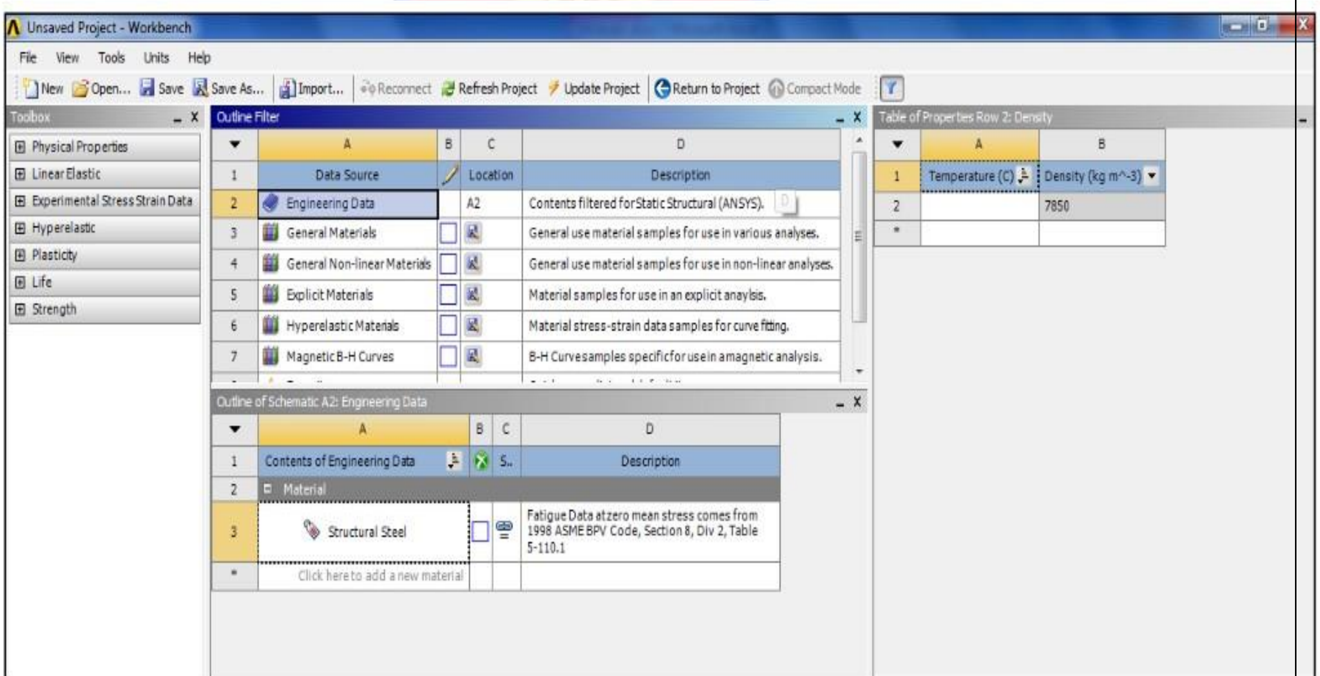
<u>Static Structural Analysis</u>

1. As you open ANSYS you can see the entire array of problems on the left had side this software can help you solve. The problem at hand is a *Static Structural* problem. Double click **Static Structural (ANSYS)** to open the task manager for your problem set in the Project Schematic area.
2. ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see **Static Structural (ANSYS),** double click this to change the name. For this problem choose "*3D Cantilever beam*."



Engineering Data

To begin setup for your cantilever beam, double click or right click on ***Engineering Data*** and click ***edit***. This will bring up another screen.

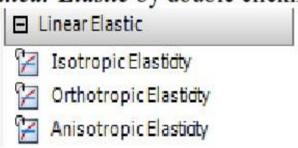


This new window will allow you to alter the material properties of your cantilever beam. Under **Outline of Schematic A2: Engineering Data**, it shows ***click here to add a new material***, this menu allows you to input the material of your cantilever beam, double click and type ***Aluminum***.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Now expand ***Linear Elastic*** by double clicking on 

or on the plus symbol shown.

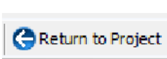


Double click on Isotropic Elasticity to give the material the same properties across the beam. This action brought up a new table on the right; this allows us to add necessary properties. As show on the top right of the screen in ***Table of Properties Row 2: Isotropic Elasticity***:

1. Click in Temperature and type 25
2. Click in Young's Modulus and type 70E9 or 7E10
3. Click in Poisson's Ratio and type 0.33
4. Make sure to DELETE the Temperature entry after property input before continuing! Failure to do so will lead to errors later.

Table of Properties Row 2: Isotropic Elasticity

	A	B	C
1	Temperature (C)	Young's Modulus (Pa)	Poisson's Ratio
2	25	7E+10	0.33
*			

After filling in the properties, this concludes the Engineering Data, to return to the project schematic area, click on 

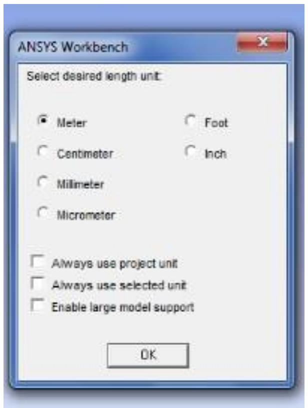
seen on the upper tab.

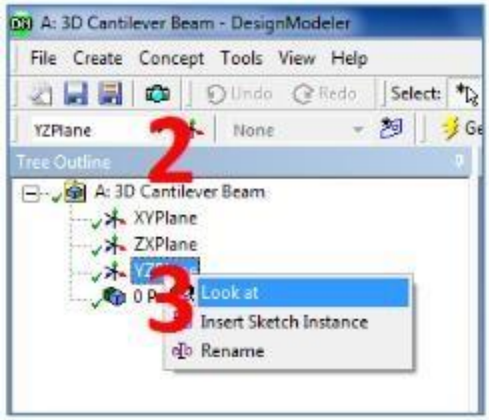
<u>Geometry</u>

Base Geometry

1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR





Note: Select meters and hit ok

2. In the new window, click the **Display Plane** icon to toggle the coordinate system.

3. Go to **Design Modeler > Tree Outline >** right click on YZPlane. Click **Look At** to view the YZ plane.

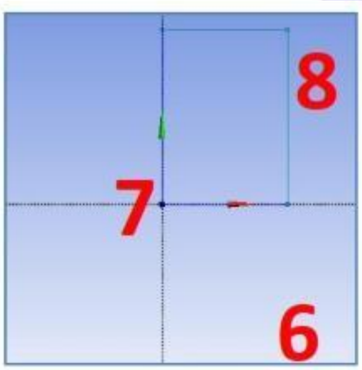
4. Go to **Design Modeler -> Tree Outline -> Sketching**

5. Click on **Rectangle** and Click off **Auto-Fillet:**

6. Bring your cursor into the workspace at point 0,0, over the origin until 'P' appears directly above the origin.

7. Click on the origin to place the lower left corner of our rectangle on the origin.

8. Click on a point in the first quadrant to define the top right corner of our rectangle. The point is arbitrary as we will be fixing dimensions momentarily.



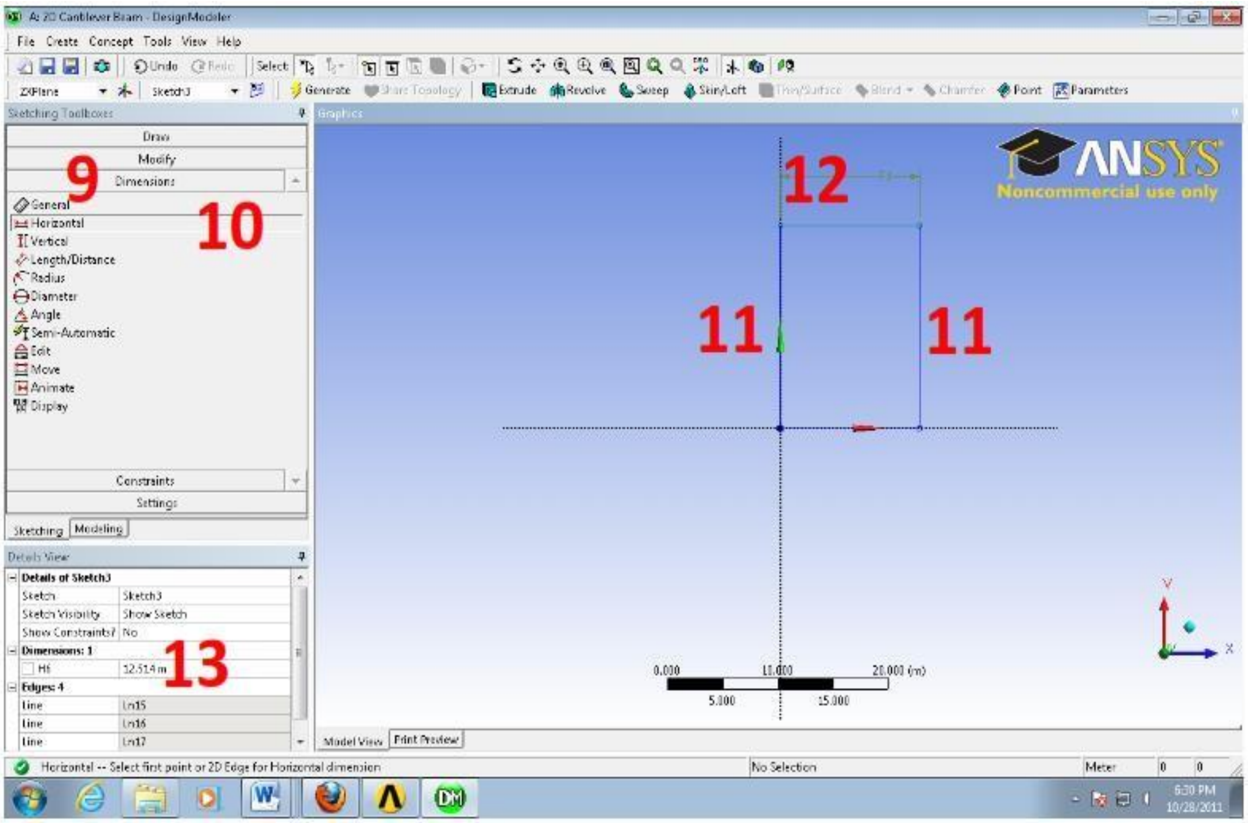
9. Go to **Sketching Toolboxes -> Dimensions**

10. Click **Horizontal** to specify a horizontal dimension.

11. Click the left and right faces of the rectangle in the sketch to specify that we will be dimensioning this horizontal length. A green line with a symbol should appear.

12. Drag the green line above the sketch and click to set its location.

13. Go to **Detail View -> Dimension 1.** In the first subcategory, replace the current dimension with 10. The units should populate automatically.



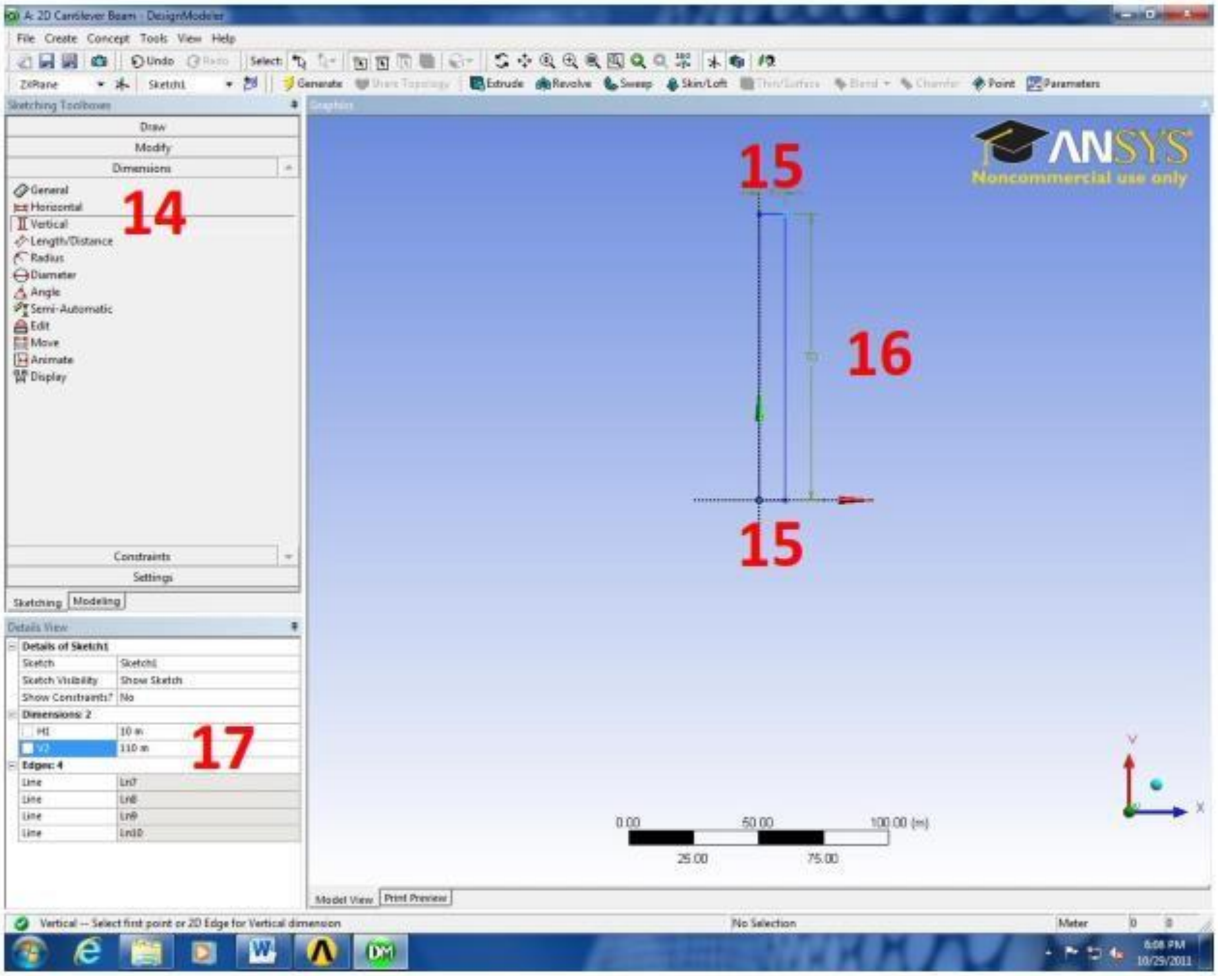
14. Go to **Sketching Toolboxes -> Dimensions -> Vertical** to specify the vertical dimension.

15. Click the bottom and top faces of the sketch to specify the vertical dimension. A green line should appear.

16. Drag the green line to the right of the sketch and click.

17. Go to **Detail View -> Dimension 2**. Replace the value with 10. The units should populate automatically (meters).

Now that we have modeled the base geometry, we will extrude it to create a 3D volume.



Extrude Sketch

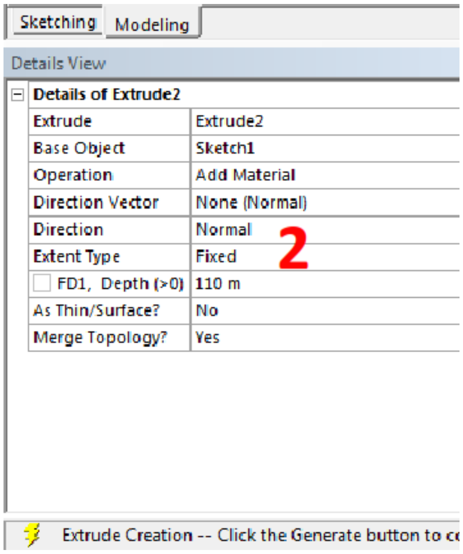
1. Go to **Main Toolbar ->** and select **Extrude**

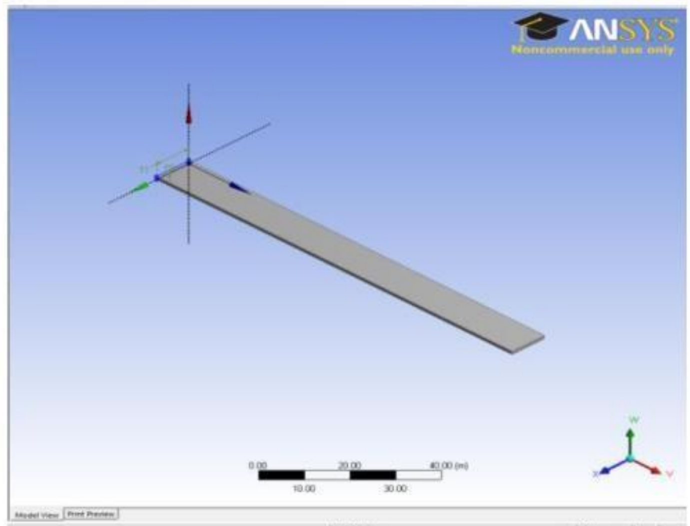
2. Go to **Modeling -> FD1, Depth (>0) ->** enter in **110**

3. Go to **Design Modeler ->** **Generate**.

4. To verify our geometry, look at the isometric view. Click the *blue dot* in the **triad** in the lower right corner of the screen to look at the isometric view.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR





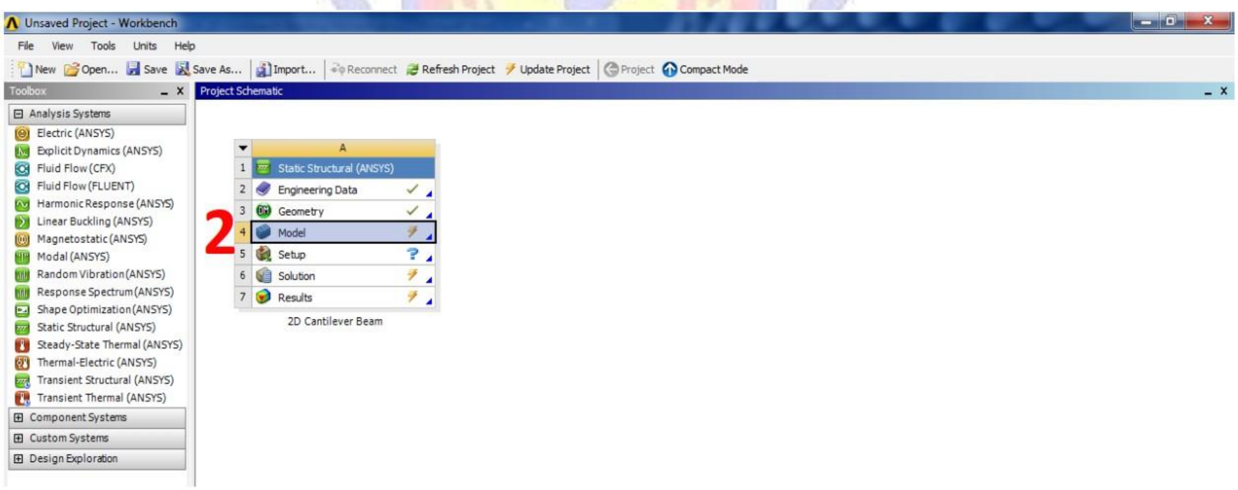
Now that we have the geometry, we will mesh the beam using 3D Elements.

Model

Open ANSYS Mechanical

1.  out of Design Modeler. Don't worry, your work will be saved.

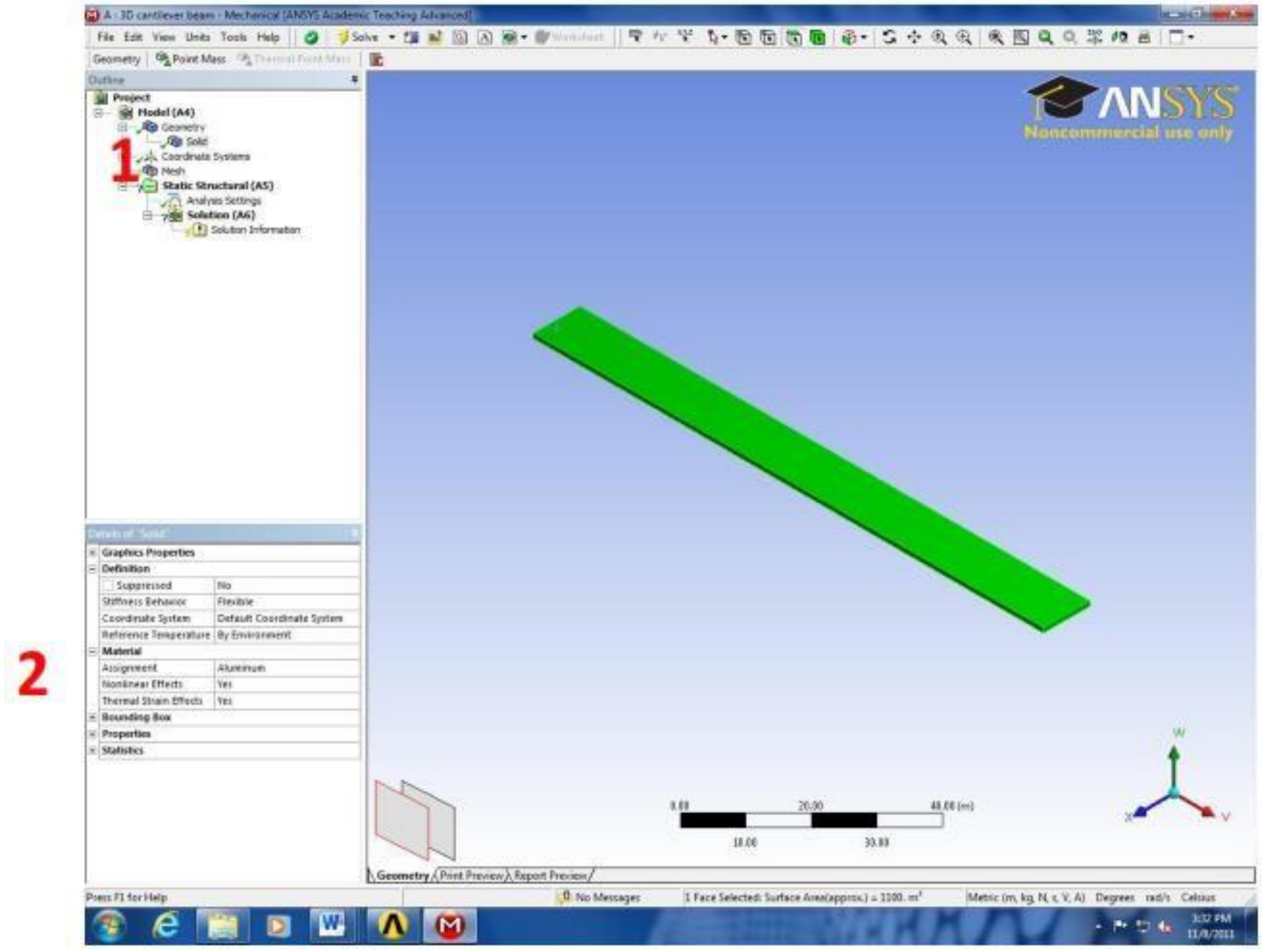
2. Go to **Workbench -> Project Schematic -> Model** This will open *ANSYS Mechanical*



Material Assignment

1. Go to **Mechanical -> Outline -> Project -> Model -> Geometry -> Solid Body**

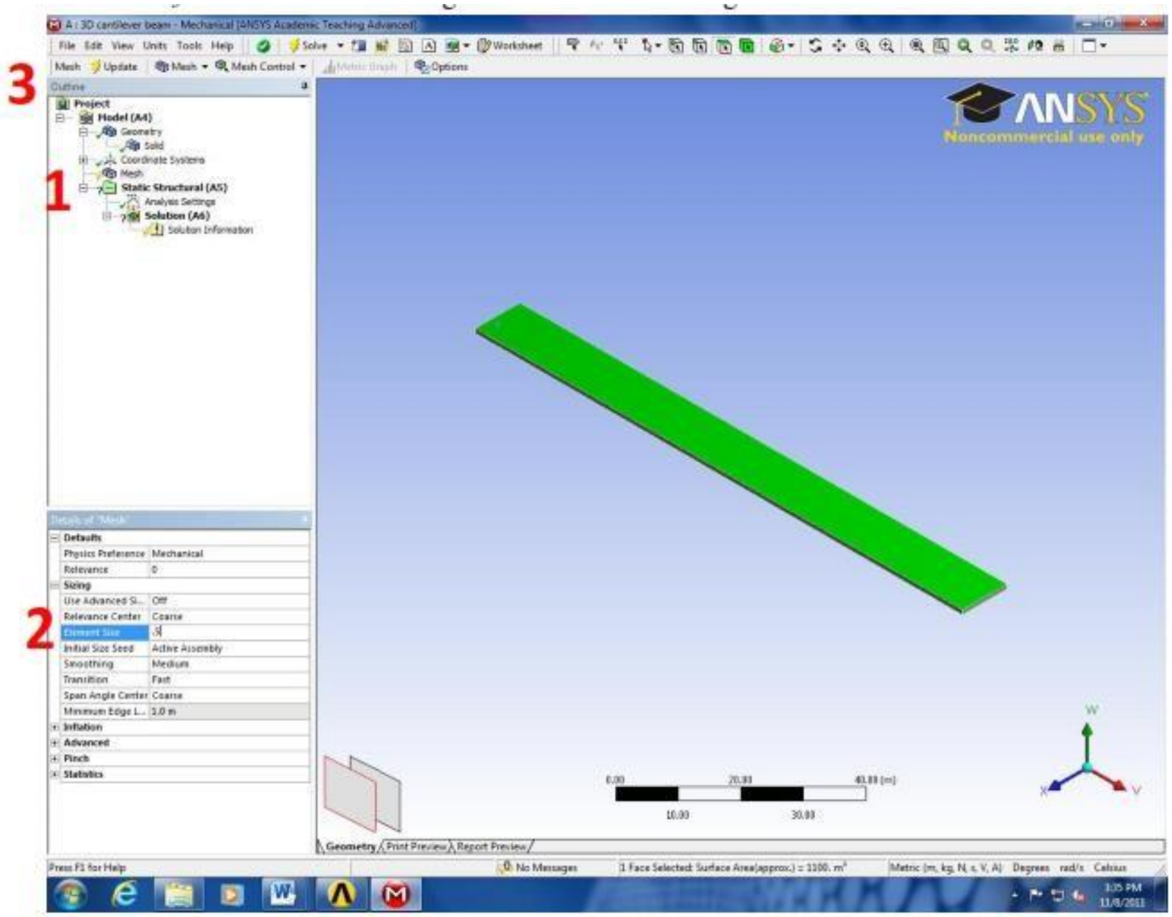
2. Under **Mechanical -> Details of "SolidvBody" -> Material -> Assignment**, change *Structural Steel* to *Aluminum*.



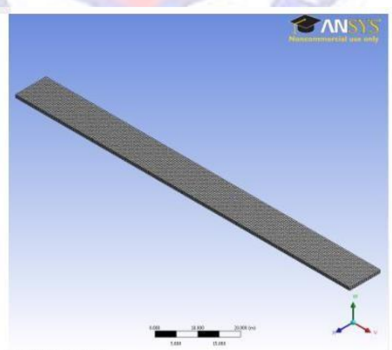
Mesh

1. Go to **Mechanical -> Outline -> Project -> Model -> Mesh**

2. Go to **Mechanical -> Details of 'Mesh' -> Sizing -> Element Size** and change the value from *Default* to .5 m. This will give us 2 elements through the thickness of the beam.



3. Click **Mechanical ->Update**. This may take some time. Your mesh should look as shown below:



Setup

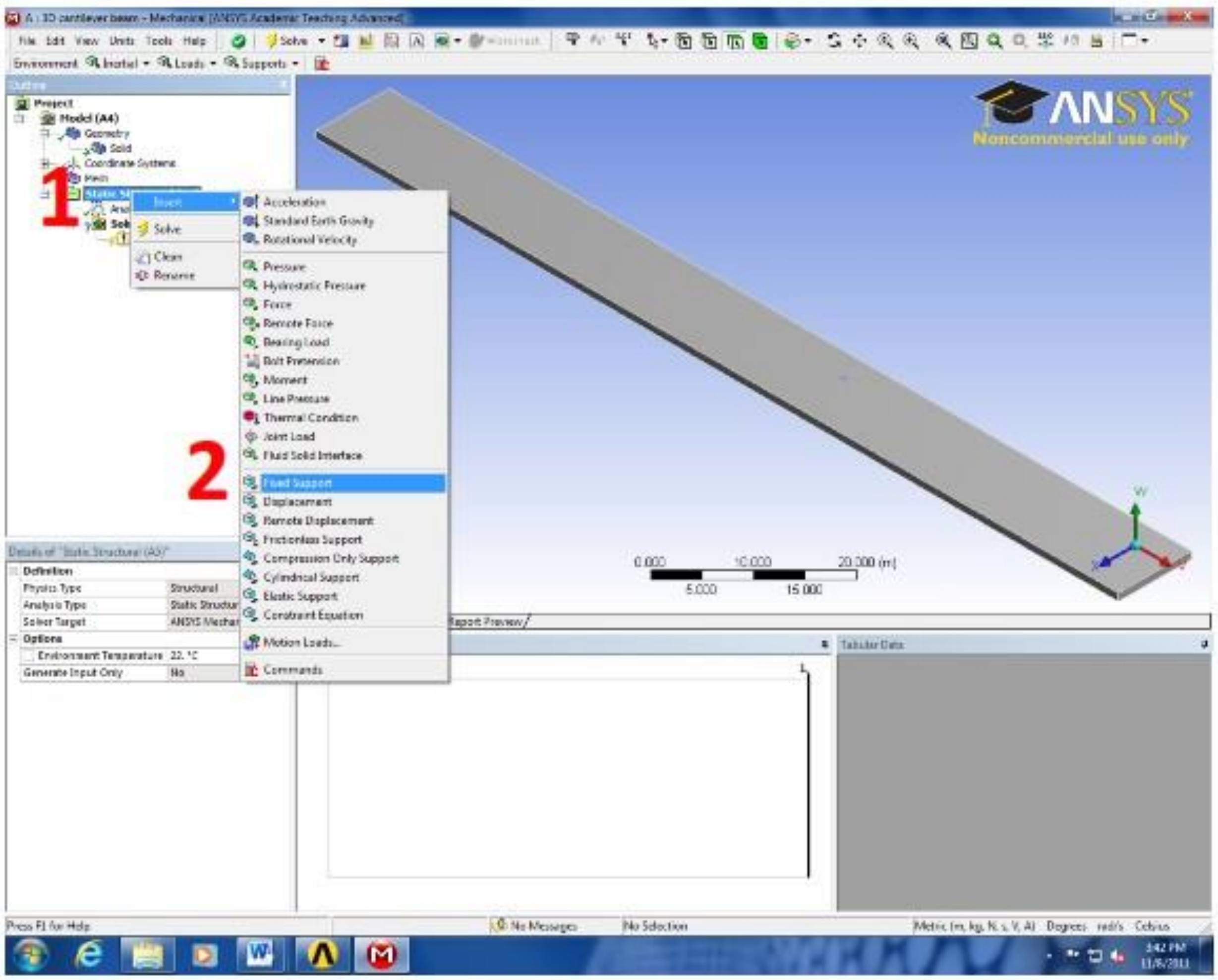
You can perform the rest of your analysis for this problem in the *ANSYS Mechanical* window. The other options in the *Workbench* window will link you back to the same screen (i.e. Setup, Solution, Results)

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Fixed Support

1. Go to **Mechanical -> Outline ->** right click **Static Structural (A5)**

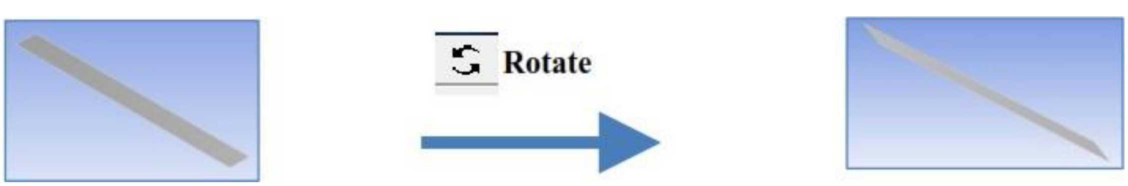
2. Go to **Insert -> Fixed Support**



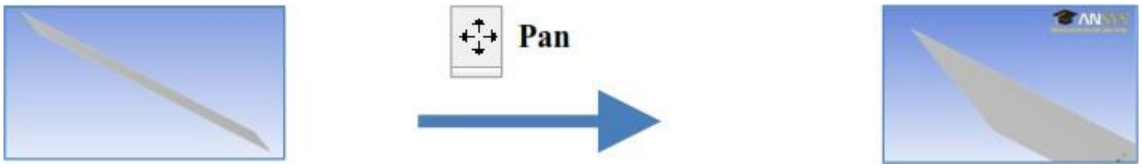
We are going to fix the elements at the left end of the beam. In order to do this, we will use the

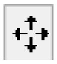
 **Edge** tool to select the left edge. However, from the current orientation of the beam, it is difficult to select this surface.

3. Using the  **Rotate** tool click on the graphic area and move the mouse to the right. This will cause the left end of the beam to be oriented in a manner that can be clicked



Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



4. Using the  **Pan** tool, click the graphic area and drag the left face to the center of the graphic window. Use the *mouse scroll* to zoom in on the left face.

5. Click the  **Edge** tool.

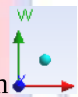
6. Go to **Mechanical -> Outline -> Static Structural (A5) -> Fixed Support**

7. Run the cursor across the left end face. When it becomes red, click it to select it.

8. Go to **Mechanical -> Details of "Fixed Support" -> Geometry** and select **Apply**

While in the Project Schematic double click Setup This will open a new window similar to Model Space

Loads

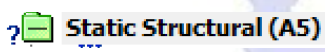
1. Click the x-axis icon to get a side view of the cantilever beam 

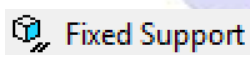
2. Click Fixed end On the tool bar, make sure vertex option is selected. 

3. Go to **Mechanical -> Outline -> Static Structural (A5) -> Fixed Support**

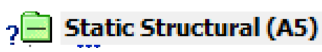
4. Run the cursor across the left end face. When it becomes red, click it to select it.

5. Click the left side of the geometry; this will add a green box to select the point.

6. Right click 

7. Click insert, and 

8. This will add a fixed end to your cantilever beam in the work space.

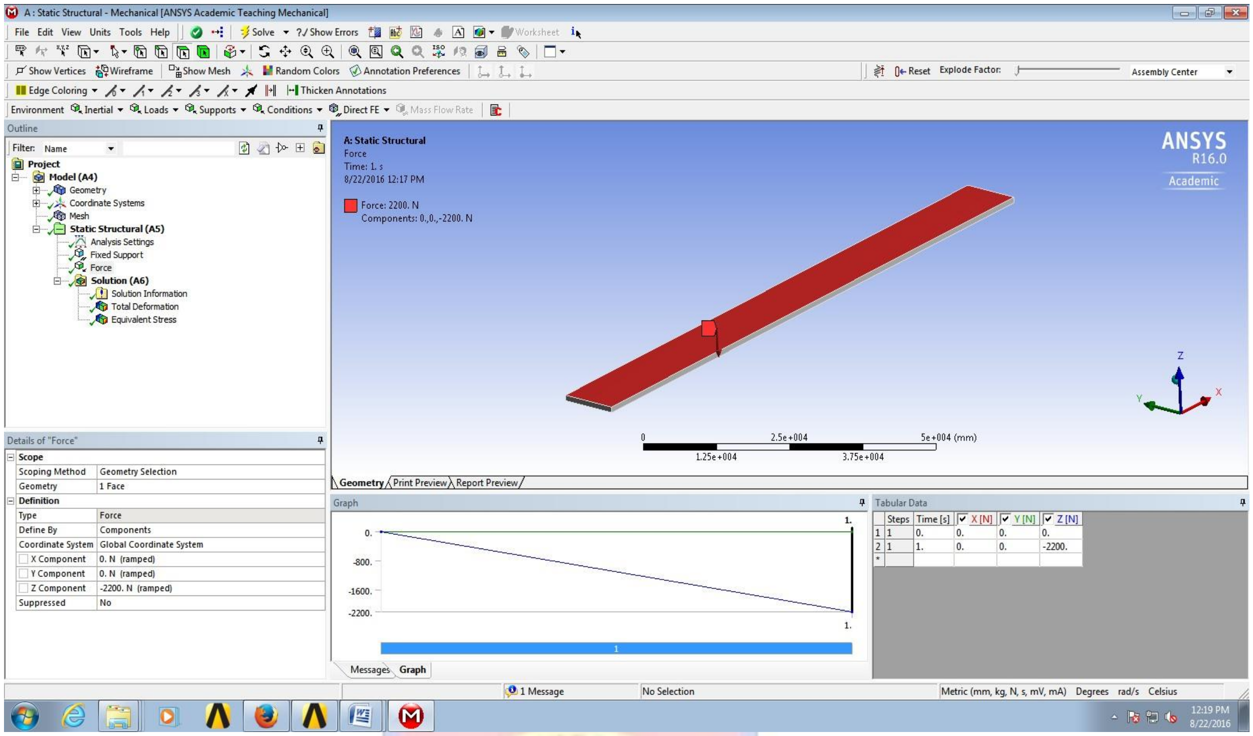
10. Right click 

11. click insert , and A table will appear "Details of Force"

12. Under "Definition" you will see "Defined by" Change this to "Components"

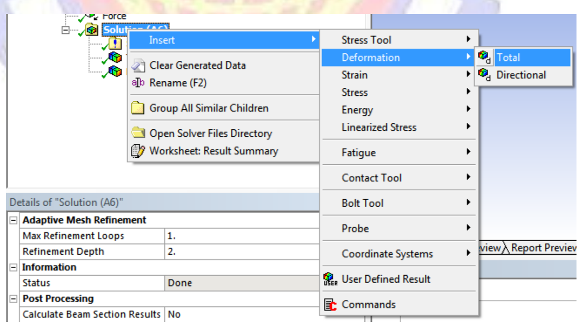
13. As shown, Y Component force is zero. Change this to value to -2200 **(as distributed load)**

14. This will show your cantilever beam with a load applied as shown. Leave the Setup screen open this time.



Solution

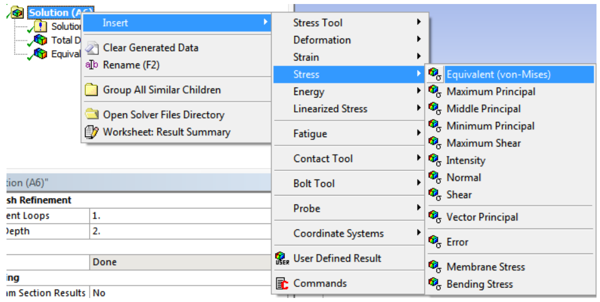
Deformation Go to **Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Insert -> Deformation -> Total**



Equivalent Stress

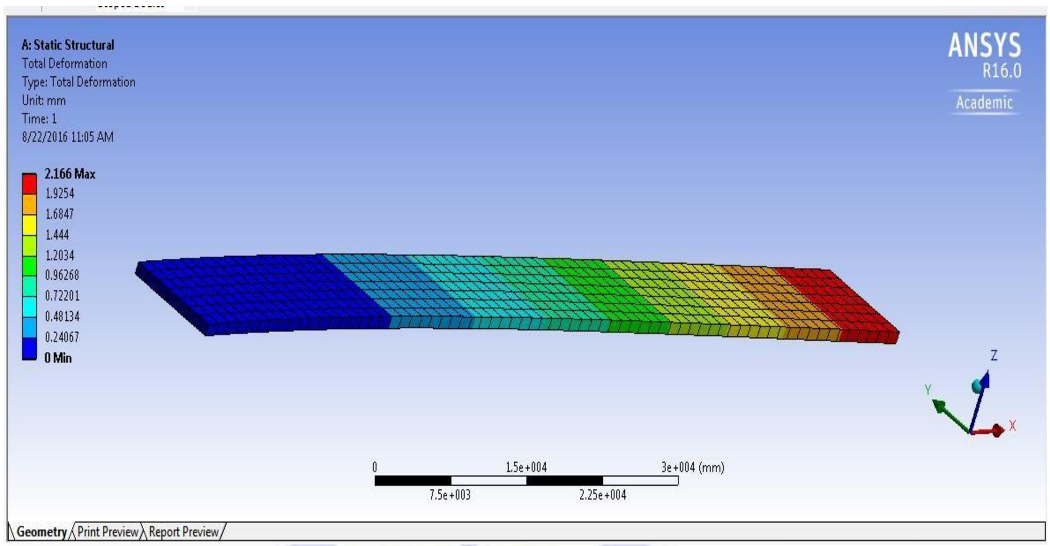
Go to **Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Insert -> Stress -> Equivalent (Von Mises)**

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



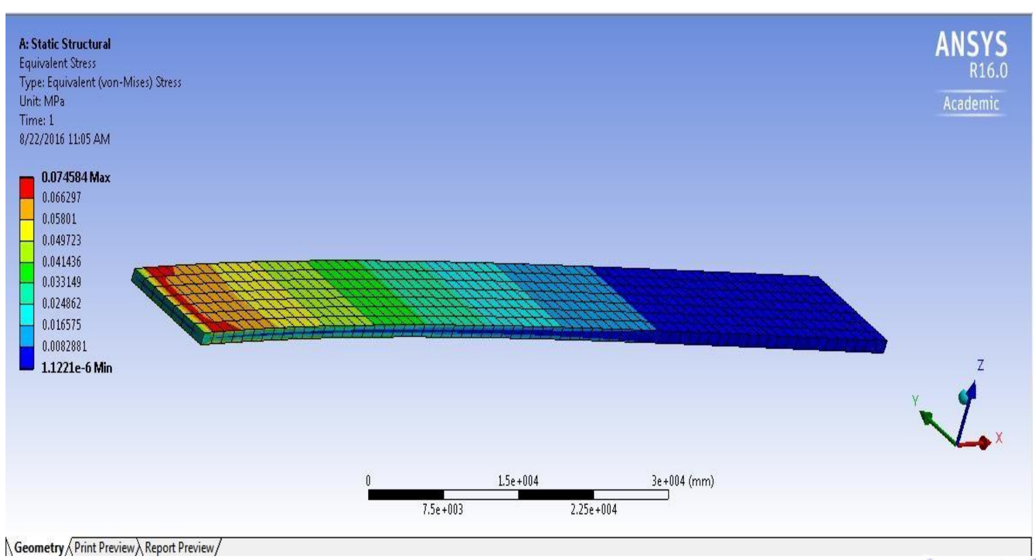
Now that our solvers have been defined, go to **Mechanical ->** **Solve**. The calculations in Workbench may take up to a minute to solve.

Your Stress plot should look as shown below:



Go to **Mechanical -> Outline -> Project -> Model(A4) -> Solution(A6) -> Total Deformation**

Your Von-Mises plot should look as shown below:



Results:-

Max Deformation According to equation, the theoretical max deflection is 7.6 mm. By Ansys it is 6.1 mm. Equivalent Von Mises Stress, theoretically we got 66 KPa and by Ansys we have 76.5 Kpa. The difference is due to Meshing & discritization in Ansys as we have not done theoretical solution

Viva Questions:-

1. What is point load (P)?
2. What is meant by plane stress?
3. Define plane strain.
4. What are isotropic and orthotropic materials?
5. Define term element
6. What do you mean by cantilever beam?
7. Define Young's Modulus and Poisson's Ratio.
8. What is factor of safety?
9. What is bending moment diagram?
10. What is shear force diagram?

Experiment No.3 Steady State Thermal Analysis

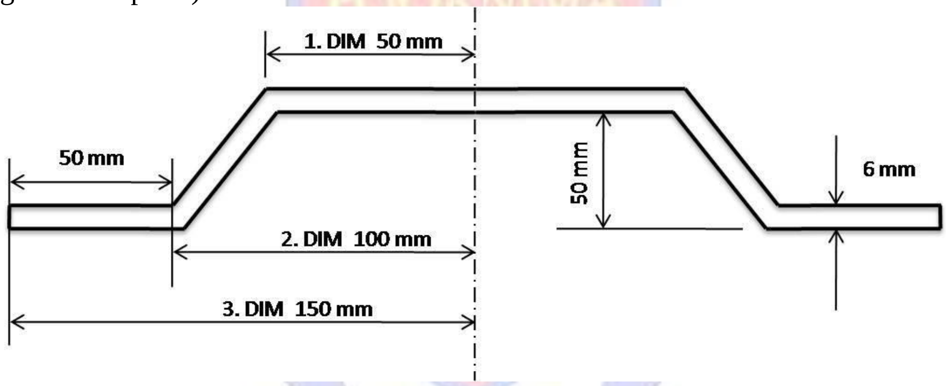
Objective:- Steady State Thermal Analysis by using ANSYS workbench. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. You can use steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries.

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects also makes the analysis nonlinear.

