

A Laboratory Manual for

Computer Aided Design

(3161903)

B.E. Semester 6 (Mechanical)



Directorate of Technical Education, Gandhinagar,
Gujarat

Vision of the DTE

- To provide globally competitive technical education;
- Remove geographical imbalances and inconsistencies;
- Develop student friendly resources with a special focus on girls' education and support to weaker sections;
- Develop programs relevant to industry and create a vibrant pool of technical professionals.

Vision of the Institute

Mission of the Institute

MECHANICAL ENGINEERING DEPARTMENT

Vision of the Department

Mission of the Department

Lukhdhirji Engineering College, Morbi

Certificate

This is to certify that Mr./Ms. _____
_____ Enrollment No. _____ of B.E. Semester
_____ Mechanical Engineering of this Institute (GTU Code: _____) has
satisfactorily completed the Practical / Tutorial work for the subject **Computer
Aided Design (3161903)** for the academic year 2024-24.

Place: _____

Date: _____

Name and Sign of Faculty member

Head of the Department

LIST OF EXPERIMENTS with CO Mapping-3161903(CAD)

Sr.No	Name of Practical	CO MAPPING
01	Prepare a program for plotting lines and curves using algorithms learned. <ul style="list-style-type: none"> Students will prepare the Program in C Language for the assignment given. (Bresenham's Line , DDA Line , One C-Graphics Object etc) 	CO1
02	Introductory exercise for 3-D modeling <ul style="list-style-type: none"> Students will draw 2D-sketches first using the basic commands in the sketch module in 3D-Modeling software to get acquainted or use too. Students will generate the 3D- Solid model using the commands learned in the lab session. 	CO2
03	Exercise for advanced 3-D modeling. <ul style="list-style-type: none"> Students will generate / draw the advanced 3D model given in the assignment. 	CO2
04	Exercise for 3-D editing options <ul style="list-style-type: none"> Students will learn and operate the editing features available in the 3D modeling software on the model created previously. 	CO2
05	Exercise for Assembly modeling. <ul style="list-style-type: none"> Students will prepare the 3D Assembly given in the assignment in 3D Modeling software. 	CO2
06	Exercise for FEA of 1-D structural problems. <ul style="list-style-type: none"> Students will perform analysis in licensed Ansys Software for simple problems given in assignment and verify the manual calculation. 	CO4
07	Exercise for FEA of trusses. <ul style="list-style-type: none"> Students will perform analysis in licensed Ansys Software for simple problems given in assignment and verify the manual calculation. 	CO4
08	Exercise for FEA using Beam Element <ul style="list-style-type: none"> Students will perform analysis in licensed Ansys Software for simple problems given in assignment and verify the same via manual calculation. 	CO4
09	Exercise for FEA of 1-D thermal problems. <ul style="list-style-type: none"> Students will perform analysis in licensed Ansys Software for simple problems given in assignment and verify the manual calculation. 	CO4
10	Exercise for FEA of 2-D structural problems <ul style="list-style-type: none"> Students will perform analysis in licensed Ansys Software for simple problems given in assignment and verify the manual calculation. 	CO4
11	Exercise for developing the optimization model of machine elements using Johnson Method.	CO5

Course Outcome: (As per GTU)

3161903.1	Demonstrate basic concept of computer aided design and its applications.
3161903.2	Make use of various concepts and characteristics in geometric modeling.
3161903.3	Analyze geometric transformations
3161903.4	Determine stress and strain in structural elements through FEA
3161903.5	Summarized optimization techniques for design of machine elements.

Government Engineering College, Dahod
DEPARTMENT OF MECHANICAL ENGINEERING
Computer Aided Design-3161903

INDEX

Sr. No.	Experiment	Page Number		Dates		Sign	Grades/ marks
		Start	End	Start	End		
1	Prepare a programme for plotting lines and curves using algorithms learned.						
2	Introductory exercise for 3D modeling.						
3	Exercise for advanced 3-D modeling.						
4	Exercise for 3-D editing options.						
5	Exercise for Assembly modeling.						
6	Exercise for FEA of 1-D structural problems						
7	Exercise for FEA of trusses.						
8	Exercise for FEA using Beam Element.						
9	Exercise for FEA of 1-D thermal problems.						
10	Exercise for FEA of 2-D structural problems.						
11	Exercise for developing the optimization model of machine element using Johnson Method.						

Following Instructions are mandatory for Laboratory/Term Work

1. Students are required to solve their tutorials/assignments as per instruction of the faculty and get them signed before due date.
2. Students are required to bring a note book for rough work, pencil, eraser, sharpener, compass, foot rule, and calculator along with a reference book for the subject in the laboratory.
3. Experiments and tutorials/assignments/quiz will be regularly assessed and the cumulative marks will form your final score of term work at the end of semester.
4. Students should prepare their journals in best possible way. Your journal should be simple yet attractive. Getting your journal checked regularly by concerned faculty is an indication of regularity of work.
5. Students must attend each and every practical that will help you in answering your oral exams. Ask the questions to your faculty whenever you find difficulty in understanding the Practical's.
6. Every student should maintain the laboratory manual properly.
7. Students must come to the laboratory in proper dress code. Half pants, loosely hanging garments and slippers are not allowed.
8. Students must ensure that their work areas are clean and dry to avoid any accident.
9. In CAD Lab, all students must enter the details in the entry register available on lab entrance.
10. Keep your bags in the stand or rack provided.
11. Properly operate the PC as per the instruction of lab assistant/faculty. For storage of files/work use only Drive of Computer.
12. Do not misuse the internet (i.e, do not upload/download any unlawful/unwanted material)

In this laboratory you will be exposed to the modeling and analysis software. Laboratory experiments will consist of hands on session and demonstration of the above mentioned experiments.

You are required to submit a report on each laboratory session in the following that session if asked. You have to refer to the textbooks / reference book for appropriate answers to the questions.

EXPERIMENT: 01

Aim: Prepare a programme for plotting lines and curves using algorithms learned.

Theory:

Introduction

Computer-aided design is essentially based on a versatile and powerful technique called computer graphics, which basically means the creation and manipulation of pictures on a display device with the aid of a computer. Computer graphics originated at the Massachusetts Institute of Technology (MIT) in 1950 when the first computer-driven display, linked to a Whirlwind 1 computer, and was used to generate some pictures. The first important step forward in computer graphics came in 1963 when a system called SKETCHPAD was demonstrated at the Lincoln Laboratory of MIT. This system consists of a cathode ray tube (CRT) driven by TX2 computer. The CRT had a keyboard and a light pen. Pictures could be drawn on the screen and then manipulated interactively by the user via the light pen.

This demonstration clearly showed that the CRT could potentially be used as a designer's electronic drawing board with common graphic operations such as scaling, translation, rotation, animation and simulation automatically performed at the 'push of a button'. At that time, these systems were very expensive; therefore they were adopted only in such major industries as the aircraft and automotive industries where their use in design justified the high capital costs. Another crucial factor preventing computer graphics from being generally applied to engineering industries was that there was a lack of appropriate graphics and application software to run on these systems. However, a computer-based design system was clearly emerging. Since these pioneering developments in computer graphics, which had captured the imagination of the engineering industry all over the world, new and improved hardware, which is faster in processing speed, larger in memory, cheaper in cost and smaller in size, have become widely available. Sophisticated software techniques and packages have also been gradually developed. Consequently, the application of CAD in industry has been growing rapidly. Initially CAD systems primarily were automated drafting stations in which computer controlled plotters produced engineering drawings. The system were later linked to graphic display terminals where geometric model describing part dimensions were created, and the resulting database in the computer was then used to produce drawings.

Nowadays, CAD systems can do much more than mere drafting. Some systems have analytical capabilities that allow parts to be evaluated with techniques such as the finite element method. There is also

kinematics analysis Programs that enable the motion of mechanism to be studied. In addition, CAD system includes testing techniques to perform model analysis on structures, and to evaluate their response to pinpoint any possible defects.

The term computer graphics includes almost everything on computers that is not text or sound. Today almost every computer can do some graphics, and people have even come to expect to control their computer through icons and pictures rather than just by typing. Here in our lab at the Program of Computer Graphics, we think of computer graphics as drawing pictures on computers, also called rendering. The pictures can be photographs, drawings, movies, or simulations - pictures of things, which do not yet exist and maybe could never exist. Or they may be pictures from places we cannot see directly, such as medical images from inside your body. We spend much of our time improving the way computer pictures can simulate real world scenes. We want images on computers to not just look more realistic, but also to be more realistic in their colors, the way objects and rooms are lighted, and the way different materials appear. We call this work “realistic image synthesis”.

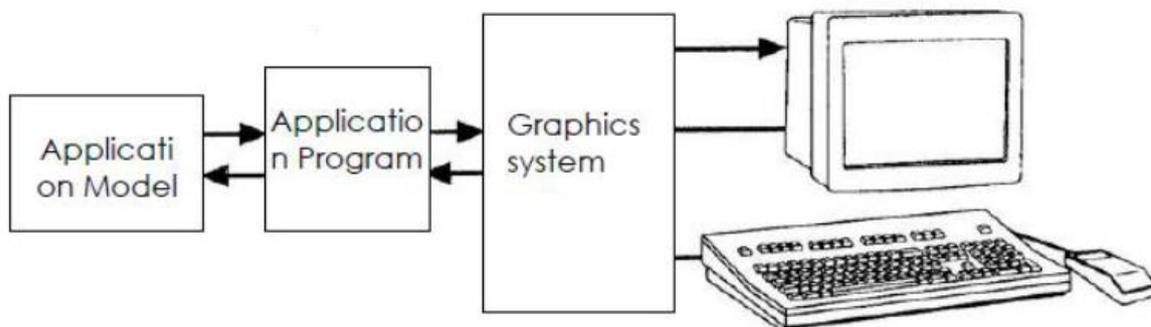


Fig 1.1: Conceptual Model of Interactive Graphics

Thus the graphics system is a layer in between the application program and the display hardware that effects an output transformation from objects in the application model to a view of the model. The objective of the application model is to captures all the data, objects, and relationships among them that are relevant to the display and interaction part of the application program and to any non graphical post processing modules.

- **DDA Line Drawing Algorithm:**

Digital differential analyzer (DDA) is hardware or software used for linear interpolation of variables over an interval between start and end point. DDAs are used for rasterization of lines, triangles and polygons. DDA algorithm is an incremental scan conversion method. Here we perform calculations at each step using the

results from the preceding step. The characteristic of the DDA algorithm is to take unit steps along one coordinate and compute the corresponding values along the other coordinate.

Algorithm:

1. Read the line end points (x_1, y_1) and (x_2, y_2) such that they are not equal. [If equal then plot that point and exit]
2. $dx = |x_2 - x_1|$ and $dy = |y_2 - y_1|$
3. If $(dx \geq dy)$ then
 Length=dx
 else
 Length=dy
 end if
4. $x_{inc} = (x_2 - x_1)/length$
 $y_{inc} = (y_2 - y_1)/length$

[This makes either dx or dy equal to 1 because length is either $|x_2 - x_1|$ or $|y_2 - y_1|$ Therefore, the incremental value for either x or y is one.]

5. $i=1$ [begins the loop, in this points are plotted]
 - While $(i \leq length)$
 - {
 - Plot (integer(x), integer(y))
 - $x = x + x_{inc}$
 - $y = y + y_{inc}$
 - $i = i + 1$
 - }

Advantages of DDA Algorithm:

- It is the simplest algorithm and it does not require special skills for implementation.
- It is a faster method for calculating pixel positions than the direct use of equation.
 $y = mx + b$.
- It eliminates the multiplication in the equation by making use of raster Characteristics, so that appropriate increments are applied in the x or y direction to Find the pixel positions along the line path.

Disadvantages of DDA Algorithm:

- Floating point arithmetic in DDA algorithm is still time-consuming.
- The algorithm is orientation dependent. So the end point accuracy is poor.

Bresenham's line algorithm:

Bresenham's line algorithm is an algorithm that determines which points in an n -dimensional raster should be plotted in order to form a close approximation to a straight line between two given points. It is

commonly used to draw lines on a computer screen, as it uses only integer addition, subtraction and bit shifting, all of which are very cheap operations in standard computer architectures. It is one of the earliest algorithms developed in the field of computer graphics. A minor extension to the original algorithm also deals with drawing circles.

1. Read the line end points (x_1, y_1) and (x_2, y_2) such that they are not equal. [If equal then plot that point and exit]
2. plot start point
3. calculate the constant $dx, dy, 2dx, 2dy$ and get the first value for initial parameter

$$P_k = 2dy - dx$$
4. at each x_k along the line starting at $k=0$ perform the following test $p_k < 0$, the next point to plot is

$$\begin{aligned} &(x_k + 1, y_k) \text{ and } p_{k+1} = p_k + 2dy \\ &\text{else } (x_k + 1, y_k + 1) \\ & \quad p_{k+1} = p_k + 2dy - 2dx \end{aligned}$$
5. Repeat step 4 till $(dx-1)$ times

Output primitives

- A picture can be described in several ways.
 In a raster display, a picture is completely specified by the set of intensities for the pixel positions in the display.

 At the other extreme, a picture can be described as a set of complex objects, such as trees and terrain or furniture and walls, positioned at specified coordinate locations within the scene.
- Shapes and colors of the objects can be described internally with pixel arrays or with sets of basic geometric structures, such as straight line segments and polygon color areas.
- The scene is then displayed either by loading the pixel arrays into the frame buffer or by scan converting the basic geometric-structure specifications into pixel patterns.
- Graphics programming packages provide functions to describe a scene in terms of these basic geometric structures, referred to as output primitives, and to group sets of output primitives into more complex structures.
- Each output primitive is specified with input coordinate data and other information about the way that object is to be displayed.
- Points and straight line segments are the simplest geometric components of pictures.
- Additional output primitives that can be used to construct a picture include circles and other conic sections, quadric surfaces, spline curves and surfaces, polygon color areas, and character strings.

Point Plotting:

- Point plotting is attained by converting a single coordinate position furnished by an application program into appropriate operations for output device in use.
- For example, with a CRT monitor, the electron beam is turned on to illuminate the screen phosphor at the selected location. The electron beam is positioned based on the display technology.
- For a random-scan(vector) system :
 - This stores point-plotting instructions in the display list.
 - Coordinate values in these instructions are converted to deflection voltages that position the electron beam at the screen locations to be plotted during each refresh cycle.
- For a black-and-white raster system,
 - a point is plotted by setting the bit value corresponding to a specified screen position within the frame buffer to 1.
 - Then, as the electron beam sweeps across each horizontal scan line, it emits a burst of electrons (plots a point) whenever a value of 1 is encountered in the frame buffer.
- With an RGB system raster system
 - The frame buffer is loaded with the color codes for the intensities that are to be displayed at the screen pixel positions.

Line Drawing:

- The line is the most fundamental drawing primitive with many uses - charts, engineering drawings, illustrations, 2D pencil-based animation and curve approximation.
- Desired properties of line drawing algorithms :
 - Lines should appear sharp and straight
 - Line should terminate accurately
 - Line should have constant density
 - Line density should be independent of line, length and angle.
 - Efficient
- Line drawing is accomplished by calculating intermediate positions along directed to fill in these positions between the endpoint positions. An output device is then directed to fill in these positions between the endpoints.
- For analog devices, such as a vector pen plotter or a random-scan display, a straight line can be drawn smoothly from one endpoint to the other. Linearly varying horizontal and vertical deflection voltages are generated that are proportional to the required changes in the x and y directions to produce smooth lines.
- Digital devices display a straight line segment by plotting discrete points between two endpoints. Discrete coordinate positions along the line path are calculated from the equation of the line.
- For a raster video display, the line color (intensity) is then loaded into the frame buffer at the corresponding pixel coordinates. Reading from the frame buffer, the video controller then "plots" the screen pixels. Screen locations are referenced with integer values, so plotted positions may only

approximate actual Line positions between two specified endpoints. A computed line position of (10.48, 20.51), for example, would be converted to pixel position (10,21). Thus rounding of coordinate values to integers causes lines to be displayed with a stair step appearance Figure 1.2

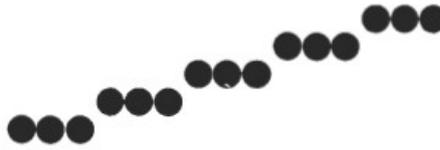


Fig 1.2 Stair step effect (jaggies) produced when a line is generated as a series of pixel positions.

- The characteristic stair step shape of raster lines is particularly noticeable on systems with low resolution; their appearance is improved by displaying them on high-resolution systems.
- More effective techniques for smoothing raster lines are based on adjusting pixel intensities along the line paths.
- For the raster-graphics device-level algorithms, object positions are specified directly in integer device coordinates. The pixel positions are referenced according to scan-line number and column number (pixel position across a scan line). This addressing scheme is shown in Figure 1.3.

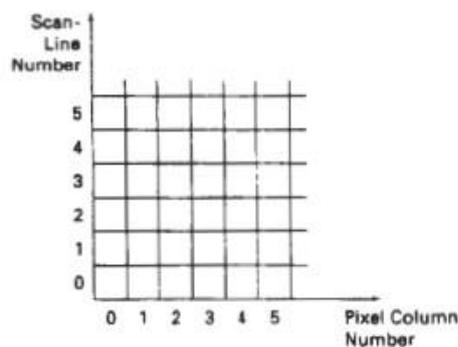


Fig 1.3 Pixel positions referenced by scan line number and column number.

- Scan lines are numbered consecutively from 0, starting at the bottom of the screen; and pixel columns are numbered from 0, left to right across each scan line.
- To load a specified color into the frame buffer at a position corresponding to column x along scan line y, a low-level procedure set Pixel of the form is used

set Pixel(x,y)

- To retrieve the current frame buffer intensity setting for a specified location the low-level Function get Pixel is used get

Pixel(x,y)

Procedure:

1. Students are informed to prepare the algorithm for line using the DDA and Bresenham's method.
2. Students are informed to prepare the algorithm for drawing the circle using Bresenham's method.
3. Prepare the C Program of above algorithm, run the program in computer.
4. Plot the output result and take printout.
5. Students also have to write the program in the files pages with the below mention assignment questions.

Questions for Experiment-1:

1. Prepare flow chart for DDA, Bresenham's algorithm for rasterizing a line and Write a C program for the same
2. Prepare flow chart for Bresenham's Circle algorithm and write a C program for the same
3. Write the advantages and disadvantages of Bresenham's algorithm over DDA Algorithm for drawing a line
4. Write the advantages and limitations of DDA line algorithm.
5. Determine the raster scan locations selected by Bresenham's algorithm while generating a line from (10, 10) To (30, 50).
6. Explain in brief about the coordinate systems-MCS, WCS, SCS.
7. List the various functions and commands of graphics mode available in C Language and explain in brief.
8. Write a C-Graphics Program to draw the line from (0,0) origin to (500,500).

Suggested References:

- <https://nptel.ac.in/courses/106/102/106102063/>
- https://www.vssut.ac.in/lecture_notes/lecture1424803788.pdf
- <https://cse.iitkgp.ac.in/~pb/pb-graphics-2018.pdf>
- <https://www.javatpoint.com/computer-graphics-programs>
- <https://www.mycplus.com/tutorials/c-programming-tutorials/graphics-programming/>
- <https://developerinsider.co/download-turbo-c-for-windows-7-8-8-1-and-windows-10-32-64-bit-full-screen/> (For Downloading Turbo C)
- <https://www.youtube.com/watch?v=A3i2qFx7d5g> (Video)
- <https://www.programmingsimplified.com/c/graphics.h#functions> (C-Graphics Help)
- <https://www.dgp.toronto.edu/~hertzman/418notes.pdf> (Lecture Notes of other Uni.)

Sign of Faculty/Lab In charge:

Rubric wise marks obtained:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 02

INTRODUCTORY EXERCISE FOR 3-D MODELLING

Aim: To study brief about the 3D modeling software and exercise/practice on software available.

Tool used: Extrude, Revolve, Fillets, Chamfers, Ribs, Shell

Introduction:

3D Modeling is used in a variety of applications to make representations of physical objects on the computer. 3D modeling is a subset of Computer Aided Design (CAD), in which you use a computer to assist in the design process for any type of design work. It is used in a variety of applications, mostly when it comes to designing parts on the computer to assist in the making or visualization of those parts. The computer model is used to communicate dimensions, material types, etc. to anyone viewing the design, and can be used to make control paths for Computer Numerical Controlled (CNC) machines. 3D Modeling Services is the blend of developing a geometrical representation of any facet of an object or a product in three dimensions via specialized software. In 3D Computer graphics, the product is called 3D Model. It includes 3D Modeling of an object or a product by manipulating polygons, circumference and vertices in simulated 3D space.

3D Modeling in general makes the product design process more efficient. Modeling programs allow you to create and visualize final products, modify and optimize the designs, and document designs, measurements and materials easily. If you've heard of 3D printing, 3D modeling is what is used to design objects before they are 3D printed.

From movies to manufacturing, 3D modeling is incredibly useful. There are hundreds of different 3D design programs out there, each for a specific application. 3D modeling allows engineers to flush out their ideas before they become reality, so most objects that you see around you were first designed in 3D design software by engineers before they were made. 3D design is extremely important for this reason: engineers, architects, and the like use 3D CAD programs to design things before building them all the time. For example, each component of your computer was modeled in 3D modeling software, each part's shape and cost was optimized for its use, and all of the models were put together in an assembly in the software to ensure that they all fit together properly. The files were then all sent to a manufacturer, where computer controlled machines were able to make all of the parts, and workers used the files to follow the assembly steps to physically build the computer that you have sitting in front of you.

Working with a 3D model **helps you render three-dimensional objects more realistically**. That way, you can evaluate the design before working on the prototype. The design industry is currently seeing a

decline in the use of physical models, with more focus on the digital stage to reduce spending and maximize time.

Arguments to move up from 2D to 3D include:

- Better visualization: the model looks like the real live object.
- Lesser risk for mistakes: a 3D-system will only accept spatial structures that can (geometrically) exist in space.
- Faster dataflow in between steps in the design process: as all information (geometry, administrative and technological features) is part of the object, all this information will pass automatically to the next step in design or manufacturing.
- Automated drawing extraction: spatial objects are the basis for projection views, sectioning and hidden line removal.
- Faster design: editing possibilities lead to the recycling of former designs.

Description of Tools:

- 1. Extrude:** The extrude tool is used to define the extrusion depth of the sketch by specifying a depth value.
 - **Blind Extrude:** The tool is used to define the extrusion depth of the sketch by specifying a depth value.
 - **Symmetric:** The tool is used to add the material the equally in both the directions of the sketching plane.
 - **Through All:** The tool is used to set the extrusion depth of the sketch through all the surfaces of the model in the direction of extrusion.
 - **Through until:** The tool is used to select the surface up to which the sketch is to be extruded.
 - **To next:** The tool is used to set the extrusion depth of the sketch up to the next surface of the model in the direction of extrusion.
- 2. Revolve:** The revolve tool allows you to revolve the sketched section through the specified angle about an axis of the coordinate system or of a geometric centerline.
- 3. Chamfers:** The chamfer tool is used to bevel the selected edges and corners as per some specified parameters.
 - **Corner chamfer:** The tool is used to create a chamfer on a corner.
 - **Edge chamfer:** The tool is used to create a chamfer on an edge.
- 4. Ribs:** Ribs are defined as thin wall like structures used to bind the joints together so that they do not fail under an increased load. The section for the rib is sketched as an open section and can be extruded equally in both directions of the sketch plane or an either side.
- 5. Fillets:** The fillet tool is used to round the edges or faces of a part to a specified radius.
- 6. Shell:** Shell features remove surfaces to hollow out a design model, leaving walls with specified thickness values.

There are two parts for the creation of a basic shell feature:

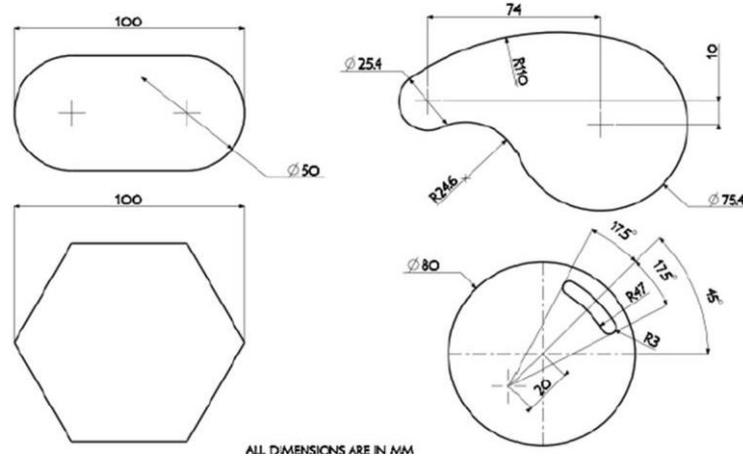
- **Select Surface for Removal** – Select the surface or surfaces want to remove from the model. You may decide not to remove any surfaces from the shell, which results in the creation of a

closed shell, with the whole inside of the part hollowed out and no access to the hollow.

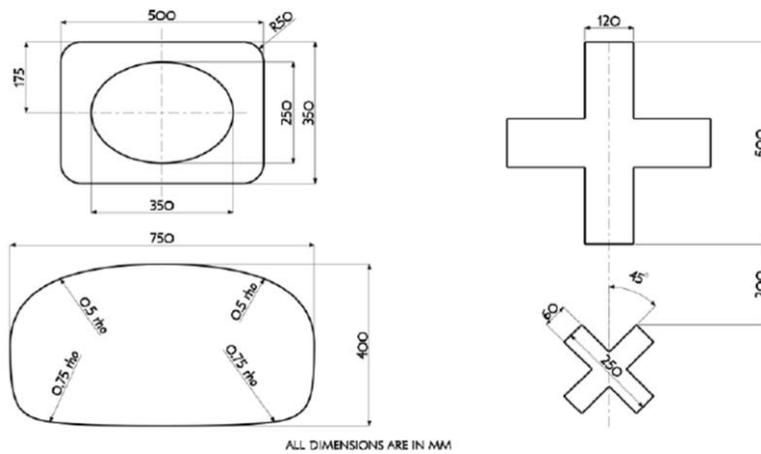
- Thickness – Specify the thickness of the model walls that remain. You create shells the design process to support your design intent. However, be aware that several features could reference a shell created early in the design process.

Exercise: Create 3-dimensional solid model using above mentioned tools in modeling software.

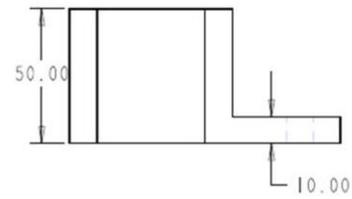
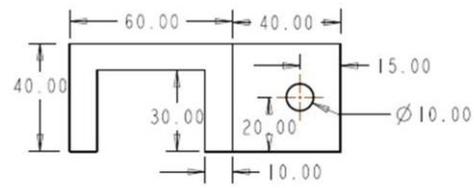
1.



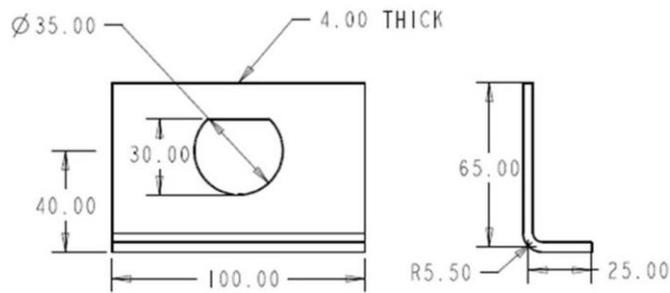
2.



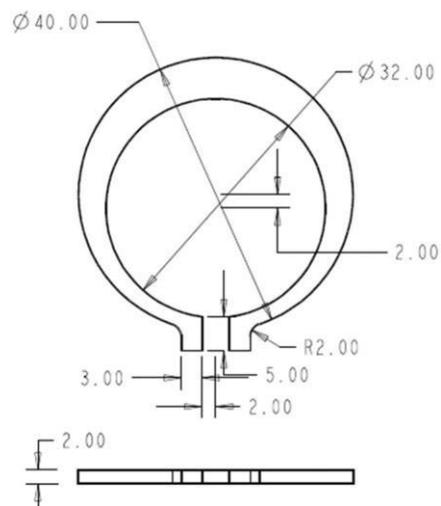
3.



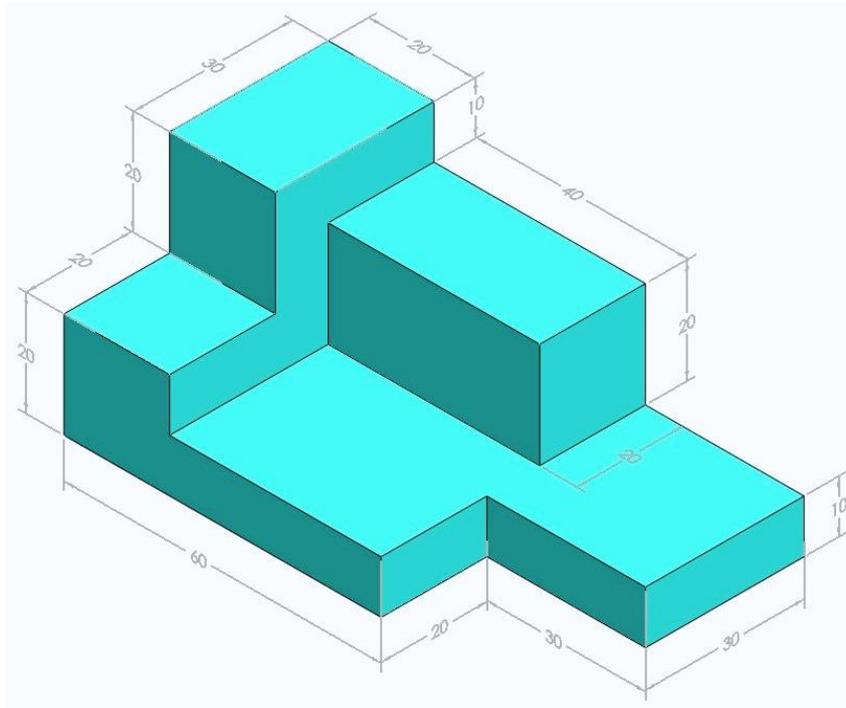
4.



5.



6.



