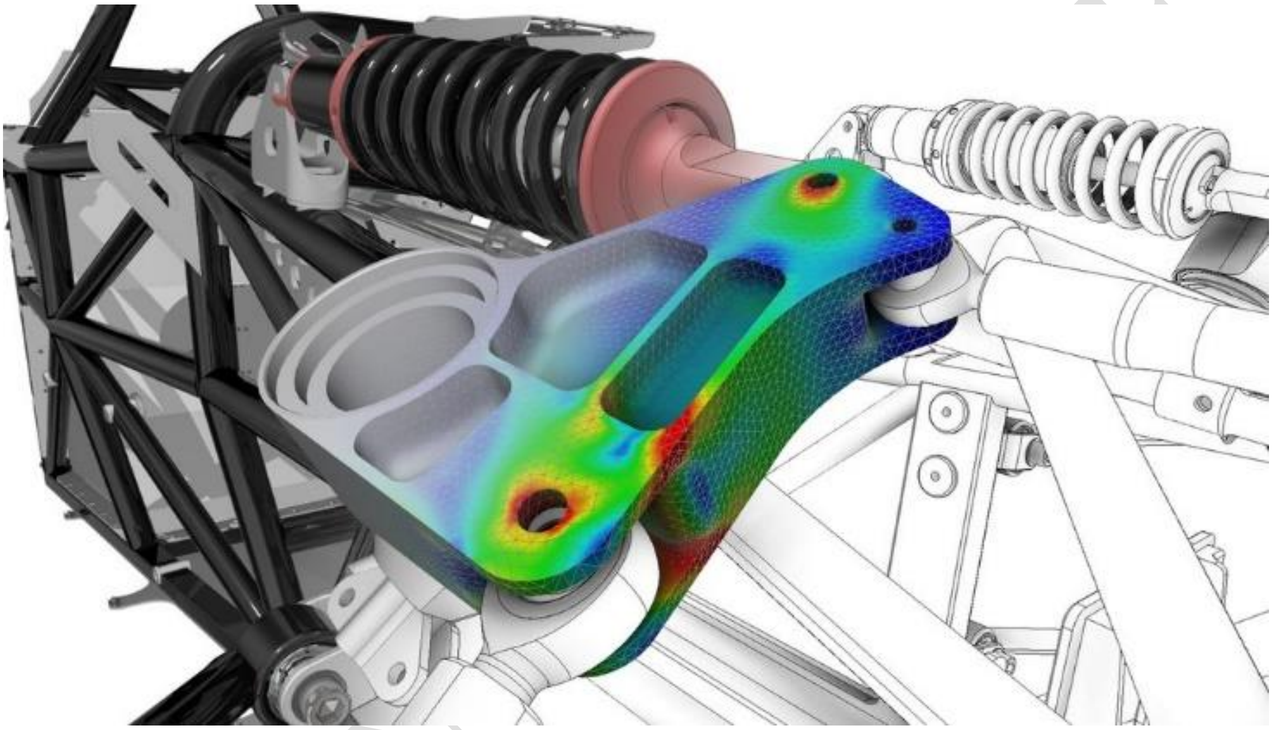
16ME77 – COMPUTER AIDED SIMULATION AND ANALYSIS LABORATORY MANUAL

(FOR SEVENTH SEMESTER B.E MECHANICAL ENGINEERING STUDENTS)



DEPARTMENT OF MECHANICAL ENGINEERING

REV : 02
JUNE: 2021

P.S.R ENGINEERING COLLEGE, SIVAKASI – 626 140.
(An Autonomous Institution)
DEPARTMENT OF MECHANICAL ENGINEERING
(2021 - 2022)

P.S.R. ENGINEERING COLLEGE, SIVAKASI – 626 140.
(An Autonomous Institution)
DEPARTMENT OF MECHANICAL ENGINEERING

SERVICE IS OUR MOTTO

161ME77-COMPUTER AIDED SIMULATION & ANALYSIS LABORATORY MANUAL

For Seventh Semester B.E Mechanical Engineering Students
(2021-2022)

REV: 02
JUNE: 2021

1/7/2021
Prepared by

Dr.P. Shenbaga Velu ASP/MECH
Dr. Srinivasagam Ramesh ASP/MECH

Approved by
1/7/2021
HOD/MECH
Head Of the Department
Department of Mechanical Engineering
P.S.R. Engineering College
Sevalpatti, Sivakasi - 626 140

INDEX

Ex.No	Date	Name of the Exercise	Page.No	Staff Initial

P.S.R. Engineering College

Vision & Mission Statement

Vision

- To contribute to the society through excellence in technical education with societal values and thus a valuable resource for industry and the humanity

Mission

- To create an ambience for quality learning experience by providing sustained care and facilities
- To offer higher level training encompassing both theory and practices with human and social values
- To provide knowledge-based services and professional skills to adapt tomorrow's technology and embedded global changes

Department of Mechanical Engineering

Vision & Mission Statement

Vision

- To provide broad-based education and training in mechanical engineering and its applications to enable the graduates to meet the demands in a rapidly changing needs in industry, academia and society

Mission

- To impart high quality technical education and training that encompasses both theory and practices with human and social values
- To equip the students to face tomorrow's technology embedded global changes
- To create, explore, and develop innovations in mechanical engineering research

Department of Mechanical Engineering

Programme Specific Outcomes

PSO 1 - Apply the concepts of mathematics and science in mechanical systems
PSO 2 - Design and analyze components and systems for mechanical engineering applications
PSO 3 - Synthesis data and technical concepts for application to mechanical engineering software
PSO 4 - Apply manufacturing and management practices in industries

Programme Outcomes of Mechanical Engineering

1. **Engineering Knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem Analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design/ Development of Solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct Investigations of Complex Problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern Tool Usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The Engineer and Society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and teamwork:** Function effectively as an individual, and as a member or leader in diverse teams, and in multi-disciplinary settings.
10. **Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one’s own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
12. **Lifelong learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

16ME77 COMPUTER AIDED SIMULATION AND ANALYSIS LABOROTARY **L T P C 0 0 3 2**

SYLLABUS

ANALYSIS (SIMPLE TREATMENT ONLY) 37

1. Introduction of ANSYS - 14.5
2. Stress Analysis of Cantilever Beam
3. Stress Analysis of Fixed End with Point Load Beam
4. Stress Analysis of Simply Supported with UDL Load Beam
5. Stress Analysis of Plate with hole
6. Stress Analysis of an Axis – Symmetric Component
7. Modal Analysis of Cantilever 2d Plate
8. Modal Analysis of Cantilever Beam
9. Modal Analysis of Simply supported Beam
10. Modal Analysis of Fixed End Beam
11. Harmonic Analysis of Cantilever Beam
12. Thermal Mixed Boundary (Conduction/Convection/Insulation)

SYSTEM REQUIREMENTS
(for a batch of 30 students)

Description of Equipment	Quantity Required
HARDWARE	
Computer Server	1 NOS
Computer System	
17" VGA Colour Monitor	
Pentium IV processor	
40 GB HDD	
512 MB RAM	30 NOS
Colour Desk Jet Printer	1 NOS
SOFTWARE	
Suitable Analysis Software	30 Licenses
C / MATLAB	5 Licenses

Course Outcomes	Program Outcomes (POs)												Program Specific Outcomes (PSOs)			
	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	2	3	3						2	3		2	2			2
CO2	2	1	1		2				2	2		3	3	3	1	2
CO3	2	3	3		2					2		2	3	3	2	2
CO4	2	3	3		2					2		2	3	3	2	2
CO5	2	3	3		2				2	2		2	3	3	2	2
CO6	2	3	3		2				2	1		3	2	3	3	3

1: Slight (Low) 2: Moderate (Medium) 3: Substantial (High)

SI.NO	LIST OF EXERCISE	PERIODS	CUMULATIVE PERIODS
1.	Introduction to ANSYS -14.5	3	3
2.	Stress Analysis of Cantilever beam	3	6
3.	Stress Analysis of Fixed end with point load beam	2	8
4.	Stress Analysis of Simply supported with UDL load beam	2	10
5.	Stress Analysis of plate with hole	3	13
6.	Stress Analysis of Axis-Symmetric component	3	16
7.	Modal Analysis of Cantilever 2D plate	3	19
8.	Modal Analysis of Cantilever beam	2	21
9.	Modal Analysis of simply supported beam	2	23
10.	Modal Analysis of Fixed end beam	2	25
11.	Harmonic Analysis of Cantilever beam	3	28
12.	Thermal mixed boundary (Conduction /Convection/Insulation)	3	31

EXTRA EXERCISE (Beyond the Syllabus)

1.	Stress Analysis on 2D Truss	2	33
2.	Stress Analysis in2D Fixed beam with UDL	2	35
3.	Modal Analysis	2	37
4.	Stress Analysis on Cantilever beam-pipe element	2	39
5.	Effect of Self Weight on a Cantilever beam	2	41
6.	Application of Distributed Loads	2	43
7.	Simple Conduction	2	45
		TOTAL PERIODS	45

Introduction to ANSYS

ANSYS is a general-purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

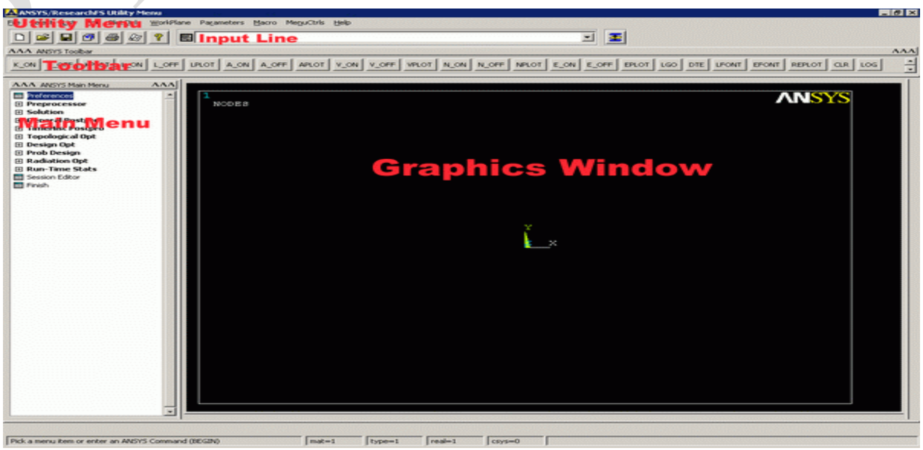
In general, a finite element solution may be broken into the following three stages. This is a general guideline that can be used for setting up any finite element analysis.

1. **Preprocessing: defining the problem**; the major steps in preprocessing are given below:
    a. Define keypoints/lines/areas/volumes
    b. Define element type and material/geometric properties
    c. Mesh lines/areas/volumes as required
    d. The amount of detail required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axisymmetric, 3D).
2. **Solution: assigning loads, constraints and solving**; here we specify the loads (point or pressure), constraints (translational and rotational) and finally solve the resulting set of equations.
3. **Postprocessing: further processing and viewing of the results;** in this stage one may wish to see:
    a. Lists of nodal displacements
    b. Element forces and moments
    c. Deflection plots
    d. Stress contour diagrams

ANSYS 14.5 Environment

The ANSYS Environment for ANSYS 14.5 contains 2 windows: the Main Window and an Output Window. Note that this is somewhat different from the previous version of ANSYS which made use of 6 different windows.

1. **Main Window**



Within the Main Window are 5 divisions:

a. **Utility Menu**

The Utility Menu contains functions that are available throughout the ANSYS session, such as file controls, selections, graphic controls and parameters.

b. **Input Window**

The Input Line shows program prompt messages and allows you to type in commands directly.

c. **Toolbar**

The Toolbar contains push buttons that execute commonly used ANSYS commands. More push buttons can be added if desired.

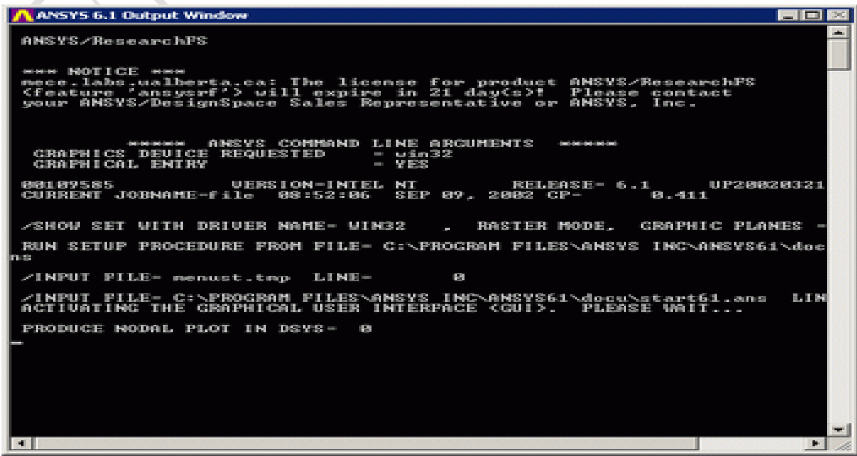
d. **Main Menu**

The Main Menu contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor, design optimizer. It is from this menu that the vast majority of modeling commands are issued. This is where you will note the greatest change between previous versions of ANSYS and version 14.5. However, while the versions appear different, the menu structure has not changed.

e. **Graphic Window**

The Graphic Window is where graphics are shown and graphical picking can be made. It is here where you will graphically view the model in its various stages of construction and the ensuing results from the analysis.

2. **Output Window**



The Output Window shows text output from the program, such as listing of data etc. It is usually positioned behind the main window and can depute to the front if necessary.

ANSYS Interface
Graphical Interface vs. Command File Coding

There are two methods to use ANSYS. The first is by means of the graphical user interface or GUI. This method follows the conventions of popular Windows and X-Windows based programs.

The second is by means of command files. The command file approach has a steeper learning curve for many, but it has the advantage that an entire analysis can be described in a small text file, typically in less than 50 lines of commands. This approach enables easy model modifications and minimal file space requirements.

The tutorials in this website are designed to teach both the GUI and the command file approach, however, many of you will find the command file simple and more efficient to use once you have invested a small amount of time into learning the code.

For information and details on the full ANSYS command language, consult:

Help > Table of Contents > Commands Manual.

FEM Convergence Testing

Introduction

A fundamental premise of using the finite element procedure is that the body is sub-divided up into small discrete regions known as finite elements. These elements defined by nodes and interpolation functions. Governing equations are written for each element and these elements are assembled into a global matrix. Loads and constraints are applied and the solution is then determined.

The Consequences

Finer meshes come with a cost however: more calculational time and large memory requirements (both disk and RAM)! It is desired to find the minimum number of elements that give you a converged solution.

Beam Models

For beam models, we actually only need to define a single element per line unless we are applying a distributed load on a given frame member. When point loads are used, specifying more than one element per line will not change the solution, it will only slow the calculations down. For simple models it is of no concern, but for a larger model, it is desired to minimize the number of elements, and thus calculation time and still obtain the desired accuracy.

General Models

In general however, it is necessary to conduct convergence tests on your finite element model to confirm that a fine enough element discretization has been used. In a solid mechanics problem, this would be done by creating several models with different mesh sizes and comparing the resulting deflections and stresses, for example. In general, the stresses will converge more slowly than the displacement, so it is not sufficient to examine the displacement convergence.

ANSYS: Saving and Restoring Jobs

Saving Your Job

It is good practice to save your model at various points during its creation. Very often you will get to a point in the modeling where things have gone well and you like to save it at the point. In that way, if you make some mistakes later on, you will at least be able to come back to this point.

To save your model, select `Utility Menu Bar -> File -> Save AsJobname.db`. Your model will be saved in a file called `jobname.db`, where `jobname` is the name that you specified in the `Launcher` when you first started ANSYS.

It is a good idea to save your job at different times throughout the building and analysis of the model to backup your work incase of a system crash or other unforeseen problems.

Recalling or *Resuming* a Previously Saved Job

Frequently you want to start up ANSYS and recall and continue a previous job. There are two methods to do this:

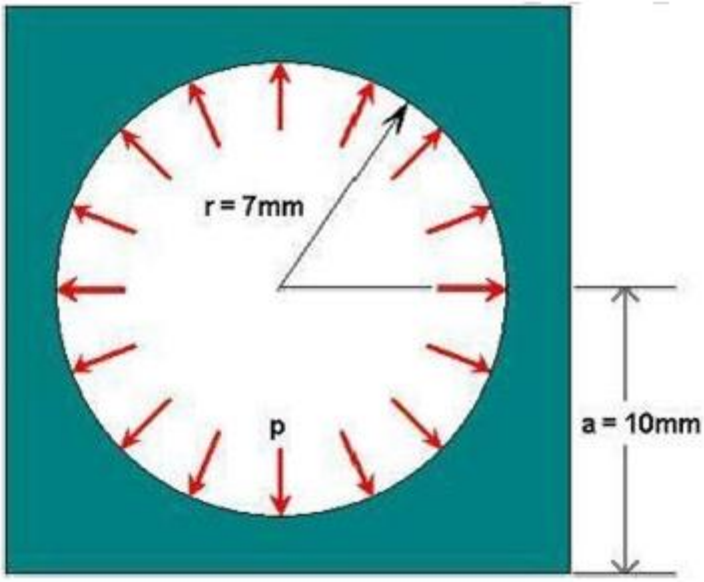
1. Using the `Launcher`...
    - In the ANSYS `Launcher`, select `Interactive...` and specify the previously defined jobname.
    - Then when you get ANSYS started, select `Utility Menu -> File -> Resume Jobname.db` .
    - This will restore as much of your database (geometry, loads, solution, etc) that you previously saved.
2. Or, start ANSYS and select `Utility Menu -> File -> Resume from...` and select your job from the list that appears.

Ex. No. : 1
Date:

STRESS ANALYSIS OF PLATE WITH HOLE

Problem Specification:

Consider the square plate of uniform thickness with a circular hole with dimensions shown inthe figure below. The thickness of the plate is 1 mm. The Young's modulus E =10 e7 MPa and the Poisson ratio is 0.3. A uniform pressure p=1 MPa acts on the boundary of the hole. Assume that plane stress conditions prevail. The stress and displacement fields are to be determined usingANSYS.



Step 1: Start-up and preliminary set-up

Create a folder

Create a folder called plate at a convenient location. We'll use this folder to store filescreated during the ANSYS session.

Start ANSYS

Start > Programs > ANSYS > ANSYS Product launcher

In the window that comes up, enter the location of the folder you just created as your**Working directory** by browsing to it. All files generated during the ANSYS run willbe stored in this directory.

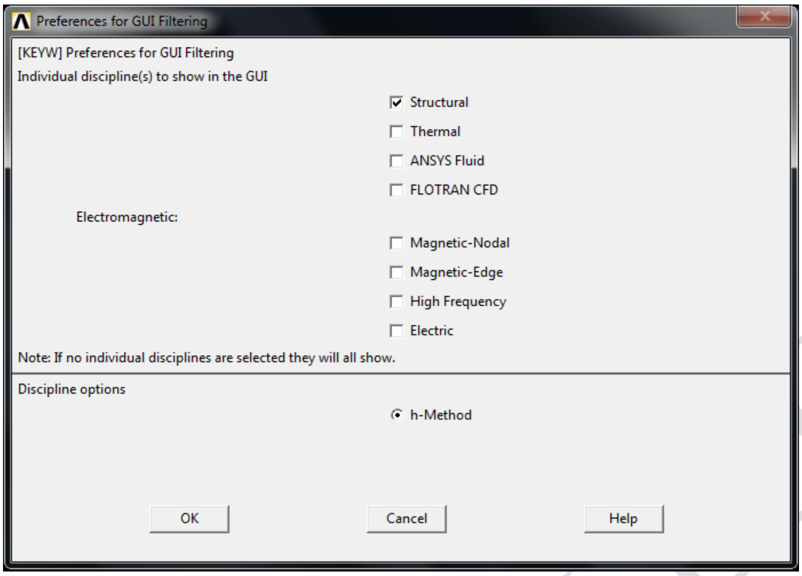
Specify plate as your **Initial job name**. The job name is the prefix used for all filesgenerated during the ANSYS session. For example, when you perform a save operation in ANSYS, it'll store your work in a file called plate.db in your working directory.

For this tutorial, we'll use the default values for the other fields. Click **Run**. Forthis tutorial, we'll use the default values for the other fields. Click **Run**. Thisbrings up the ANSYS interface. To make best use of screen real estate, move thewindows around and resize them so that you approximate this screen arrangement.This way you can read instructions in the browser window and implement them inANSYS.

Set Preferences

As before, we'll more or less work our way down the Main Menu.
Main Menu > Preferences

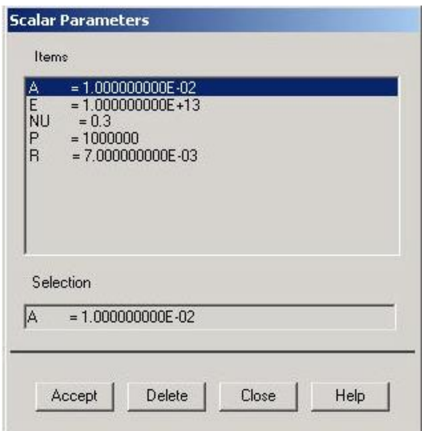
In the Preferences for GUI Filtering dialog box, click on the box next to **Structural** Soa tick mark appears in the box. Click **OK**.



Enter Parameters
For convenience, we'll create scalar parameters corresponding to the plate half-widtha, hole radius r, pressure p, and material properties E and v.
Utility Menu > Parameters > Scalar Parameters

Enter the parameter value for a:
a=10e-3
Click **Accept**.

Similarly, enter the other parameter values and click **Accept** after each.
r=7e-3
p=1e6
E=1e13
Nu=0.3



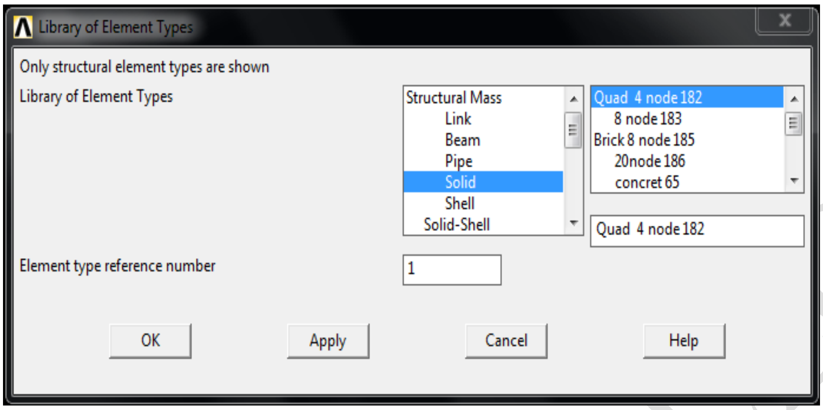
Close the Scalar Parameters window.

Step 2: Specify element type and constants

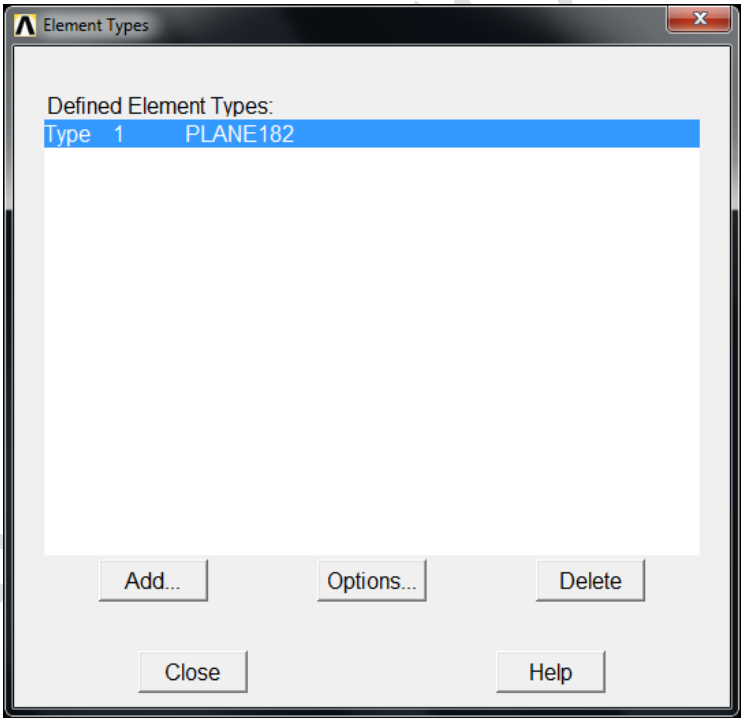
Specify Element Type

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

Pick **Structural Solid** in the left field and **Quad 4 node 182** in the right field. Click**OK** to select this element.



You'll now see the Element Types menu with PLANE182 as the only defined elementtype.



Let's take a look at the online help pages to learn about the properties of this element.

Utility Menu > Help > Help Topics

Select the **Search** tab, type in pictorial summary as the keyword and click **ListTopics**. You should see **Pictorial Summary** as one of the topics listed; double-clickon this. This brings up the Pictorial Summary of Element Types help page. Scroll down to Plane182 under Structural 2-D Solid. Note that the PLANE182 element isdefined by four nodes with two degrees of freedom at each node: translations UXand UY in the (nodal) x and y-directions.

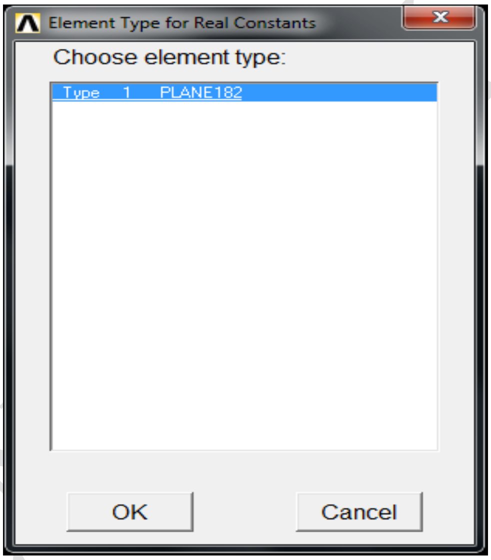
Click on the PLANE182 box to bring up the help page for this element. Read theElement Description and take a look at the figure of the element. Think about whythis element is appropriate for the problem at hand. Minimize the help window.

> If you actually read the Element Description for PLANE182, you'd have noticed that this element can also be used for axisymmetric problems also. In the axi symmetric case, you would choose **Options** for the element in the Element Types menu. Note that in the PLANE182 element type options menu that comes up, under **Elementbehavior**, you have the option of **Axisymmetric**. For the current problem, we'll of course use the default of **Plane stress**. Click **Cancel** to exit the PLANE182 element type options menu retaining the defaults.

Close the Element Types menu.

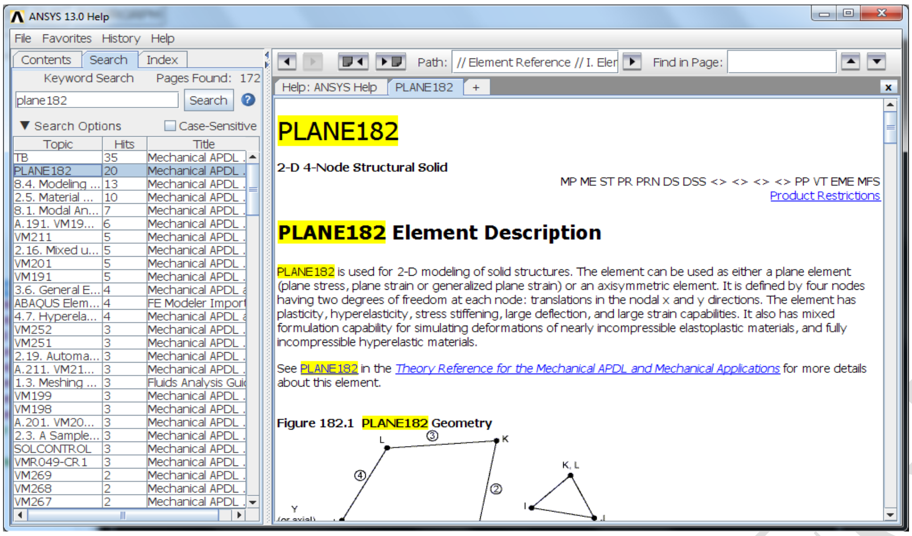
Specify Element Constants

Main Menu > Preprocessor> Real Constants > Add/Edit/Delete > Add
This brings up the Element Type for Real Constants menu with a list of the element types defined in the previous step. We have only one element type and it is automatically selected.



Click **OK**.

You should get a note saying "Please check and change key point setting for elementPLANE182 before proceeding." Close the yellow warning window and the Real Constants menu. To see what this message implies, let's again take a look at the help pages for PLANE182.



Under PLANE182 Input Summary, the documentation says that there are no real constants for this element when KEYOPT (3) =0, 1, 2.

To see what the value of KEYOPT (3) is, bring up the Element Type menu again:

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Options

K3 i.e. KEYOPT (3) is set to **Plane stress**. In the help page, under PLANE182 Input Summary, you can check that plane stress corresponds to KEYOPT (3) =0. Thus, there are no real constants to be specified. That's why we got the "Please check and change key point settings..." warning message. Of course, the ANSYS warning could have been less cryptic but what fun would that be.

Cancel the PLANE182 element type options menu, **Close** the Element Types menu and close the Element Type sticky menu.

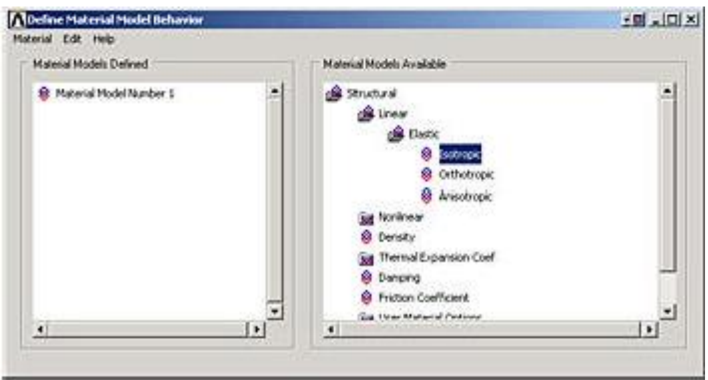
Save your work

Toolbar > SAVE_DB

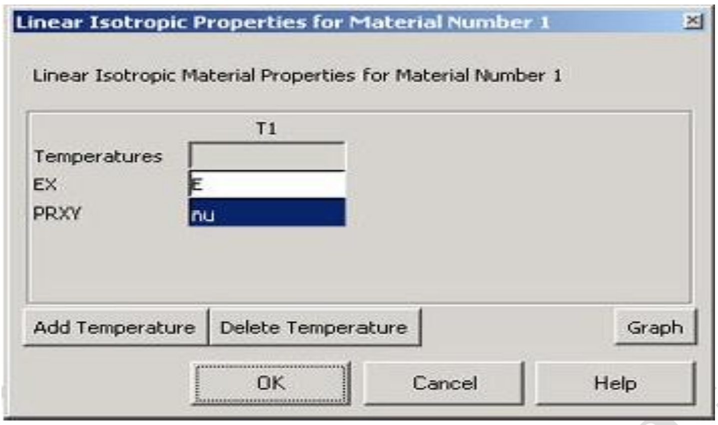
Step 3: Specify material properties.

Main Menu > Preprocessor >Material Props > Material Models....

In the Define Material Model Behavior menu, double-click on **Structural**, **Linear**, **Elastic**, and **Isotropic**.



We'll use the previously defined parameter names while specifying the materialproperties. Enter E for Young's modulus **EX**, nu for Poisson's Ratio **PRXY**. Click**OK**.



To double-check the material property values, double-click on **Linear Isotropic**under **Material Model Number 1** in the Define Material Model Behavior menu.This will show you the current values for EX and PRXY. **Cancel** the LinearIsotropic Properties window.

When you enter parameter names, ANSYS substitutes the corresponding parameter values as soon as you click OK or Apply.

This completes the specification of Material Model Number 1. When we mesh thegeometry later on, we'll use the reference no. 1 to assign this material model. Closethe Define Material Model Behavior menu.

Save your work

Toolbar > SAVE_DB

Step 4: Specify geometry.