Problem Description:- Steady state thermal analysis of "car brake disc" by using the given below parameters & sketch.
Parameters:- Thermal Material: Aluminum (from Engineering data of Ansys)

Sketch:- (by using revolve option)



Roll No	1. Dimension	2. Dimension	3. Dimension
1-6-11-16-21-26-31-36-41-46-56-61-66-71	50	100	150
2-7-12-17-22-27-32-37-42-47-57-62-67-72	51	101	151
3-8-13-18-23-28-33-38-43-48-58-63-68-73	52	102	152
4-9-14-19-24-29-34-39-44-49-59-64-69-74	53	103	153
5-10-15-20-25-30-35-40-45-50-60-65-70-75	54	106	156

Result:- 1. Maximum Temperature 2. Total Heat Flux

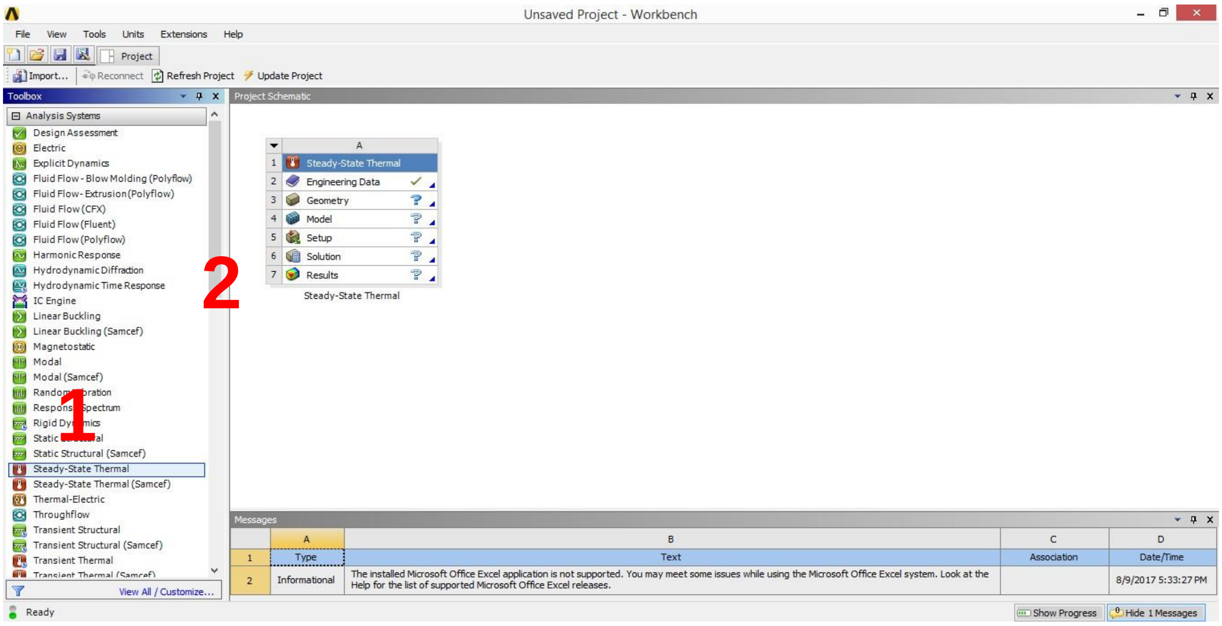
Procedure:-

Opening Workbench

1.On your Windows 7 Desktop click the **Start** button.
2.Under **Search Programs and Files** type "*ANSYS*"
3.Click on **ANSYS Workbench** to start workbench. This step may take time.

Static Structural Analysis

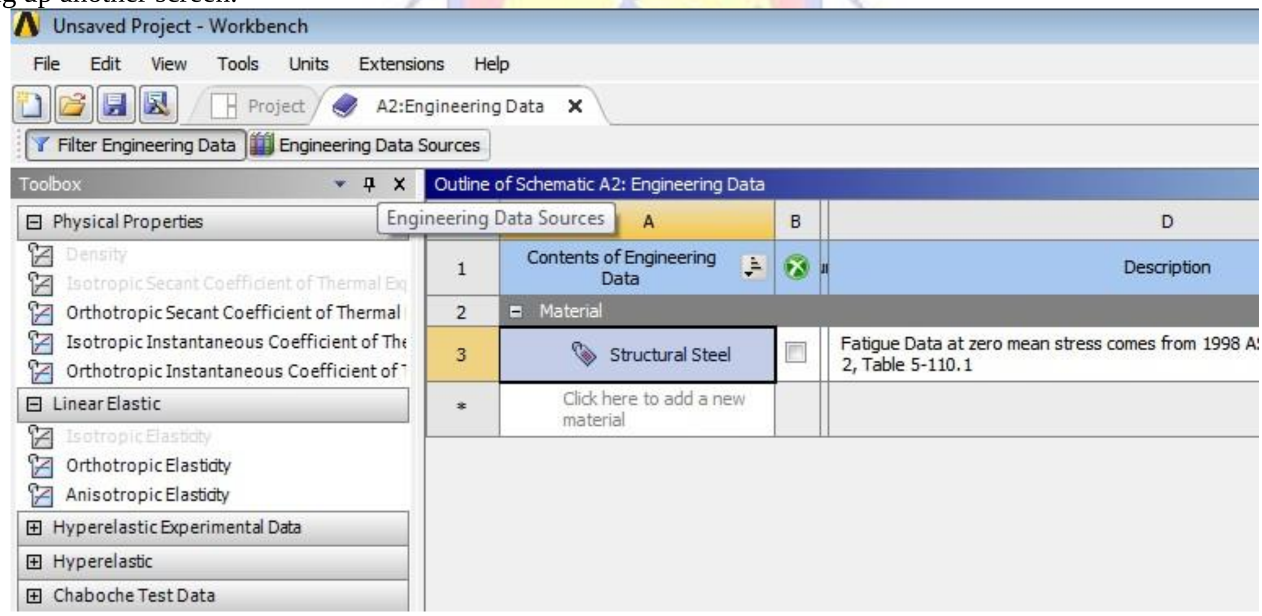
1.As you open ANSYS you can see the entire array of problems on the left had side this software can help you solve. The problem at hand is a *Thermal* problem. Double click **Steady State Thermal (ANSYS)** to open the task manager for your problem set in the Project Schematic area.



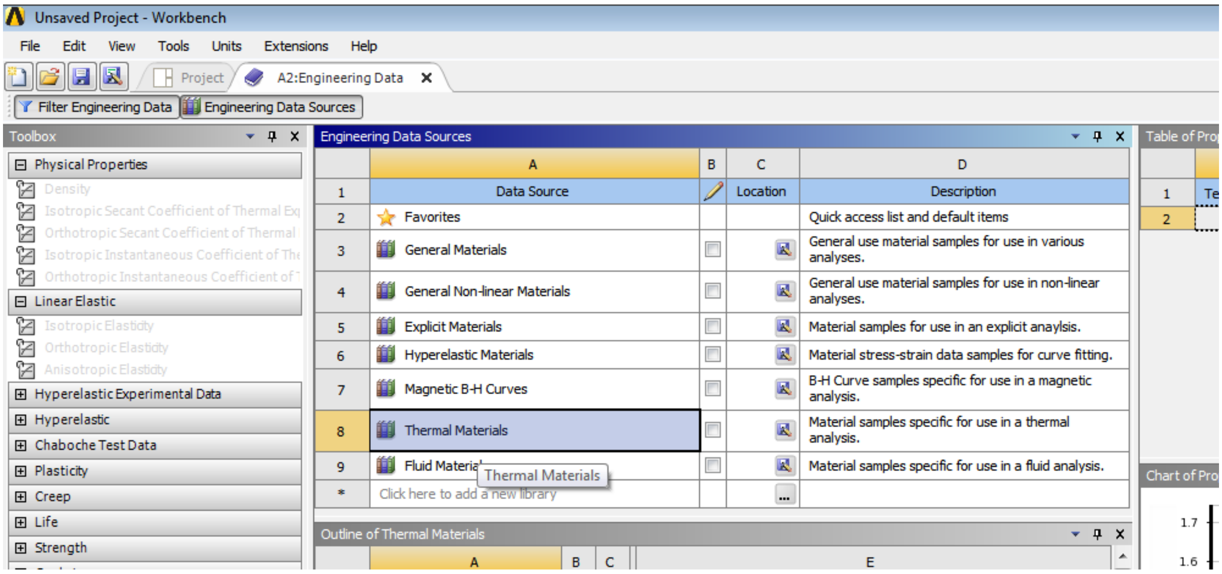
2.ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see **Steady State Thermal (ANSYS),** double click this to change the name. For this problem choose "*Car Brake Disc*."

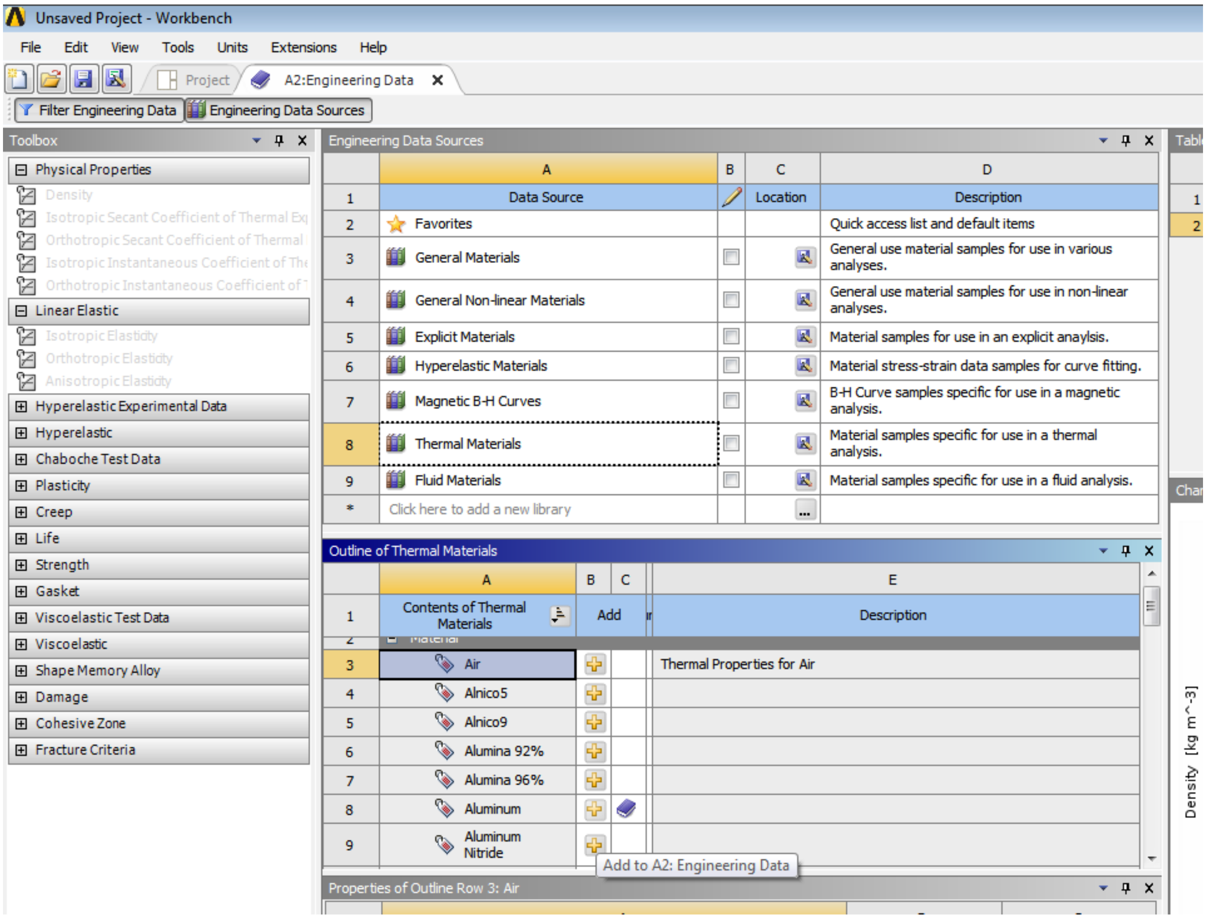
Engineering Data

To begin setup for your cantilever beam, double click or right click on ***Engineering Data*** and click ***edit***. This will bring up another screen.



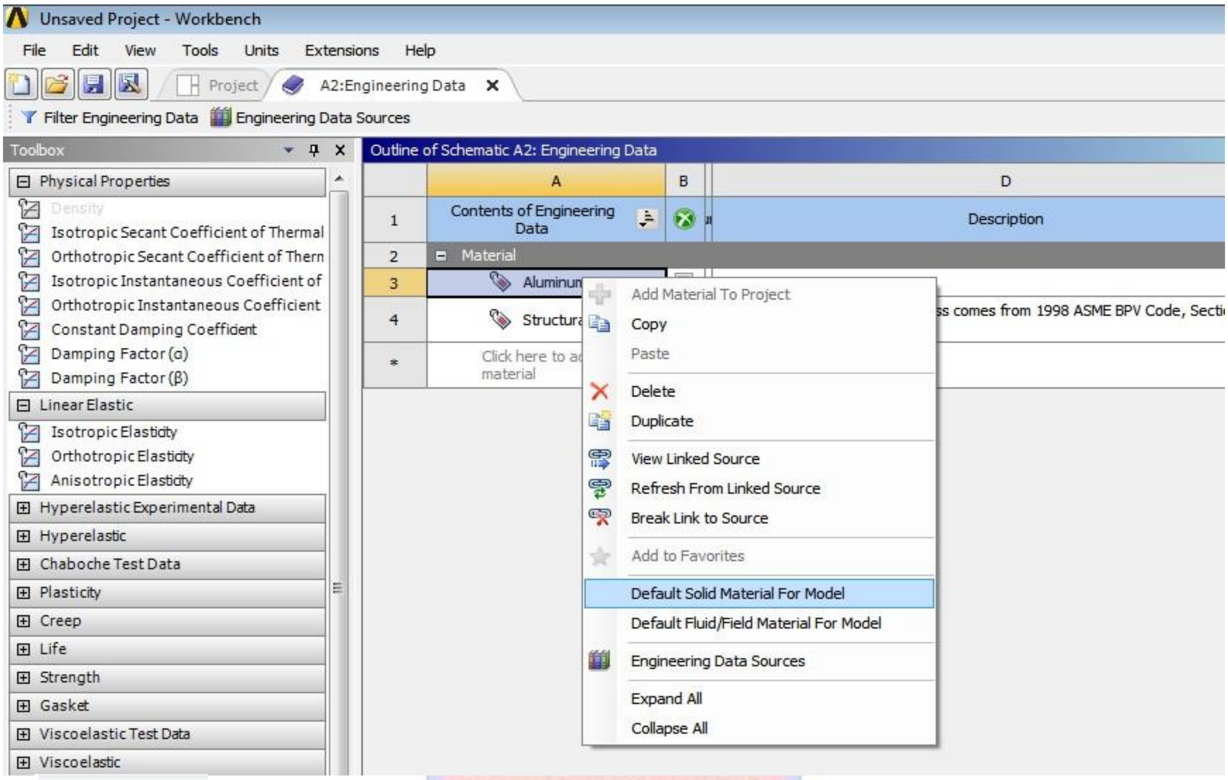
This new window will allow you to alter the material properties of your cantilever beam. Under **Outline of Schematic A2: Engineering Data**, it shows ***click here to Thermal material***, this menu allows you to input the material of your cantilever beam, add the material.

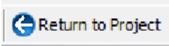




Click on + Sign and click on Filter engineering data

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



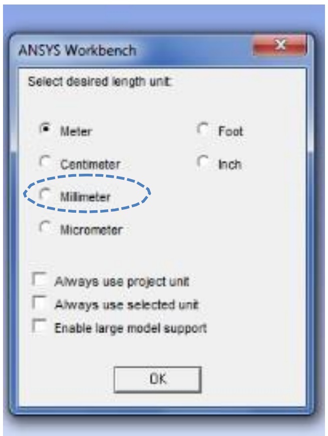
click on 

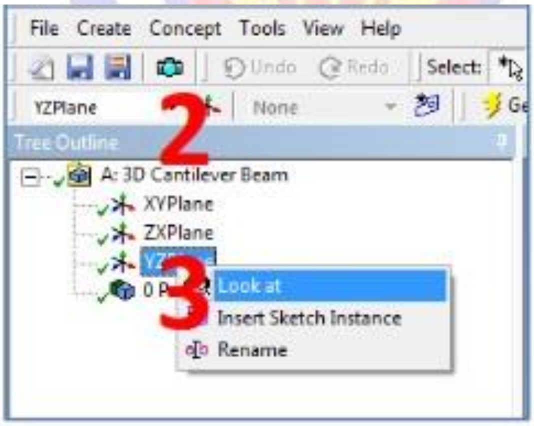
seen on the upper tab.

Geometry

Base Geometry

1.Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.





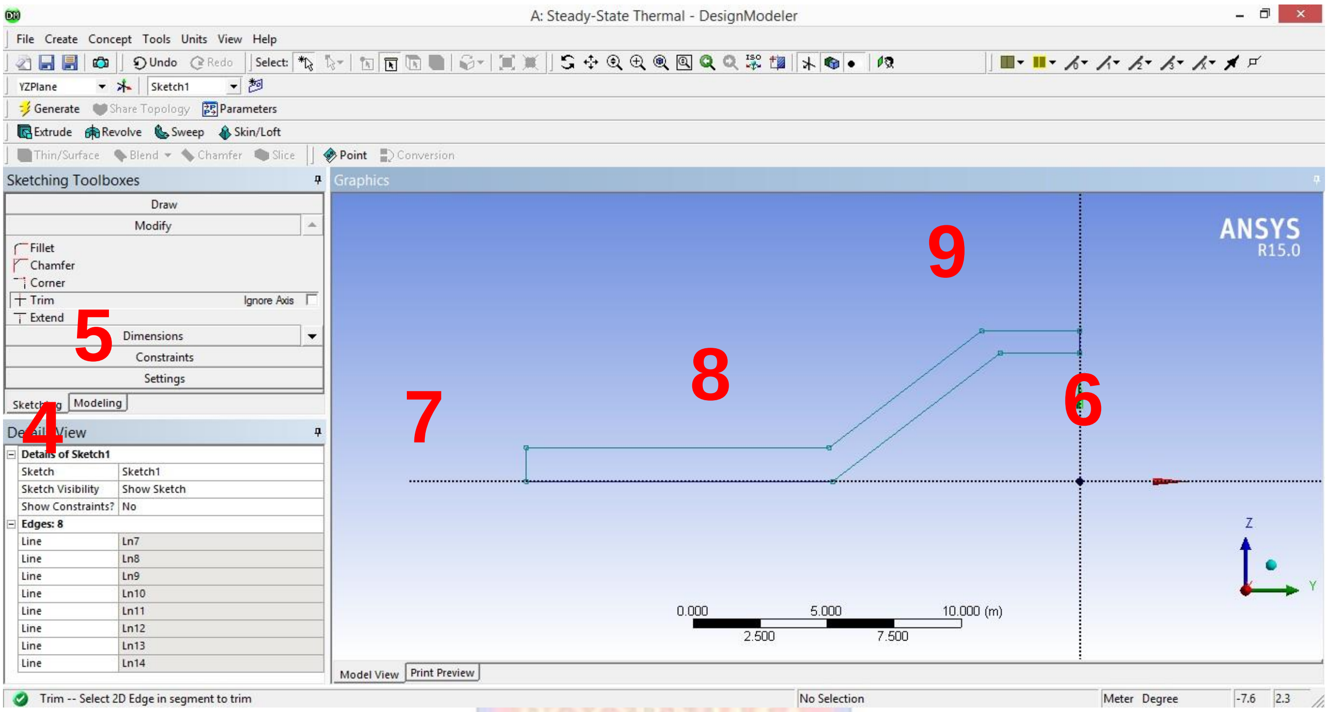
Note: Select meters and hit ok

2.In the new window, click the **Display Plane** icon to toggle the coordinate system.

3.Go to **Design Modeler > Tree Outline** >right click on YZ Plane. Click **Look At** to view the YZ plane.

4.Go to **Design Modeler -> Tree Outline -> Sketching**

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



5. Click on **Polyline.**

6. Bring your cursor into the workspace at point 0,0, over the origin until ‘P’ appears directly above the origin.

7. Click on the origin to place the lower left corner.

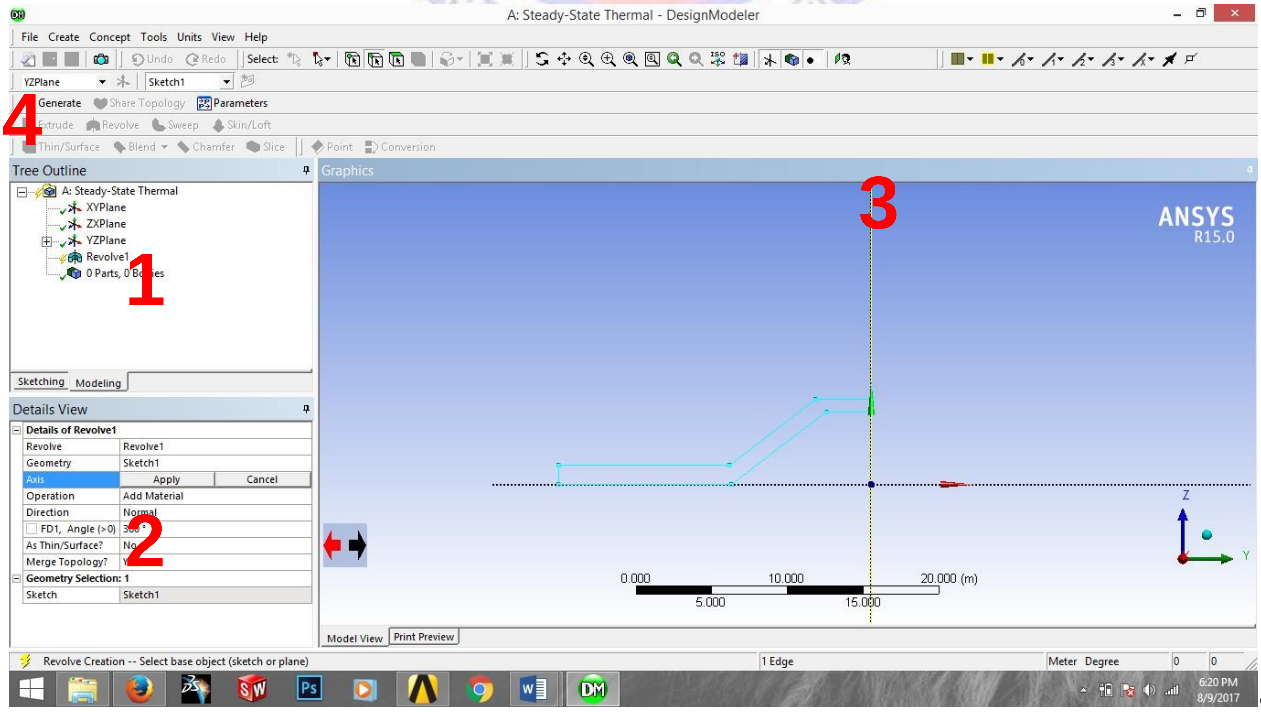
8. Click on a point in the first quadrant to define the top right corner of our rectangle. The point is arbitrary as we will be fixing dimensions momentarily.

9. Draw the geometry as shown.

Revolve Sketch

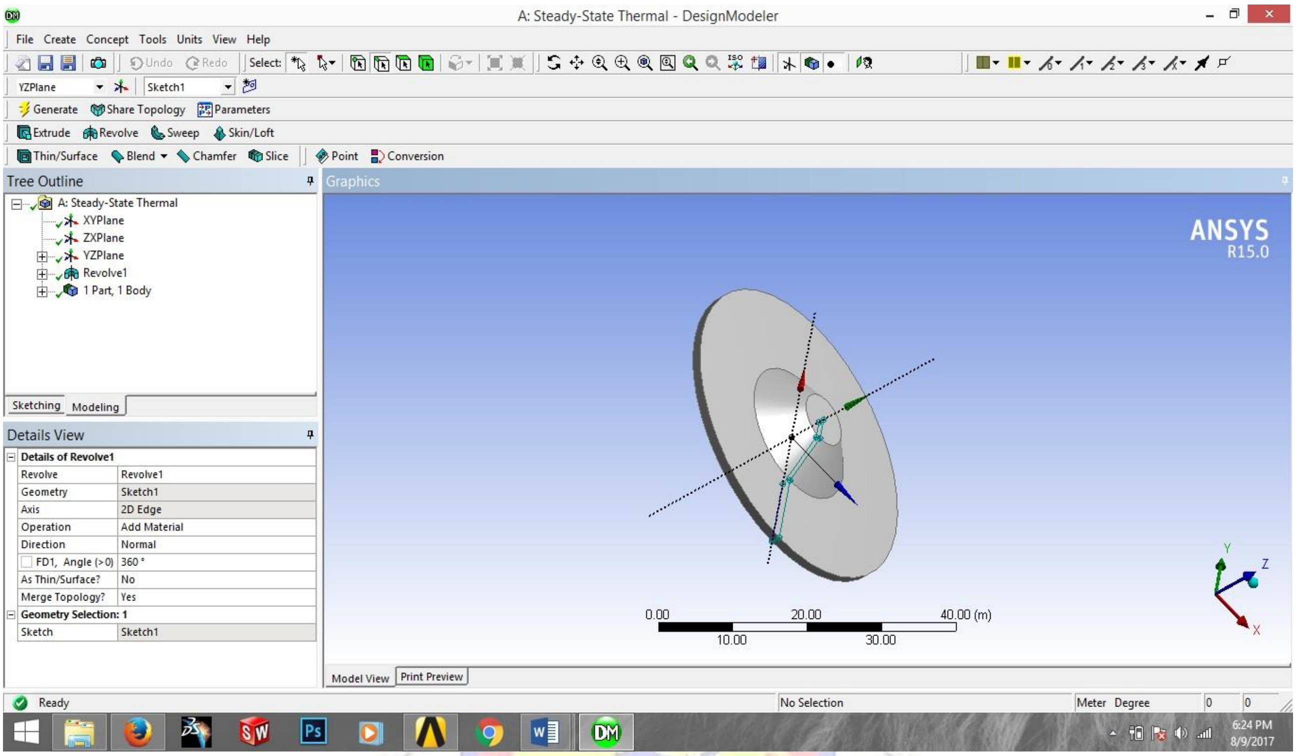
1. Go to **Main Toolbar ->** and select **Revolve** Revolve

2. Go to **Modeling -> FD1, Angle (>0) ->** enter in **360**



3. Choose Z axis as Axis and click Apply

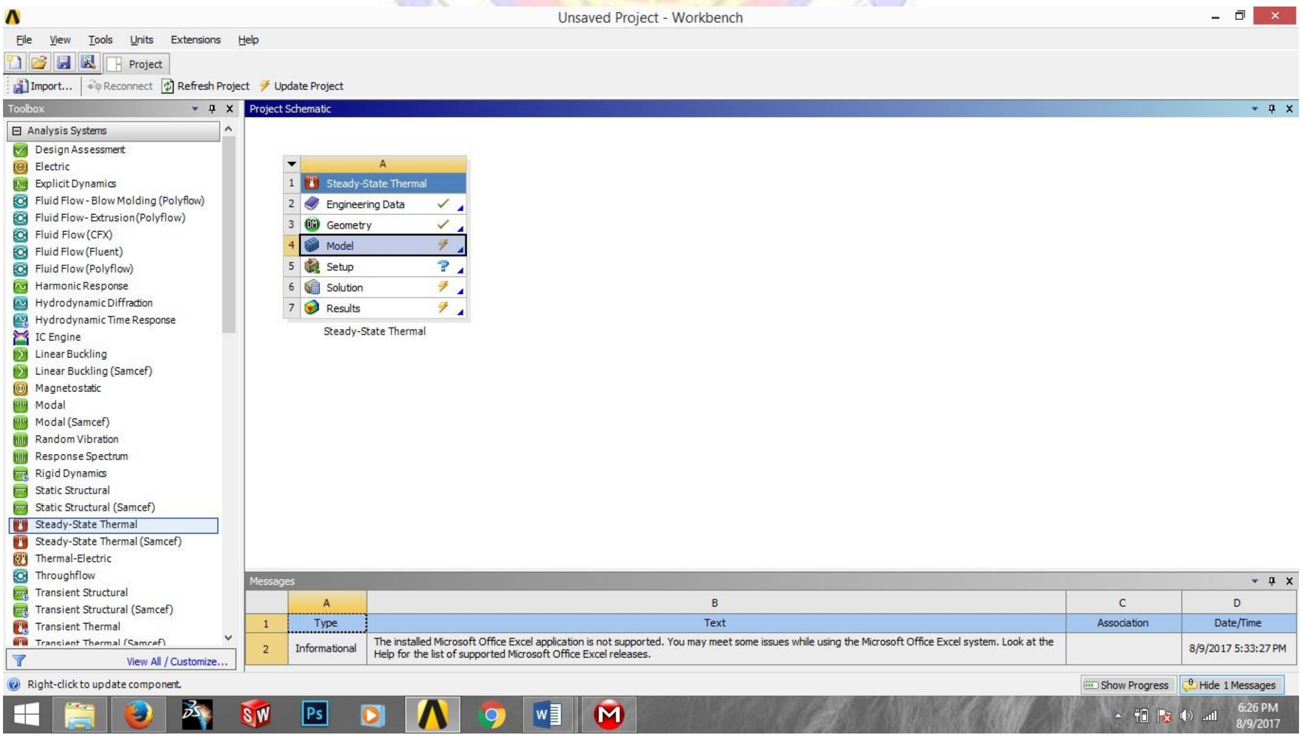
4. Go to **Design Modeler ->** **Generate**.
5. To verify our geometry, look at the isometric view. Click the *blue dot* in the **triad** in the lower right corner of the screen to look at the isometric view.



Model

Open ANSYS Mechanical

1. out of Design Modeler. Don't worry, your work will be saved.



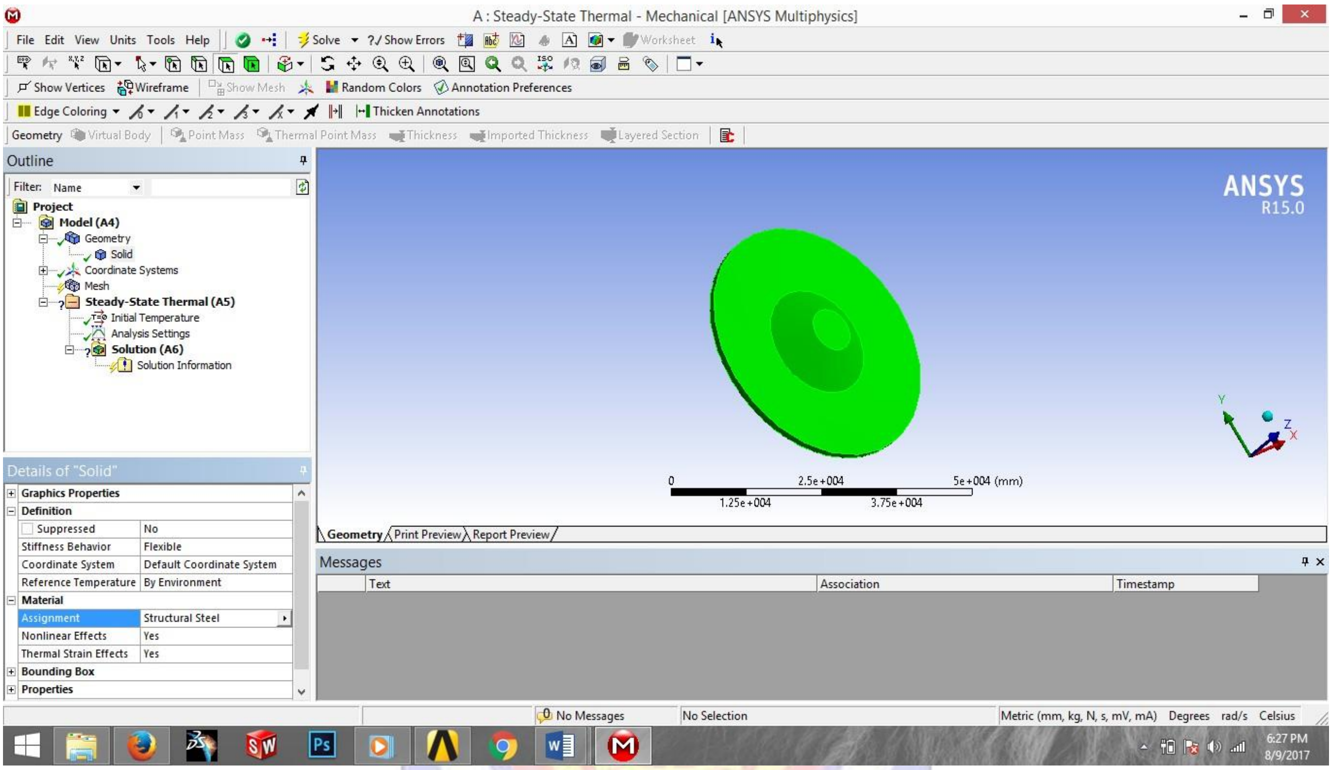
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

2.Go to **Workbench -> Project Schematic -> Model** This will open *ANSYS Mechanical*

Material Assignment

1.Go to **Mechanical -> Outline -> Project -> Model -> Geometry -> Solid Body**

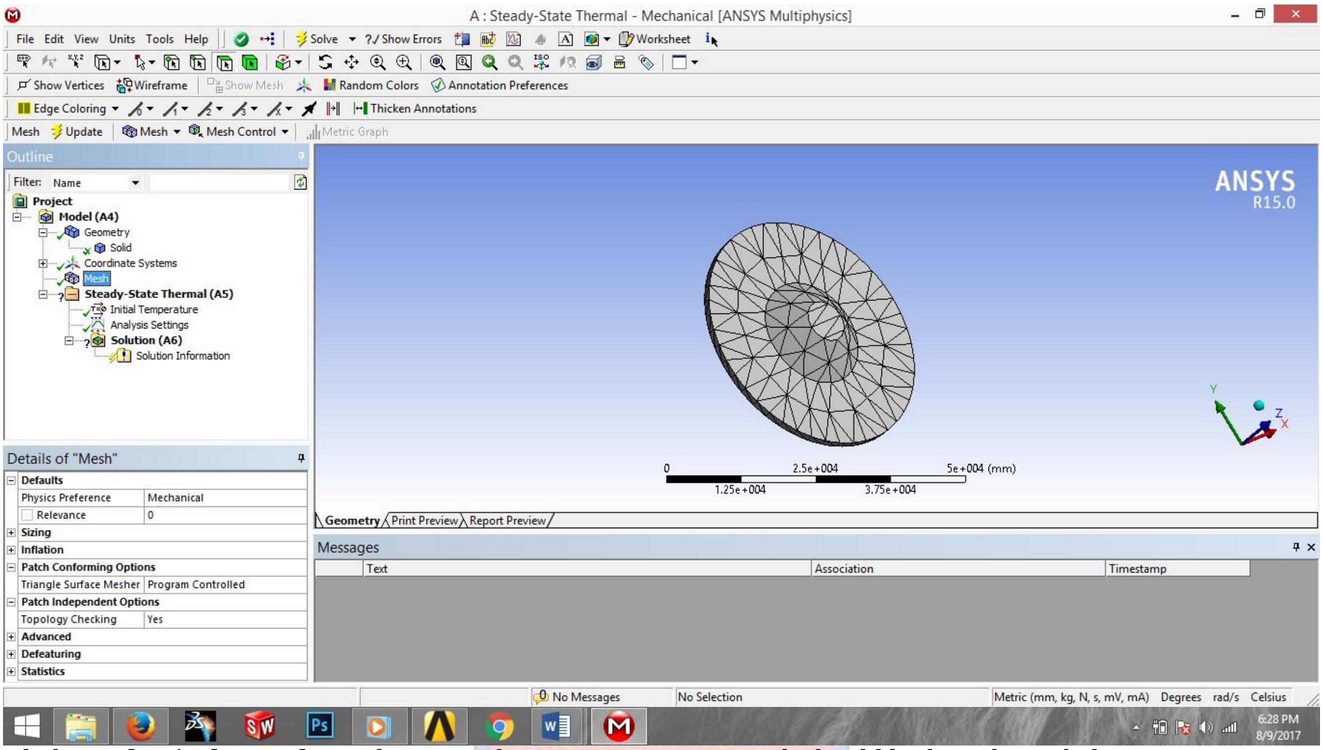
2.Under **Mechanical -> Details of "Solid Body" -> Material -> Assignment**, change *Structural Steel* to *Aluminum*.



Mesh

1.Go to **Mechanical -> Outline -> Project -> Model -> Mesh**

2.Go to **Mechanical -> Details of 'Mesh' -> Sizing -> Element Size** and change the value from *Default* to .5 m. This will give us 2 elements through the thickness of the beam.



3.Click **Mechanical ->Update**. This may take some time. Your mesh should look as shown below:

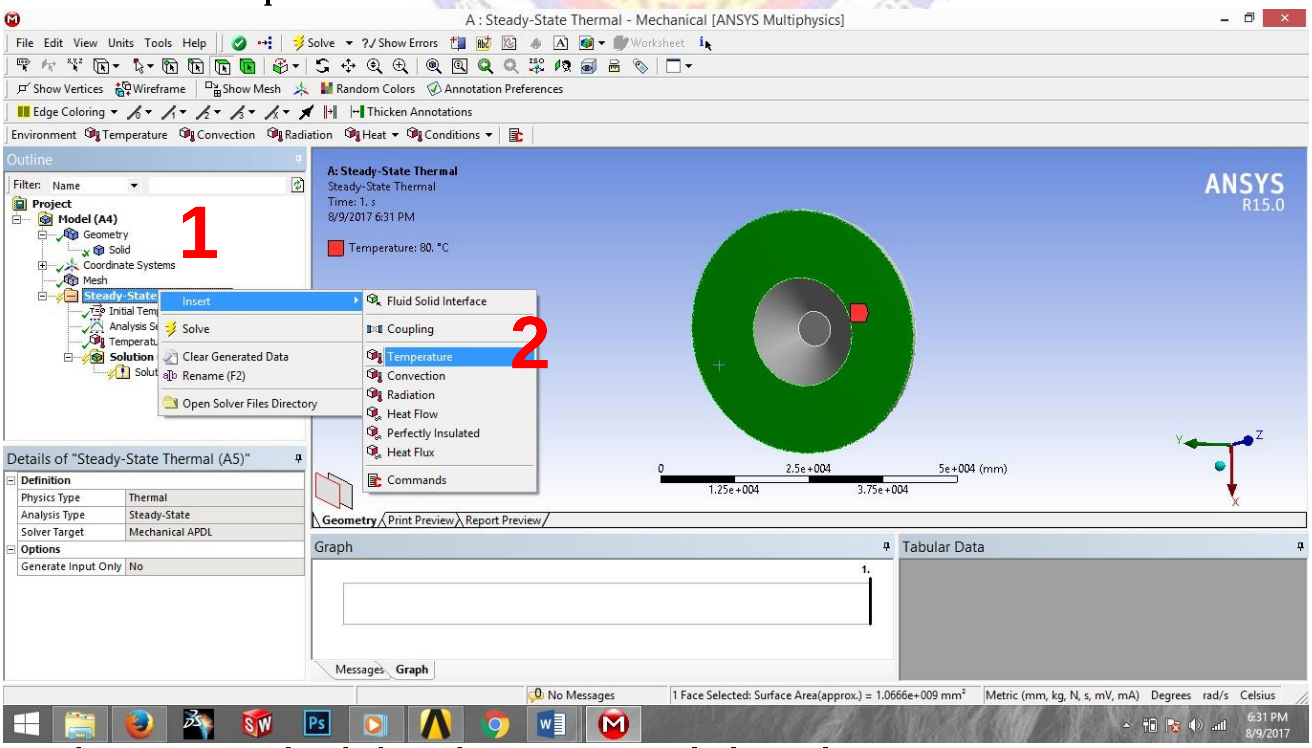
<u>Setup</u>

You can perform the rest of your analysis for this problem in the *ANSYS Mechanical* window. The other options in the *Workbench* window will link you back to the same screen (i.e. Setup, Solution, Results)

Fixed Support

1.Go to **Mechanical -> Outline ->** right click **Steady State Thermal**

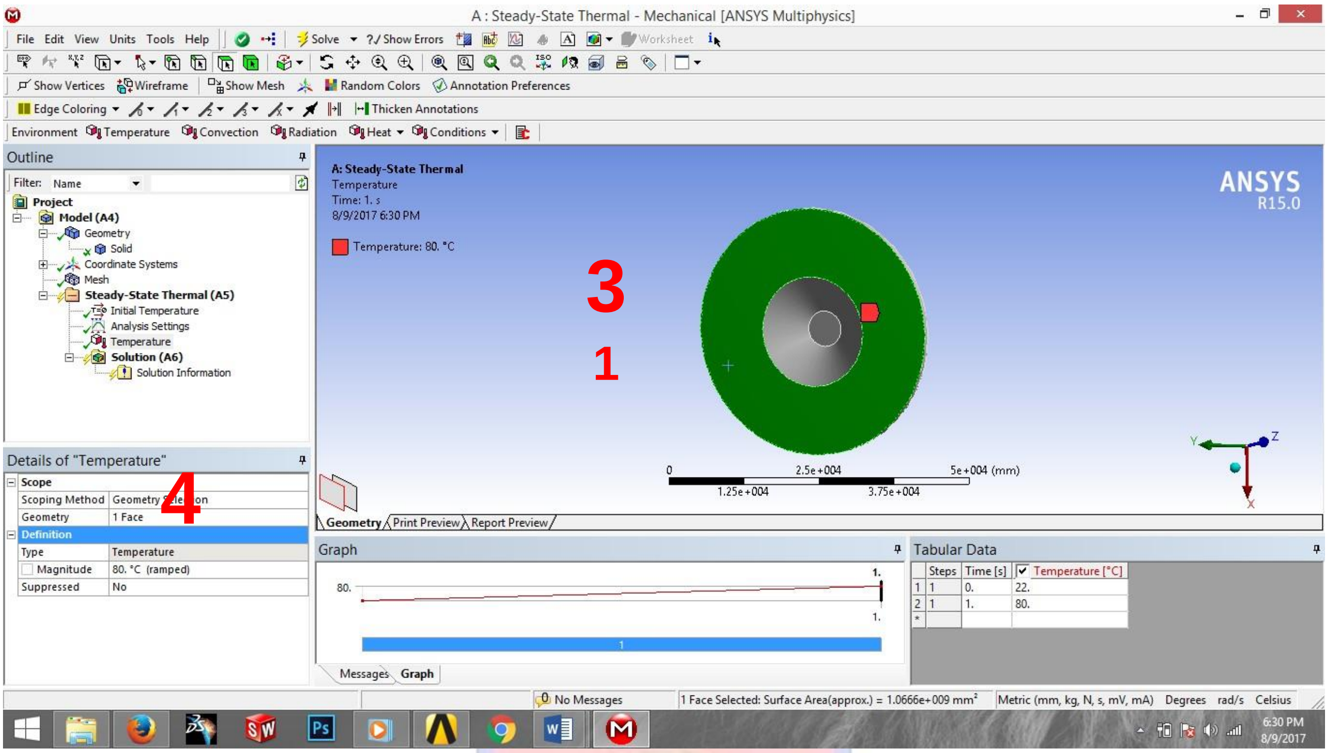
2.Go to **Insert ->Temperature**



3. Under temperature select the lower face in geometry and select apply.

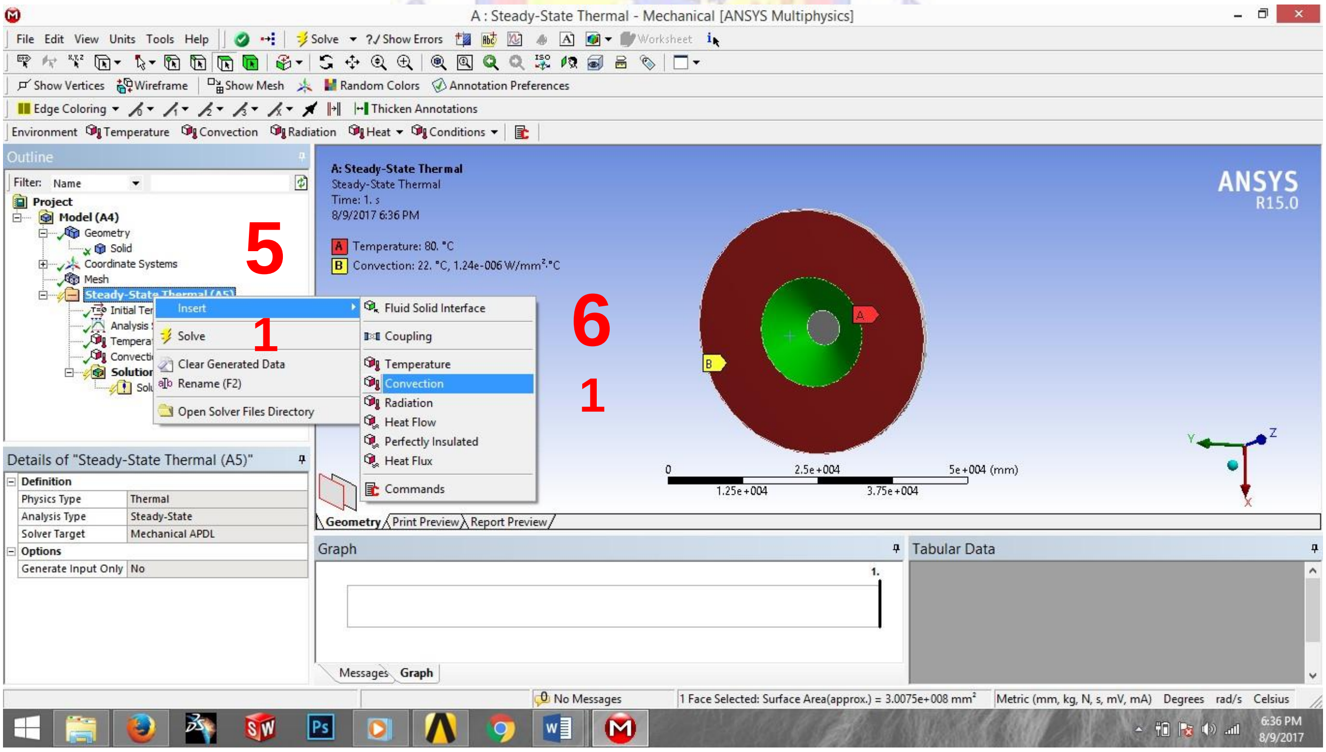
4.Go to **Temperature ->80**

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

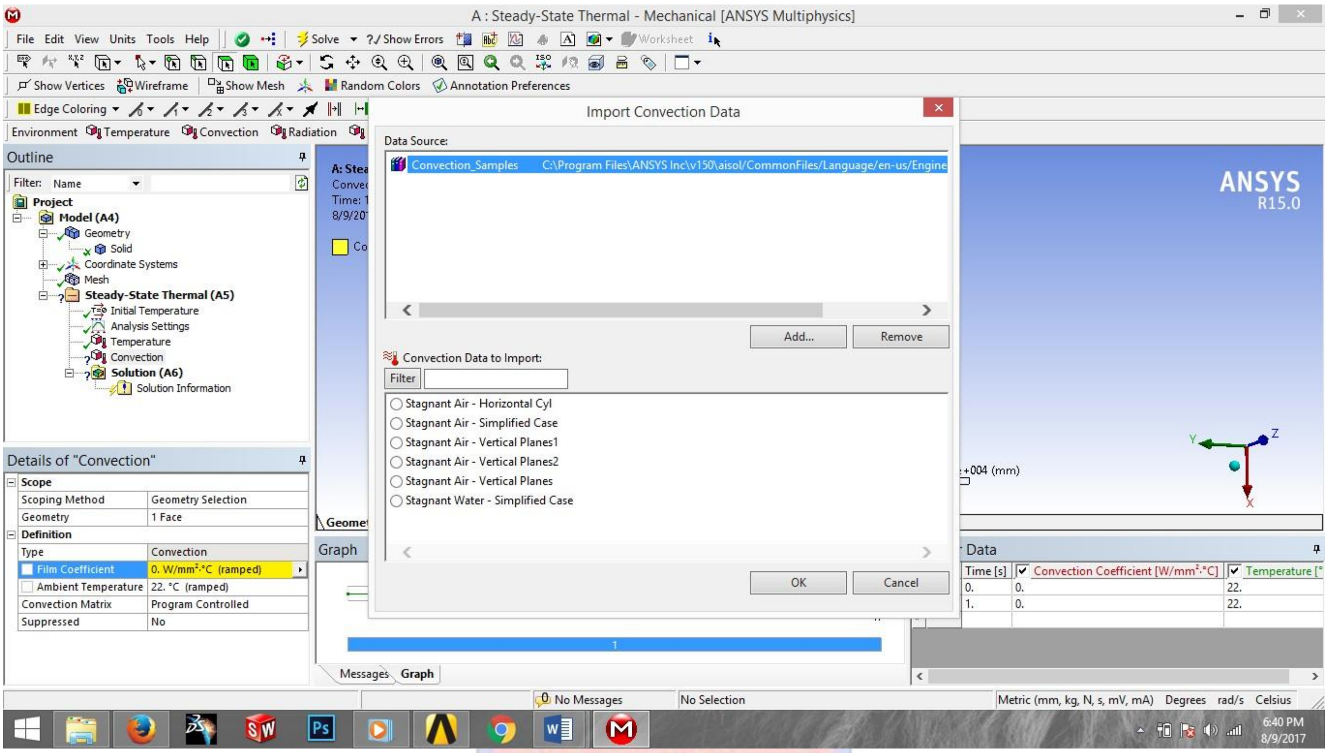


5. Go to **Mechanical -> Outline ->** right click **Steady State Thermal**

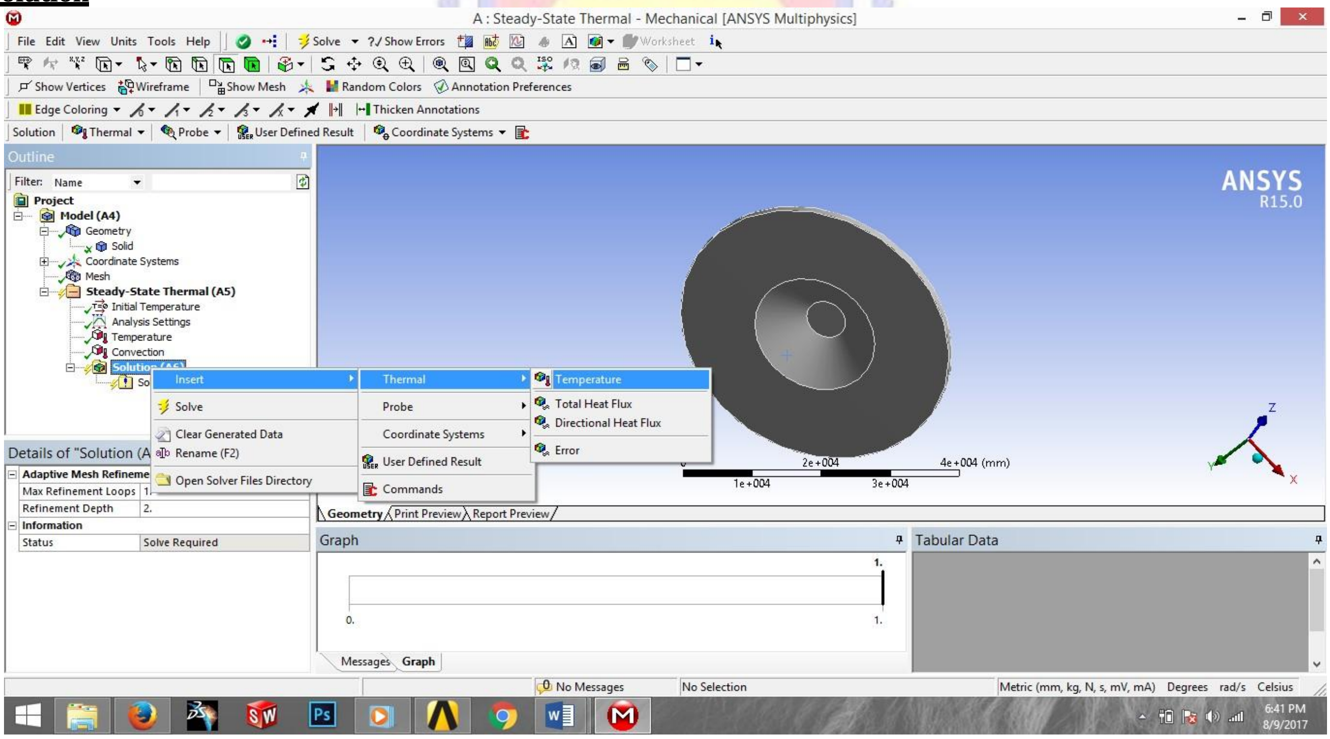
6. Go to **Insert -> Convection**



7. Under temperature select the middle face in geometry and select apply.
8. Go to **Film Coefficient (click on arrow mark)** and select **Import Convection Data,** Select the **Stagnant Air-Horizontal Cyl** Radio Button.