Questions:

1. Explain parametric representation?
2. Write the difference between 2D and 3D solid modeling.
3. List the various free and paid software for 2D Sketching and 3D part modeling.

References:

1. <https://academy.autodesk.com/>
2. <https://www.ptc.com/en/ptc-university>
3. <https://drive.google.com/file/d/1NLAKSve8OR9t87HpjKp4HeOrdNQ8WuAb/view?usp=sharing>
4. <https://www.youtube.com/watch?v=ZlzwJdW-UDE> (Installation)
5. <https://www.youtube.com/watch?v=X0AMdUMNsDI> (Practice)
6. <https://www.youtube.com/watch?v=BF1SZv8KhfM> (Practice)
7. <https://www.youtube.com/watch?v=GS0CvBh0xW8> (Video by Faculty)

Sign of Faculty/Lab in-charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 03

EXERCISE FOR ADVANCED 3D-MODELING

Aim: Students have to create the part files of the 3D Model of the part drawing provided.

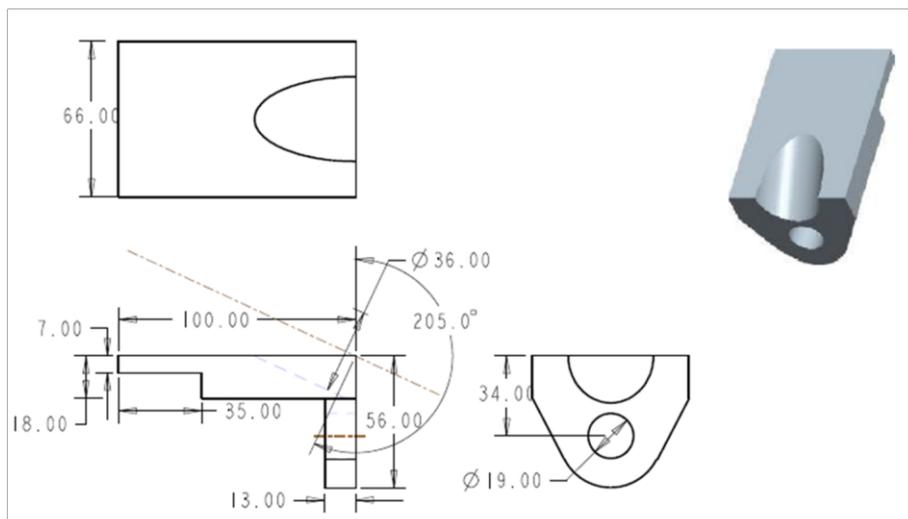
Objectives:

- Create an object using pattern
- Sketch a **Trajectory** for a sweep
- Sketch and locate a **Sweep section**

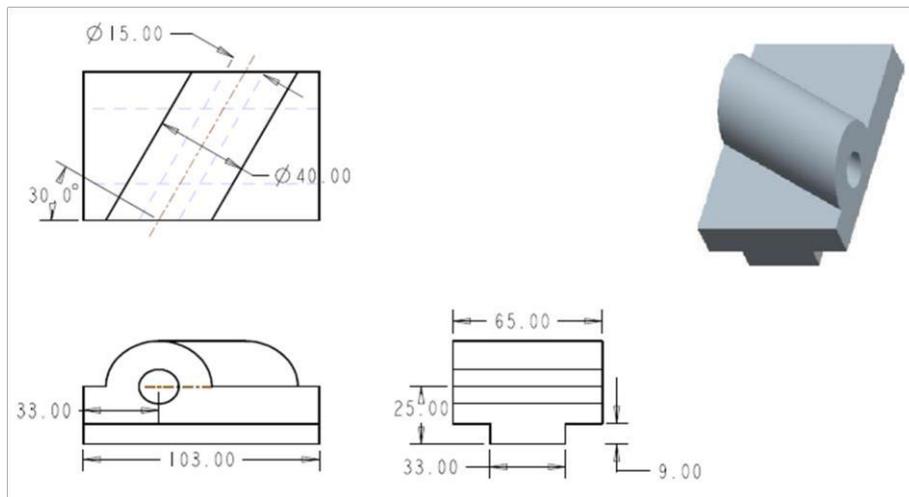
Instructions:

Students are informed to attend the video/practical hands on session in CAD laboratory for learning, how to create the 3D part model in the software. According to the steps learn, he/she draw the 3D Model and save the part file in desktop PC for the following problems. (Assume suitable dimensions where ever necessary)

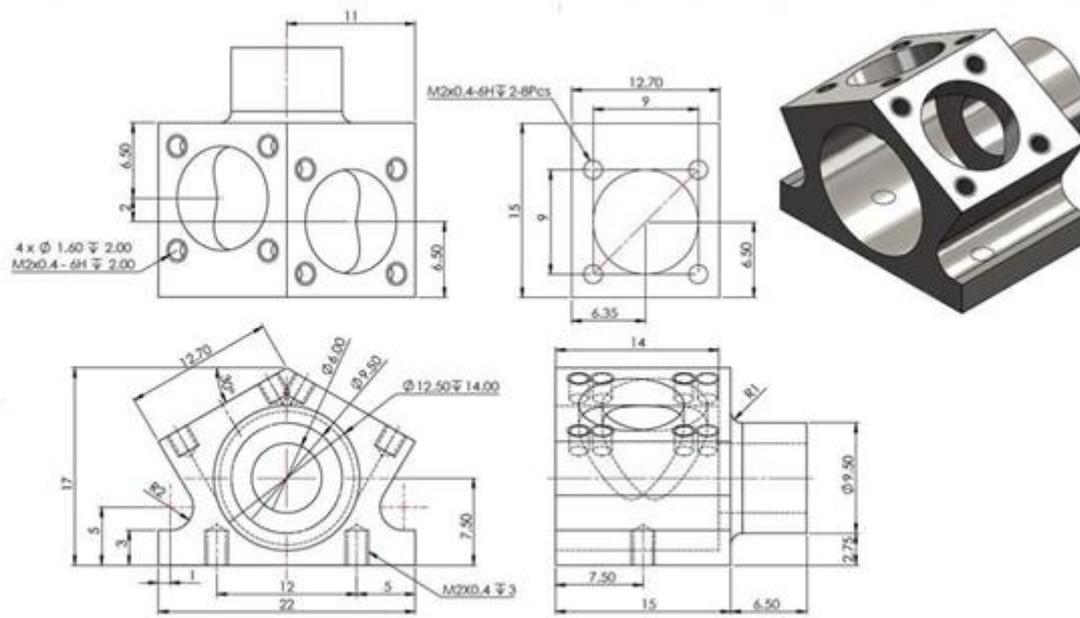
1.



2.



3.



Questions:

1. List the command used for making the above 3D Model part files.
2. Take the screenshot of each step for any one problem from above for making the 3D model, and take printout of screenshot pdf file and also submit the soft copy PDF file of it.

References:

1. <https://www.youtube.com/watch?v=KkzQoij-TJo> (Cup Modeling)
2. https://www.youtube.com/watch?v=fWTz9Fv94_k
3. <https://www.youtube.com/watch?v=JHAZ9L5XaNE>
4. https://support.ptc.com/help/creo/creo_pma/usascii/index.html#page/tutorials_pma/online_help/aux_files/pma_tutorials.html

Sign of Faculty/Lab In charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 04

EXERCISE FOR 3D-EDITING OPTIONS

Aim: Exercise for 3-D editing options.

Tool used: Reordering, Rerouting, Suppressing, Deleting, Modifying

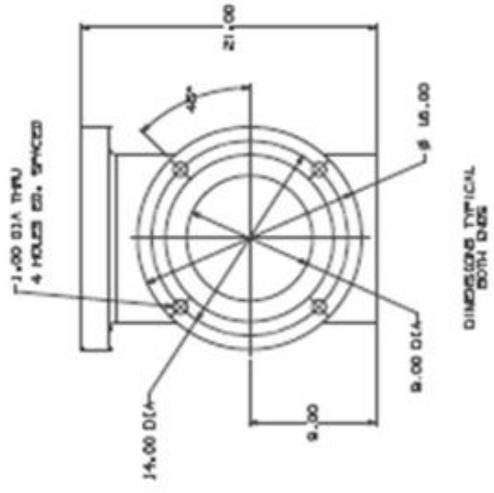
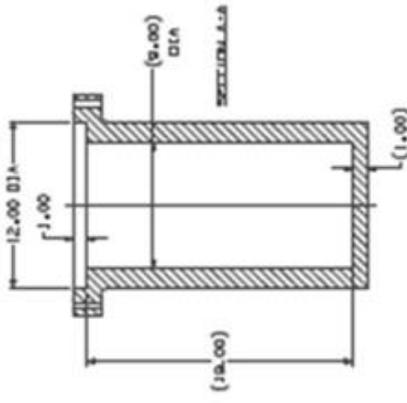
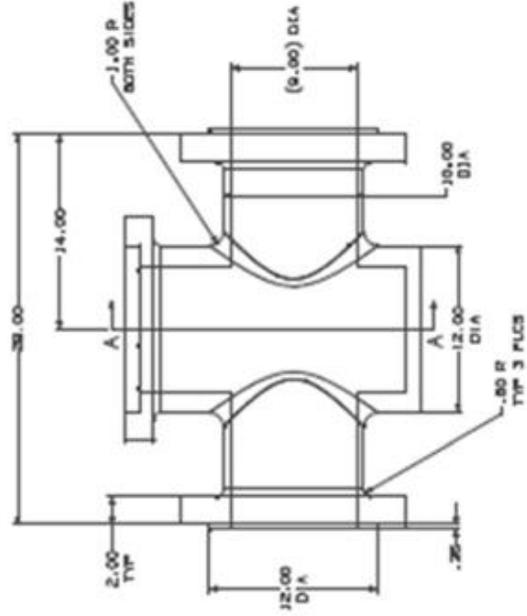
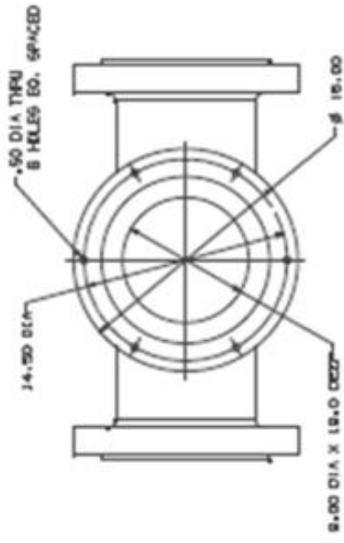
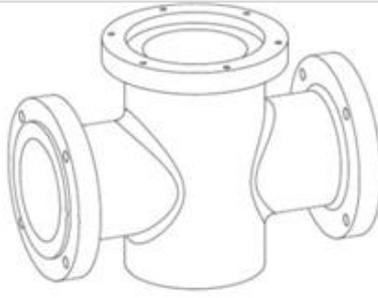
Introduction: Editing is one of the most important aspects of solid modeling. Most of the design requires the editing during their creation or after they are created.

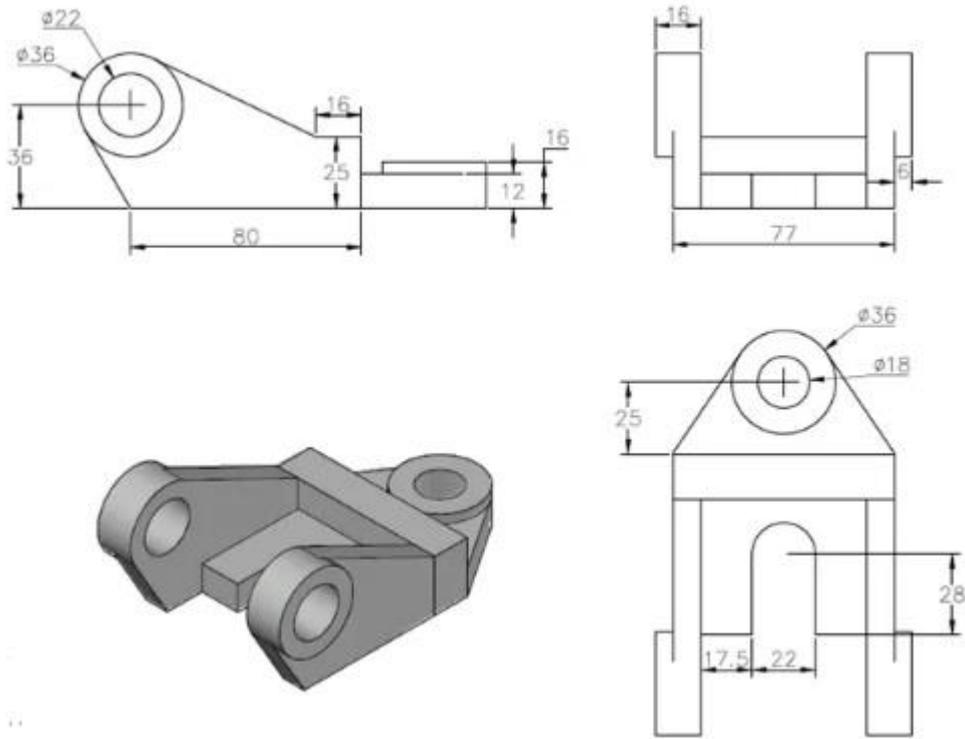
Description of Tools:

- 1. Redefining Features:** Editing the definition of features allows you to make changes in the parameters that were used to create it. Sketches of the sketched features can also modify by editing their definition.
- 2. Reordering Featured:** It is defined as the process of changing the sequence of feature creation in a model. Sometimes after creating a model, it may be required to change the order in which the features of the model were created. A feature can be placed before or after another feature.
- 3. Rerouting Features:** Rerouting of features is required when there is a need of changing its sketch plane. References can also redefine that were selected while creating the sketches.
- 4. Suppressing Features:** The Suppressed feature is used to hide the features in a model. Once the feature is suppressed, it will neither be displayed in the drawing area nor in the drawing views. The feature is not deleted, only its visibility is turned off. When a model has many features, then suppressing some features decreases its regeneration time.
- 5. Deleting Features:** The feature that is not required in the model can be deleted from the model.
- 6. Modifying Features:** Dimensions of a model are easy to modify by using modifying features.

Exercise:

Create 3-Dimensional solid model and modify it using above mentioned tools in modeling software.





(Ref <https://www.thsourcecad.com/autocad-practice-drawings-with-pdf-ebook/>)

Questions:

1. How in parametric software you can edit the 3D Model? (You can take the screen shot of the steps also).
2. Develop the 3D Model of the above drawing/Figure and use the “Edit Definition and Edit References” Option to the dimensions, references of the features of 3D Model.

References:

1. https://www.youtube.com/watch?v=ttMs393y8_Q
2. <https://www.youtube.com/watch?v=kRxBD-B5IrY> (Edit Definition)
3. <https://www.youtube.com/watch?v=V47f--6EfFM> (Edit Definition)
4. <https://www.youtube.com/watch?v=EP7WYFV1fkU> (Edit References)
5. http://support.ptc.com/help/creo/creo_pma/usascii/index.html#page/tutorials_pma%2Ftutorials_overview.html%23wwconnect_header (PTC Tutorials for beginners) .

Sign of Faculty/Lab In charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 05

EXERCISE FOR ASSEMBLY MODELLING

Aim: Students have to create the assembly files of the 3D Component, after that plot the assembly with dimensions.

Introduction:

CAD solid model programs use a database of primitive forms of solid geometry like cubes, cones, cylinders, pyramids, spheres, etc. These are combined in a variety of ways (adding or subtracting) to produce more complex models.

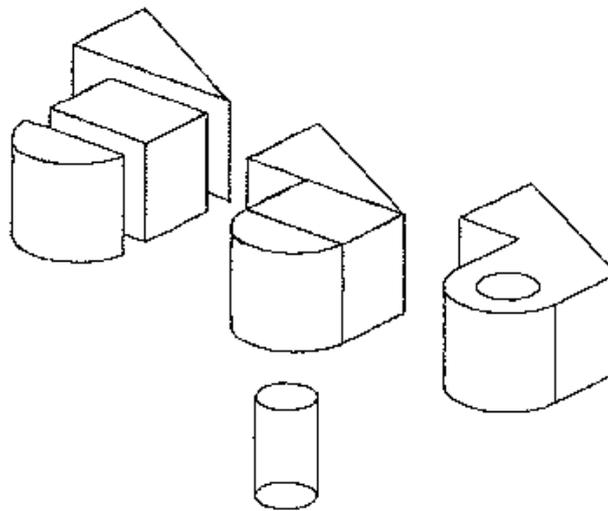


Fig 5.1: Assembly modeling using operation

This gives a well defined model. All volumes, surfaces and angles are well defined and known unlike surface modelers or wire frame modelers. Since all solids are represented inside the system as combinations of primitives or parts of primitives a task like calculating the volume of a complex space present few problems. It is just a question of taking the volumes of the constituent primitives and adding or subtracting them as appropriate. So volumes and weights are well known in the beginning of the design process and lead no room for misinterpretation.

Also designers found out that building a shape up out of primitives allows them to concentrate on essentials. The whole process brings them closer to the actual manufacturing process, and is in some cases almost a literal simulation of it. The parallel between boring a hole in an object and subtracting a model of a cylinder from a model of that object is after all very close and adding a model cogwheel to a model cylinder is very like slipping a real cogwheel onto a real axle.

The solid models are representing the geometry and can be the base for engineering analysis because the geometry is well defined. This integration between CAD and Engineering is very important in Concurrent Engineering because it makes more iteration steps possible during the

design. So many changes during the design process are possible. This will help to optimize the product, which will give an increasing product quality and a reduction in the product development time.

The analyzing methods that are mostly used can calculate and the mechanical, kinematics and dynamic behavior of the simulated software prototypes.

The ability to develop 3-D solid models is bridging the gaps between design, engineering, and manufacturing in a concurrent engineering environment. To save time and costs 3-D solid model files are electronically transferred to molders, and used to cut tools directly (CAD/CAM); the same files can also be used for rendering and analysis to reduce the risk of failure.

Assembly modeling further builds on solid modeling by combining solid part models together into an overall assembly model. The individual part solid model data files describing the 3D geometry of individual components are assembled together to create an assembly describing the whole product or assembly. Components can be positioned within the product assembly using absolute coordinate placement methods or by means of mating conditions. Mating conditions are definitions of the relative position of components between each other; for example alignment of two holes or location of two part faces to one another. Solid and assembly modeling can capture more than just the part geometry. Tolerance, geometric dimensioning, and notations can also be captured.

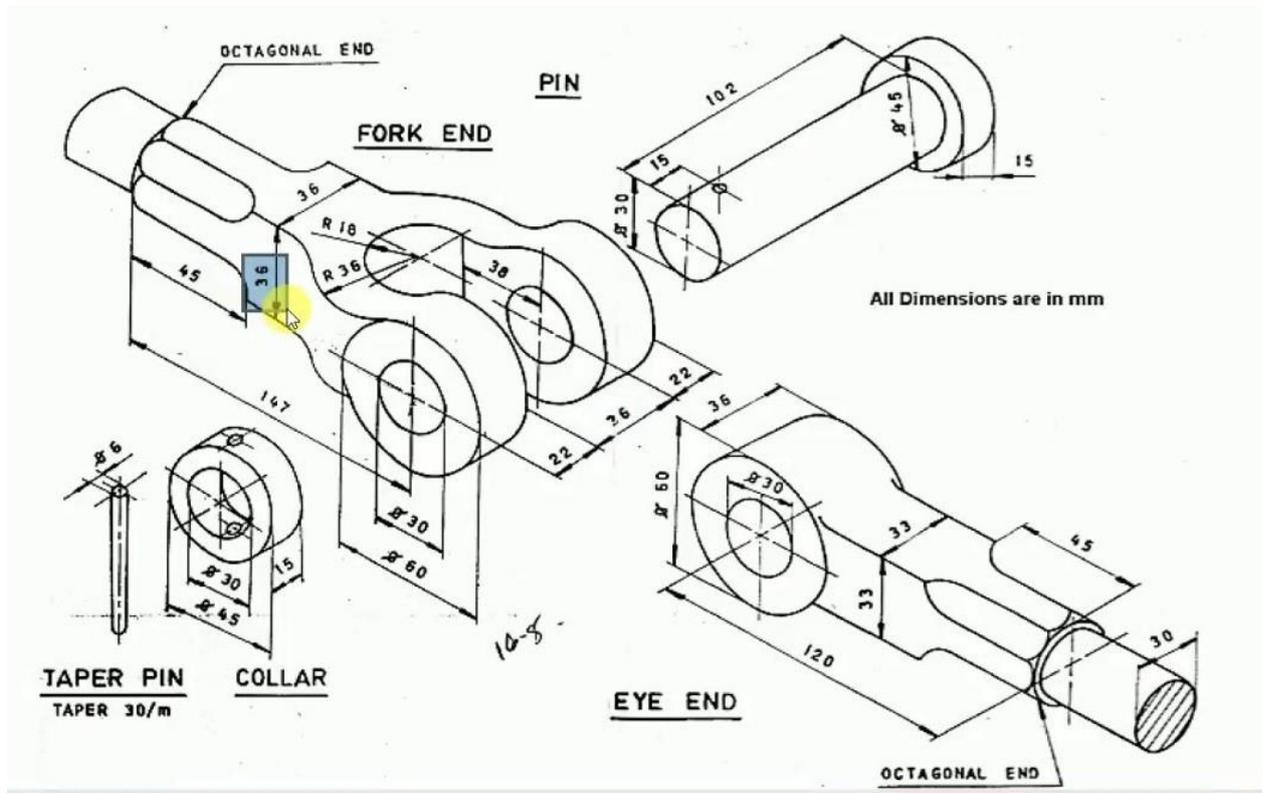
The individual data files describing the 3D geometry of individual components are assembled together through a number of sub-assembly levels to create an assembly describing the whole product. All CAD and CPD systems support this form of bottom-up construction. Some systems, via associative copying of geometry between components also allow top-down method of design.

Components can be positioned within the product assembly using absolute coordinate placement methods or by means of mating conditions. Mating conditions are definitions of the relative position of components between each other; for example alignment of axis of two holes or distance of two faces from one another. The final position of all components based on these relationships is calculated using a geometry constraint engine built into the CAD or visualization package.

Assembly:

- 1. Top down approach:** This is the method of assembling the components in which the components of the assembly are created in the same assembly file. In this type of assembly modeling approach, the components are created in the assembly file and then assembled using the assembly constraints.
- 2. Bottom down approach:** This is the method of assembling the components that are created as separate parts in the part mode. Once all parts of an assembly have been created, you will create a new assembly file and then assemble the parts using the assembly constraints.

Exercise: Create all the components of Knuckle Joint and assemble them as shown in figure. The dimension of all the components is given in figure below



Questions:

1. What is the difference between top down and bottom-up approach?
2. Develop the 3D Assembly of any one component in the solid modeling software and plot with dimension on sheet.

References:

1. <https://www.npd-solutions.com/solid-modeling.html>
2. <https://www.youtube.com/watch?v=3Td6VqUxunQ&t=224s>
3. <https://www.youtube.com/watch?v=6E05PBZ0ySY>
4. <https://www.youtube.com/watch?v=6FZ5XxF4hZs&list=PL8MELgWj0RxO5hLayfg1ka8qW8Q26Kpfx>
5. <https://nptel.ac.in/courses/112/102/112102101/>
6. https://www.ccri.edu/faculty_staff/engt/jsrobinson/Spring%202016%20Transfer/ENGR%201030%20Spring%202016/ENGR%201030%20Fall%202015/SWG2015.pdf
7. <http://www.cad-resources.com/CREO Parametric/Creo Parametric Download Lessons-Projects/CREO Projects Download.pdf>
8. <http://www.cad-resources.com/>

Sign of Faculty/Lab In charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 06

Exercise for FEA of 1-D structural problems.

Aim: Students have to solve the 1-D Structural problem in ANSYS Software.

Introduction:

The Finite Element Method (FEM) is a numerical technique to find approximate solutions of partial differential equations. It was originated from the need of solving complex elasticity and structural analysis problems in Civil, Mechanical and Aerospace engineering. In a structural simulation, FEM helps in producing stiffness and strength visualizations. It also helps to minimize material weight and its cost of the structures. FEM allows for detailed visualization and indicates the distribution of stresses and strains inside the body of a structure. Many of FE software are powerful yet complex tool meant for professional engineers with the training and education necessary to properly interpret the results. Several modern FEM packages include specific components such as fluid, thermal, electromagnetic and structural working environments. FEM allows entire designs to be constructed, refined and optimized before the design is manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The use of FEM has significantly decreased the time to take products from concept to the production line. One must take the advantage of the advent of faster generation of personal computers for the analysis and design of engineering product with precision level of accuracy.

Background of Finite Element Analysis:

The finite element analysis can be traced back to the work by Alexander Hrennikoff (1941) and Richard Courant (1942). Hrennikoff introduced the framework method, in which a plane elastic medium was represented as collections of bars and beams. These pioneers share one essential characteristic: mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.

- In 1950s, solution of large number of simultaneous equations became possible because of the digital computer.
- In 1960, Ray W. Clough first published a paper using term “Finite Element Method”.
- In 1965, First conference on “finite elements” was held.
- In 1967, the first book on the “Finite Element Method” was published by Zienkiewicz and Chung.

- In the late 1960s and early 1970s, the FEM was applied to a wide variety of engineering problems.
- In the 1970s, most commercial FEM software packages (ABAQUS, NASTRAN, ANSYS, etc.) originated. Interactive FE programs on supercomputer lead to rapid growth of CAD systems.
- In the 1980s, algorithm on electromagnetic applications, fluid flow and thermal analysis were developed with the use of FE program.
- Engineers can evaluate ways to control the vibrations and extend the use of flexible, deployable structures in space using FE and other methods in the 1990s. Trends to solve fully coupled solution of fluid flows with structural interactions, bio-mechanics related problems with a higher level of accuracy were observed in this decade.

The formulation for structural analysis is generally based on the three fundamental relations: equilibrium, constitutive and compatibility. There are two major approaches to the analysis: Analytical and Numerical. Analytical approach which leads to closed-form solutions is effective in case of simple geometry, boundary conditions, loadings and material properties. However, in reality, such simple cases may not arise. As a result, various numerical methods are evolved for solving such problems which are complex in nature. For numerical approach, the solutions will be approximate when any of these relations are only approximately satisfied. The numerical method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity. It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques. Few numerical methods which are commonly used to solve solid and fluid mechanics problems are given below.

- Finite Difference Method
- Finite Volume Method
- Finite Element Method
- Boundary Element Method
- Mesh less Method

Concepts of Elements and Nodes:

Any continuum/domain can be divided into a number of pieces with very small dimensions. These small pieces of finite dimension are called 'Finite Elements' (Fig. 6.1). A field quantity in each element is allowed to have a simple spatial variation which can be described by polynomial terms. Thus the original domain is considered as an assemblage of number of such small elements. These elements are connected through number of joints which are called 'Nodes'. While discretizing the structural system, it is assumed that the elements are attached to the adjacent elements only at the nodal points. Each element contains the material and geometrical properties. The material properties inside an element are assumed to be constant. The elements may be 1D elements, 2D elements or 3D elements. The physical object can be modeled by choosing appropriate element such as frame element, plate element, shell element, solid element, etc. All elements are

then assembled to obtain the solution of the entire domain/structure under certain loading conditions. Nodes are assigned at a certain density throughout the continuum depending on the anticipated stress levels of a particular domain. Regions which will receive large amounts of stress variation usually have a higher node density than those which experience little or no stress.

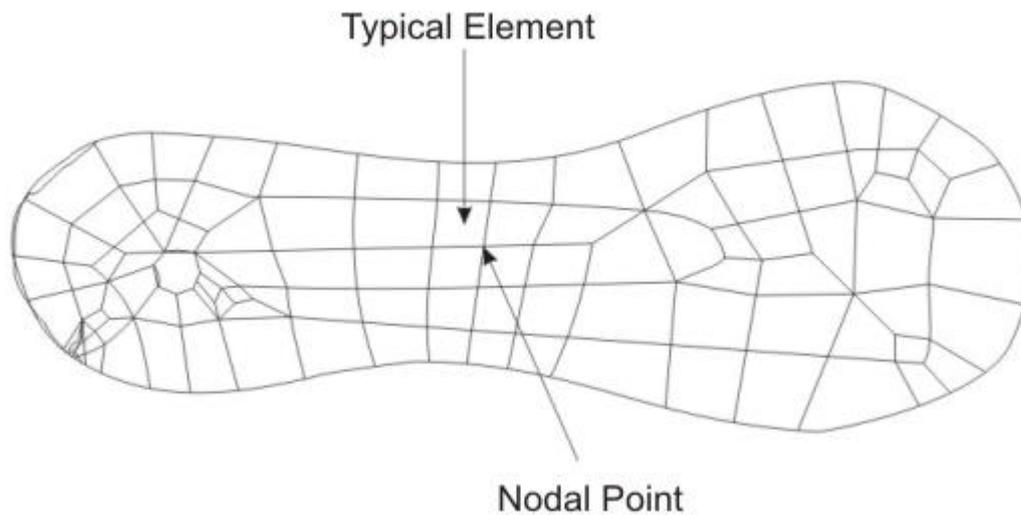


Fig. 6.1 Finite element discretization of a domain

Degrees of Freedom:

A structure can have infinite number of displacements. Approximation with a reasonable level of accuracy can be achieved by assuming a limited number of displacements. This finite number of displacements is the number of degrees of freedom of the structure. For example, the truss member will undergo only axial deformation. Therefore, the degrees of freedom of a truss member with respect to its own coordinate system will be one at each node. If a two dimension structure is modeled by truss elements, then the deformation with respect to structural coordinate system will be two and therefore degrees of freedom will also become two. The degrees of freedom for various types of element are shown in Fig. 6.2 for easy

understanding. Here (u,v,w) and $(\theta_x,\theta_y,\theta_z)$ represent displacement and rotation respectively.