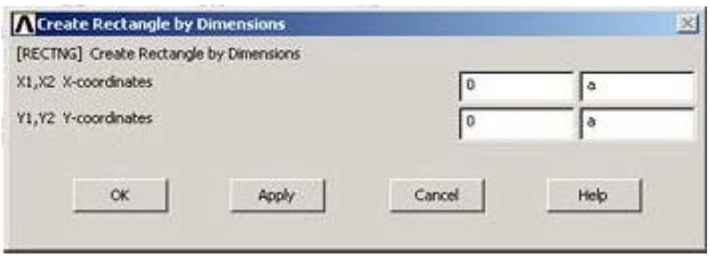
Since the geometry, material properties and loading are all symmetric with respect tothe horizontal and vertical centerlines, we need to model only a quarter of the plate. We will take the origin of the coordinate system to be at the center of the hole andmodel only the top right quadrant. We'll create the geometry by creating a square areaof side a and subtracting the circular sector of radius r from it.

Create the Square

Main Menu > Preprocessor > Modeling >Create > Areas >Rectangle > ByDimensions
X1 and **X2** are the x-coordinates of the left and right edges of the square, respectively.

Enter 0 for **X1**,a for**X2**.

Y1 and **Y2** are the y-coordinates of the bottom and top edges of the squarerespectively. Enter 0 for **Y1**, a for**Y2**.



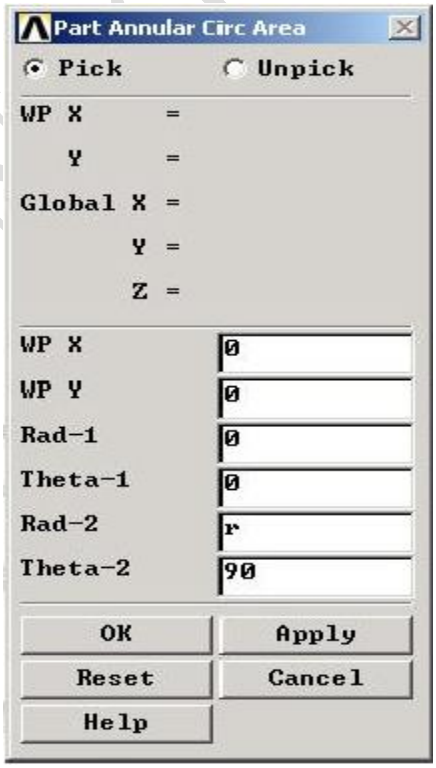
Click **OK**. You should see a square appear in the graphics window.

Create the Circular Sector

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > PartialAnnulus

WP X and **WP Y** are the x- and y-coordinates of the center of the circular arc. Soenter 0 for both **WP X** and **WP Y**. (WP refers to the Working Plane which by defaultcoincides with the global Cartesian coordinate system. We won't have to worry aboutthe working plane in this friendly example.)

Rad-1 is the radius of the inner circular arc. We want to create a solid rather than anannular arc. Enter 0 for **Rad-1** to create a solid arc.
Rad-2 is the (outer) radius of the arc. Since we had defined the whole radius asParameter r earlier, enter r for **Rad-2**.
Theta-1 and **Theta-2** are the starting and ending angles of the arc, respectively.These angles need to be specified in degrees. Enter 0 for **Theta-1** and 90 for **Theta-2**. Click **OK**.

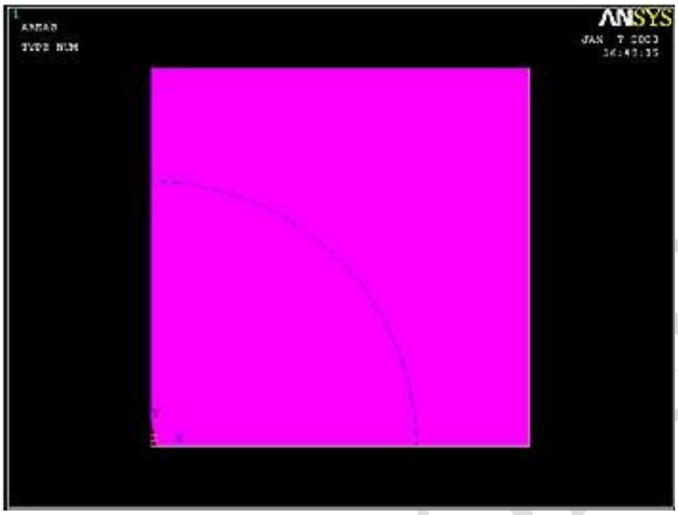


This will create and draw the circular sector. You'll see a white line denoting thecircular sector.

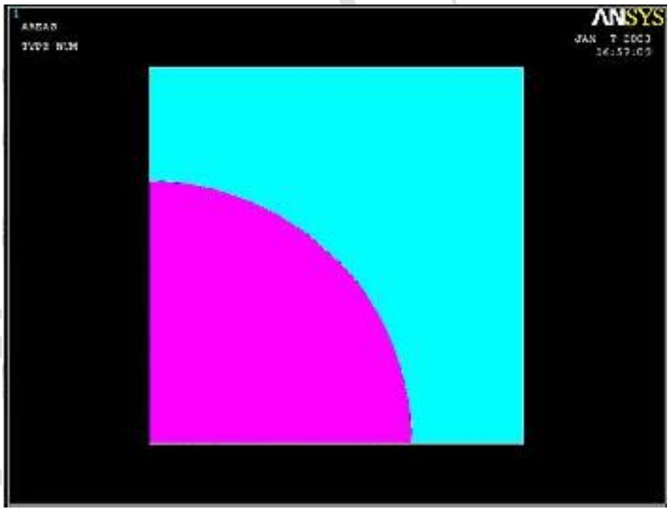
Subtract Circular Sector from Square

Main Menu > Preprocessor >Modeling > Operate > Booleans > Subtract >Areas

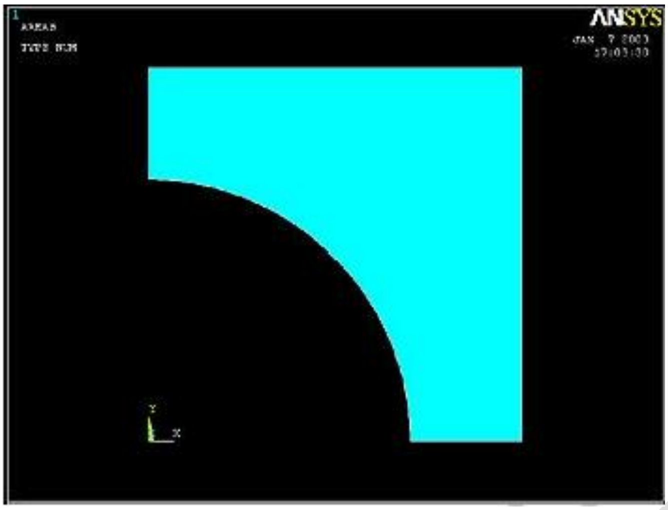
In the Input window, ANSYS tells you to "pick or enter base areas from which tosubtract". So we pick the square area as follows: Hold down the left mouse button,move the cursor over the areas until the square is selected (it will change color) andrelease the left mouse button. Click **OK**.



In the Input window, ANSYS now tells you to "pick or enter areas to be subtracted".So, select the circular sector by holding down and releasing the left mouse button.Click **OK**.



If you did this correctly, you will see that the circular sector has been subtracted outfrom the square area.



You can also select areas during the Boolean subtract operation by simply clicking on them but it becomes difficult to select areas (and other components) in this fashion in more complicated geometries. That's why I made you use the "holding-down-the-mouse-and-releasing" technique.

If you picked an area incorrectly, you can unpick it by clicking the right mouse button and selecting the area. The cursor changes to a downward arrow during an unpick operation. Right-click to return pick mode.

Save Your Work

Toolbar > SAVE_DB

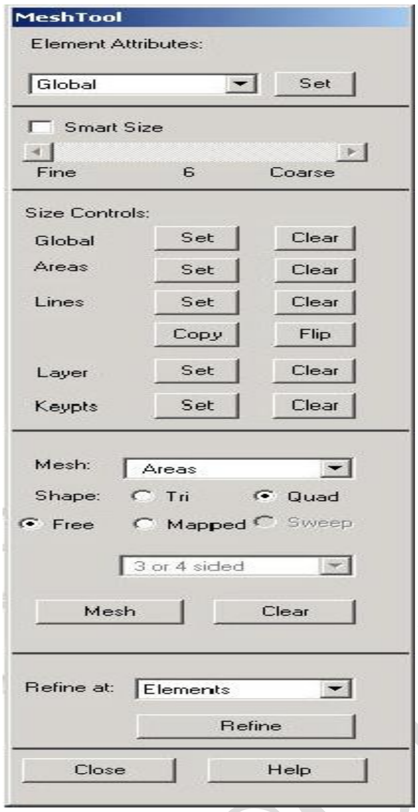
Step 5: Mesh geometry.

Bring up the MeshTool:

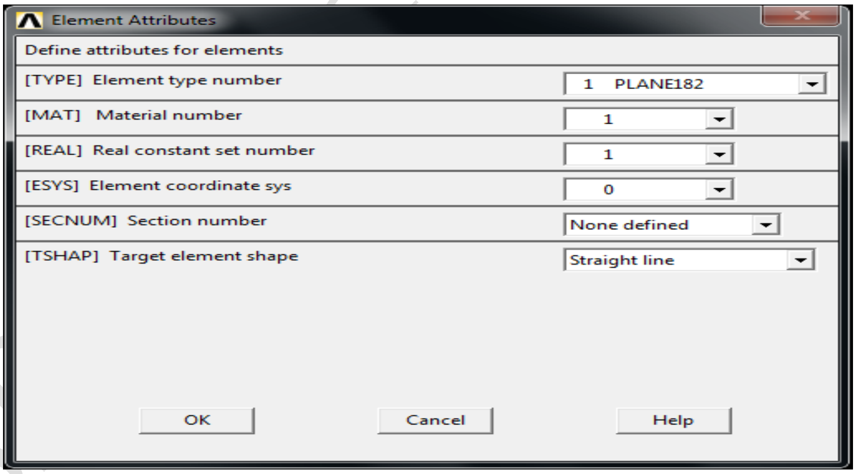
Main Menu > Preprocessor >MeshTool
The MeshTool is used to control and generate the mesh.

<u>Set Meshing Parameters</u>

We'll now specify the element type, real constant set and material property set to beused in the meshing. Since we have only one of each, we can assign them to theentire geometry using the **Global** option under **Element Attributes**.
Make sure **Global** is selected under **Element Attributes** and click on **Set**.



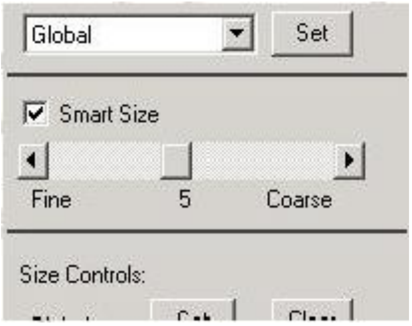
This brings up the Meshing Attributes menu. You will see that the correct elementtype and material number are already selected since we have only one of each. Recallthat no real constants need to be defined for PLANE182 element type with the planestress key option.



Click **OK**. ANSYS now knows what element type and material type to use for themesh.

Set Mesh Size

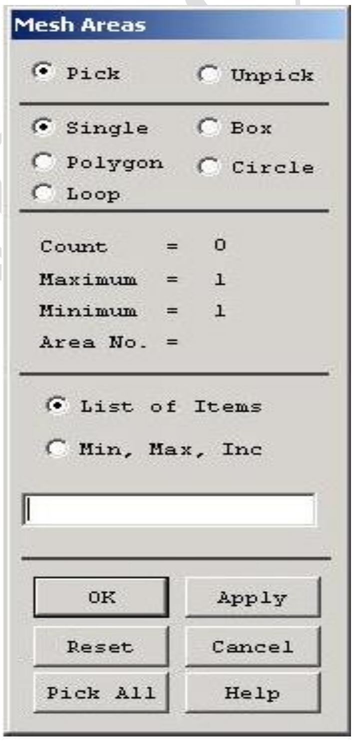
Instead of setting the mesh size at each boundary, we'll use the Smart Size optionwhich enables automatic element sizing. Click on the SmartSize checkbox so that atickmark appears in it.



The only input necessary for the SmartSize option is the overall element size level formeshing. The element size level determines the fineness of the mesh. Its value iscontrolled by the slider shown in the above picture. Change the setting for the overallelement size level to **5** by moving the slider under **SmartSize**from the left.

Mesh Areas

In the MeshTool, make sure **Areas** is selected in the drop-down list next to **Mesh**.This means the geometry components to be meshed are areas (as opposed to lines orvolumes). We'll use quadrilateral elements. So make sure the default option of **Quad**is selected under **Shape**. We'll also use the default of**Free**meshing.
Click on the **Mesh** button. This brings up the pick menu.



In the Input window, ANSYS tells you to "pick or enter areas to be meshed". Sincewe have only one area to be meshed, click on **Pick All**. The geometry has beenmeshed and the elements are plotted in the Graphics window. **Close** the MeshTool.

Save Your Work

Toolbar > SAVE_DB

Step 6: Specify boundary conditions.

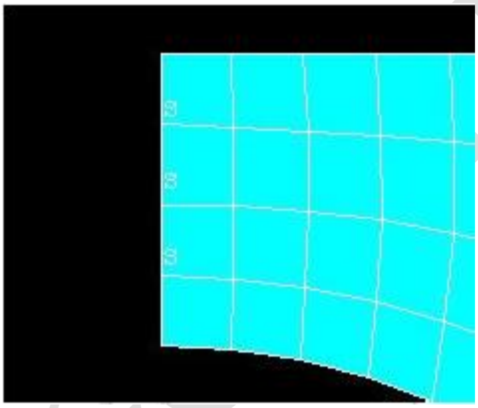
Next, we step up to the plate to define the displacement constraints and loads. Recallthat in ANSYS terminology, the displacement constraints are also "loads". As in the truss tutorial, we'll apply the loads to the geometry rather than the mesh. That way wewon't have to reapply the loads on changing the mesh.

Apply Symmetry Boundary Conditions

ANSYS provides the option of applying a "symmetry boundary condition" along lines of symmetry.

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural >Displacement > Symmetry B.C. > On Lines

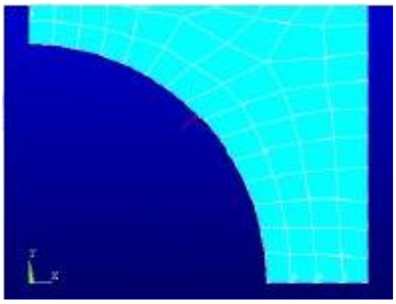
Select the straight lines corresponding to the left and bottom edges (which are the lines of symmetry for this problem) by clicking on them. Click **OK** in the pick menu. The symbol s appears along these lines indicating that the symmetry B.C. is applied along these lines.



Apply Pressure

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural >Pressure > On Lines

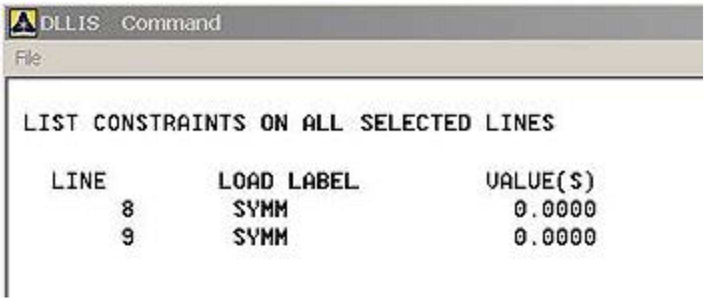
Select the circular arc and click **OK**. This brings up the Apply Pressure on Linesmenu. Enter p for **Value** and click **OK**. A single red arrow denotes the pressure and the direction in which it is acting.



Check Loads

Let's check that the displacement constraints have been applied correctly.

Utility Menu > List > Loads > DOF constraints > On All Lines



Symmetry BCs are applied on lines 8 and 9. Turn on line numbering:

Utility Menu >PlotCtrls> Numbering

Turn on **Line numbers** and click **OK**. Are lines L8 and L9 the ones on which you want the symmetry BCs?

Similarly, check that the pressure is applied correctly using **Utility Menu > List >Loads > Surface Loads > On All Lines**. Note that **VALI** and **VALJ** would be different if the applied pressure were linearly varying along the line.

Turn off line numbering: **Utility Menu >PlotCtrls> Numbering**. Turn off **Line numbers**and click **OK**.

Save Your Work

Toolbar > SAVE_DB

Step 7: Solve.

Enter solution module:

Main Menu > Solution

Enter check in the Input window. If the problem has been set up correctly, there will be no errors or warnings reported. If you look in the Output window, you should seethe message: The analysis data was checked and no warnings or errors were found.

Main Menu > Solution > Solve > Current LS

Recall from the truss tutorial that this solves the current load step (LS) i.e. the current loading conditions. In this problem also, there is only one load step.

Review the information in the /STATUS Command window. Close this window.

Click **OK** in Solve Current Load Step menu.

ANSYS performs the solution and a yellow window should pop up saying "Solution is done!”

Verify that ANSYS has created a file called plate.rst in your working directory. This file contains the results of the (previous) solve.

Step 8: Postprocessor the Results

Enter the Postprocessing module to analyze the solution.

Main Menu > General Postproc
Plot Deformed Shape

Main Menu > General Postproc> Plot Results > Deformed Shape

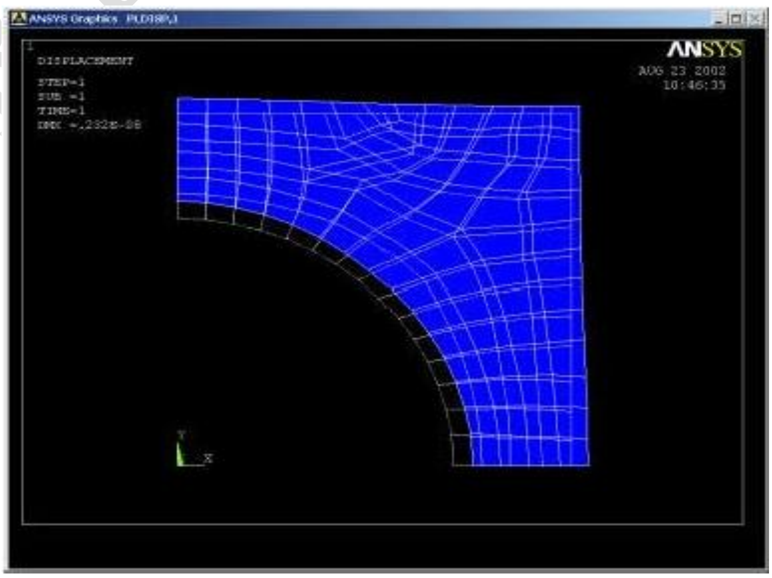
Select **Def + undeformed**and click **OK**.

This plots the deformed and undeformed shapes in the Graphics window. The maximum deformation DMX is 0.232E-08m as reported in the Graphics window. Note that the deformation is magnified in the plot so as to be visible.

The deformation would be better visible if the foreground and background were not of the same color. Turn off the background:

Utility Menu >PlotCtrls> Style > Background > Display Picture Background

To get the background back, you just have to select this again.



Animate the deformation:

Utility Menu >PlotCtrls> Animate > Deformed Shape...
Select **Def + undeformed**and click **OK**. Select **Forward Only** in the AnimationController.

The left and bottom edges move parallel to them which means that the full deformed plate is also symmetric about these edges. This shows that the symmetry boundary condition at these edges is imposed correctly. The circular edge of the holemoves outward which is what one would expect from the outward pressure acting along it. Thus, the deformation of the structure agrees with the applied boundary conditions and matches with what one would expect from intuition.

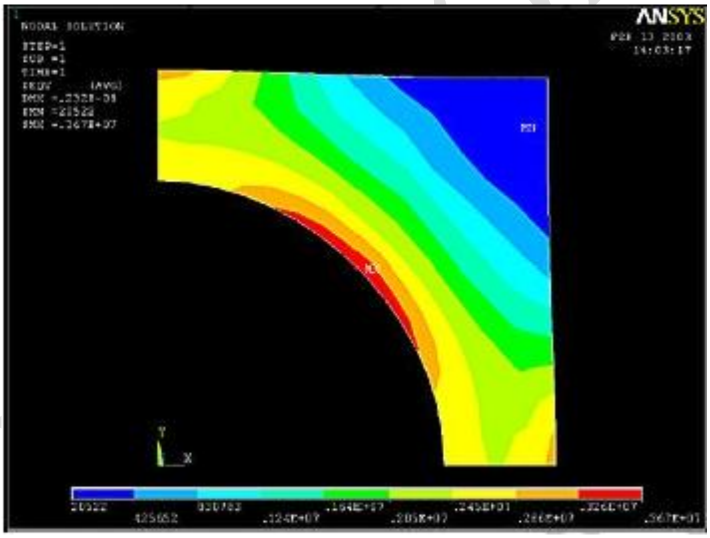
Close the Animation Controller.

Plot Nodal Solution of von Mises Stress

To display the von Mises stress distribution as continuous contours, select

Main Menu > General Postproc> Plot results > Contour Plot > Nodal Solution

Select **Stress** from the left list, **von Mises SEQV** from the right list and click **OK**.



The contour plot will show you the locations of the maximum and minimum values with the labels MX and MN, respectively. Are these locations where you expect them? SMX and SMN values reported in the Graphics window are the corresponding maximum and minimum stress values.

The diagonal is an additional line of symmetry. How symmetric is your result about the diagonal?

Save this plot to a file:

Utility Menu >PlotCtrls> Hard Copy > To File

Select the file format you want and type in a filename of your choice under **saveto:** and click **OK.** Check that the file has been created in your working directory.

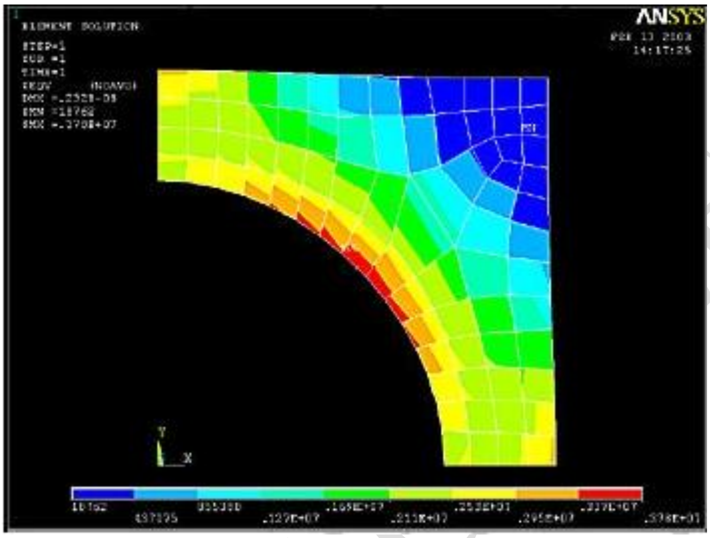
When you plot the "Nodal Solution", ANSYS obtains a continuous distribution as follows: 1. It determines the average at each node of the values of all elements connected to the node. 2. Within each element, it linearly interpolates the average nodal value obtained in the previous step.

Plot Element Solution of von Mises Stress

To obtain results without nodal averaging, select

Main Menu > General Postproc> Plot results > Contour Plot > Element Solution

Select **Stress** from the left list, **von Mises SEQV** from the right list and click **OK**.This displays the von Mises stress results as discontinuous element contours.



Save this plot to a file: **Utility Menu >PlotCtrls> Hard Copy > To File**

Element solution contours are determined by linear interpolation within each elementbut no nodal averaging is performed. The discontinuity between contours of adjacent elements is an indication of the gradient across elements. The inter-element discontinuities in our solution are relatively small compared to the stress levels. This indicates that the mesh resolution is reasonably good.

Query Results

To determine the value of the first principal stress sigma1 at a selected location, select

Main Menu > General Postproc> Query Results >SubgridSolu

This brings up the Query Subgrid Solution Data menu. Select **Stress** from the left list, **1st principal S1** from the right list and click **OK**.
This brings up the pick menu. You can click on any location in the geometry andANSYS will print the sigma1 value at that location. Try querying the values at a fewlocations. Note that the coordinates of the picked location and the corresponding solution value are reported in the pick menu.

Cancel the pick menu.

Step 9: Validate the results

It is **very important** that you take the time to check the validity of your solution.This section leads you through some of the steps you can take to validate yoursolution.

Simple Checks

Does the deformed shape look reasonable and agree with the applied boundaryconditions? We checked this in step 8.
Do the reactions at the supports balance the applied forces for static equilibrium? Tocheck this, select

Main Menu > General Postproc> List Results > Reaction Solution

Select **allstrucforc F** for **Item to be listed** and click **OK.**

The total reaction force in the x-direction is -7000 N.

Applied force = (pressure) x (projected distance in x-direction of the line along whichthe constant pressure acts) = (p) (r) = 7000 N in positive x-direction.

So the reaction cancels out the applied force in the x-direction. Similarly, you cancheck that this is true in the y-direction also.

Refine Mesh

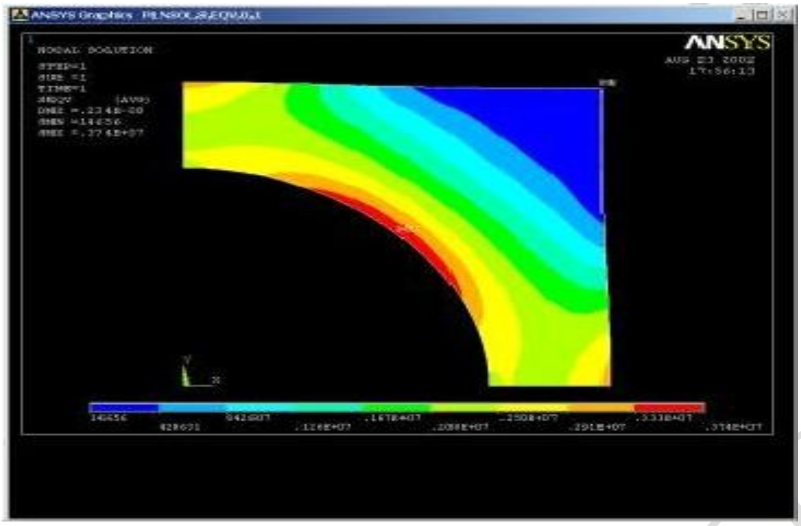
Let's repeat the calculations on a mesh with overall element size level underSmartSize set to 4 instead of 5 and compare the results on the two meshes. Delete thecurrent mesh:

Main Menu > Preprocessor > Mesh Tool

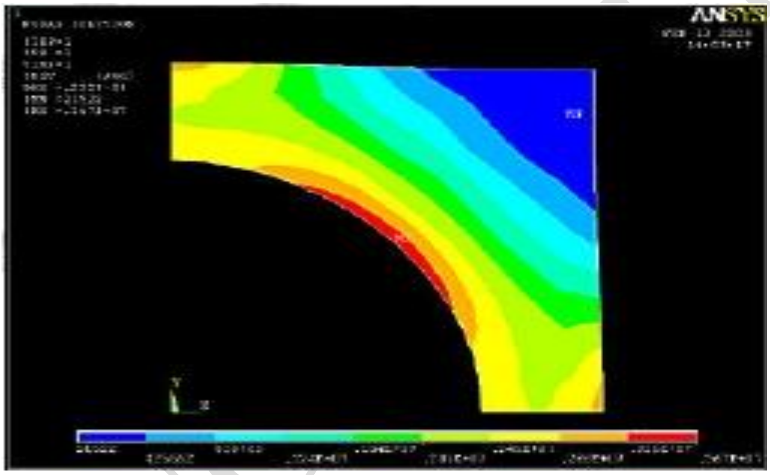
Select **Clear** under **Mesh:** and **Pick all**in the pick menu. The mesh is deleted.
Set the overall element size level under SmartSize to 4 by dragging the slider to theleft. Click on **Mesh** and **Pick All**.
In the Output window, check how many elements are contained in this mesh? Yournew mesh should have 276 quadrilateral elements.
Obtain a new solution: **Main Menu > Solution > Solve > Current LS**

Plot nodal solution of the von Mises stress:
Main Menu > General Postproc> Plot results > Contour Plot > Nodal Solution

Select **Stress** from the left list, **von Mises SEQV** from the right list and click **OK**.



Compare this with the von Mises contours for the previous mesh:



The two results compare well with the finer mesh contours being smoother asexpected. Compare the maximum stress and displacement values:

.	Coarser Mesh	Finer Mesh
DMX	0.232e-8m	0.234e-8m
SMX	3.64MPa	3.74MPa

The maximum displacement value changes by less than 1% and the maximum vonMises stress value by less than 3%. This indicates that the meshes used provideadequate resolution.

Exit ANSYS

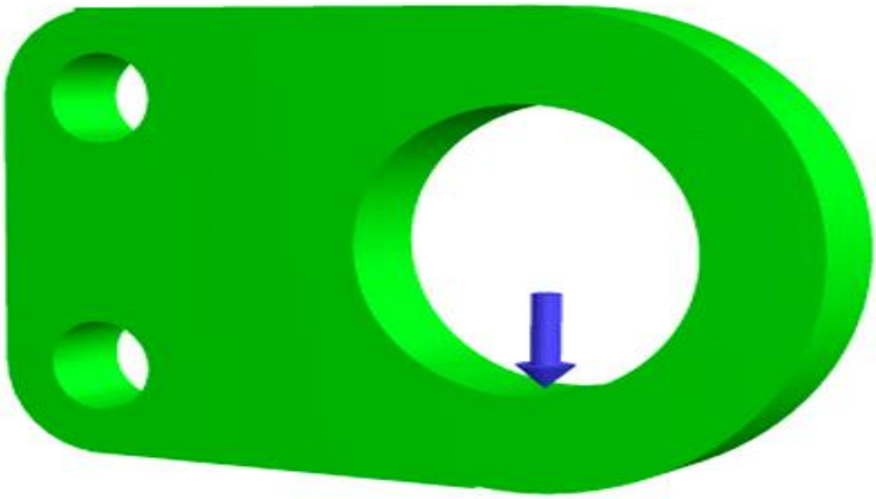
Utility Menu > File > Exit

Select **Save Everything** and click **OK.**

Ex. No. : 2
Date:

STRESS ANALYSIS OF A BRACKET

The problem to be modeled in this example is a simple bracket shown in the following figure. This bracket is to be built from a 20 mm thick steel plate. A figure of the plate is shown below.



This plate will be fixed at the two small holes on the left and have a load applied to the larger hole on the right.

Preprocessing: Defining the Problem

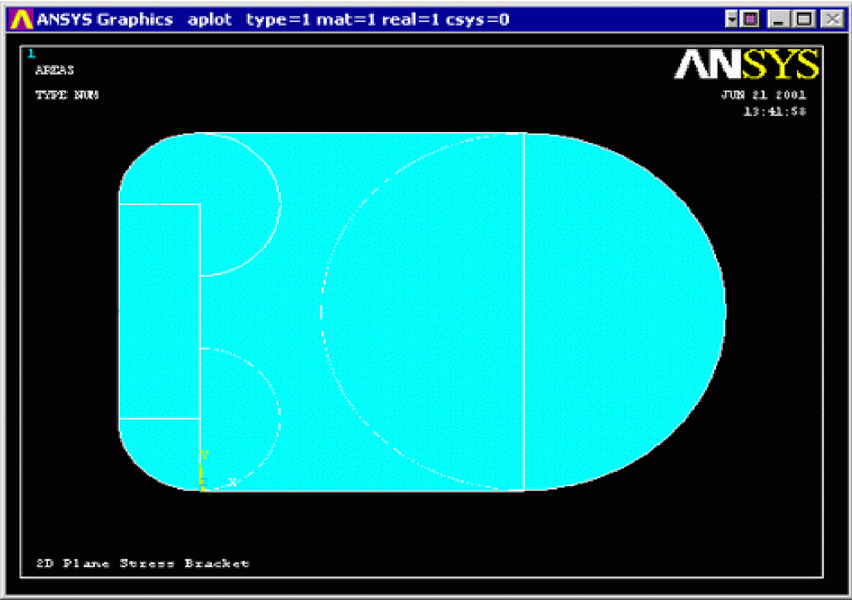
1. **Give the Bracket example a Title**
   Utility Menu > File > Change Title
2. **Form Geometry**
   Boolean operations will be used to create the basic geometry of the Bracket.
   a. Create the main rectangular shape
      The main rectangular shape has a width of 80 mm, a height of 100mm and the bottom left corner is located at coordinates (0,0)
   b. Create the circular end on the right hand side
      The center of the circle is located at (80,50) and has a radius of 50 mm
   c. Now create a second and third circle for the left hand side using the following dimensions:

parameter	circle 2	circle 3
XCENTER	0	0
YCENTER	20	80
RADIUS	20	20

   d. Create a rectangle on the left hand end to fill the gap between the two small circles.