Solution



Solution->Insert->Thermal->Temprature

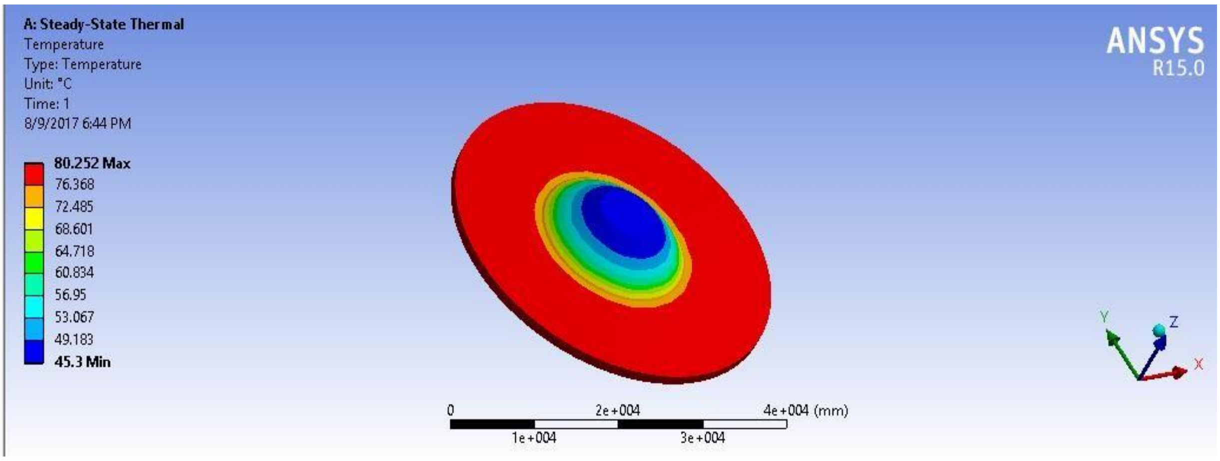
1. **Solution->Insert->Thermal->Thermal Flux**

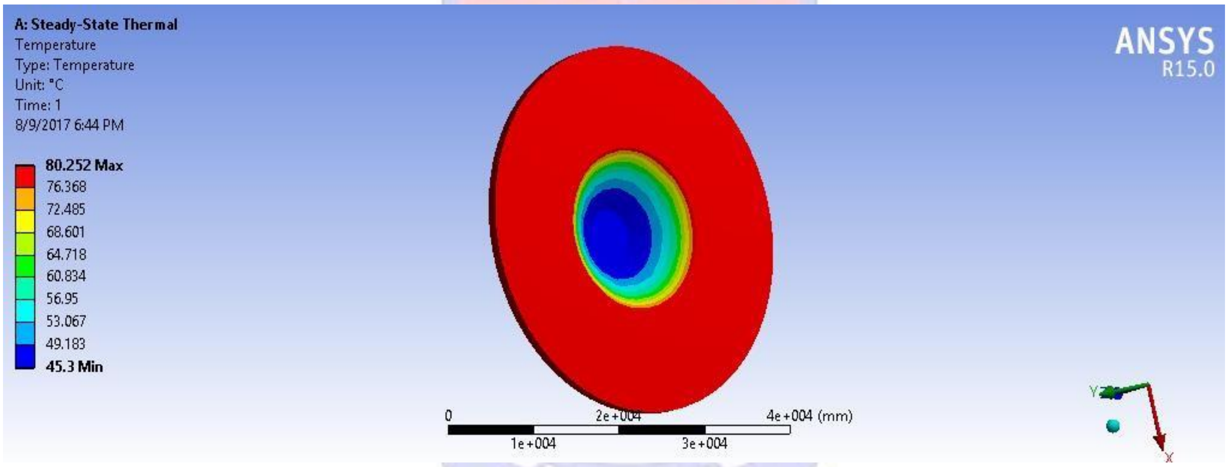
Now that our solvers have been defined, go to **Mechanical ->** ⚡ **Solve**. The calculations in Workbench may take up to a minute to solve.

Go to **Mechanical -> Outline -> Project -> Model (A4) ->Solution (A6) -> Maximum**

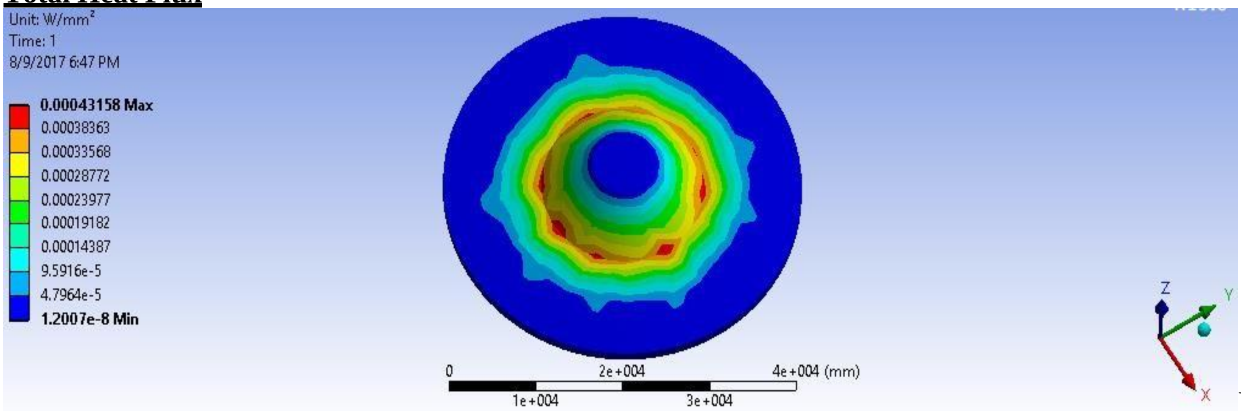
Temperature

Your Temperature plot should look as shown below:





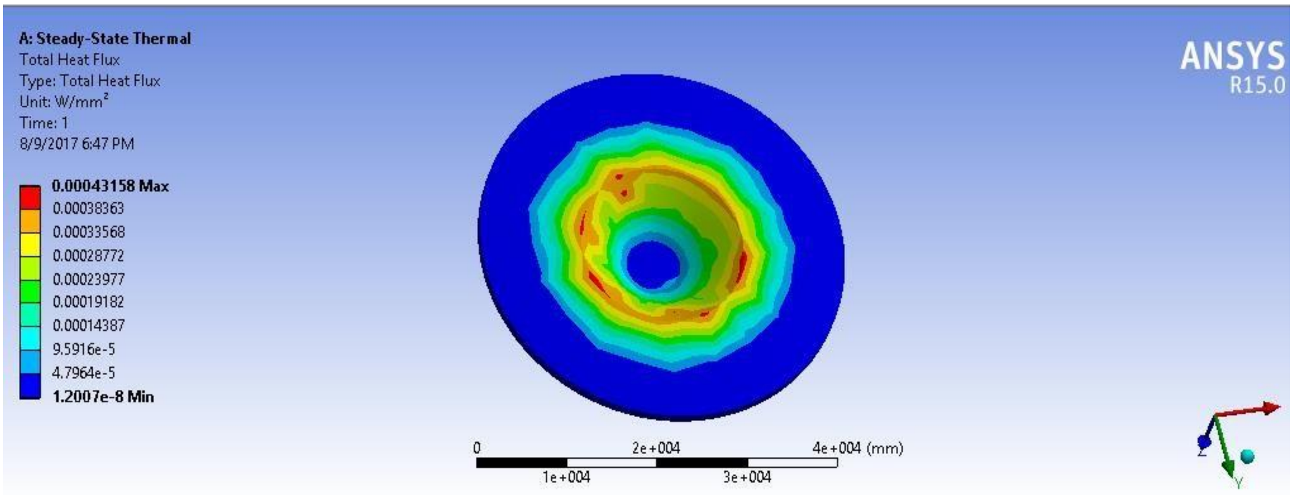
Total Heat Flux



Your

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Temperature plot should look as shown below:



Viva Questions:-

1. How u discretize a model?
2. What is coefficient of thermal expansion?
3. Mention different types of elements
4. Define steady state analysis.
5. Is stress depends on material property?
6. What are the different types of analysis?
7. Define mesh plotting
8. What are the types of boundary conditions?
9. What is P-method and H-method?
10. What do you mean by continuum?

Experiment No.4. Cantilever Beam Modal Analysis

Objective:- To find the Natural frequency & mode shapes of a cantilever beam

- Modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration. Modal analysis is used to determine a structure's vibration characteristics — natural frequencies and mode shapes.
- It is the most fundamental of all dynamic analysis types and is generally the starting point for other, more detailed dynamic analyses.
- **Modal analysis**, or the mode-superposition method, is a linear dynamic-response procedure which evaluates and superimposes free-vibration mode shapes to characterize displacement patterns. Mode shapes describe the configurations into which a structure will naturally displace. Typically, lateral displacement patterns are of primary concern. Mode shapes of low-order mathematical expression tend to provide the greatest contribution to structural response. As orders increase, mode shapes contribute less, and are predicted less reliably. It is reasonable to truncate analysis when the number of mode shapes is sufficient.
- A structure with N degrees of freedom will have N corresponding mode shapes. Each mode shape is an independent and normalized displacement pattern which may be amplified and superimposed to create a resultant displacement pattern.

Problem Specification:-

Consider an aluminum beam that is clamped at one end, with the following dimensions.

Length	4 m
Width	0.346 m
Height	0.346 m

Roll No	Length
1-6-11-16-21-26-31-36-41-46-56-61-66-71	4
2-7-12-17-22-27-32-37-42-47-57-62-67-72	4.5
3-8-13-18-23-28-33-38-43-48-58-63-68-73	5
4-9-14-19-24-29-34-39-44-49-59-64-69-74	5.5
5-10-15-20-25-30-35-40-45-50-60-65-70-75	6

The aluminum used for the beam has the following material properties.

Density	2,700 kg/m^3
Young's Modulus	70x10^9 Pa
Poisson Ratio	0.35

Using ANSYS Workbench find the first six natural frequencies of the beam and the mode shapes.

Results Required:- 1. Frequency tabular data 2. Mode shapes (6)

Pre-Analysis & Start-Up:-

Pre-Analysis

The following equations give the frequencies of the modes and the mode shapes and are derived from Euler-Bernoulli Beam Theory.

$$w_n = \alpha_n^2 \sqrt{\frac{EI}{ml^3}}$$

$$n = 1, 2, 3, \ldots$$

$$\alpha_n = 1.875, 4.694, 7.855, \ldots$$

$$m = \rho V = \rho \cdot l \cdot h \cdot w$$

$$I = \frac{w \cdot h^3}{12}$$

$$w_1 = 1.875^2 \sqrt{\frac{70\ E9\ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{2.7\ E3\ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 111.7\ \frac{rad}{s} = 17.8\ Hz$$

$$w_2 = 4.694^2 \sqrt{\frac{70\ E9\ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{2.7\ E3\ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 700.4\ \frac{rad}{s} = 111.5\ Hz$$

$$w_3 = 7.855^2 \sqrt{\frac{70\ E9\ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{2.7\ E3\ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 1961.2\ \frac{rad}{s} = 312.1\ Hz$$

$$y_i(x) = cosh(\frac{\alpha_i x}{L}) - cos(\frac{\alpha_i x}{L}) - \sigma_i(sinh(\frac{\alpha_i x}{L}) - sin(\frac{\alpha_i x}{L}))$$

$$\alpha_i = 1.875, 4.694, 7.855, \ldots$$

$$\sigma_i = 0.73409,\ 1.018647,\ 0.9992245,\ \ldots$$

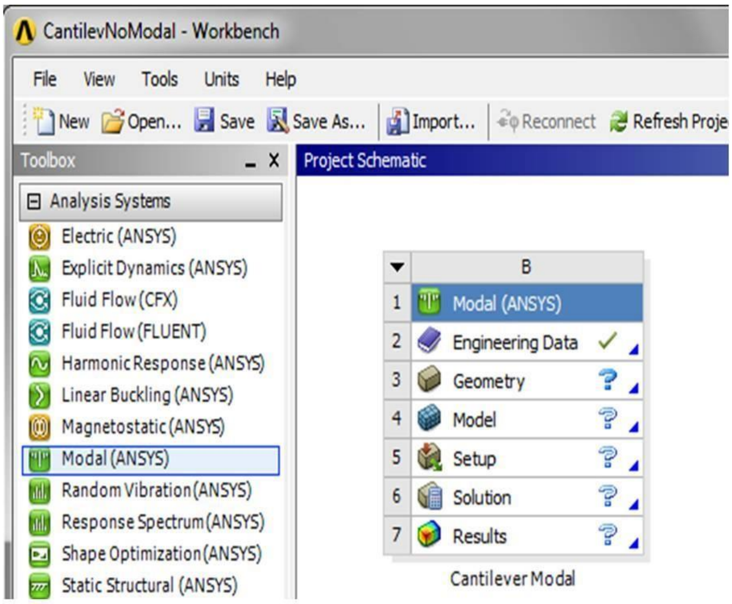
Start ANSYS Workbench & Load Files

In this section we will launch ANSYS Workbench and then load the project file, "cantilever.wbpj" that was created in the "Cantilever Beam" tutorial.

Start > All Programs > ANSYS 16.0 > Workbench

Rename Modal (ANSYS)

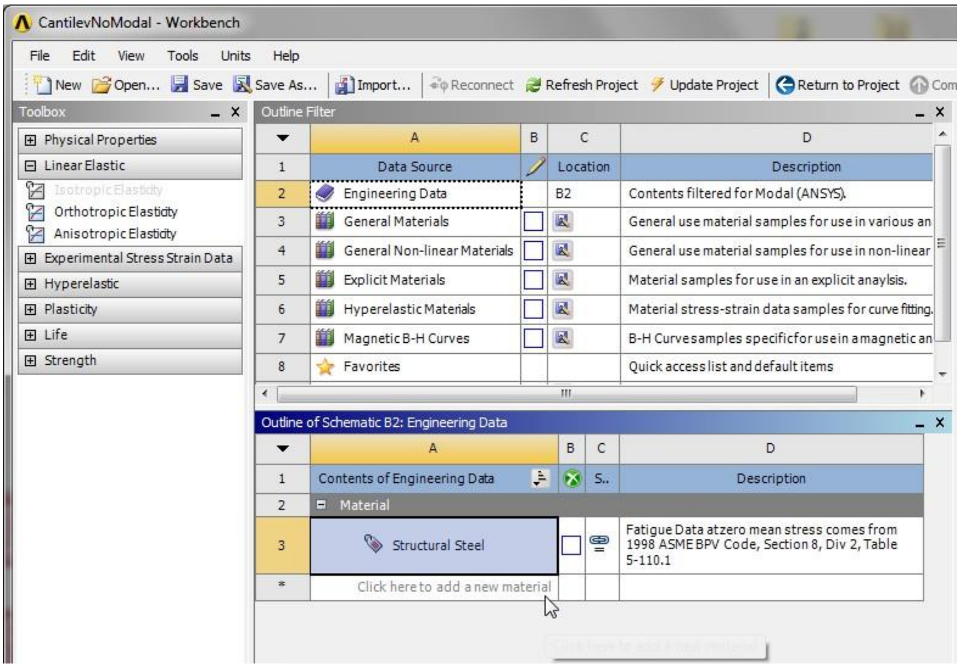
Double click on Modal (ANSYS) and rename it to "Cantilever Modal".



In this section we will input the properties of aluminum (as defined in the the Problem Specification) in to ANSYS. First, double click ***Engineering Data***,

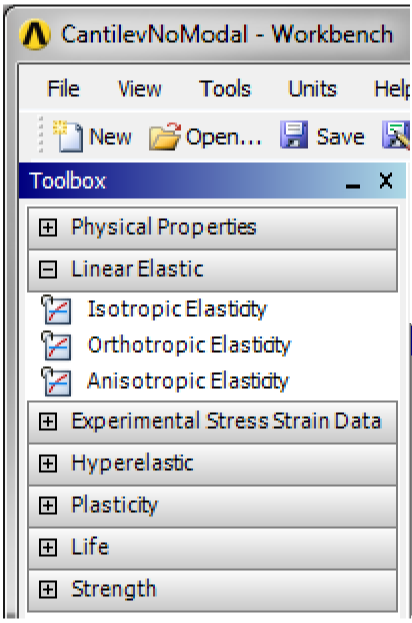
, in the "Cantilever Modal" Project. Next, click where it says "Click here to add a new material" as shown in the image below.

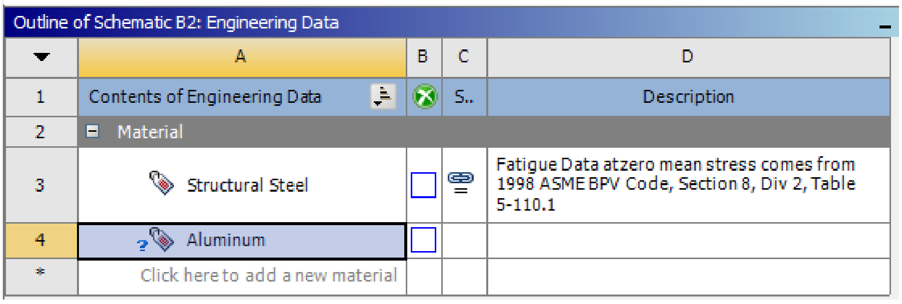
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



Next, enter "Aluminum" and press **enter**. You should now have Aluminum listed as one of the materials in table called "Outline of Schematic B2: Engineering Data", as shown below.

Then, ***(expand) Linear Elastic***, as shown below.

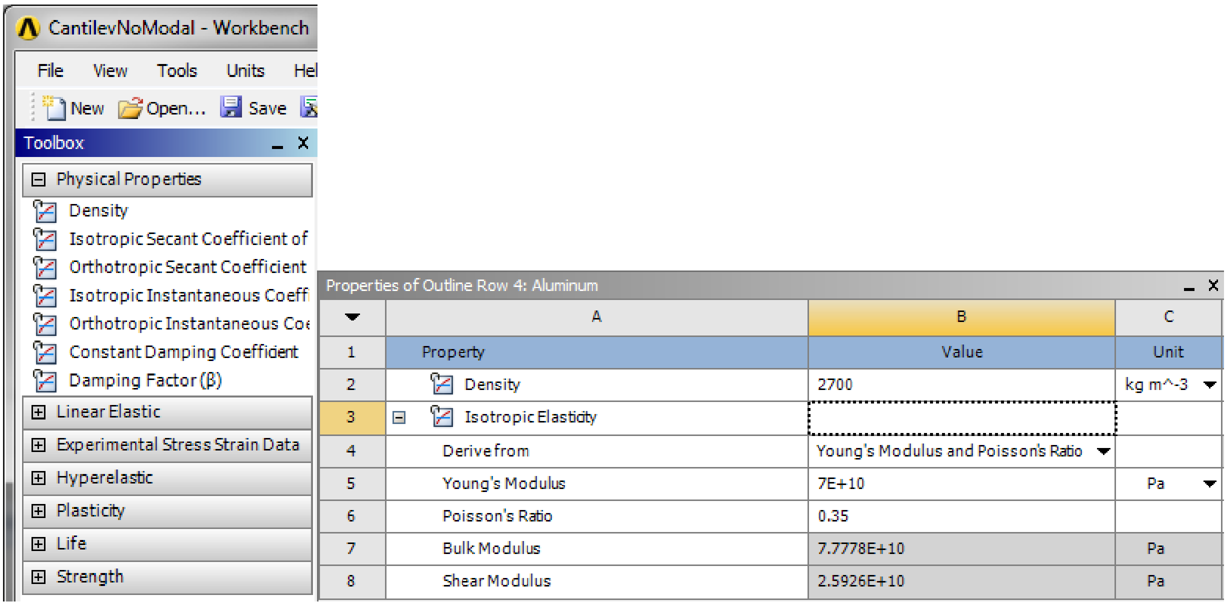




Now, ***(Double Click) Isotropic Elasticity***. Then set ***Young's Modulus*** to 70e9 Pa and set ***Poisson's Ratio*** to 0.35 , as shown below.

Next, ***(expand) Physical Properties***, as shown below.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



Now, ***(Double Click) Density***. Then, set ***Density*** to 2,700 kg / m^3 , as shown below.

Now, the material properties for Aluminum have been specified. Lastly, ***(Click) Return To Project***, Return to Project .

Save

Save your project now and periodically, as you work. ANSYS does not have an auto-save feature.

Go to Step 2: Geometry Create geometry as same as we have done in experiment no 3.

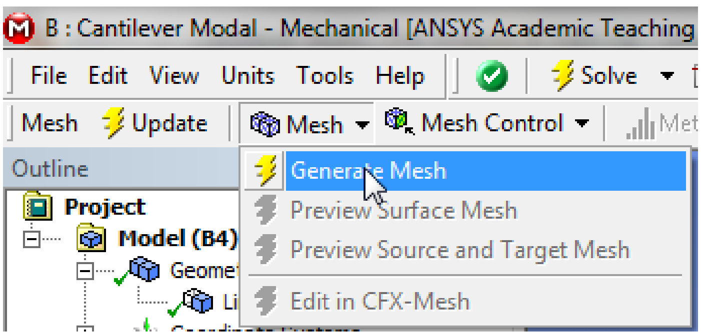
Mesh

Launch Mechanical

(double click) Model, Model , in the "Cantilever Modal" project.

Generate Default Mesh

First, ***(click) Mesh*** in the tree outline. Next, ***(click) Mesh > Generate Mesh*** as shown below.

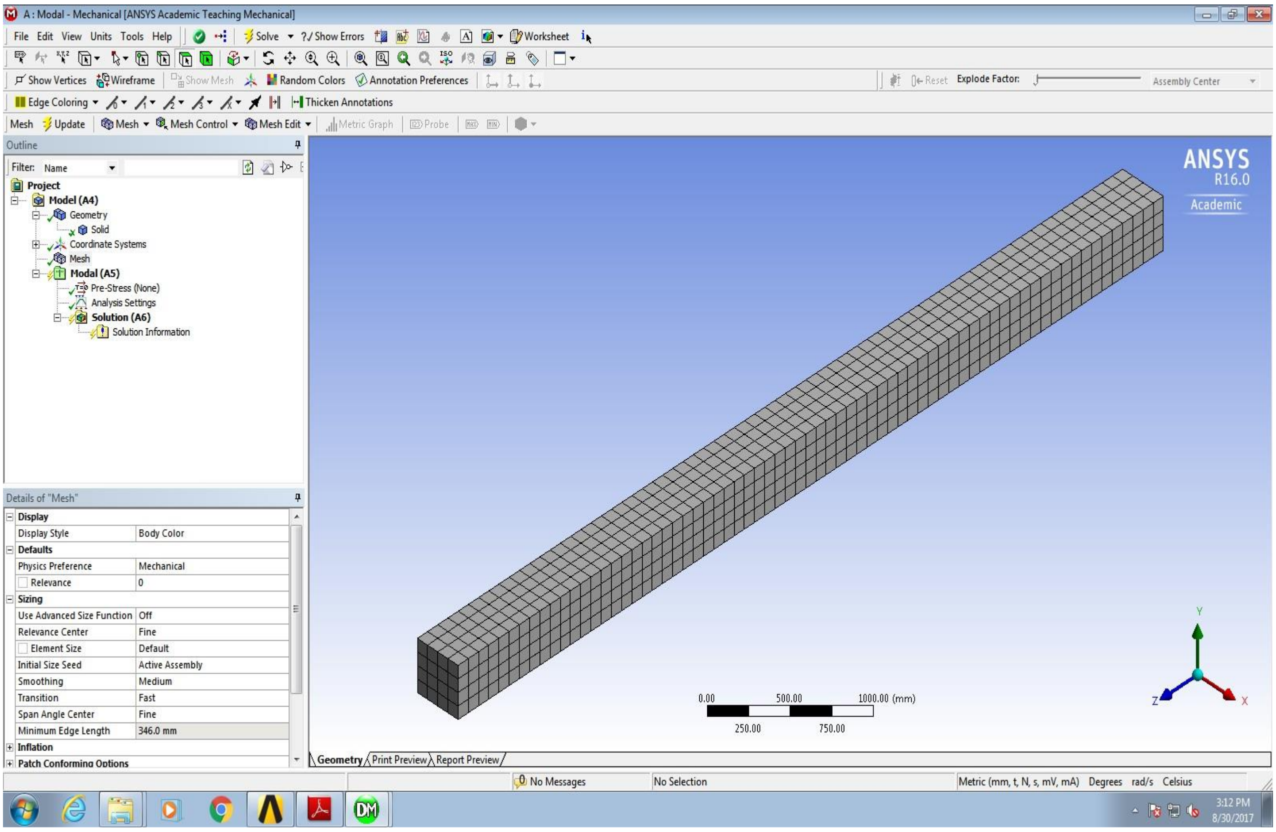


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Size Mesh

In this section we will size the mesh, such that it has ten uniform elements. In order to size the mesh, first expand ***Sizing*** located within the ***Relevance centre fine and Smoothing High***

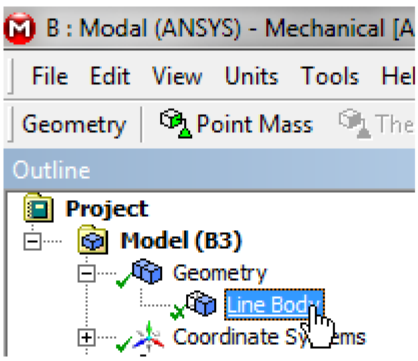
Now, ***(click) Mesh > Generate Mesh*** in order to generate the new mesh. You should obtain the mesh that is shown in the following image.



Physics Setup

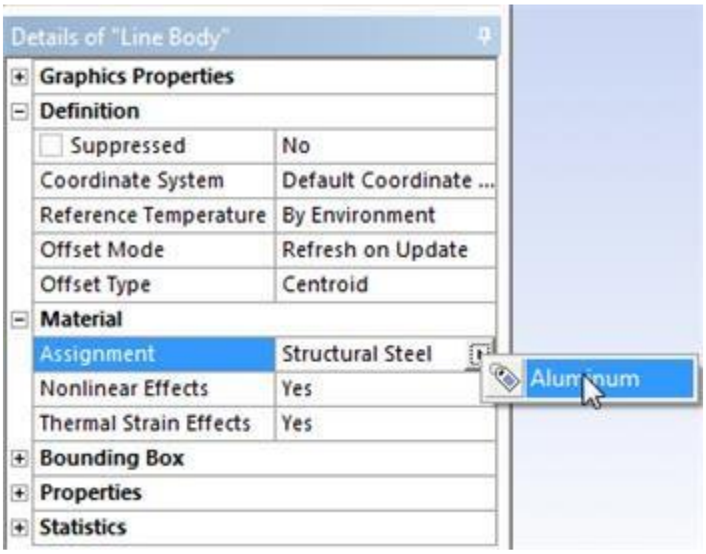
Material Assignment

At this point, we will tell ANSYS to assign the Aluminium material properties that we specified earlier to the geometry. First, ***(expand) Geometry*** then ***(click) Line Body***, as shown below.

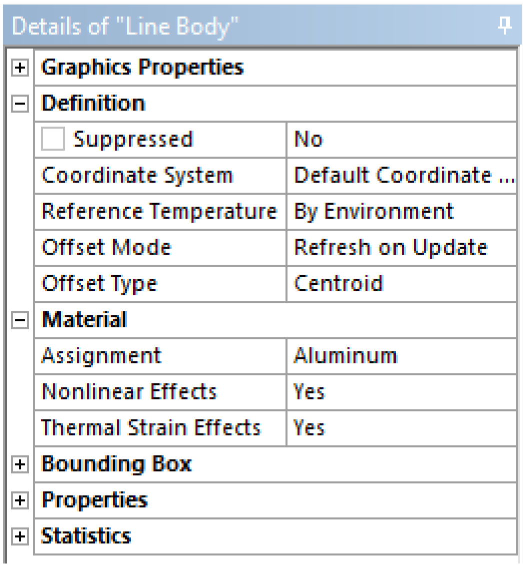


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Then, ***(expand) Material*** in the "Details of Line Body" table and set ***Assignment*** to Aluminium, as shown below.



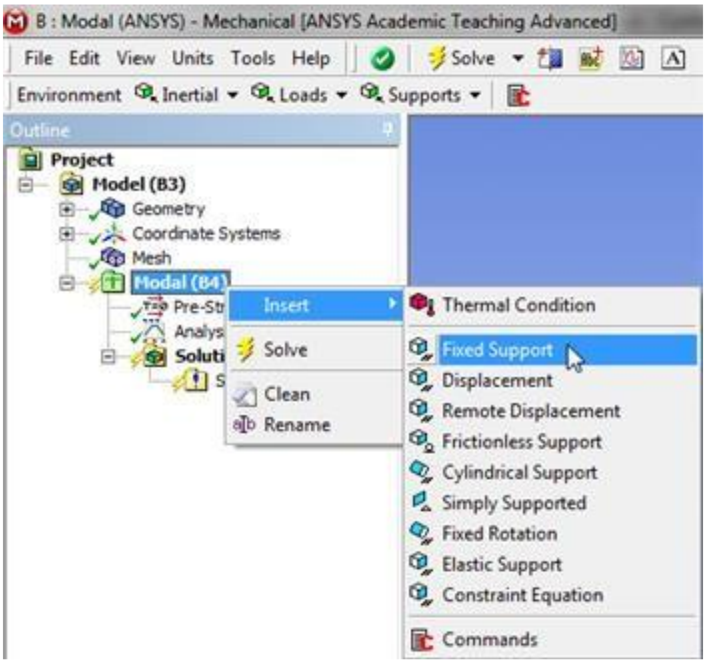
At this point your "Details of Line Body" table, should look comparable to the following image.



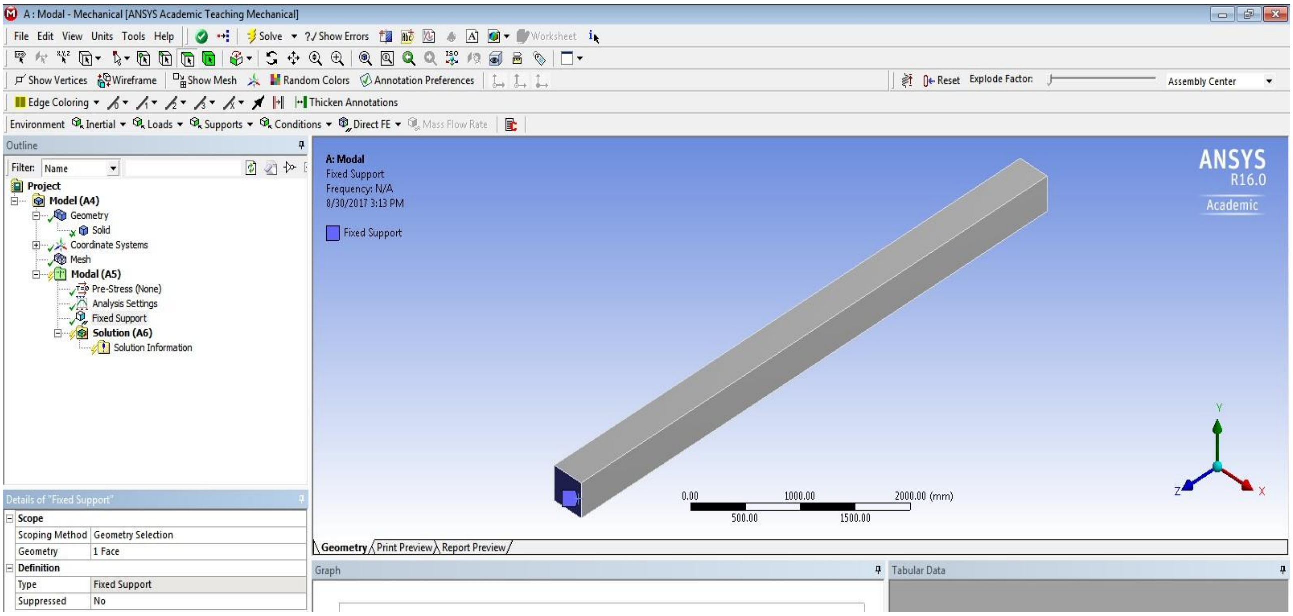
Fixed Support

First, ***(right click) Modal > Insert > Fixed Support***, as shown below.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



Next, click the ***vertex selection filter*** button, . Then, click on the left end of the beam and apply it as the ***Geometry*** in the "Details of Fixed Support" table.



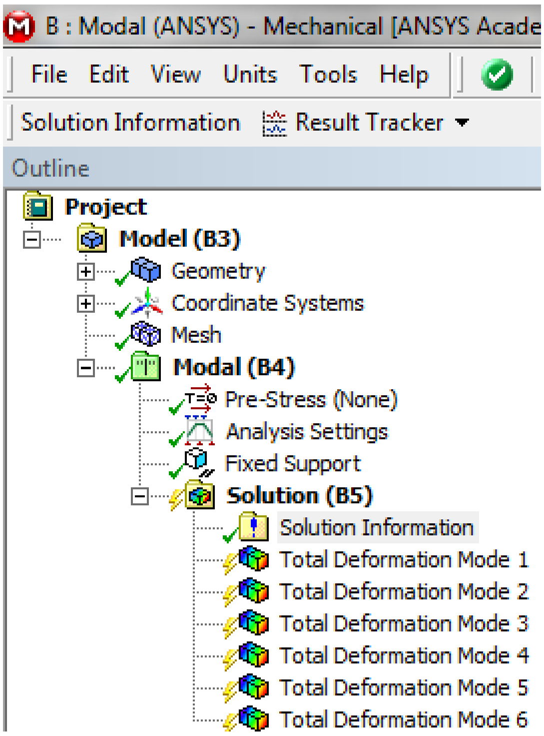
Numerical Solution

Specify Results (Deformation)

Here, we will tell ANSYS to find the deformation for the first six modes. Then, we will be able to see the shapes of the six modes. Additionally, we will be able to watch nice animations of the six modes.

Make sure that you set ***Mode*** to the respective mode number. At this point, your ***Outline*** should look the same as the following image.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



In order to run the simulation and calculate the specified outputs, click the ***Solve*** button, 

.

Select all natural frequency and right click to solve for results.

Tabular Data

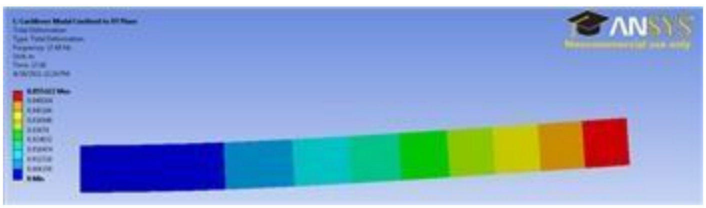
	Mode	Frequency [Hz]
1	1.	17.68
2	2.	107.04
3	3.	179.16
4	4.	285.26
5	5.	318.23
6	6.	525.41

Right click on Solution and evaluate all results

Numerical Results

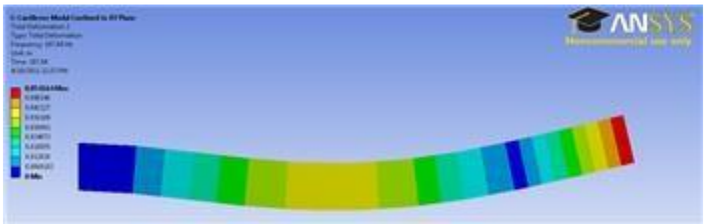
Mode Shapes

Mode 1

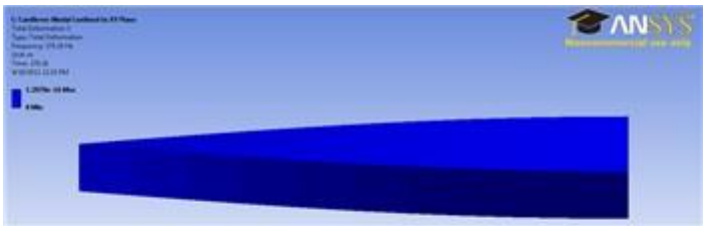


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

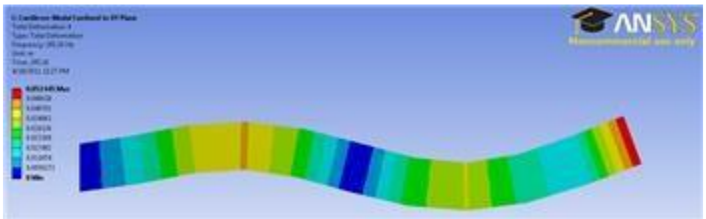
Mode 2



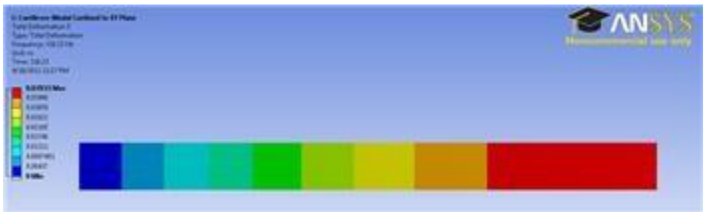
Mode 3



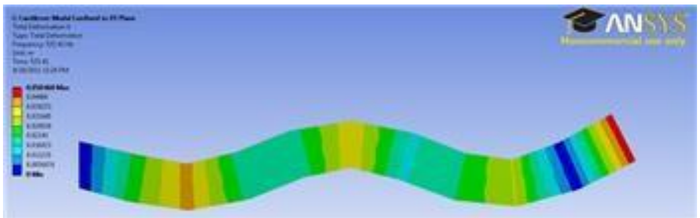
Mode 4



Mode 5



Mode 6



Verification & Validation

For our verification, we will focus on the first 3 modes. ANSYS uses a different type of beam element to compute the modes and frequencies, which provides more accurate results for relatively short, stubby beams such as the one examined in this tutorial. However, for these beams, the Euler-Bernoulli beam theory breaks down and is no longer valid for higher order modes.

Verification

Comparison with Euler-Bernoulli Theory

From our Pre-Analysis, based on Euler-Bernoulli beam theory, we calculated frequencies of 17.8, 111.5 and 312.1 Hz for the first three bending modes. The ANSYS frequencies for the first three bending modes are 17.7, 107.0 and 285.2 Hz. Note that in the ANSYS results, the third mode is NOT a bending mode. So the fourth mode reported by ANSYS is the third bending mode. These results give percent differences of 0.6%, 4.2% and 8.7% between ANSYS and theory. Thus the ANSYS results match quite well with Euler-Bernoulli beam theory. Note that the ANSYS beam element formulation used here is based on Timoshenko beam theory which includes shear-deformation effects (this is neglected in the Euler-Bernoulli beam theory).

Viva Questions:-

1. What is modal analysis?
2. What is Natural frequency?
3. How many natural frequencies and mode shapes can beam has? Why?
4. How will you verify whether the simulation is correct? Explain methods.
5. What do you mean by Degree of Freedom?
6. What is the total degree of freedom of ANSYS commercial package?
7. What is discretization?
8. Mention different types of elastic constants.
9. Specify the terms required to solve FEA problem.
10. Define: Modulus of rigidity, Bulk modulus.