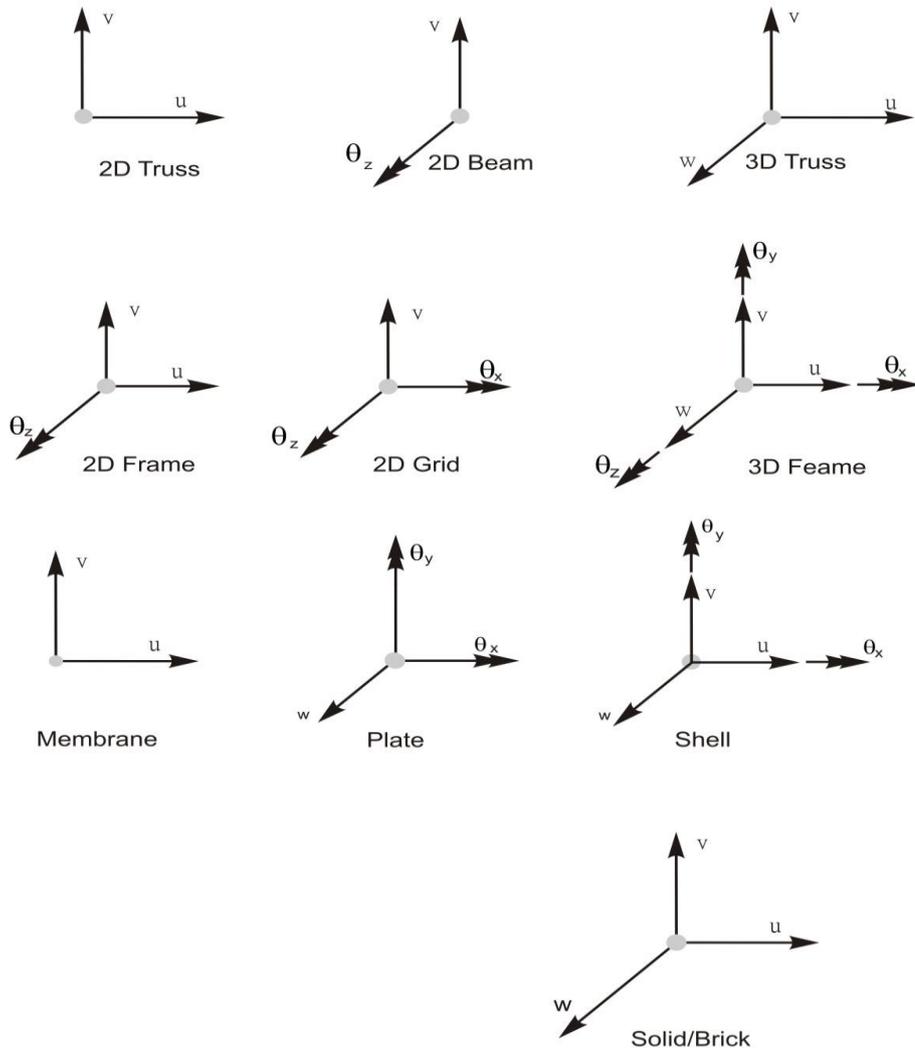


Fig. 6.2 Degrees of Freedom for Various Elements

Advantages of FEA:

- The physical properties, which are intractable and complex for any closed bound solution, can be analyzed by this method.
- It can take care of any geometry (may be regular or irregular).
- It can take care of any boundary conditions.
- Material anisotropy and non-homogeneity can be catered without much difficulty.
- It can take care of any type of loading conditions.
- This method is superior to other approximate methods like Galerkin and Rayleigh-Ritz methods.
- In this method approximations are confined to small sub domains.
- In this method, the admissible functions are valid over the simple domain and have nothing to do with boundary, however simple or complex it may be.

- Enable to computer programming.

Disadvantages of FEA

- Computational time involved in the solution of the problem is high.
- For fluid dynamics problems some other methods of analysis may prove efficient than the FEM.
- Proper engineering judgment is to be exercised to interpret results.
- It requires large computer memory and computational time to obtain intend results.
- There are certain categories of problems where other methods are more effective, e.g., fluid problems having boundaries at infinity are better treated by the boundary element method.
- For some problems, there may be a considerable amount of input data. Errors may creep up in their preparation and the results thus obtained may also appear to be acceptable which indicates deceptive state of affairs. It is always desirable to make a visual check of the input data.
- In the FEM, many problems lead to round-off errors. Computer works with a limited number of digits and solving the problem with restricted number of digits may not yield the desired degree of accuracy or it may give total erroneous results in some cases. For many problems the increase in the number of digits for the purpose of calculation improves the accuracy.

This experiment has given the following skills.

1. The ability to model 1-D problems in ANSYS.
2. The ability to adapt element types to specific situations by suppressing degrees of freedom (we used a 3D element as a 1D element).
3. The ability to generate finite element models using the direct method (i.e. defining nodes and then defining elements linking those nodes, as opposed to taking a solid model and dividing it up into elements which we will do in subsequent tutorials).
4. The ability to define element types, real constants and material parameters for a finite element model.
5. The ability to apply boundary conditions and loads to specific nodes in a finite element model.
6. The ability to run a simple linear static analysis.
7. The ability to list displacement results for each node in the finite element model.
8. The ability to create an element table to obtain additional results from a finite element model and to list these results.

9. Experience in comparing the results obtained from your finite element model with other results and validating your results against the other results.

- **Solve the tutorials as given below and compare with results of software.**

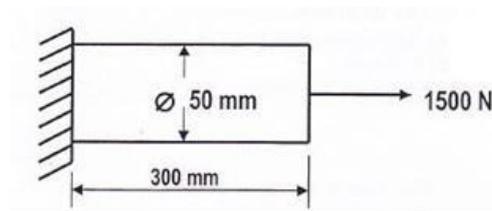
Problem Statement:

Find Total Displacement, Stress in the Element & Reaction Forces for the following Problem using Link 1 Element.

Given Data

$$E=2.1 \times 10^5 \text{ N/mm}^2$$

$$\nu = 0.3 \text{ (Poisson's Ratio).}$$



Overview

In this tutorial we will examine the 1-D displacement of a shaft in tension using

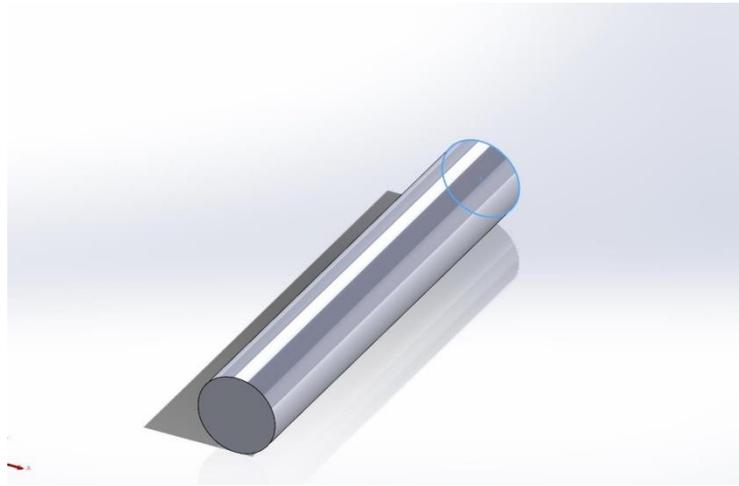
ANSYS.. You will determine the displacement distribution and stress distribution in the bar due to the applied loading and boundary conditions. A one-dimensional structural truss element (often also called a "spar", "spring" or "link" element) will be used for this analysis. We will use SI system units for this tutorial:

Procedure

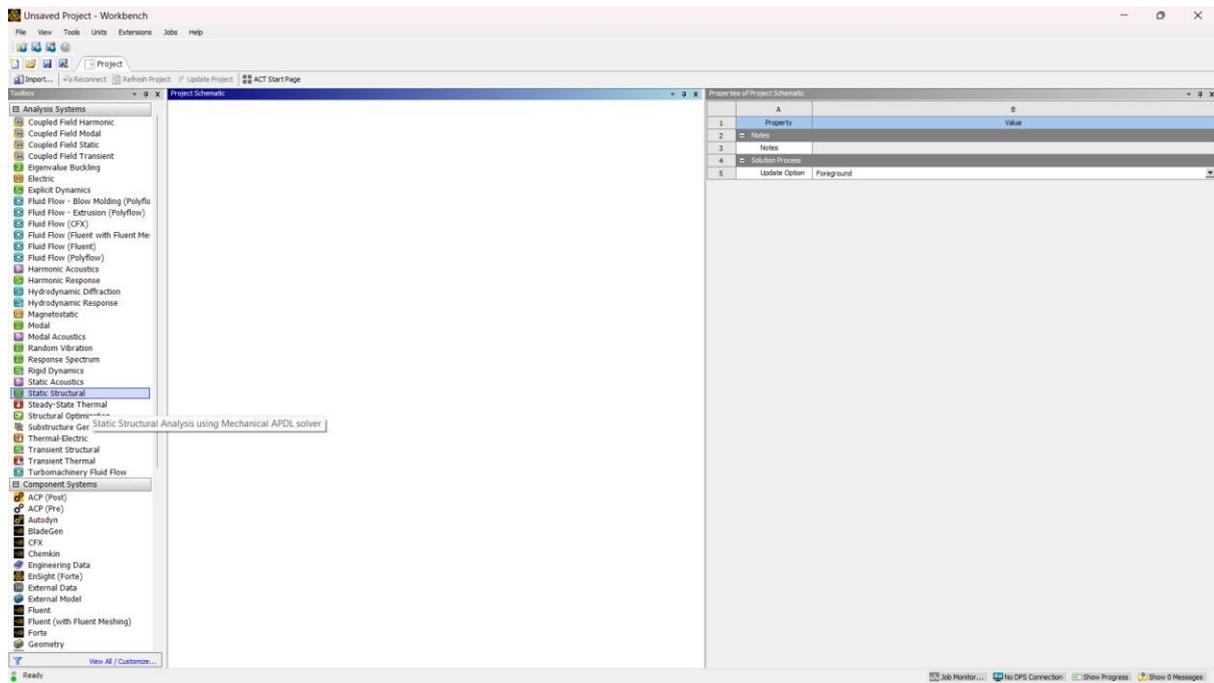
Step 1: create geometry

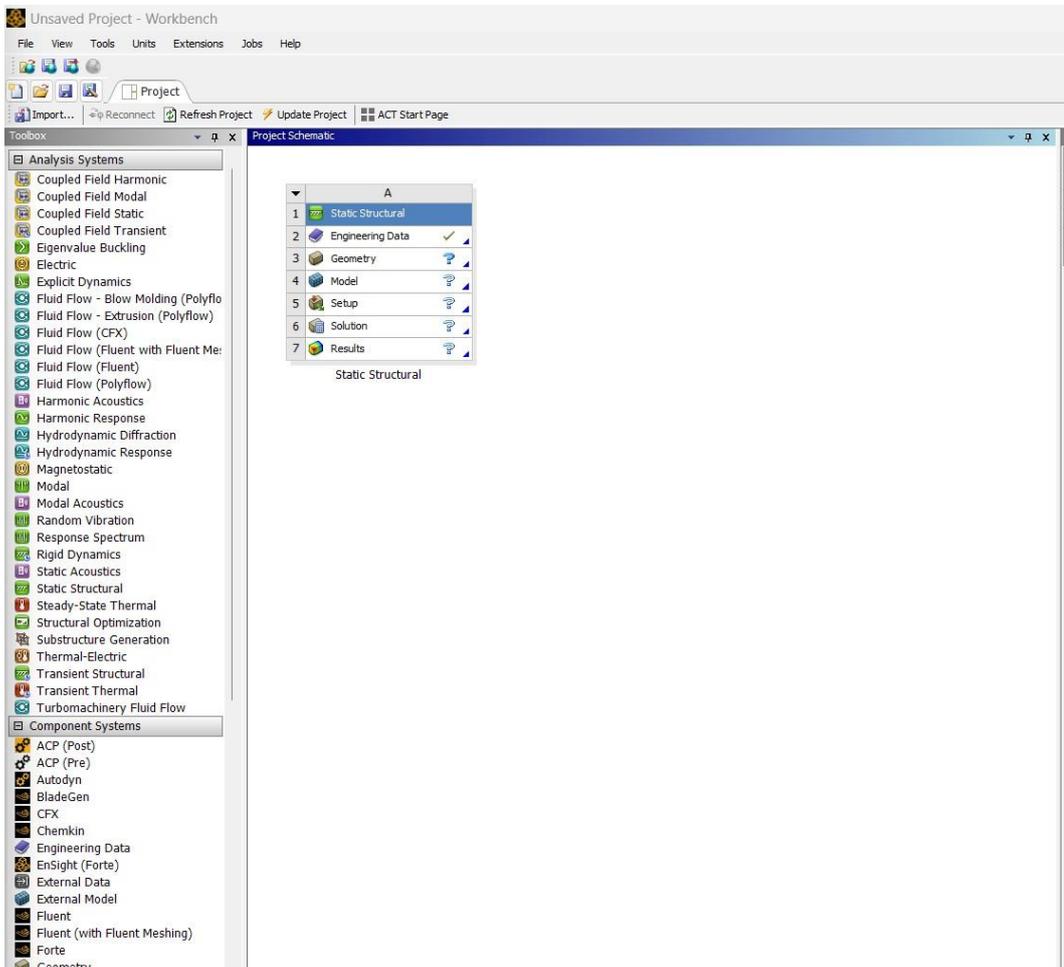
Use any modeling software(solid works, nx, fusion 360 etc.) or inbuilt space clam module to create the model of the problem

Make sure to save the model file as IGES/IGS format to successfully import the model into Ansys workbench



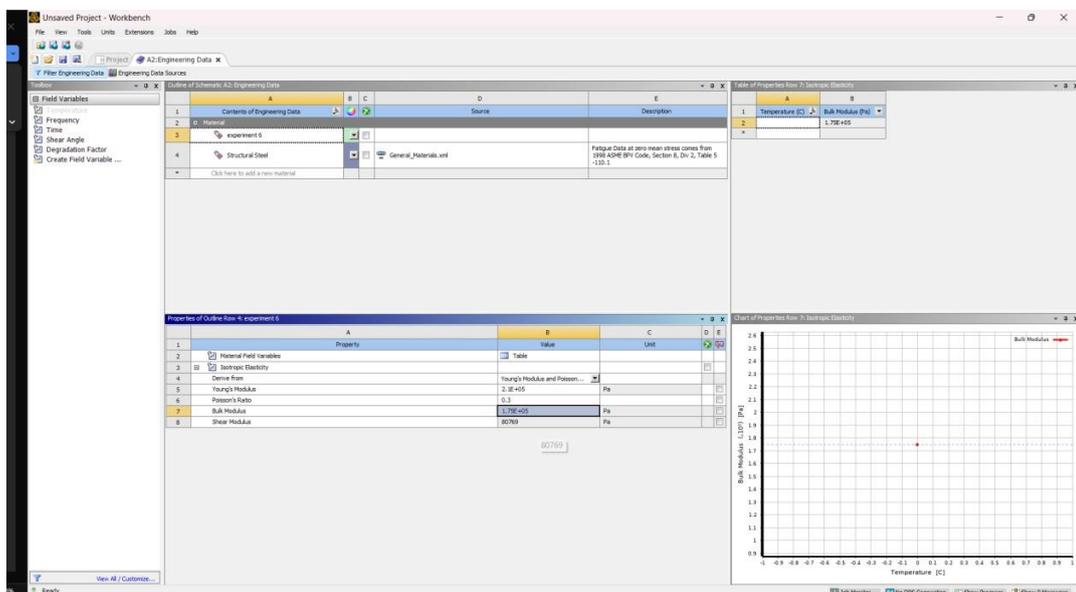
Step 2: open Ansys workbench and drag static structural module into project schematic





Step 3: add material

Double click on engineering data, then click on the blank box which says “click here to add a new material” and put the name of the material there. After that on left side go on linear Elastic properties and drag and drop “Isotropic Elasticity” on your material then put values of poisson’s ratio and young’s modulus (E) in yellow boxes (to add exponential values use “e” ex.50000=5e4)

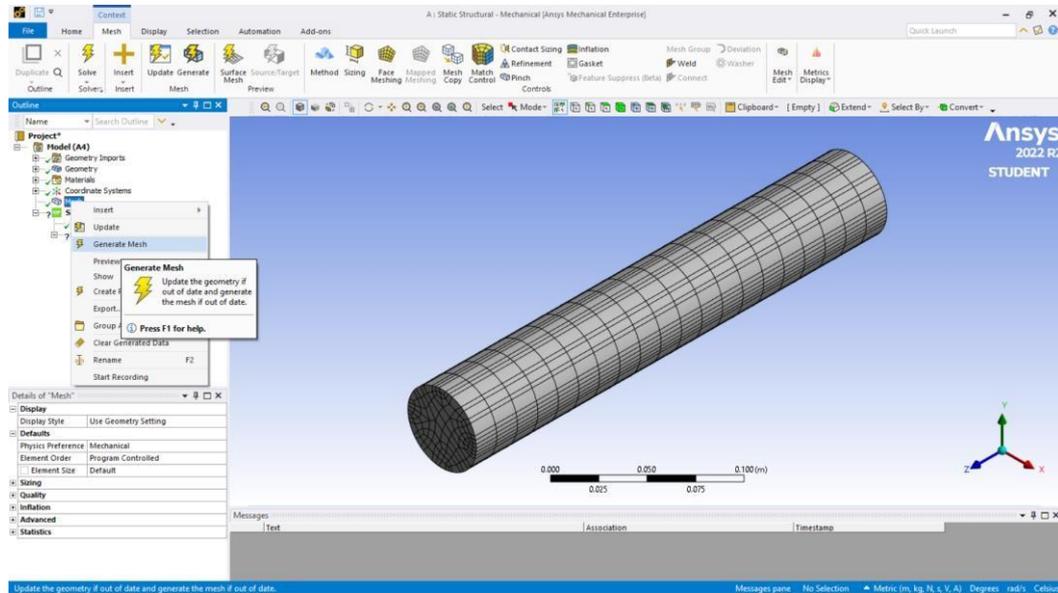


Step4: add geometry

To add geometry right click on geometry>import geometry>browse and then locate your file where you saved your geometry

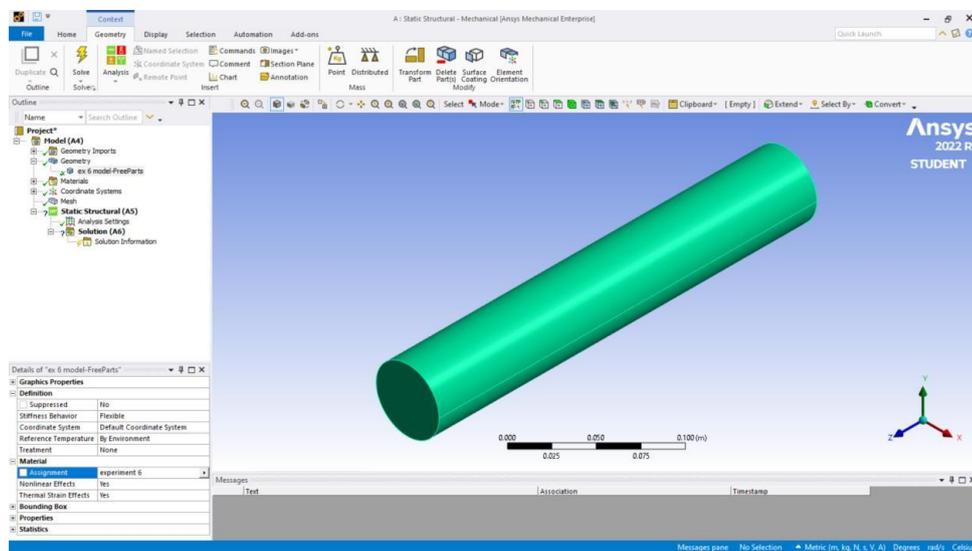
Step 5: open design modeler and generate mesh

After importing the geometry double click on the “Model ” to open design modeler. On the left side on the “outline” space right click on the “mesh” then “generate mesh”



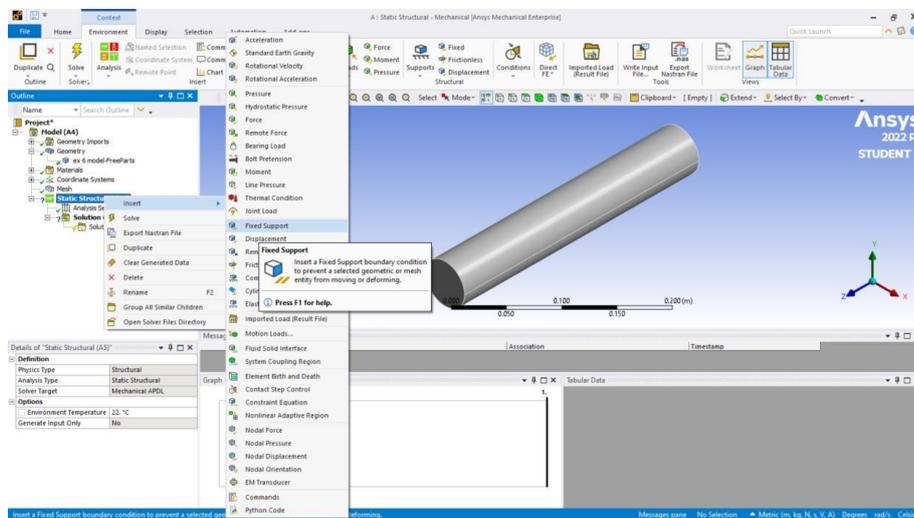
Step 6: assigning material

Click on the plus icon near “geometry” then select the geometry, then on the bottom left go to the material and change it to the defined material using the arrow on the right side

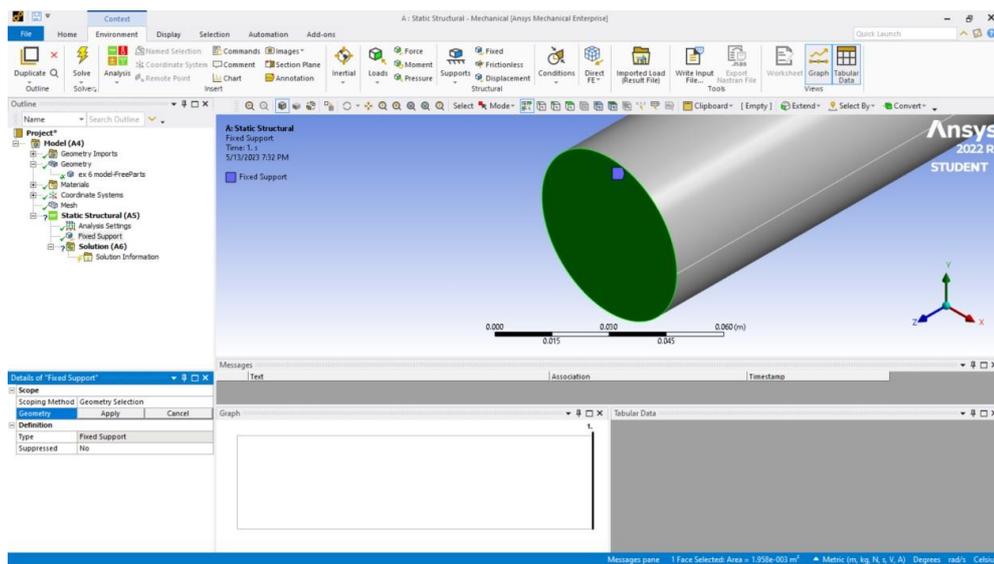


Step 7: boundary conditions

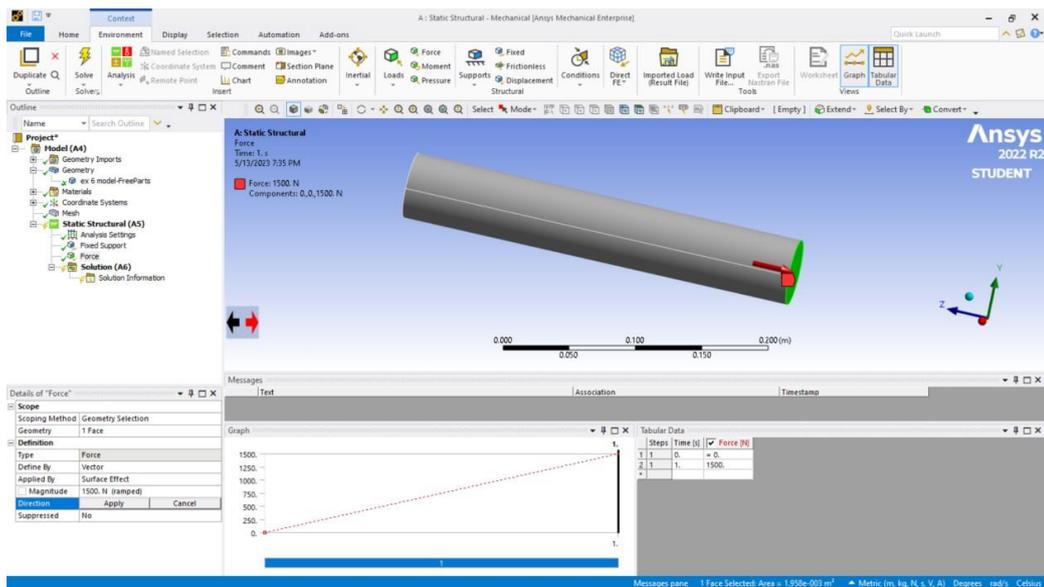
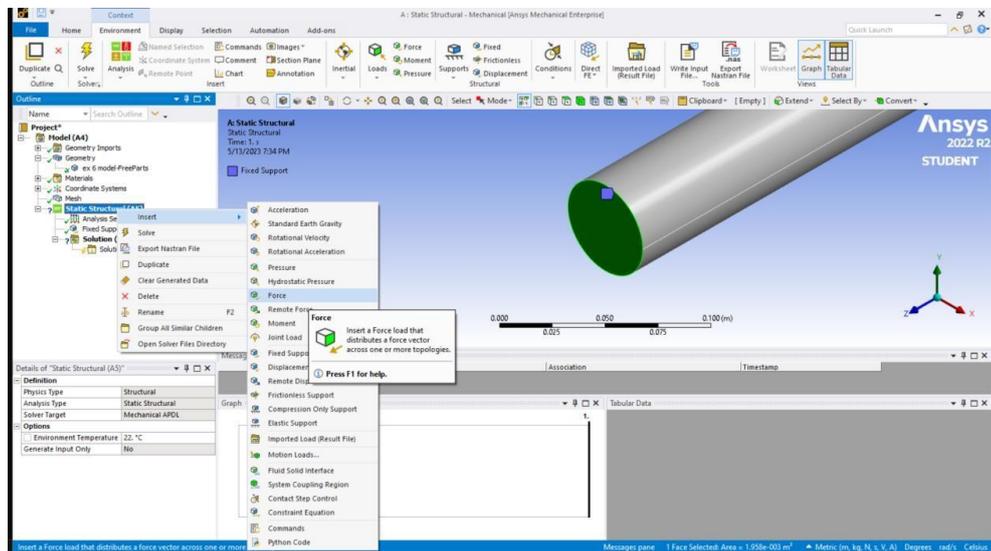
Right click on the static structural>insert then click on the “fixed support”



Then select the face and press apply to give boundary condition

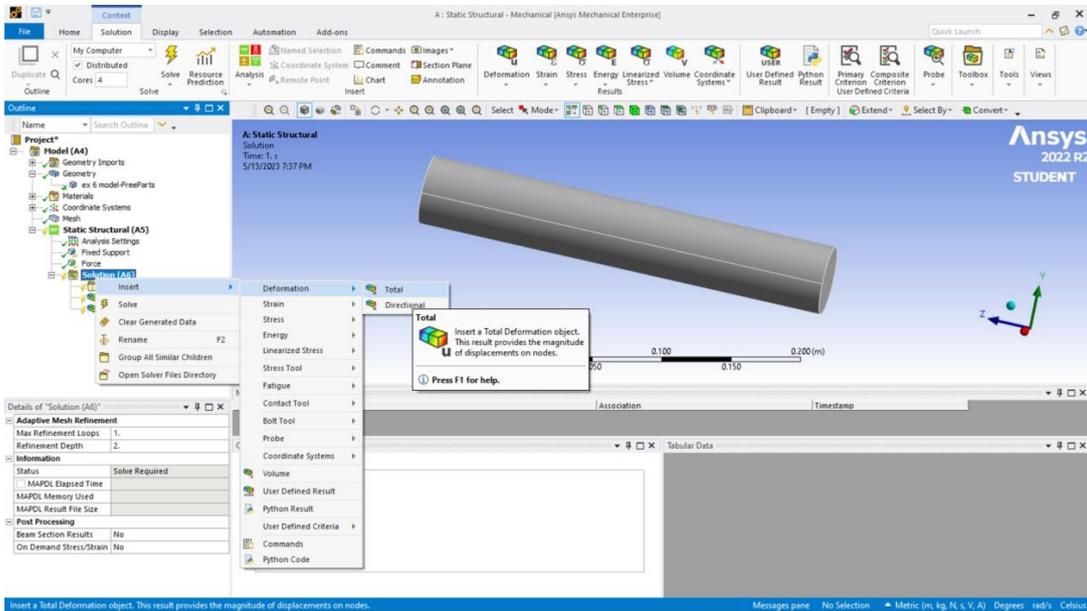


Similarly give force and the magnitude of the force in bottom left details box and set the direction of the force