XCORNER	-20
YCORNER	20
WIDTH	20
HEIGHT	60

   e. Screen should now look like the following

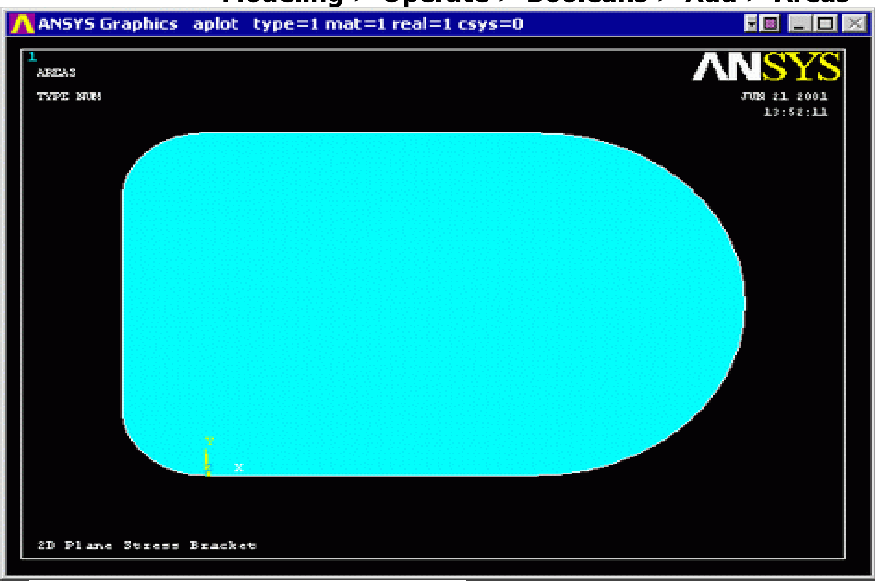


f. Boolean Operations - Addition

Now add these five discrete areas together to form one area.

- To perform the Boolean operation, from the Preprocessor menu select:

  Modeling > Operate > Booleans > Add > Areas



g. Create the Bolt Holes from this plate.

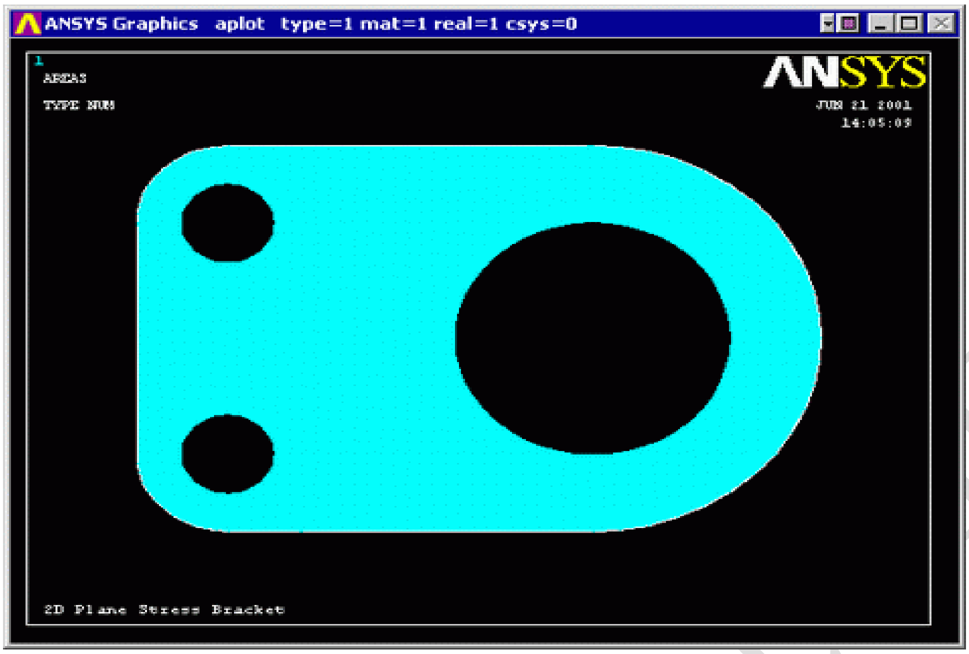
- Create the three circles with the parameters given below:

parameter	circle 1	circle 2	circle 3
WP X	80	0	0
WP Y	50	20	80
radius	30	10	10

- Select

  Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

- Select the base areas from which to subract (the large plate that was created)
- Next select the three circles that we just created. Click on the three circles that you just created and click 'OK'.



3. **Define the Type of Element**
   As in the verification model, PLANE82 will be used for this example
   - **Preprocessor > Element Type > Add/Edit/Delete**
   - Use the 'Options...' button to get a plane stress element with thickness

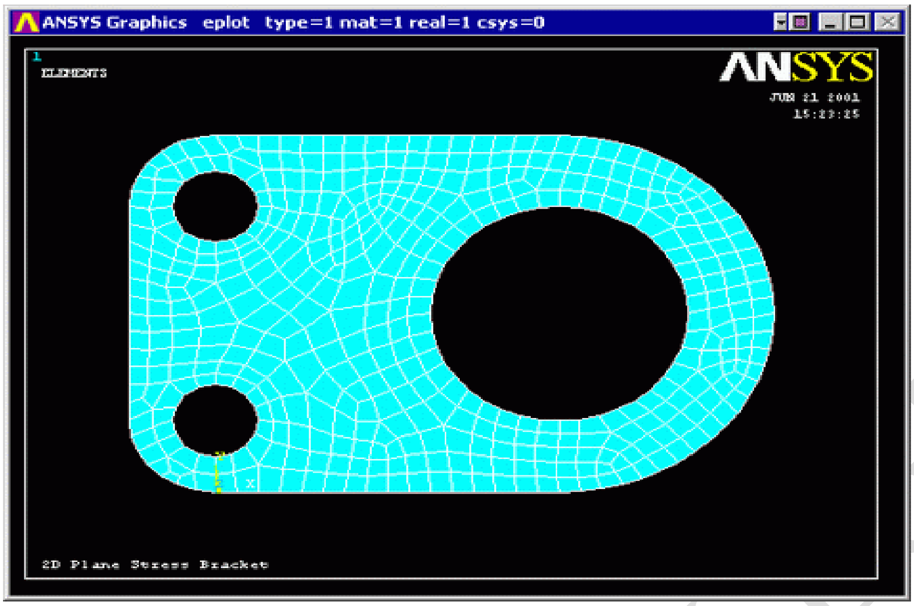
   Define Geometric Constraints
   - **Preprocessor > Real Constants > Add/Edit/Delete**
   - Enter a thickness of 20mm.

   Element Material Properties
   - **Preprocessor > Material Props > Material Library > Structural > Linear > Elastic > Isotropic**

Properties of Steel.	EX 200000
	PRXY 0.3

   Mesh Size
   - **Preprocessor > Meshing > Size Cntrls> Manual Size > Areas > All Areas**
   - Select an element edge length of 5. Again, we will need to make sure the model has converged.

   Mesh
   - **Preprocessor > Meshing > Mesh > Areas > Free** and select the area when prompted
     (Alternatively, the command line code for the above step is **AMESH,ALL**)

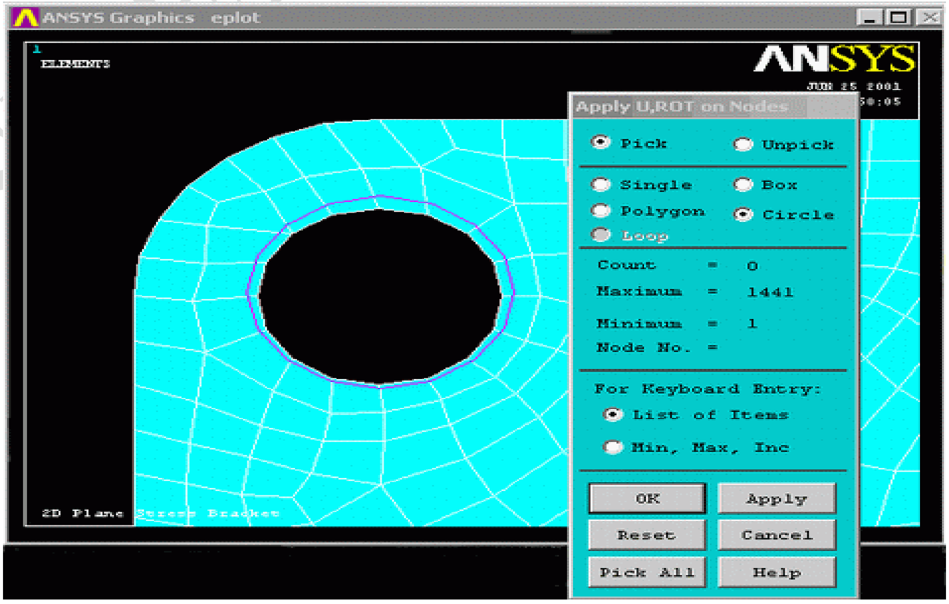


Saving Your Job
Utility Menu > File > Save as

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.

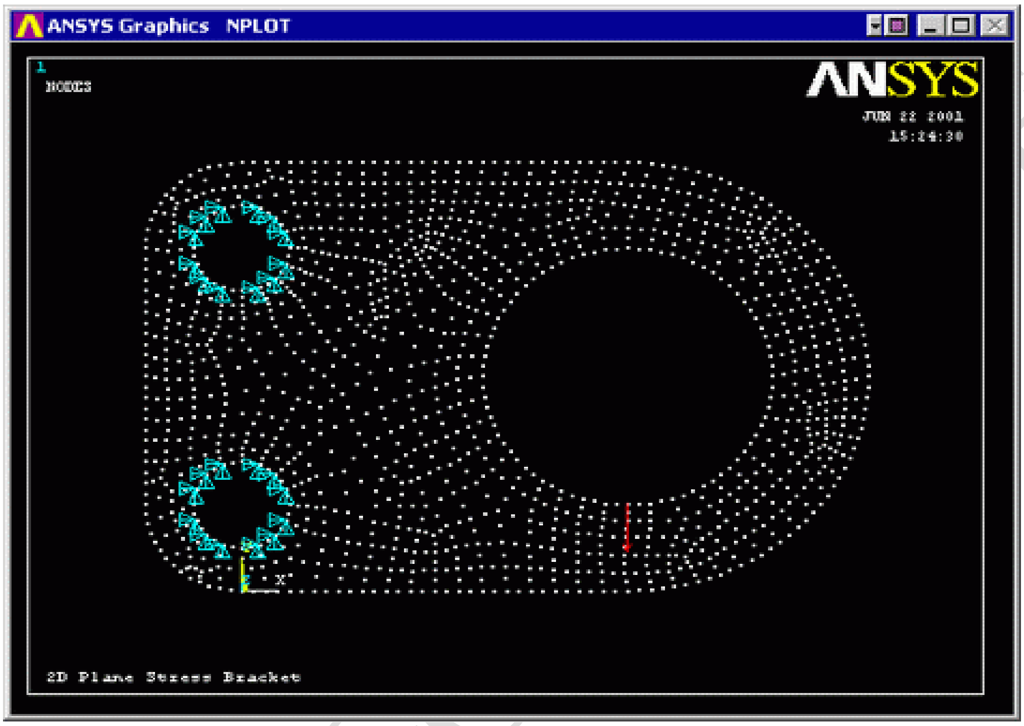
1. **Define Analysis Type**
    - **'Solution' > 'New Analysis'** and select 'Static'.

2. **Apply Constraints**
   As illustrated, the plate is fixed at both of the smaller holes on the left hand side.
    - **Solution > Define Loads > Apply > Structural > Displacement > On Nodes**
    - Instead of selecting one node at a time, you have the option of creating a box, polygon, or circle of which all the nodes in that area will be selected. For this case, select 'circle' as shown in the window below. Click at the center of the bolt hole and drag the circle out so that it touches all of the nodes on the border of the hole.



- Click on 'Apply' in the 'Apply U,ROT on Lines' window and constrain all DOF's in the 'Apply U,ROT on Nodes' window.
- Repeat for the second bolt hole.

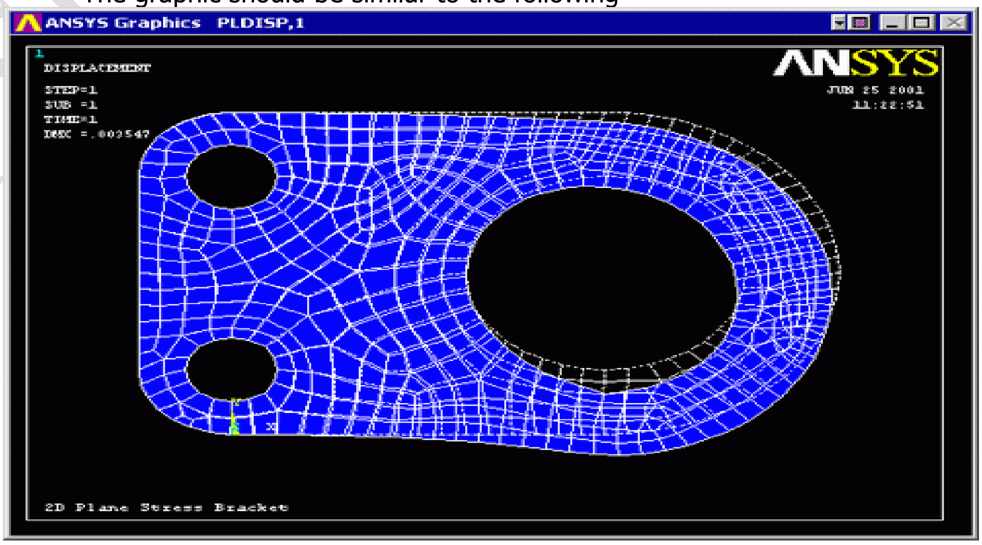
3. **Apply Loads**
   As shown in the diagram, there is a single vertical load of 1000N, at the bottom of the large bolt hole. Apply this force to the respective keypoint (**Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints** Select a force in the y direction of -1000)
   The applied loads and constraints should now appear as shown below.



4. **Solving the System**
   Solution > Solve > Current LS

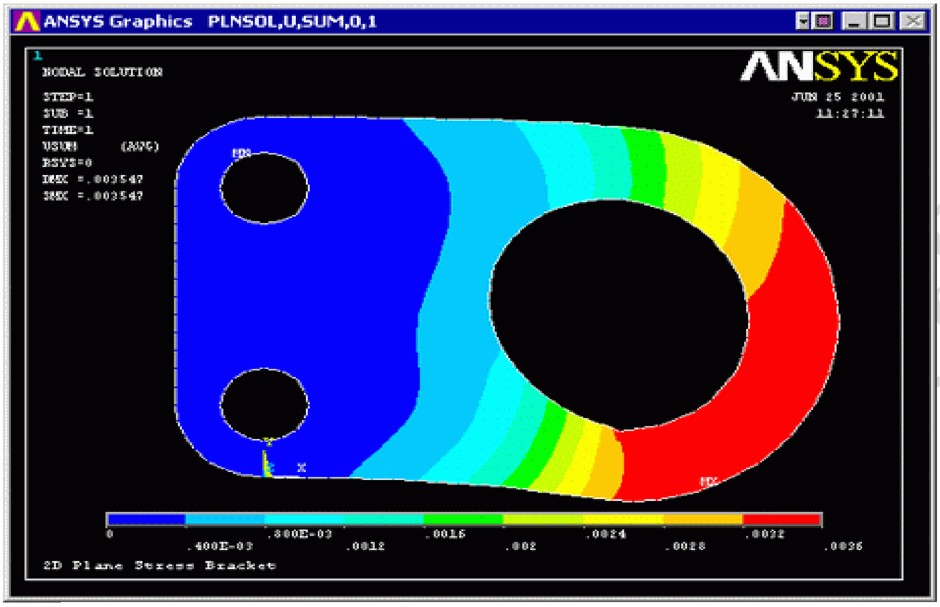
Post-Processing: Viewing the Results

1. **Convergence using ANSYS**
   As shown previously, it is necessary to prove that the solution has converged. Reduce the mesh size until there is no longer a sizeable change in your convergence criteria.
2. **Deformation**
   - **General Postproc> Plot Results >Def + undeformed** to view both the deformed and the undeformed object.
     The graphic should be similar to the following



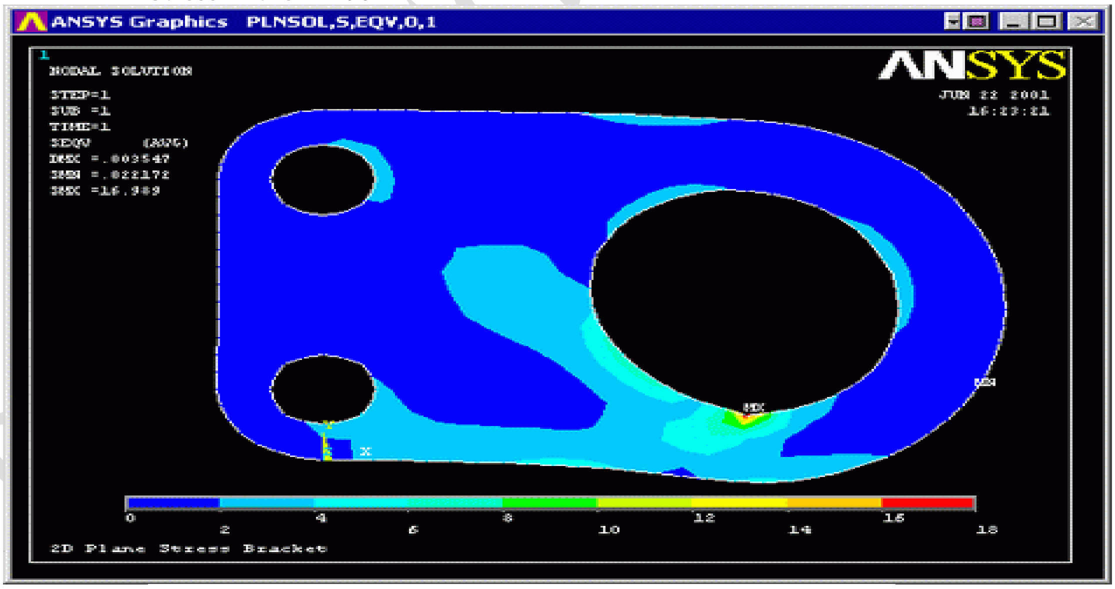
- Observe the locations of deflection. Ensure that the deflection at the bolt hole is indeed 0.

3. **Deflection**
   - To plot the nodal deflections use **General Postproc> Plot Results > Contour Plot > Nodal Solution** then select DOF Solution - USUM in the window.



   - Alternatively, obtain these results as a list. (**General Postproc> List Results > Nodal Solution...**)
   - Are these results what you expected? Note that all translational degrees of freedom were constrained to zero at the bolt holes.

4. **Stresses**
   - **General Postproc> Plot Results > Nodal Solution...** Then select von Mises Stress in the window.



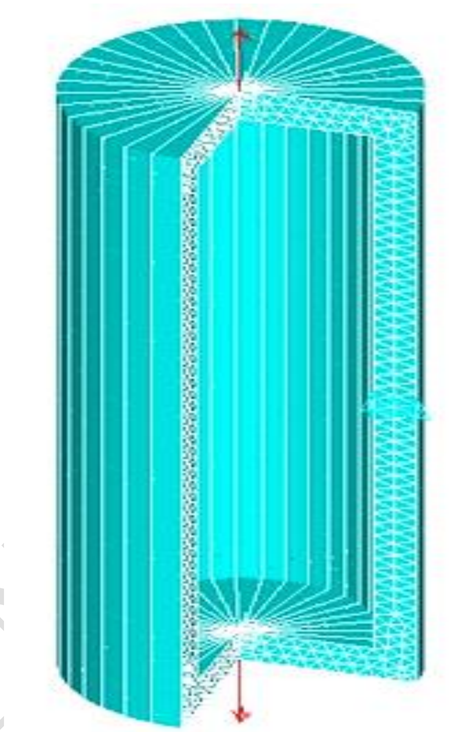
   - List the von Mises stresses to verify the results at certain nodes
     General Postproc> List Results. Select **Stress, Principals SPRIN**

Ex. No. : 3 **STRESS ANALYSIS OF AN AXI-SYMMETRIC COMPONENT**
Date:

Problem Description:

The model will be that of a closed tube made from steel. Point loads will be applied at the center of the top and bottom plate to make an analytical verification simple to calculate. A 3/4 cross section view of the tube is shown below.

As a warning, point loads will create discontinuities in the model near the point of application. If you chose to use these types of loads in your own modeling, bevery careful and be sure to understand the theory of how the FEA package is applyingthe load and the assumption it is making. In this case, we will only be concernedabout the stress distribution far from the point of application, so the discontinuities will have a negligible effect.



Preprocessing: Defining the Problem

1. **Give example a Title**
Utility Menu > File > Change Title .../title, Axisymmetric Tube

2. **Open preprocessor menu**
ANSYS Main Menu > Preprocessor/PREP7

3. **Create Areas**
Preprocessor > Modeling > Create > Areas > Rectangle > By DimensionsRECTNG, X1, X2, Y1, Y2.
For an axisymmetric problem, ANSYS will rotate the area around the y-axis at x=0.Therefore, to create the geometry mentioned above, we must define a U-shape.We are going to define 3 overlapping rectangles as defined in the following table:

Rectangle	X1	X2	Y1	Y2
1	0	20	0	5
2	15	20	0	100
3	0	20	95	100

4. **Add Areas Together**
Preprocessor > Modeling > Operate > Booleans > Add >Areas ADD, ALL

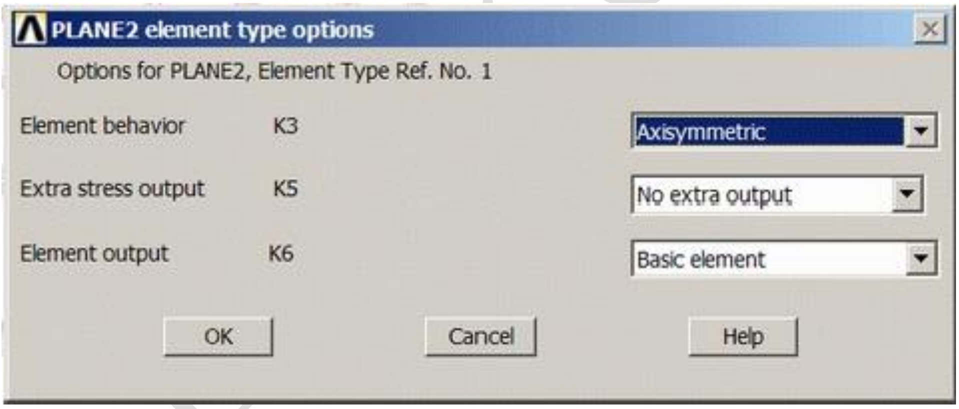
Click the Pick All button to create a single area.

5. **Define the Type of Element**
Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use the PLANE182 (Structural, Solid, Triangle 6node) element. This element has 2 degrees of freedom (translation along the X and Y axes).

Many elements support axisymmetry, however if the Ansys Elements Reference (which can be found in the help file) does not discuss axisymmetric applications for aparticular element type, axisymmetry is not supported.

6. **Turn on Axisymmetry**
While the Element Types window is still open, click the **Options...** button.Under Element behavior K3 select **Axisymmetric**.



7. **Define Element Material Properties.**

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic >Isotropic
In the window that appears, enter the following geometric properties for steel:
1. Young's modulus EX: 200000
2. Poisson's Ratio PRXY: 0.3

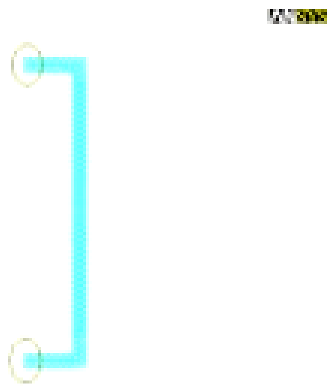
8. **Define Mesh Size**
Preprocessor > Meshing > Size Cntrls>ManualSize> Areas > All Areas
For this example we will use an element edge length of 2mm.
9. **Mesh the frame**
Preprocessor > Meshing > Mesh > Areas > Free > click 'Pick All'
Your model should know look like this:

Solution Phase: Assigning Loads and Solving

1. **Define Analysis Type**
    - Solution > Analysis Type > New Analysis > StaticANTYPE,0

2. **Apply Constraints**
    - Solution > Define Loads > Apply > Structural > Displacement >Symmetry B.C. > On Lines

Pick the two edges on the left, at x=0, as shown below. By using the symmetry B.C.command, ANSYS automatically calculates which DOF's should be constrained forthe line of symmetry. Since the element we are using only has 2 DOF's per node, wecould have constrained the lines in the x-direction to create the symmetric boundaryconditions.



- Utility Menu > Select > Entities

Select **Nodes** and **By Location** from the scroll down menus. Click **Y coordinates**and type **50** into the input box as shown below, then click OK.

- Solution > Define Loads > Apply > Structural > Displacement > On Nodes >PickAll

Constrain the nodes in the y-direction (UY). This is required to constrain the model in space; otherwise it would be free to float up or down. The location to constrain the model in the y direction (y=50) was chosen because it is along a symmetry plane. Therefore, these nodes won't move in the y-direction according to theory.

3. Utility Menu > Select > Entities

In the select entities window, click **Select All** to reselect all nodes. It is important to always reselect all entities once you've finished ensuring future commands are applied to the whole model and not just a few entities. Once you've clicked Select All, click on **Cancel** to close the window.

4. Apply Loads:

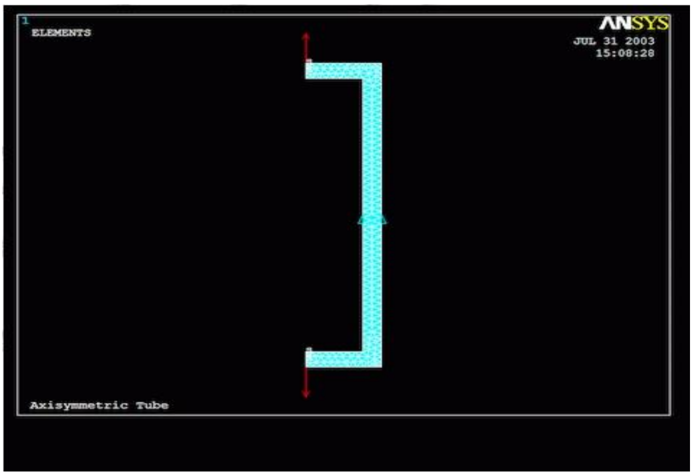
- Solution > Define Loads > Apply > Structural > Force/Moment > On Key points

Pick the top left corner of the area and click OK. Apply a load of 100 in the FY direction.

- Solution > Define Loads > Apply > Structural > Force/Moment >On Key points

Pick the bottom left corner of the area and click OK. Apply a load of -100 in the FY direction.

- The applied loads and constraints should now appear as shown in thefigure.



5. Solve the System:

- Solution > Solve > Current LS SOLVE

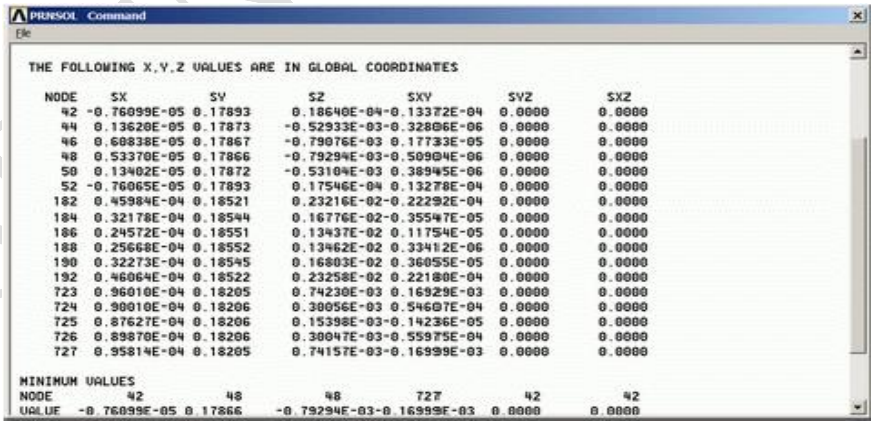
Determine the Stress Through the Thickness of the Tube.

- Utility Menu > Select > Entities...

Select **Nodes > By Location > Y coordinates** and type **45,55**in the Min,Max box, as shown below and click OK.

- General Postproc> List Results > Nodal Solution > Stress >Components SCOMP

The following list should pop up.


PRNSOL Command

File

THE FOLLOWING X,Y,Z VALUES ARE IN GLOBAL COORDINATES

NODE	SX	SY	SZ	SXY	SYZ	SXZ
42	-0.76099E-05	0.17893	0.18640E-04	-0.13372E-04	0.0000	0.0000
44	0.13620E-05	0.17873	-0.52933E-03	-0.32806E-06	0.0000	0.0000
46	0.60838E-05	0.17867	-0.79076E-03	0.17733E-05	0.0000	0.0000
48	0.53370E-05	0.17866	-0.79294E-03	-0.50904E-06	0.0000	0.0000
50	0.13402E-05	0.17872	-0.53104E-03	0.38945E-06	0.0000	0.0000
52	-0.76065E-05	0.17893	0.17546E-04	0.13278E-04	0.0000	0.0000
182	0.45984E-04	0.18521	0.23216E-02	-0.22292E-04	0.0000	0.0000
184	0.32178E-04	0.18544	0.16776E-02	-0.35547E-05	0.0000	0.0000
186	0.24572E-04	0.18551	0.13437E-02	0.11754E-05	0.0000	0.0000
188	0.25668E-04	0.18552	0.13462E-02	0.33412E-06	0.0000	0.0000
190	0.32273E-04	0.18545	0.16803E-02	0.36055E-05	0.0000	0.0000
192	0.46064E-04	0.18522	0.23258E-02	0.22180E-04	0.0000	0.0000
723	0.96010E-04	0.18205	0.74230E-03	0.16929E-03	0.0000	0.0000
724	0.90010E-04	0.18206	0.30056E-03	0.54607E-04	0.0000	0.0000
725	0.87627E-04	0.18206	0.15398E-03	-0.14236E-05	0.0000	0.0000
726	0.89870E-04	0.18206	0.30047E-03	-0.55975E-04	0.0000	0.0000
727	0.95814E-04	0.18205	0.74157E-03	-0.16999E-03	0.0000	0.0000

MINIMUM VALUES

NODE	42	48	48	727	42	42
VALUE	-0.76099E-05	0.17866	-0.79294E-03	-0.16999E-03	0.0000	0.0000

If you take the average of the stress in the y-direction over the thickness of the tube, (0.18552 + 0.17866)/2, the stress in the tube is 0.182 MPa, matching the analytical solution. The average is used because in the analytical case, it is assumed the stress is evenly distributed across the thickness. This is only true when the location is far from any stress concentrators, such as corners. Thus, to approximate the analytical solution, we must average the stress over the thickness.

Ex. No. : 4
Date:

MODAL ANALYSIS OF A CANTILEVER 2D-PLATE

Step 1: Set preferences.

1. **Main Menu>Preferences**
2. Turn on structural filtering. The options may differ from what is shown here since they depend on theANSYS product you are using.
3. OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. Add an element type.
3. Structural solid family of elements.
4. Choose the 8-node quad (PLANE183).
5. OK to apply the element type and close the dialog box.
6. Options for PLANE183 are to be defined.
7. Choose plane stress with thickness option for element behavior.
8. OK to specify options and close the options dialog box.
9. Close the element type dialog box.

Step 3: Define material properties.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. Double-click on Structural, Linear, Elastic, Isotropic.
3. Enter 30e6 for EX
4. Enter 0.27 for PRXY.
5. OK to define material property set and close the dialog box.
6. Density = 7.8e6
7. **Material> Exit**

Step 4: Build Geometry.

1. **Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle> By Dimensions**
2. Enter the following:
   X1 = 0 (Note: Press the Tab key between entries)
   X2 = 10
   Y1 = 0
   Y2 = 2.5
3. Apply to create the first rectangle.
4. OK to create the second rectangle and close the dialog box.