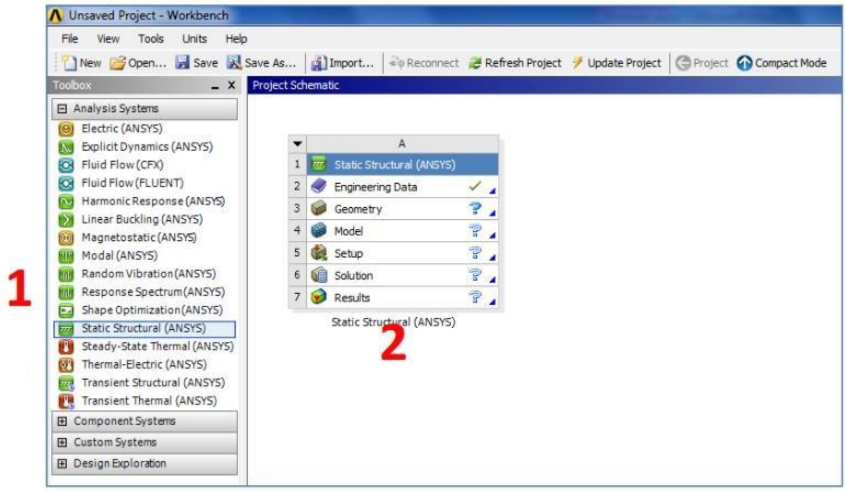
Experiment no-5 Comparison of Stress Concentrated Loads

Opening Workbench

1.On your Windows 7 Desktop click the **Start** button.
2.Under **Search Programs and Files** type "*ANSYS*"
3.Click on **ANSYS Workbench** to start workbench. This step may take time.

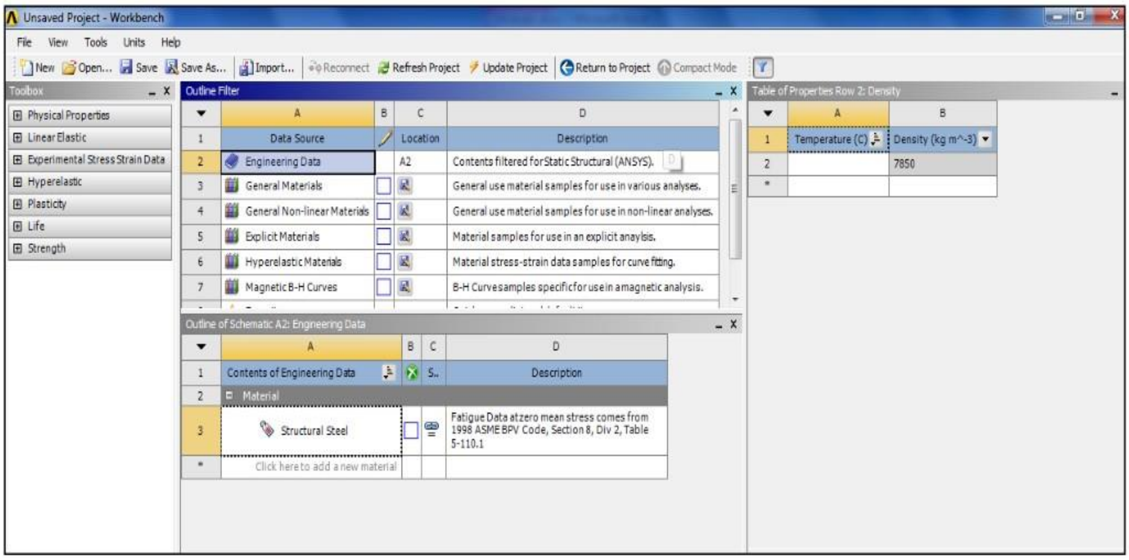
Comparison of Stress Concentrated Loads

1.As you open ANSYS you can see the entire array of problems on the left had side this software can help you solve. The problem at hand is a *Static Structural* problem. Double click **Static Structural (ANSYS)** to open the task manager for your problem set in the Project Schematic area.
2.ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see **Static Structural (ANSYS),** double click this to change the name. For this problem choose "*3DCantilever beam*."



Engineering Data

To begin setup for your cantilever beam, double click or right click on ***Engineering Data*** and click ***edit***. This will bring up another screen.

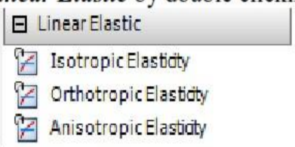


This new window will allow you to alter the material properties of your cantilever beam. Under **Outline of Schematic A2: Engineering Data**, it shows ***click here to add a new material***, this menu allows you to input the material of your cantilever beam, double click and type ***Aluminum***.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Now expand ***Linear Elastic*** by double clicking on 

or on the plus



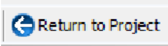
symbol shown.

Double click on Isotropic Elasticity to give the material the same properties across the beam. This action brought up a new table on the right; this allows us to add necessary properties. As shown on the top right of the screen in ***Table of Properties Row 2: Isotropic Elasticity***:

1. Click in Temperature and type 25
2. Click in Young's Modulus and type 70E9 or 7E10
3.Click in Poisson's Ratio and type 0.33
4. Make sure to DELETE the Temperature entry after property input before continuing! Failure to do so will lead to errors later.

Table of Properties Row 2: Isotropic Elasticity

	A	B	C
1	Temperature (C)	Young's Modulus (Pa)	Poisson's Ratio
2	25	7E+10	0.33
*			

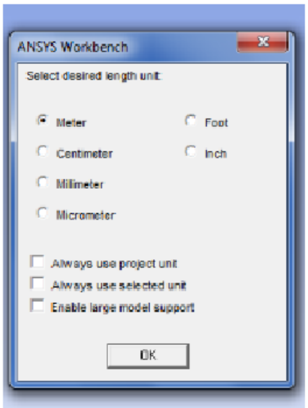
After filling in the properties, this concludes the Engineering Data, to return to the project schematic area, click on 

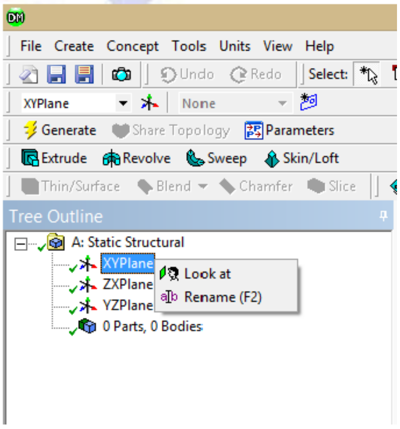
seen on the upper tab.

Geometry-GEOMETERY 1(Non Filleted)

Base Geometry

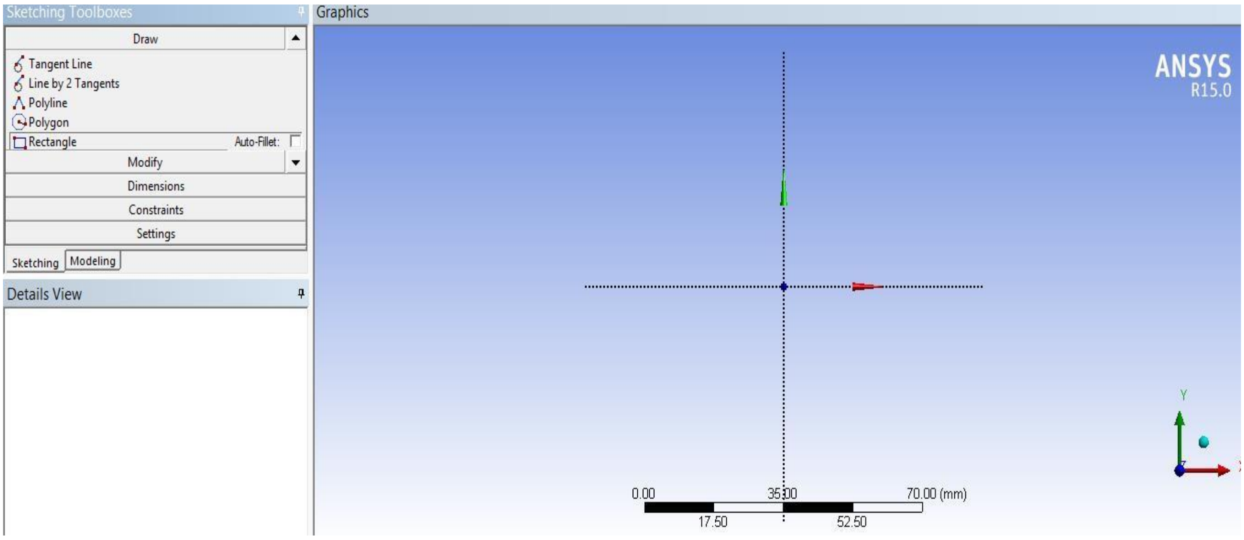
1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.





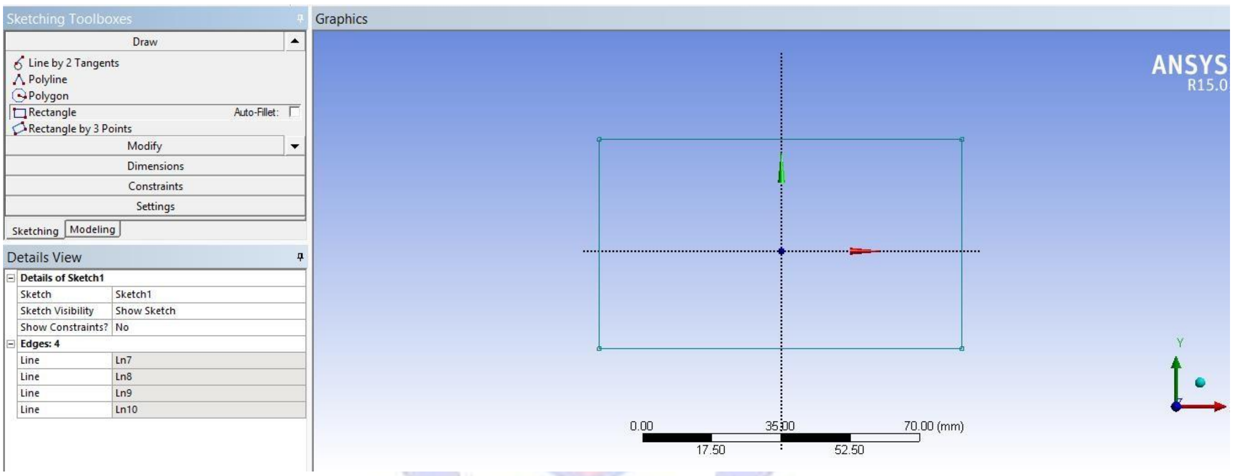
2. Note: Select meters and hit ok
3. In the new window, click the **Display Plane** icon to toggle the coordinate system.
4. Go to **Design Modeler > Tree Outline** >right click on XY Plane. Click **Look At** to view the YZ plane.
5. Go to **Design Modeler -> Tree Outline -> Sketching**

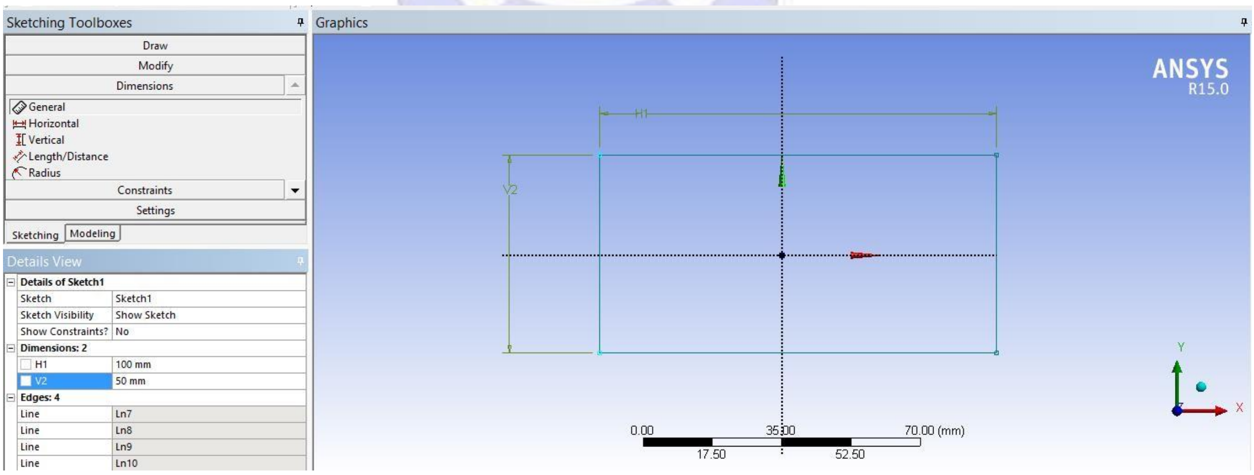
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



6. Click on **Rectangle** and Click off **Auto-Fillet:**

7. Draw a rectangle by free hand.





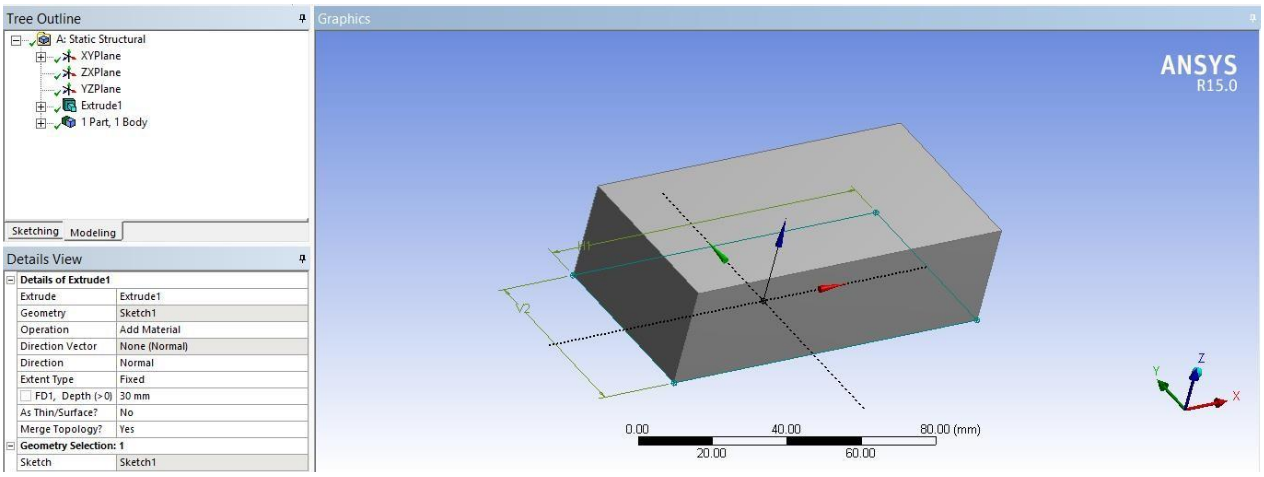
Give the geometry dimensions i.e. ***V2=50mm*** and ***H1=100mm***.

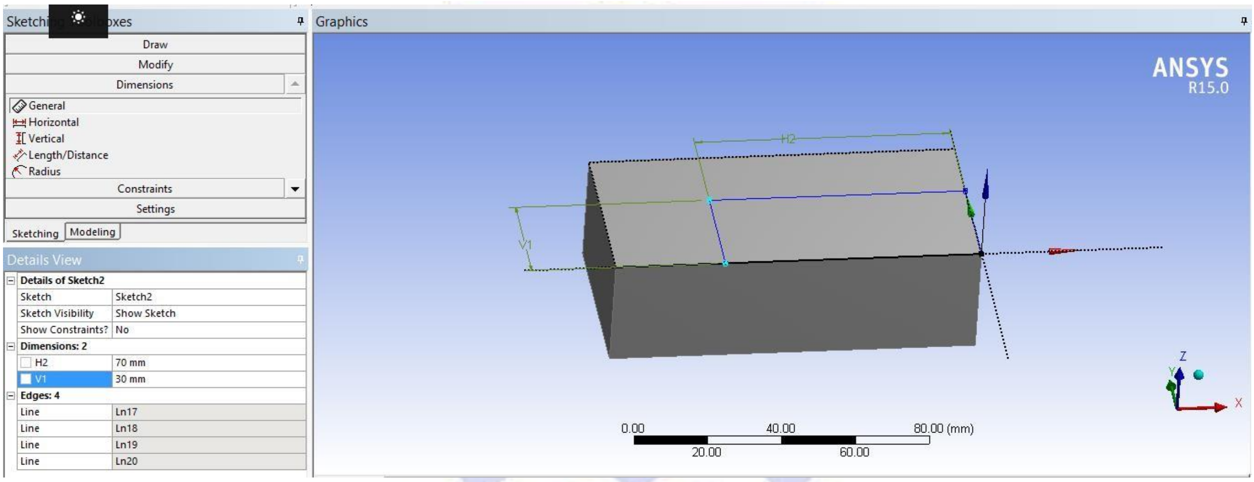
8. Extrude the geometry to the height=***30mm***.

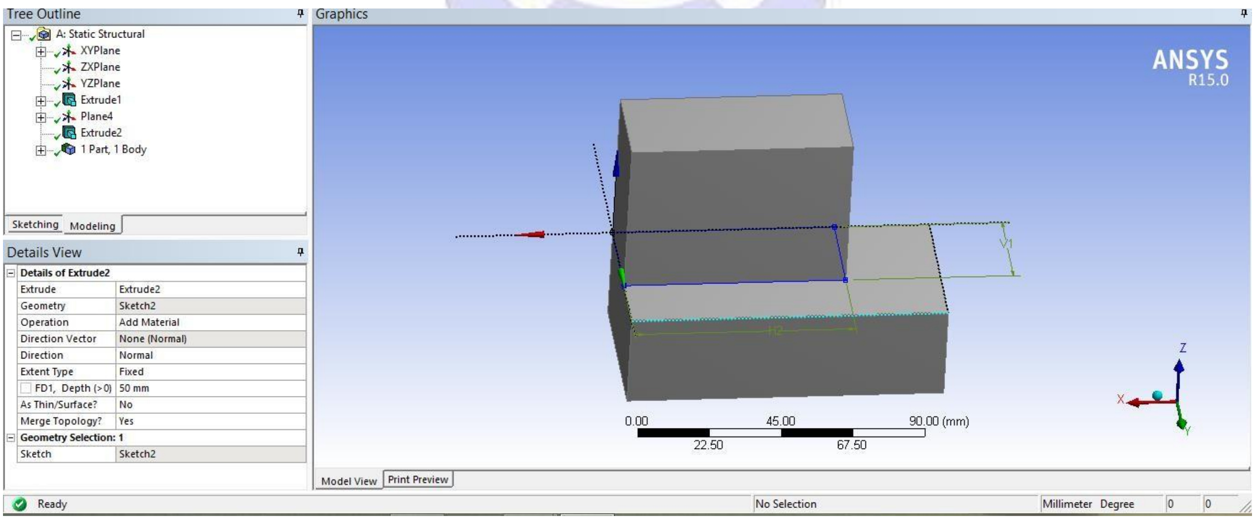
9. Select the **Face Element**  or press**Ctrl+F.**

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

10. Select the longer face of the solid, and click on sketching. Draw another rectangle over the longer face and give dimensions ***V1=30mm*** and ***H2=70mm***.
11. Extrude the face at height **50mm**.

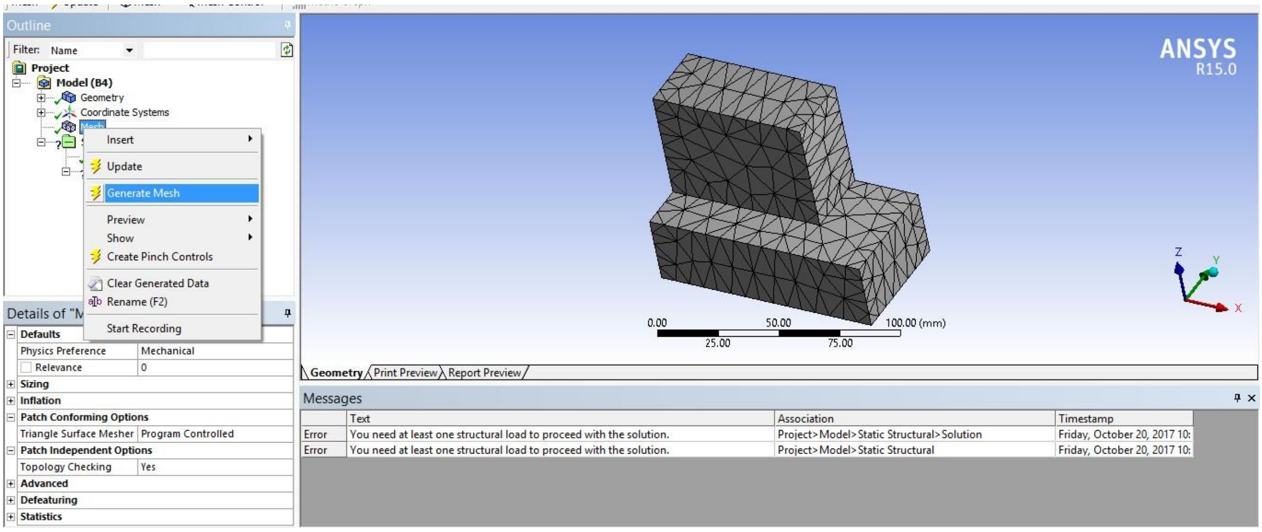




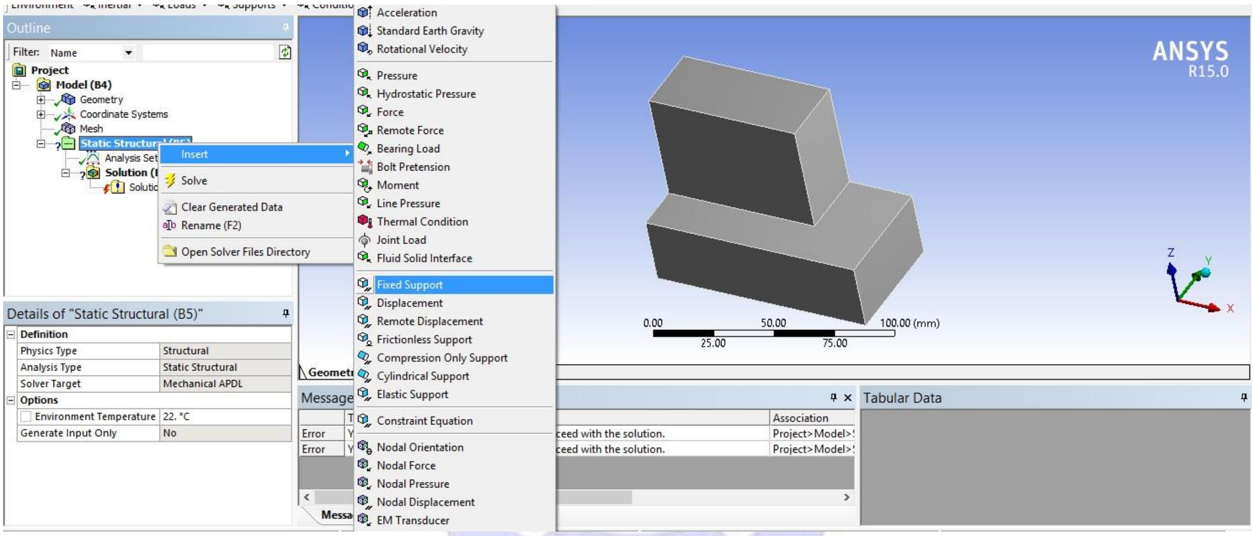


12. Close the **Design Modeler** and switch to the **Model** from *Project Schematic.*

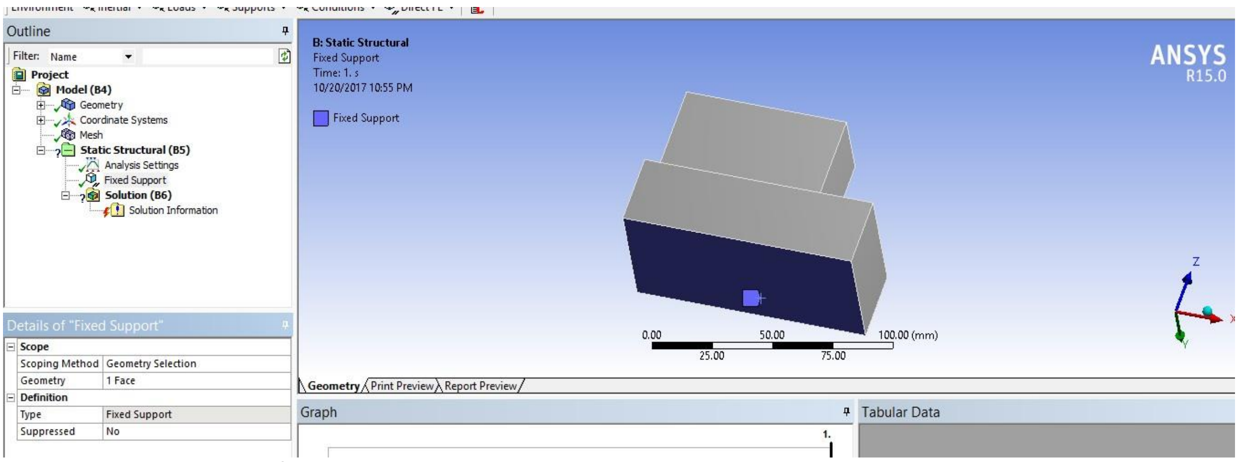
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



13. Select **Mesh(*right click*)>>Generate Mesh.**



14. Select **Static Structural**(Right Click)>>**Insert>>Fixed Support.**



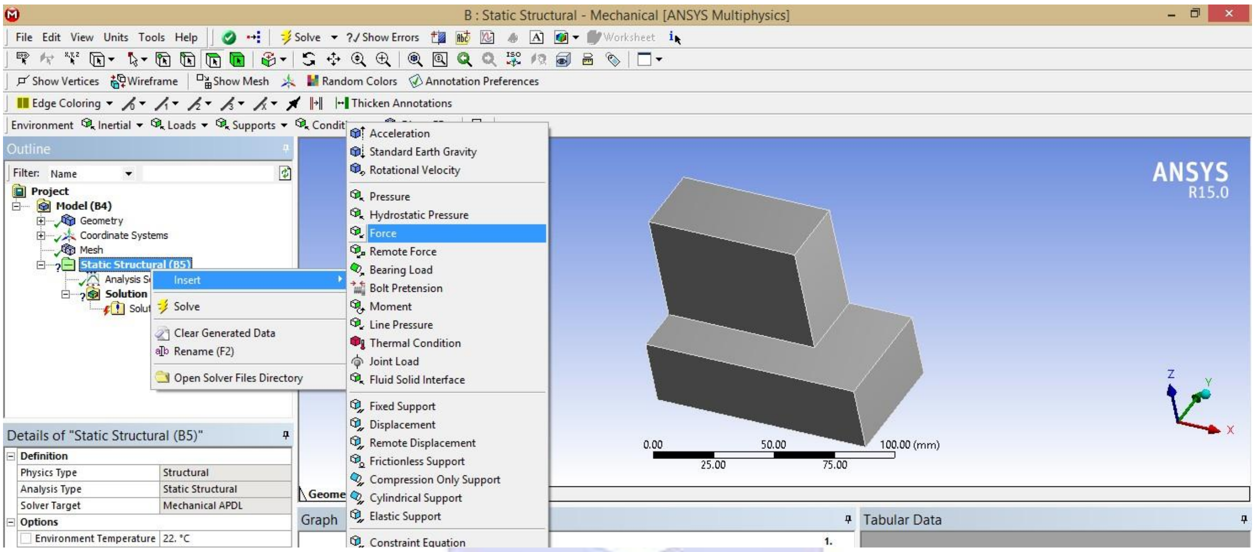
15. Choose the base as the fixed support and select apply.

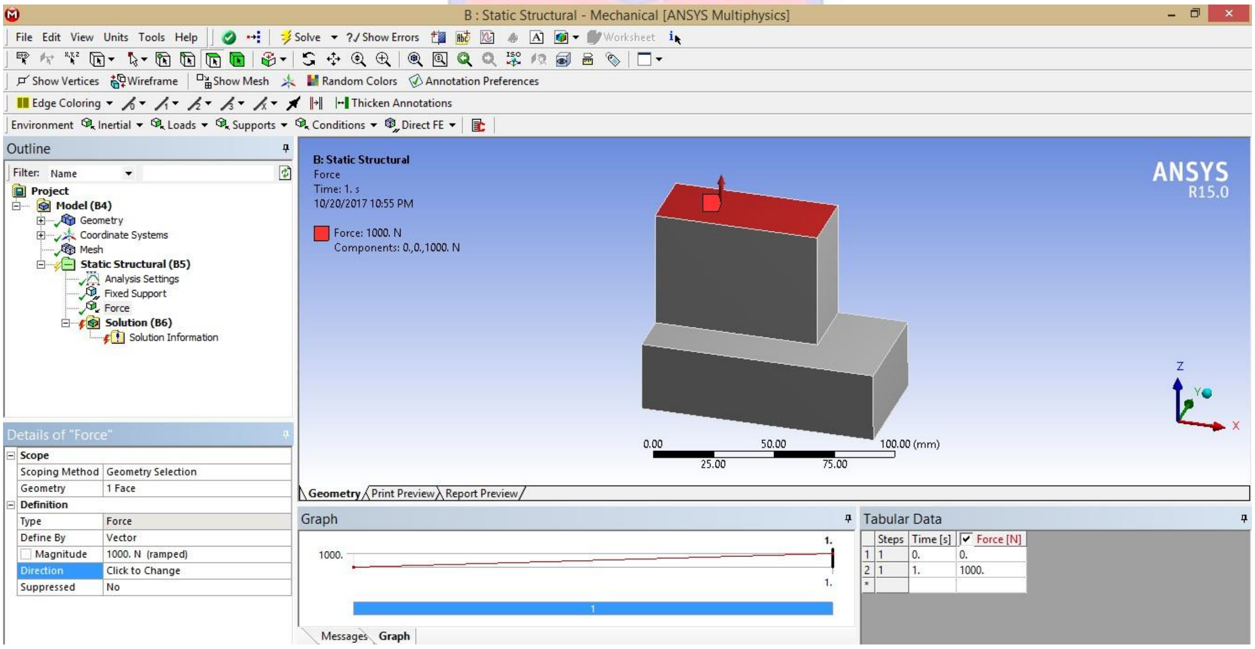
16. Select **Static Structural**(Right Click)>>**Insert>>Force.**

17. Choose the top surface of the as geometry to apply.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

18. Select the **Magnitude** and give it the value of ***1000N***.
19. Give the direction as shown.

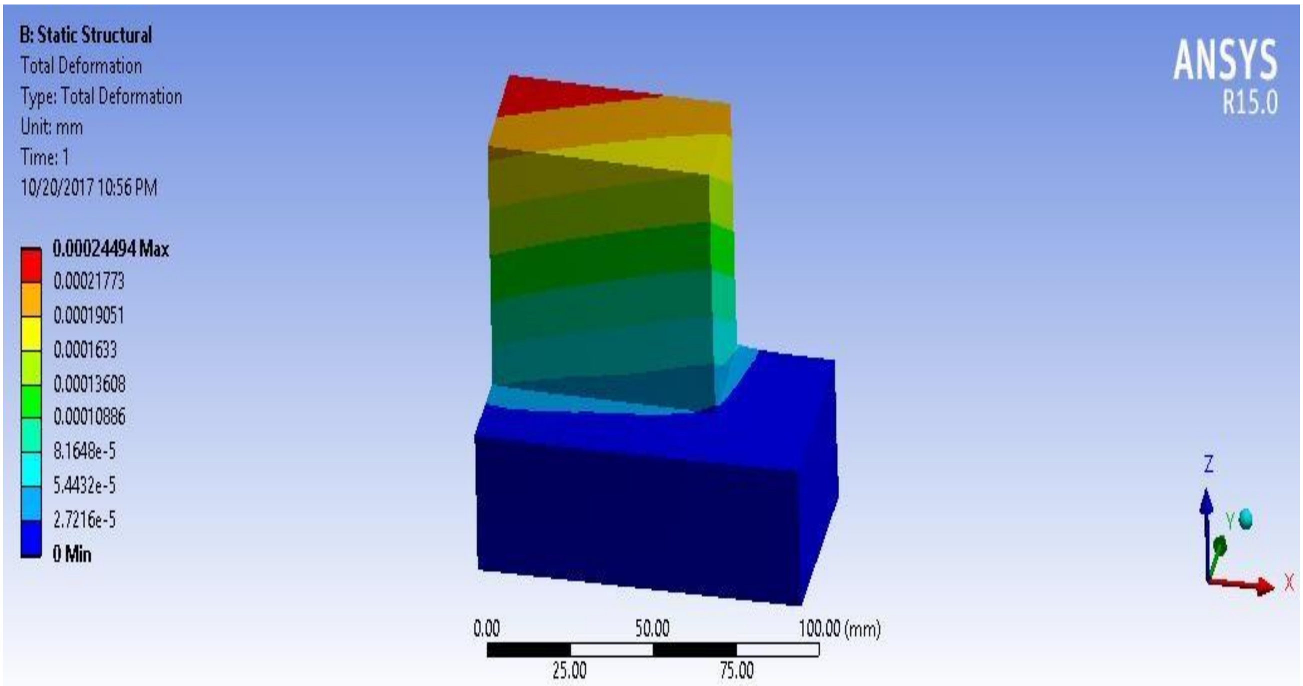




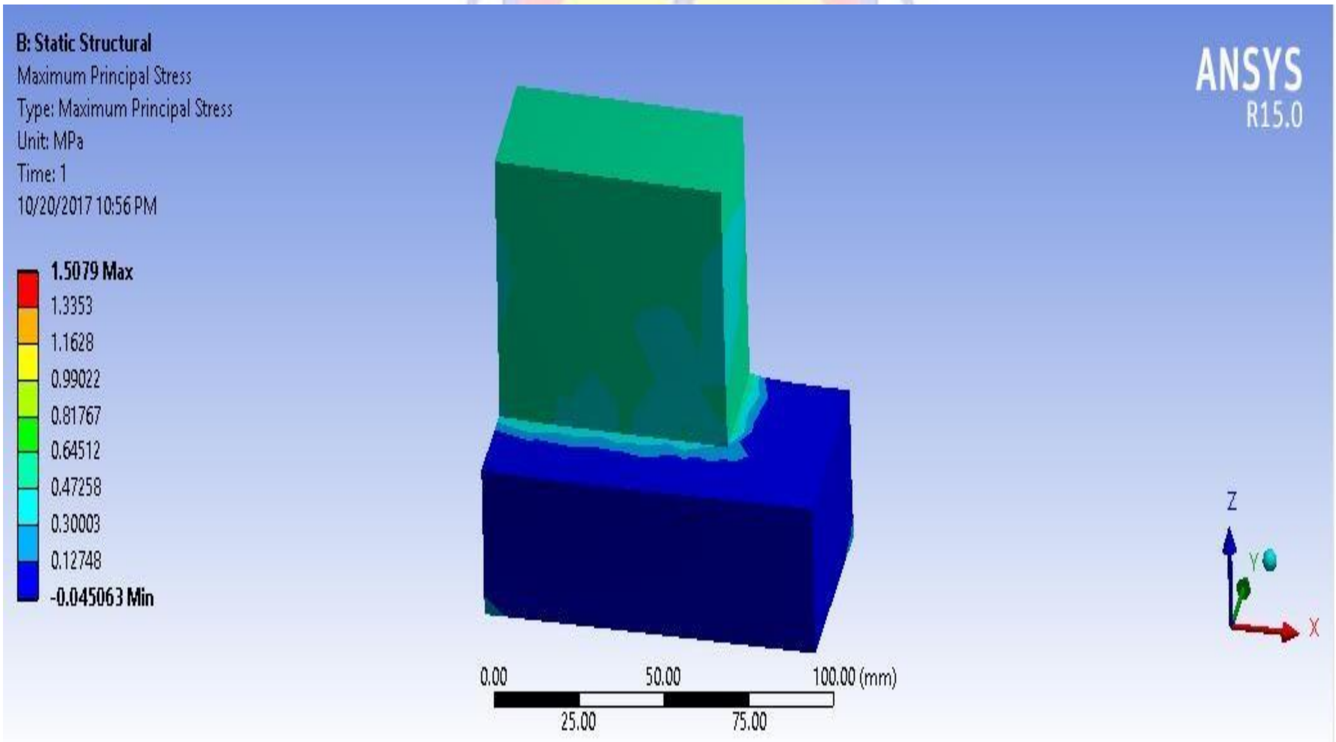
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Choose *Total Deformation* and *Max. Principle Stress* from solver and click on ***SOLVE.***

20. Note the values of the *Total Deformation*=***8.164x10^{-5}***and the value for *Max. Principle Stress*=



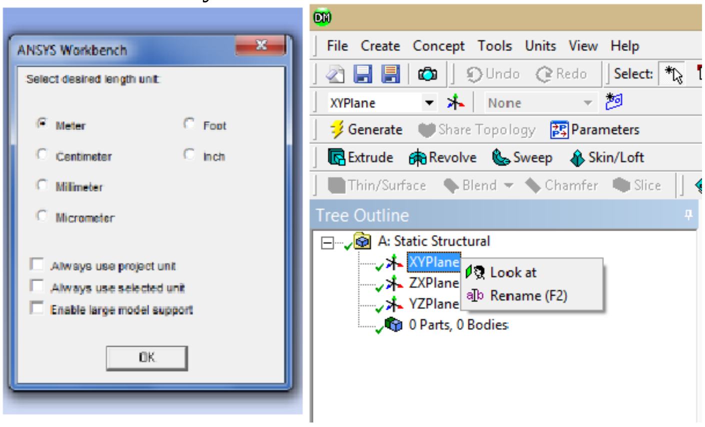
0.47258MPa at the meeting edges.



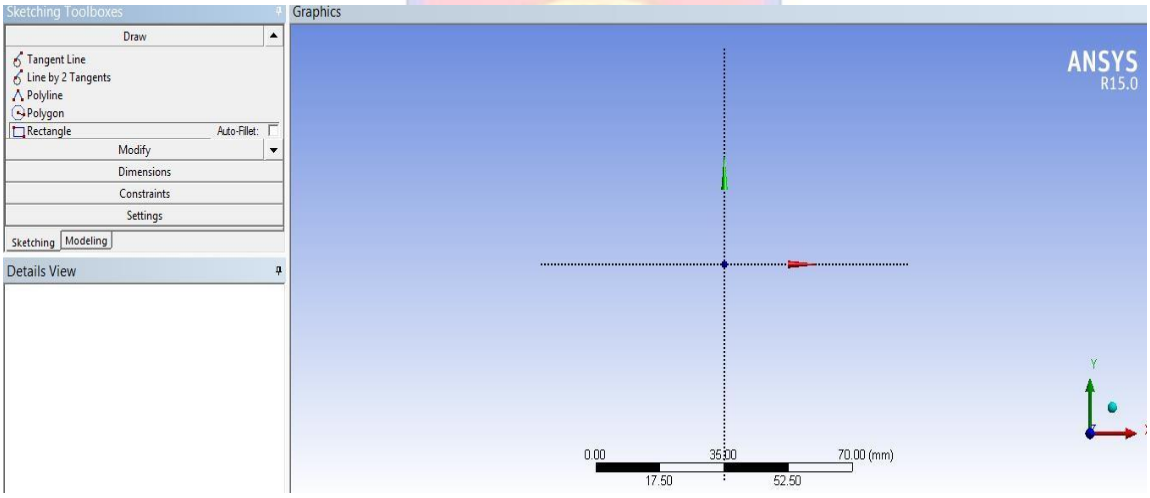
<u>Geometry-GEOMETERY 2(Filleted)</u>

Base Geometry

1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.

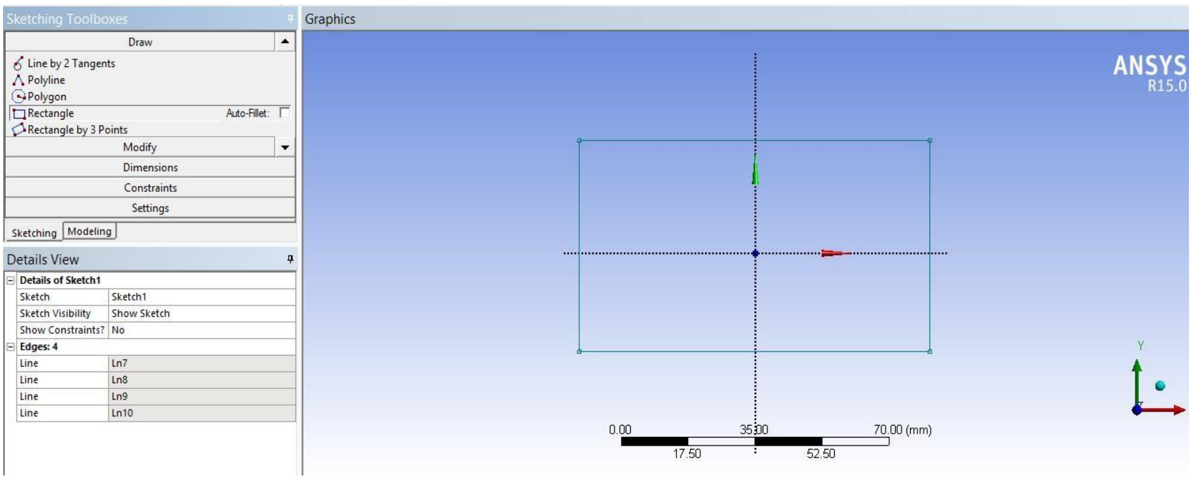


2. Note: Select meters and hit ok
3. In the new window, click the **Display Plane** icon to toggle the coordinate system.
4. Go to **Design Modeler** > **Tree Outline** >right click on XY Plane. Click **Look At** to view the YZ plane.
5. Go to **Design Modeler -> Tree Outline -> Sketching**

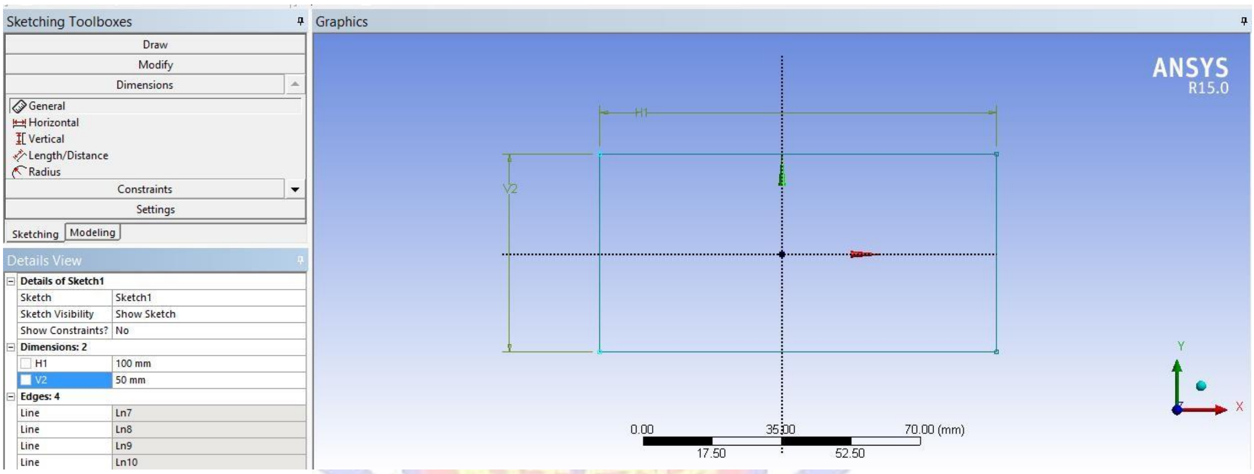


6. Click on **Rectangle** and Click off **Auto-Fillet:**
7. Draw a rectangle by free hand.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



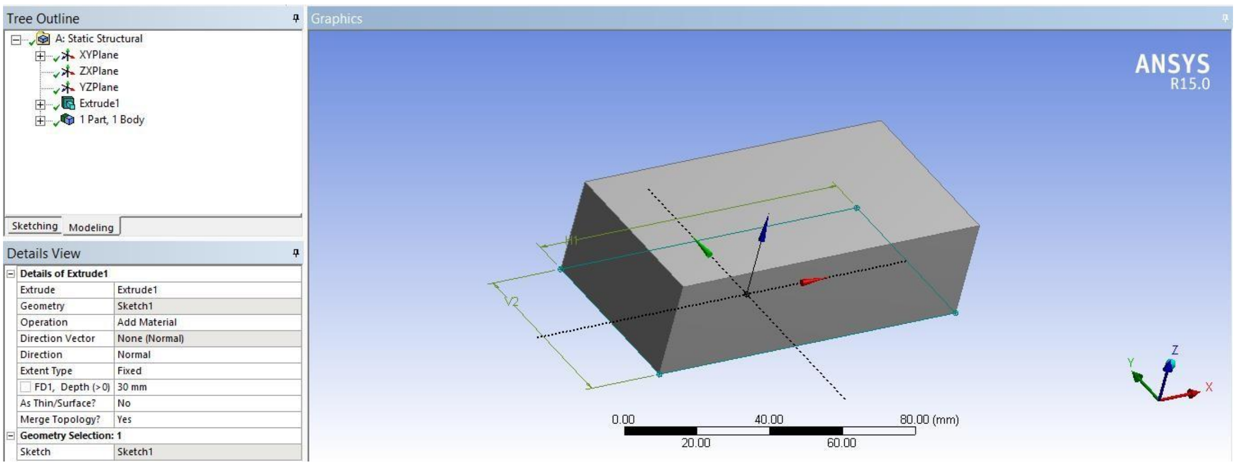
8. Give the geometry dimensions i.e. ***V2=50mm*** and ***H1=100mm***.