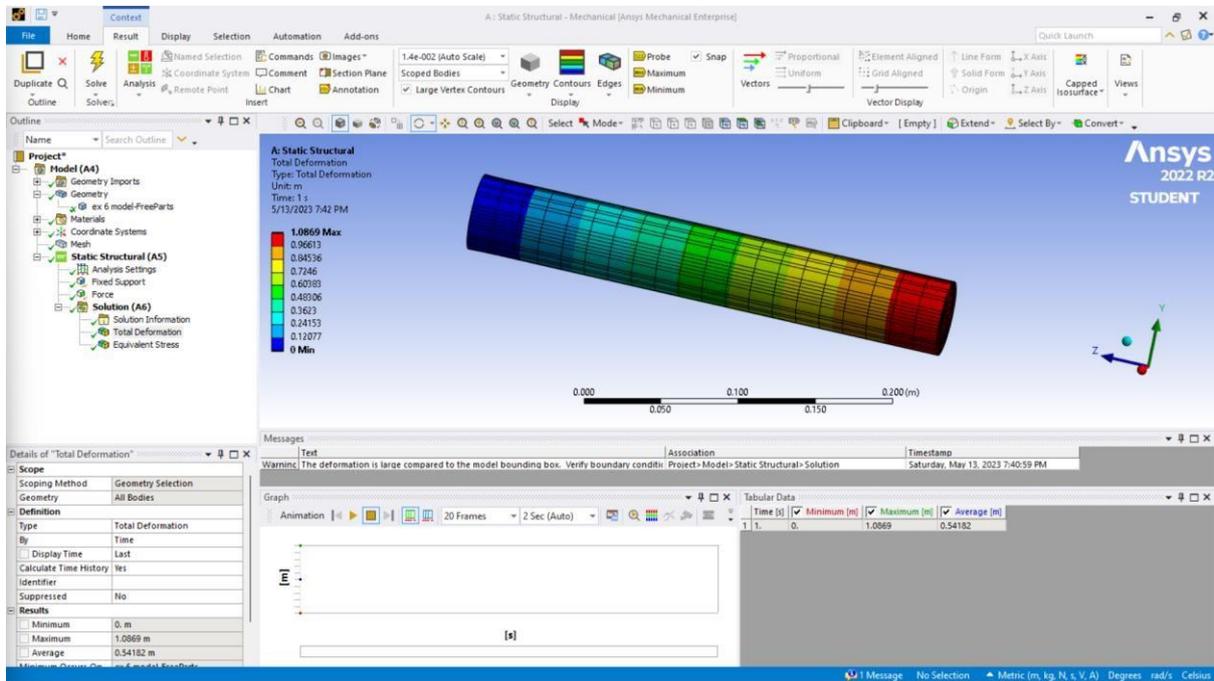
Step 8:solve the problem

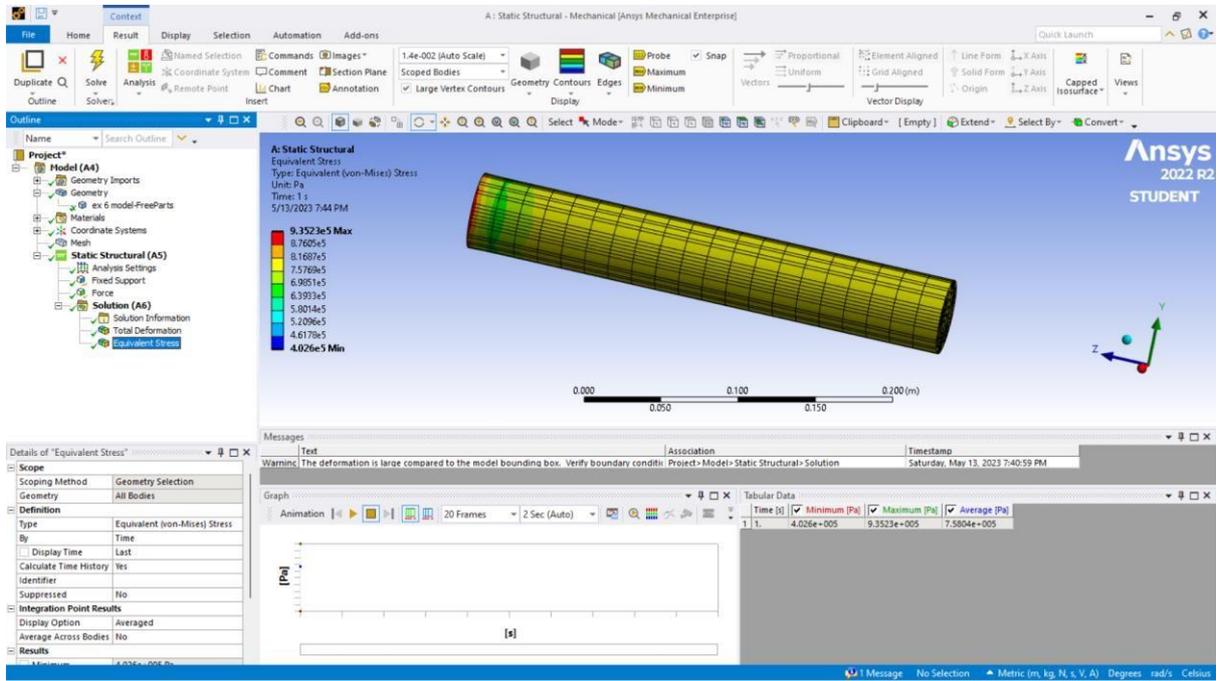
Right click on solution then insert>deformation>total deformation and insert>stress>von-mises stress after selection them right click on solution and then solve



Step 9: Examine the Results

To see the results click on the total deformation and equivalent stress to see the results

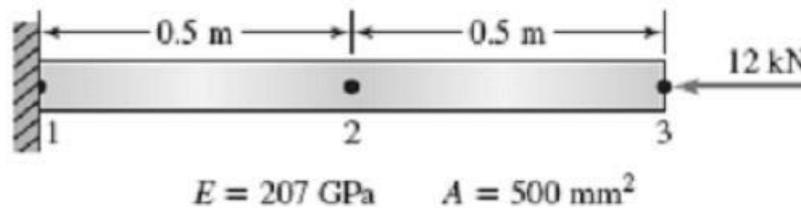




Max Stress in Element	$9.35 \times 10^5 \text{ Pa}$
Min Stress in Element	$4.026 \times 10^5 \text{ Pa}$
Total deformation	1.0869m

Problem 1:

A steel rod subjected to compression is modeled by two bar elements, as shown in Figure. . Using the steps and references learned solve the problem in Ansys Mechanical APDL. Calculate the Deformation Value, Stresses in axial direction and also capture the image of the result. $E=207\text{GPa}$, $A = 500 \text{ mm}^2$



Result Table:

	Ansys	Theoretical
Deformation		
Stress		
Reaction		

Questions:

1. Write the basic steps in Finite Element Analysis?
2. List the Finite element analysis software available commercially and freely?
3. List and draw the element types in Ansys Software with details?
4. Write down brief history of Finite Element Method (FEM)?
5. Solve the problem of 1-D Structural in Ansys or in any other analysis software?

References:

1. <https://www.youtube.com/watch?v=TWEQWnXGOHU>
2. <https://nptel.ac.in/courses/112/104/112104193/> (FEA Course)
3. <https://nptel.ac.in/courses/105/105/105105041/> (FEA Course)
4. http://mech.iust.ac.ir/files/mech/madoliat_bcc09/pdf/yijun_liu_nummeth_20040121_fem.pdf
5. <https://www.youtube.com/watch?v=v5hr0Cb9CdQ>
6. <https://www.youtube.com/watch?v=TWEQWnXGOHU>
7. https://www.cae.tntech.edu/~chriswilson/FEA/ANSYS/ANSYSguide_fea-concepts.pdf
8. <https://www.youtube.com/watch?v=AE29Dnc6j94> (In Creo)
9. https://drive.google.com/drive/folders/1-s72t2HJxD2FUk7hzsXWf_a9TUIuEzw?usp=share_link
10. <https://www.youtube.com/watch?v=DBtFzp1-EGM>
11. <https://www.ansys.com/training-center/course-catalog/structures/introduction-to-ansys-mechanical-apdl>
12. <https://www.youtube.com/watch?v=VJxxnmpVWac> (ANSYS TUTOR CHANNEL)
13. <https://www.youtube.com/watch?v=olw0ADO3RPo>
14. <https://www.youtube.com/watch?v=kx1jdxzDXVQ> (Example Video for Help)

Sign of Faculty/Lab in charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 07

Exercise for FEA of trusses

Aim: Students have to solve the truss structural problem in ANSYS Software.

Introduction:

A truss is a structure that consists of members organized into connected triangles so that the overall assembly behaves as a single object. Trusses are most commonly used in bridges, roofs and towers. A truss is made up of a web of triangles joined together to enable the even distribution of weight and the handling of changing tension and compression without bending or shearing. The triangle is geometrically stable when compared to a four (or more) -sided shape which requires that the corner joints are fixed to prevent shearing.

Trusses consist of triangular units constructed with straight members. The ends of these members are connected at joints, known as nodes. They are able to carry significant loads, transferring them to supporting structures such as load-bearing beams, walls or the ground.

In general, trusses are used to:

- Achieve long spans.
- Minimize the weight of a structure.
- Reduced deflection.
- Support heavy loads.

Trusses are typically made up of three basic elements:

- A top chord which is usually in compression.
- A bottom chord which is usually in tension.
- Bracing between the top and bottom chords.

The top and bottom chords of the truss provide resistance to compression and tension and so resistance to overall bending, whilst the bracing resists shear forces. The efficiency of trusses means that they require less material to support loads compared with solid beams. Generally, the overall efficiency of a truss is optimized by using less material in the chords and more in the bracing elements.

Trusses are structures that are composed entirely of two-force members. Each member of a truss is assumed to be a straight member that can only have forces applied on the ends of that member. The ends are pinned together so that they allow rotation. A simple truss might look like this:

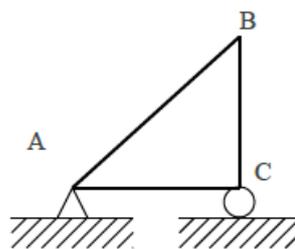


Fig 7.1 Truss

Truss



Fig7.1 Truss Example (Application) Ref: <https://www.civil.iitb.ac.in/~minamdar>

Problem Description:

Determine the nodal deflections, reaction forces, and stress for the truss system shown below ($E = 200\text{GPa}$, $A = 3250\text{mm}^2$).

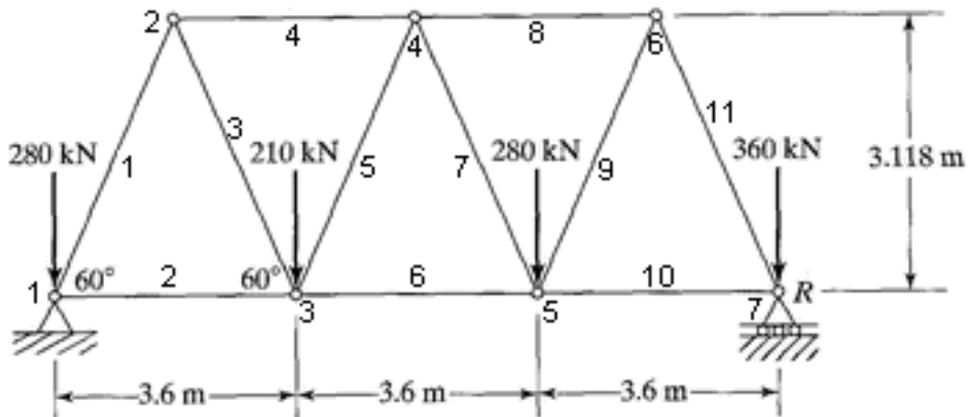


Fig7.2 Truss problem with dimensions

(Ref: Chandrupatla & Belegunda, Introduction to Finite Elements in Engineering)

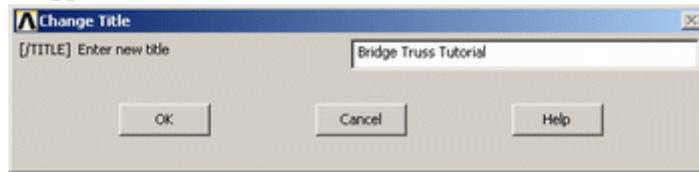
Preprocessing: Defining the Problem

1. Give the Simplified Version a Title (such as 'Bridge Truss Tutorial').

In the Utility menu bar select File > Change Title:



The following window will appear:



Enter the title and click 'OK'. This title will appear in the bottom left corner of the 'Graphics' Window once you begin.

Note: to get the title to appear immediately, select **Utility Menu > Plot > Replot**

2. Enter Keypoints

The overall geometry is defined in ANSYS using key points which specify various principal coordinates to define the body. For this example, these key points are the ends of each truss.

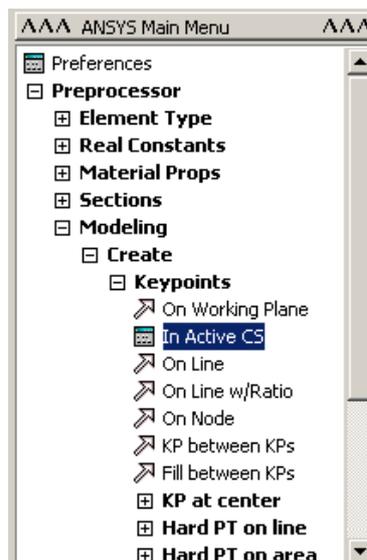
We are going to define 7 key points for the simplified structure as given in the following table

keypoint	coordinate	
	x	y
1	0	0
2	1800	3118
3	3600	0
4	5400	3118
5	7200	0
6	9000	3118
7	10800	0

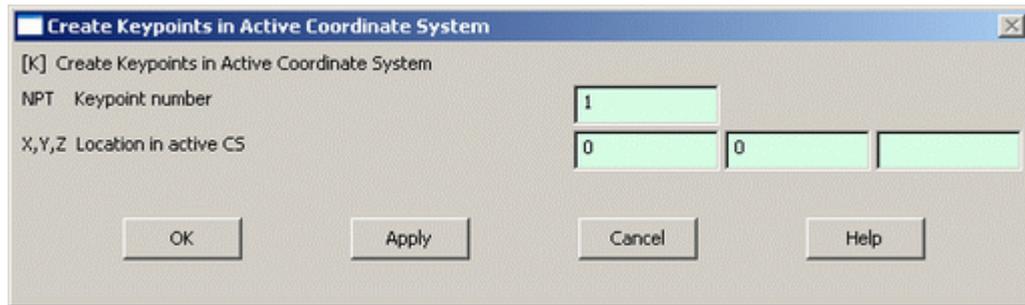
Table 7.1 – Key points coordinates

From the 'ANSYS Main Menu' select:

Preprocessor > Modeling > Create > Keypoints > In Active CS



The following window will then appear:



To define the first key point which has the coordinates $x = 0$ and $y = 0$:

Enter key point number 1 in the appropriate box, and enter the x,y coordinates: 0, 0 in their appropriate boxes (as shown above).

Click 'Apply' to accept what you have typed.

Enter the remaining key points using the same method.

Note: When entering the final data point, click on 'OK' to indicate that you are finished entering key points. If you first press 'Apply' and then 'OK' for the final key point, you will have defined it twice! If you did press 'Apply' for the final point, simply press 'Cancel' to close this dialog box.

Units

Note the units of measure (ie mm) were not specified. It is the responsibility of the user to ensure that a consistent set of units are used for the problem; thus making any conversions where necessary.

Correcting Mistakes

When defining key points, lines, areas, volumes, elements, constraints and loads you are bound to make mistakes. Fortunately these are easily corrected so that you don't need to begin from scratch every time an error is made! Every 'Create' menu for generating these various entities also has a corresponding 'Delete' menu for fixing things up.

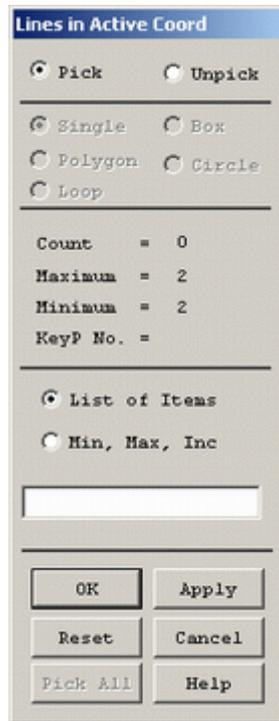
3. Form Lines

The key points must now be connected

We will use the mouse to select the key points to form the lines.

In the main menu select: **Preprocessor > Modeling > Create > Lines > Lines > In Active Coord.**

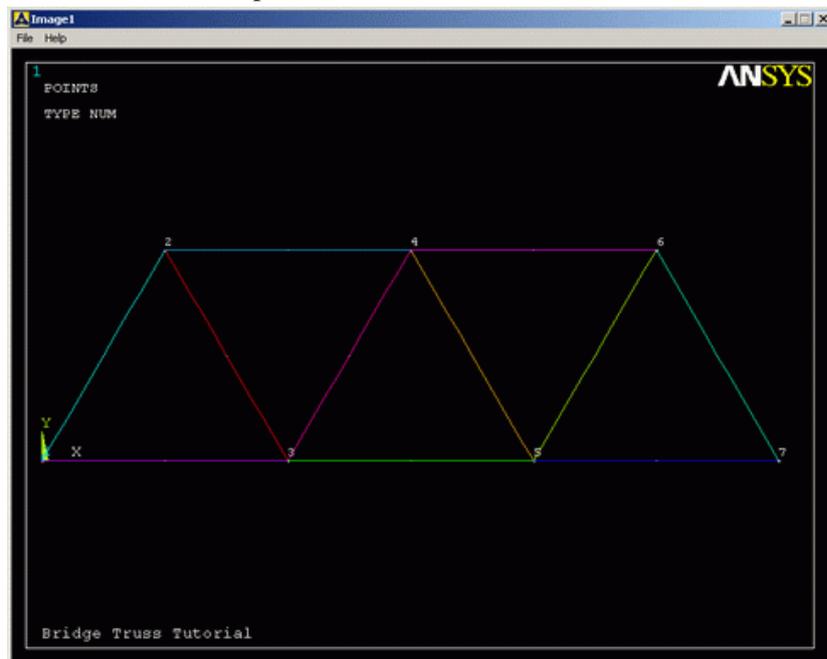
The following window will then appear:



Now move the mouse toward key point #2.

A line will now show on the screen joining these two points. Left click and a permanent line will appear. Connect the remaining key points using the same method.

When you're done, click on 'OK' in the 'Lines in Active Coord' window, minimize the 'Lines' menu and the 'Create' menu. Your ANSYS Graphics window should look similar to the following figure.

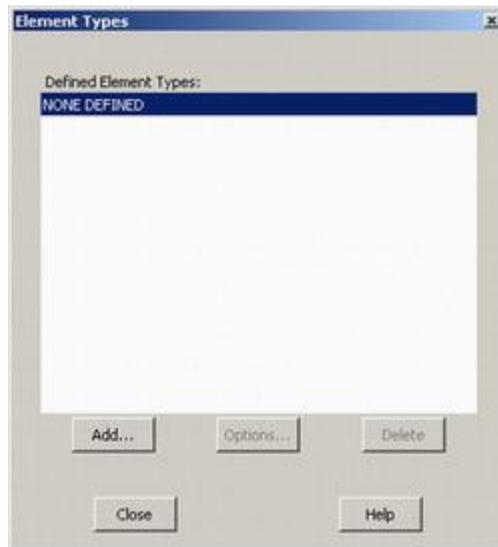


Disappearing Lines

Please note that any lines you have created may 'disappear' throughout your analysis. However, they have most likely **NOT** been deleted. If this occurs at any time from the **Utility Menu** select: **Plot > Lines**

4. Define the Type of Element

It is now necessary to create elements. This is called 'meshing'. ANSYS first needs to know what kind of elements to use for our problem: From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**. The following window will then appear:



Click on the 'Add...' button. The following window will appear:



For this example, we will use the 2D spar element as selected in the above figure. Select the Element shown and click 'OK'. You should see 'Type 1 LINK1' in the 'Element Types' window. Click on 'Close' in the 'Element Types' dialog box.

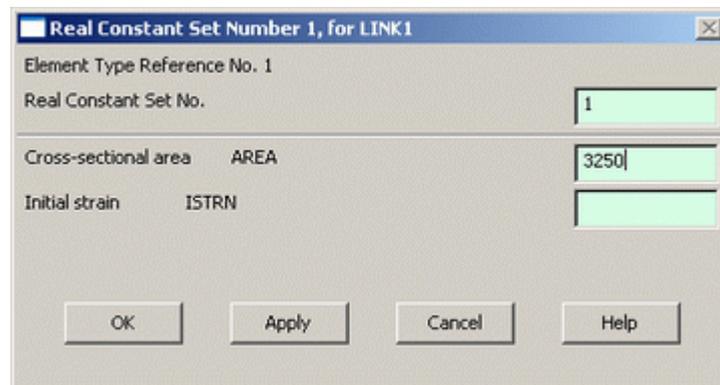
5. Define Geometric Properties

We now need to specify geometric properties for our elements:

In the Preprocessor menu, select **Real Constants > Add/Edit/Delete**



Click **Add...** and select 'Type 1 LINK1' (actually it is already selected). Click on 'OK'. The following window will appear:



Above window is optional, in newer version not require based on element selection type. As shown in the window above, enter the cross-sectional area (3250mm):

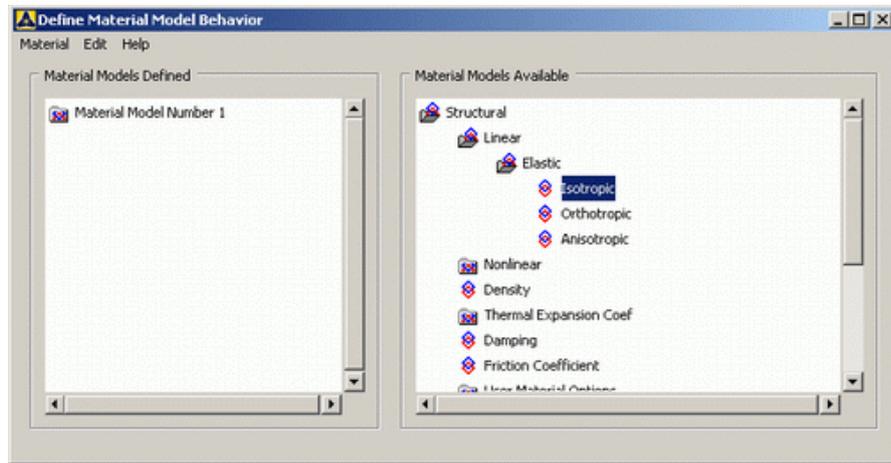
Click on 'OK'.

'Set 1' now appears in the dialog box. Click on 'Close' in the 'Real Constants' window.

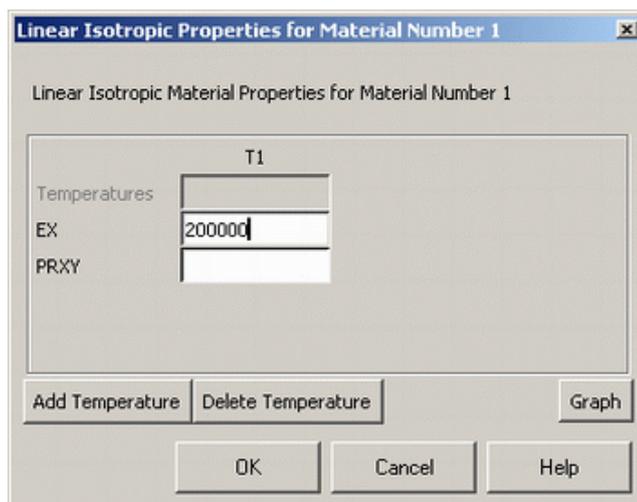
6. Element Material Properties

You then need to specify material properties:

In the 'Preprocessor' menu select **Material Props > Material Models**



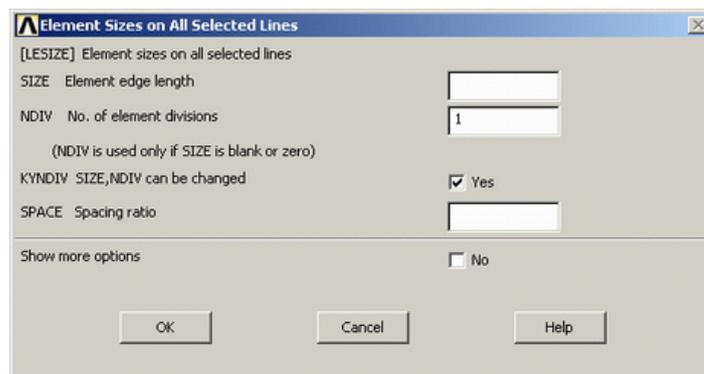
Double click on **Structural > Linear > Elastic > Isotropic**



We are going to give the properties of Steel. Enter the following field: $EX = 2e5$
 Set these properties and click on 'OK'. Note: You may obtain the note 'PRXY will be set to 0.0'. This is poisson's ratio and is not required for this element type. Click 'OK' on the window to continue. Close the "Define Material Model Behavior" by clicking on the 'X' box in the upper right hand corner.

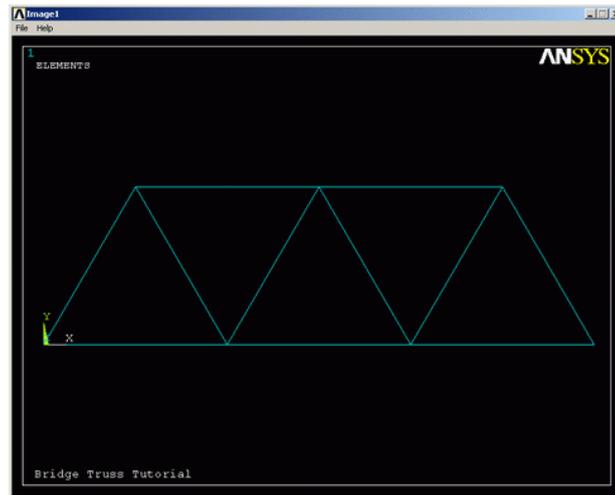
7. Mesh Size

The last step before meshing is to tell ANSYS what size the elements should be. There are a variety of ways to do this but we will just deal with one method for now. □ In the Preprocessor menu select **Meshing > Size Cntrls > ManualSize > Lines > All Lines**



In the size 'NDIV' field, enter the desired number of divisions per line. For this example we want only 1 division per line, therefore, enter '1' and then click 'OK'. Note that we have not yet meshed the geometry, we have simply defined the element sizes.

8. Mesh Now the frame can be meshed. In the 'Preprocessor' menu select **Meshing > Mesh > Lines** and click 'Pick All' in the 'Mesh Lines' Window Your model should now appear as shown in the following

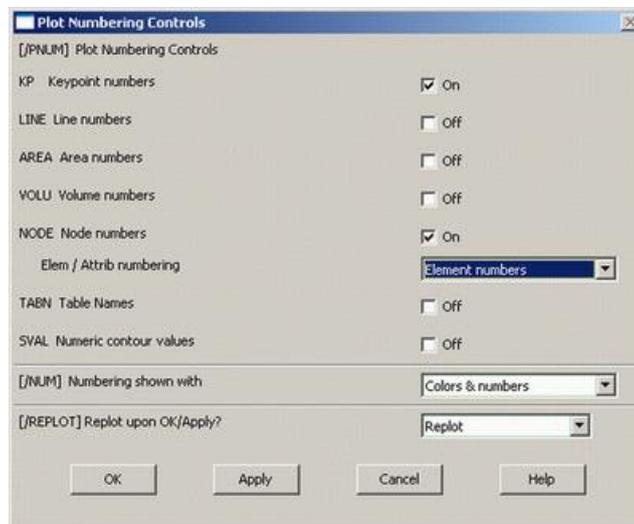


Plot Numbering

To show the line numbers, keypoint numbers, node numbers...

From the **Utility Menu** (top of screen) select **PlotCtrls > Numbering...**

Fill in the Window as shown below and click 'OK'



Now you can turn numbering on or off at your discretion **Saving Your Work**

Save the model at this time, so if you make some mistakes later on, you will at least be able to come back to this point. To do this, on the **Utility Menu** select **File > Save as....** Select the name and location where you want to save your file.

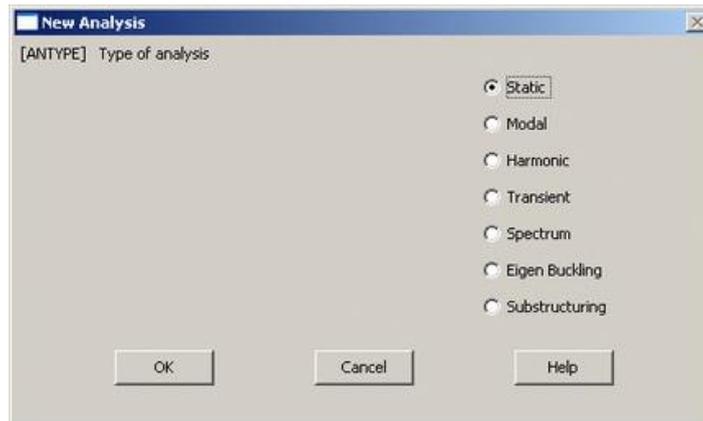
It is a good idea to save your job at different times throughout the building and analysis of the model to backup your work in case of a system crash or what have you.

Solution Phase: Assigning Loads and Solving

You have now defined your model. It is now time to apply the load(s) and constraint(s) and solve the the resulting system of equations. Open up the 'Solution' menu (from the same 'ANSYS Main Menu').

1. Define Analysis Type

First you must tell ANSYS how you want it to solve this problem:
From the **Solution** Menu, select **Analysis Type > New Analysis**.

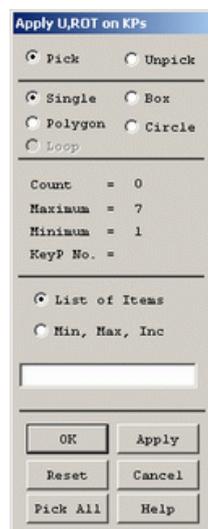


Ensure that 'Static' is selected; i.e. you are going to do a static analysis on the truss as opposed to a dynamic analysis, for example.
Click 'OK'.

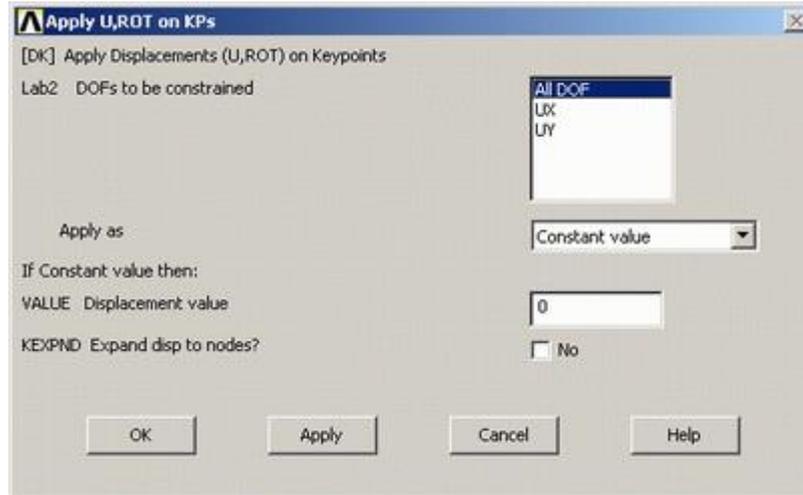
2. Apply Constraints

It is necessary to apply constraints to the model otherwise the model is not *tied down* or *grounded* and a singular solution will result. In mechanical structures, these constraints will typically be fixed, pinned and roller-type connections. As shown above, the left end of the truss bridge is pinned while the right end has a roller connection.