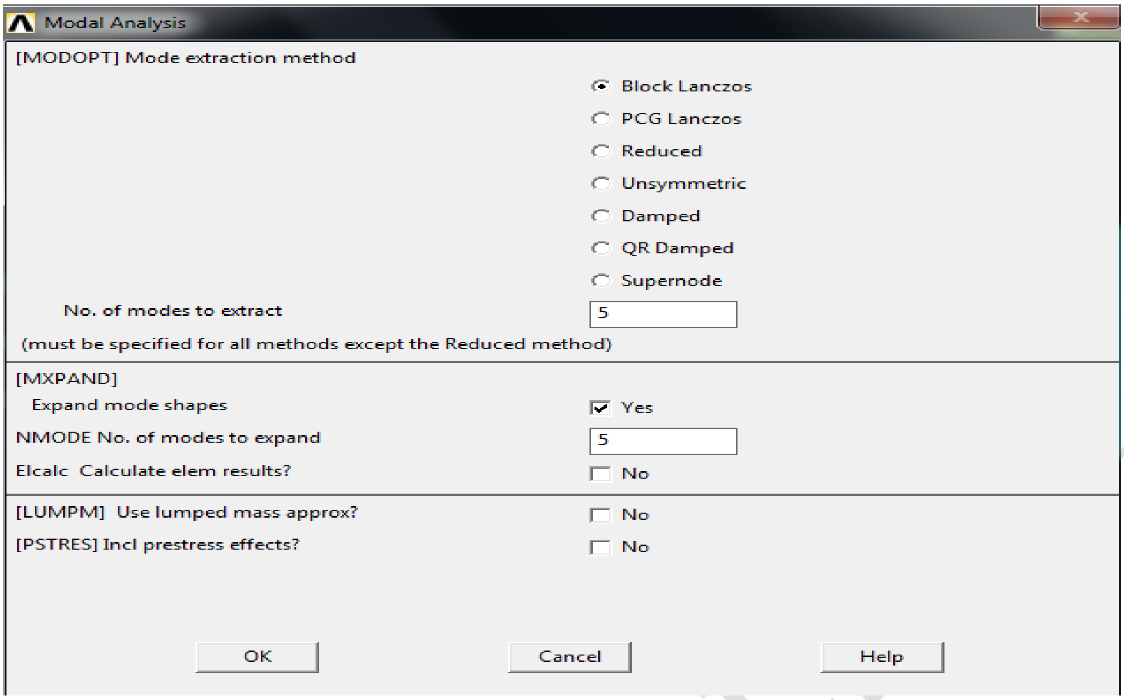
Step 5: Generate Mesh.

Main Menu> Preprocessor> Meshing> Mesh Tool

- Set area element edge length to 1 and select mapped mesh option and mesh the area.

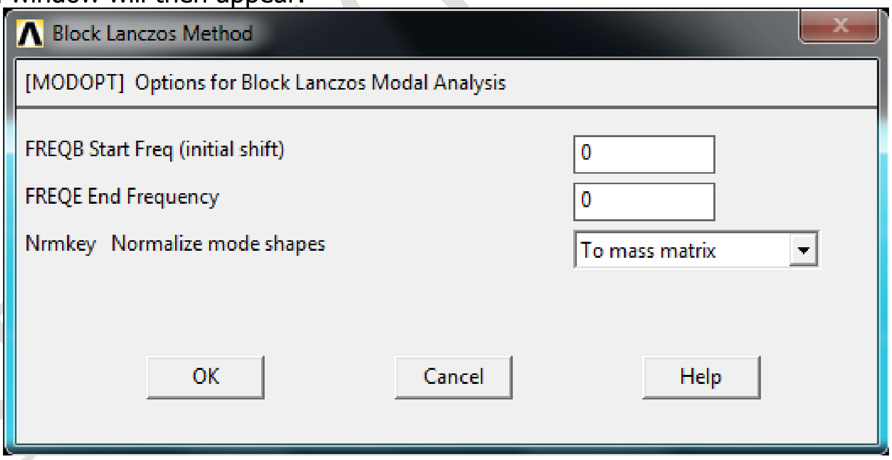
Step 6: Solution: Assigning Loads and Solving.

1. **Define Analysis Type**
   - Solution > Analysis Type > New Analysis > Modal ANTYPE,2
2. **Set options for analysis type:**
   - Select: **Solution > Analysis Type > Analysis Options...**
   - The following window will appear.



- As shown, select the **Block Lanczos**method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'.

The following window will then appear.



- For a better understanding of these options see the Commands manual.
- For this problem, we will use the default options so click on OK.

3. Apply Constraints

- Solution > Define Loads > Apply > Structural > Displacement > On Keypoints
- Fix Keypoint 1 (i.e. all DOFs constrained).

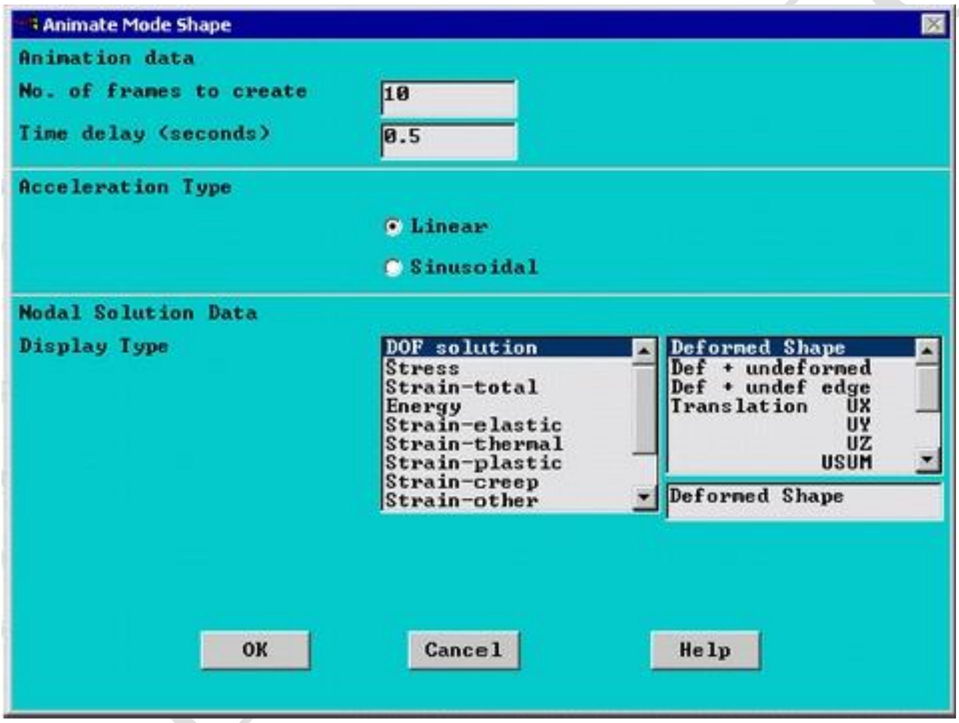
4. Solve the System

- Solution > Solve > Current LS

Step 7: Postprocessing: Viewing the Results.

1. **Verify extracted modes against theoretical predictions**
    - Select: **General Postproc> Results Summary**
2. **View Mode Shapes**
    - Select: **General Postproc> Read Results > First Set**
    - This selects the results for the first mode shape
    - Select **General Postproc> Plot Results > Deformed shape.** Select 'Def + undefedge'
    - The first mode shape will now appear in the graphics window.
    - To view the next mode shape, select **General Postproc>ReadResults> Next Set.** As above choose **General Postproc>PlotResults> Deformed shape.** Select 'Def + undef edge'.

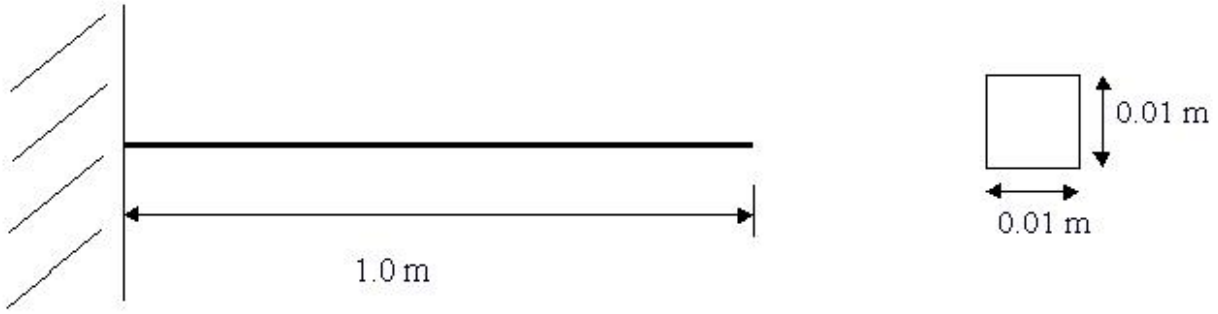
3. **Animate Mode Shapes.**
    - Select **Utility Menu (Menu at the top) > Plot Ctrls> Animate > Mode Shape**
    - The following window will appear



- Keep the **default setting** and click 'OK'. Then animated mode shapes

Ex. No. : 5 MODAL ANALYSIS OF A CANTILEVER BEAM

Date:



Modulus of Elasticity (E) = $206800(10^6)$ N/m^2

Density = 7830 kg/m^3

Step 1: Preprocessing: Defining the Problem.

1. **Main Menu>Preferences**
2. Turn on structural filtering. The options may differ from what is shown here since they depend on the ANSYS product you are using.
3. OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. Add an element type.
3. Structural beam family of elements.
4. Choose the BEAM189.
5. OK to apply the element type and close the dialog box.

   Main Menu> Preprocessor> Preprocessor > Sections > Beam > Common Sections
   1. Enter B=0.01
   2. Enter H=0.01
   3. Ok

Step 3: Define material properties.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. Double-click on Structural, Linear, Elastic, Isotropic.
3. Enter 206800e6 for EX.
4. Enter 0.27 for PRXY.
5. Ok
6. Enter 7830 for density
7. OK to define material property set and close the dialog box.
8. **Material> Exit**

Step 4: Modeling.

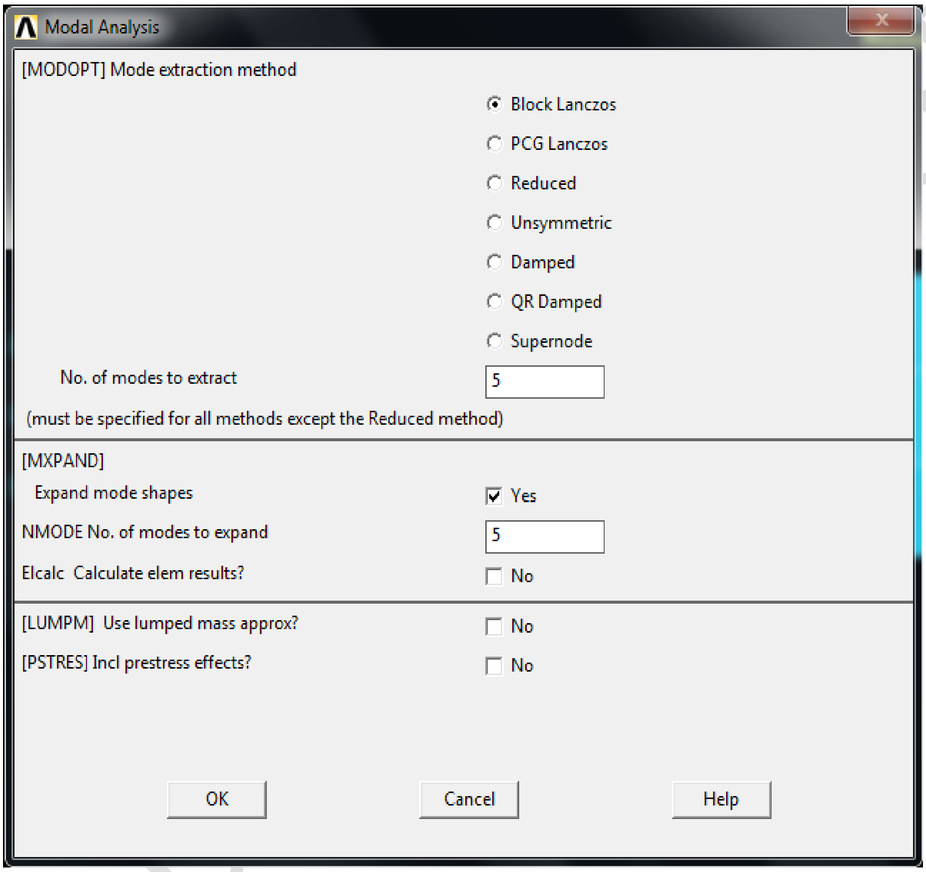
1. Main Menu> Preprocessor> Modeling > Create >keypoints> in active cs
   Createkeypoint 1 at 0,0,0 and create keypoint 2 at 1,0,0
2. Main Menu> Preprocessor> Modeling> Create>line>straight line
   Create line throughkeypoint 1 to 2

Step 5: Meshing.

1. Main Menu> Preprocessor>meshing> mesh tool
   - Set the no of element division to 100 in line set controls and mesh the line

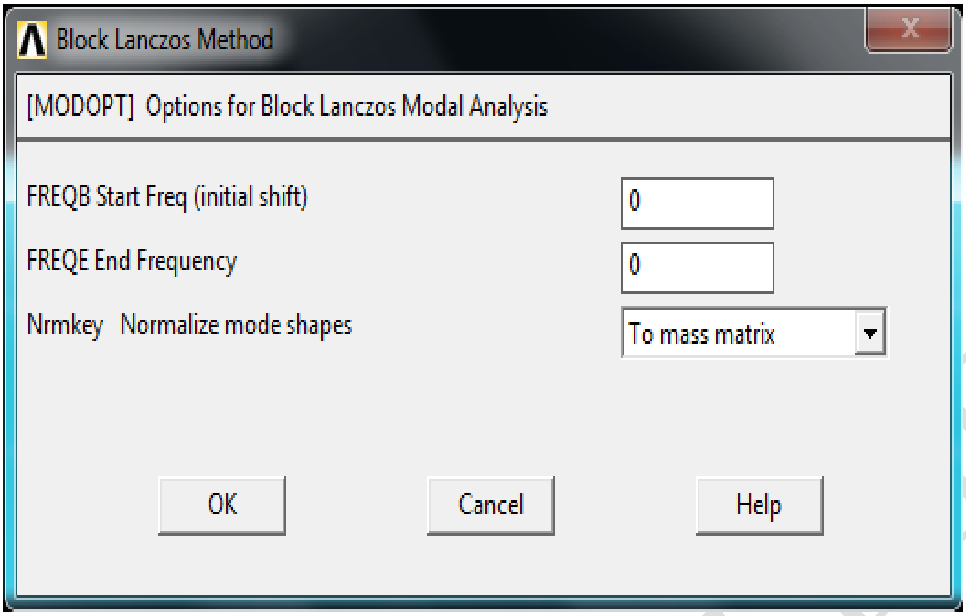
Step 6: Solution: Assigning Loads and Solving

1. **Define Analysis Type**
   - Solution > Analysis Type > New Analysis > Modal ANTYPE,2
2. **Set options for analysis type**
   - Select: **Solution > Analysis Type > Analysis Options**
   - The following window will appear



   - As shown, select the **Block Lanczos**method and enter 5 in the 'No. of modes to extract'
   - Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
   - Click 'OK'

   - The following window will then appear



- For a better understanding of these options see the Commands manual.
- For this problem, we will use the default options so click on OK.

3. Apply Constraints

- Solution > Define Loads > Apply > Structural > Displacement > On Keypoints
- Fix Keypoint 1 (i.e. all DOFs constrained).

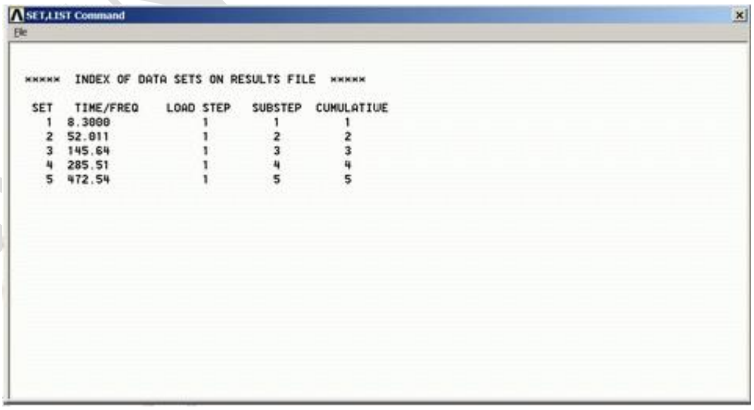
4. Solve the System

- Solution > Solve > Current LS SOLVE

Step 7: Postprocessing: Viewing the Results.

1. Verify extracted modes against theoretical predictions

- Select: **General Postproc> Results Summary...**
- The following window will appear



SET,LIST Command

File

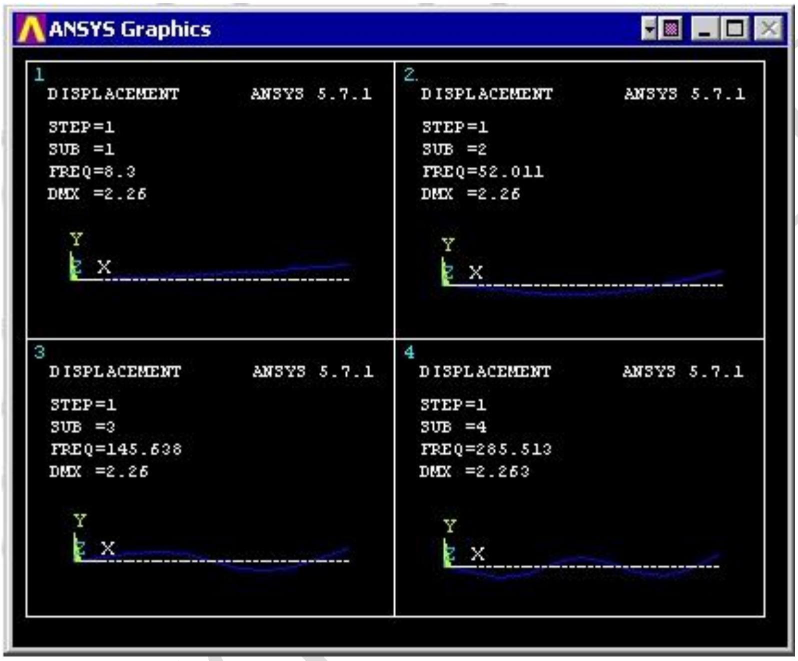
***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	8.3080	1	1	1
2	52.011	1	2	2
3	145.64	1	3	3
4	285.51	1	4	4
5	472.54	1	5	5

- The following table compares the mode frequencies in Hz predicted by theory and ANSYS.

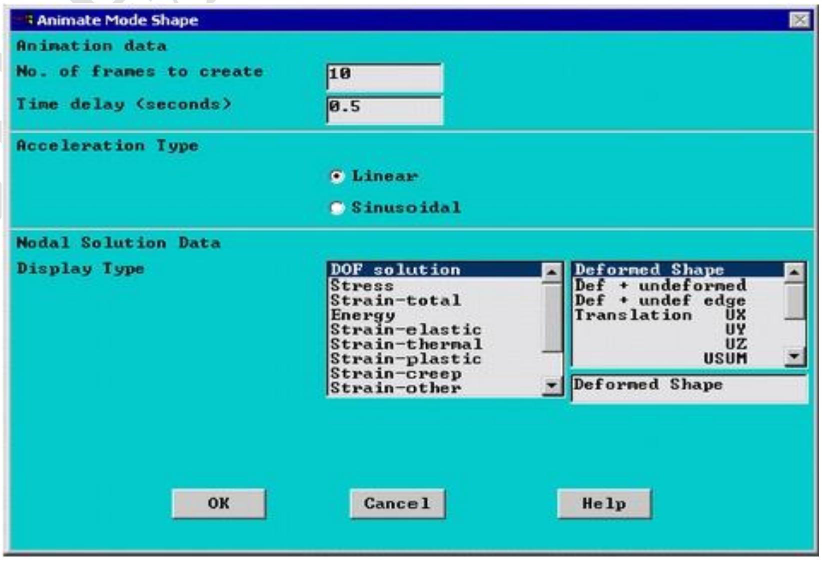
Note: To obtain accurate higher mode frequencies, this mesh would have to be refined even more (i.e. instead of 10 elements, we would have to model the cantilever using 15 or more elements depending upon the highest mode frequency of interest).

2. **View Mode Shapes**
    - Select: **General Postproc> Read Results > First Set**
        - This selects the results for the first mode shape
    - Select **General Postproc> Plot Results > Deformed shape.** Select 'Def + undef edge'
    - The first mode shape will now appear in the graphics window.
    - To view the next mode shape, select **General Postproc> Read Results > Next Set** . As above choose **General Postproc> Plot Results > Deformed shape.** Select 'Def + undef edge'.
    - The first four mode shapes should look like the following:



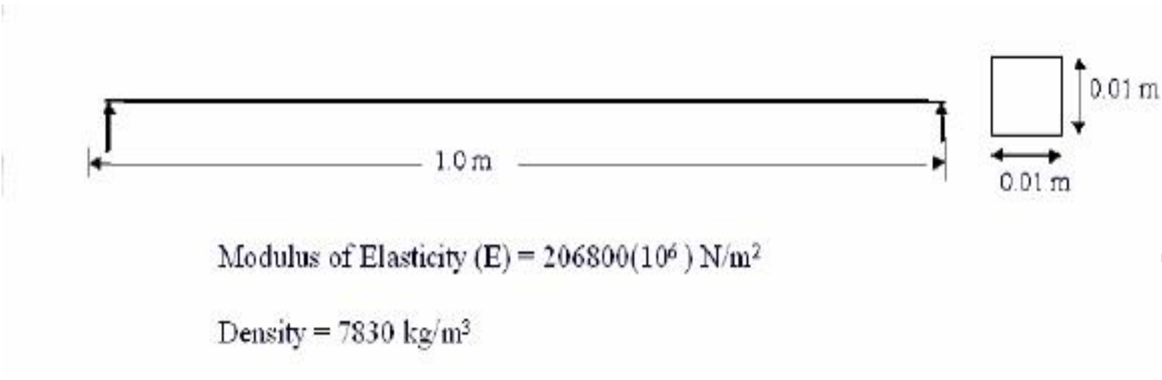
3. **Animate Mode Shapes**
    - Select **Utility Menu (Menu at the top) > Plot Ctrls> Animate > Mode Shape**

The following window will appear



- Keep the default setting and click 'OK'
- The animated mode shapes are shown below.

Ex. No. : 6 **MODAL ANALYSIS OF A SIMPLY SUPPORTED BEAM**
Date:



Step 1: Preprocessing: Defining the Problem.

1. **Main Menu>Preferences**
2. Turn on structural filtering. The options may differ from what is shown here since they depend on the ANSYS product you are using.
3. OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. Add an element type.
3. Structural beam family of elements.
4. Choose BEAM189.
5. OK to apply the element type and close the dialog box.

Main Menu> Preprocessor>Preprocessor> Sections> Beam> Common Sections

1. Enter B=0.01
2. Enter H=0.01
3. Ok

Step 3: Define material properties.

1. **Main Menu> Preprocessor>Material Props> Material Models**
2. Double-click on Structural, Linear, Elastic, Isotropic.
3. Enter 206800e6 for EX.
4. Enter 0.27 for PRXY.
5. Enter 7830 for density.
6. OK to define material property set and close the dialog box.
7. **Material> Exit**

Step 4: Modeling.

1. Main Menu> Preprocessor> Modeling> Create>key points>in active cs
   Create key point 1 at 0,0,0 and create key point 2 at 1,0,0
2. Main Menu> Preprocessor> Modeling> Create>line>straight line
   Create line through key point 1 to 2

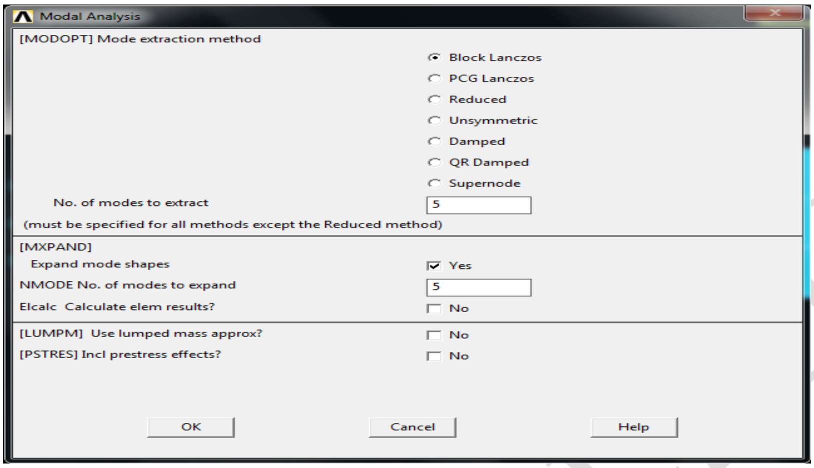
Step 5: Meshing.

1. Main Menu> Preprocessor>meshing>mesh tool
   - Set the no of element division to 100 in line set controls and mesh the line

Step6: Solution: Assigning Loads and Solving.

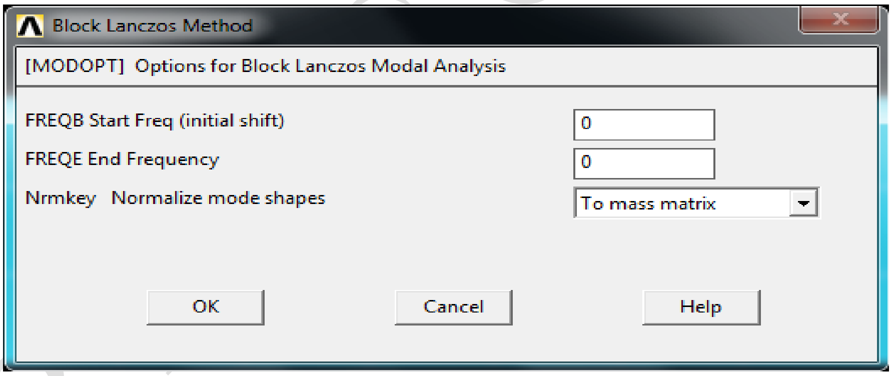
1. **Define Analysis Type**
   - **Solution > Analysis Type > New Analysis > Modal ANTYPE,2**
2. **Set options for analysis type:**
   - Select: **Solution > Analysis Type > Analysis Options**

   The following window will appear



   - As shown, select the **Block Lanczos**method and enter 5 in the 'No. of modes to extract'
   - Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
   - Click 'OK'

   The following window will then appear.



   - For a better understanding of these options see the Commands manual.
   - For this problem, we will use the default options so click on OK.

3. **Apply Constraints**
   - **Solution > Define Loads > Apply > Structural > Displacement >On Keypoints**

     fix Key point 1 and Key point 2 (i.e.**ux,uy constrained).**

4. **Solve the System**
   - **Solution > Solve > Current LS**
   - **SOLVE**

Step7: Postprocessing: Viewing the Results.

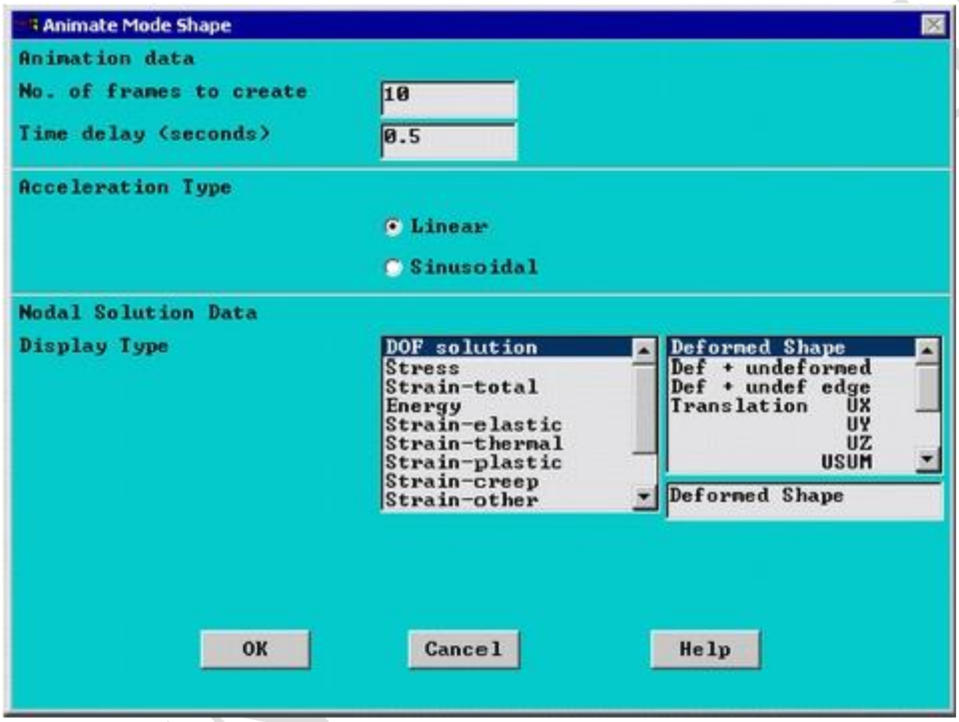
1. **Verify extracted modes against theoretical predictions**
   - Select: **General Postproc> Results Summary**
2. **View Mode Shapes**

- Select: **General Postproc> Read Results > First Set**
  This selects the results for the first mode shape
- Select **General Postproc> Plot Results > Deformed shape.**
  Select 'Def + undef edge'
- The first mode shape will now appear in the graphics window.
- To view the next mode shape, select **General Postproc> Read Results > Next Set.**
- As above choose **General Postproc> Plot Results > Deformed shape.**
  Select 'Def + undef edge'.

3. Animate Mode Shapes

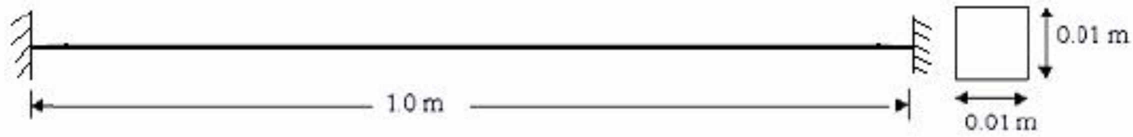
- Select **Utility Menu (Menu at the top) > Plot Ctrls> Animate > Mode Shape**

The following window will appear



- Keep the default setting and click 'OK'
- The animated mode shapes

Date:



Step 1: Preprocessing: Defining the Problem.

1. **Main Menu>Preferences**
2. Turn on structural filtering. The options may differ from what is shown here since they depend on the ANSYS product you are using.
3. OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. Add an element type.
3. Structural beam family of elements.
4. Choose BEAM189.
5. OK.

Main Menu > Preprocessor > Preprocessor > Sections > Beam > Common Sections

1. Enter B=0.01
2. Enter H=0.01
3. Ok

Step 3: Define material properties.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. Double-click on Structural, Linear, Elastic, Isotropic.
3. Enter 206800e6 for EX.
4. Enter 0.27 for PRXY.
5. Enter 7830 for density.
6. OK to define material property set and close the dialog box.
7. **Material> Exit**

Step 4: Modeling.