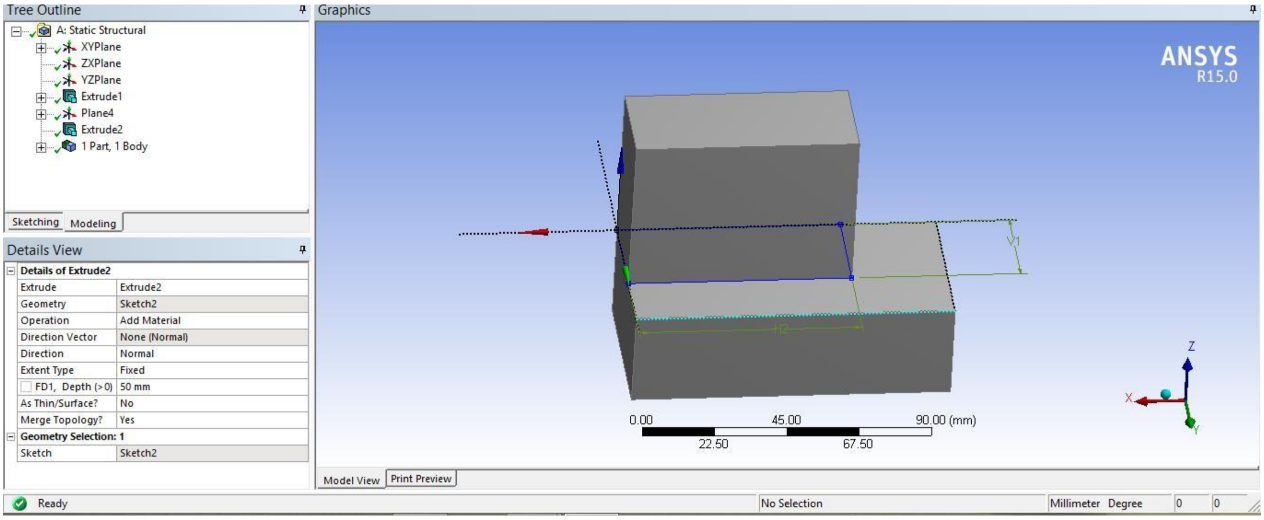


9. Extrude the geometry to the height=***30mm***.

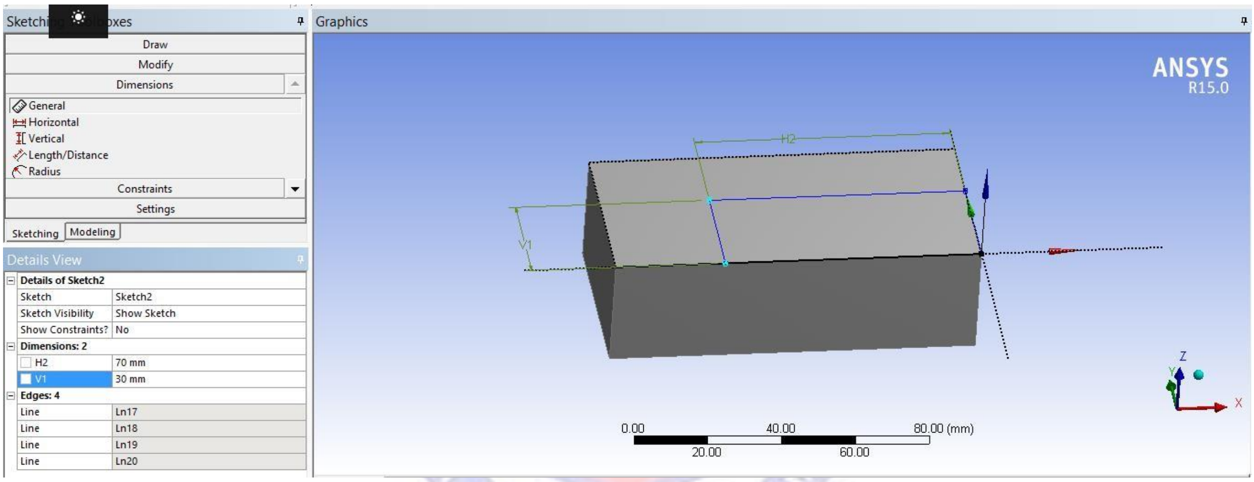
10. Select the **Face Element**  or press **Ctrl+F.**

11. Select the longer face of the solid, and click on sketching. Draw another rectangle over the longer face and give dimensions ***V1=30mm*** and ***H2=70mm***.

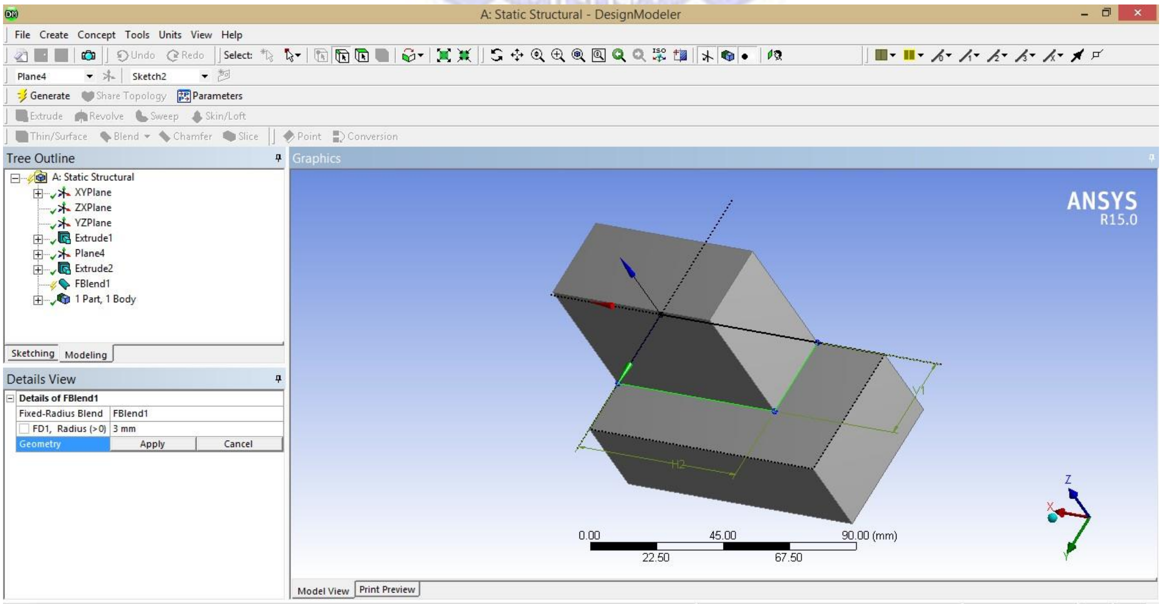




12. Extrude the face at height **50mm**.

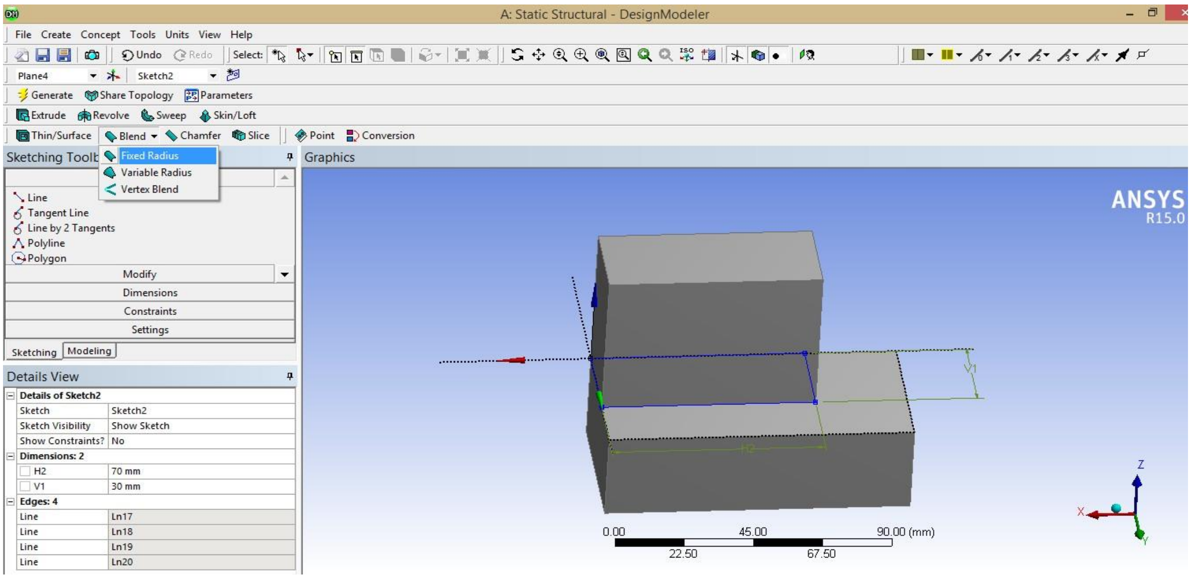


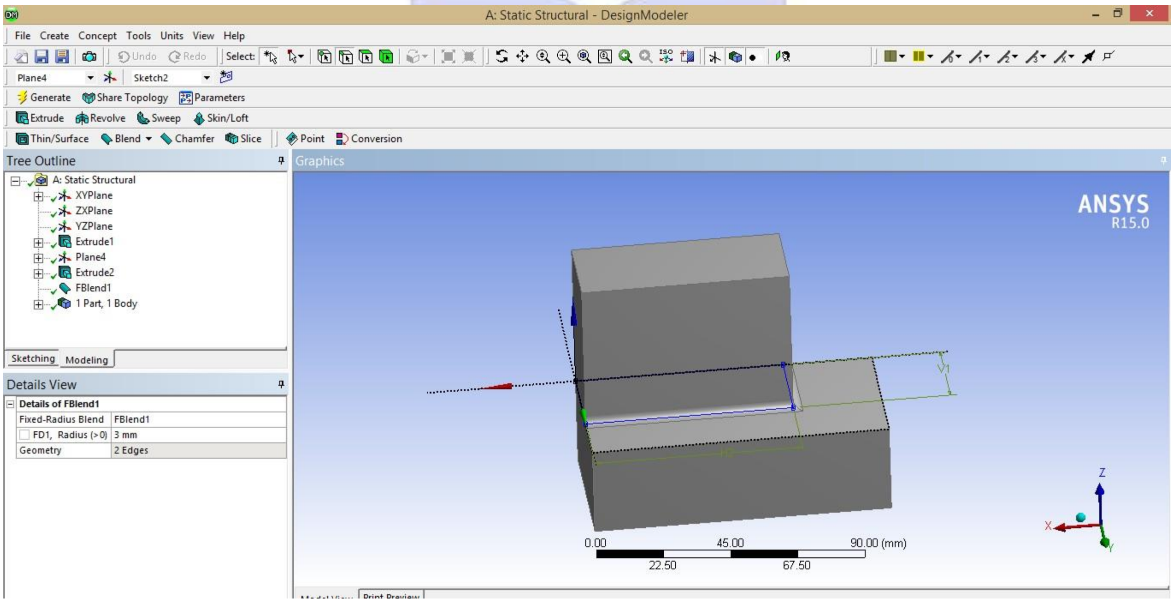
13. Now add a fillet over the meeting edges to the solid.



Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

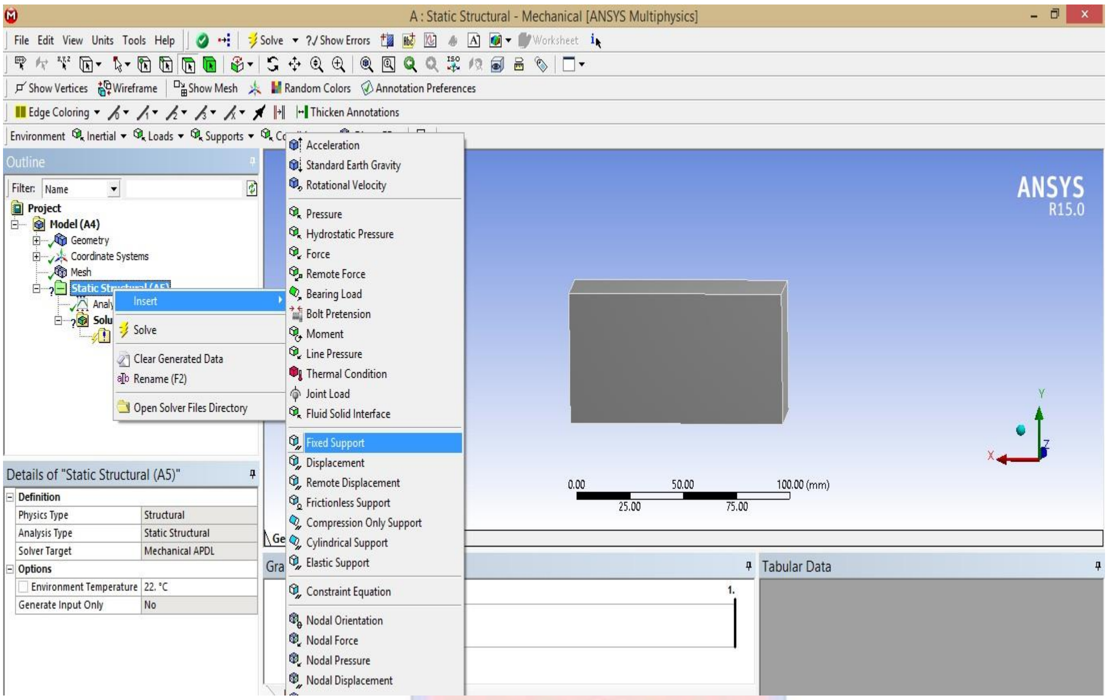
14. Give the value of the filleted radius equal **3mm**.



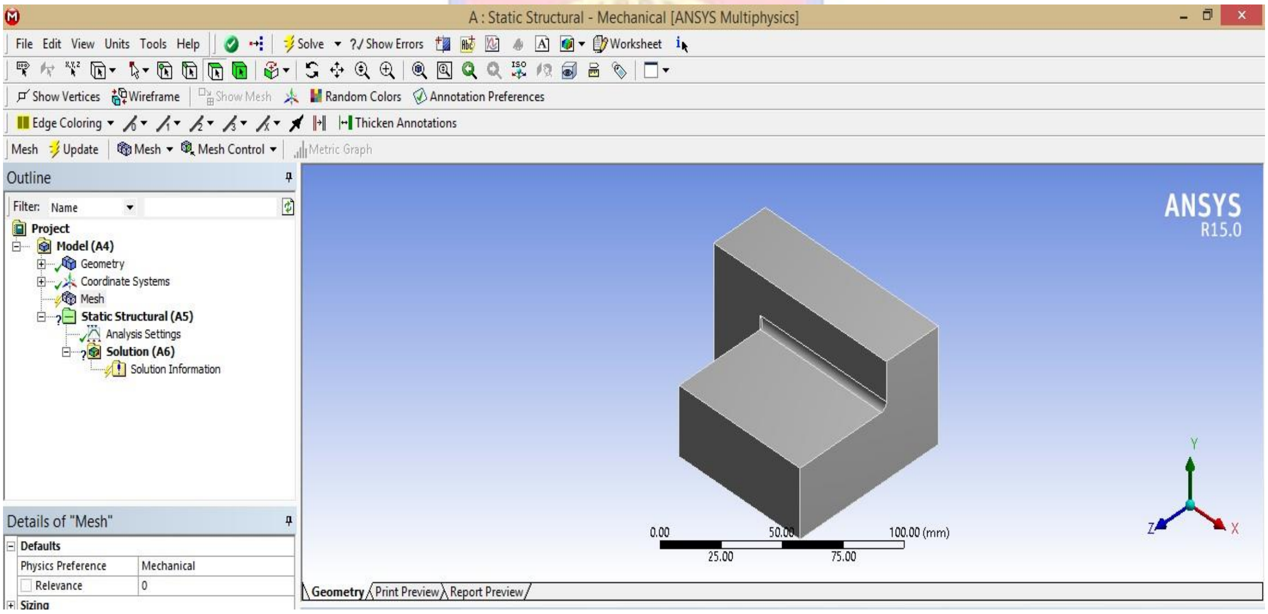


15. Select the meeting edges and click generate.
16. Close the **Design Modeler** and open **MODEL.**
17. Select **Static Structural**(Right Click)>>**Insert>>Fixed Support.**

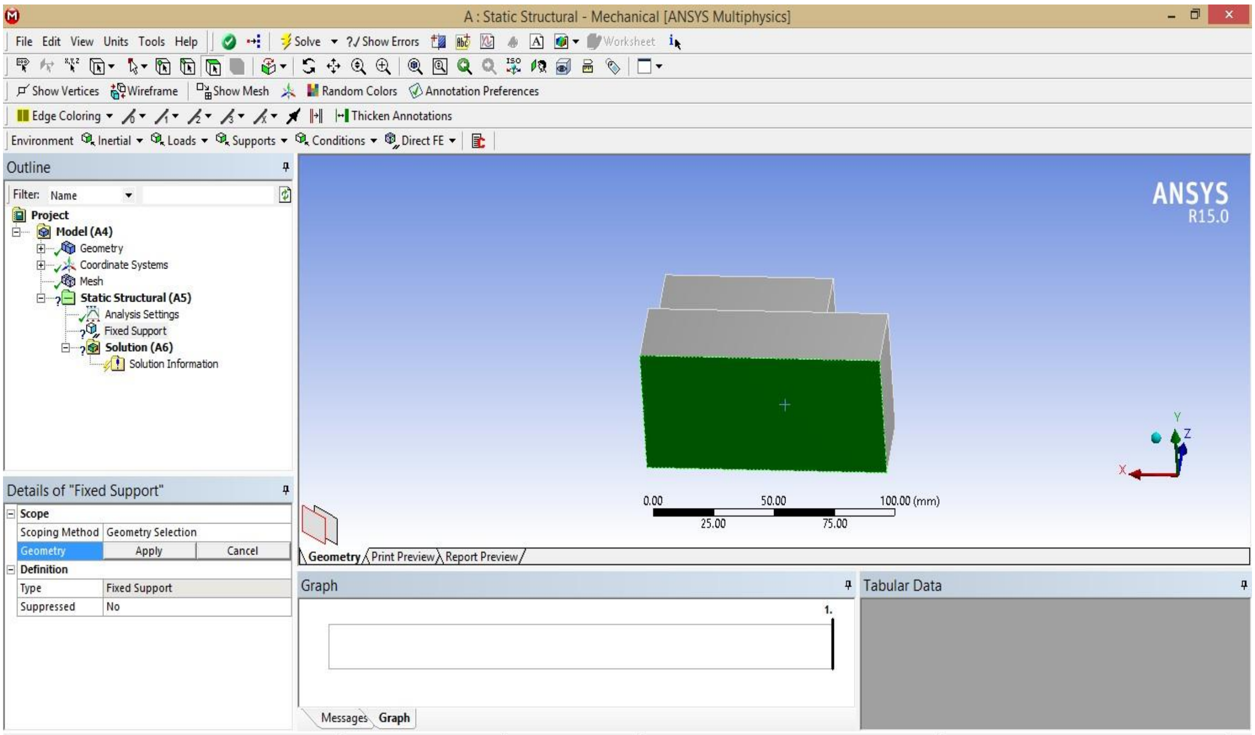
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



18. Choose the base as the fixed support and select apply.



Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

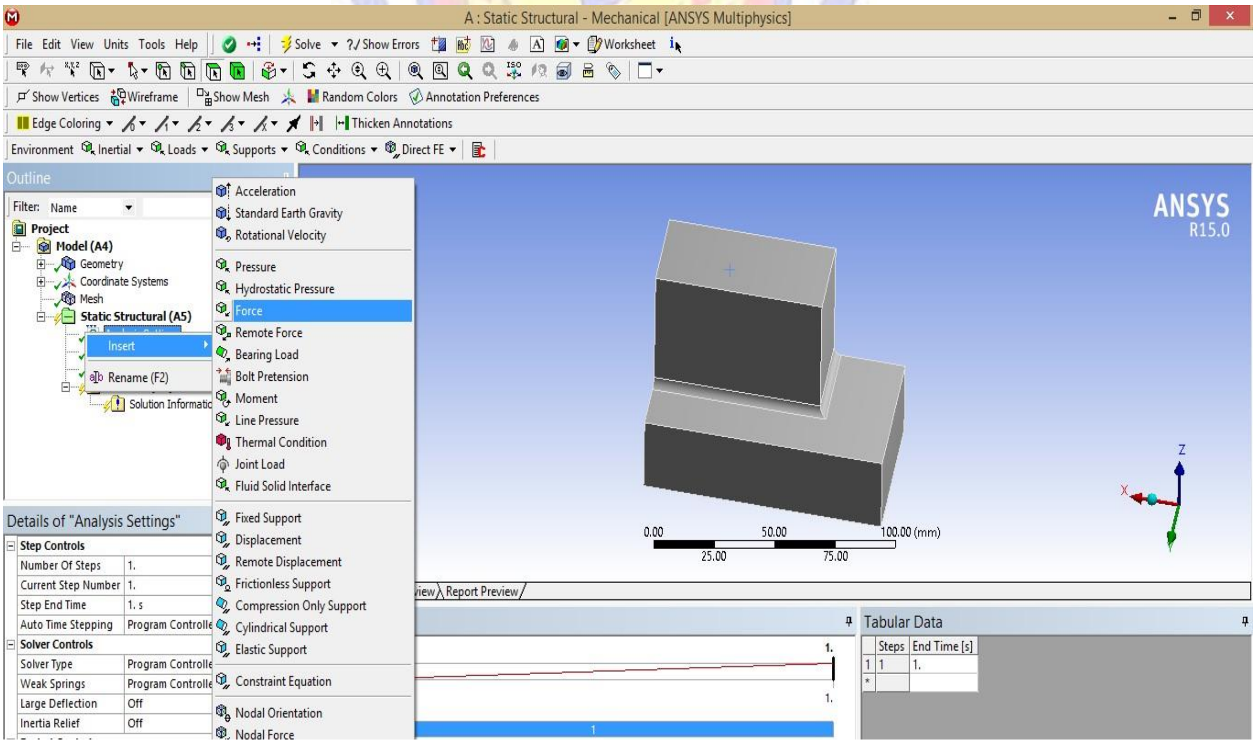


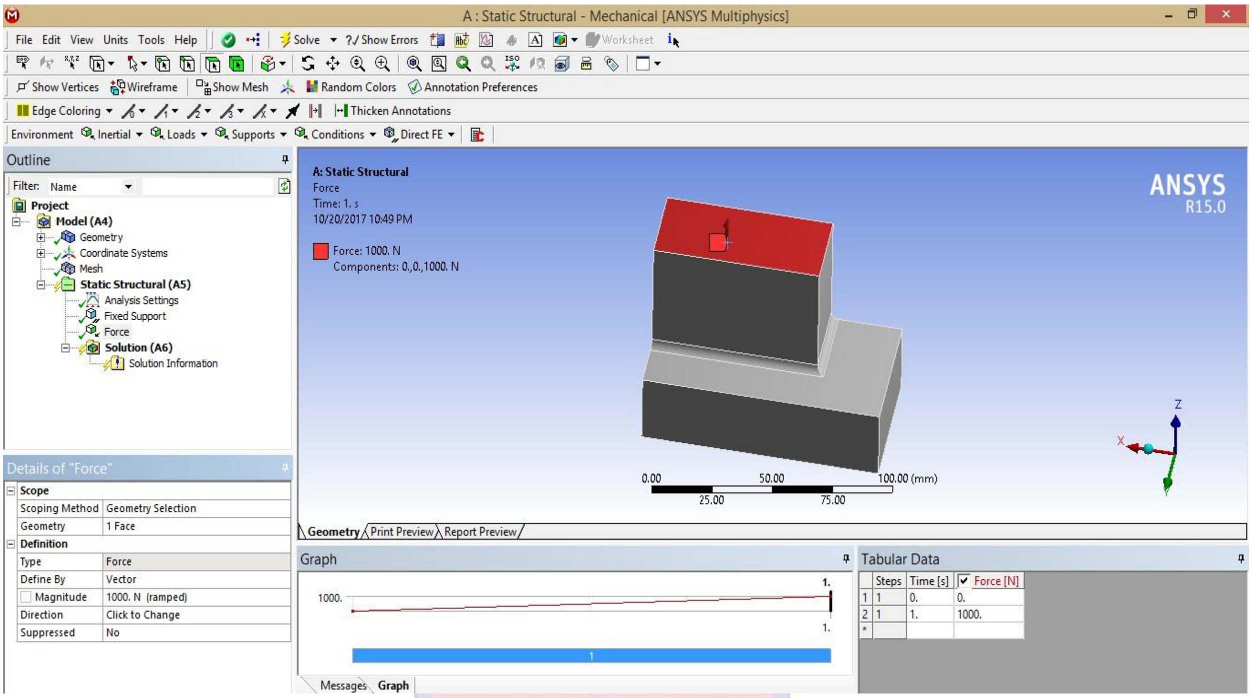
19. Select **Static Structural** (Right Click)>>**Insert>>Force.**

20. Choose the top surface of the as geometry to apply.

21. Select the **Magnitude** and give it the value of ***1000N***.

22. Give the direction as shown.





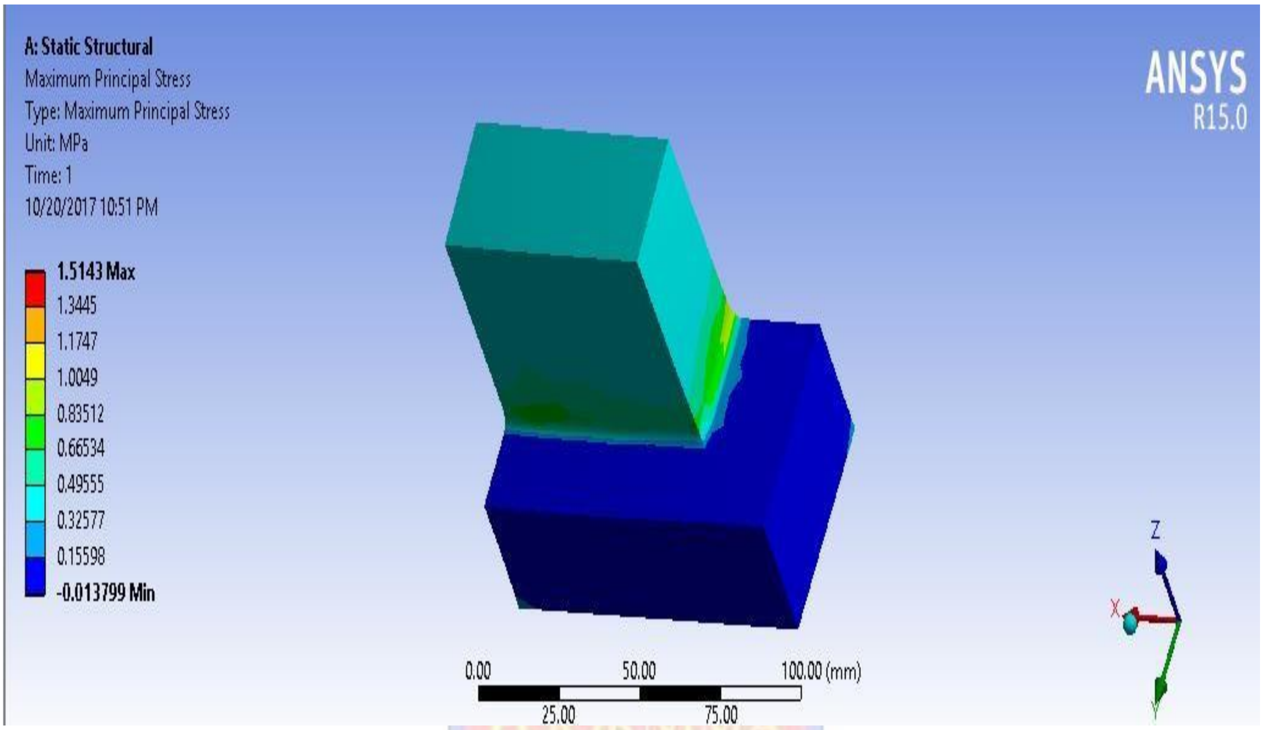
23. Choose *Total Deformation* and *Max. principle Stress* from solver and click on ***SOLVE.***

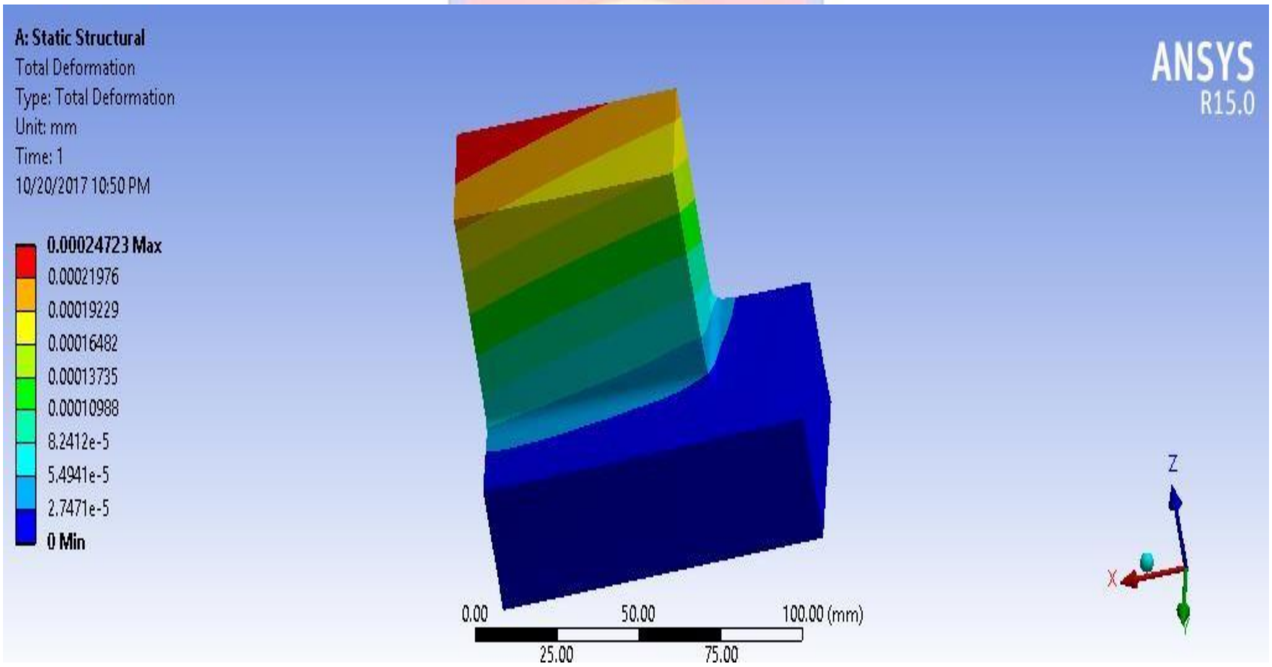
24. Note the Values of *Total Deformation*=***0.00024723 mm*** and that of *Max. Principle Stress* =***1.5143MPa*** at the fillet point.

Now compare both the values.

Sr. No.	TOTAL DEFORMATION	MAX. Principle Stress
GEOMETERY 1	***8.24x10^{-5}***	***0.4258MPa***.
GEOMETERY 2	***8.124x10^{-5}***	***0.32577MPa.***

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR


A: Static Structural
Maximum Principal Stress
Type: Maximum Principal Stress
Unit: MPa
Time: 1
10/20/2017 10:51 PM
1.5143 Max
1.3445
1.1747
1.0049
0.83512
0.66534
0.49555
0.32577
0.15598
-0.013799 Min
ANSYS
R15.0
0.00
50.00
100.00 (mm)
25.00
75.00


A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
10/20/2017 10:50 PM
0.00024723 Max
0.00021976
0.00019229
0.00016482
0.00013735
0.00010988
8.2412e-5
5.4941e-5
2.7471e-5
0 Min
ANSYS
R15.0
0.00
50.00
100.00 (mm)
25.00
75.00

Viva Questions:-

1. What is **Poisson's ratio** and its significance?
2. What do u means by stress concentration & u mesh for such regions?
3. What is shape function?
4. How do verify and validate the results obtained from FEA?
5. Are there any recommended commercial FEM packages that are versatile in handling a wide range of problems?
6. What is optimization in ANSYS?
7. What is Endurance limit?
8. Define truss.
9. Mention some common material properties.
10. What is the purpose of meshing or dividing the component into finite elements?

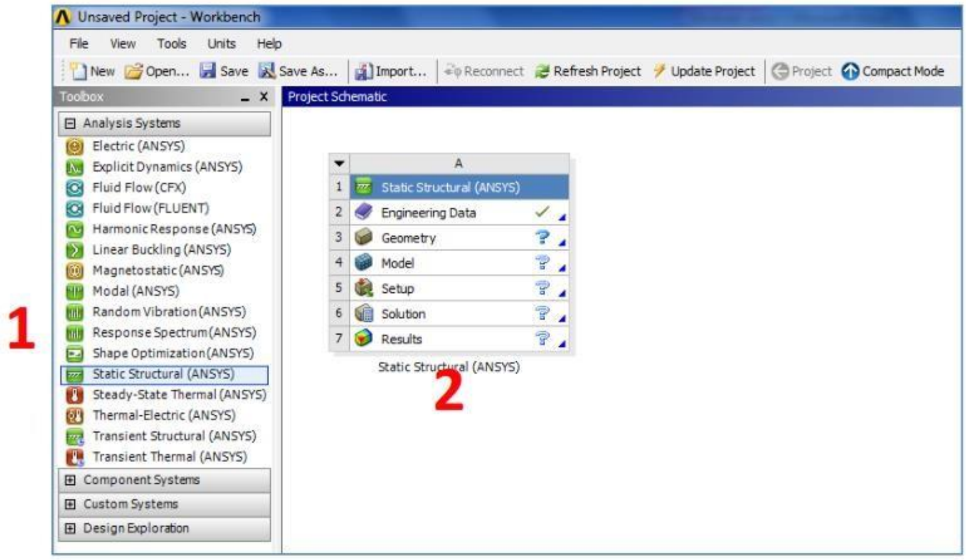
Experiment No.6 Analysis of Axis-Symmetric Solids

Opening Workbench

1.On your Windows 7 Desktop click the **Start** button.
2.Under **Search Programs and Files** type "*ANSYS*"
3.Click on **ANSYS Workbench** to start workbench. This step may take time.

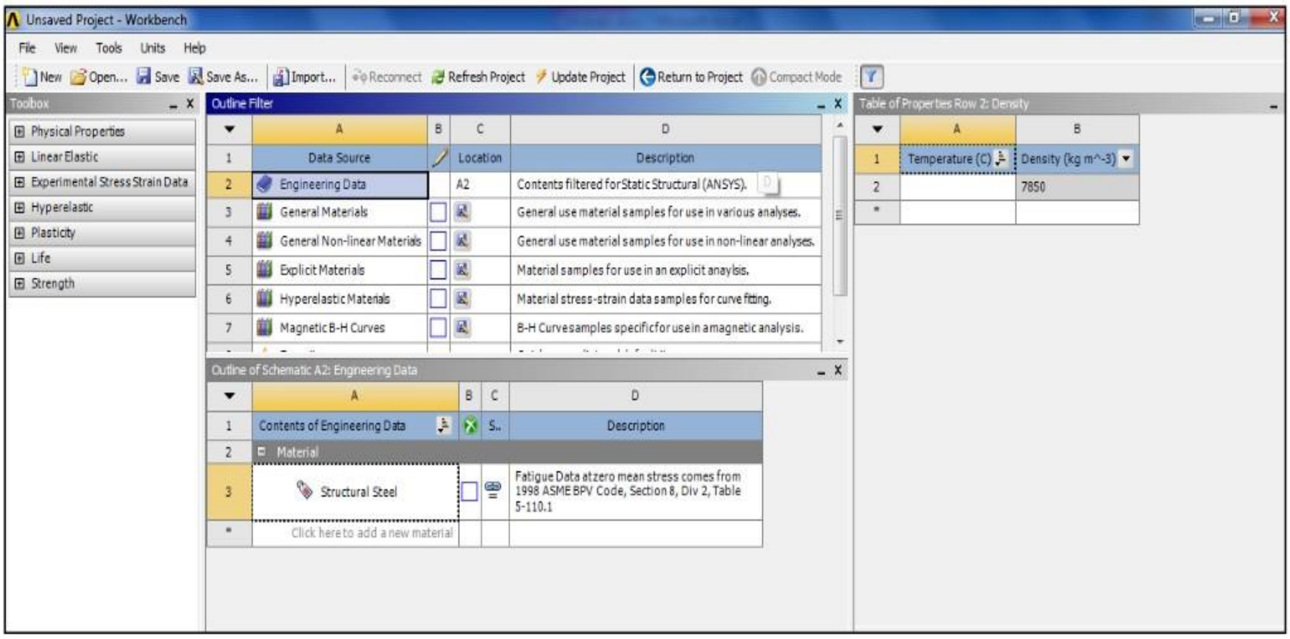
Analysis of Axis-Symmetric Solids

1.As you open ANSYS you can see the entire array of problems on the left had side this software can help you solve. The problem at hand is a *Static Structural* problem. Double click **Static Structural (ANSYS)** to open the task manager for your problem set in the Project Schematic area.
2.ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see **Static Structural (ANSYS),** double click this to change the name. For this problem choose "*3DCantilever beam*."



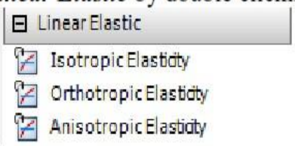
Engineering Data

To begin setup for your cantilever beam, double click or right click on ***Engineering Data*** and click***edit***. This will bring up another screen.



This new window will allow you to alter the material properties of your cantilever beam. Under **Outline of Schematic A2: Engineering Data**, it shows ***click here to add a new material***, this menu allows you to input the material of your cantilever beam, double click and type ***Aluminum***.

Now expand ***Linear Elastic*** by double clicking on 

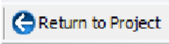
or on the plus symbol shown. 

Double click on Isotropic Elasticity to give the material the same properties across the beam. This action brought up a new table on the right; this allows us to add necessary properties. As show on the top right of the screen in ***Table of Properties Row 2: Isotropic Elasticity***:

1. Click in Temperature and type 25
2. Click in Young's Modulus and type 70E9 or 7E10
3.Click in Poisson's Ratio and type 0.33
4. Make sure to DELETE the Temperature entry after property input before continuing! Failure to do so will lead to errors later.

Table of Properties Row 2: Isotropic Elasticity

	A	B	C
1	Temperature (C)	Young's Modulus (Pa)	Poisson's Ratio
2	25	7E+10	0.33
*			

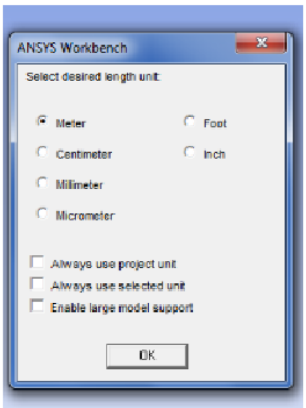
After filling in the properties, this concludes the Engineering Data, to return to the project schematic area, click on 

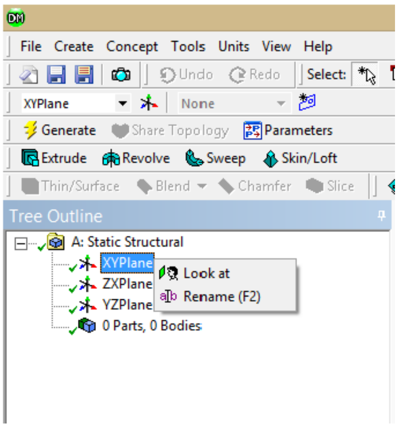
seen on the upper tab.

Geometry

Base Geometry

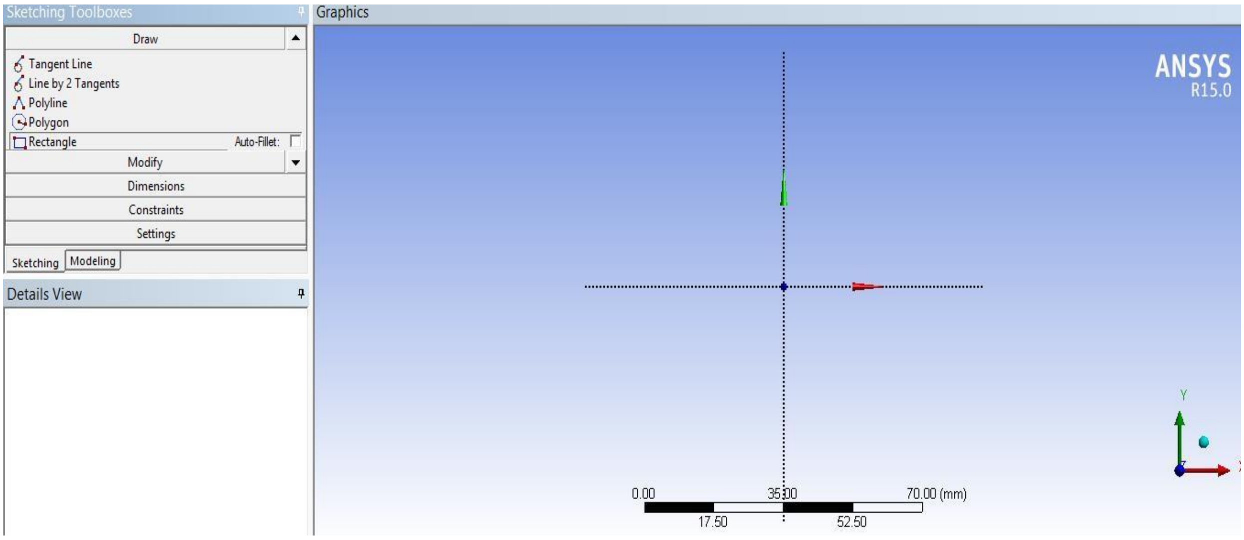
1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.