In the **Solution** menu, select **Define Loads > Apply > Structural > Displacement > On Keypoints**



Select the left end of the bridge (Keypoint 1) by clicking on it in the Graphics Window and click on 'OK' in the 'Apply U,ROT on KPs' window.



This location is fixed which means that all translational and rotational degrees of freedom (DOFs) are constrained. Therefore, select 'All DOF' by clicking on it and enter '0' in the Value field and click 'OK'.

You will see some blue triangles in the graphics window indicating the displacement constraints.

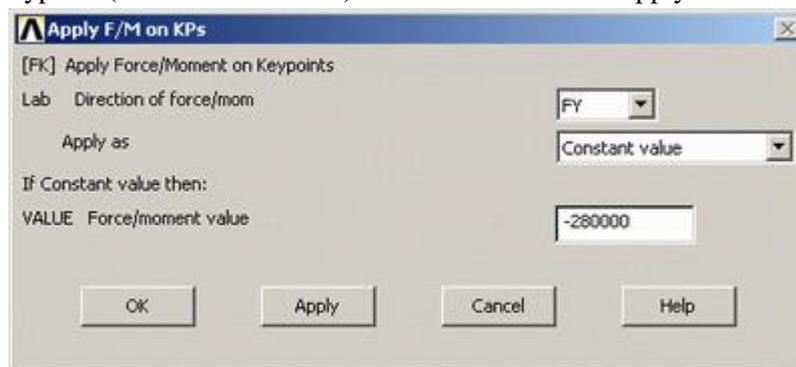
Using the same method, apply the roller connection to the right end (UY constrained). Note that more than one DOF constraint can be selected at a time in the "Apply U,ROT on KPs" window. Therefore, you may need to 'deselect' the 'All DOF' option to select just the 'UY' option.

3. Apply Loads

As shown in the diagram, there are four downward loads of 280kN, 210kN, 280kN, and 360kN at keypoints 1, 3, 5, and 7 respectively.

Select **Define Loads > Apply > Structural > Force/Moment > on Keypoints**.

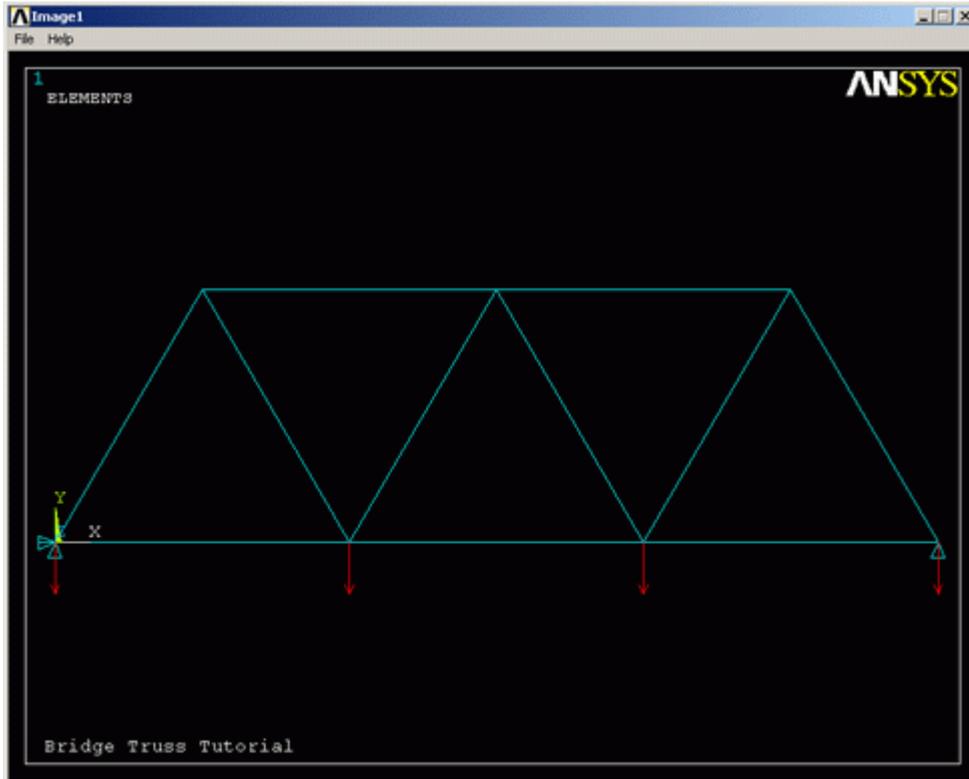
Select the first Keypoint (left end of the truss) and click 'OK' in the 'Apply F/M on KPs' window.



Select FY in the 'Direction of force/mom'. This indicate that we will be applying the load in the 'y' Direction Enter a value of -280000 in the 'Force/moment value' box and click 'OK'. Note that we are using units of N here, this is consistent with the previous values input.

The force will appear in the graphics window as a red arrow. Apply the remaining loads in the same manner.

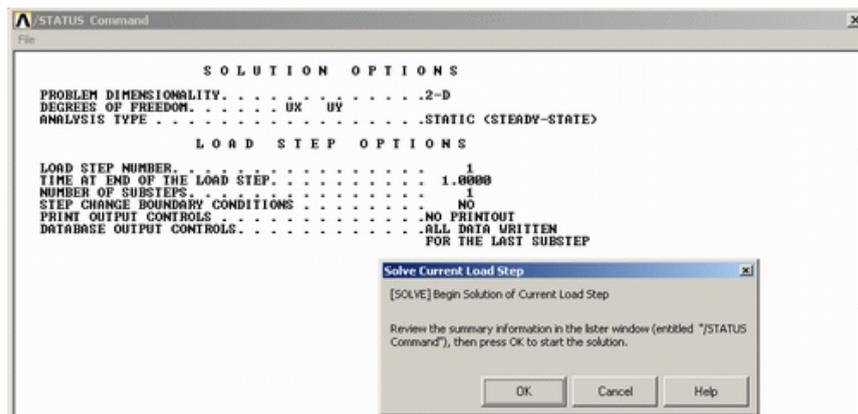
The applied loads and constraints should now appear as shown below.



4. Solving the System

We now tell ANSYS to find the solution:

In the 'Solution' menu select **Solve > Current LS**. This indicates that we desire the solution under the current Load Step (LS).



The above windows will appear. Ensure that your solution options are the same as shown above and click 'OK'.

Once the solution is done the following window will pop up. Click 'Close' and close the /STATUS



1. Hand Calculations

We will first calculate the forces and stress in element 1 (as labeled in the problem description).

$$\begin{aligned} \circlearrowleft \sum M_1 = 0 &= -210 \text{ kN}(3.6 \text{ m}) - 280 \text{ kN}(7.2 \text{ m}) - 360 \text{ kN}(10.8 \text{ m}) + F_7(10.8 \text{ m}) \\ F_7 &= \frac{210 \text{ kN}(3.6 \text{ m}) + 280 \text{ kN}(7.2 \text{ m}) + 360 \text{ kN}(10.8 \text{ m})}{10.8 \text{ m}} = 617 \text{ kN} \end{aligned}$$

$$\begin{aligned} \uparrow \sum F_y = 0 &= -280 \text{ kN} - 210 \text{ kN} - 280 \text{ kN} - 360 \text{ kN} + 617 \text{ kN} + F_1 \\ F_1 &= 280 \text{ kN} + 210 \text{ kN} + 280 \text{ kN} + 360 \text{ kN} - 617 \text{ kN} = 513 \text{ kN} \end{aligned}$$

Element 1 Forces/Stress

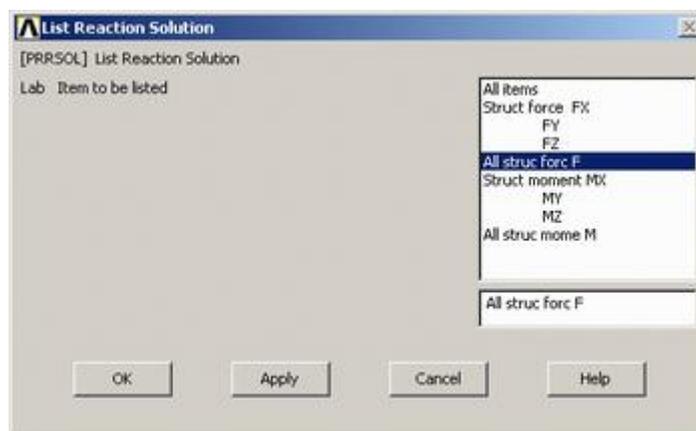
$$\begin{aligned} F_{E1} &= \frac{513 \text{ kN} - 280 \text{ kN}}{\cos(30)} = 269 \text{ kN} \\ \sigma_{E1} &= \frac{F_{E1}}{A} = \frac{269 \text{ kN}}{3250 \text{ mm}^2} = 82.8 \text{ MPa} \end{aligned}$$

2. Results Using ANSYS

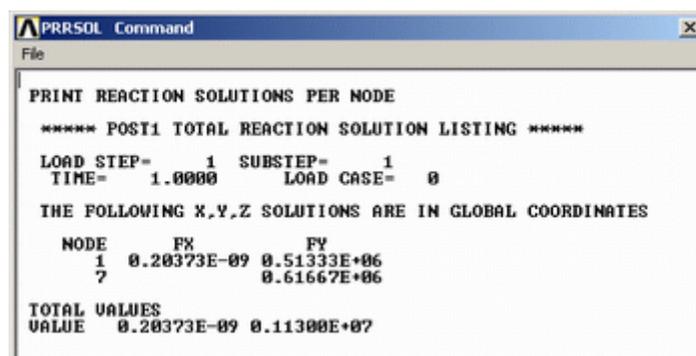
Reaction Forces

A list of the resulting reaction forces can be obtained for this element

- from the Main Menu select **General Postproc > List Results > Reaction Solution**



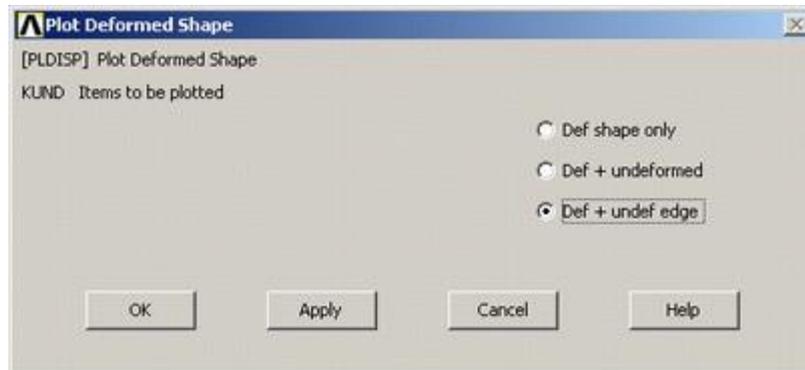
Select 'All struc forc F' as shown above and click 'OK'



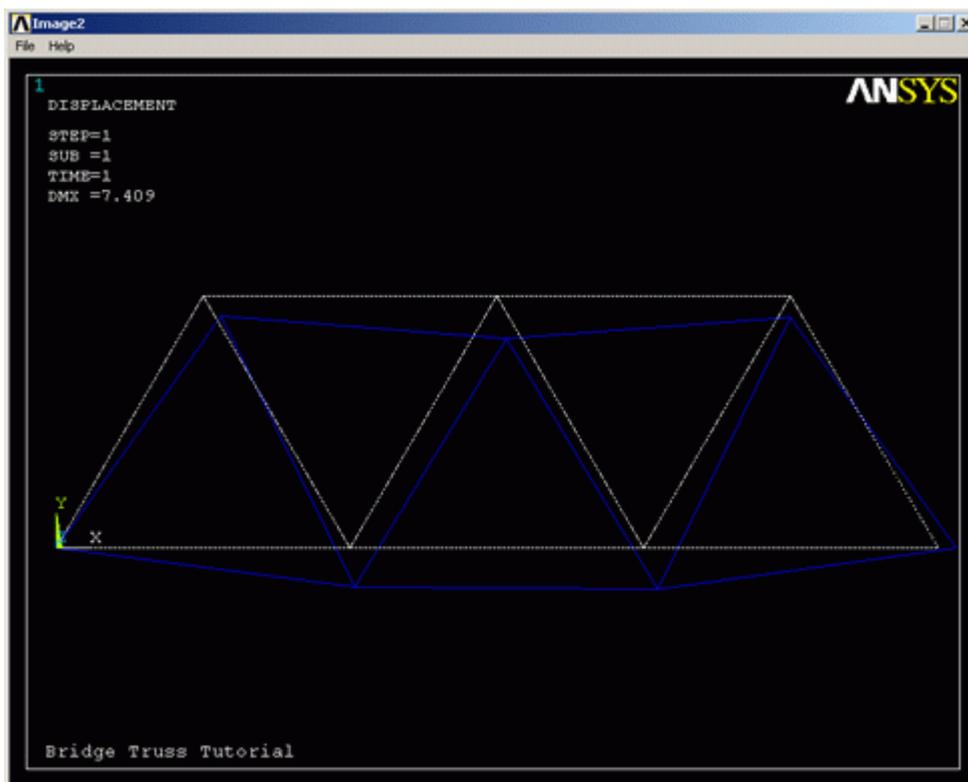
These values agree with the reaction forces calculated by hand above.

Deformation

In the General Postproc menu, select **Plot Results > Deformed Shape**. The following window will appear.



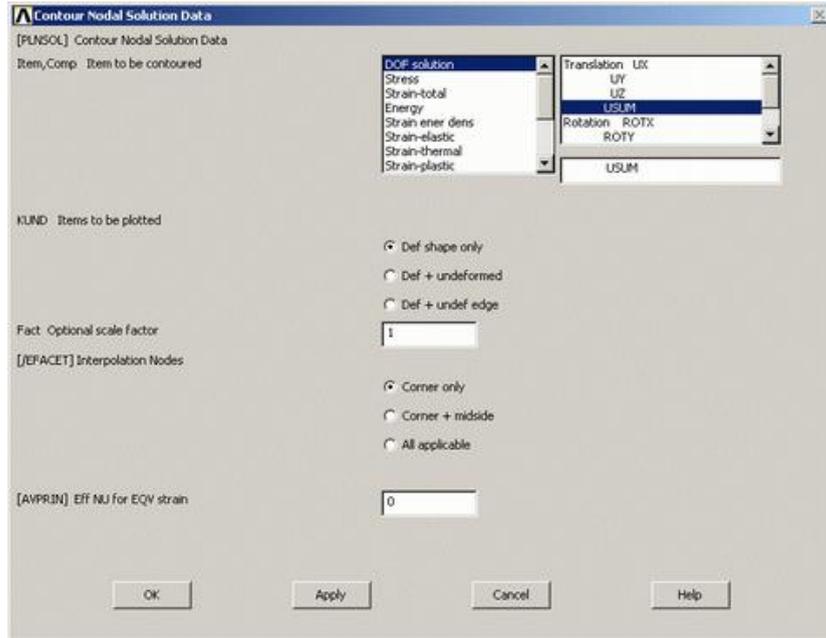
Select 'Def + undef edge' and click 'OK' to view both the deformed and the un deformed object.



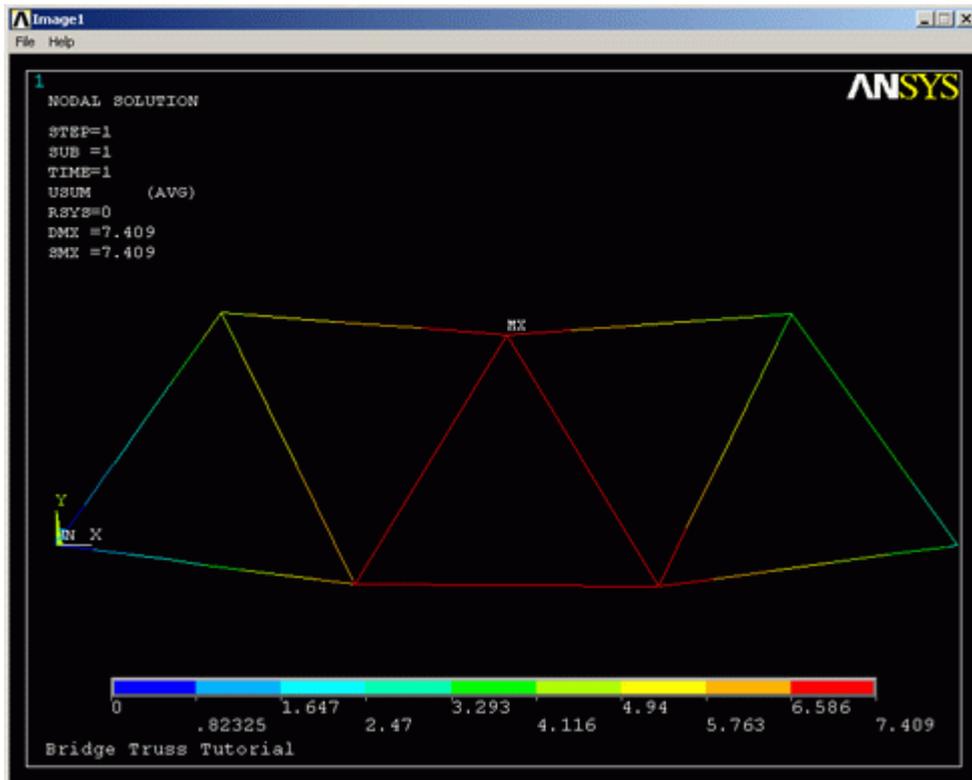
Observe the value of the maximum deflection in the upper left hand corner (DMX=7.409). One should also observe that the constrained degrees of freedom appear to have a deflection of 0 (as expected!)

Deflection

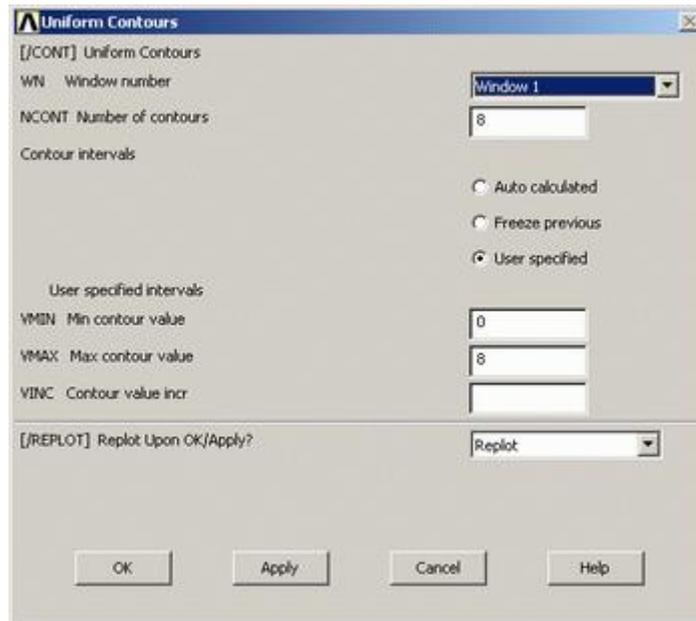
For a more detailed version of the deflection of the beam,
From the 'General Postproc' menu select **Plot results > Contour Plot > Nodal Solution**. The following window will appear.



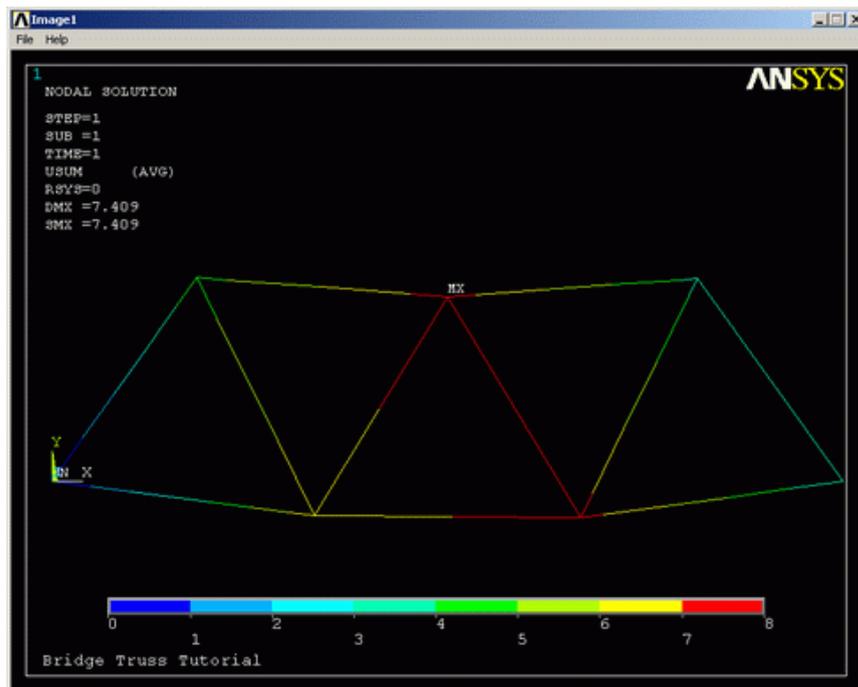
Select 'DOF solution' and 'USUM' as shown in the above window. Leave the other selections as the default values. Click 'OK'.



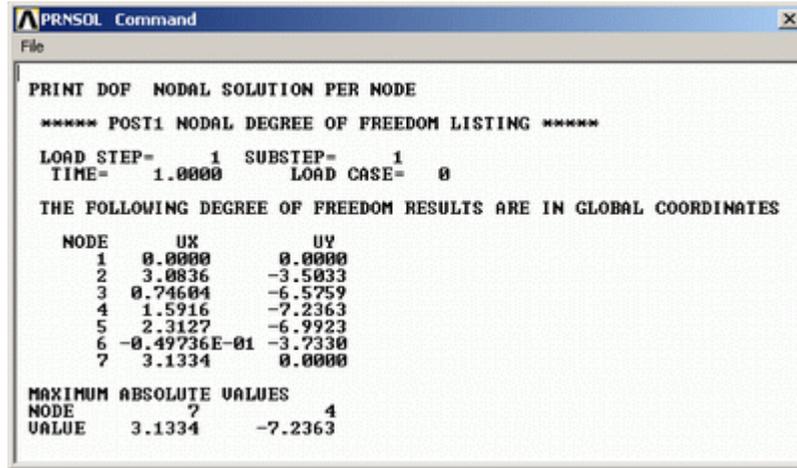
Looking at the scale, you may want to use more useful intervals. From the **Utility Menu** select **Plot Controls > Style > Contours > Uniform Contours...**
Fill in the following window as shown and click 'OK'.



You should obtain the following.



The deflection can also be obtained as a list as shown below. **General Postproc > List Results > Nodal Solution** select 'DOF Solution' and 'ALL DOFs' from the lists in the 'List Nodal Solution' window and click 'OK'. This means that we want to see a listing of all degrees of freedom from the solution.



Are these results what you expected? Note that all the degrees of freedom were constrained to zero at node 1, while UY was constrained to zero at node 7.

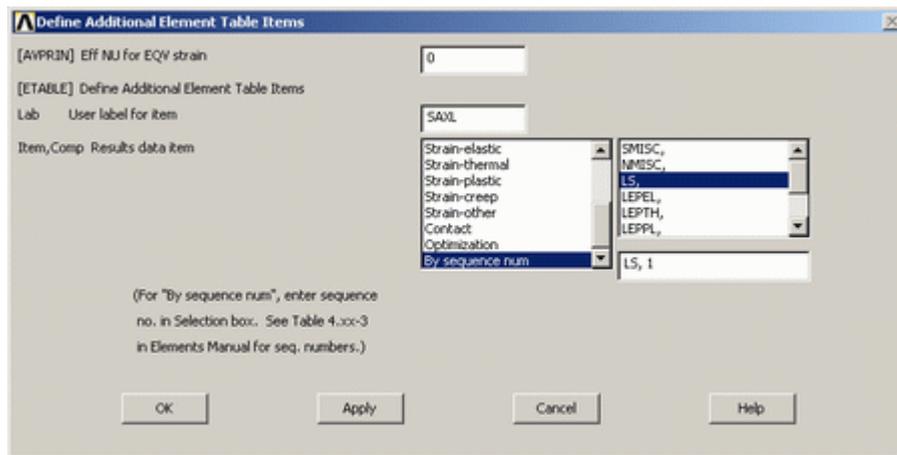
If you wanted to save these results to a file, select 'File' within the results window (at the upper left hand corner of this list window) and select 'Save as'.

Axial Stress

For line elements (ie links, beams, spars, and pipes) you will often need to use the **Element Table** to gain access to derived data (ie stresses, strains). For this example we should obtain axial stress to compare with the hand calculations. The Element Table is different for each element, therefore, we need to look at the help file for LINK1 (Type **help link1** into the Input Line). From Table 1.2 in the Help file, we can see that SAXL can be obtained through the ETABLE, using the item 'LS,1'

From the **General Postprocessor** menu select **Element Table > Define Table**

Click on 'Add...'

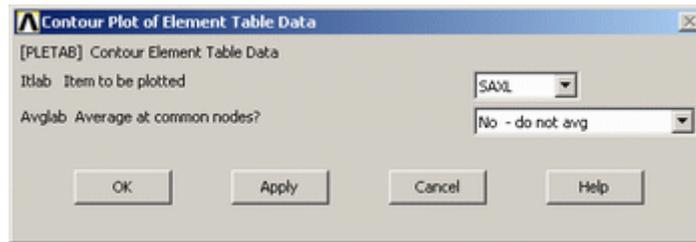


As shown above, enter 'SAXL' in the 'Lab' box. This specifies the name of the item you are defining. Next, in the 'Item,Comp' boxes, select 'By sequence number' and 'LS,.'. Then enter 1 after LS, in the selection box

Click on 'OK' and close the 'Element Table Data' window.

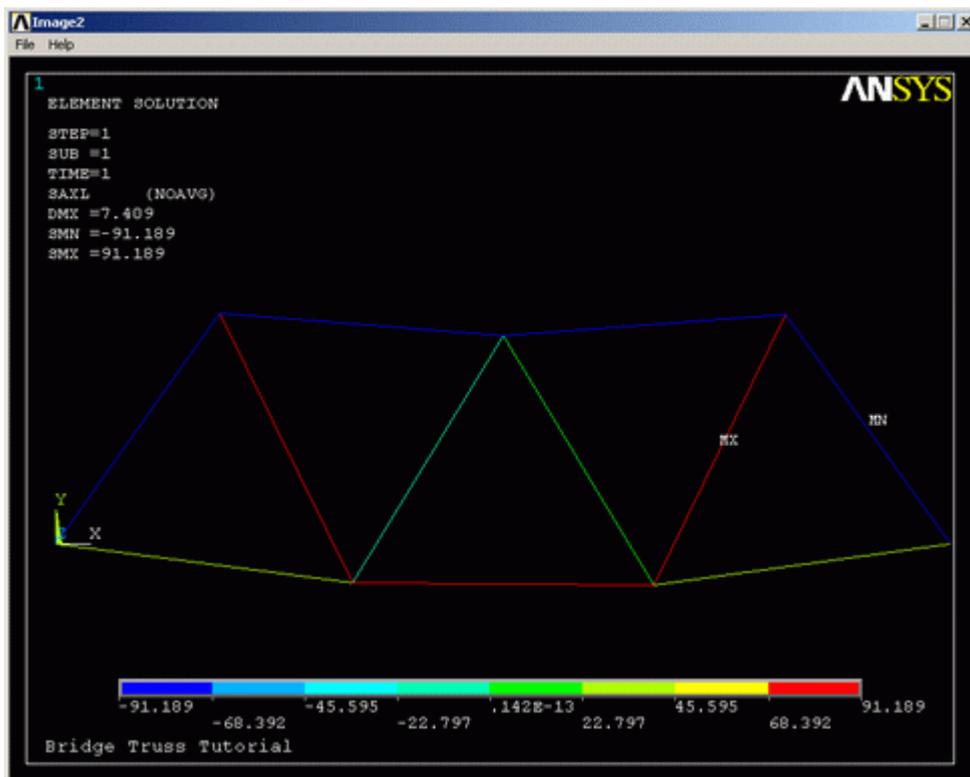
Plot the Stresses by selecting **Element Table > Plot Elem Table**

The following window will appear. Ensure that 'SAXL' is selected and click 'OK'



Because you changed the contour intervals for the Displacement plot to "User Specified" – you need to switch this back to "Auto calculated" to obtain new values for VMIN/VMAX.

Utility Menu > PlotCtrls > Style > Contours > Uniform Contours ...



Again, you may wish to select more appropriate intervals for the contour plot

List the Stresses

From the 'Element Table' menu, select '**List Elem Table**'

From the 'List Element Table Data' window which appears ensure 'SAXL' is highlighted

Click 'OK'

```

PRETAB Command
File
PRINT ELEMENT TABLE ITEMS PER ELEMENT
***** POST1 ELEMENT TABLE LISTING *****
STAT      CURRENT
ELEM      SAKL
 1      -82.900
 2      41.447
 3      82.900
 4      -82.893
 5      -8.2900
 6      87.038
 7      8.2900
 8      -91.183
 9      91.189
10      45.591
11      -91.189

MINIMUM VALUES
ELEM      11
VALUE     -91.189

MAXIMUM VALUES
ELEM      9
VALUE     91.189

```

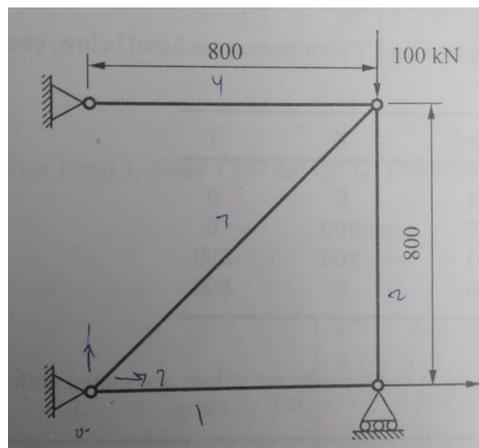
Note that the axial stress in Element 1 is 82.9MPa as predicted analytically. The above are the steps to solve the Truss problem in Ansys using the APDL module. Try to solve any two book problem in Ansys

Instructions:

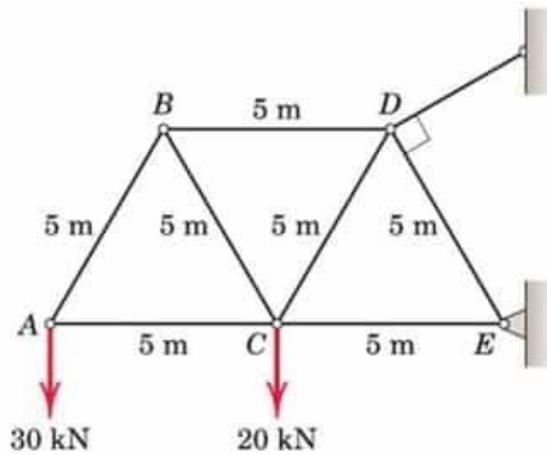
Students are informed to attend the video/practical hands on session in CAD laboratory for learning, how to create and analyze the truss in the software. According to the steps mention above practice the truss in Ansys software.