- Main Menu> Preprocessor> Modeling> Create>key points>in active cs
  Create key point 1 at 0,0,0 and create key point 2 at 1,0,0
- Main Menu> Preprocessor > Modeling > Create > line >straight line
  Create line through key point 1 to 2

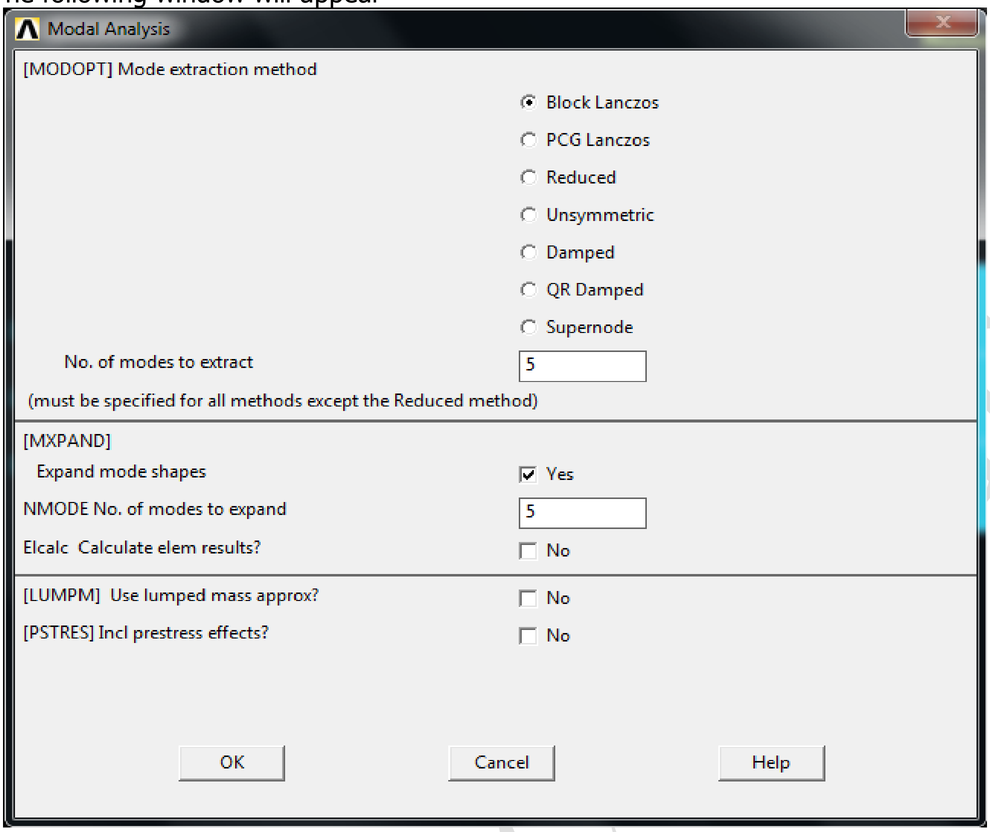
Step 5: Meshing.

- Main Menu> Preprocessor>meshing>mesh tool
  Set the no of element division to 100 in line set controls and mesh the line

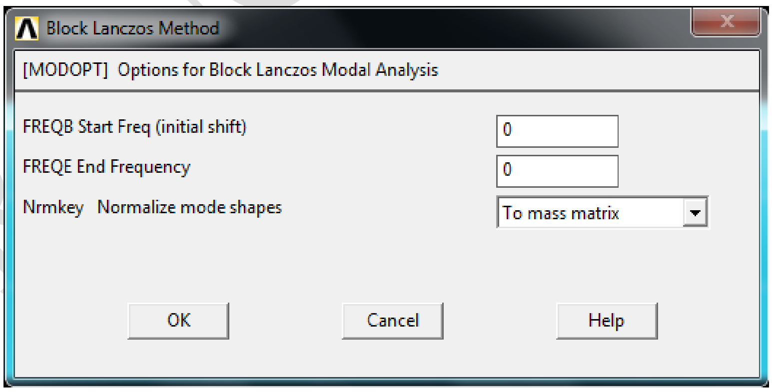
Step6: Solution: Assigning Loads and Solving.

1. **Define Analysis Type**
   - Solution > Analysis Type > New Analysis > Modal ANTYPE,2
2. **Set options for analysis type:**
   - Select: **Solution > Analysis Type > Analysis Options.**

The following window will appear



- As shown, select the **Block Lanczos**method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'
- The following window will then appear



- For a better understanding of these options see the Commands manual.
- For this problem, we will use the default options so click on OK.

3. Apply Constraints.

- Solution > Define Loads > Apply > Structural > Displacement > On Keypoints fix Key point 1 and Key point 2.

4. Solve the System

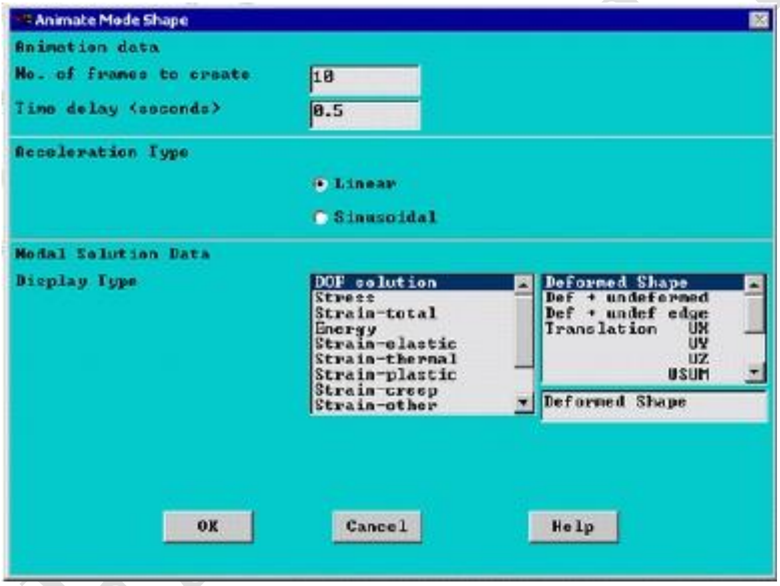
- Solution > Solve > Current LS

- SOLVE

Step7: Postprocessing: Viewing the Results.

1. **Verify extracted modes against theoretical predictions**
    - Select: **General Postproc> Results Summary**
2. **View Mode Shapes**
    - Select: **General Postproc> Read Results > First Set**
    - This selects the results for the first mode shape
    - Select **General Postproc> Plot Results > Deformed shape**. Select 'Def + undef edge'
    - The first mode shape will now appear in the graphics window.
    - To view the next mode shape, select **General Postproc> Read Results > Next Set.** As above choose **General Postproc> Plot Results > Deformed shape.** Select 'Def + undef edge'.
3. **Animate Mode Shapes**
    - Select **Utility Menu (Menu at the top) > Plot Ctrls> Animate >Mode Shape**

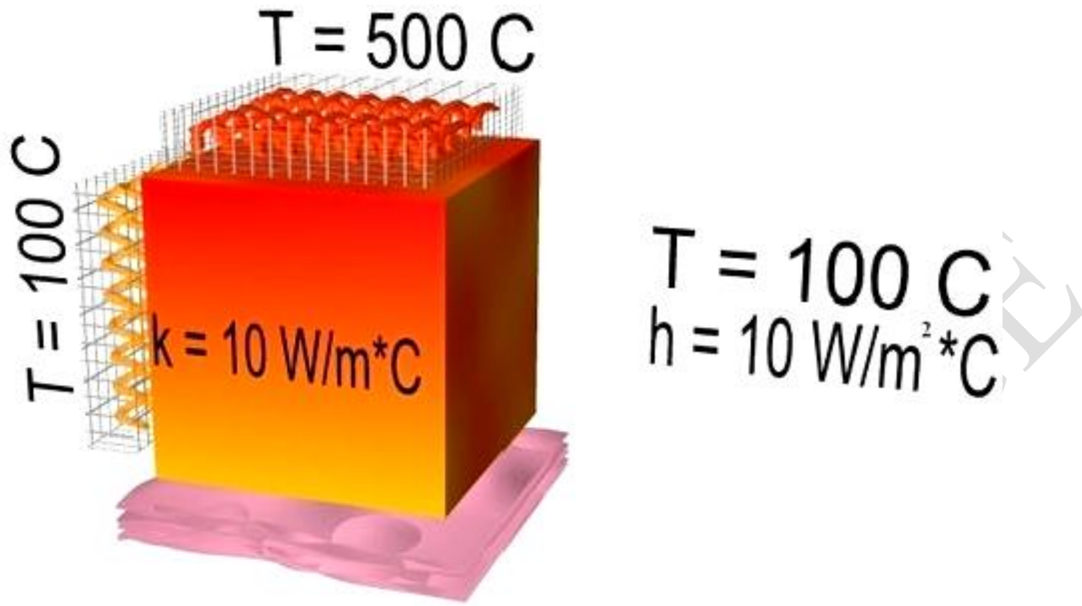
The following window will appear



- Keep the default setting and click 'OK'
- The animated mode shapes

Ex. No. : 8 THERMAL-MIXED BOUNDARY

Date: **(Conduction/Convection/Insulation)**



Preprocessing: Defining the Problem

1. **Give example a Title**

2. **Open preprocessor menu**
   ANSYS Main Menu > Preprocessor
   /PREP7

3. **Create geometry**
   - Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners
     X=0, Y=0, Width=1, Height=1 BLC4,0,0,1,1

4. **Define the Type of Element**
   - Preprocessor > Element Type > Add/Edit/Delete... > click 'Add'
     Select Thermal Mass Solid, Quad 4Node 55
     ET, 1, PLANE55

   As in the conduction example, we will use PLANE55 (Thermal Solid, Quad 4node 55). This element has 4 nodes and a single DOF (temperature) at each node. PLANE55 can only be used for 2 dimensional steady-state or transient thermal analysis.

5. **Element Material Properties**
   - Preprocessor > Material Props > Material Models > Thermal >Conductivity > Isotropic > KXX = 10
   - MP,KXX,1,10
   - This will specify a thermal conductivity of 10 W/m*C.

6. **Mesh Size**
   - Preprocessor > Meshing > Size Cntrls>Manual Size> Areas > All Areas > 0.05
     AESIZE,ALL,0.05

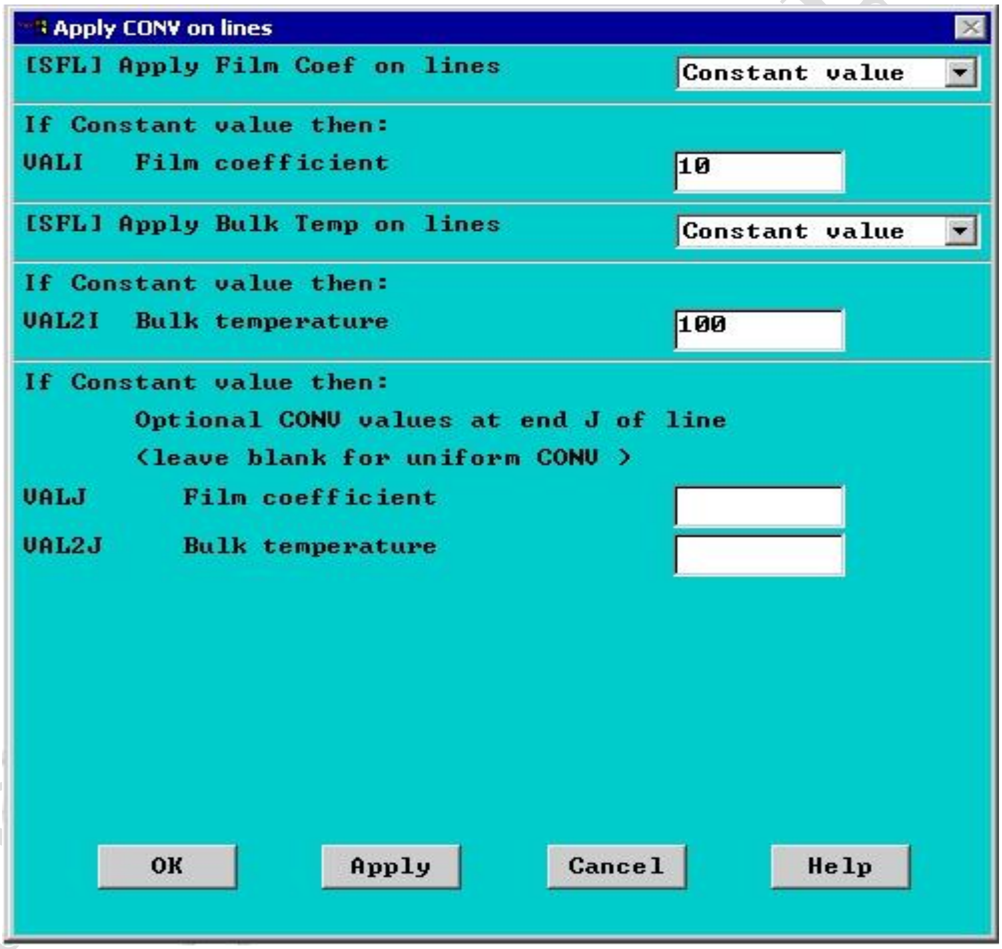
7. **Mesh**

- Preprocessor > Meshing > Mesh > Areas > Free > Pick All
  AMESH,ALL

Solution Phase: Assigning Loads and Solving

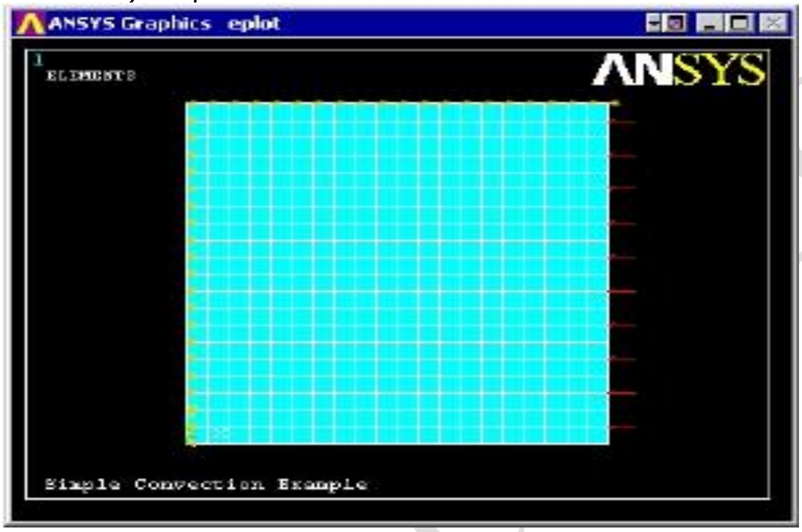
1. **Define Analysis Type**
   - Solution > Analysis Type > New Analysis > Steady-State
     ANTYPE,0
2. **Apply Conduction Constraints**
   - In this example, all 2 sides of the block have fixed temperatures, while convection occurs on the other 2 sides.
   - **Solution > Define Loads > Apply > Thermal > Temperature > On Lines**
   - Select the top line of the block and constrain it to a constant value of 500^{0}C Using the same method, constrain the left side of the block to a constant value of 100^{0}C
3. **Apply Convection Boundary Conditions**
   - **Solution > Define Loads > Apply > Thermal > Convection > On Lines**
   - Select the right side of the block.

The following window will appear:



- Fill in the window as shown. This will specify a convection of 10 W/m2*C and an ambient temperature of 100 degrees Celsius. Note that VALJ and VAL2J have been left blank. This is because we have uniform convection across the line.

4. **Apply Insulated Boundary Conditions**
   - **Solution > Define Loads > Apply > Thermal > Convection > On Lines**
   - Select the bottom of the block.
   - Enter a constant Film coefficient (VALI) of 0. This will eliminate convection through the side, thereby modeling an insulated wall. Note: you do not need to enter a Bulk (or ambient) temperature

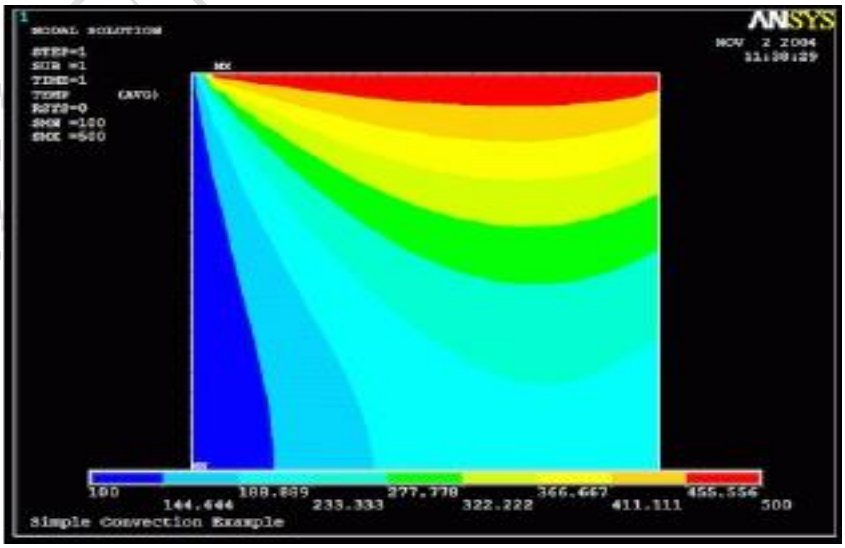


5. **Solve the System**
   - Solution > Solve > Current LS
   - SOLVE

Postprocessing: Viewing the Results

1. **Results Using ANSYS**
   - **Plot Temperature**

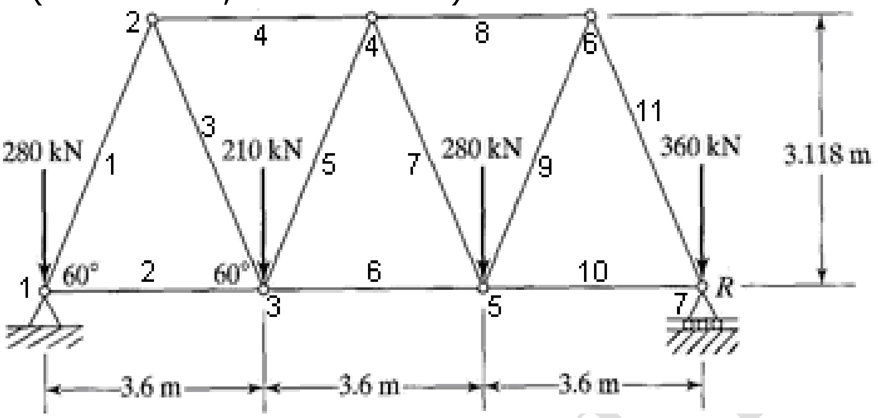
     General Postproc> Plot Results > Contour Plot > Nodal Solution> DOF solution, Temperature TEMP



Ex. No. : 9 **Stress Analysis on 2D Truss**
Date:

Problem Description

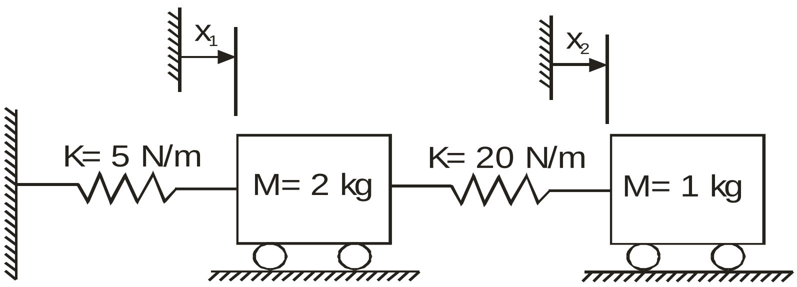
Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200 GPa, A = 3250 mm^2).



Ex. No. : 10 **Modal Analysis**
Date:

Problem Description

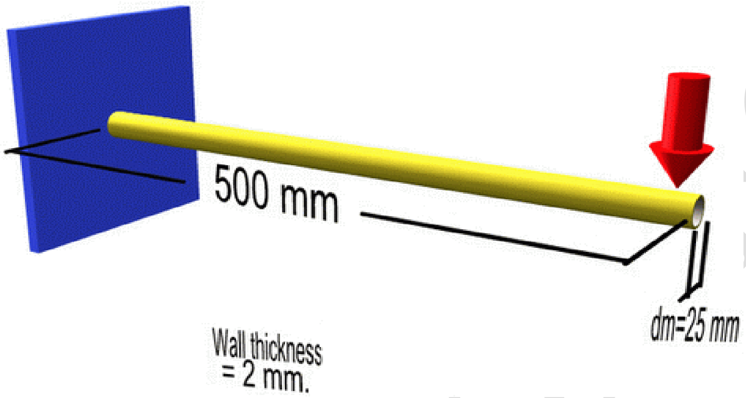
ANSYS has an option that will allow for the masses to be modeled as point masses. Using this option, no dimensions or material properties would be necessary for the masses. However, so that the mode shapes are easier to understand when they are animated, in this case, the masses will be modeled as blocks, 1 m x 1 m, with unit thickness. So, they have a volume of 1 m^3. The densities will be specified to produce the correct masses. Also, the free lengths of the springs are irrelevant in this analysis, as only the stiffness matter. But, they will both be assumed to have a length of 5 m.



Ex. No. : 11 **Stress Analysis on a Cantilever beam with Pipe Element**
Date:

Problem Description

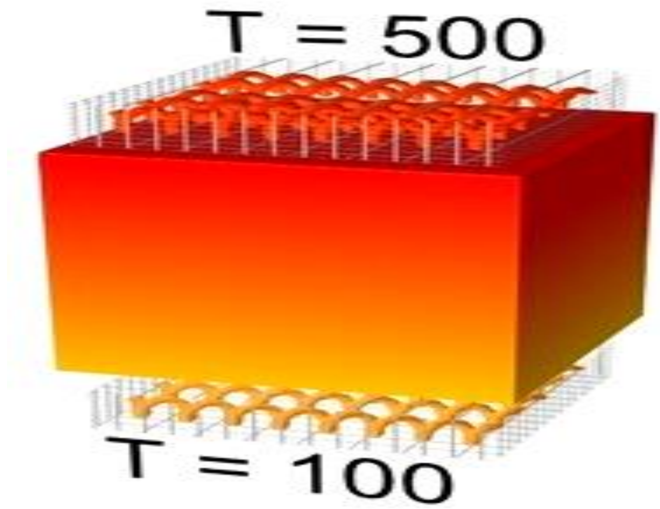
EX = 70000, PRXY=0.33,



Ex. No. : 12 **Simple Conduction**
Date:

Problem Description

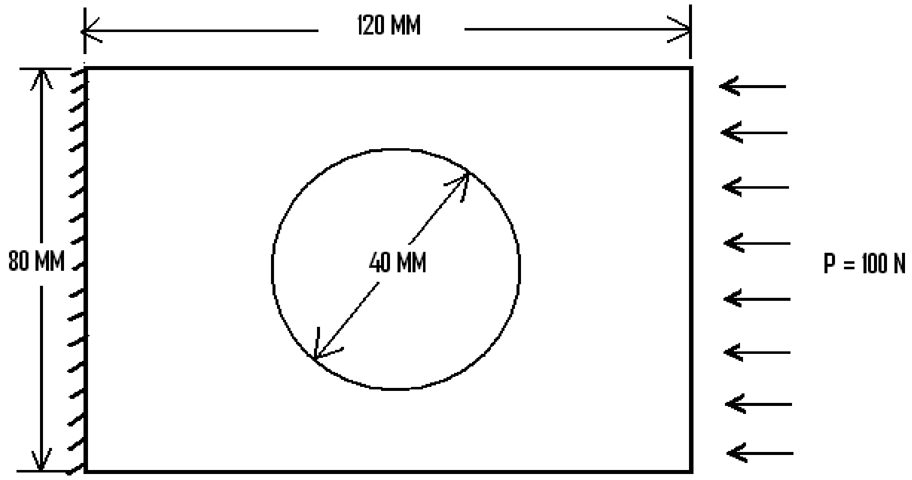
Find the temperature distribution for the given figure. Thermal conductivity (k) of the material is5W/mK and the block is assumed to be infinitely long. Also, the density of the material is 920 kg/m^3 and the specific heat capacity (c) is 2.040 kJ/kgK



Ex. No. : 13 **Stress Analysis of a plate with hole**
Date:

Problem Description

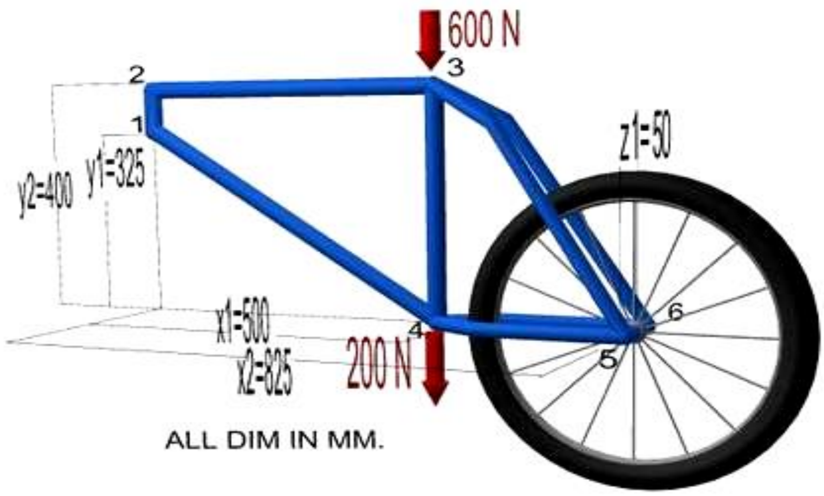
Consider a rectangle plate of uniform thickness with a circular hole of radius 20mm as shown in figure. The thickness of the plate is 1 mm. The Young's modulus E =10e7 MPa and the Poisson ratio is 0.33. Assume plane stress condition.



Ex. No. : 14 **Stress Analysis in a bicycle frame**
Date:

Problem Description

Find the stress distribution in the bicycle frame with outer diameter as 30mm and wall thickness as 3mm. Take E = 70000 N/mm^2, υ=0.33



P.S.R. ENGINEERING COLLEGE

SEVALPATTI - 626 140
SIVAKASI
Virudhunagar District.

BONAFIDE CERTIFICATE

Certified that this is a Bonafide Record of work done

by M: BASILAR

Roll No 18ME009 *in the* 16IME71 - Computer Aided Analysis

Laboratory of this College during the academic year 2021 - 2022

P.S 2/11/21
Staff - in - Charge

Head Of the Department
Department of Mechanical Engineering
P.S.R. Engineering College
Sevalpatti, Sivakasi - 626 140
Head of the Department

Register No. 1806009

Submitted for the Practical Examination held on 17/11/2021

P.S 17/11
Internal Examiner

External Examiner

CONTENTS

Expt. No.	Date	Title of the Experiment	Page No.	Marks	Signature
	15/9/2021	introduction of analysis	1		
1	15/9/2021	stress analysis of cantilever beam	9	21	P&L 22/9
2	15/9/2021	stress analysis of Fixed beam	13,	21	P&L 22/9
3,	22/9/21	stress analysis of simple supported beam with udl	17	21	P&L 29/9
4,	22/9/2021	stress analysis of Truss element	21	26	P&L 29/9
5	29/9/21	stress analysis of plate with hole	25	24	P&L 13/10
6	29/9/21	Analysis of AXI-symmetric element	29	24	P&L 13/10
7,	13/10/21	modal analysis of Fixed beam	33.	28	P&L 20/10
8	13/10/21	modal analysis of cantilever beam	35	24	P&L 20/10
9	20/10/21	thermal mixed boundary	37.	25	P&L 27/10
10,	20/10/21	harmonic analysis of cantilever beam	43,	26	P&L 27/10
11,	27/10/21	modal analysis of simple supported beam	45	24	P&L 30/10
12	27/10/21	modal analysis of 2D plate with cantilever support	49.	24	P&L 30/10

completed

24/30 P&L 30/10/21

INTRODUCTION OF ANSYS

Ansys is a general purpose final element modelling packing for numerically solving a wide velocity of material problem. Mora problem include static dynamic structural analysis for both linier and non-slinear heat transfer & fluid problems

in general a finite element solution may be broken into following three signals this is a general guideline that can be used for setting up any finite element analysis

Processing defining position:-

The major step in pre-processing are given below

i) define key point /lines / area / volume
ii) define element type and material / geometric propel
iii) mesh lines / area / volume are required
iv) the amount of details required depends on dimensionally of the analysis (1D, 2D, 3D)

Solution assing load construct:-

here we specify the load (points) pressure constrain finally solve the resulting set of question

post processing:-

in this stage one may wish to
i) list of load displacement.
ii) element of force and moment.
iii) deflection.

iv) stress combon diagram.

ANSYS 14.5 Environment:-

the ansys environment for "ANSYS 14.5" contain 2 window the main window and an output window note that is some what differs from the previous version of ansys which make use of diff window.

main widow:-

the main menu is division

a) utility menu:-

it contain function such as available throughout the ansys session such as file analysis control selection. Graphic contains.

b) inspect window:-

the output line show program oration and allows you to type is commands directly.

c) tool box:-

it contains push button that execute commonly 'ANSYS' commands can be added if defined.

d) main menu:-

it contains the primary ANSYS software organized by pre professor solution general commands are issued this is where you will note the greatest change b/w previous session ANSYS 14.5

Out put window:-

it show the output from program such as listening & data etc. it will usually positioned behind the main window and can desposite to front.

ANSYS interface:-

GRAPHICAL INTERFACE (VS) COMMAND FILE COEDING:-

There are two methods to use ANSYS the first is by means of graphical user interface connection to popular window and y-direction programs teach both recommended file many you will file command file.

help -> table to content -> command manual.

FEM convosemo Techniques:-

A fundamental promise of using to files element produre is that the today is sub divided upto small discrete region and these elements are assembled into global matrix

Consequences:-

Fiber meshes comes with a cost allowance more calculation time and large memory requirement

Beam models:-

For beam models was actually needs to define a single element for line unless we for applying a distributor load. this will only flow the Calusion down for simple means an concerns but the larger means.

Exp. No.: 7 | Page No.: 7

Date :

General models:.

In general the small will conversions more slowly than the displacement so it not sufficient to examine.

Solving and Resolving Jobs:.

Solving your Job:-

it will be good practice to save your model at various point during its creations very often you may like to solve it all the point in this you make some mistakes often on you allow to able to common back to this point

Resuming a preliminary solved Job:.

Recalling frequently you want to start up ANSYS and recall and continue a previous Job there are two methods are using the launcher.

in the ansys to the interactive and specify the defind Job.

then the get started select utility menu -> File -> Resume Jobs.

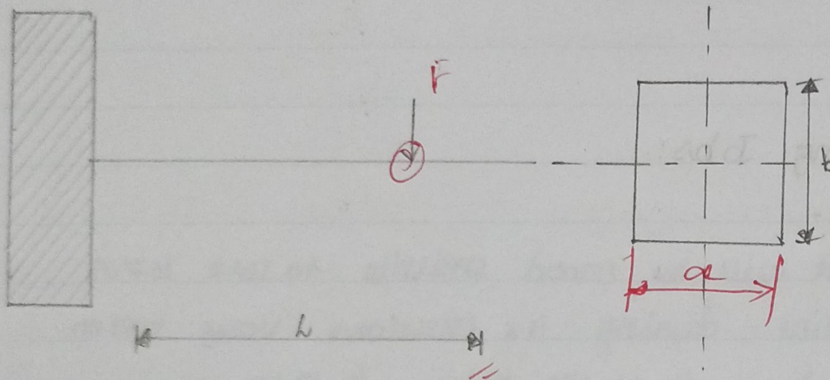
on start ANSYS and select utility File -> resume from and select your Job from list that appearance.

R.S 25/9/2021

Result:.

Thus the detailed study of "ANSYS" was studied.

stress analysis of cantilever beam:.



ALL DIMENSION ARE IN MM.

$E = 8 \times 10^5 \ N/mm^2$

$\nu = 0.3$

$L = 100mm$

$a = 10mm$

$b = 10mm.$

$F = -100N$

Exp. No.: 1
Date : 15/9/2021

STRESS ANALYSIS OF CANTILEVER BEAM

Aim:-

to analysis the stress induced on the given beam with the given load condition using ANSYS software.

Software used:-

ANSYS 14.5

PROCEDURE:-

utility menu -> File - change title -> Enter new title.
utility menu -> change Job name -> enter Job name

Step 2:- PREFERENCE:-

PREFERENCE -> STRUCTURED -> OK.

Step 3:- Preprocessor:-

Pre processor -> element type -> Add/Edit/delete/Beam
Pre processor -> section -> Beam -> common se
Enter B=10
Enter H=10
Preprocessor -> modeling -> create -> key point > in active

Pre processor -> modeling -> Create -> line -> straight line
Preprocessor -> meshing -> size Control -> manual size line ->
Pre processor -> load -> define load -> apply -> structural ->
displacement on key point.