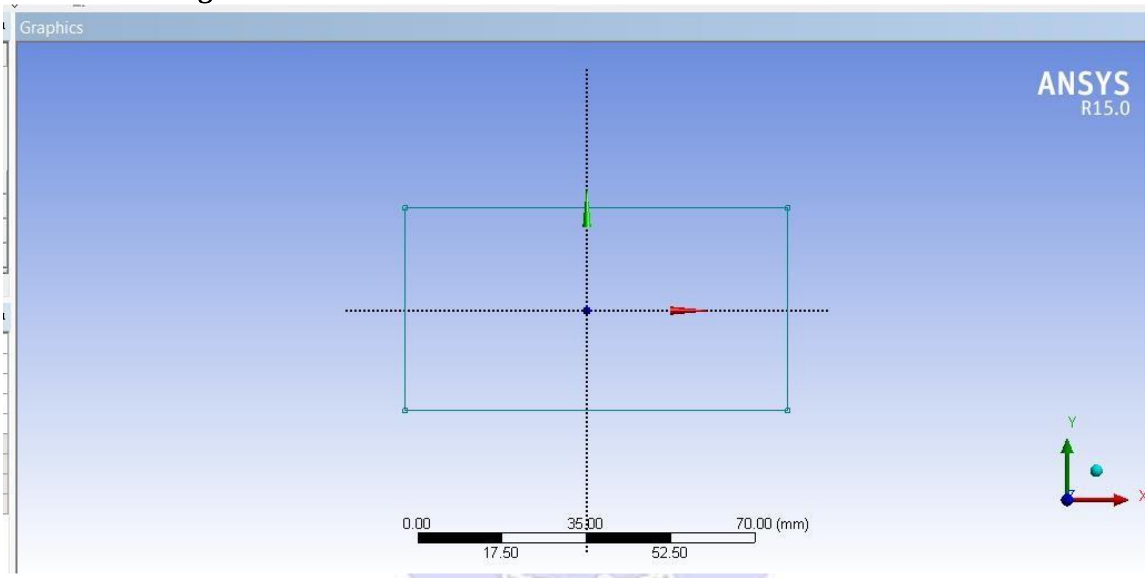


2. Note: Select meters and hit ok
3. In the new window, click the **Display Plane** icon to toggle the coordinate system.
4. Go to **Design Modeler** > **Tree Outline** >right click on XY Plane. Click **Look At** to view the YZ plane.
5. Go to **Design Modeler -> Tree Outline -> Sketching**

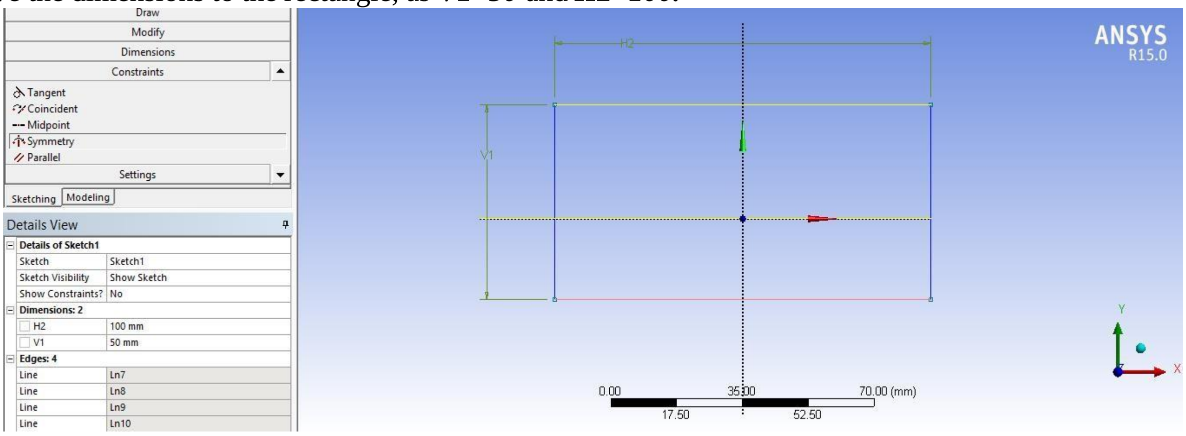
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

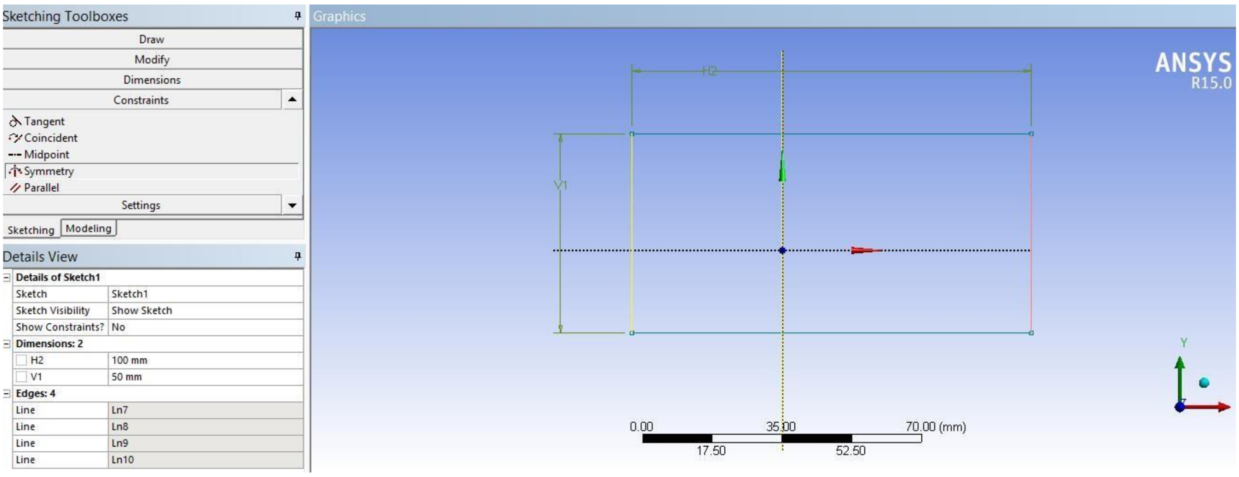


6. Click on **Rectangle** and Click off **Auto-Fillet:**

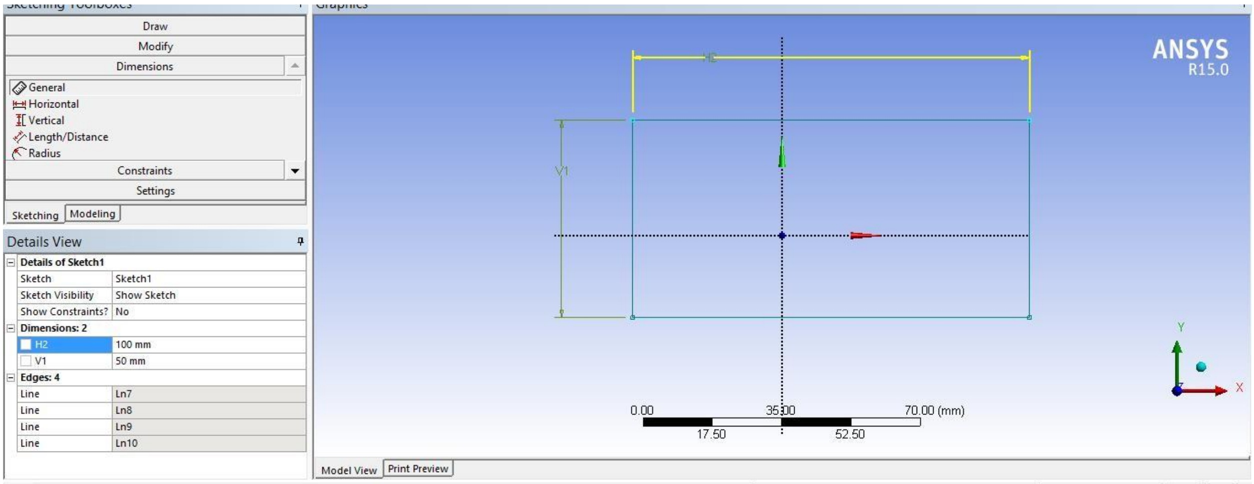


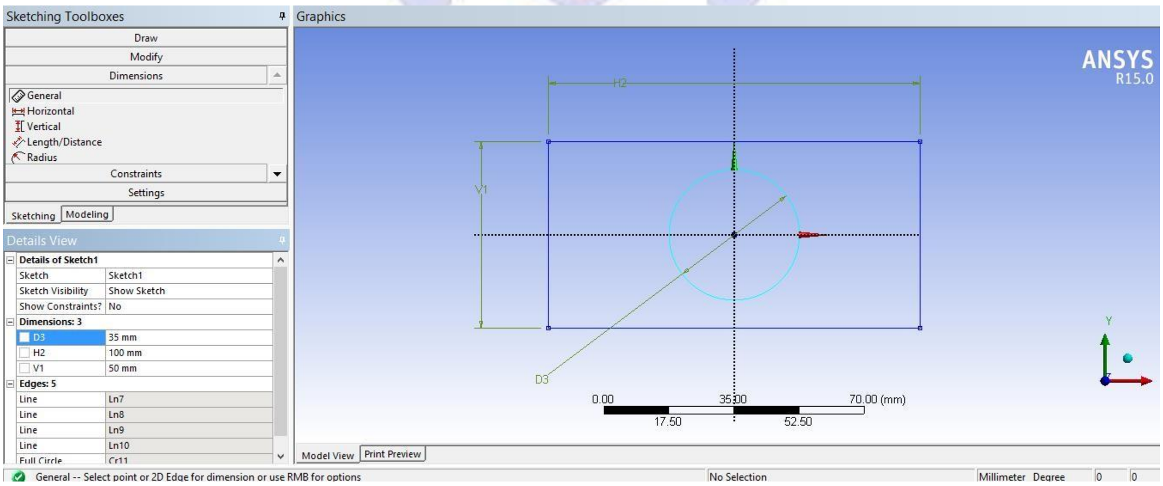
7. Draw a rectangle taking origin into consideration.
8. Give the dimensions to the rectangle, as ***V1=50*** and ***H2=100.***





9. Using *symmetry* constraint bring the intersection of the diagonals of the rectangle at the origin.

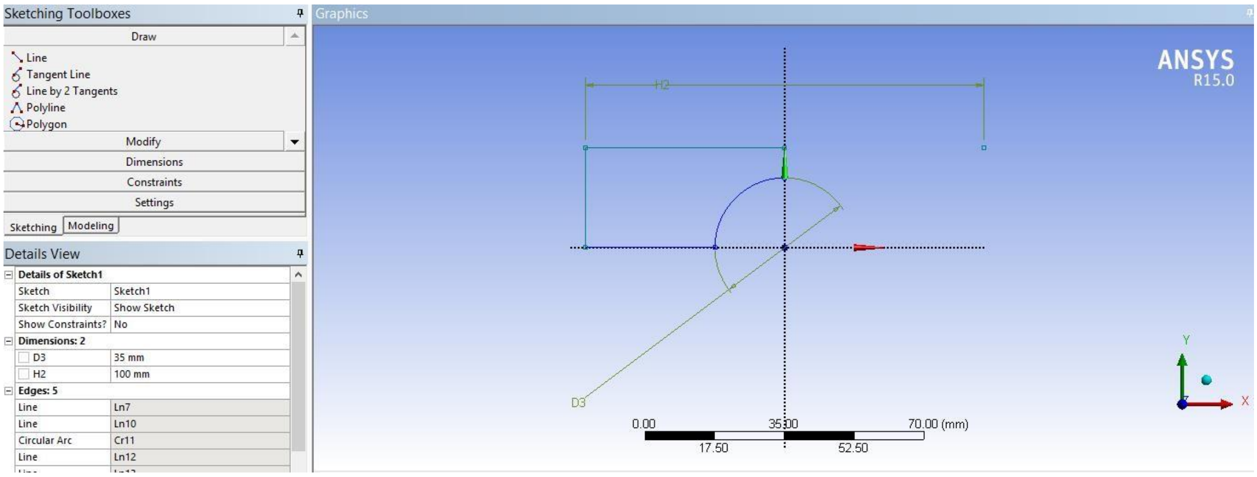




10. Draw a circle of 35mm in the center of rectangle, or at the origin.

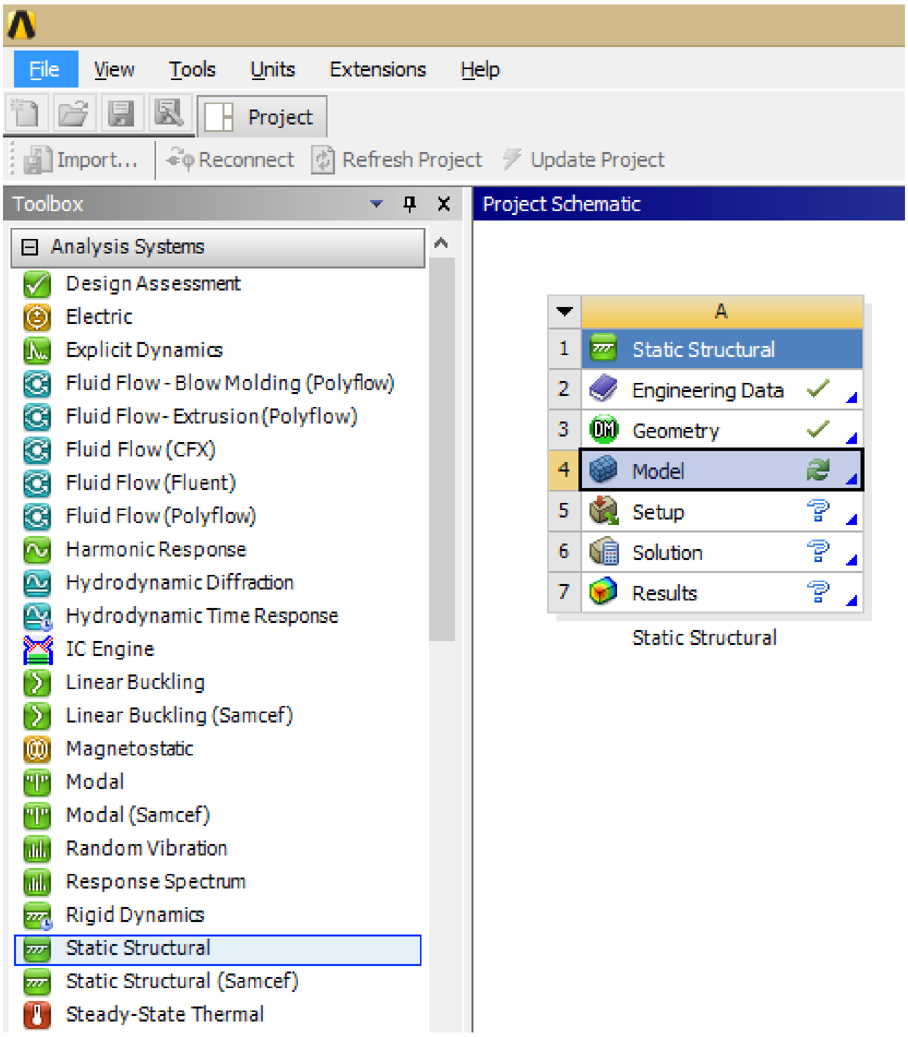
NOTE: Now the generated drawing is axis-symmetric, hence the meshed model of the quarter will be equal to the four times the actual model. Hence to reduce calculations and to speed the work, axis-symmetric solids are used.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

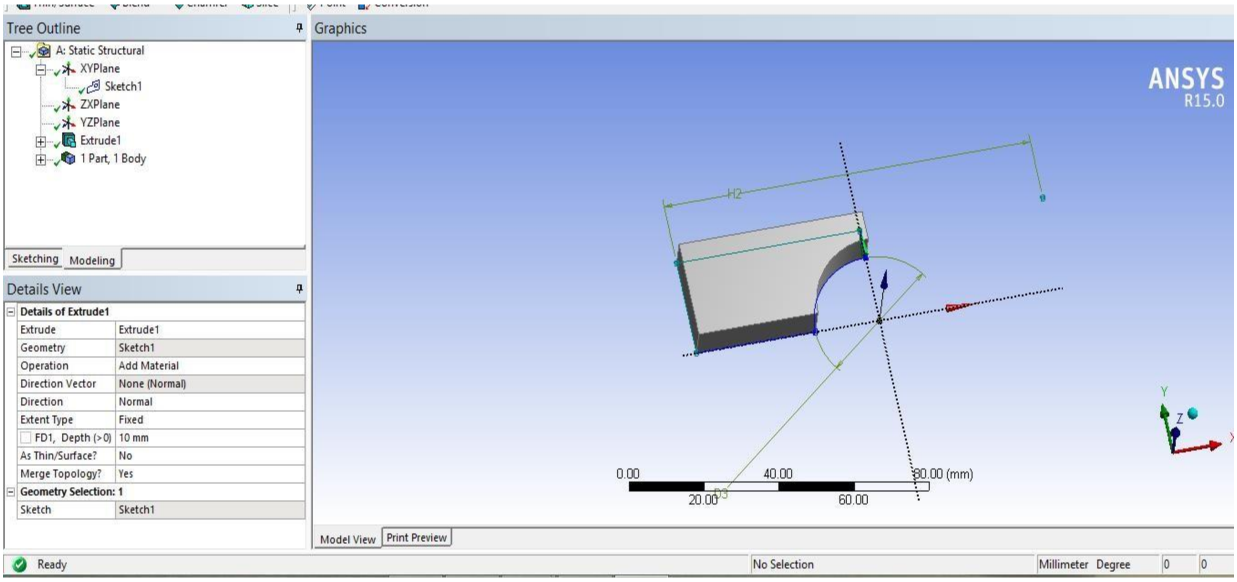


Remove any 3 quarters of the drawing, as it is axis-symmetric hence, will be similar to all the quarters.

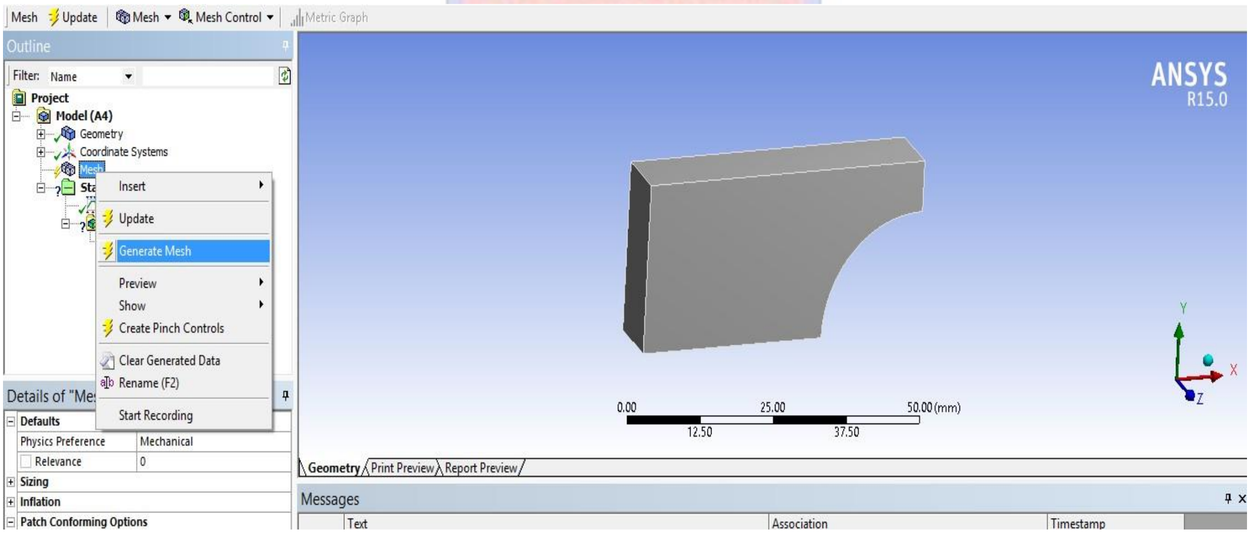
11. Extrude the model up to 10 mm.
12. Close the *Design Modeler* and start *Solver/MODEL*.

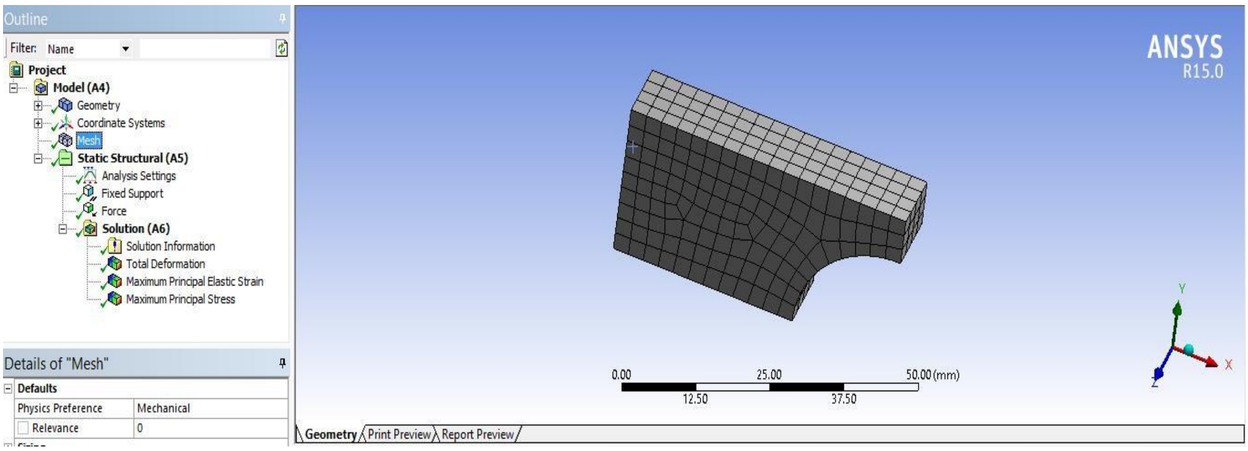


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

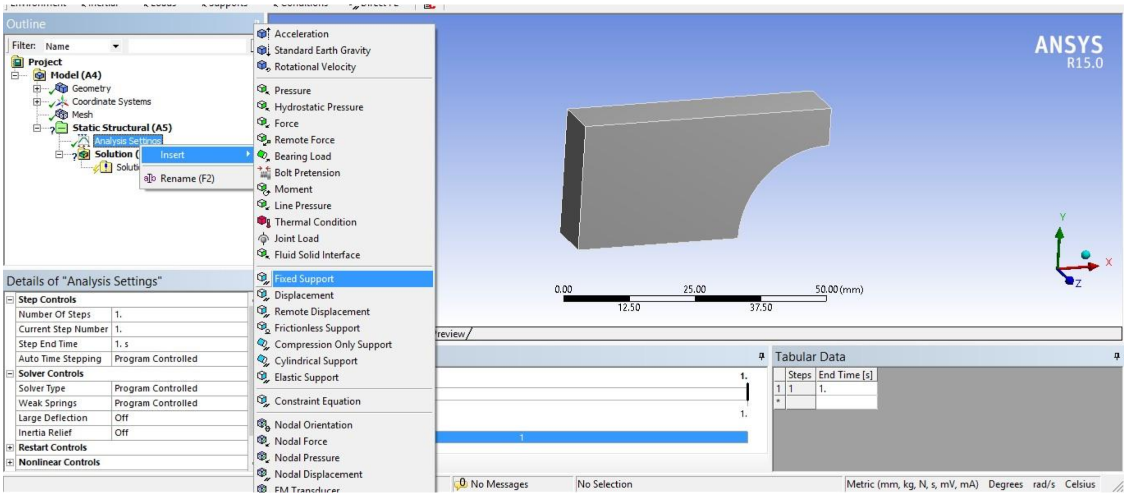


13. Right click on ***Mesh>Generate Mesh***
14.



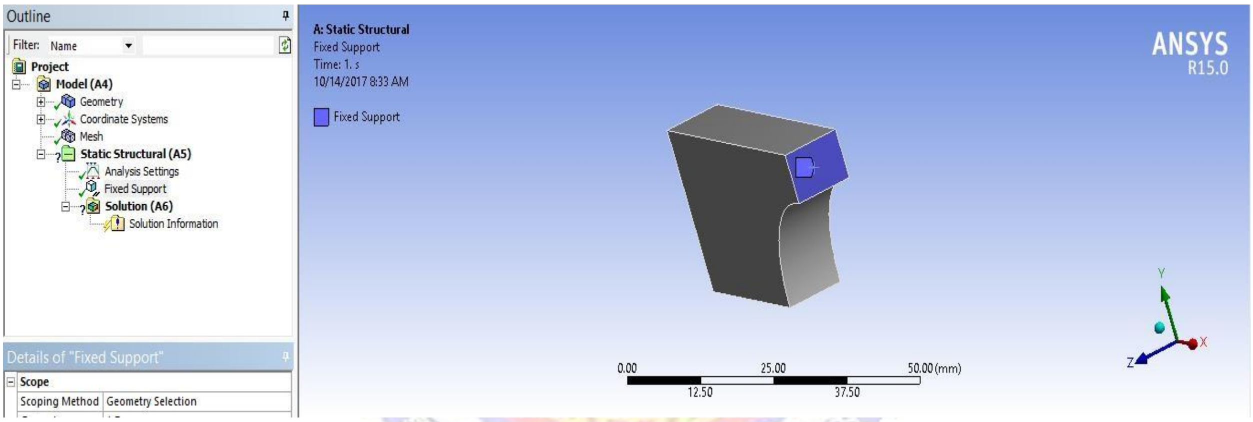


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



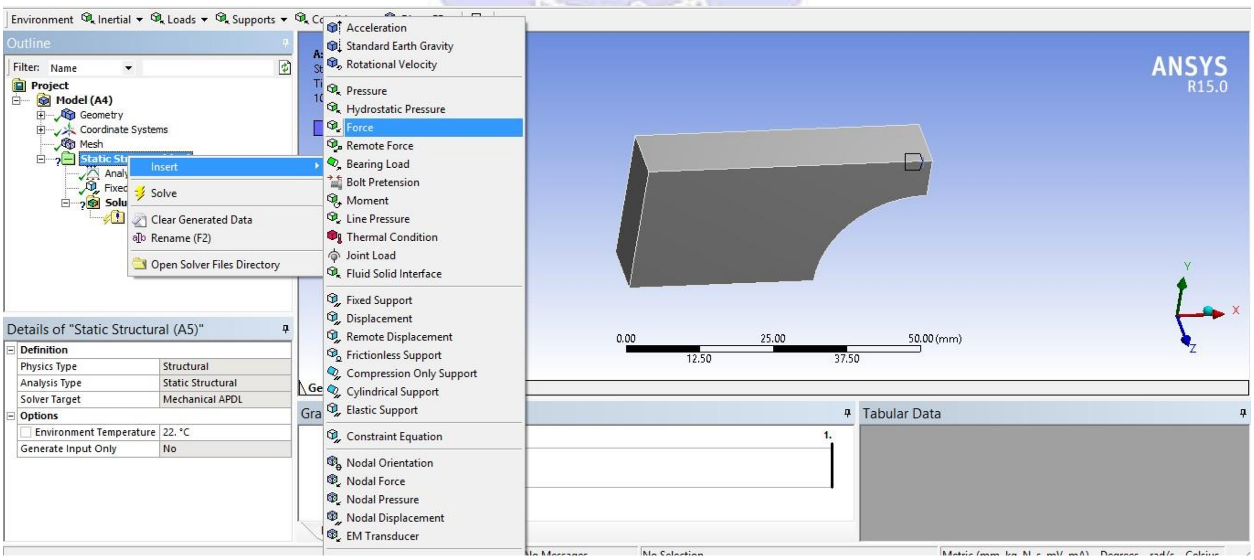
15. Select the Fixed Support from the ***Analysis Setting>Insert>Fixed Support***

16. Select the face as shown below, and click on apply in detail pane.

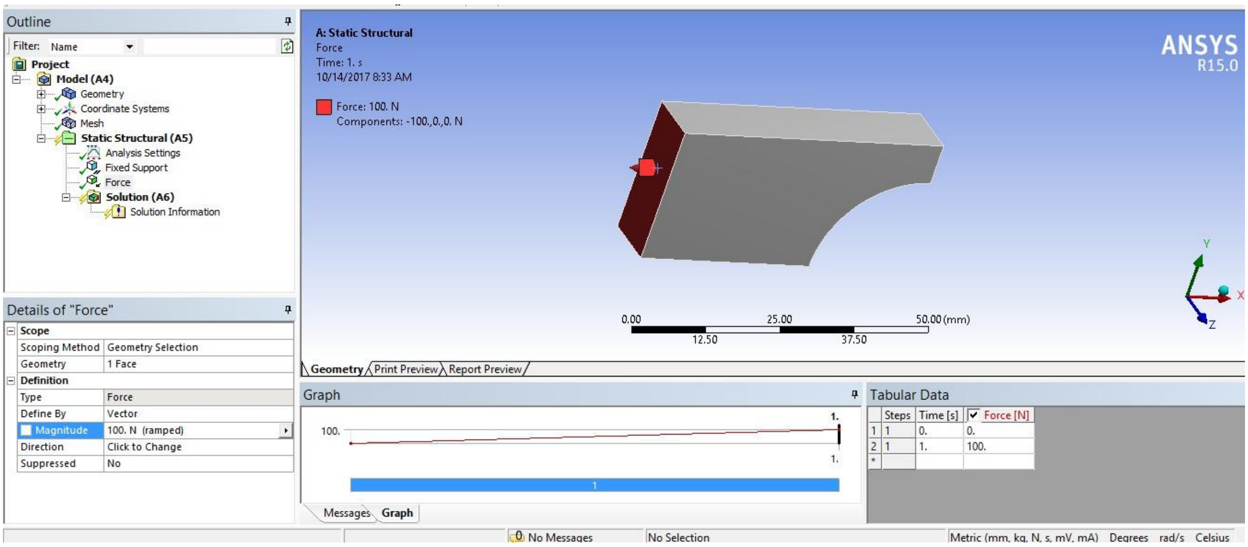


17. Select the Force from the ***Analysis Setting>Insert>Force***

18. Select the face as shown, the value of force as ***100N*** and the direction as shown.



Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



19. Select the ***Total Displacement, Maximum Principle Stress, and Maximum Principle Strain,*** and click on **SOLVE.**

SOLUTION:-

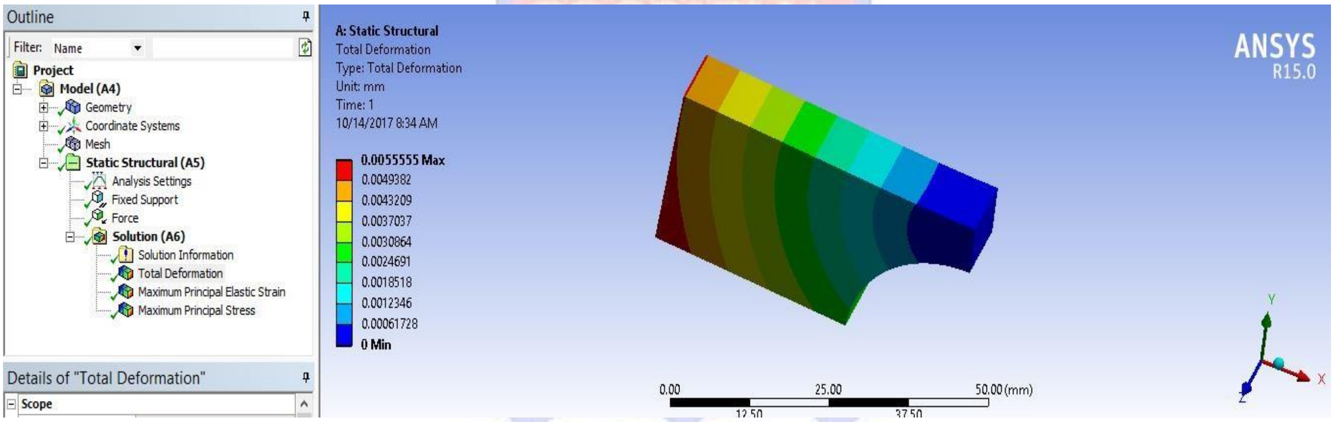


Figure 1: Value of Total Deformation =0.0055555 mm(max)

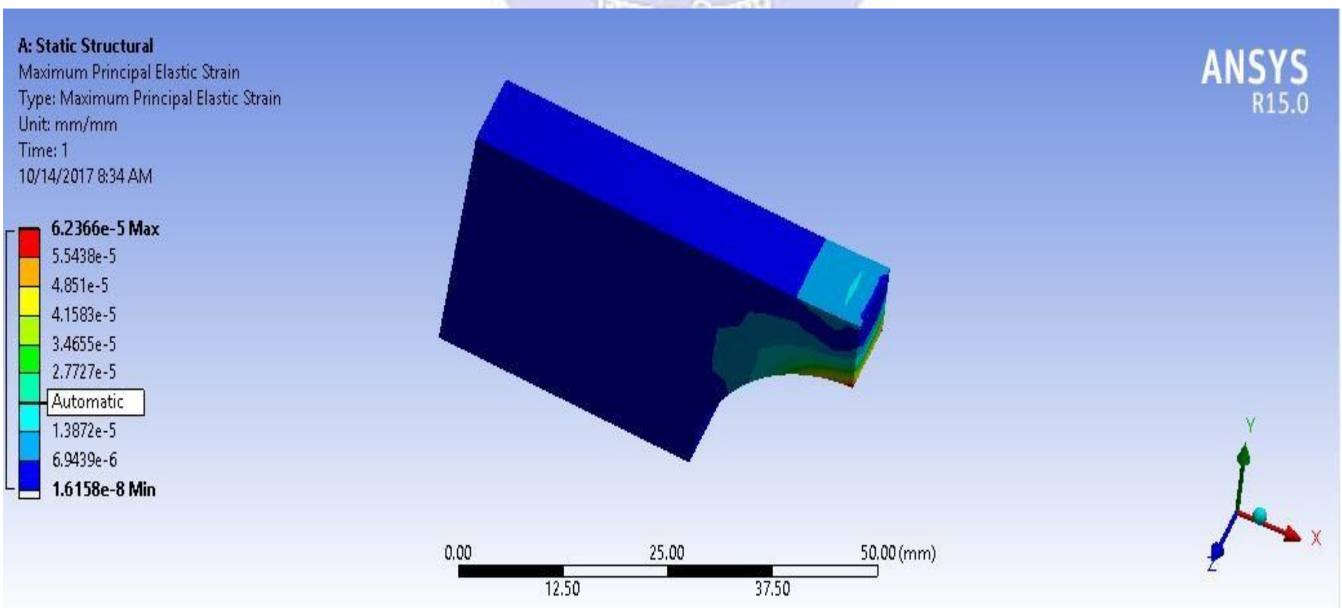


Figure 2: Value of Max. Principle Strain = 5.236 (max)

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

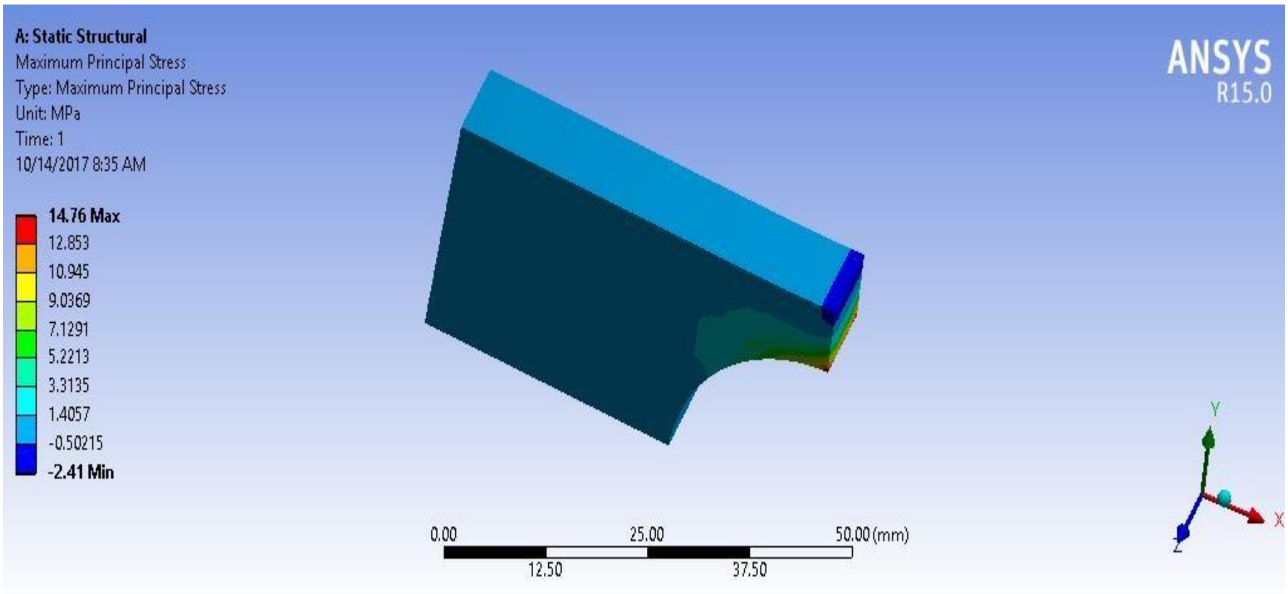


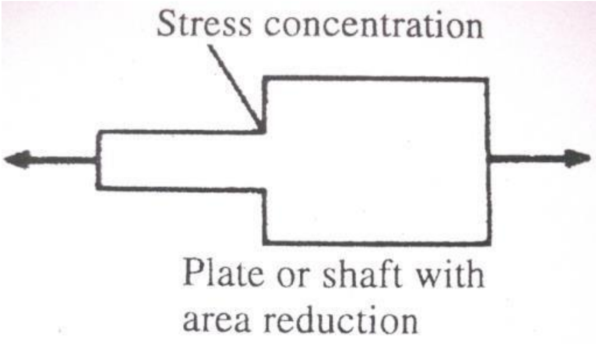
Figure 3: Value of Max. Principle Stress = 14.76 MPa

Viva Questions:-

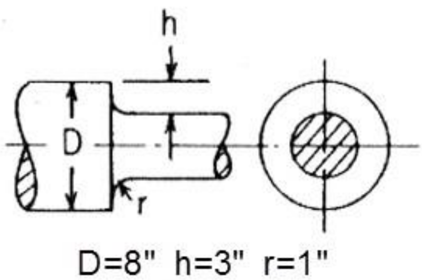
1. What is aspect ratio?
2. Differentiate between global and local axes.
3. What is the difference between linear and non-linear finite element analysis?
4. What is the difference between static and dynamic FEA?
5. What is the difference between implicit and explicit finite element analysis?
6. What is the purpose of meshing or dividing the component into finite elements?
7. When can we use symmetric boundary in the analysis?
8. Explain force method and stiffness method?
9. How do you compute strains and stresses in FEA?
10. What are the types of boundary conditions?

Experiment No.7 Analysis of A Stepped Shaft in Axial Tension

Problem Specification Consider a stepped shaft under an applied axial load, P. A stress concentration is apparent at the step where the cross-sectional area is discontinuous. The cross section is circular.



So the problem becomes amenable, let's consider a relatively small fillet placed at the step to reduce the stress concentration to a finite value.



The pressure load P/A on the smaller cross-section is 1000 psi.

Calculate the axial stress concentration factor and compare it to the formula

Go to Step 1: Pre-Analysis & Start-Up

Pre-Analysis & Start-Up

Pre-Analysis

It is recommended that you make some back-of-the-envelope estimates of expected results before launching into your computer solution. Here

$$h = 3\ in$$

$$r = 1\ in$$

$$D = 8\ in$$

$$\frac{h}{r} = 3$$

$$\frac{2h}{D} = \frac{3}{4} = 0.75$$

for which the following formula for the axial stress concentration factor, K, holds (*Roark's Formulas for Stress and Strain*, Warren C. Young and Richard G. Budynas, 2002):

$$K = C_1 + C_2\frac{2h}{D} + C_3\left(\frac{2h}{D}\right)^2 + C_4\left(\frac{2h}{D}\right)^3$$

$$C_1 = 1.225 + 0.831\sqrt{h/r} - 0.010\left(h/r\right) = 2.634$$

$$C_2 = -1.831 - 0.318\sqrt{h/r} - 0.049\left(h/r\right) = -2.529$$

$$C_3 = 2.236 - 0.5220\sqrt{h/r} + 0.176\left(h/r\right) = 1.8599$$

$$C_4 = -0.63 + 0.009\sqrt{h/r} - 0.117\left(h/r\right) = -0.965411543$$

$$\Rightarrow K = C_1 + C_2\frac{2h}{D} + C_3\left(\frac{2h}{D}\right)^2 + C_4\left(\frac{2h}{D}\right)^3 = 1.3766$$

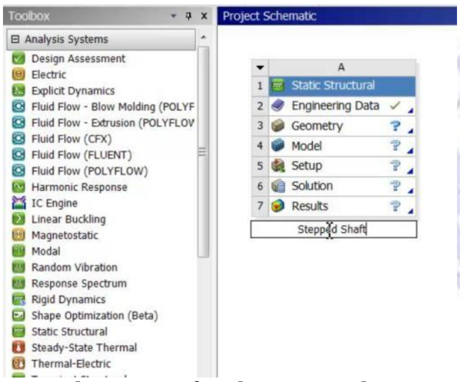
where

$$\sigma_{MAX} = K\sigma_{NOM} = K\frac{P}{A_{MIN}} = K\frac{4P}{\pi\left(D-2h\right)^2} = 1376\ psi$$

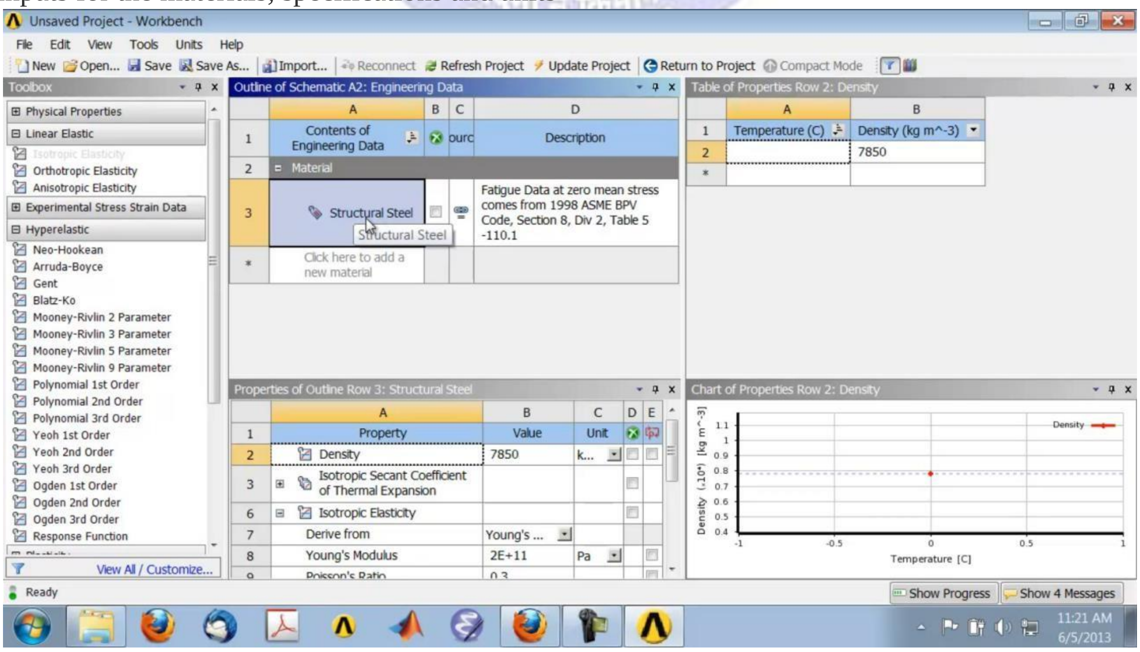
We'll compare the above axial stress concentration factor to the value obtained from ANSYS.

Start-Up

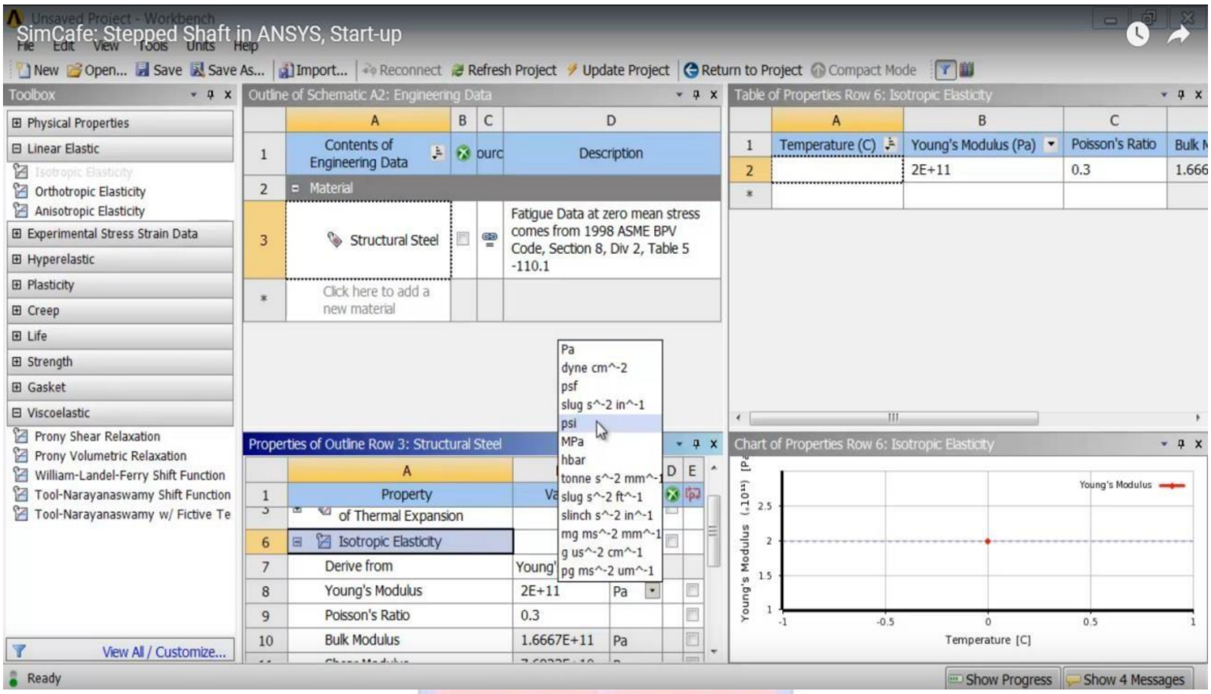
Launch ANSYS Workbench and start a "Static Structural" analysis



Give the inputs for the materials, specifications and units

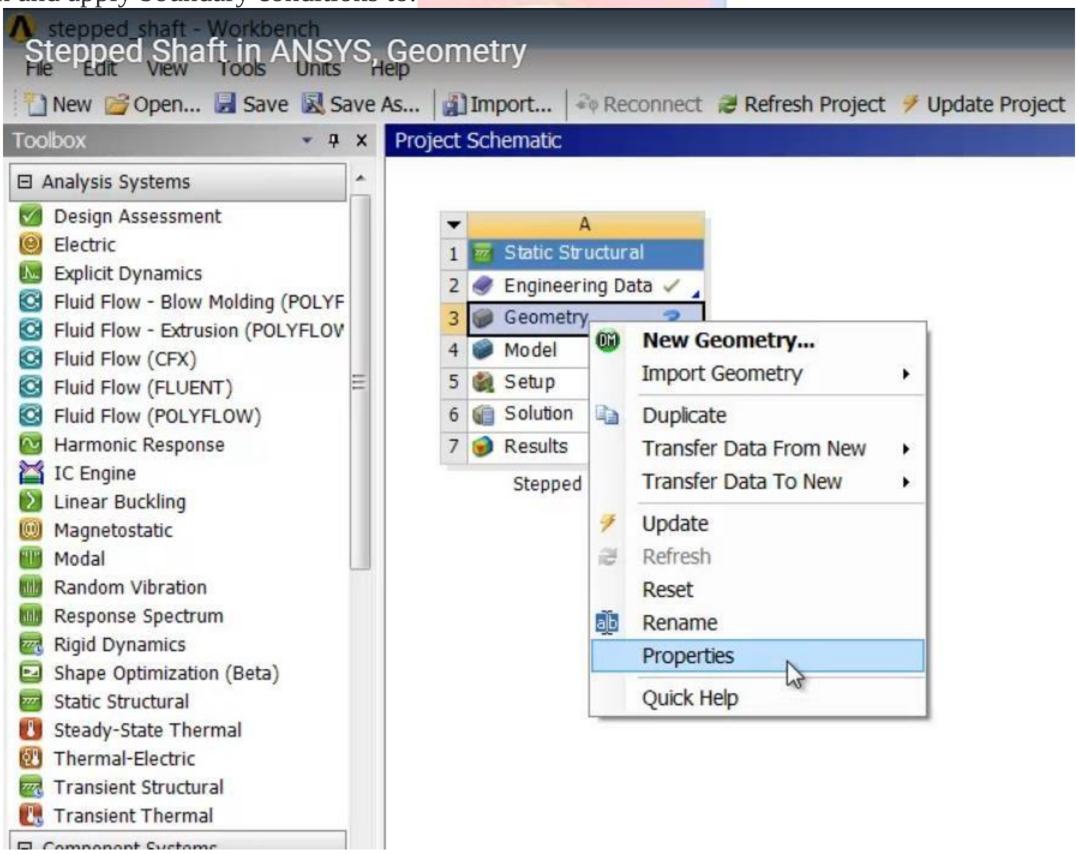


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

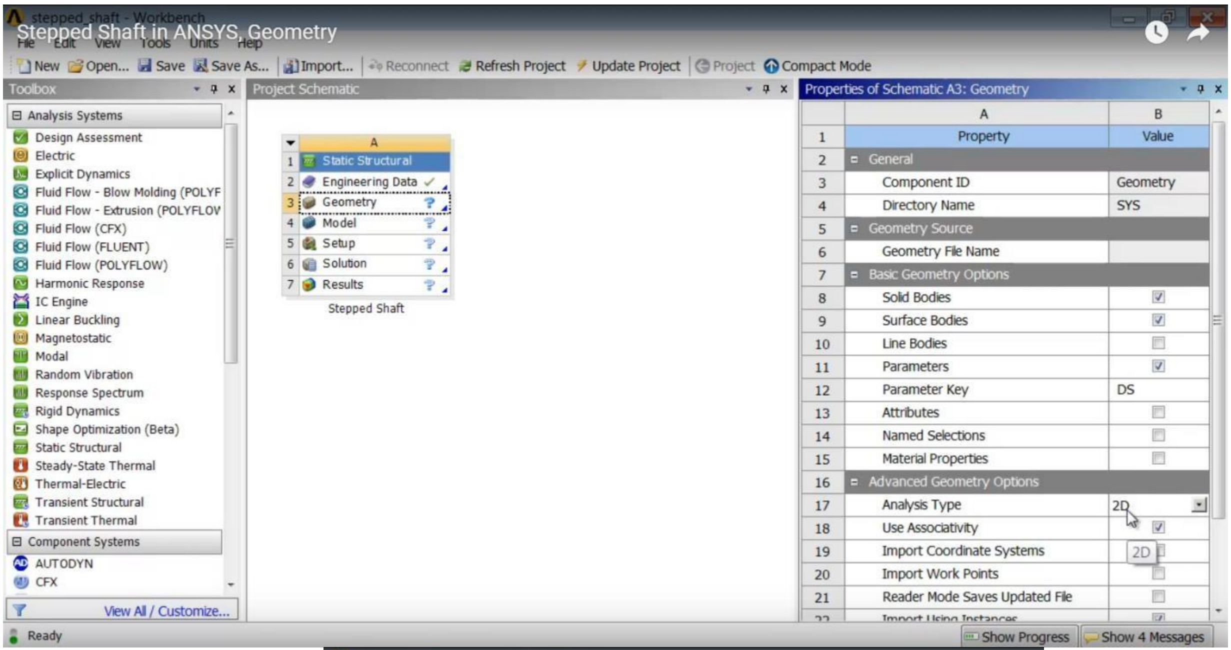


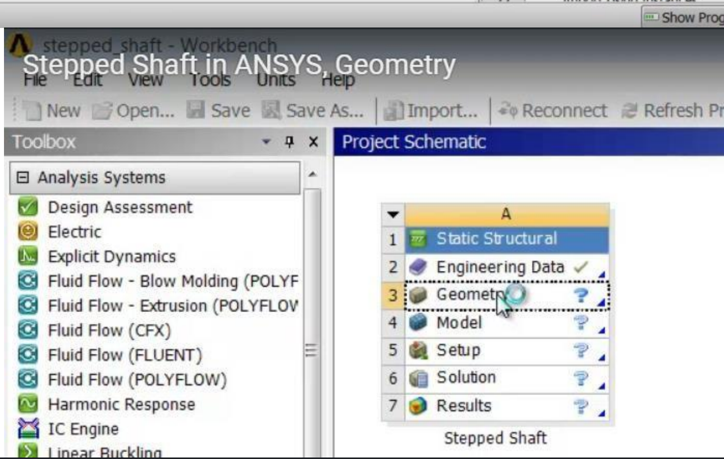
Go to Step 2: Geometry

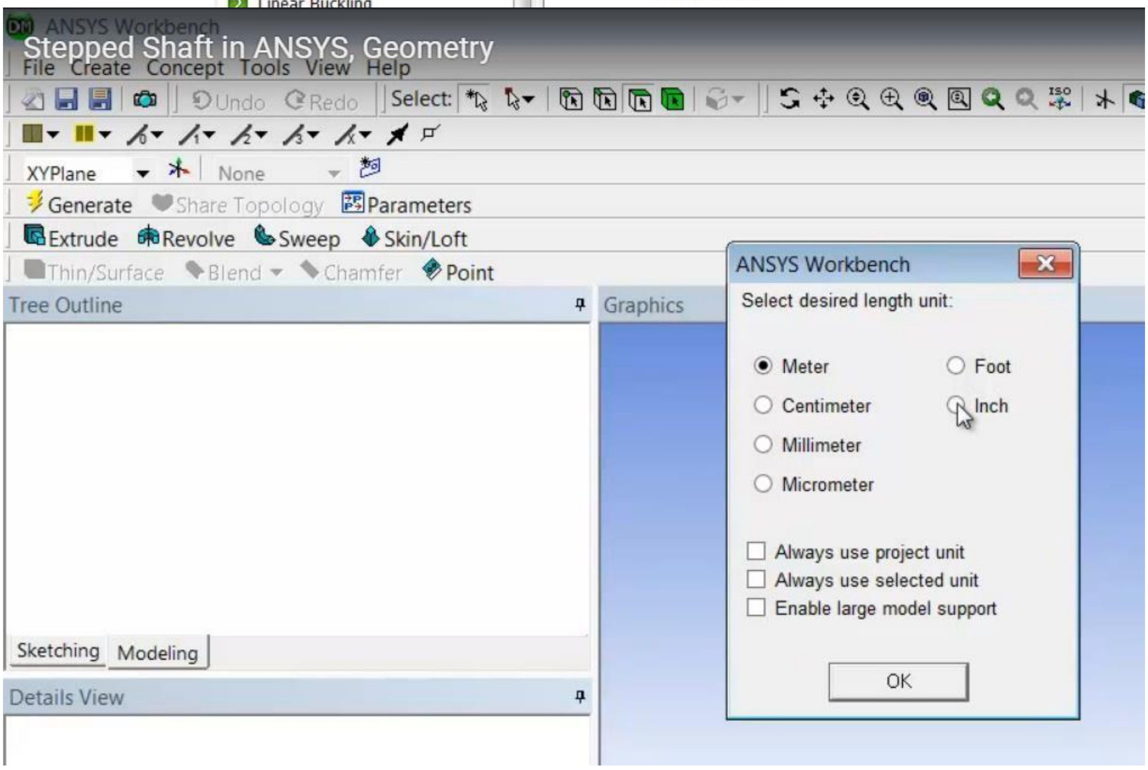
We'll first create a sketch and then a "surface body" from the sketch. The "surface body" is nothing but an area that we can mesh and apply boundary conditions to.