Questions:

1. A four bar truss as shown in figure below. Assuming that each element having the cross-sectional area is 400mm² and modulus of elasticity is 200GPa. Determine the deflections, reaction forces and stresses in each elements.



2. Consider the similar data of above question and solve the following truss problem



3. List the types of trusses and its brief?

References:

1. <https://www.youtube.com/watch?v=cEr9MnnXME0> (2D truss problem in Workbench)
2. <https://www.youtube.com/watch?v=IXFN0-yLAI0> (Truss analysis using the APDL Ansys)
3. <http://web.engr.uky.edu/~gebland/CE%20382/CE%20382%20Four%20Slides%20per%20Page/L5%20-%20Truss%20Structures.pdf>
4. <http://www.steel-insdag.org/TeachingMaterial/chapter27.pdf>
5. <https://www.civil.iitb.ac.in/~minamdar/ce102/Files/Trusses.pdf>
6. <https://www.sjsu.edu/people/steven.vukazich/docs/160.4.1%20Trusses.pdf>
7. <https://www.youtube.com/watch?v=s3xKv5QussM>
8. https://www.youtube.com/watch?v=lz11_NxgsaY

Sign of Faculty/Lab In charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 08

Exercise for FEA using Beam Element

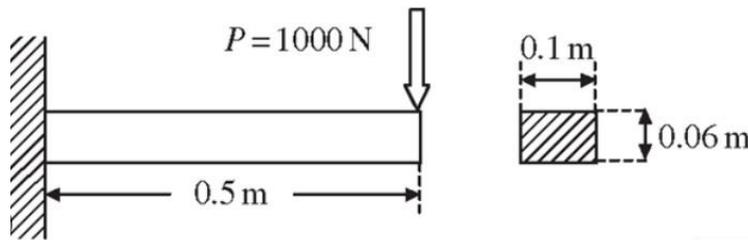
Aim: Students have to solve the cantilever beam structural problem in ANSYS Software

Objective:

- Create beam-meshable geometry.
- Create standard beam cross-sections.
- Create beam meshes.
- Define beam offset and orientation.
- Override the mesh display.
- Create a local coordinate system.
- Request element forces and solve the model.
- Calculate complete beam stress components.
- Create a beam cross-section display

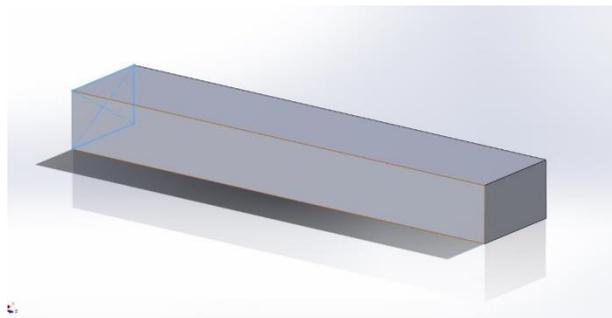
Problem Statement:

Solve the following data for Finite Element Analysis of a Cantilever beam:
Material : structural steel , find total deformation and stress



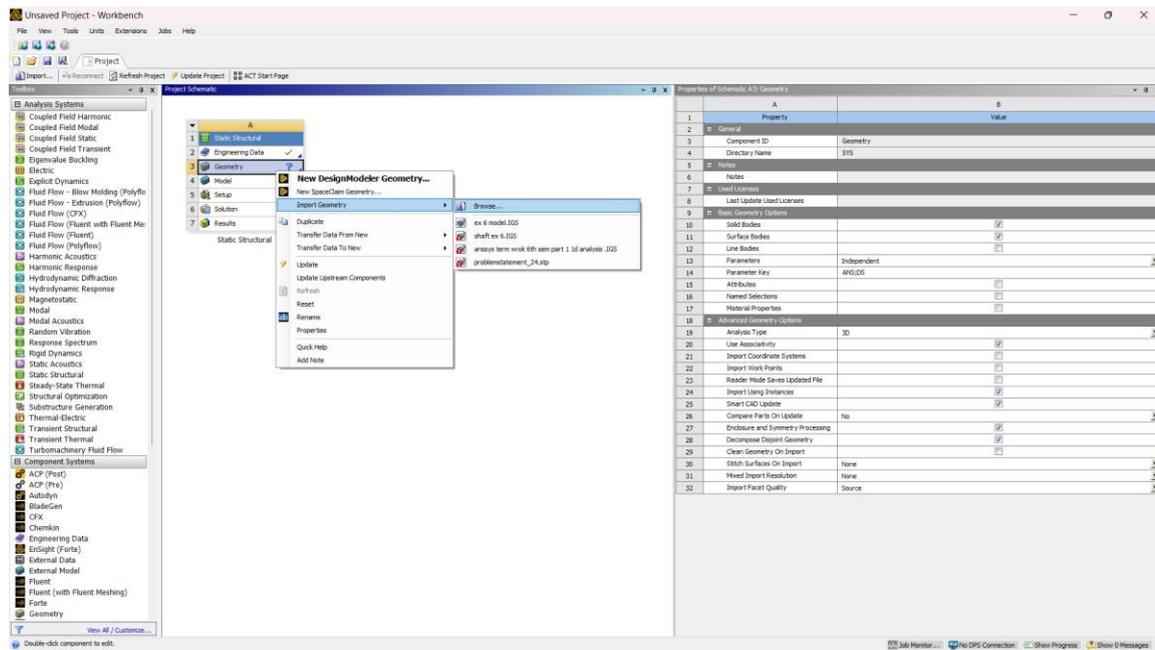
Step1:

Create the given geometry in any modeling software of your choice and save the file in IGES/IGS format.



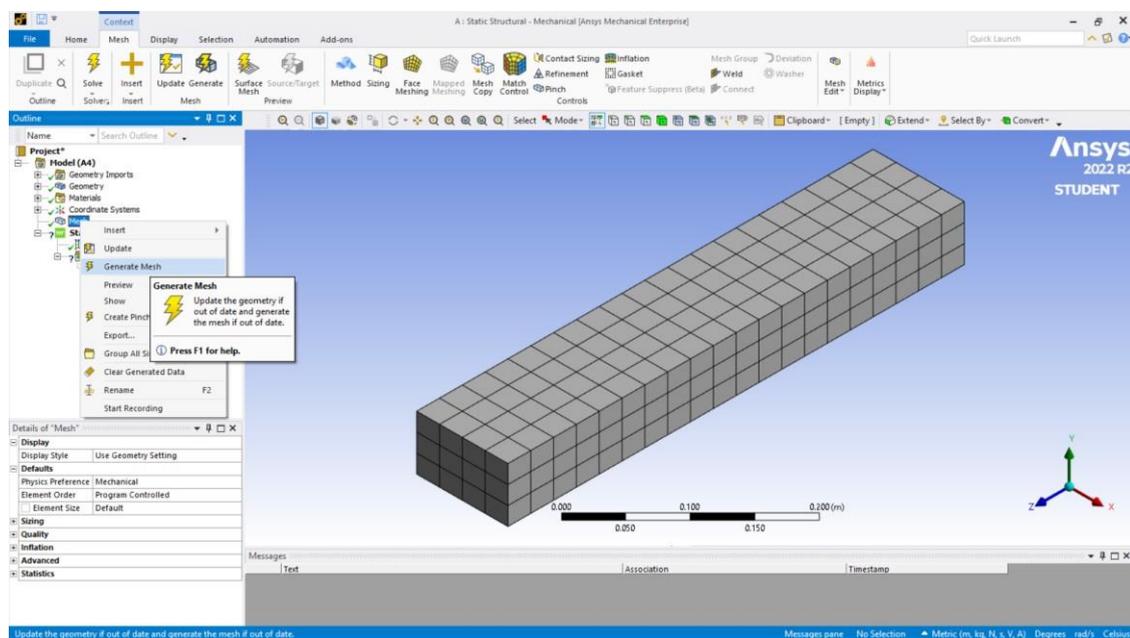
Step 2:

open ansys workbench, drag and drop the static structural module onto project schematic and import the geometry(because static structural is default material we do not have to select material form engineering data)



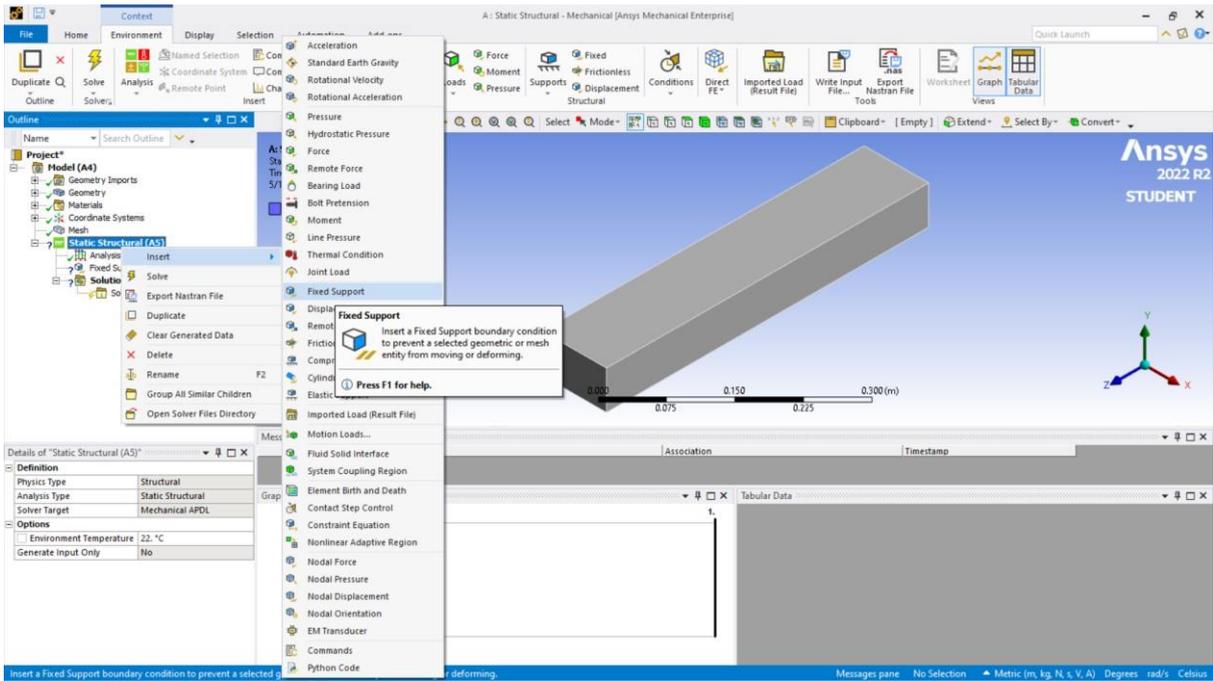
Step 3 :

open the modeler and right click on the “mesh” icon, left side on the outline tree, and then click on “generate mesh”

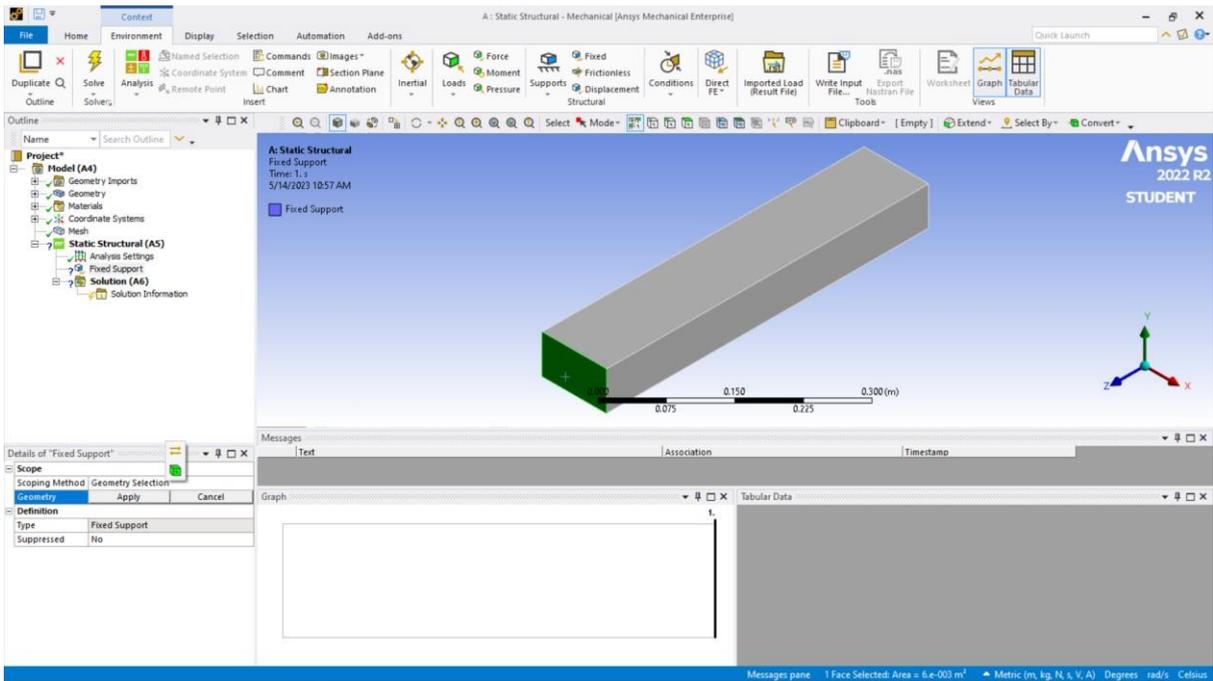


Step 4 : boundary conditions

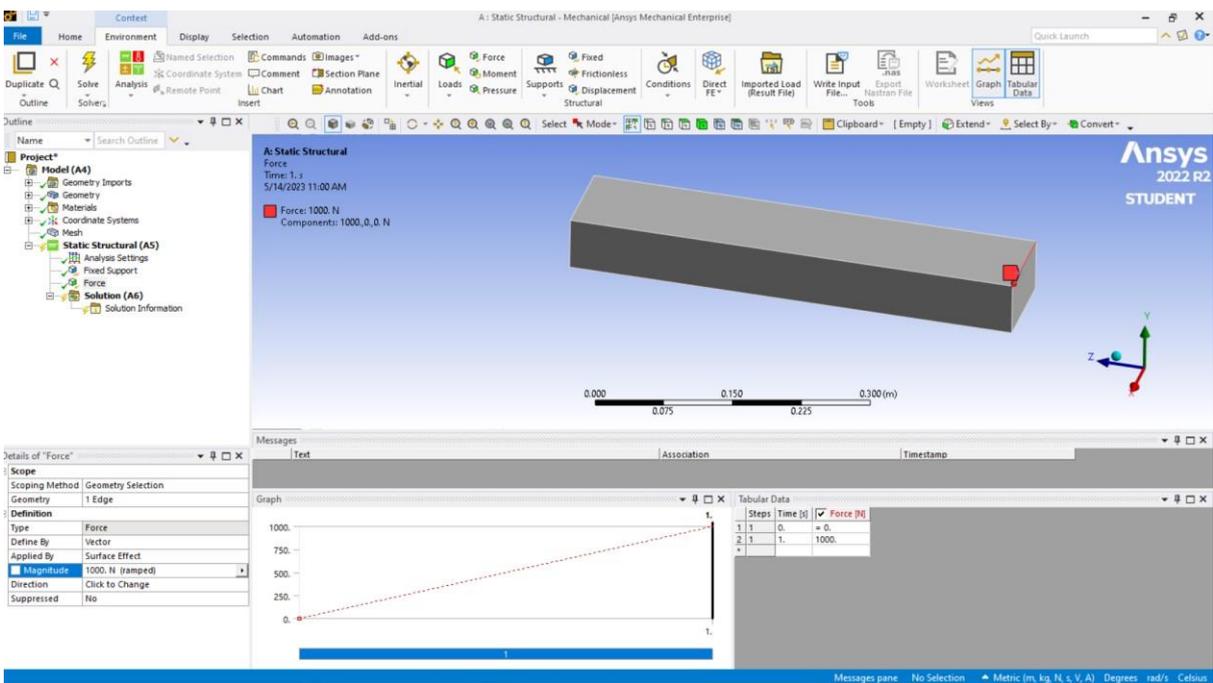
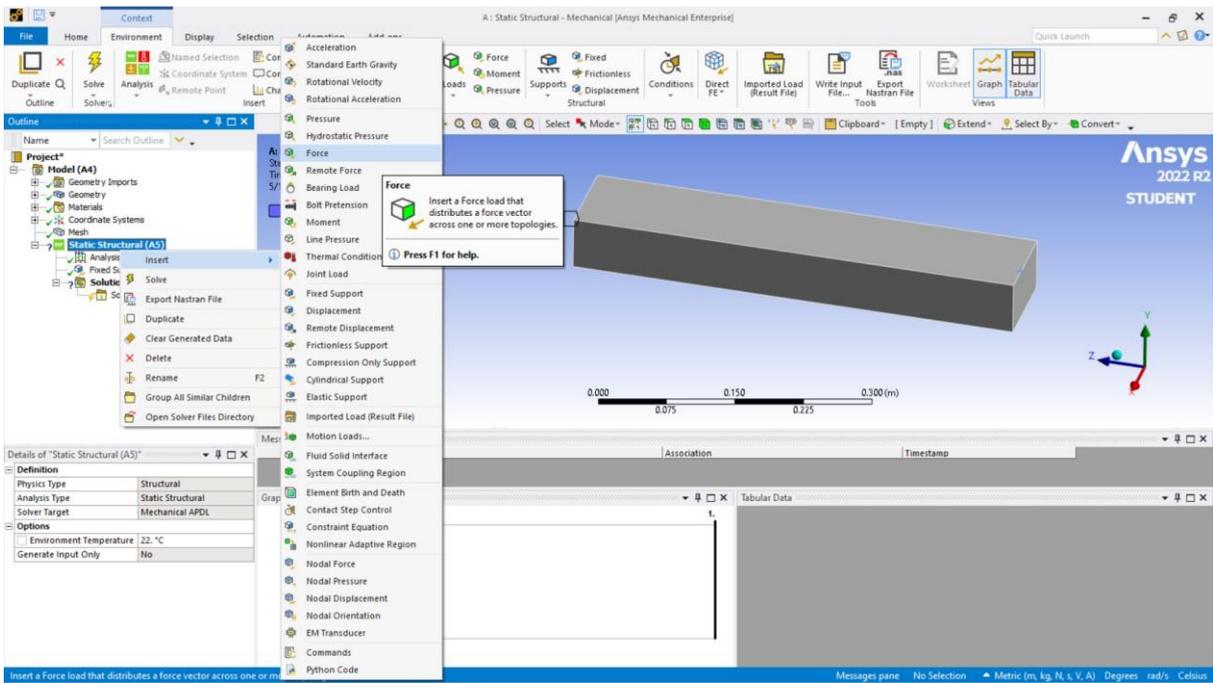
Right click on static structural then insert>fixed support



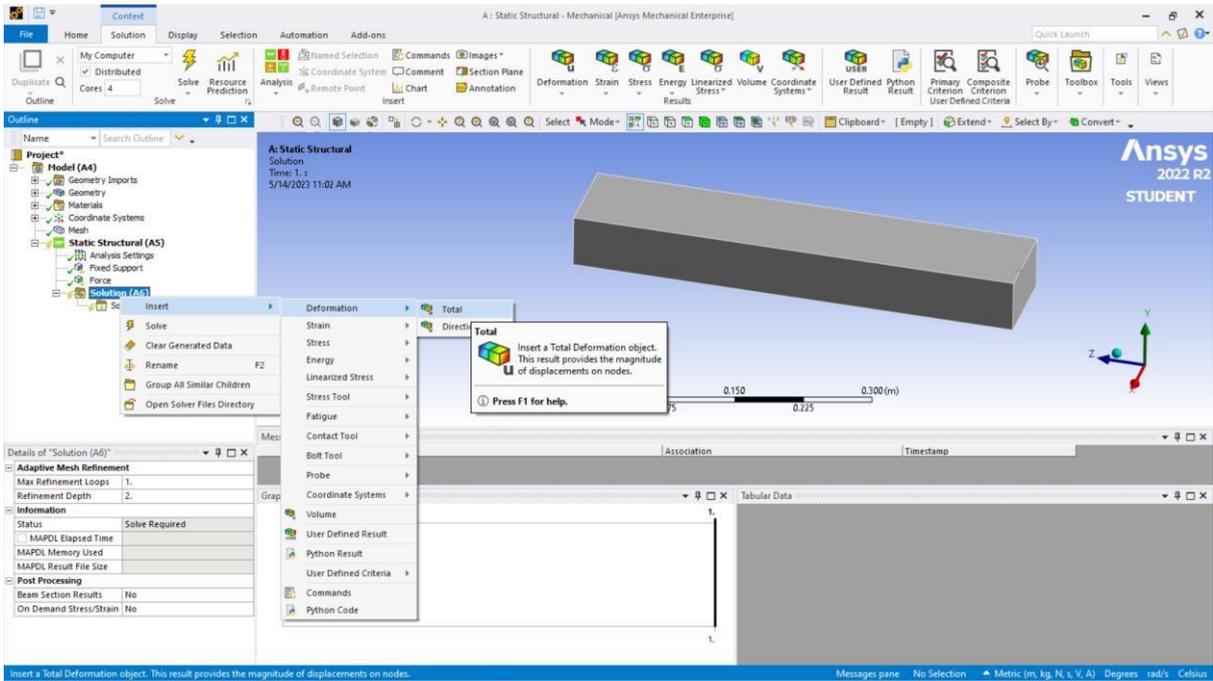
Then select the face and then apply the support



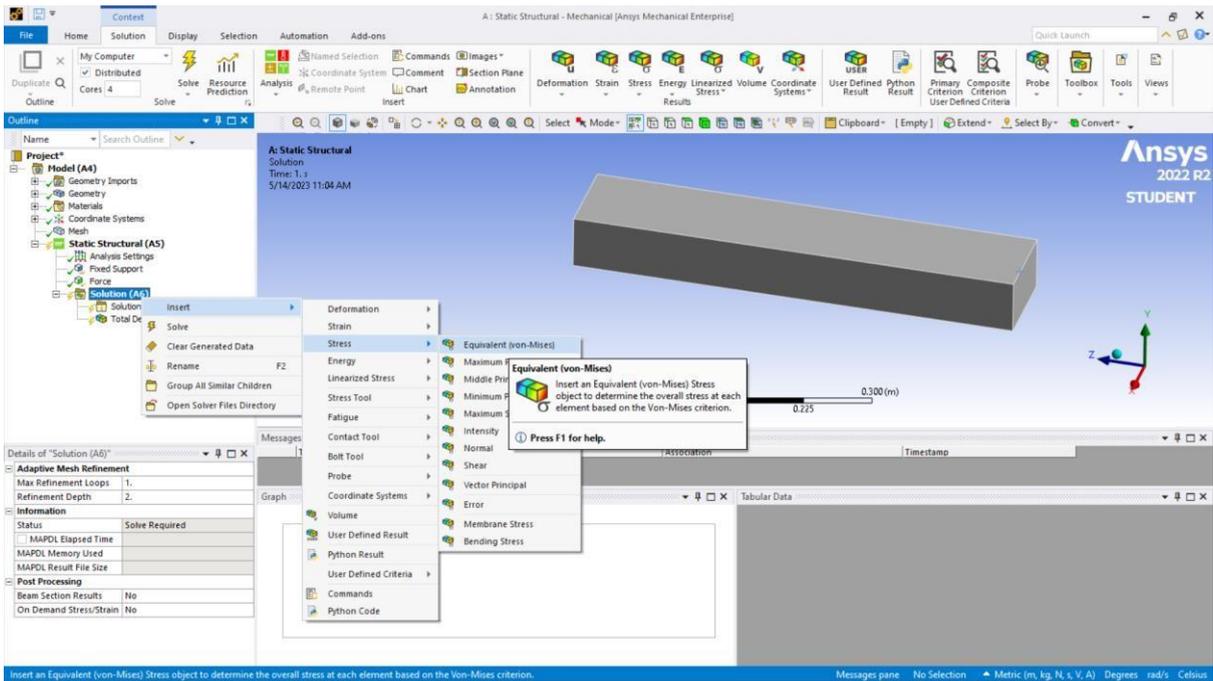
Similarly apply force on the edge of the beam



Step5: solve the problem , right click on solution>insert>deformation>total



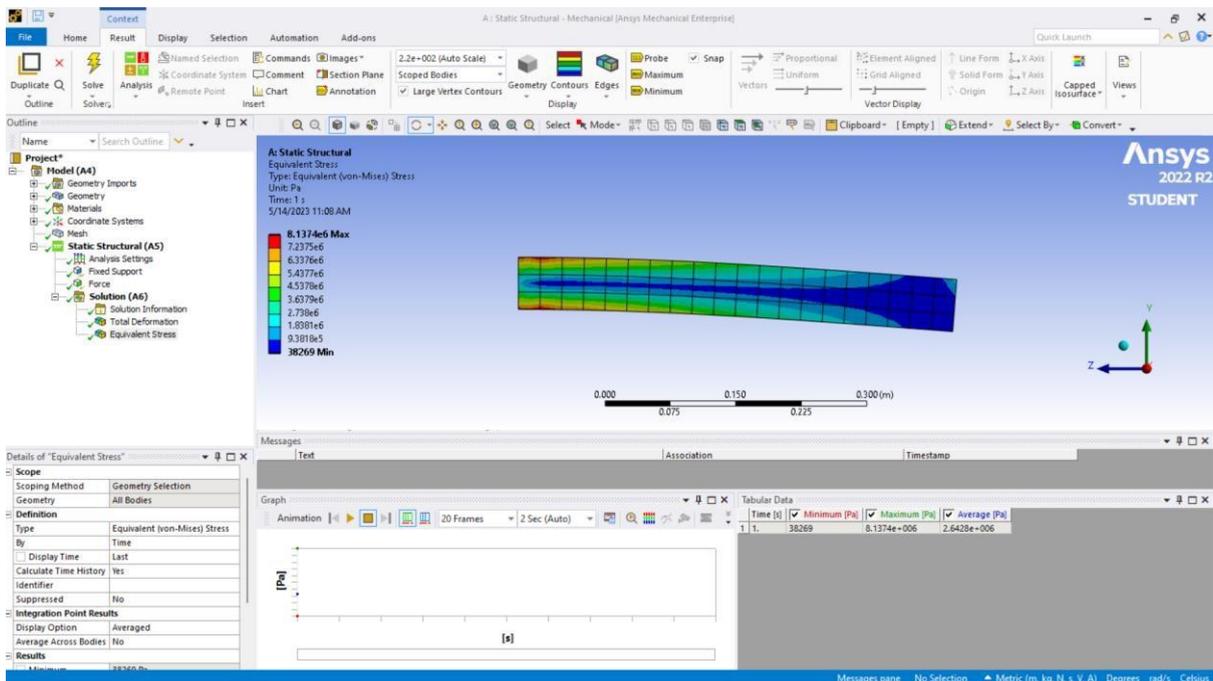
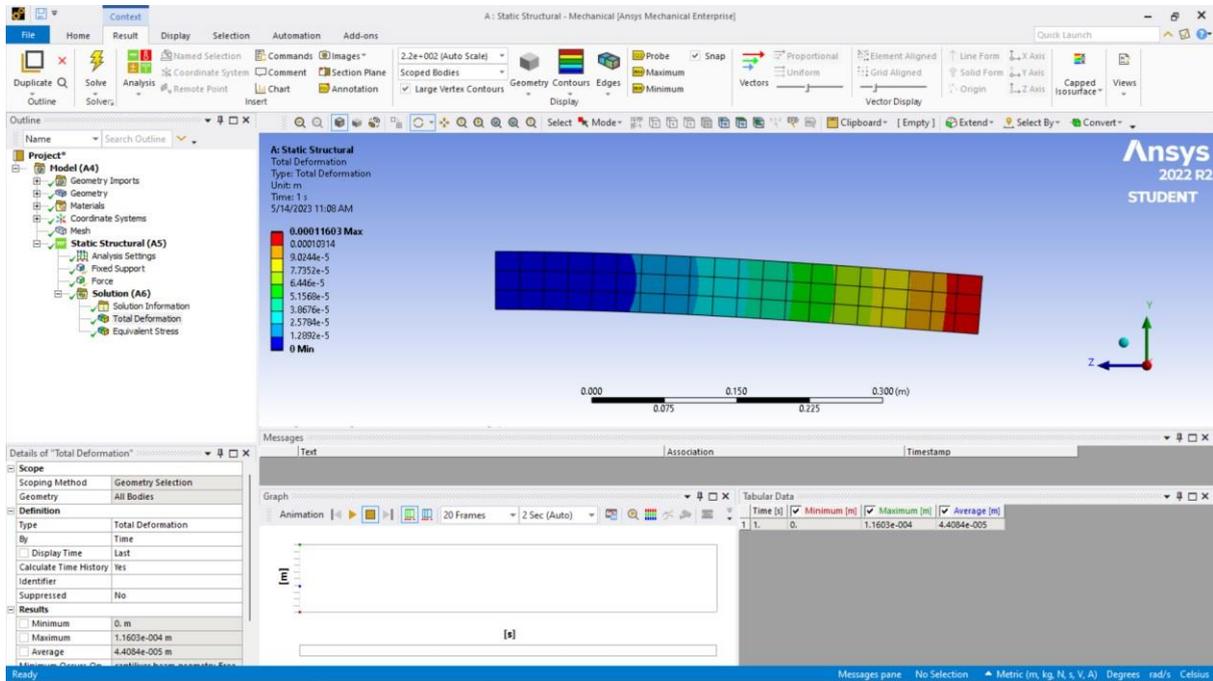
Similarly , right click on solution>insert>stress>equivalent stress



Then click on the “solve “ icon

Step6 : results

Total deformation



Conclusion:

References:

1. <https://www.youtube.com/watch?v=1e4pAYXFWLo>
2. <https://www.youtube.com/watch?v=3Ct0eVArooU>
3. https://my.eng.utah.edu/~me7540/ANSYS_intro2.pdf (From ansys manual)
4. https://www.engr.uvic.ca/~mech410/lectures/FEA_Theory.pdf
5. https://drive.google.com/file/d/1adyPIBFbT_Q6hkhvbHCBcgCHy8PafpWD/view?usp=sharing
6. <https://www.youtube.com/watch?v=-qNFGxjXx-s> (Beam Element Video)
7. <https://www.youtube.com/watch?v=L9rwxJ7SLqw> (Beam Element FEA Ansys Video)
8. <https://www.youtube.com/watch?v=0Jrh3k34dNQ>

Sign of Faculty/Lab In charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 09

Exercise for FEA of 1D thermal problems

Aim: Students have to solve the 1D thermal problem in ANSYS Software.