ANALYTICAL METHODE :-

shear force $SF = -N$

$SF = -100N$

Bending moment $= WL$

$= -100 \times 100$

$= -10000$ N-mm

moment of inertia $I = \frac{bd^3}{12}$

$= \frac{10 \times 10^3}{12}$

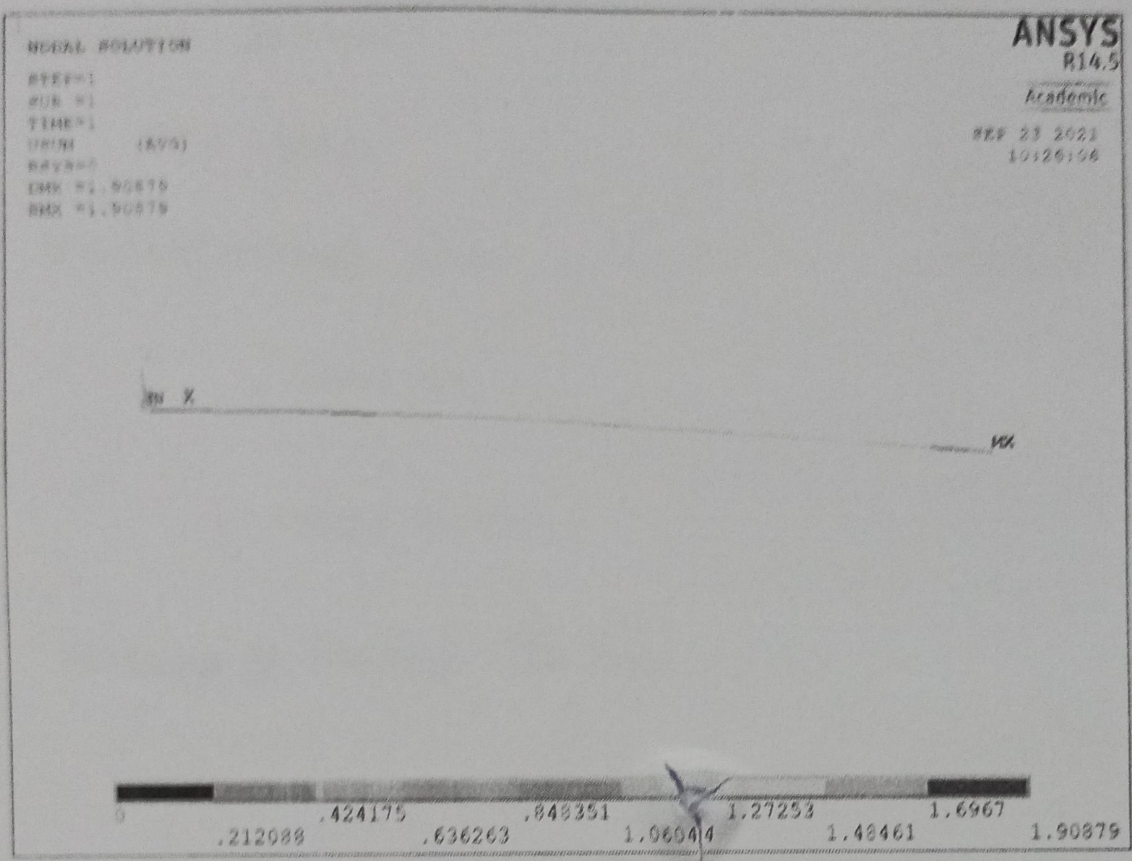
$I = 8333.33 \; mm^3$

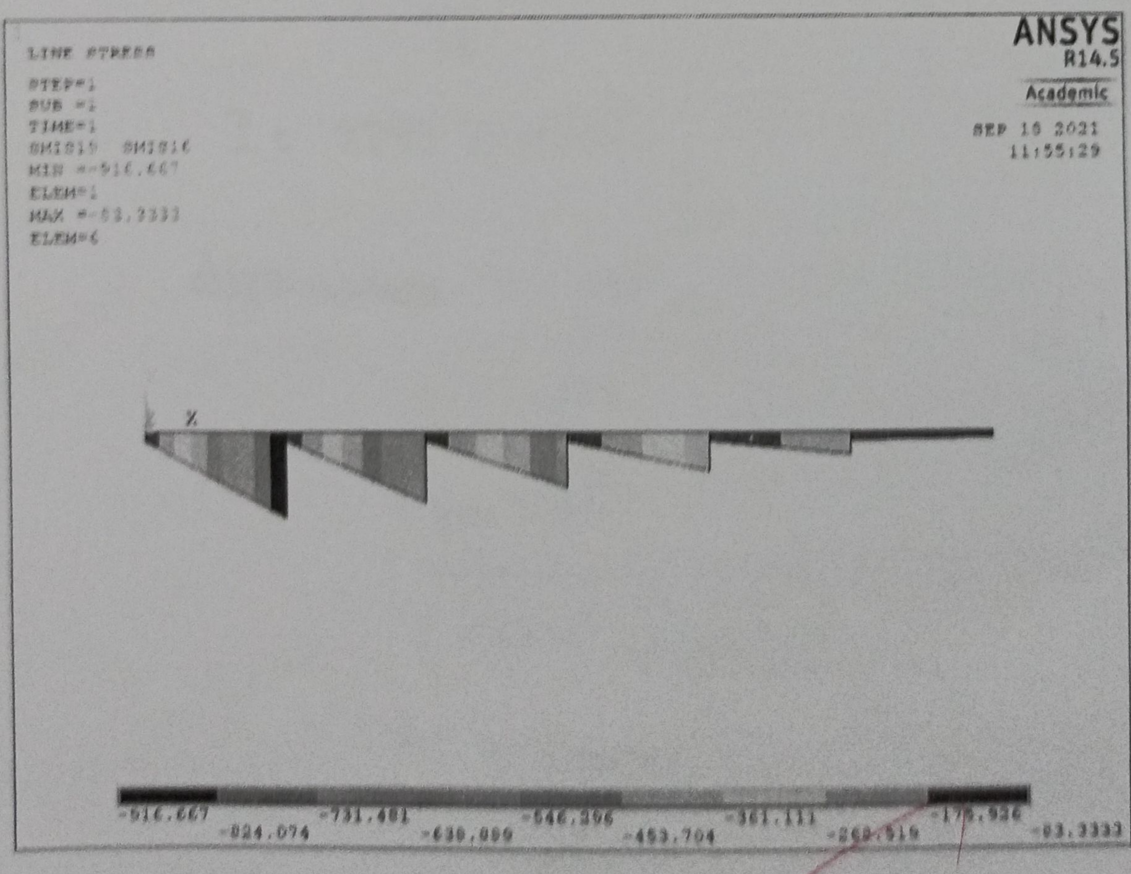
displacement $x = \frac{WL^3}{3EI}$

$= \frac{100 \times 10^3}{3 \times 2 \times 10^5 \times 833.33}$

$x = 0.20mm.$

STRESS ANALYSIS OF CANTILIVER BEAM





Exp. No.:
Date : 15/9/2021

Page No.: 11

preprocessor -> load -> define load -> Apply -> structure -> force moment -> on keypoint.

Step4:- solution:-

solutions -> solve -> current LS -> ok.

Step5:- General post processing:-

General post proc -> plot nodes -> deformed shape.

General post pro -> plots -> results -> contour plots -> line element.

General post pro -> element table -> define table . add 3,6,16,19.

Result:-

description	displacement (mm)	shear force (N)	Bending moment
Theoritical value			

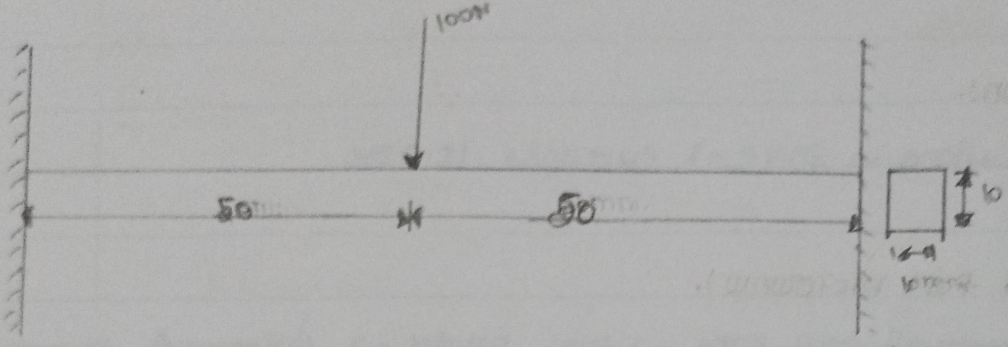
Exprimental value:-

Thus the stress analysis was done for the given cantilever beam in ANSYS V14.5 software.

PRACTICAL EVALUATION

Observation (10)	7
Calculation / Result (10)	7
Viva (10)	7
Total Marks (30)	21
Signature of Faculty	

Stress analysis of fixed beam:



ALL DIMENSIONS ARE IN MM.

$E = 2 \times 10^5 N/mm^2$

$\nu = 0.3$

$a = 10mm$

$b = 10mm$

Exp. No.: 8
Date : 15/9/2021

STRESS ANALYSIS OF FIXED BEAM.

Aim:-

to analysis the shear induced on the given beam with the given condition using analys software.

Software used:-

ANSYS 14.5

Procedure:-

Step 1: utility menu:-

utility menu -> File -> change title -> Enter new title.
utility menu -> change Job name -> enter Job name.

Step 2: Preferences:.

Preference -> Structural -> ok.

Step 3:. Preprocessor:-

Preprocessor -> element type -> add/edit/delete -> Beam 188 - close.

Preprocessor -> section beam -> Common section.

B=1

H=1

Preprocessor -> material props -> material models ->

Ex = 2.6 e11

PRXY = 0.3.

preprocessor -> modeling -> create -> key point -> inactive.

ANALYTICAL METHODE:-

deflection $y = \frac{wl^3}{192EI}$

moment of inertia $I = \frac{bd^3}{12}$.

$= \frac{10 \times 10^3}{12} = 833.33 mm^4$

$y = \frac{100 \times 100^3}{192 \times (2 \times 10^5) \times 833.33}$

$y = 0.001 mm \Rightarrow 5 \cancel{\times 10^{-7}} mm$

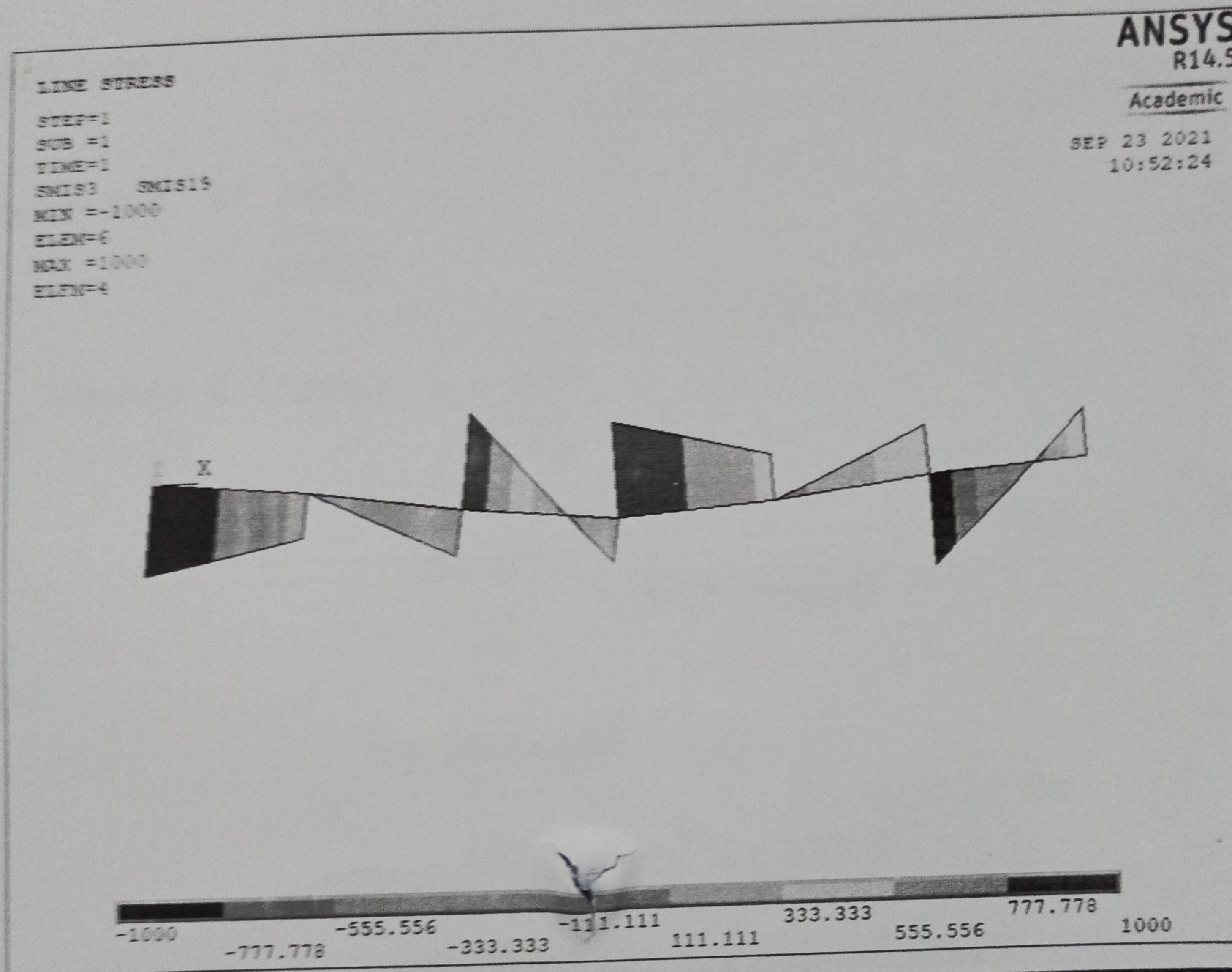
shear force $(SF) = W/2 = \frac{100}{2}$

$SF = 50N$

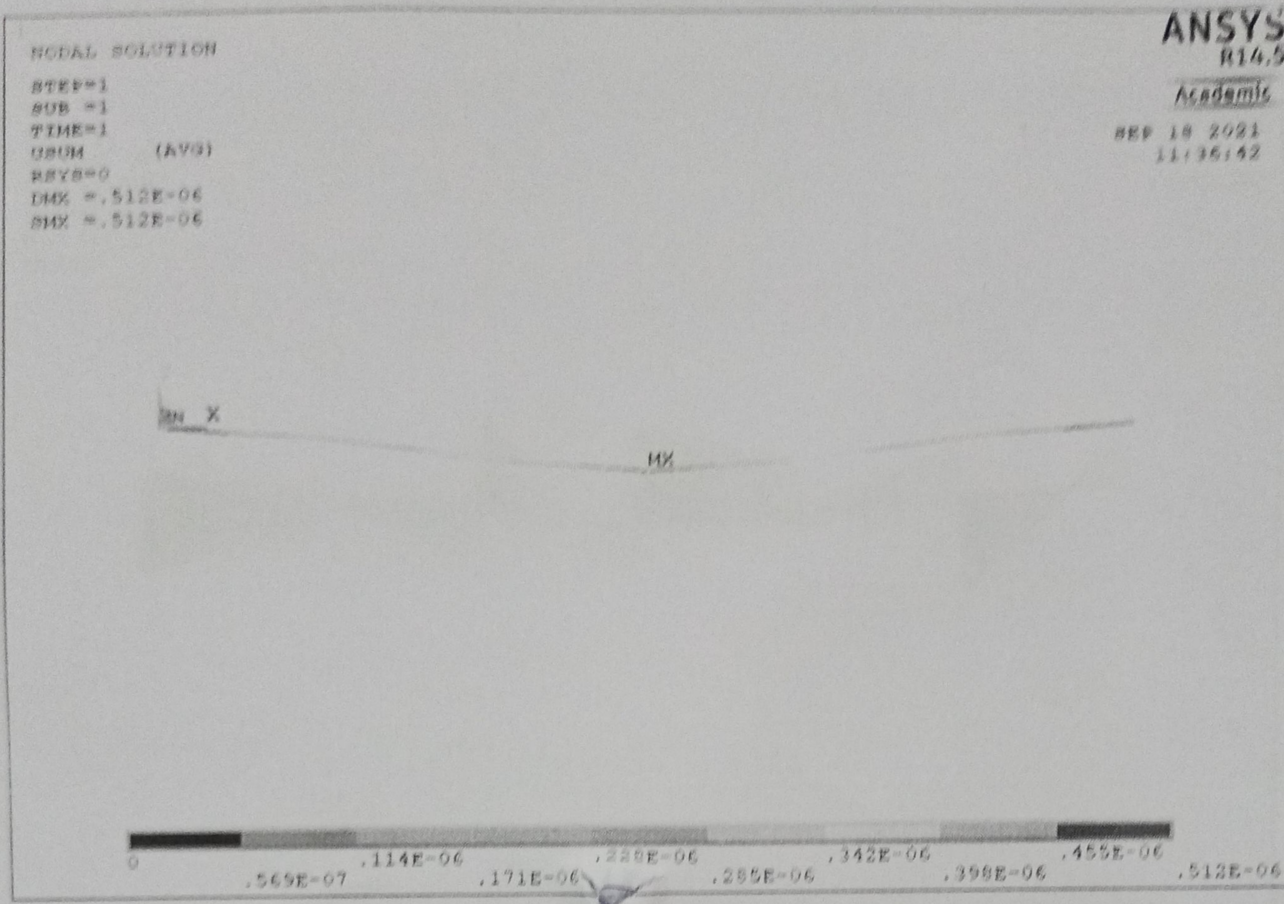
Bending moment $(mb) = \frac{WL}{8}$

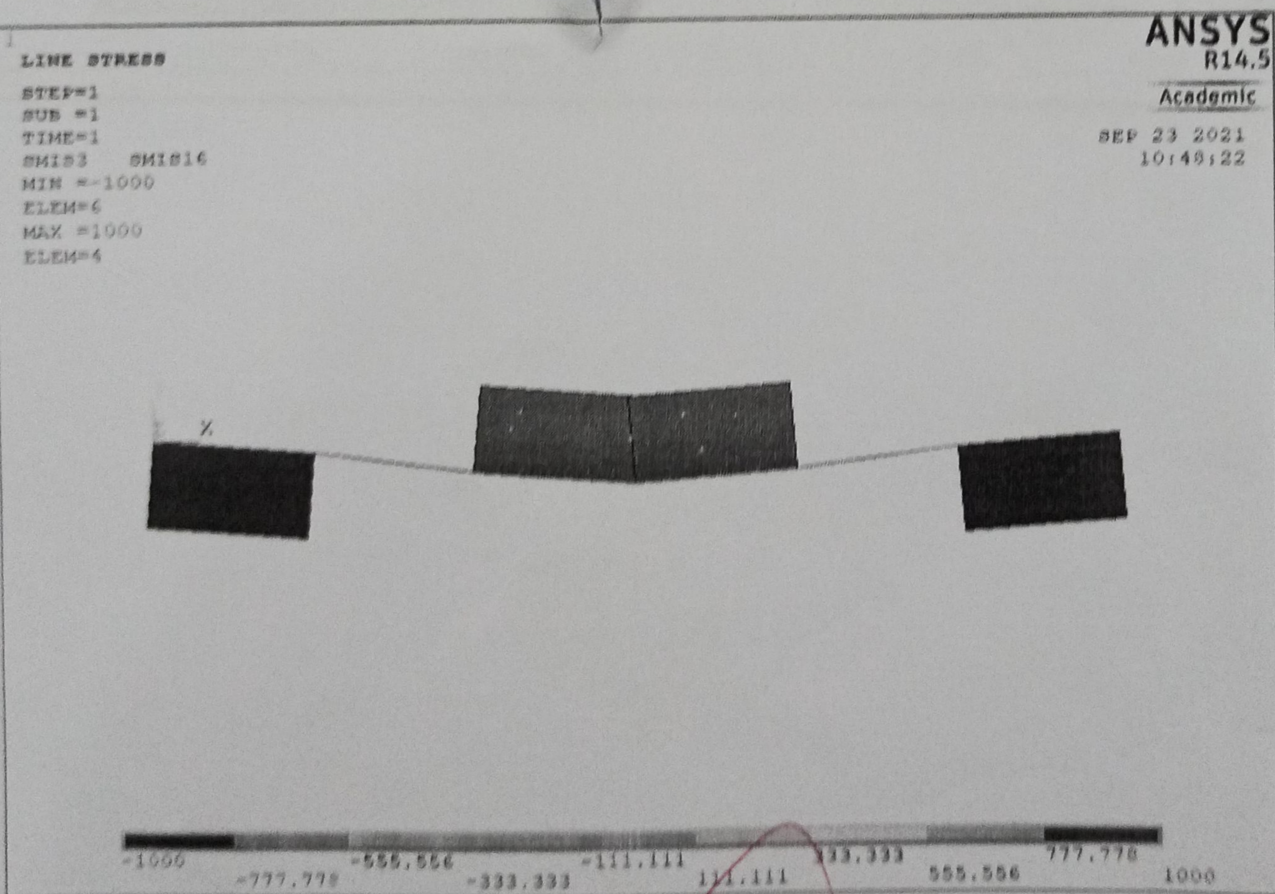
$= \frac{10.0 \times 1000}{8}$

$\boxed{mb = 1250 mm}$


LINE STRESS
STEP=1
SUB =1
TIME=1
SMIS3 SMIS15
MIN =-1000
ELEM=6
MAX =1000
ELEM=4
ANSYS
R14.5
Academic
SEP 23 2021
10:52:24
X
-1000
-777.778
-555.556
-333.333
-111.111
111.111
333.333
555.556
777.778
1000

STRESS ANALYSIS OF FIXED BEAM





Page No.

Exp. No.

Date

keypoint 1 at (0,0,0)

key point at (10,0,0)

key point at (12,0,0)

preprocessor -> modelling -> node -> key point -> line -> st line

preprocessor -> meshing -> mesh tool

pre processor -> force / moment on key point -> Apply -> fy -100 -ok.

Step4: Solution:-

Solution -> solve -> current LS -> ok.

Step5:- General post processor:-

General post proc -> plot results -> deformed shape deformed -> undeformed -> ok.

General post proc -> contour plot -> nodal solution DOF solution -> ok.

General pre post -> contour -> nodal solution is stress component

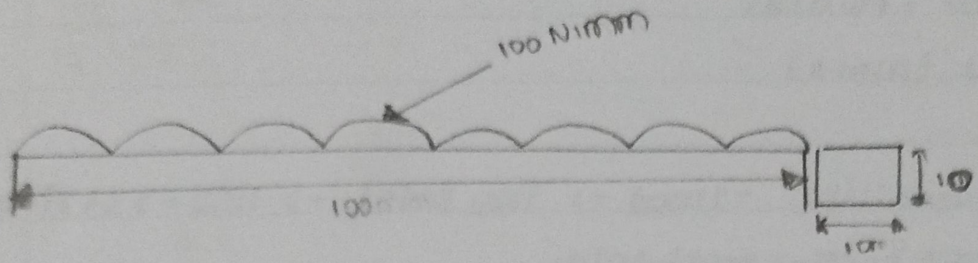
Result:-

description	displacement (mm)	shear force (N)	Bending moment N-m
Theoretical value-			
Experimental value			

Thus the stress analysis was done for the given fixed beam in ANSYS V14's Software

PRACTICAL EVALUATION	
Observation (10)	7
Calculation / Result (10)	7
Viva (10)	7
Total Marks (30)	21
Signature of Faculty	P&L

stress analysis simply supported beam with UDL.



$E = 2\times10^5$ N/mm

$\nu = 0.3$

a = 10mm

b = 10mm.

ALL DIMENSIONS ARE IN MM.

Exp. No.: 8
Date: 22/9/2021

STRESS ANALYSIS OF SSB BEAM WITH UDL LOAD

Aim:-

to analysis the stress induced on the given beam with the given condition using analysis software.

Software used:-

ANSYS 14.5

Procedure:-

Step1:- utility menu:-

utility menu -> File -> change title -> Enter new title

utility menu -> change Job Name -> Enter Job Name.

Step2: preference:-

preference -> structural -> ok.

Step3: preprocessor:-

preprocessor -> element type -> Add / Edit / delete -> Beam.

preprocessor -> selection -> Beam -> common selection.

Enter B = 1

Enter H = 1

preprocessor -> material props -> maintained model -> linear elastic 2e5 and 0.3 for PRXY

preprocessor -> modeling -> create -> keypoint -> In active CS

keypoint (0,0,0) and enter keypoint (2,0,0)

preprocessor modeling -> create line -> straight line.

ANALYTICAL METHODE :.

shear force (SF) $= \frac{wl}{2} = \frac{1000 \times 100}{2}$

$SF = 50000\, N$

Bending moment $(M_x) = \frac{wl^2}{8}$

$= \frac{1000 \times 100^2}{8}$

$M_x = 0.125 \times 10^7\, mm$

displacement $x = \frac{5wl^4}{384EI}$

moment of inertia $I = \frac{bd^3}{12}$

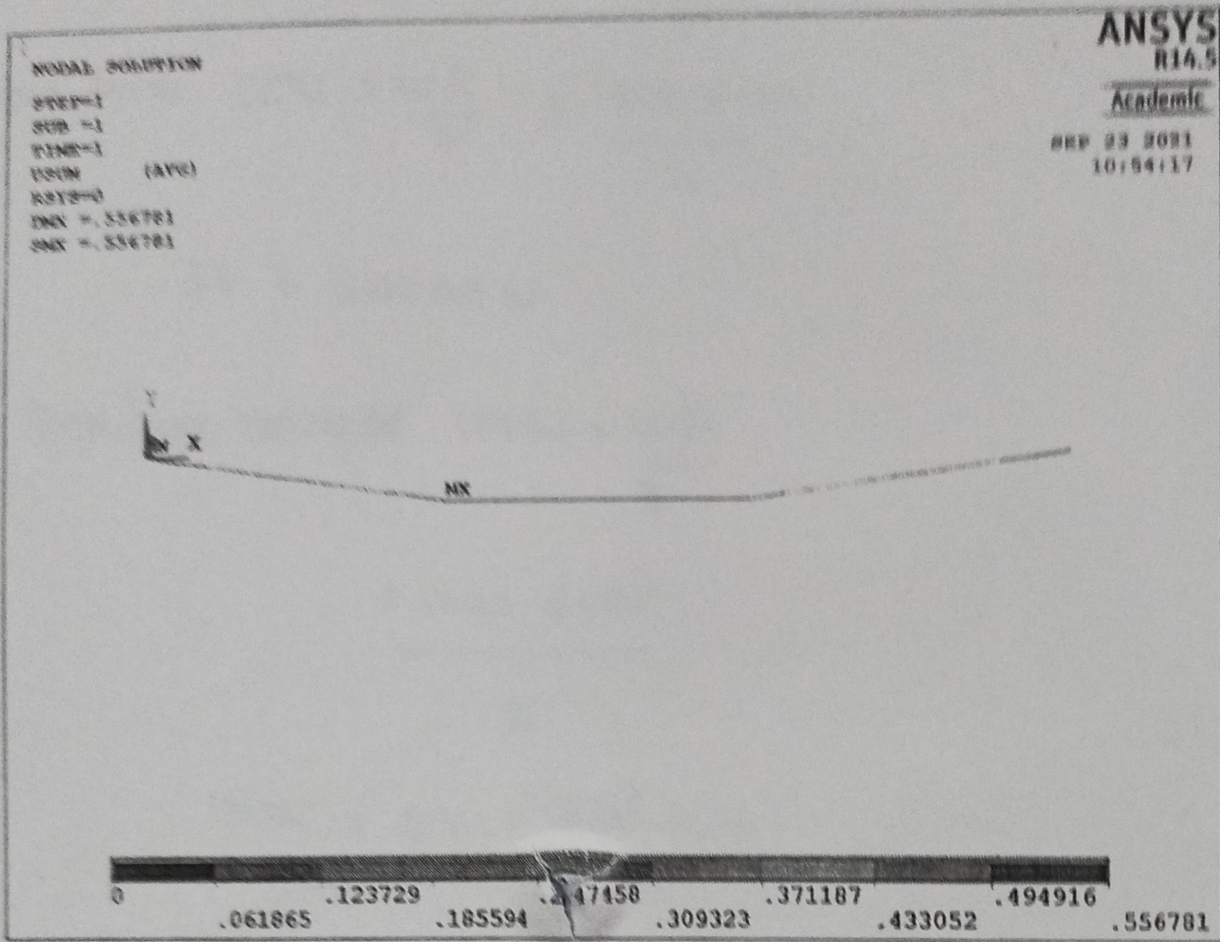
$= \frac{10 \times 10^3}{12}$

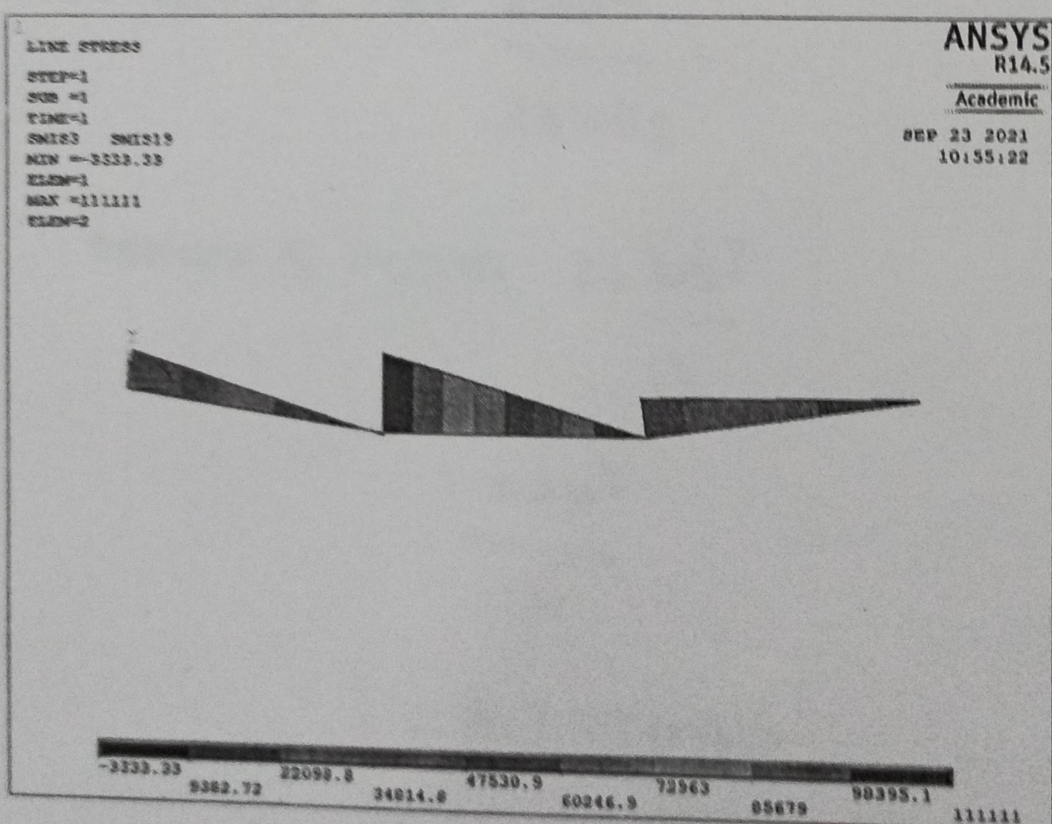
$I = 833.33\, mm^4.$

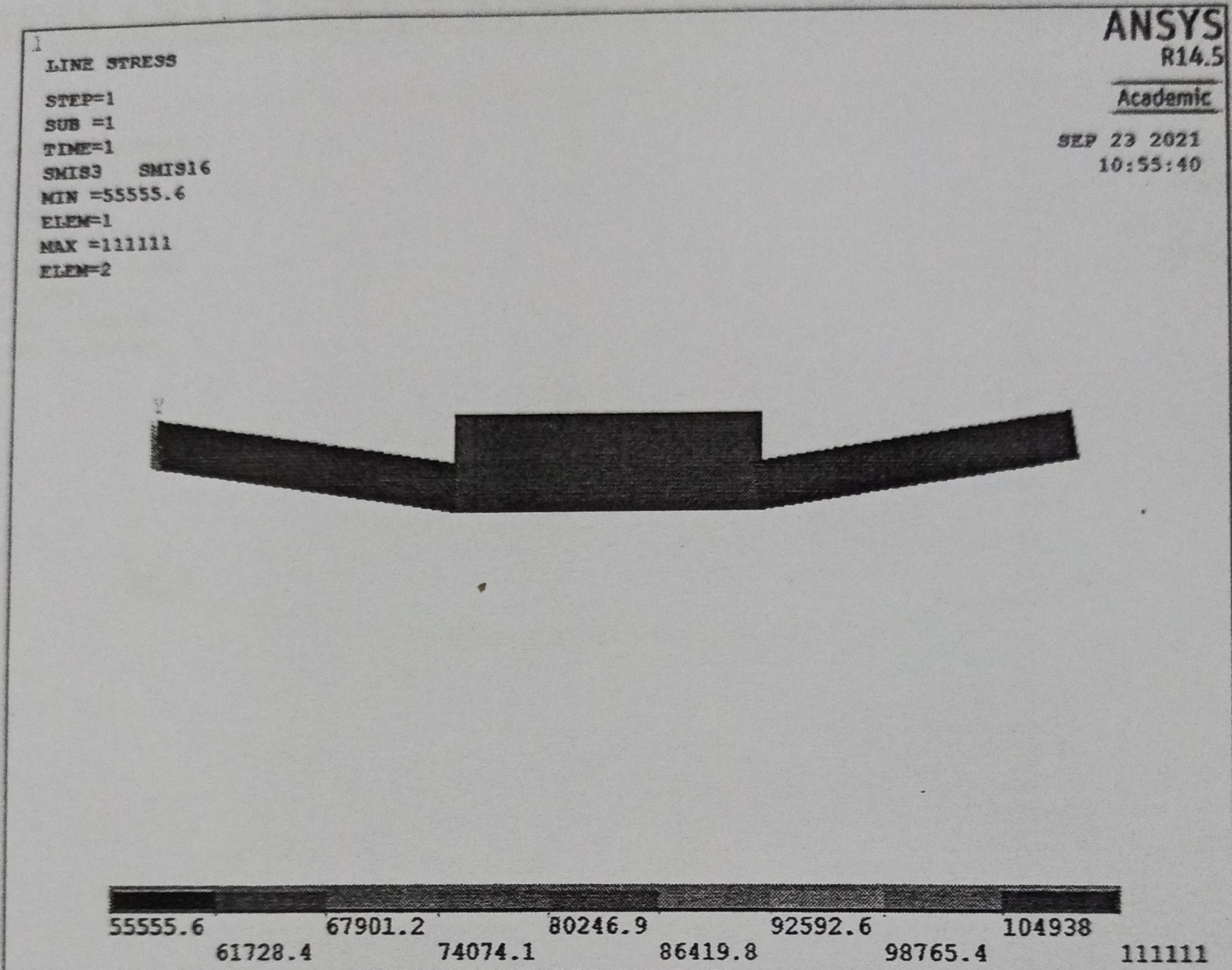
$x = \frac{3 \times 1000 \times 100^4}{384 \times 2 \times 10^5 \times 833.3}$

$\boxed{x = 0.798\, mm}$

STRESS ANALYSIS OF SIMPLY SUPPORTED BEAM WITH UDL






LINE STRESS
STEP=1
SUB =1
TIME=1
SMIS3 SMIS16
MIN =55555.6
ELEM=1
MAX =111111
ELEM=2
ANSYS
R14.5
Academic
SEP 23 2021
10:55:40
55555.6
61728.4
67901.2
74074.1
80246.9
86419.8
92592.6
98765.4
104938
111111

Exp. No.:
Date :

Step4 solution:-

solution -> solve -> current LS

Step4:- General post proc:-

General post pro -> plot results -> deformed -> undeformed

General post pro -> element table -> define table -> ok.