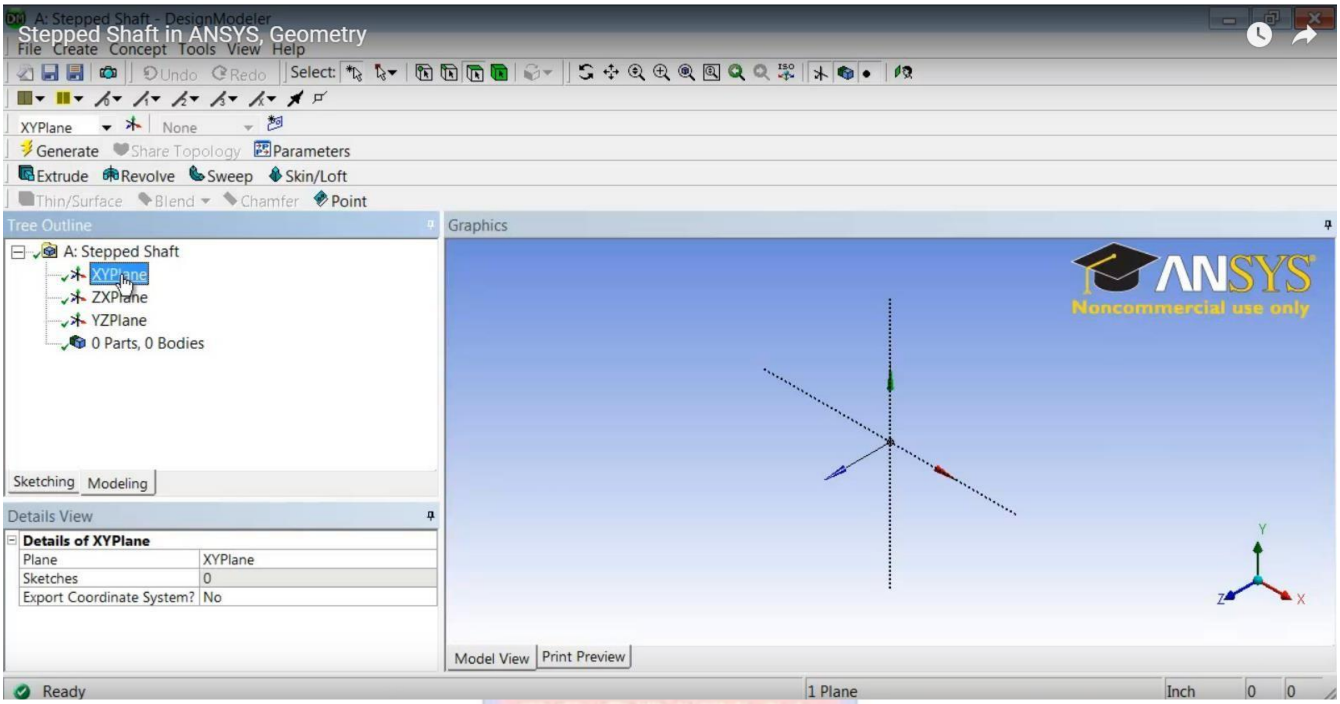
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

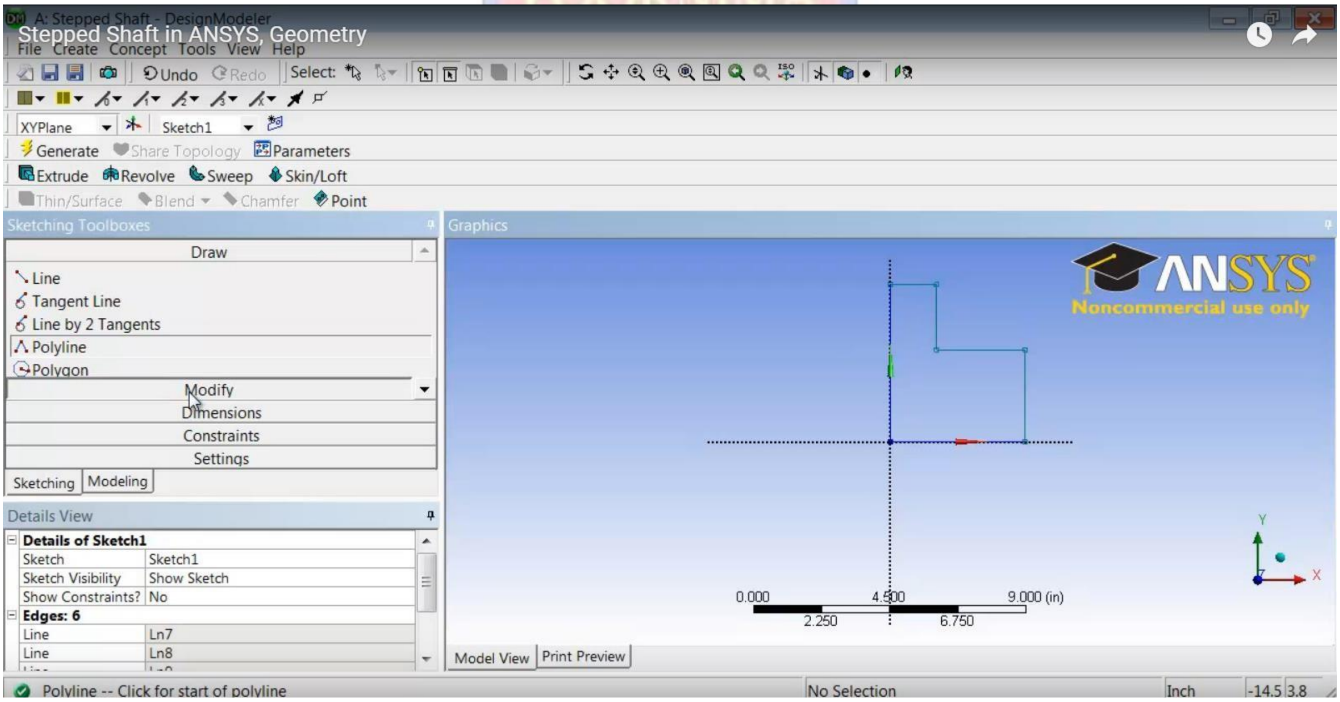




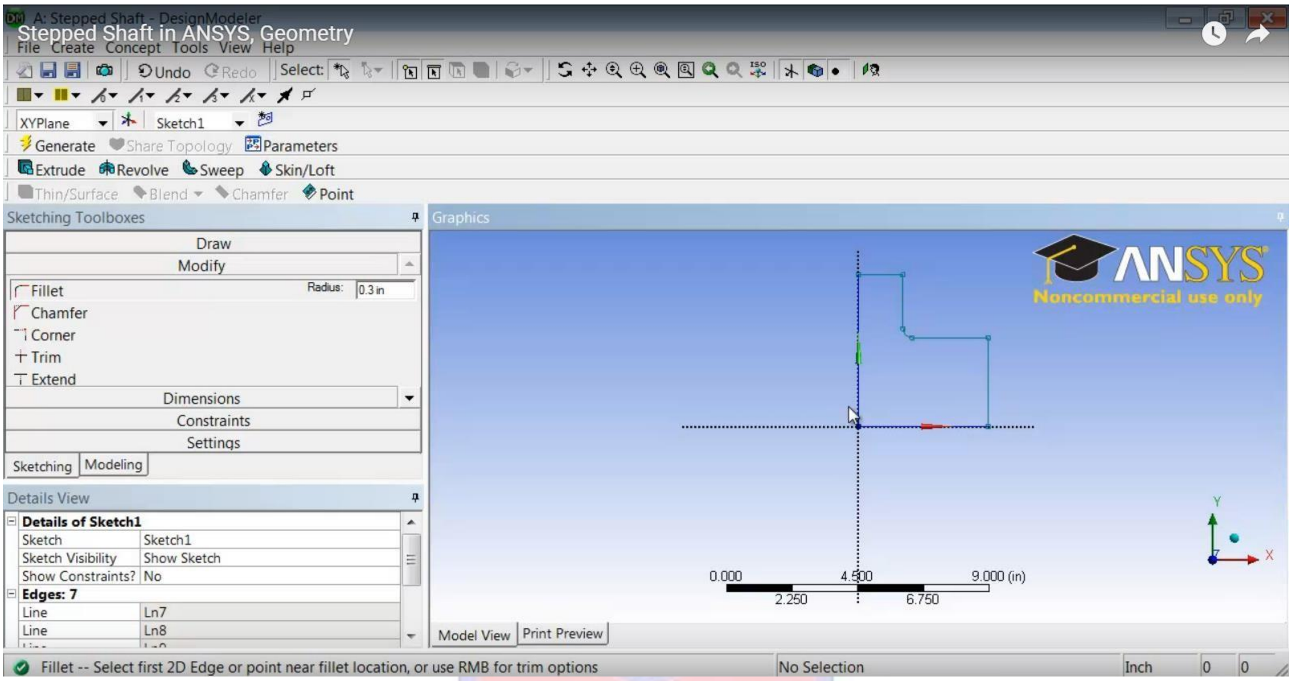


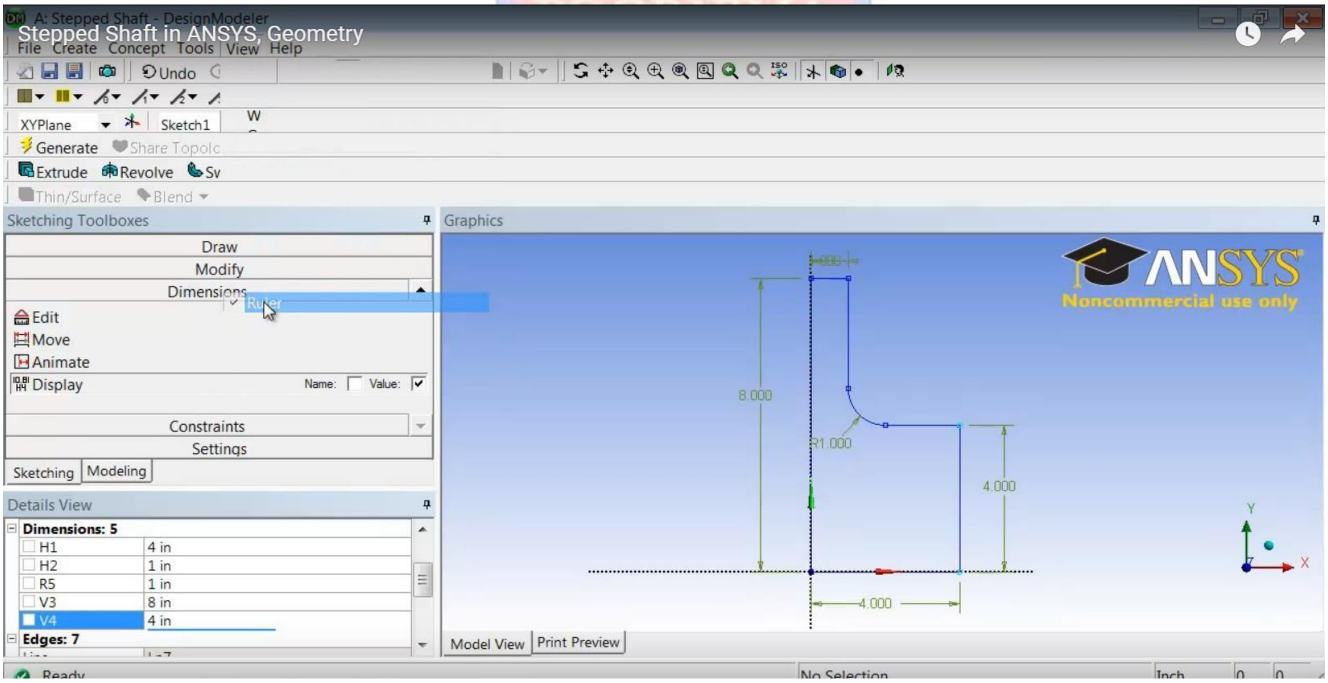
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

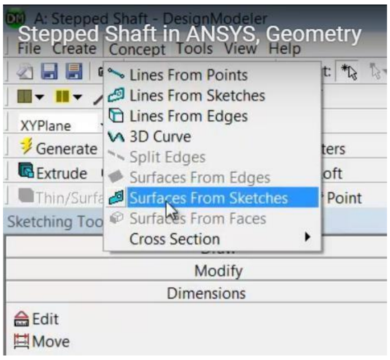




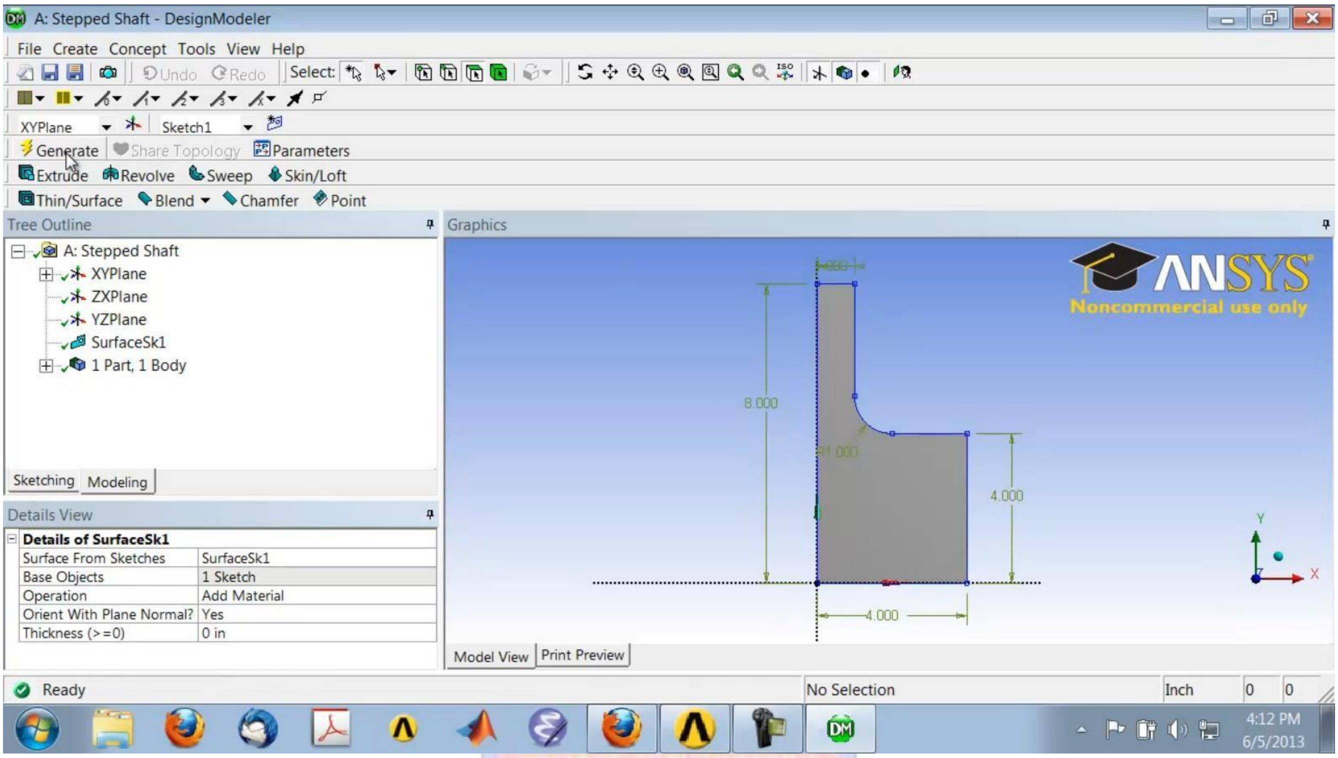
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR





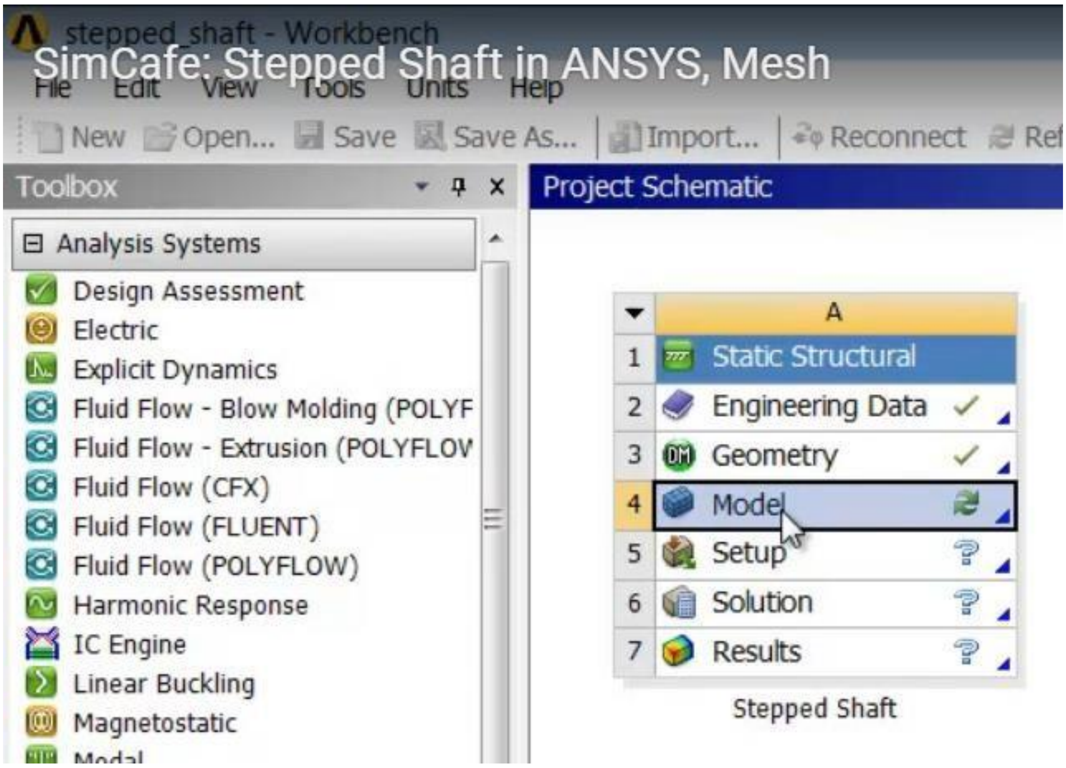


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

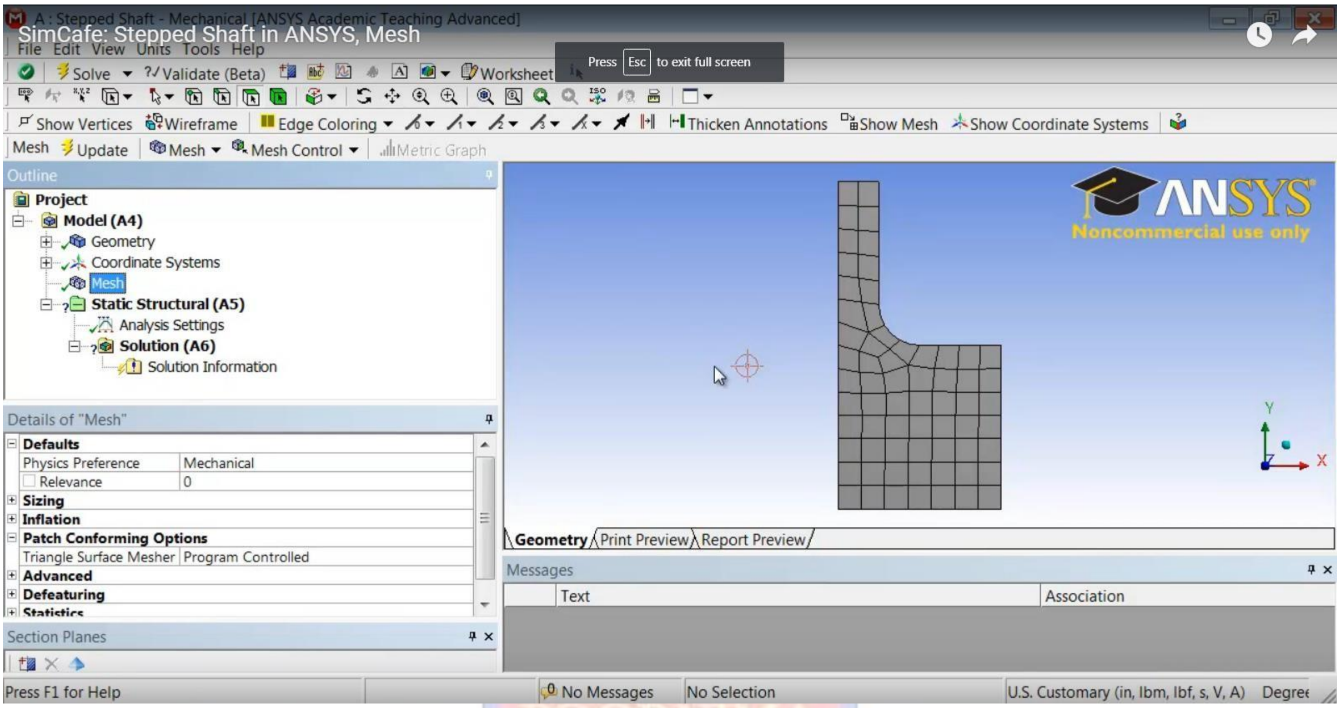


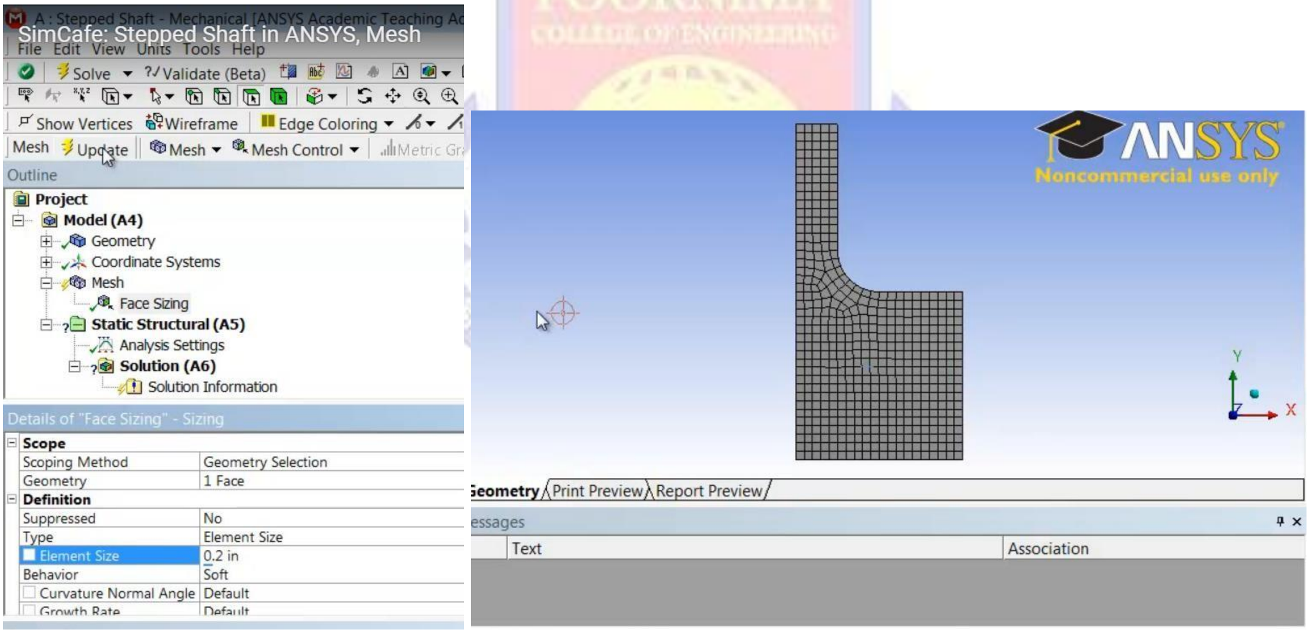
Mesh

The following video shows how to create a mesh for the 2D geometry. We stick with the default "Q8" elements (eight nodes per element including mid-side nodes). The element edge sizes are controlled by inserting a "face sizing".



Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

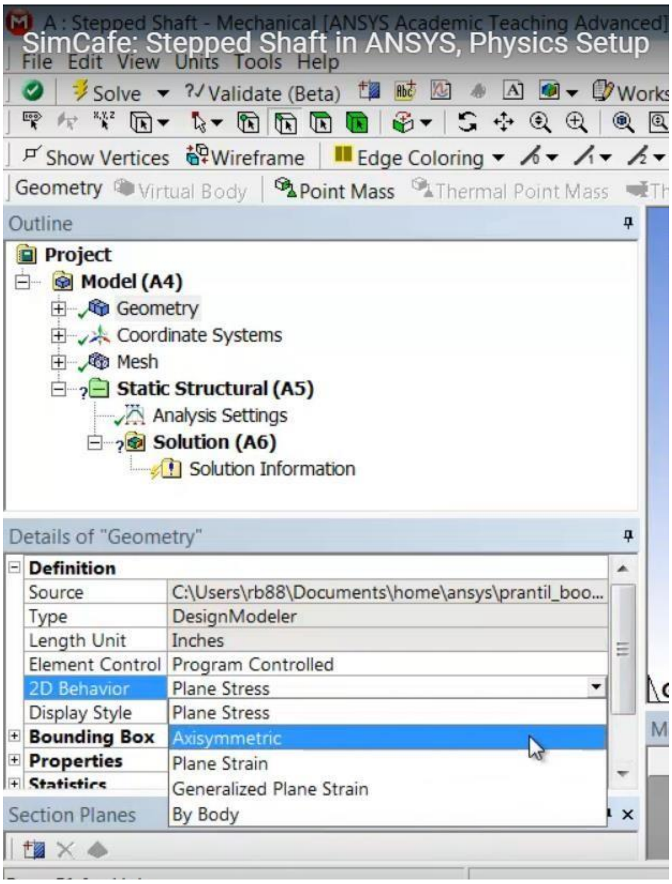


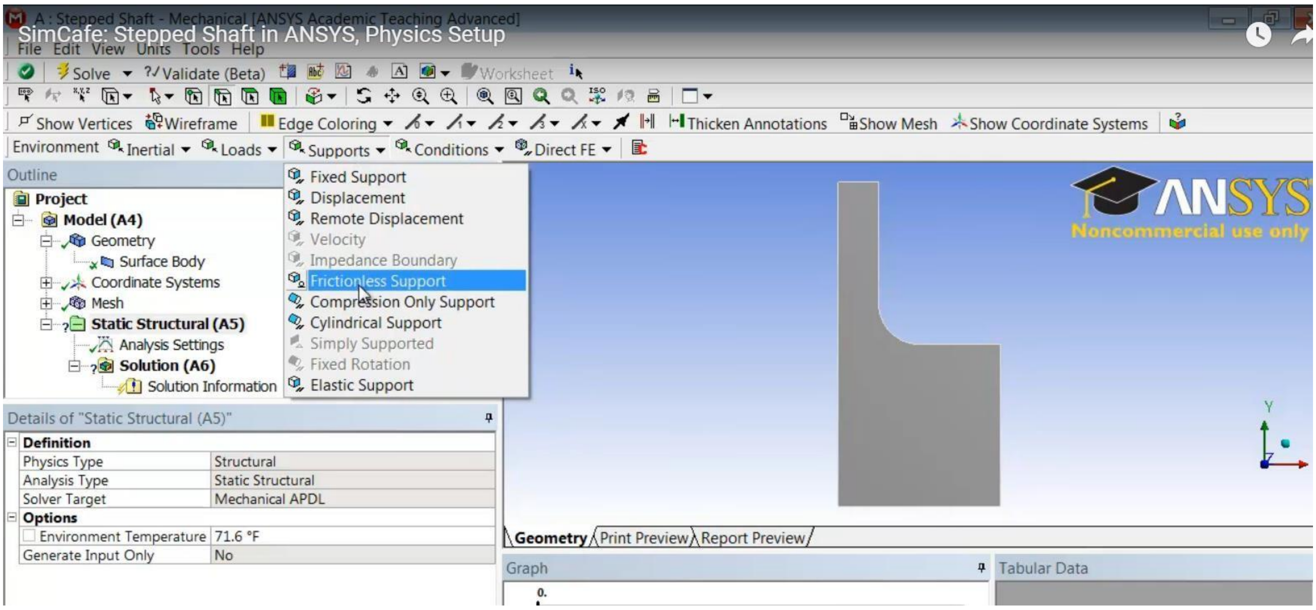


Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

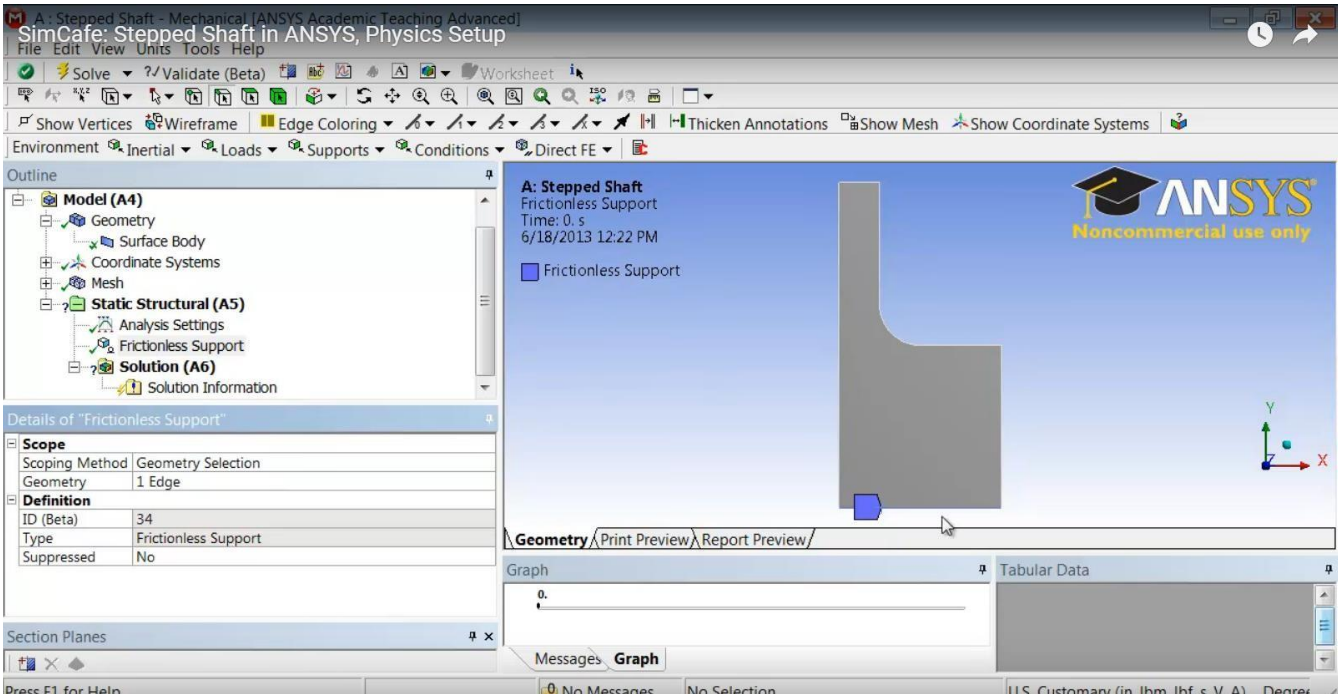
Physics Setup

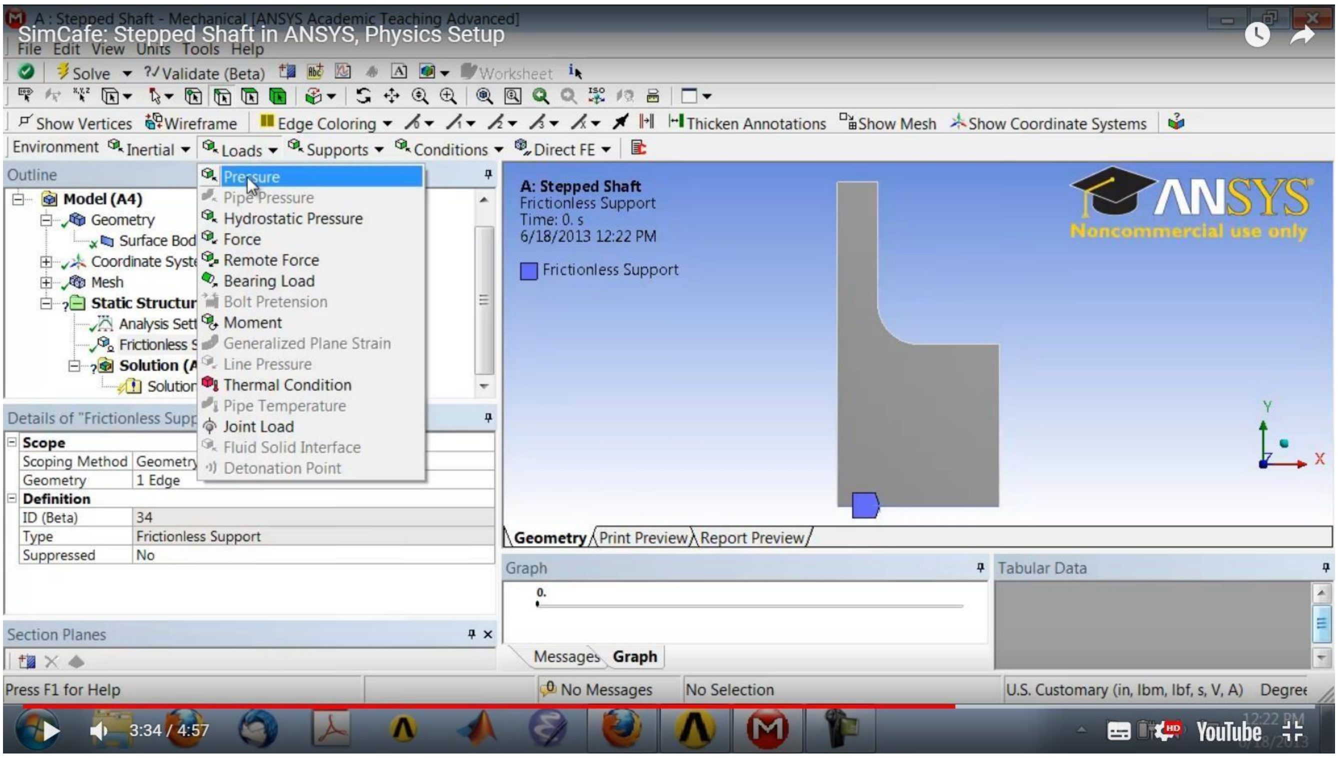
Specify the physics of the problem: axisymmetric approximation, material properties (Young's modulus and Poisson ratio) and boundary conditions.