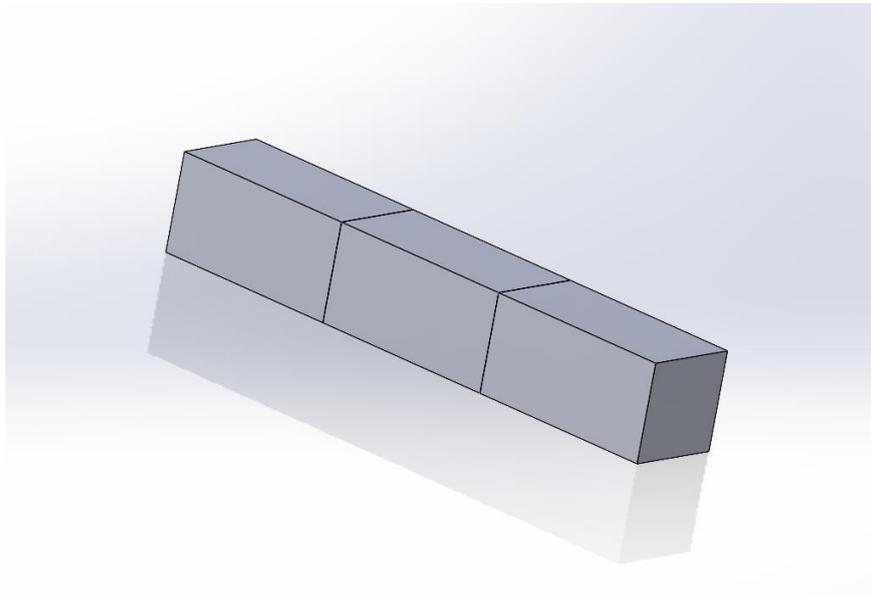
Objective:

- Create a convection boundary condition
- Create a cylindrical coordinate system.
- Create a spatially varying temperature constraint.
- Plot temperature variation.
- Use thermal results in a structural solution.

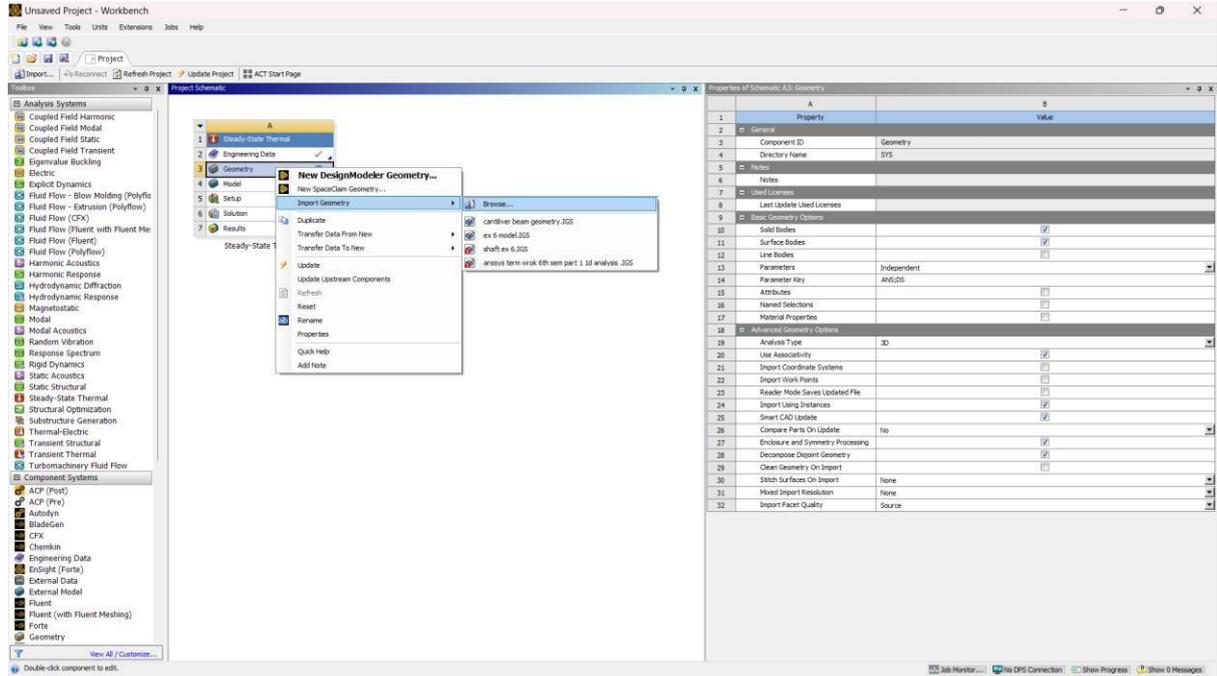
Problem Statement:

For a composite slab perform steady state thermal 1-D Finite Element Analysis. Procedure:

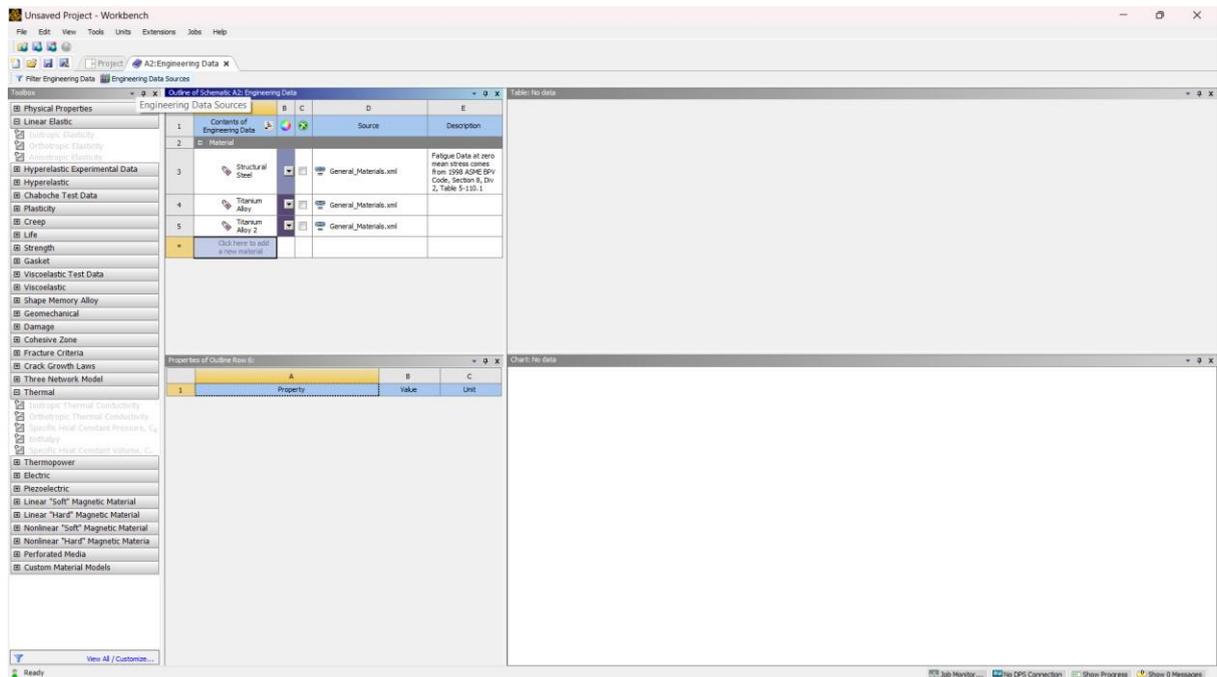
Step 1: create any geometry of your choice with multiple materials/bodies and save the file
in IGES/IGS format



Step2: open Ansys Workbench, select “steady-state thermal” analysis system and drag and drop it onto project schematic and import the geometry



Step3: double click on engineering data and go to engineering data sources on upper left corner



Then select materials of your choice

Here from thermal materials, aluminum and cast iron is selected

To add them click on the yellow "+" icon besides them

Unsaved Project - Workbench

File Edit View Tools Units Extensions Jobs Help

Project A2: Engineering Data X

Filter Engineering Data Engineering Data Sources

Field Variables

Frequency

Coordinate X

Coordinate Y

Coordinate Z

Shear Angle

Degradation Factor

Create Field Variable ...

Physical Properties

Linear Elastic

Hyperelastic

Hyperelastic Elasticity

Hyperelastic Elasticity

Hyperelastic Experimental Data

Hyperelastic

Chaboche Test Data

Plasticity

Creep

Life

Strength

Gasket

Viscoelastic Test Data

Viscoelastic

Shape Memory Alloy

Geomechanical

Damage

Cohesive Zone

Fracture Criteria

Crack Growth Laws

Three Network Model

Thermal

Isotropic Thermal Conductivity

Hyperelastic Thermal Conductivity

Specific Heat Constant Pressure, Cp

Thermal Expansion

Thermopower

Electric

Piezoelectric

Linear "Soft" Magnetic Material

Linear "Hard" Magnetic Material

Nonlinear "Soft" Magnetic Material

Nonlinear "Hard" Magnetic Material

View All / Customize...

Ready

Job Monitor No DPS Connection Show Progress Show 0 Messages

Unsaved Project - Workbench

File Edit View Tools Units Extensions Jobs Help

Project A2: Engineering Data X

Filter Engineering Data Engineering Data Sources

Field Variables

Frequency

Coordinate X

Coordinate Y

Coordinate Z

Shear Angle

Degradation Factor

Create Field Variable ...

Physical Properties

Linear Elastic

Hyperelastic

Hyperelastic Elasticity

Hyperelastic Elasticity

Hyperelastic Experimental Data

Hyperelastic

Chaboche Test Data

Plasticity

Creep

Life

Strength

Gasket

Viscoelastic Test Data

Viscoelastic

Shape Memory Alloy

Geomechanical

Damage

Cohesive Zone

Fracture Criteria

Crack Growth Laws

Three Network Model

Thermal

Isotropic Thermal Conductivity

Hyperelastic Thermal Conductivity

Specific Heat Constant Pressure, Cp

Thermal Expansion

Thermopower

Electric

Piezoelectric

Linear "Soft" Magnetic Material

Linear "Hard" Magnetic Material

Nonlinear "Soft" Magnetic Material

Nonlinear "Hard" Magnetic Material

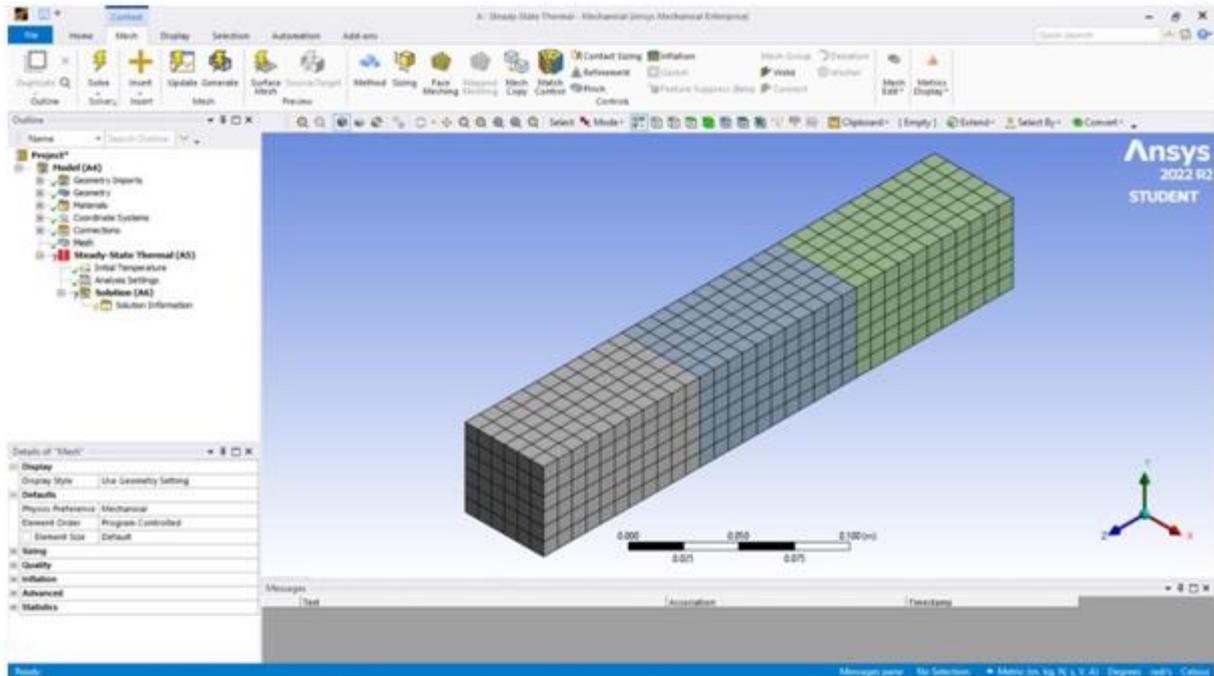
View All / Customize...

Ready

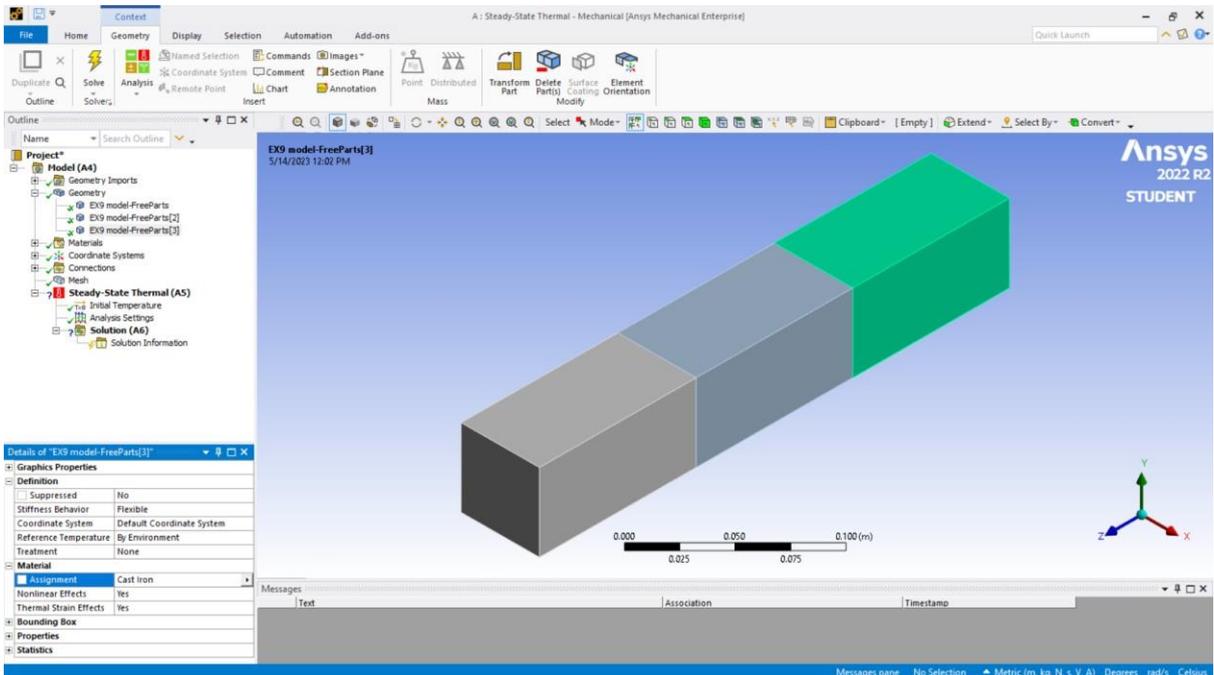
Job Monitor No DPS Connection Show Progress Show 0 Messages

After selecting close the engineering data tab from upper left corner

Step 4: open modeler from "model" and generate mesh by right clicking on "mesh" in outline tree

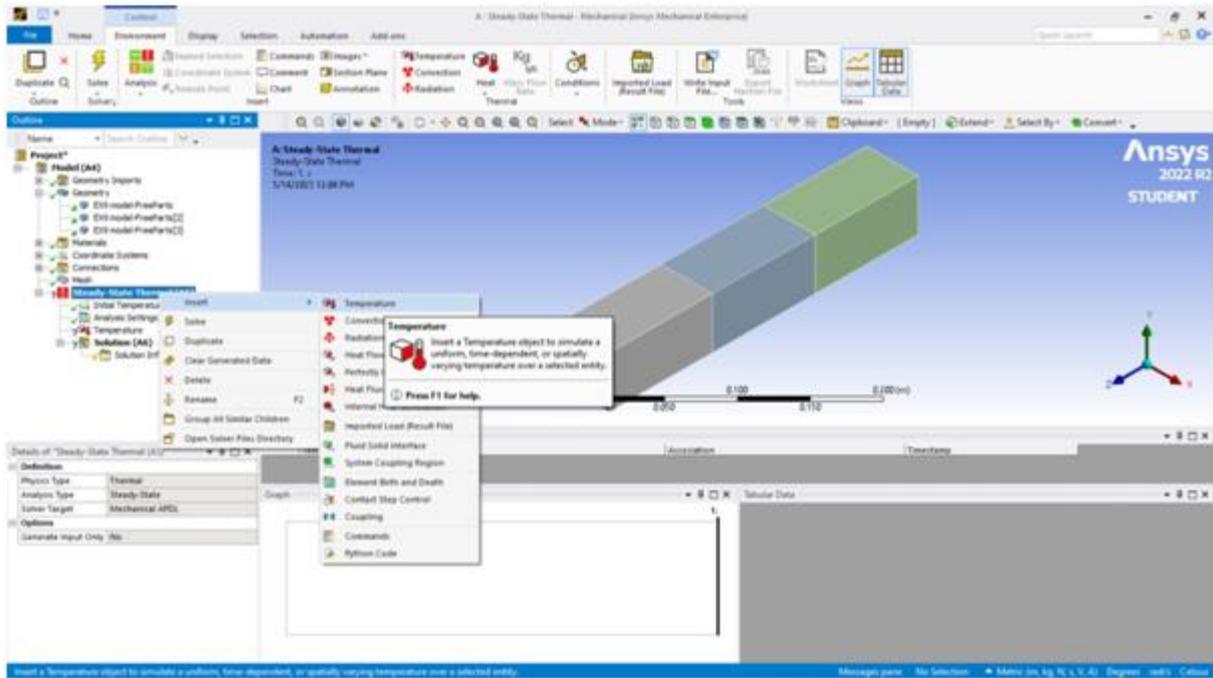


Step5: to add materials, click on the “+” near geometry in outline tree then select the geometry portion. On bottom left corner detail box will appear for that part and there in ”material” select the material of choice for each segment

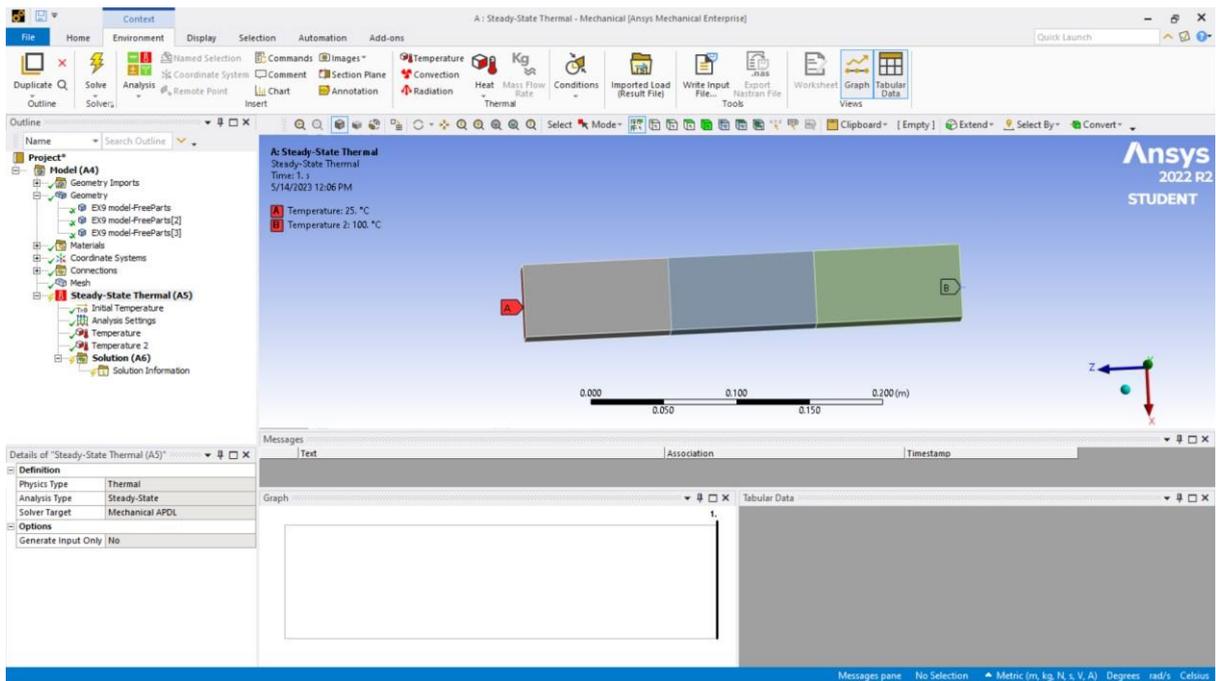


Step6: boundary conditions

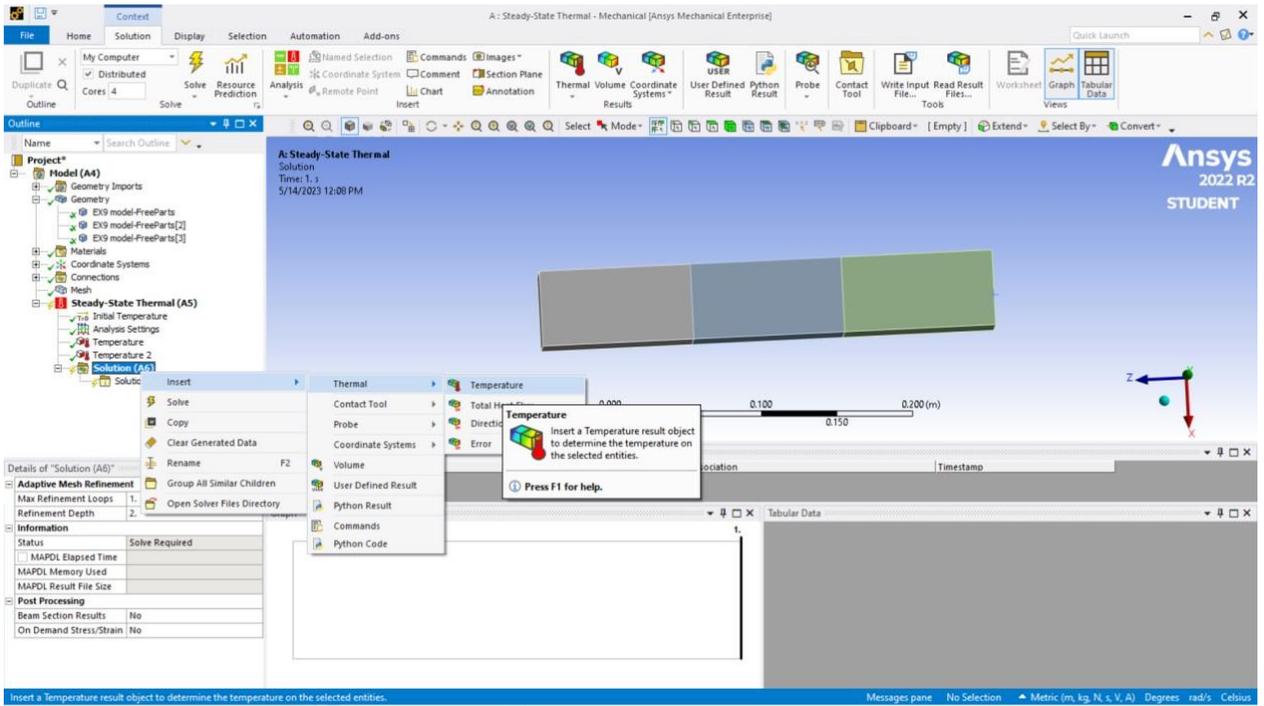
Right click on “steady state thermal ” then insert and select the boundary condition of choice. here temperature on two faces are given as boundary condition



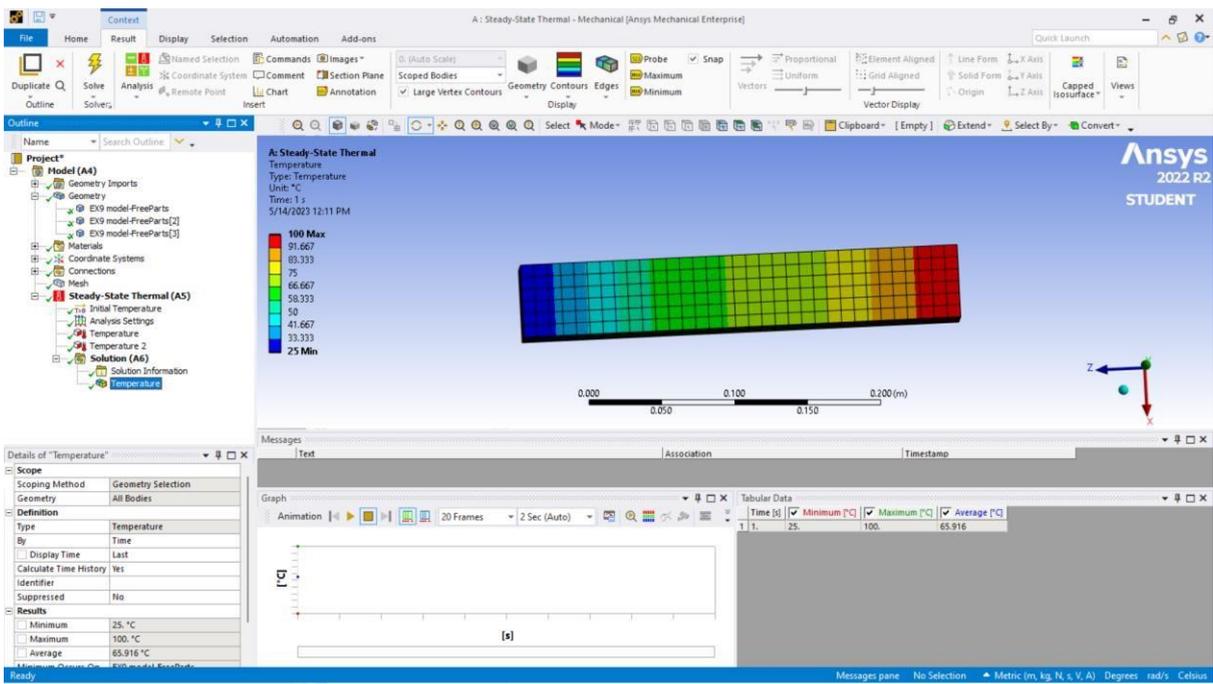
One face is set at 25° and other side is set at 100°



Step 7: solution , right click on “solution”>insert> temperature to measure temperature variation in the body then click on solve to solve the problem



Step 8: results



Conclusion:

References:

1. <https://www.youtube.com/watch?v=DXhpDia5RPk> (1D Thermal)
2. https://www.youtube.com/watch?v=oHYVzAih_VM
3. <https://www.youtube.com/watch?v=pA2th222ej4> (Composite Wall, Cylinder and Sphere)
4. <https://www.youtube.com/watch?v=miI1t7jyBQ>
5. Book : Finite element methods , by C.R.Alavala
6. https://drive.google.com/file/d/1aOUppYEd7UMLLEvW_OTBNSXfQ24ivvN9/view?usp=sharing

Sign of Faculty/Lab in charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 10

Exercise for FEA of 2-D structural problems

Aim: Students have to solve the 2D structural problem using ANSYS Software.

Objective:

- Create a new part from Parasolid geometry.
- Define boundary conditions.
- Solve the model.
- Display the results.