General post plot results -> contour plot -> line element results

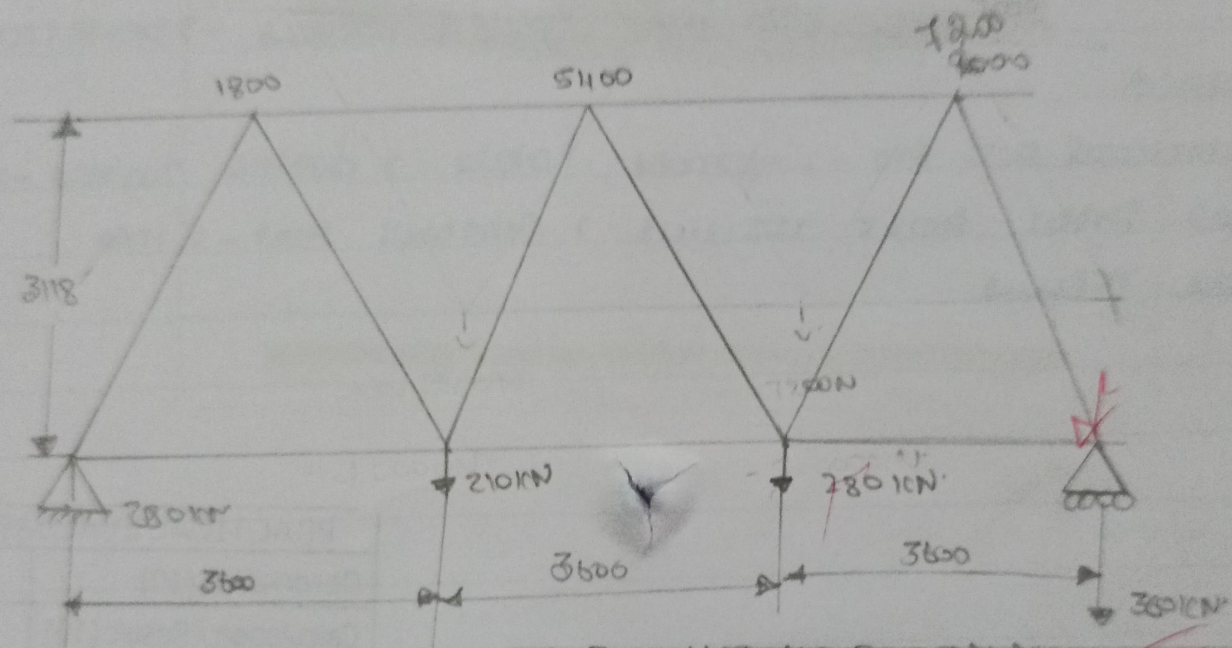
PRACTICAL EVALUATION	
Observation (10)	7
Calculation / Result (10)	7
Viva (10)	7
Total Marks (30)	21
Signature of Faculty	PS 29/9

Result:-

description	displacement (mm)	shear force (N)	Bending moment
Theoretical value.			
analytical value.			

Thus the stress analysis was done for the given SSB with UDL beam using Analysis 14.5 software.

Stress Analysis in Truss element.



ALL DIMENSIONS ARE IN MM.

STRESS ANALYSIS IN TRUSS ELEMENT

Aim:-

to analysis the stress induced in the given truss element for given loading condition using ansys

Software used:-

ANSYS 14.5

Procedure:-

Step 1:- change directory -> creat Folder -> change menu.

Step 2:- main menu -> preference -> structural -> element type Edit 3D link.

Step 3:- material property

select real constant -> Add

command Area = 3250 mm²

select material -> new material property

Isontropic -> EX 2E5 P=0.33

Step 4:- modelling

select modelling -> key point -> in active CS

1 (0,0,0) 2 (1800, 3000, 0) 3 (3600,0,0) 4 (5400, 3100, 0)

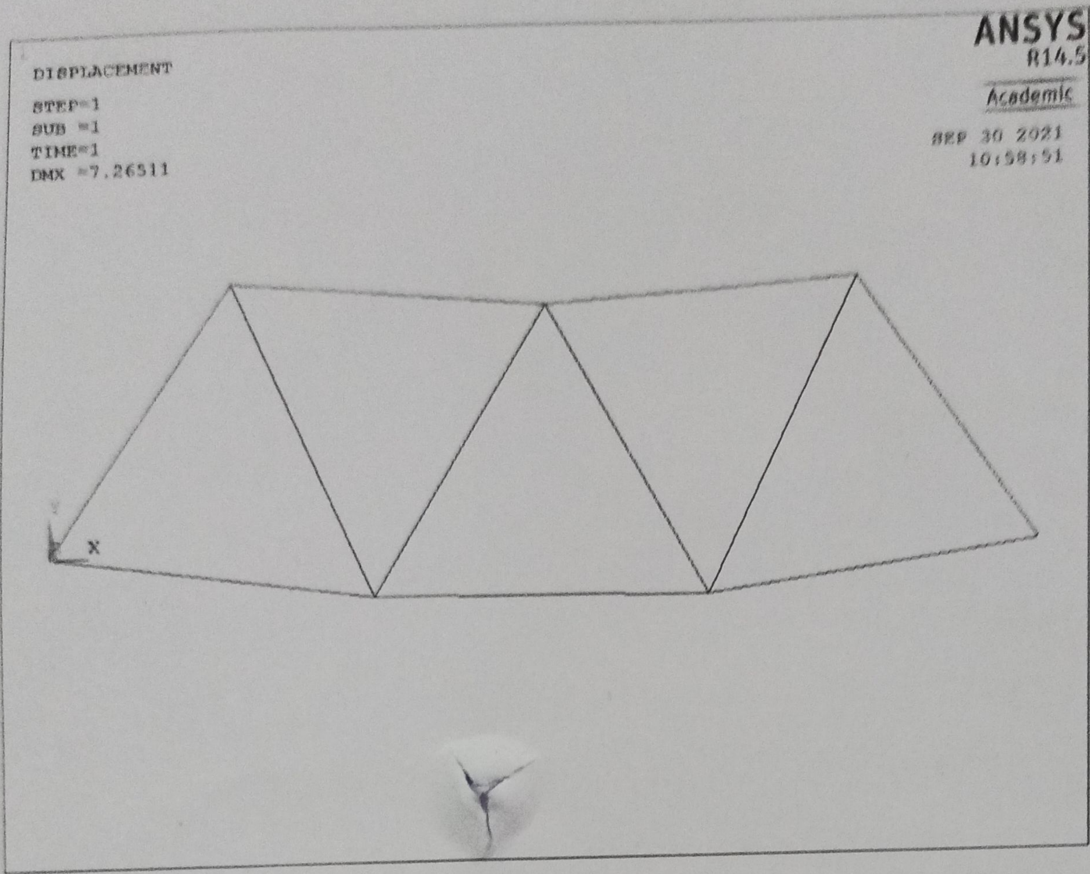
5 (7200, 0, 0) 6 (9000, 3100, 0) 7 (10800, 0, 0)

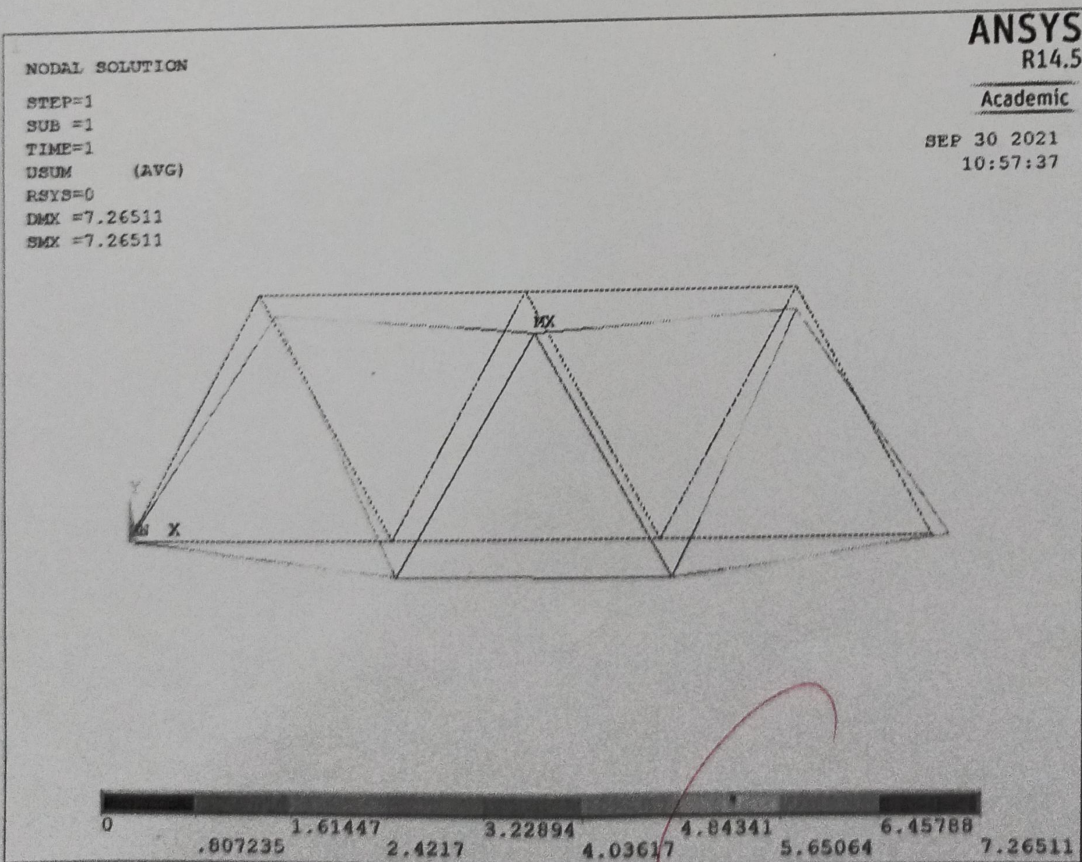
the using straight line.

Step 5:- meshing

select meshing -> mesh tool -> set -> mesh -> click -line -> ok.

STRESS ANALYSIS OF TRUSS ELEMENT





Exp. No.:
Date :

Step 6:- Load.

Load -> define load -> structural, displacement.

Force on node : 1 280000

Force on node : 2 200000

Force on node : 3 280000

Force on node : 4 300000

Step 7:-

Result

Plot result -> contour plot -> nodal solution -> solve.

current is done

Plot ctrl -> contour plot -> nodal solution.

Step 8:-

General post process.

General post processor -> plot - result -> define stress all deformation.

Scale contour plot -> nodal soln -> stress

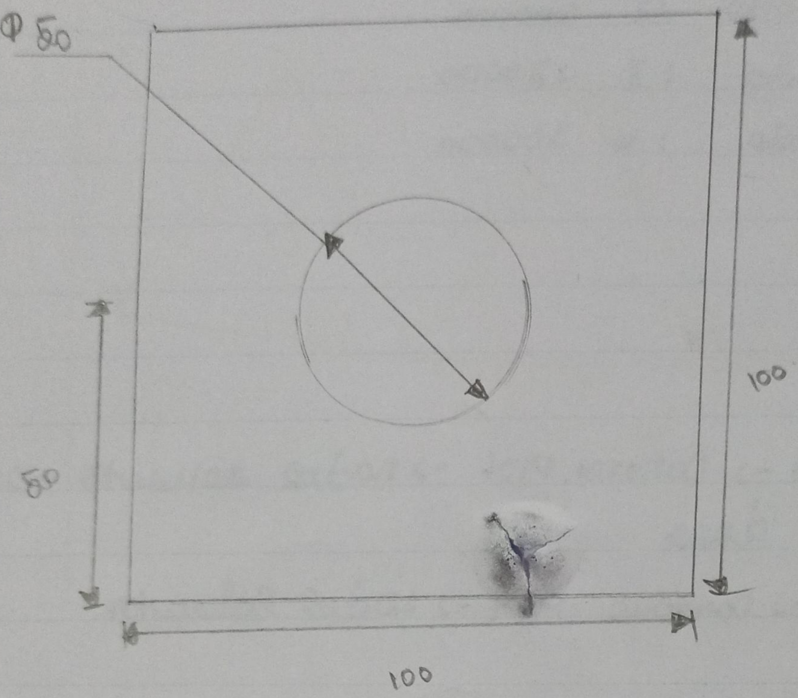
Result:-

Thus the stress analysis of given mass element was done using ANSYS 14.5 software.

PRACTICAL EVALUATION	
Observation (10)	9
Calculation / Result (10)	9
Viva (10)	8
Total Marks (30)	26
Signature of Faculty	PSC

29/9

stress analysis of plate with hole.



ALL DIMENSIONS ARE IN MM.

circle.

$x = 50$ mm

$y = 50$ mm

$r = 25$ mm

rectangle

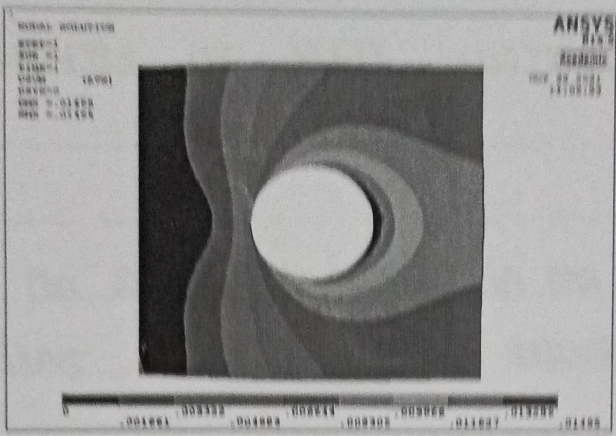
$x_1 = 0$ $x_2 = 100$

$y_1 = 0$ $y_2 = 100$

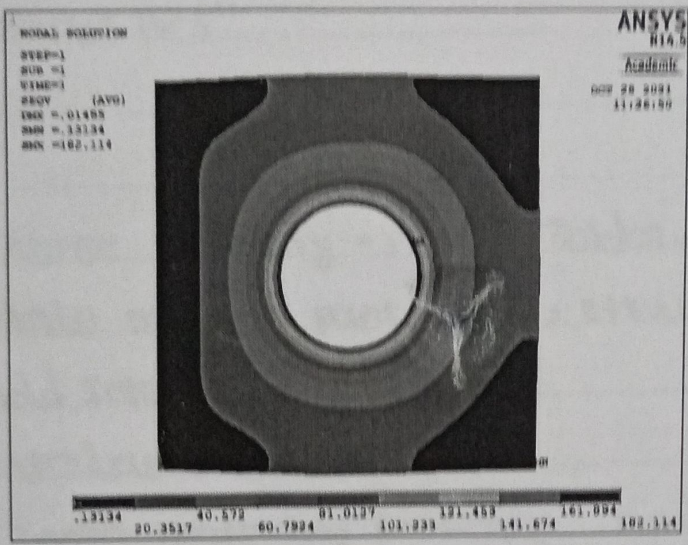
EX : 2E5 N/mm^2

PRXY = 0.3

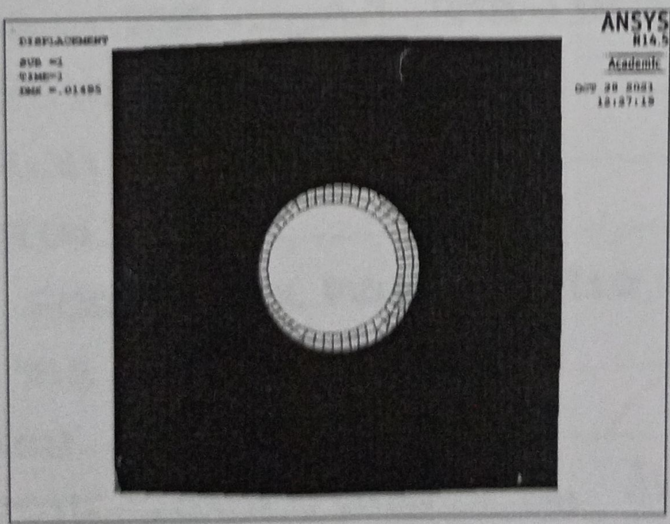
STRESS ANALYSIS OF PLATE WITH HOLE



DISPLACENENT VECTOR SUM



VON MISES STRESS



DEFORMED SHAPE

Exp. No.: 5
Date : 29/9/2021

STRESS ANALYSIS OF PLATE WITH HOLE.

Aim:-

to analysis the stress induced in the given plate when hole for given loading condition using ansys software

Software used:-

"ANSYS 14.5"

Procedure:-

Step1:- change directory -> creat folder -> change Job name

Step 2:- main menu -> preference -> structure -> element type ->
Add select 4 node 182

Step3 !- material property.
Choose material property and enter value a Ex

Step 4:- modelling
Select area -> carbit area -> rectangle
(x1, x2) -> 0.100
(y1, y2) -> 0.100

Step5 :- offset
select work plane -> offset wp
xiy offset 50, 50

Step 6:- creat
create -> area -> circle -> by dimension -> ans
select modelling -> operate -> boolean -> substract
meshing

Step7:- select the meshing -> mesh tool -> size ctrl -> manual
global -> size mesh -> Area Force mesh -> global size
Area -> 85.

Exp. No.:
Date :

Step 8:- loads.

define loads -> structure -> displacement -> nodal solution -> time pof.

Step 9:- Force

select Force -> on nodal -> ok -> enter value -> 500

Step 10:- Solve.

Solve -> current ls

Then choose general post proc -> plot result deformed size -> nodal solution.

Step 11:- Result:

Identity result -> select plot result -> contour line element.

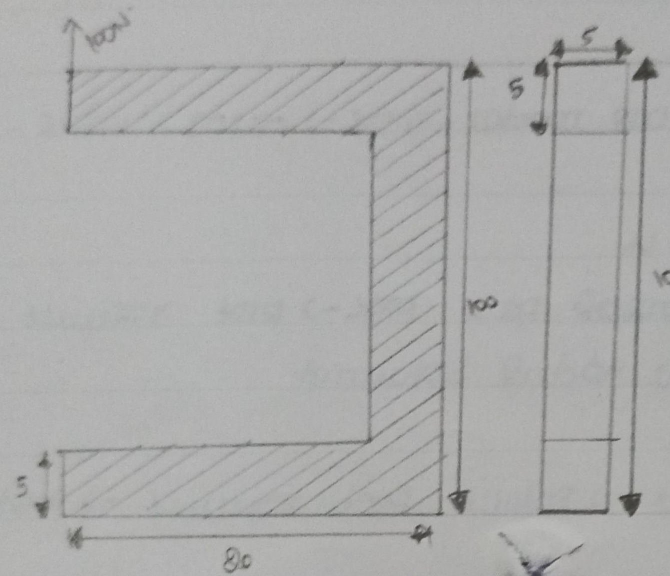
Finally take the print out of result.

PRACTICAL EVALUATION	
Observation (10)	8
Calculation / Result (10)	8
Viva (10)	8
Total Marks (30)	24
Signature of Faculty	PSL 13/10

Result:

Thus the stress analysis of the given plate with holes done by the using ANSYS 14.5 Software.

Analysis of Axisymmetric elements.

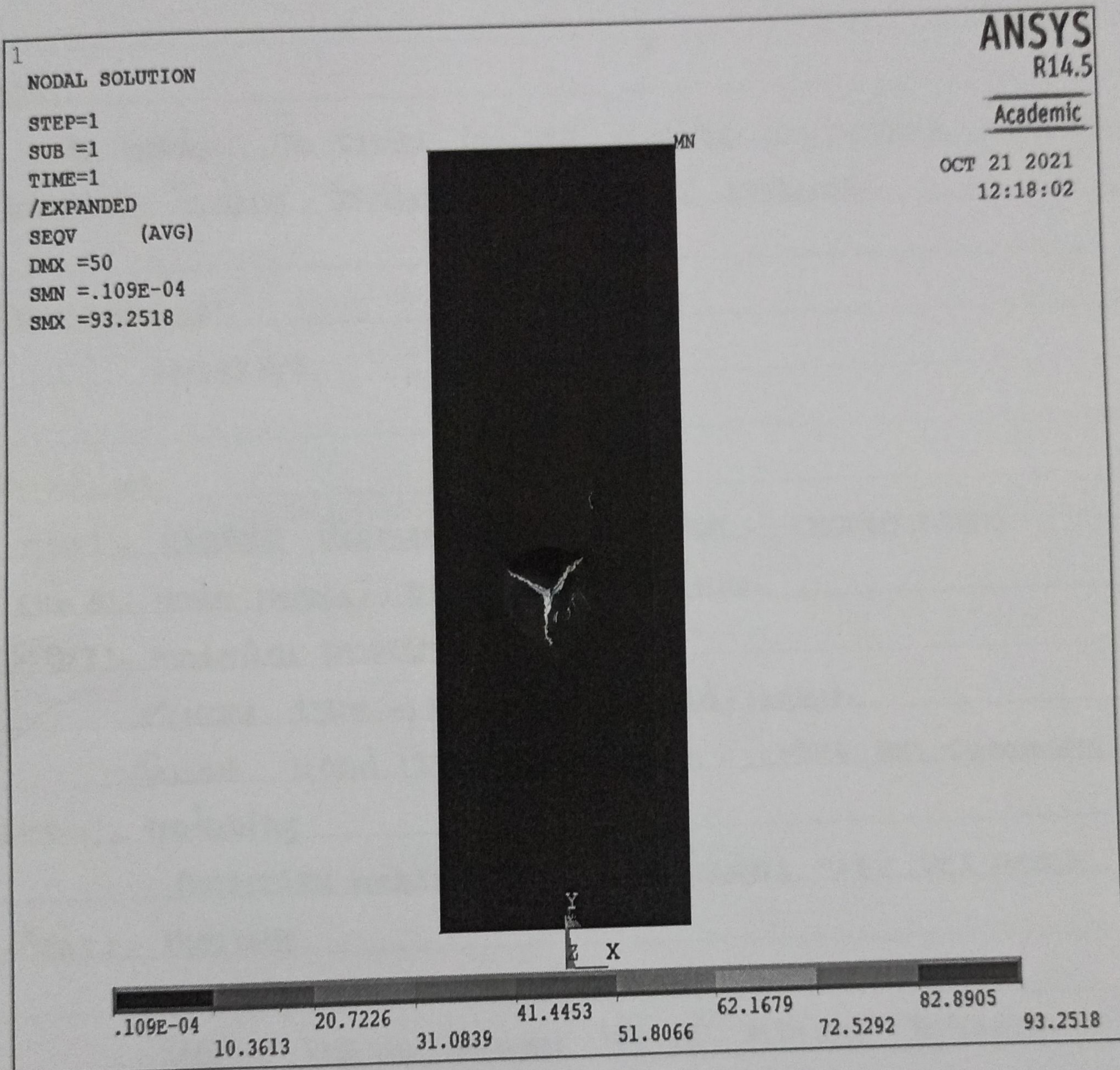


ALL DIMENSIONS ARE IN MM.

PRXY = 0.3

EX = 8E5 N/mm^2.

STRESS ANALYSIS OF AXIS SYMMETRIC COMPONENTS



Exp. No.: 6
Date : 22.9.2021

ANALYSIS OF AXI-SYMMETRIC ELEMENT:

Aim:-

To analysis the stress induced in given axysymmetric element for given loading condition using ANSYS software.

Software used:-

ANSYS 14.5

Procedure:-

Step 1:- change directory -> creat folder -> change name.

Step 2:- main models -> preferences -> structure.

Step 3:- material property.

element type -> Add -> Edit -> Add -> select.

Quad 4and 185 note -> options -> select Axi-symmetric.

Step 4:- modelling

material model -> youngs modulus -> Ex 2E5 P=0.3

Step 5:- meshing

meshing -> create -> area -> rectangle -> dimension

1(0,20) 2(5,20) 3(0,20) 4(0,5) 5(0,100) 6(45,100)

Output -> bonds -> Add.

size control -> manual size -> Area -> All Area -> mesh.

Area -> free mesh ok.

Step 6:- solution:

solution -> define load -> displacement symmetric.

boundary (41) -> oline 48.1 select circle

by -> location + v direction -> so define load ->

displacement -> on load -> Dir all zero 0.

Exp. No.:............		Page No.: 31
Date :............		

Step 7:- Force.

Force -> On keypoint -> Fy 100 at node Fx = 100 at node

Step 8:- Result.

Solve -> current LS -> close.

Step 9:- General post proc.

Post processor -> plot results -> define shape -> All dimension -> stress von mises stress

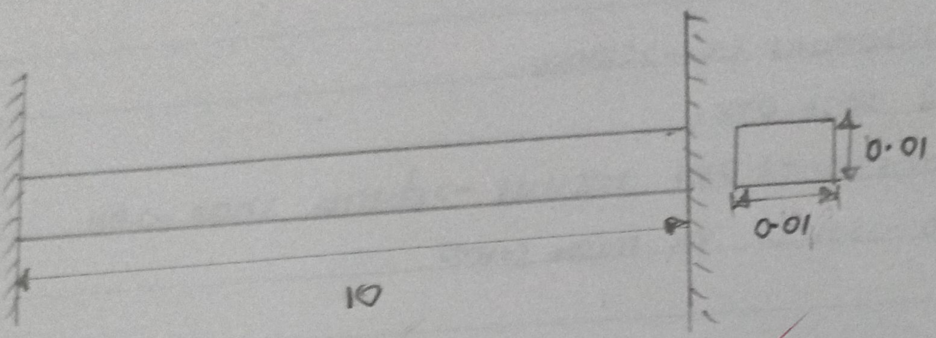
PRACTICAL EVALUATION	
Observation (10)	8
Calculation / Result (10)	8
Viva (10)	8
Total Marks (30)	24
Signature of Faculty	PSL 13/10

Result:-

Thus the stress analysis for axisymmetric element done by using "ANSYS 14.5" software

MODAL

ANALYSIS: FIXED BEAM.



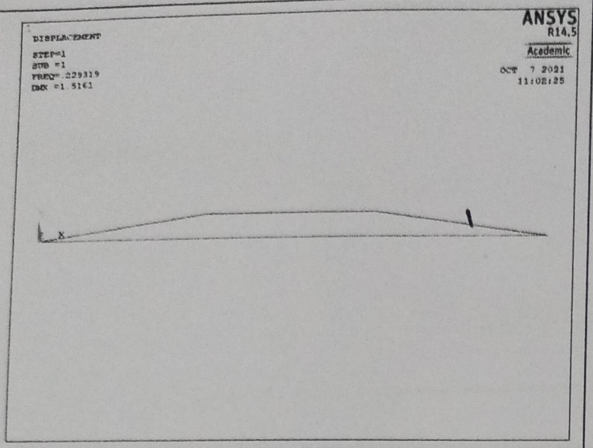
ALL DIMENSIONS ARE IN M.

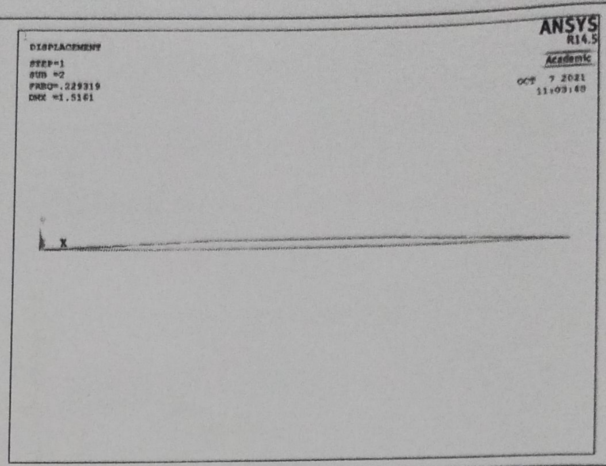
E = 206800 e6 Mm^2.