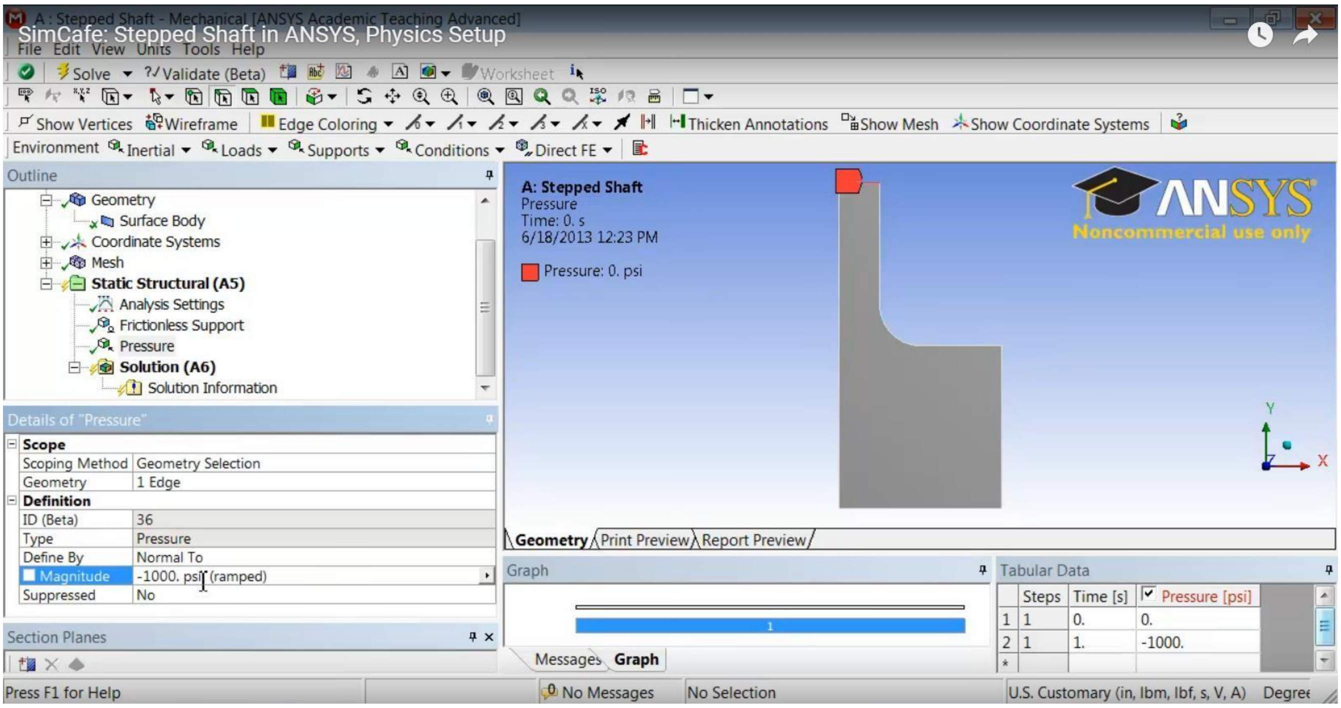


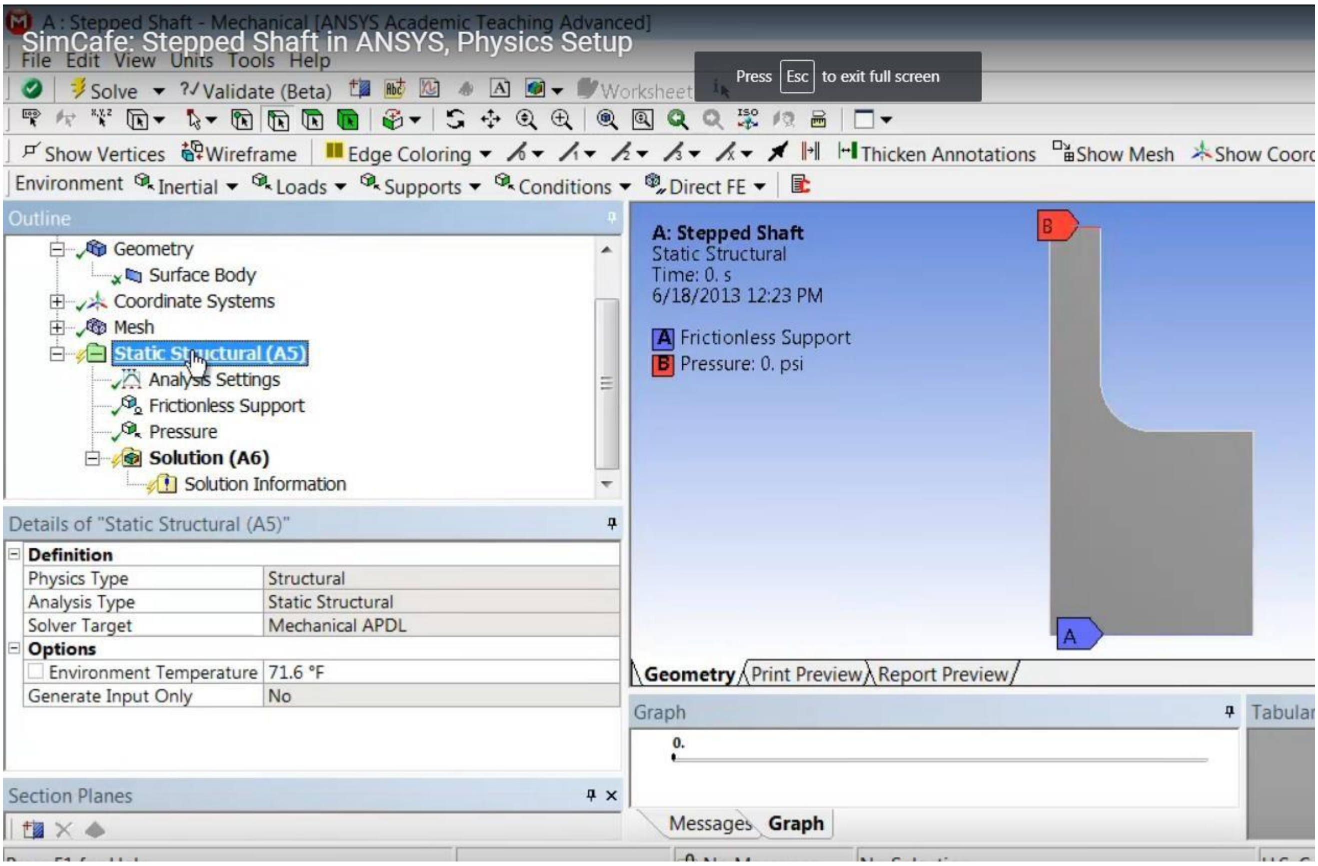
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR





Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



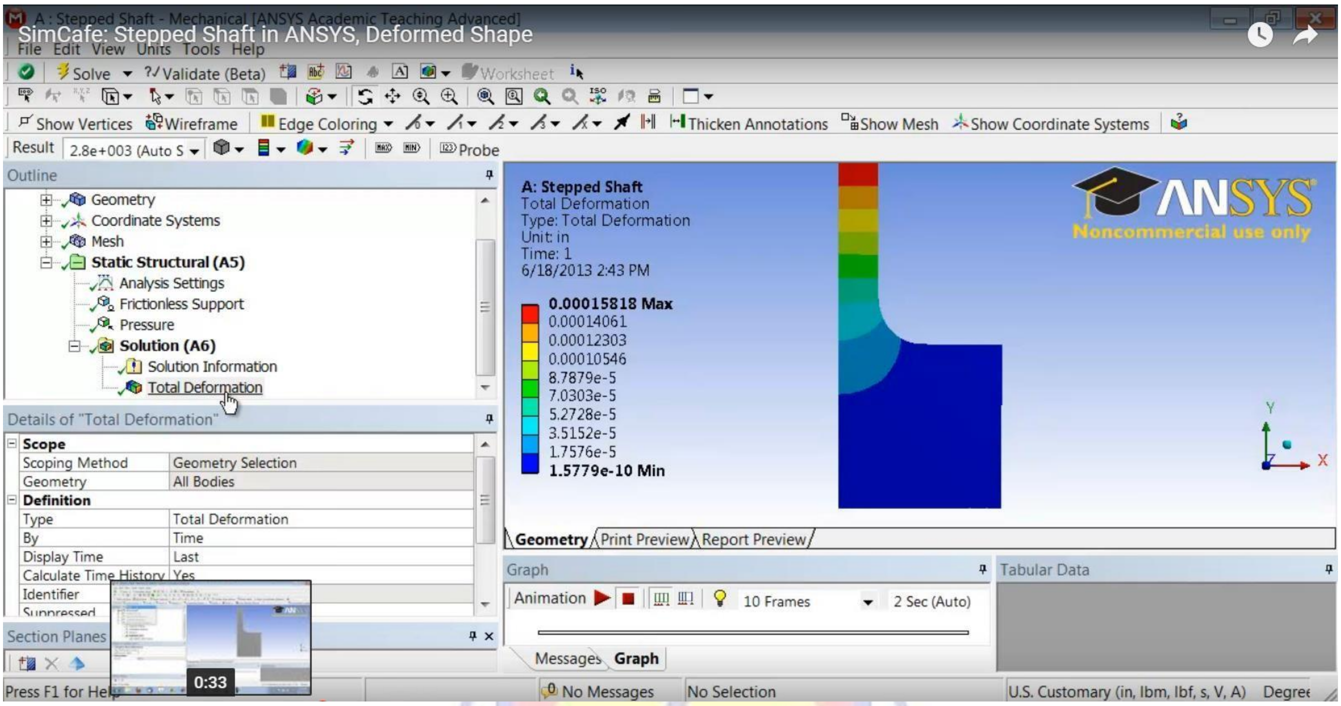


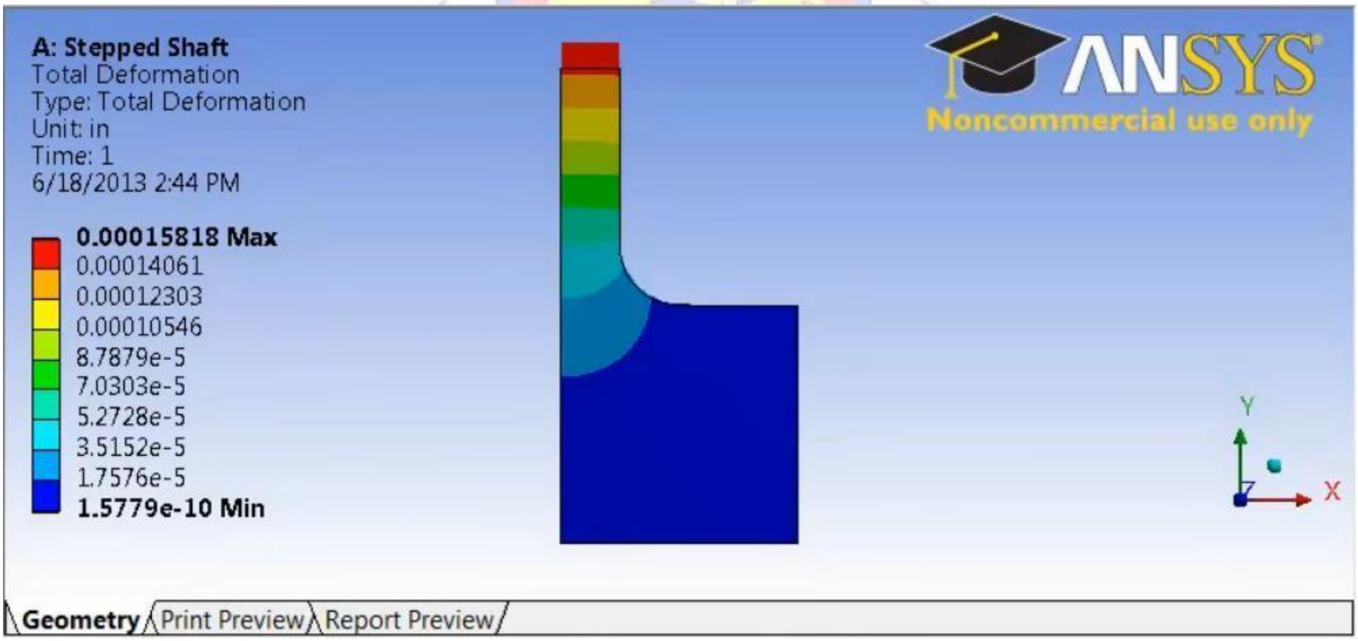
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Numerical Results

Plot the deformed shape and use it to check if the boundary conditions (displacement constraints and point load) have been applied correctly.

1. Total Deformation

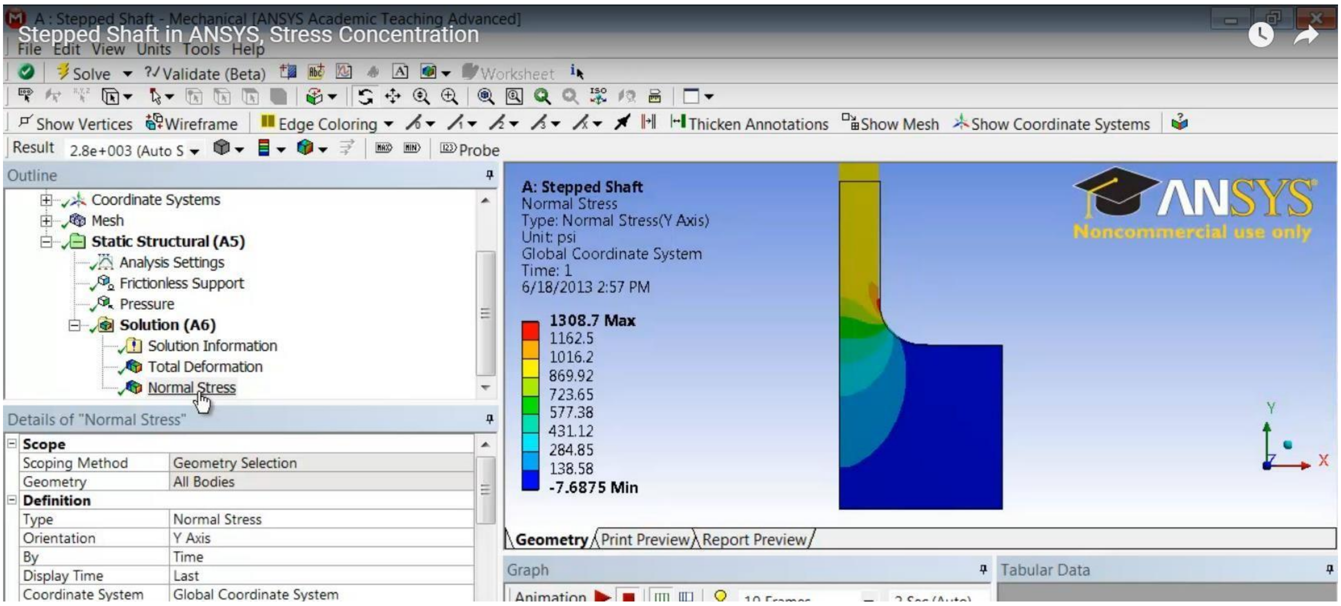


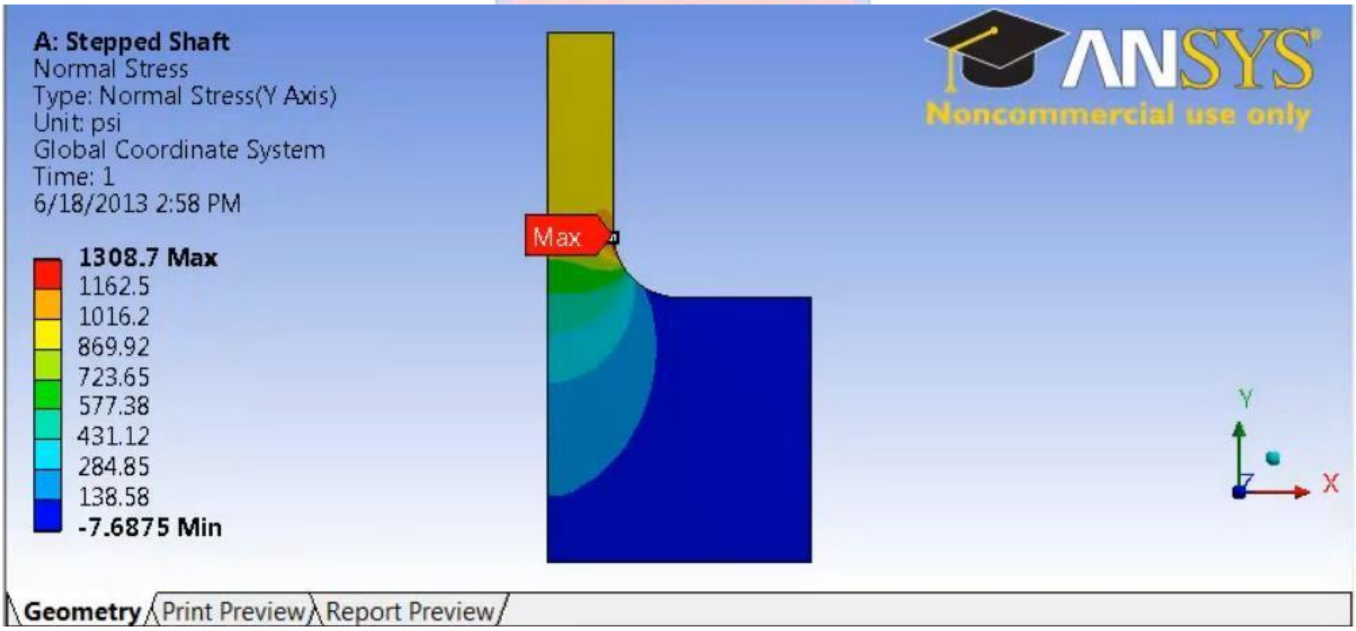


Plot the normal stress in the axial direction. The stress concentration factor can be deduced from this plot.

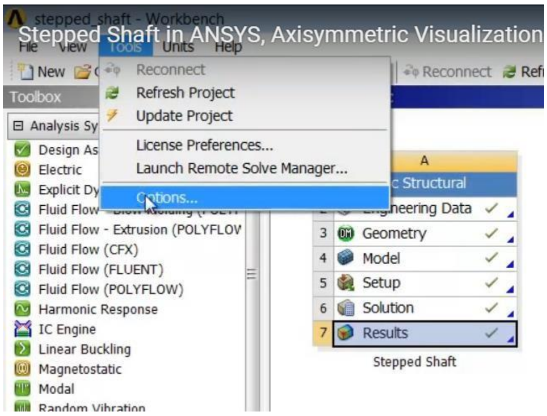
Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

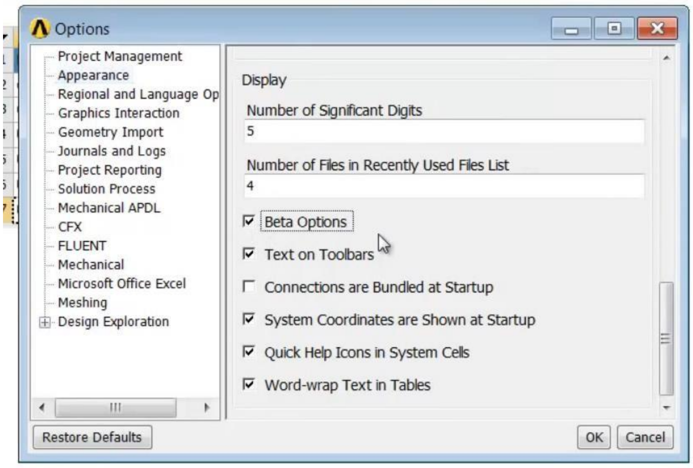
1. Normal Stress in Y-direction

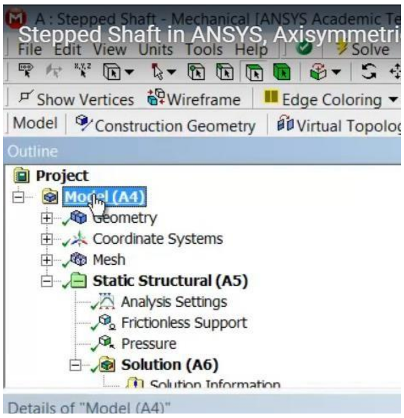


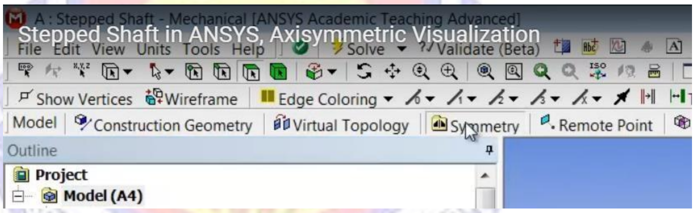


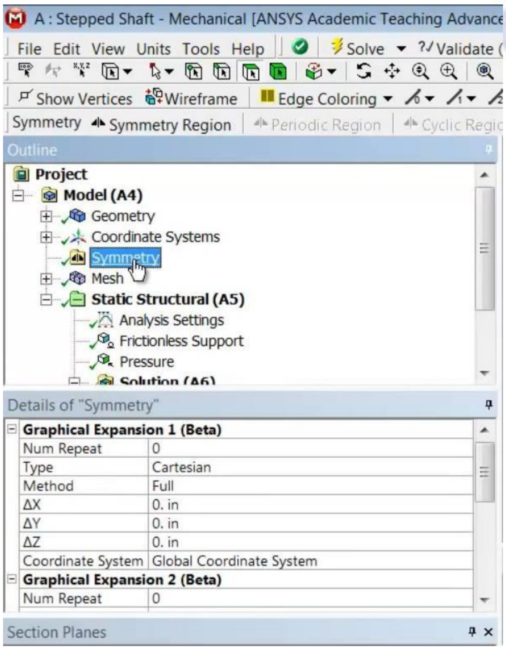
Visualize the full 3D shape by revolving the 2D (axisymmetric) geometry about the axis.

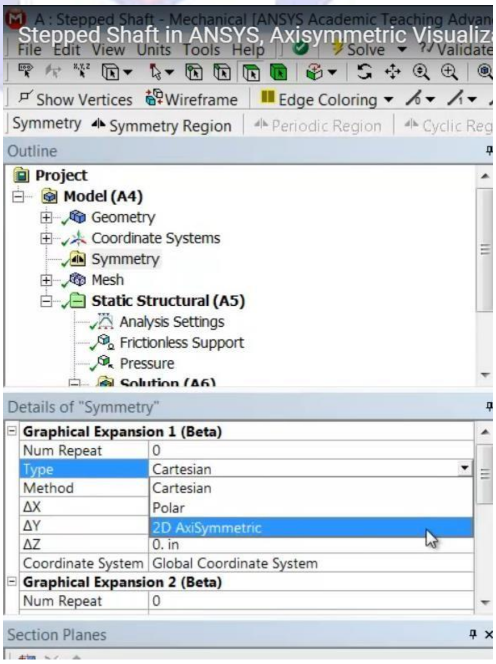




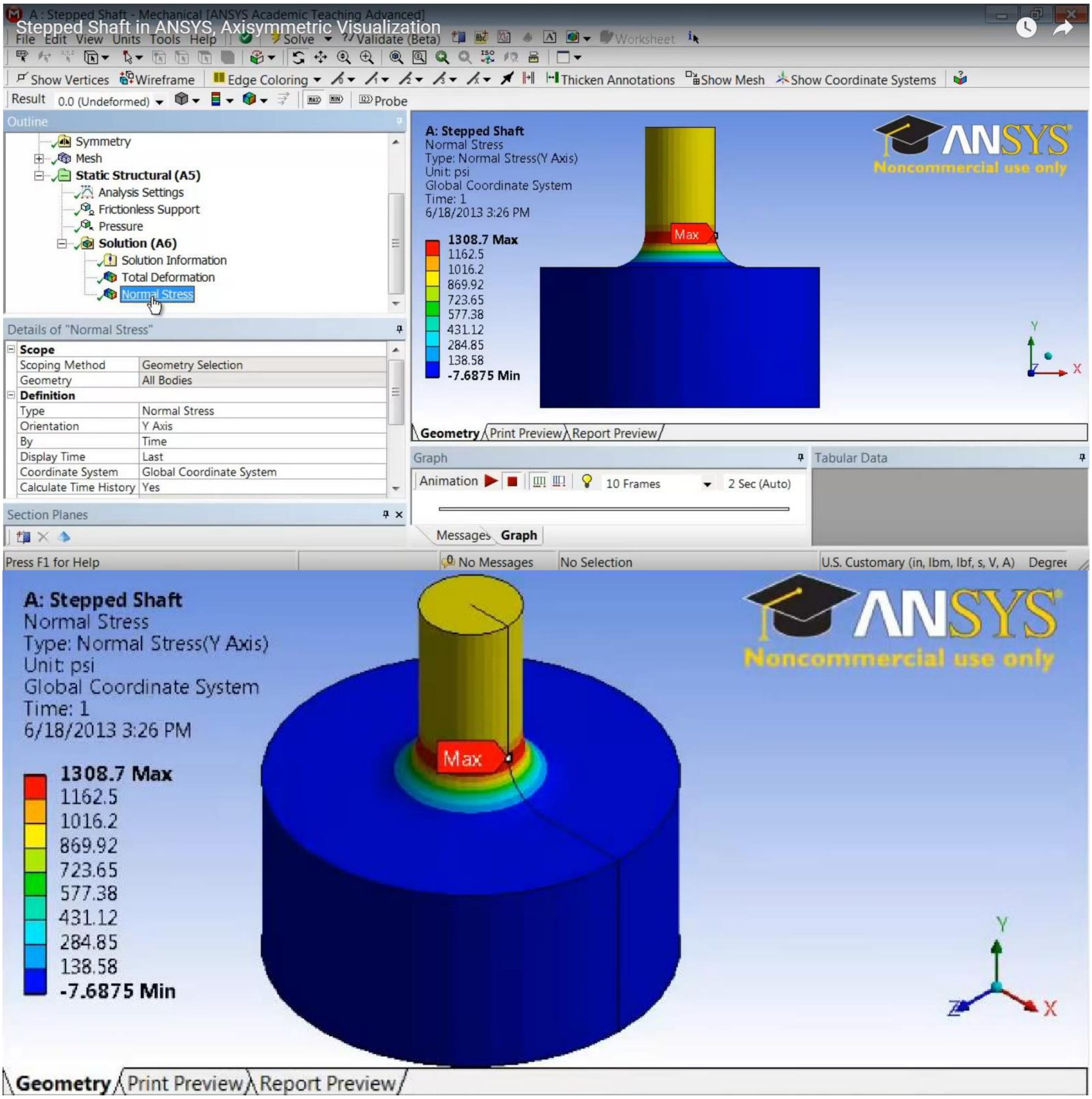








Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR



Axial Stress Concentration Factor

In the table below, the **axial stress concentration factors** on the original and refined meshes are compared with the hand calculation from the Pre-Analysis step. Recall that the hand calculation used a formula from *Roark's Formulas for Stress and Strain*.

ANSYS, Original mesh	ANSYS, Refined mesh	Hand calculation
1.309	1.316	1.377

There is only a slight change on refining the mesh. The stress concentration factor on either mesh is accurate to within the level of accuracy of the cited formula, i.e. 5%. This increases our confidence in the ANSYS results.

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Viva Questions:-

1. What are the different menus in ANSYS?
2. Give examples for the finite element.
3. What is the history of the FEM?
4. What are the three phases of finite element method?
5. What is discretization?
6. Explain the principle of virtual displacement.
7. Mention some advantages of FEA over solid mechanics.
8. What are the different types of errors in FEA?
9. What is the difference between static and dynamic FEA?
10. What is the difference between implicit and explicit finite element analysis?

Experiment no. 8- Introduction to MATLAB

Objective: To introduce the basic information on MATLAB

What Is MATLAB?

MATLAB is a high-performance language for technical computing. It integrates computation, visualization, and programming in an easy-to-use environment where problems and solutions are expressed in familiar mathematical notation. Typical uses include:

- Math and computation
- Algorithm development
- Modeling, simulation, and prototyping
- Data analysis, exploration, and visualization
- Scientific and engineering graphics
- Application development, including Graphical User Interface building

MATLAB is an interactive system whose basic data element is an array that does not require dimensioning. This allows you to solve many technical computing problems, especially those with matrix and vector formulations, in a fraction of the time it would take to write a program in a scalar non interactive language such as C or Fortran.

The name MATLAB stands for matrix laboratory. MATLAB was originally written to provide easy access to matrix software developed by the LINPACK and EISPACK projects, which together represent the state-of-the-art in software for matrix computation.

MATLAB has evolved over a period of years with input from many users. In university environments, it is the standard instructional tool for introductory and advanced courses in mathematics, engineering, and science. In industry, MATLAB is the tool of choice for high-productivity research, development, and analysis.

MATLAB features a family of application-specific solutions called toolboxes. Very important to most users of MATLAB, toolboxes allow you to *learn* and *apply* specialized technology. Toolboxes are comprehensive collections of MATLAB functions (M-files) that extend the MATLAB environment to solve particular classes of problems. Areas in which toolboxes are available include signal processing, control systems, neural networks, fuzzy logic, wavelets, simulation, and many others.

MATLAB is a fourth-generation programming language and numerical analysis environment. Uses for MATLAB include matrix calculations, developing and running algorithms, creating user interfaces (UI) and data visualization. The multi-paradigm numerical computing environment allows developers to interface with programs developed in different languages, which makes it possible to harness the unique strengths of each language for various purposes.

MATLAB is used by engineers and scientists in many fields such as image and signal processing, communications, control systems for industry, smart grid design, robotics as well as computational finance.

The MATLAB System:- MATLAB system consists of five main parts:

The MATLAB language.
This is a high-level matrix/array language with control flow statements, functions, data structures, input/output, and object-oriented programming features. It allows both "programming in the small" to rapidly create quick and dirty throw-away programs, and "programming in the large" to create complete large and complex application programs.

The MATLAB working environment.
This is the set of tools and facilities that you work with as the MATLAB user or programmer. It includes facilities for managing the variables in your workspace and importing and exporting data. It also includes tools for developing, managing, debugging, and profiling M-files, MATLAB's applications.

Handle Graphics.
This is the MATLAB graphics system. It includes high-level commands for two-dimensional and three-dimensional data visualization, image processing, animation, and presentation graphics. It also includes low-level commands that allow you to fully customize the appearance of graphics as well as to build complete Graphical User Interfaces on your MATLAB applications.

The MATLAB mathematical function library.
This is a vast collection of computational algorithms ranging from elementary functions like sum, sine, cosine, and complex arithmetic, to more sophisticated functions like matrix inverse, matrix eigenvalues, Bessel functions, and fast Fourier transforms.

The MATLAB Application Program Interface (API).
This is a library that allows you to write C and Fortran programs that interact with MATLAB. It include facilities for calling routines from MATLAB (dynamic linking), calling MATLAB as a computational engine, and for reading and writing MAT-files.

How to start MATLAB
Mac: Double-click on the icon for MATLAB.
PC: Choose the submenu "Programs" from the "Start" menu. From the "Programs" menu, open the "MATLAB" submenu. From the "MATLAB" submenu, choose "MATLAB".
Unix: At the prompt, type matlab.

You can quit MATLAB by typing exit in the command window.

The MATLAB environment

Note: From now on an instruction to press a certain key will be denoted by < >, e.g., pressing the enter key will be denoted as <enter>. Commands that should be typed at the prompt, will be written in courier font.

The MATLAB environment (on most computer systems) consists of menus, buttons and a writing area similar to an ordinary word processor. There are plenty of help functions that you are encouraged to use. The writing area that you will see when you start MATLAB, is called the *command window*. In this window you give the commands to MATLAB. For example, when you want to run a program you have written for MATLAB you start the program in the command window by typing its name at the prompt. The command window is also useful if you just want to use MATLAB as a scientific calculator or as a graphing tool. If you write longer programs, you will find it more convenient to write the program code in a separate window, and then run it in the command window.

In the command window you will see a prompt that looks like >> . You type your commands immediately after this prompt. Once you have typed the command you wish MATLAB to perform, press <enter>. If you want to *interupt a command* that MATLAB is running, type <ctrl> + <c>.

The commands you type in the command window are stored by MATLAB and can be viewed in the *Command History* window. To repeat a command you have already used, you can simply double-click on the command in the history window, or use the <up arrow> at the command prompt to iterate through the commands you have used until you reach the command you desire to repeat.

Useful functions and operations in MATLAB
Using MATLAB as a calculator is easy.

Example: Compute 5 $\sin(2.5^{3-pi})+1/75$.

In MATLAB this is done by simply typing
5*sin(2.5^(3-pi))+1/75

at the prompt. Be careful with parantheses and don't forget to type * whenever you multiply!

Note that MATLAB is *case sensitive*. This means that MATLAB knows a difference between letters written as lower and upper case letters. For example, MATLAB will understand sin(2) but will not understand Sin(2).

Here is a table of useful operations, functions and constants in MATLAB.

Operation, function or constant	**MATLAB command**
+ (addition)	+
- (subtraction)	-
� (multiplication)	*
/ (division)	/
|x| (absolute value of x)	abs(x)
square root of x	sqrt(x)
e^x	exp(x)
ln x (natural log)	log(x)
$\log_{10}$ x (base 10 log)	log10(x)
sin x	sin(x)
cos x	cos(x)
tan x	tan(x)
cot x	cot(x)
arcsin x	asin(x)
arccos x	acos(x)
arctan x	atan(x)
arccot x	acot(x)
n! (n factorial)	gamma(n+1)
e (2.71828...)	exp(1)
(3.14159265...)	pi
i (imaginary unit, sqrt(-1))	i

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Exercises

Compute the following expressions using MATLAB:

- 3cos(pi)
- 1+1+1/2+1/6+1/24-e
- ln $(1000+2^{pi-2})$
- $e^{i\ pi}$
- The number of combinations in which 12 persons can stand in line. (Hint: Use factorials.)

Obtaining Help on MATLAB commands

To obtain help on any of the MATLAB commands, you simply need to type

help <command>

at the command prompt. For example, to obtain help on the gamma function, we type at the command prompt:

help gamma

Try this now. You may also get help about commands using the "Help Desk", which can be accessed by selecting the MATLAB Help option under the Help menu.

Note that the description MATLAB returns about the command you requested help on contains the command name in ALL CAPS. This does not mean that you use this command by typing it in ALL CAPS. In MATLAB, you almost always use all lower case letters when using a command.

Variables in MATLAB

We can easily define our own variables in MATLAB. Let's say we need to use the value of 3.5sin(2.9) repeatedly. Instead of typing 3.5*sin(2.9)over and over again, we can denote this variable as x by typing the following:

x=3.5*sin(2.9)

(Please try this in MATLAB.) Now type

x+1

and observe what happens. Note that we did not need to declare x as a variable that is supposed to hold a floating point number as we would need to do in most programming languages.

Often, we may not want to have the result of a calculation printed-out to the command window. To supress this output, we put a semi-colon at the end of the command; MATLAB still performs the command in "the background". If you defined x as above, now type

y=2*x;

y

and observe what happened.

In many cases we want to know what variables we have declared. We can do this by typing whos. Alternatively, we can view the values by openning the "Workspace" window. This is done by selecting the Workspace option from the View menu. If you want to erase all variables from the MATLAB memory, type clear. To erase a specific variable, say x, type clear x. To clear two specific variables, say x and y, type clear x y, that is separate the different variables with a space. Variables can also be cleared by selecting them in the Workspace window and selecting the delete option.

Vectors and matrices in MATLAB

We create a vector in MATLAB by putting the elements within [] brackets.

Example: x=[1 2 3 4 5 6 7 8 9 10]

We can also create this vector by typing x=1:10. The vector (1 1.1 1.2 1.3 1.4 1.5) can be created by typing x=[1 1.1 1.2 1.3 1.4 1.5] or by typing x=1:0.1:1.5.

Matrices can be created according to the following example. The matrix A= $\begin{pmatrix} 1 & 2 & 3 \\ 4 & 5 & 6 \\ 7 & 8 & 9 \end{pmatrix}$ is created by typing

A=[1 2 3 ; 4 5 6; 7 8 9],

i.e., rows are separated with semi-colons. If we want to use a specific element in a vector or a matrix, study the following example:

Example:

x=[10 20 30]

A=[1 2 3 ; 4 5 6 ; 7 8 9]

x(2)

A(3,1)

Here we extracted the second element of the vector by typing the variable and the position within parantheses. The same principle holds for matrices; the first number specifies the row of the matrix, and the second number specifies the column of the matrix. **Note that in MATLAB the first index of a vector or matrix starts at 1, not 0 as is common with other programming languages**.

If the matrices (or vectors which are special cases of a matrices) are of the same dimensions then matrix addition, matrix subtraction and scalar multiplication works just like we are used to.

How to plot with MATLAB

There are different ways of plotting in MATLAB. The following two techniques, illustrated by examples, are probably the most useful ones.

Example 1: Plot sin(x^2) on the interval [-5,5]. To do this, type the following:

x=-5:0.01:5;

y=sin(x.^2);

plot(x,y)

and observe what happens.

Example 2: Plot exp(sin(x)) on the interval [-□,□]. To do this, type the following:

x=linspace(-pi,pi,101);

y=exp(sin(x));

plot(x,y)

and observe what happens. The command linspace creates a vector of 101 equally spaced values between -□ and □ (inclusive).

Ocassionally, we need to plot values that vary quite differently in magnitude. In this case, the regular plot command fails to give us an adequate graphical picture of our data. Instead, we need a command that plots values on a log scale. MATLAB has 3 such commands: loglog,semilogx, and semilogy. Use the help command to see a description of each function. As an example of where we may want to use one of these plotting routines, consider the following problem:

Example 3: Plot $x^{5/2}$ for x = 10^{-5} to 10^{5}. To do this, type the following:

x=logspace(-5,5,101);

y=x.^(5/2);

plot(x,y)

and observe what happens. Now type the following command:

loglog(x,y)

The command logspace is similar to linspace, however it creates a vector of 101 points lograthmically equally distributed between 10^{-5} and 10^{5}.

The following commands are useful when plotting:

Graphing functions	**MATLAB command**
Label the horizontal axis.	xlabel('text')
Label the vertical axis.	ylabel('text')
Attach a title to the plot.	title('text')
Change the limits on the x and y axis.	axis([xmin xmax ymin ymax])
"Keep plotting in the same window."	hold on
Turn off the "keep-plotting-in-the-same-window-command".	hold off

Note that all text must be put within ' '. The last two commands (hold on and hold off) are best explained by trying them next time you plot.

Viva Questions:-

1. What is The MATLAB working environment?
2. What is Simulink?
3. What is 3D-Visualization elements in MATLAB?
4. What are the basic Plots and Graphs of MATLAB?
5. What Is Stress Analysis in MATLAB?
6. Explain MATLAB API (Application Program Interface)?
7. What does MATLAB consist of?
8. Explain what is MATLAB? Where MATLAB can be applicable?
9. How to run MATLAB code?
10. How to write function in MATLAB?

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR

Experiment No.9. Cantilever Beam Modal Analysis by Using MATLAB

Objective:- To find the Natural frequency & mode shapes of a cantilever beam

- Modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration. Modal analysis is used to determine a structure's vibration characteristics — natural frequencies and mode shapes.
- It is the most fundamental of all dynamic analysis types and is generally the starting point for other, more detailed dynamic analyses.
- **Modal analysis**, or the mode-superposition method, is a linear dynamic-response procedure which evaluates and superimposes free-vibration mode shapes to characterize displacement patterns. Mode shapes describe the configurations into which a structure will naturally displace. Typically, lateral displacement patterns are of primary concern. Mode shapes of low-order mathematical expression tend to provide the greatest contribution to structural response. As orders increase, mode shapes contribute less, and are predicted less reliably. It is reasonable to truncate analysis when the number of mode shapes is sufficient.
- A structure with N degrees of freedom will have N corresponding mode shapes. Each mode shape is an independent and normalized displacement pattern which may be amplified and superimposed to create a resultant displacement pattern.

Problem Specification:-

Consider an aluminum beam that is clamped at one end, with the following dimensions.

Length	4 m
Width	0.346 m
Height	0.346 m

The aluminum used for the beam has the following material properties.

Density	2,700 kg/m^3
Youngs Modulus	70x10^9 Pa
Poisson Ratio	0.35

Using Matlab find the first six natural frequencies of the beam and the mode shapes.

Results Required:- 1. Frequency tabular data 2. Mode shapes (6)

Pre-Analysis

The following equations give the frequencies of the modes and the mode shapes and are derived from Euler-Bernoulli Beam Theory.

$$w_n = \alpha_n^2 \sqrt{\frac{EI}{ml^3}}$$
$$n = 1, 2, 3, \ldots$$
$$\alpha_n = 1.875, 4.694, 7.855, \ldots$$
$$m = \rho V = \rho \cdot l \cdot h \cdot w$$
$$I = \frac{w \cdot h^3}{12}$$
$$w_1 = 1.875^2 \sqrt{\frac{70\ E9\ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{2.7\ E3\ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 111.7\ \frac{rad}{s} = 17.8\ Hz$$
$$w_2 = 4.694^2 \sqrt{\frac{70\ E9\ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{2.7\ E3\ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 700.4\ \frac{rad}{s} = 111.5\ Hz$$
$$w_3 = 7.855^2 \sqrt{\frac{70\ E9\ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{2.7\ E3\ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 1961.2\ \frac{rad}{s} = 312.1\ Hz$$

$$y_i(x) = cosh(\frac{\alpha_i x}{L}) - cos(\frac{\alpha_i x}{L}) - \sigma_i(sinh(\frac{\alpha_i x}{L}) - sin(\frac{\alpha_i x}{L}))$$
$$\alpha_i = 1.875, 4.694, 7.855, \ldots$$
$$\sigma_i = 0.73409,\ 1.018647,\ 0.9992245,\ \ldots$$

MATLAB Script:-

```
function cbeam2(~)
clear all;
clc;

   W=input('Enter Width of the X-section in [m]: ');
   Th=input('Enter Thickness of the X-section in [m]:');
   L=input('Enter Length in [m]: ');
   Ix=(1/12)*W*Th^3;
   Iy=(1/12)*(W^3)*Th;
   A=W*Th;
   disp('Material properties of the beam')
   disp('Do you know your beam''s material properties, viz. Young''s modulus and density ?')

   E=input('Enter Young''s modulus in [Pa]: ');
   Ro=input('Enter material density in [kg/m^3]:  ');

display('How many modes and mode shapes would you like to evaluate ?')
HMMS=input('Enter the number of modes and mode shapes to computed: ');
if HMMS>=7
   disp('  ')
   warning('NOTE: Up to 6 mode shapes (plots) are displayed via the script. Yet, using evaluated data (Xnx) of the
script, more mode shapes can be plotted');
   disp(' ')
end
   Nm=3*HMMS;
   jj=1;
```

```
  while jj<=Nm;
            betaNL(jj)=fzero(@(betaNL)cosh(betaNL)*cos(betaNL)+1,[jj jj+3]);
       jj=jj+3;
  end
   index=(betaNL~=0);
   betaNLall=(betaNL(index))';
   %fprintf('betaNL value is %2.3f\n', betaNLall);
   betaN=(betaNLall/L)';
k=1;
wn=ones(1,length(betaN));
fn=ones(1,length(wn));
while k<=length(betaN);
   wn(k)=betaN(k)^2*sqrt((E*Ix)/(Ro*A));
   fn(k)=wn(k)/(2*pi);
   fprintf('Mode shape # %2f corresponds to nat. freq (fn): %3.3f\n', k, fn(k) );
   k=k+1;
end
x=linspace(0, L, 180);
xl=x./L;
sigmaN=zeros(1, HMMS);
for ii=1:HMMS;
   sigmaN(ii)=(sinh(betaN(ii)*L)-sin(betaN(ii)*L))/(cosh(betaN(ii)*L)+cos(betaN(ii)*L));
end
Tc='(cosh(betaN(ii).*x(jj))-cos(betaN(ii).*x(jj)))-sigmaN(ii).*(sinh(betaN(ii).*x(jj))-sin(betaN(ii)*x(jj)))';
Xnx=zeros(length(betaN),length(x));
for ii=1:length(betaN)
   for jj=1:length(x)
      Xnx(ii,jj)=eval(Tc);
   end
end
% Plot mode shapes;
MMS=HMMS;
if MMS==1
   plot(xl,Xnx(1,:), 'b-')
   title('Mode shape of the Cantilever beam')
   legend('Mode #1', 0); xlabel('x/L'); ylabel('Mode shape X_n(x)'); grid
   hold off
elseif MMS==2
   plot(xl,Xnx(1,:), 'b-'); hold on
   plot(xl,Xnx(2,:), 'r-');grid
   title('Mode shapes of the Cantilever beam')
   legend('Mode #1', 'Mode #2', 0)
   xlabel('x/L'); ylabel('Mode shape X_n(x)')
   hold off;
elseif MMS==3
   plot(xl,Xnx(1,:), 'b-'); hold on
   plot(xl,Xnx(2,:), 'r-')
   plot(xl,Xnx(3,:), 'm-');grid
   title('Mode shapes of the Cantilever beam')
   legend('Mode #1', 'Mode #2', 'Mode #3', 0)
   xlabel('x/L'); ylabel('Mode shape X_n(x)')
   hold off;
elseif MMS==4
   plot(xl,Xnx(1,:), 'b-'); hold on
```

```
    plot(xl,Xnx(2,:), 'r-')
    plot(xl,Xnx(3,:), 'm-')
    plot(xl,Xnx(4,:), 'c-'); grid
    title('Mode shapes of the Cantilever beam')
    legend('Mode #1', 'Mode #2', 'Mode #3', 'Mode #4', 0)
    xlabel('x/L'); ylabel('Mode shape X_n(x)')
    hold off;
elseif MMS==5
    plot(xl,Xnx(1,:), 'b-'); hold on
    plot(xl,Xnx(2,:), 'r-')
    plot(xl,Xnx(3,:), 'm-')
    plot(xl,Xnx(4,:), 'g-')
    plot(xl,Xnx(5,:), 'k-')
    grid
    title('Mode shapes of the Cantilever beam')
    legend('Mode #1', 'Mode #2', 'Mode #3', 'Mode #4', 'Mode #5', 0)
    xlabel('x/L'); ylabel('Mode shape X_n(x)')
    hold  off
elseif MMS>=6
    plot(xl,Xnx(1,:), 'b-'); hold on
    plot(xl,Xnx(2,:), 'r-')
    plot(xl,Xnx(3,:), 'm-')
    plot(xl,Xnx(4,:), 'g-')
    plot(xl,Xnx(5,:), 'k-')
    plot(xl,Xnx(6,:), 'c-')
    grid
    title('Mode shapes of the Cantilever beam')
    legend('Mode #1', 'Mode #2', 'Mode #3', 'Mode #4', 'Mode #5','Mode #6', 0)
    xlabel('x/L'); ylabel('Mode shape X_n(x)')
    hold off
end

end
```

MATLAB Command Prompt:- write cbeam2- Enter↵

Required Input as per the Problem Statement

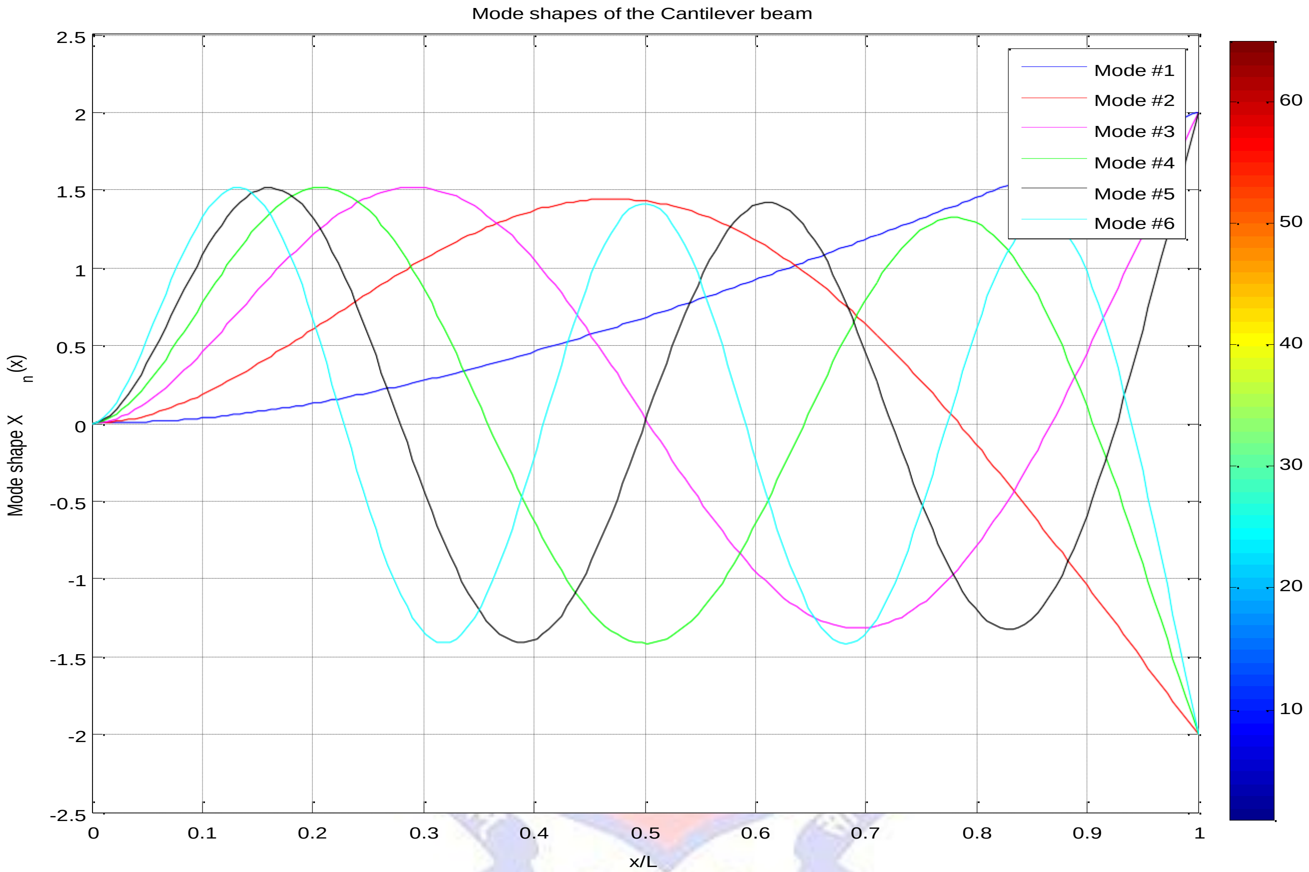
Rectangular Plate

Length (L)	= 4 m
Width (w)	= 0.346 m
Height (h)	= 0.346 m
Young Modulus	= 70 x 10^9
Density	= 2700 Kg/m^3
Mode	= 6

Results:

Mode shape # 1.000000 corresponds to nat. freq (fn): 17.787
Mode shape # 2.000000 corresponds to nat. freq (fn): 111.469

Mode shape # 3.000000 corresponds to nat. freq (fn): 312.118
Mode shape # 4.000000 corresponds to nat. freq (fn): 611.626
Mode shape # 5.000000 corresponds to nat. freq (fn): 1011.062
Mode shape # 6.000000 corresponds to nat. freq (fn): 1510.351



Viva Questions:-

1. List out the operators that MATLAB allows?
2. What is the type of program files that MATLAB allows to write?
3. How to call a function in MATLAB?
4. What is MATLAB used for?
5. How to run MATLAB code?
6. How to write function in MATLAB?
7. What is MATLAB software?
8. How to stop a program in MATLAB?
9. How to resize an image in MATLAB?
10. Explain MATLAB API (Application Program Interface)?

Dr. Mahesh Bundele
B.E., M.E., Ph.D.
Director
Poornima College of Engineering
ISI-6, RIICO Institutional Area
Sitapura, JAIPUR