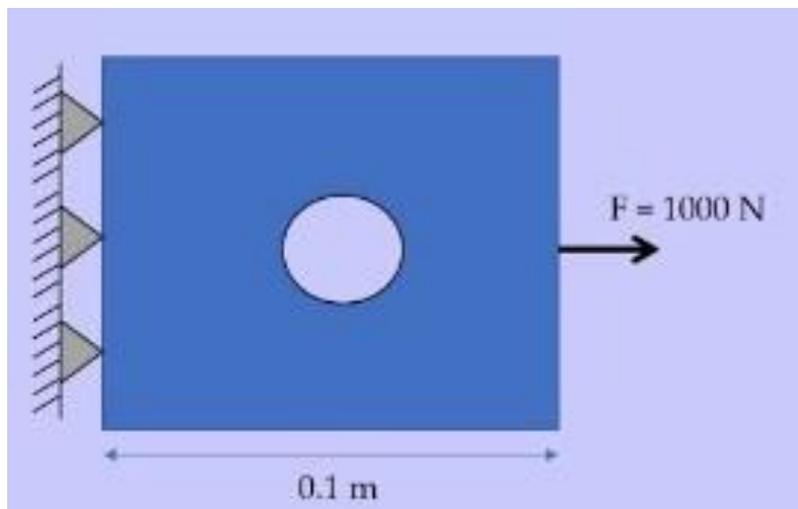
Problem Statement:

Find total deformation and von-mises stress of given square plate

Material: structural steel

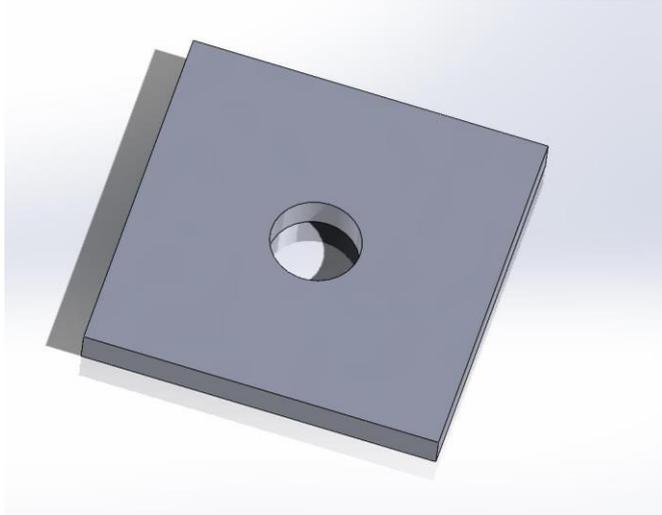
Thickness of the plate=0.01m

Hole diameter=0.025m

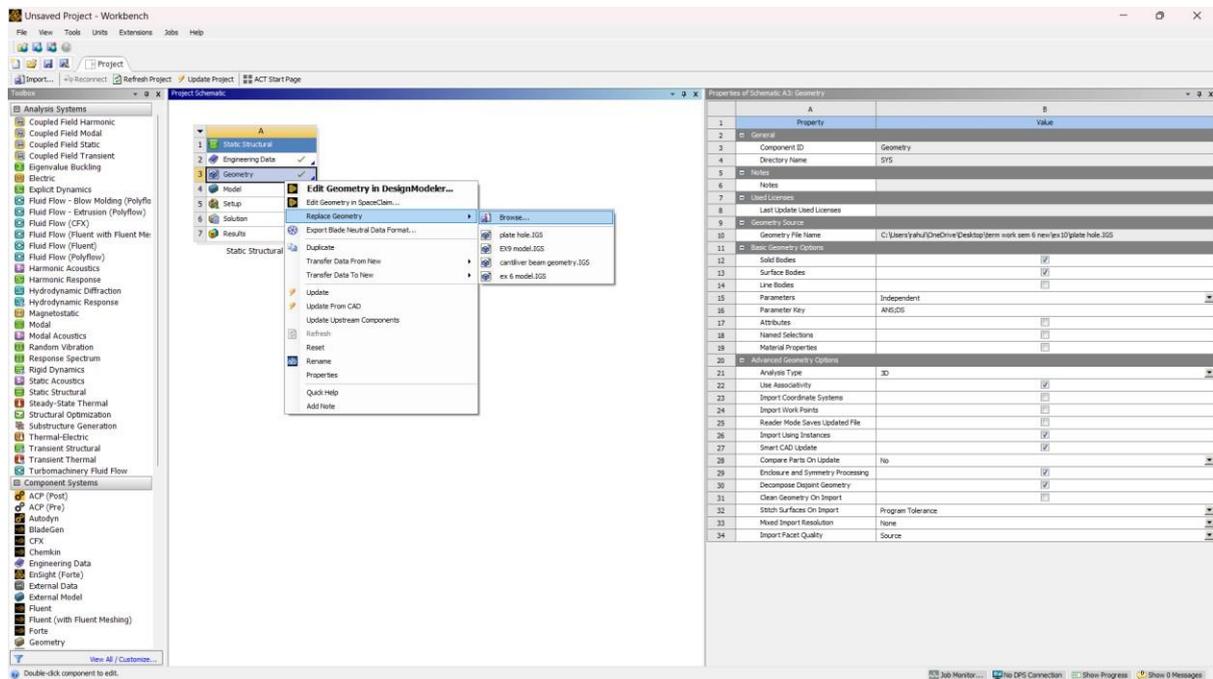


Procedure:

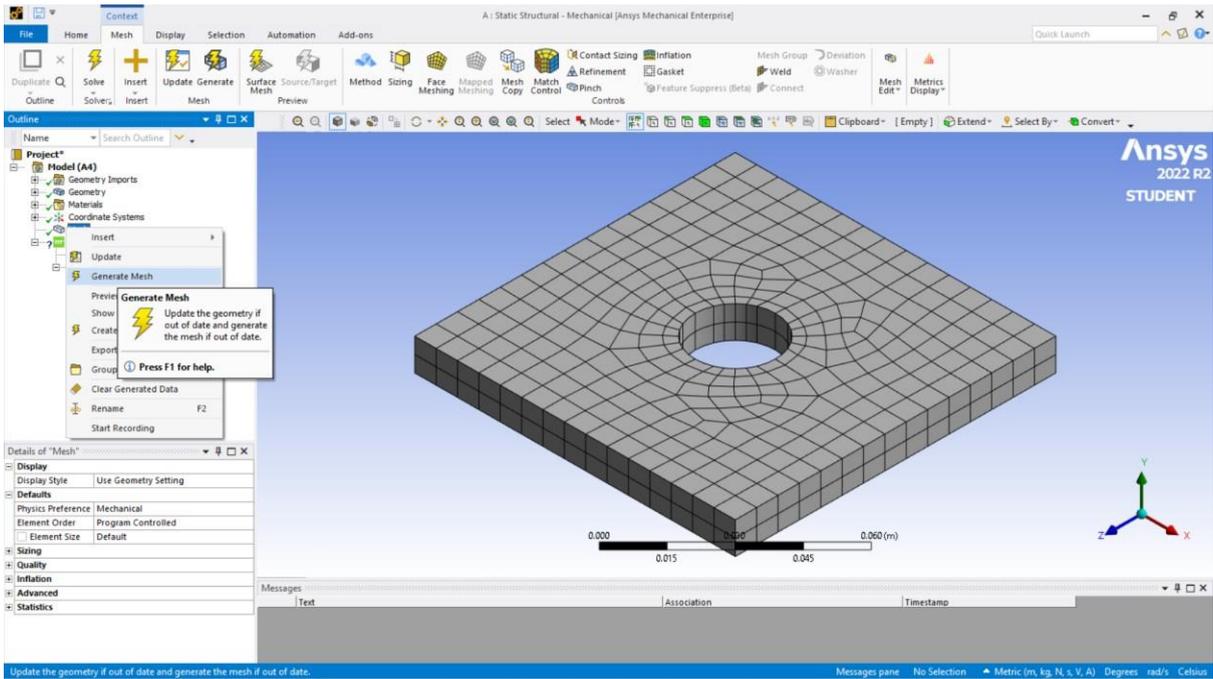
Step1: create geometry and save it in IGES/IGS format.



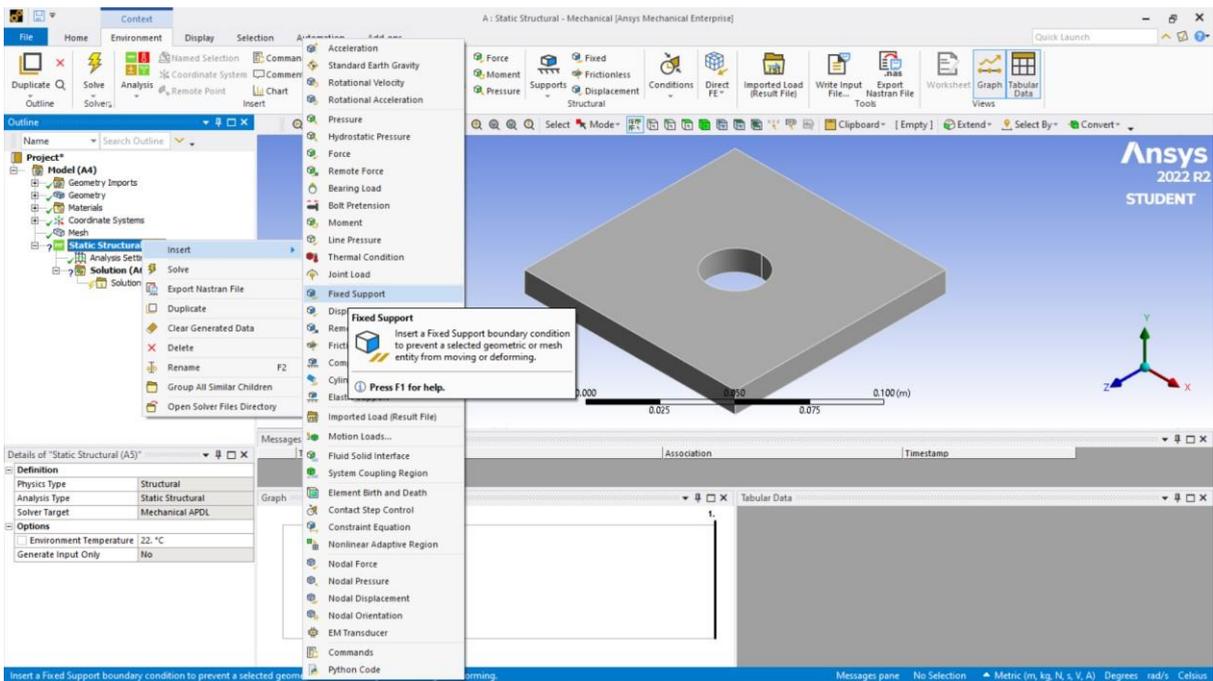
Step 2: open Ansys, drag and drop “static structural” Analysis system onto the project schematic and import the geometry



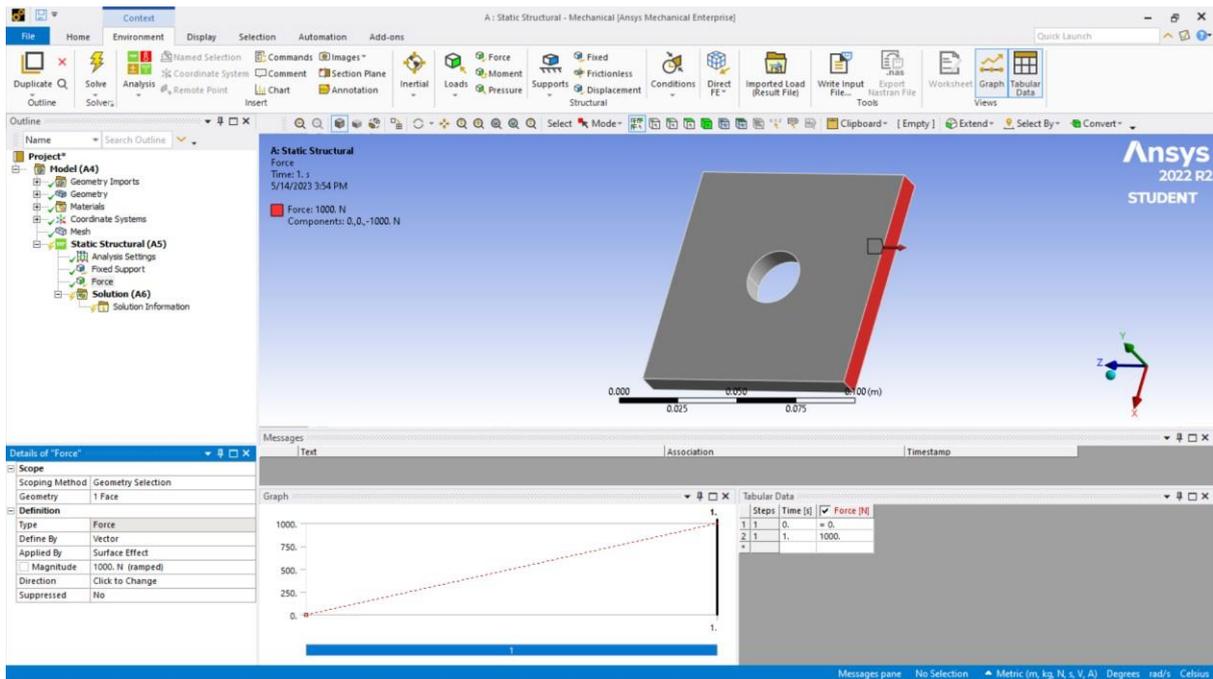
Step3: open modeler and generate the mesh from outline tree “mesh” option



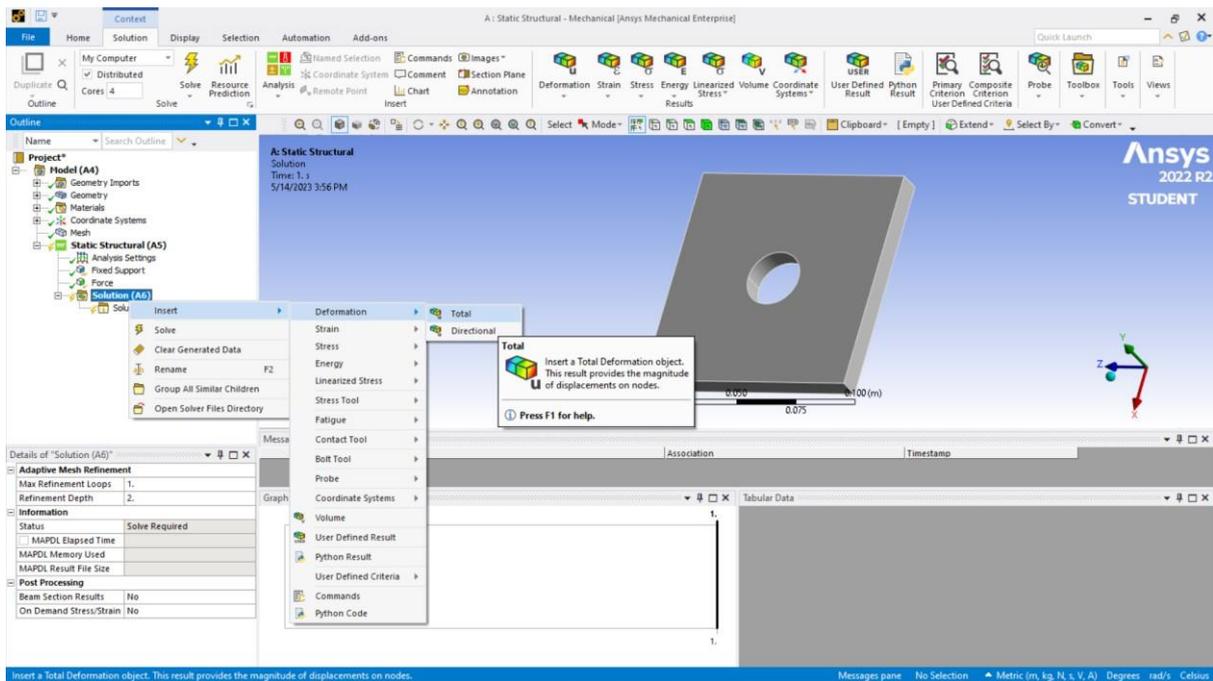
Step 4: adding boundary conditions, right click on “static structural”>insert>fixed support then select the face to be fixed and apply.

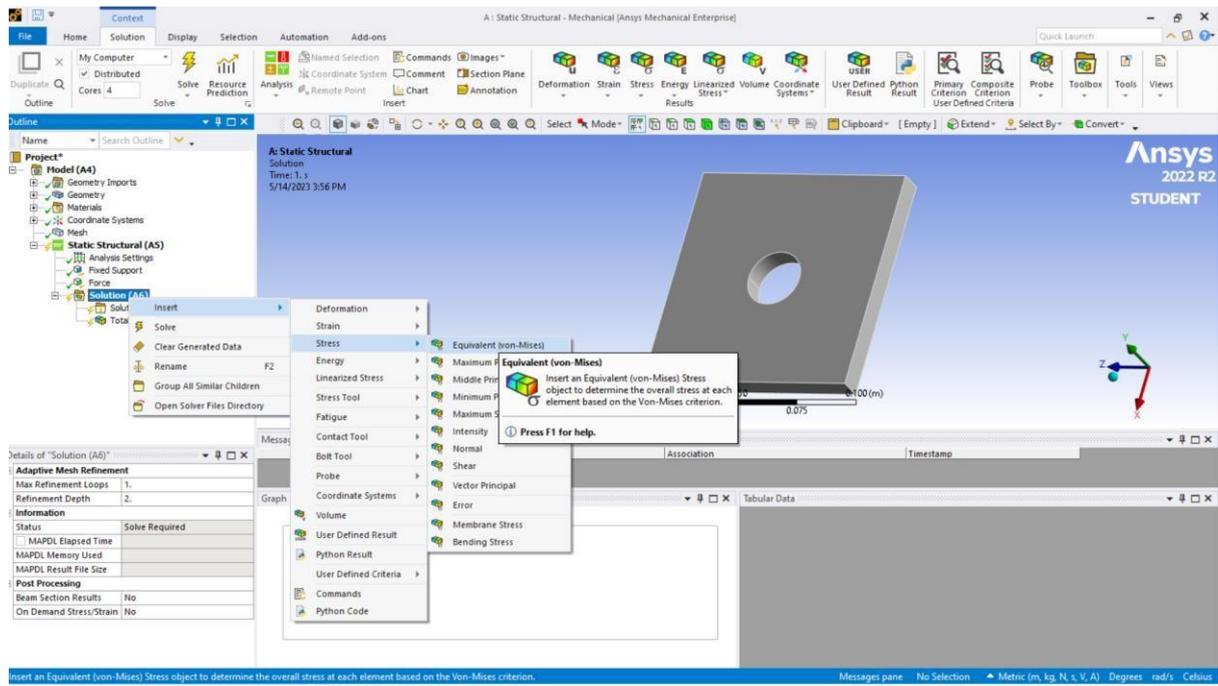


Similarly apply force in the desired face

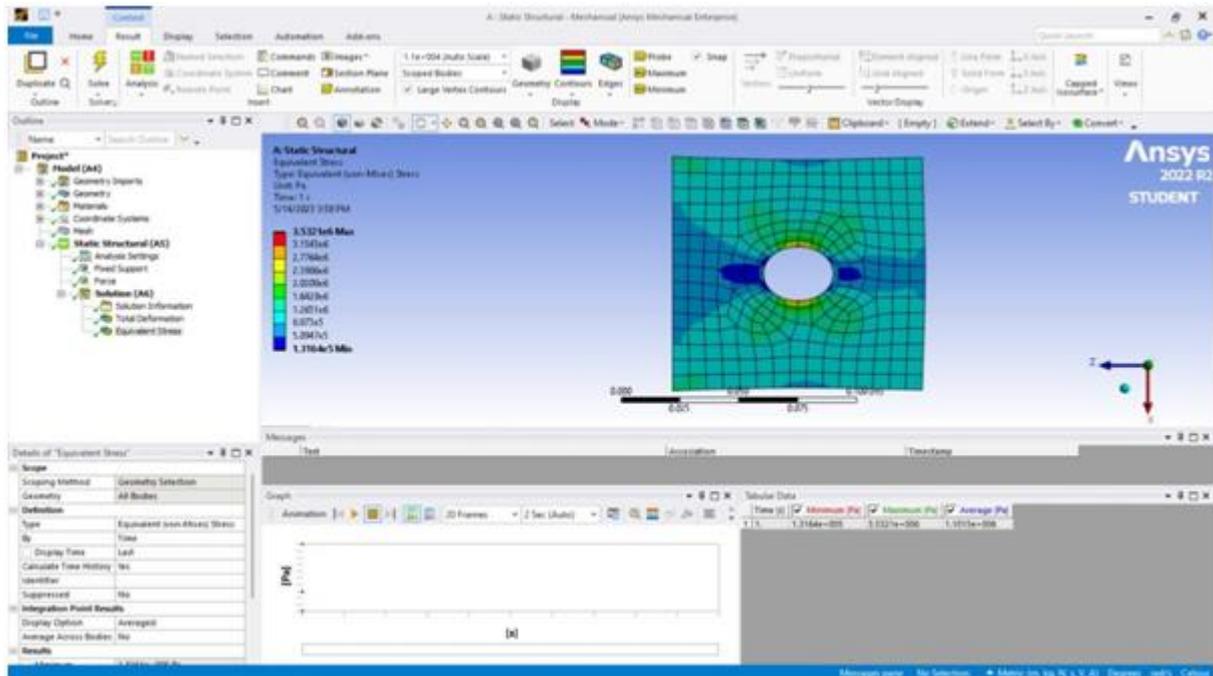


Step5: solve the problem, right click on “solution ”>insert>deformation>total and insert>stress>equivalent stress and then click on the “solve”

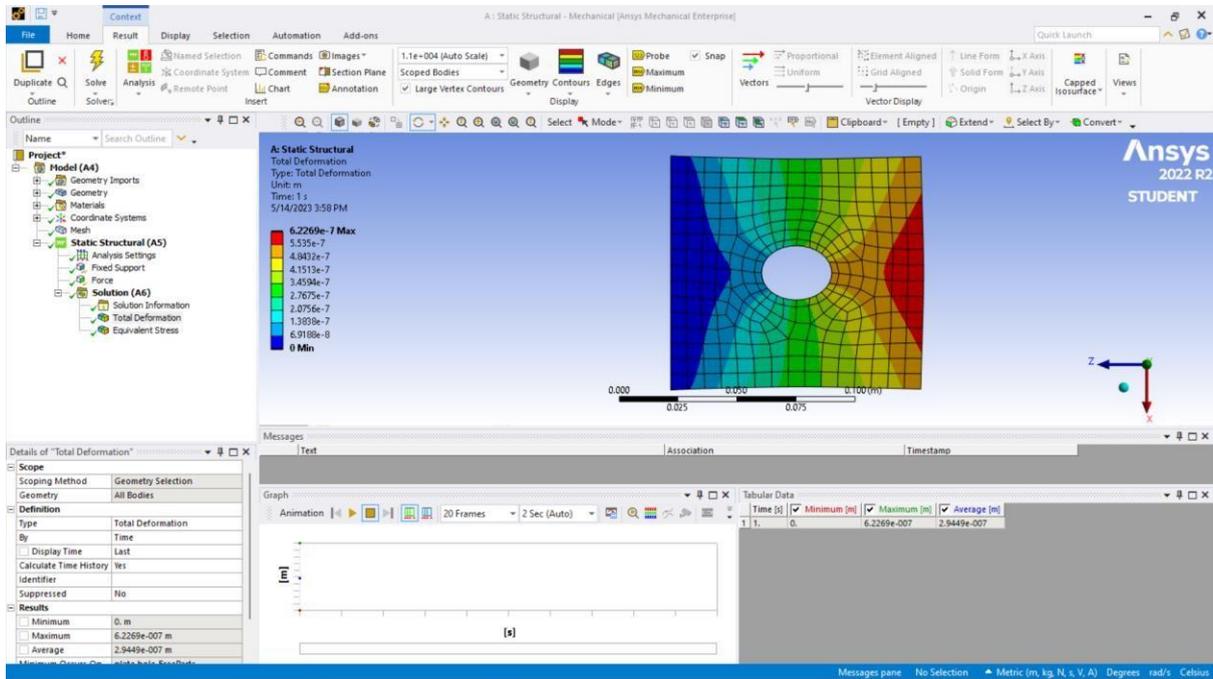




Step6:
results
stress



Deformation



Conclusion:

References:

1. <http://web.engr.uky.edu/~gebland/CE%20382/CE%20382%20Structural%20Analysis%20Handout%20Notes.pdf>
2. <https://www.youtube.com/watch?v=FtyTU5DOZtQ> (Ansys Workbench 2D Plain stress of bracket)
3. https://www.youtube.com/watch?v=qnIu2_6aAY
4. http://fea-cae-engineering.com/fea-cae-engineering/element_types.htm
5. http://mae.uta.edu/~lawrence/ansys/ansys_examples.htm
6. <https://www.youtube.com/watch?v=kcdbB0wJBXM>
7. Ref: University of Alberta (Ansys Tutorial)

Sign of Faculty/Lab In charge:

Rubrics	1	2	3	4	5	Total
Marks						

EXPERIMENT: 11

Exercise for developing the optimization model of machine element using Johnson Method.

Aim: Students have to develop the optimization model for the problem given.

1. Introduction: Adequate design and Optimum design and the differences.

Adequate design:

Adequate design is the selection of material & the values for independent geometrical parameters for a mechanical element such that the mechanical element satisfies its function requirement & the undesirable effects are kept in tolerable limit. For example, if one wants to design a shaft for a machine. The material for the shaft will be selected from the list of feasible materials and the geometrical parameter (Dia of shaft) will be determined on the basis of strength equation based on stress or rigidity. One will get different sizes for the different materials chosen. All such designs are known as adequate design.

Optimum Design:

Optimum design is the selection of material & the values for independent geometrical parameters with the explicit objective of either minimizing most significant undesirable effects or maximizing most significant functional requirement while making certain that the mechanical element satisfies function requirement & other undesirable effects are kept within tolerable limits. For example, if one wants to design a shaft for a machine with "Minimum weight criteria for that particular application" then there will be a single solution out of various options available. Such design is called optimum design. Optimum design is inclusive of adequate design.

2. The objectives of optimum design

For a design problem a large number of solutions are available which basically fulfill the functional requirement. Optimum design helps to choose the best alternative of the all available keeping a certain target in mind like minimizing weight or maximum strength or minimum cost. Following are the main objectives

- 1) To select the best possible option from the available alternatives keeping the target of maximizing a desirable effect or minimizing the undesirable effect

- 2) To reach the best solution without doing the repetitive work of detail analysis and evaluation of each alternative (If the optimum design methods would not be there, we would have to design complete component considering each alternative material and then after lot of laborious work we would reach a conclusion)

3. Johnsons Method of Optimum Design

Johnson's method involves developing three major design equations and solving these equations. These three equations are

- 1) **Primary design equation (PDE):** It is most important equation which outlines the most significant functional requirement to be maximized or the most significant undesirable effect which is to be minimized. For example, in design of a tensile bar the primary target of minimum weight, then the equation for weight is the PDE.
- 2) **Secondary design equation (SDE):** The secondary or subsidiary design equation are the important equation other than the primary design equation. For example, in case of tensile bar design the equation of tensile stress is the SDE.
- 3) **Limit Equation (LE):** The limit equations are the important design equation that define the satisfactory ranges of certain design parameters. Like the limiting value of stress, or torsional stiffness etc.

4. Step by step procedure of Johnson's method of optimum design

Following steps are followed in optimum design using Johnsons Method for normal specifications

- 1) Select independent geometrical parameters
- 2) Decide the basis of optimum design (like weight, cost, strength etc)
- 3) Write P.D.E
- 4) Write S.D.E and L.E
- 5) Classify parameters
- 6) Combine S.D.E 's with P.D. E's
- 7) Find variation of optimum design quantity
- 8) Impose L.E. on P.D.E
- 9) Write final P.D.E in terms of parameter groups
- 10) Select optimum material from comparison table for various material.
- 11) Determine optimum values of geometrical parameters

5. The various design parameters and Explain Functional requirement parameters, Material parameters and Geometric parameters.

The design parameter that make the design complete and successful are classified into three groups

- 1) **Functional requirement parameters:** These are the conditions imposed by the end applications on the component. These are the requirements which are must for the proper functioning of the component. Generally functional requirement parameters are independent of the material or geometric parameters
- 2) **Material Parameters:** The material parameters are those parameters which describe the material with values like density, modulus of elasticity, modulus of rigidity, yield strength, ultimate tensile strength, cost per kg etc
- 3) **Geometrical parameters:** These are the parameters which define/limit the geometry of the mechanical component like module of gear, length of belt, face width of gears etc.

6. Desirable and undesirable effects in optimum design

The functional requirements are the conditions which must be satisfied by the mechanical element so that it works as intended. The functional requirements are classified as Desirable and Undesirable effects.

Desirable effects: Desirable effects are the functional requirement parameters that are useful for the mechanical element to perform satisfactorily its function, like tensile strength, power transmitting capacity, storage capacity, bending resistance etc. Desirable effects are best when they are maximum in value.

Undesirable effects: Undesirable effects are also functional requirements but which are detrimental for its functioning, and are best when they have minimum in value. Like weight of component, cost of component, Imbalance in component etc.

7. Application of Johnson method:

- 1) Designing aircraft parts for minimum weight
- 2) Designing optimum production schedules
- 3) Minimum cost design of various devices
- 4) Inventory control planning
- 5) Optimum design of structural members
- 6) Optimum design of control systems and electrical circuits

- 7) Finding Optimal nose cone angle for minimum drag
- 8) Designing material handling equipment's such as conveyers, cranes, trucks etc. for material handling

CaseStudy:

In light weight Equipment, a shaft is transmitting a torque of 900 Nm. and is to have a rigidity of 90Nm/degree. Assume a factor of safety of 1.5 based on yield stress. Design the shaft with minimum weight. What will be the change in design for minimum cost? Assume maximum shear stress theory of failure. Use the following data for the materials:

Material	Mass Density(kg/m ³)	Material cost(Rs./N (N=weight in Newton)	Yield strength (Mpa)	Shear Modulus(Gpa)
Steel	8500	16	130	80
Al.Alloy	3000	32	50	26.7
Titanium Alloy	4800	480	90	40
Magnesium Alloy	2100	32	20	16

Questions:

1. Classify the optimization problem in detail?
2. Differentiate between adequate and optimum design. Also explain different types of equations that are used in 'Johnson's method of optimum design'.
3. What do you mean by primary and subsidiary design equation?
4. Draw the step by step flow chart for Optimum Design Procedure?
5. Explain the following with reference to optimization: i) Objective function ii) Constraints

References:

1. S.S.Rao, "Engineering Optimization", A wiley –Interscience
2. <https://old.mu.ac.in/wp-content/uploads/2017/10/Optimization-Models.pdf>
3. <https://doi.org/10.1115/1.3454119>
4. https://www.researchgate.net/publication/308961573_Design_Optimization_of_Mechanical_components_using_Johnson's_Optimization_Algorithm
5. <https://doi.org/10.1115/1.3267471>

Sign of Faculty/Lab in charge:

Rubrics	1	2	3	4	5	Total
Marks						