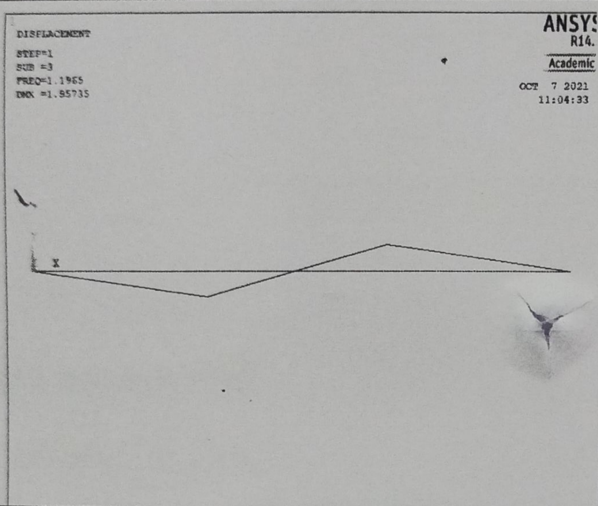
PRXY = 0.27

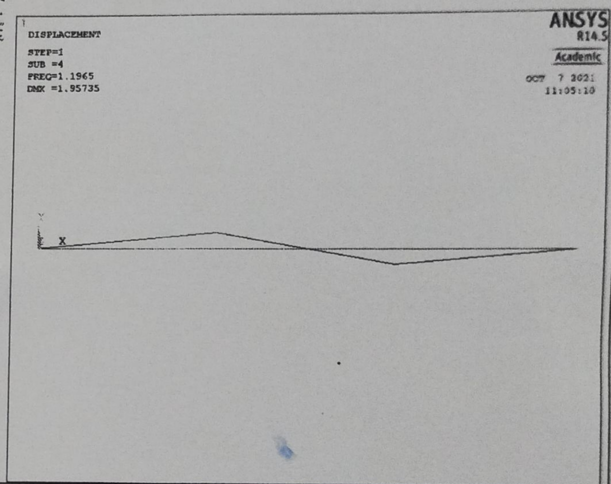
P = 7830 kg/m^3

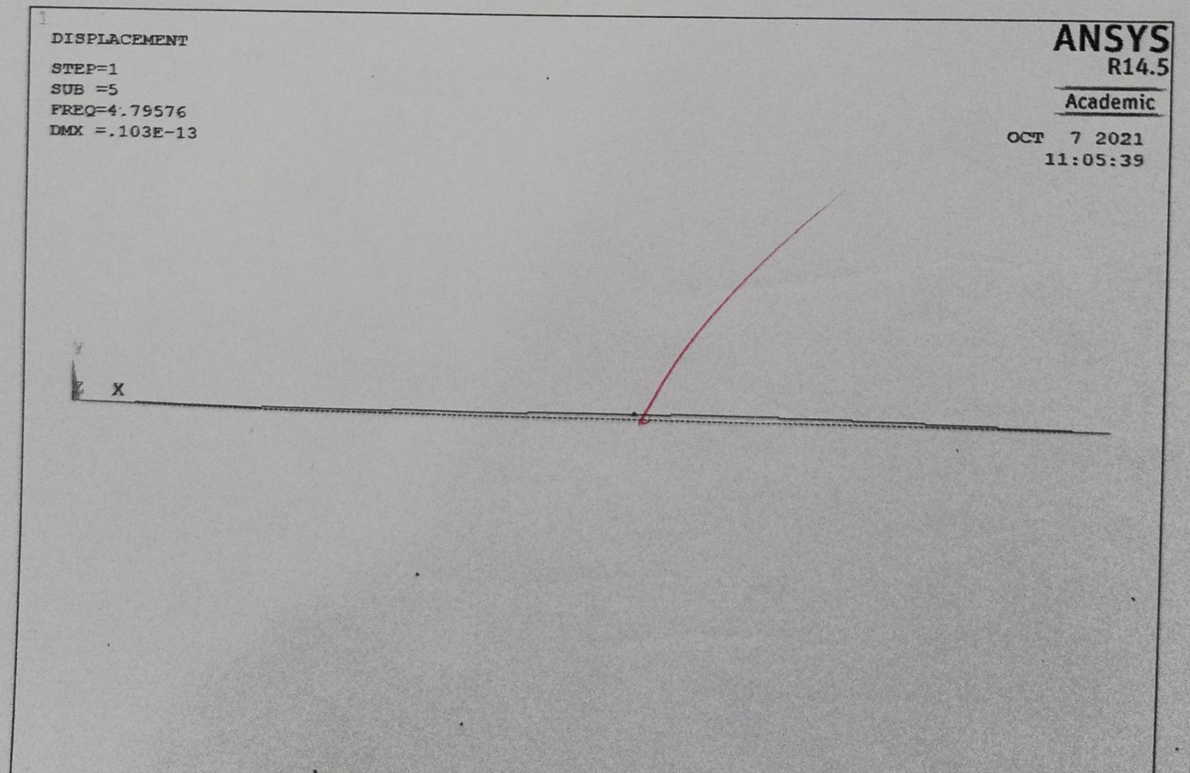
MODAL ANALYSIS OF FIXED BEAM











Exp. No.: 7
Date: 13/10/21

Page No.: 33

Modal analysis of fixed beam

Aim:

To analysis the modal frequency in the given fixed beam using ANSYS software.

Software used:

ANSYS 14.5

Procedure:

Step 1: utility menu -> change directory -> create folder change Job name -> fixed modal.

Step 2: main menu -> preference -> structural -> ok.

Step 3: preprocessor -> material props -> structural linear -> Elastic -> isotropic -> enter value as Ex D=0.37 P=7850

Step 4: preprocessor -> solution -> beam -> create -> keypoint -> individual -> Enter value as (0,0,0) (1,0,0)

Step 5: main menu -> preprocessor -> linear -> straight line plot point.

Step 6: preprocessor -> meshing -> size cntr -> manual size -> mesh.

Step 7: main menu -> preprocessor -> load -> analysis type -> apply structural -> displacement keypoint -> software

Step 8: General post processor -> analysis options -> block.

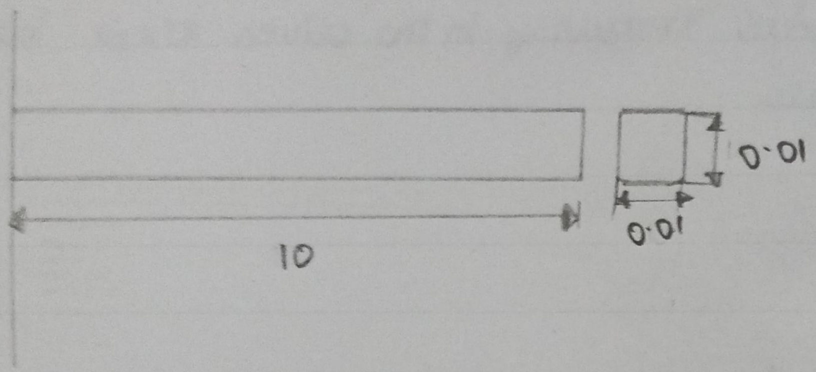
Step 9: main menu -> ~~general~~ post processor -> result summary

Result:

Thus the modal analysis of given fixed beam was analysed by using ANSYS 14.5 software.

PRACTICAL EVALUATION	
Observation (10)	10
Calculation / Result (10)	9
Viva (10)	9
Total Marks (30)	28
Signature of Faculty	

nodal displacement cantilever beam :-



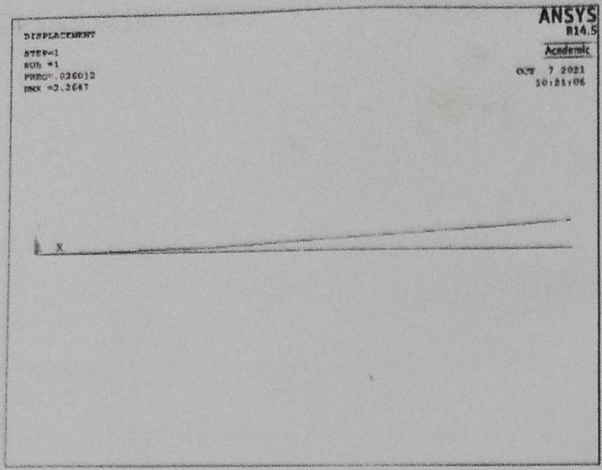
ALL DIMENSIONS ARE IN M

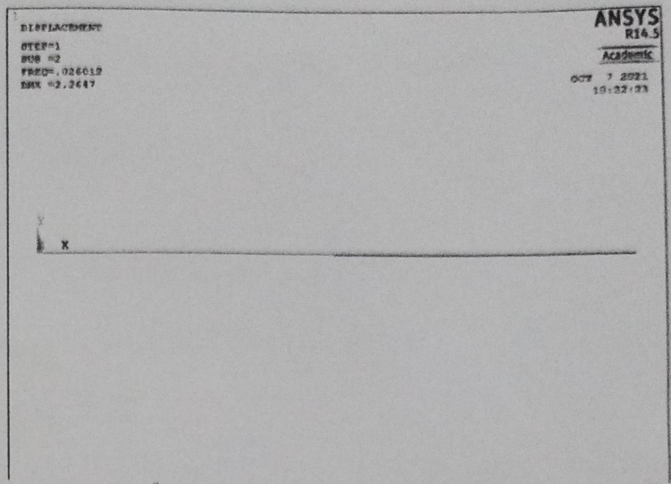
$E = 206800e6 \ N/m^2$

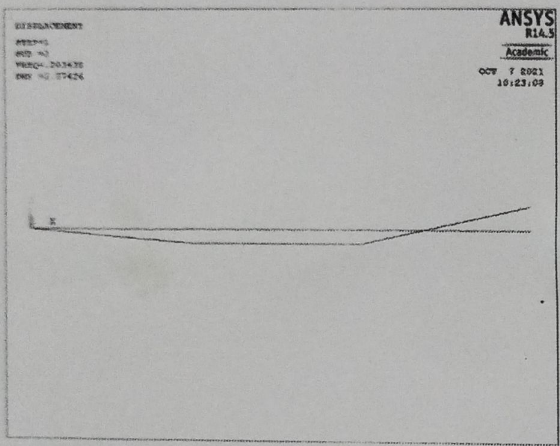
PRXY = 0.27

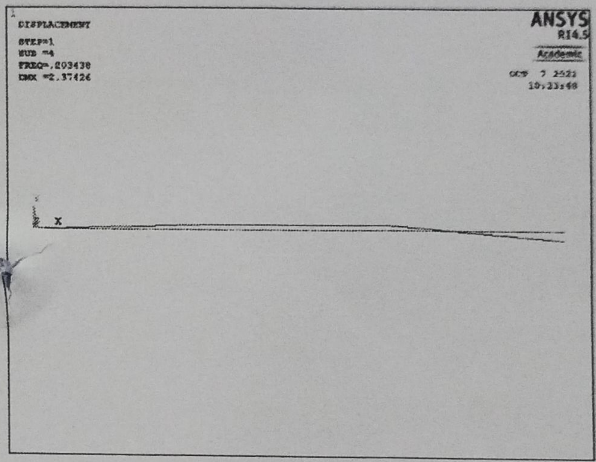
$\rho = 7836 \ kg/m^3$

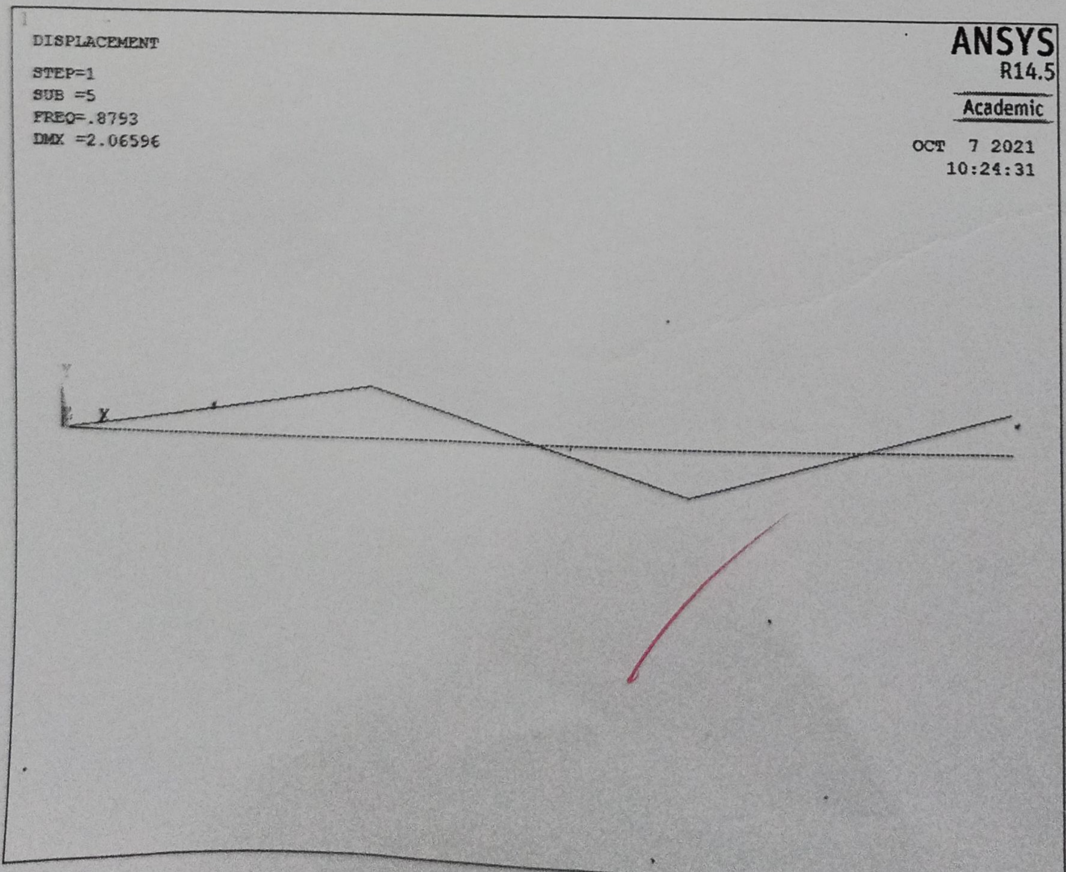
MODAL ANALYSIS OF CANTILEVER BEAM











Exp. No.: 8
Date : 13.10.21

Modal analysis of cantilever beam.

Aim:-

to perform modal analysis on a given cantilever beam using "ANALYSIS 14.5" software.

Software used:-

ANSYS 14.5

Procedure:-

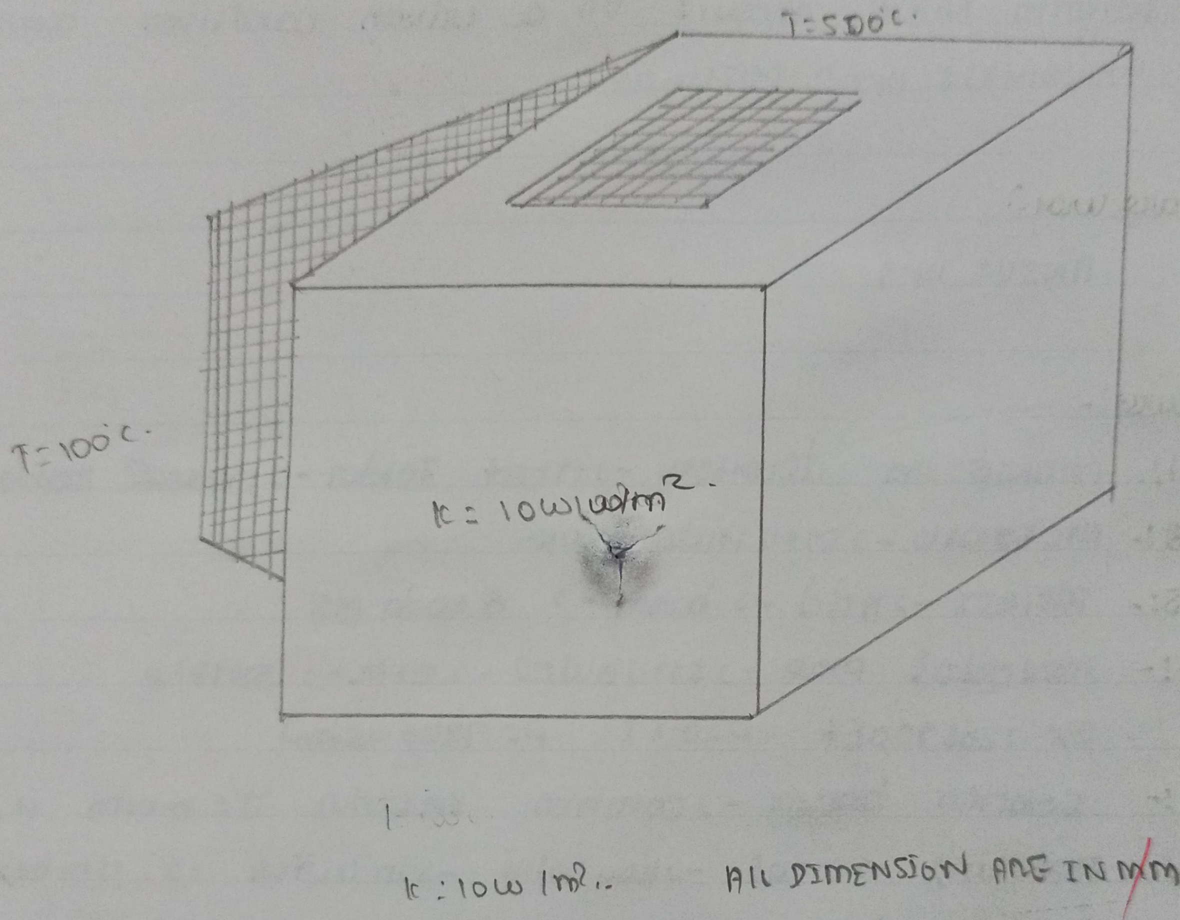
Step 1:- change the directory -> create folder -> change name.
Step 2:- preferences -> structure -> ok.
Step 3:- process -> add -> beam -> 2 node 188
Step 4:- material prop -> structural -> EX -> isotropic
EX : 206800E6 D = 0.27 P = 7800 kg/m³
Step 5:- section beam -> common section B = 0.01m H = 0.01m
Step 6:- modelling -> create -> key point -> in active (1 (0,0,0) 2 (1,0,0))
Step 7:- line -> straight line -> pick points
Step 8:- meshing -> size ctrl -> control plot -> manual size
Step 9:- load analysis type -> new analysis modal size
Step 10:- solution -> solve -> current solution done
Step 11:- load define loads -> apply -> structural -> displacement on key -> All.
Step 12:- general post processor result

PRACTICAL EVALUATION	
Observation (10)	8
Calculation / Result (10)	8
Viva (10)	8
Total Marks (30)	24
Signature of Faculty	P&L 27/10

Result:-

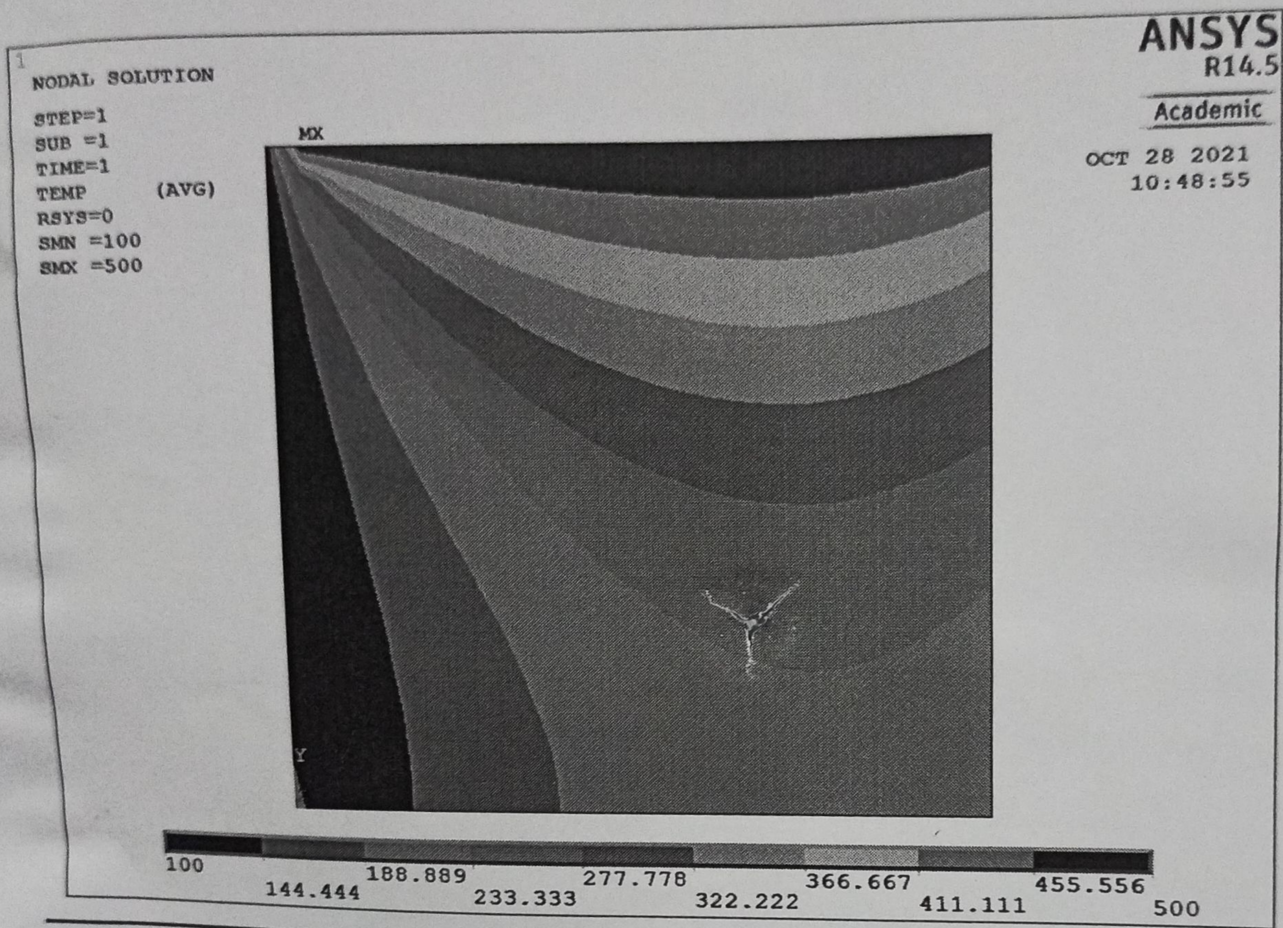
Thus the modal analysis of given cantilever beam was analysised by using "ANSYS 14.5" software.

Thermal mixed boundary :-



THERMAL MIXED BOUNDARY

NODAL TEMPERATURE



Exp. No.: 9
Date: 20/10/21

Thormal mixed boundary:.

Aim:-

to perform the thermal analysis of the given rectangular area with diff boundary temperature using ANSYS 14.5 software

Software used:.

ANSYS 14.5

Procedure:.

Step 1:- utility menu

utility menu -> change Job name -> enter Job name

utility menu -> change title -> Enter new title.

Step 2:- preference

preference - thermal - ok

Step 3:- preprocessor.

preprocessor -> element type -> Add edit / delete -> solid Quad node

preprocessor -> material prop -> thermal conduction 51

preprocess -> modeling -> create -> Area -> rectangle -> By dimen

X 0 1

Y 0 1

preprocess -> meshing -> mesh -> Area -> Mesh.

preprocessor node -> Analysis -> new analy -> steady state

preprocessor -> define load -> Apply thermal -> temperature on line -> ok -> temperature -> 100 -> ok.

Exp. No.:
Date :

pre processor -> cound load -> apply -> convection -> bottom line

Step 4 : Solution :-

General post pre -> plot results -> contour -> nodal solution
click -> next bar -> place -> node -> ok
General post proc -> map on the solution.
General post proc -> path operation place path item on graph temp -> ok.

PRACTICAL EVALUATION	
Observation (10)	8
Calculation / Result (10)	9
Viva (10)	8
Total Marks (30)	25
Signature of Faculty	PSL 27/10

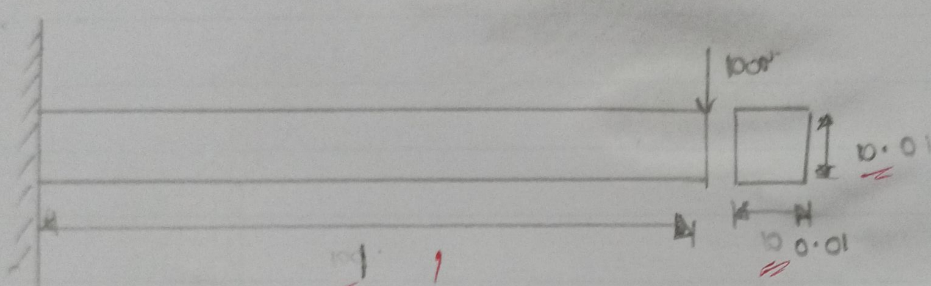
Analysis value :-

minimum temperature = ~~100°c~~ 100°c
maximum temperature = 500°c

Result :-

Thus the thermal analysis & thermal mixer in performed using the analysis 14.5 software.

hormonic analysis of cantiliver beam.



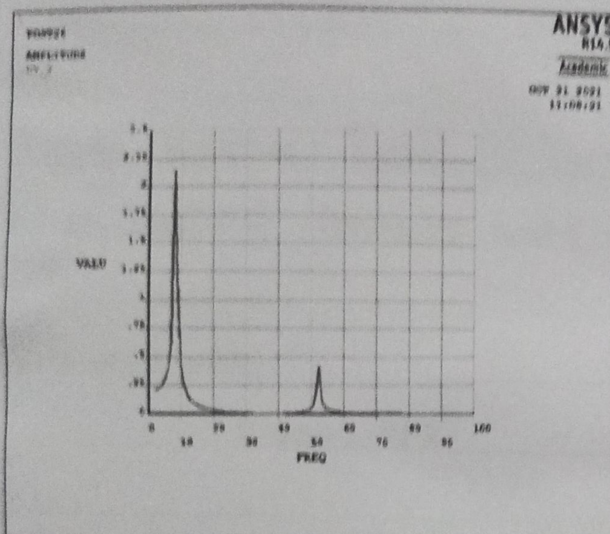
ALL DIMENSION ARE IN M

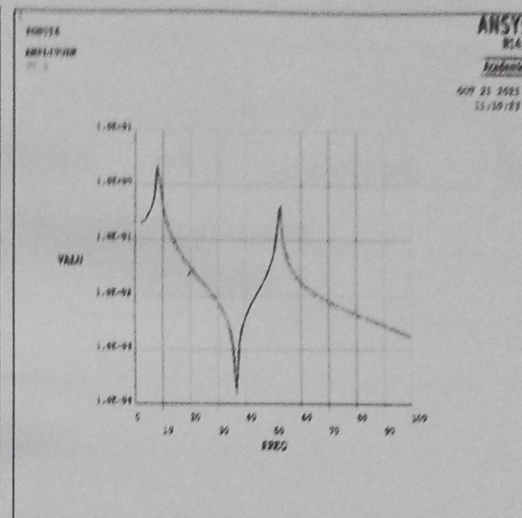
E = 206800E6 N/m^2

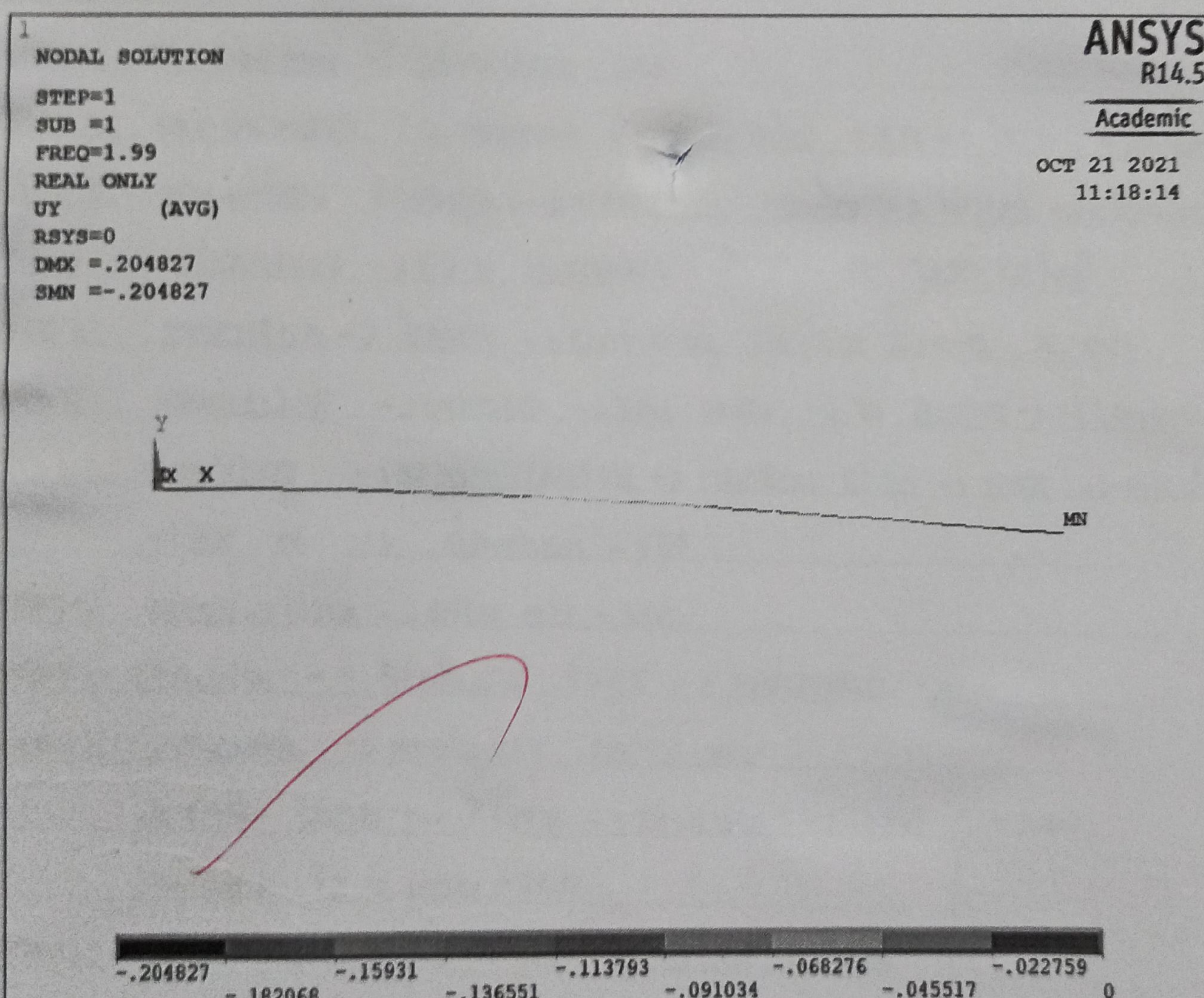
PRXY = 0.3

ρ = 7830 kg/m^3

HARMONIC ANALYSIS OF CANTILEVER BEAM







Exp. No.: 10 | Date: 20/10/21 | Page No.: 43

harmonic analysis of cantilever beam

aim:-

to perform harmonic analysis on a cantilever beam for given loading condition using "ANSYS" software

software used:

"ANSYS 14.5"

procedure:-

Step 1:- Preprocessor -> structure -> ok.

Step 2:- preprocessor -> element type -> add edit -> beam -> 3 node

Step 3:- material property -> material model -> linear -> elastic -> isotropic -> Ex = 2.068E6 P: 0.3 ρ = 7830 kg/m³

Step 4:- section -> beam -> common section B: 0.01 H: 0.01

Step 5:- modeling -> create -> key point -> in active -> 1(0,0,0) 2(1,0,0)

Step 6:- meshing -> size control -> manual size -> lines -> all lines. -> no element division -> 25

Step 7:- mesh -> lines -> pick all -> ok.

Step 8:- solution -> Analysis type -> harmonic -> ok.

Step 9:- solution -> Analysis option -> ok. define load -> Apply -> structure -> force comment -> keypoint Fy = 100N -> ok

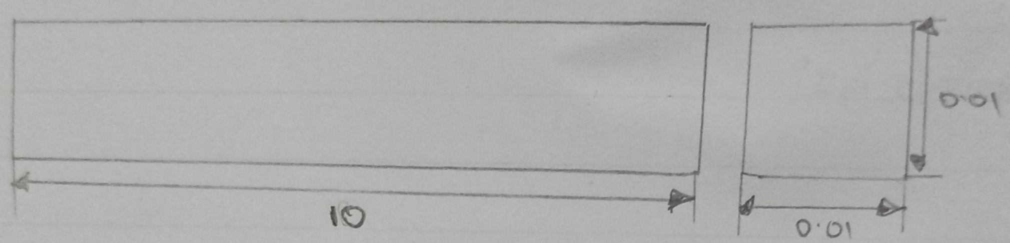
Step 10:- solve -> current -> ok solution is done

PRACTICAL EVALUATION	
Observation (10)	9
Calculation / Result (10)	9
Viva (10)	8
Total Marks (30)	26
Signature of Faculty	PSL

27/10

Result:-

Thus the harmonic analysis on a cantilever beam was performed by using ansys 14.5 software model analysis of simply supported beam.



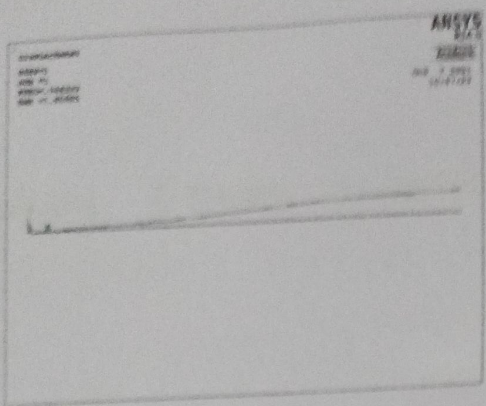
ALL DIMENSION ARE IN M

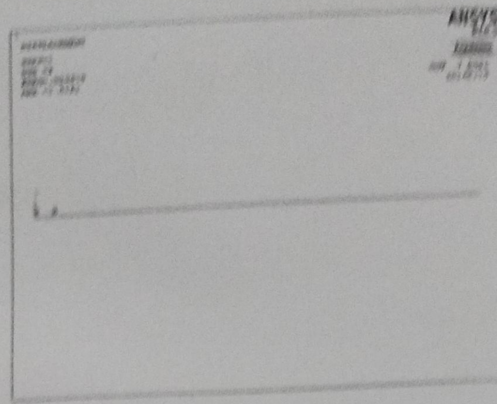
$E = 206800e6 \ N/m^2$

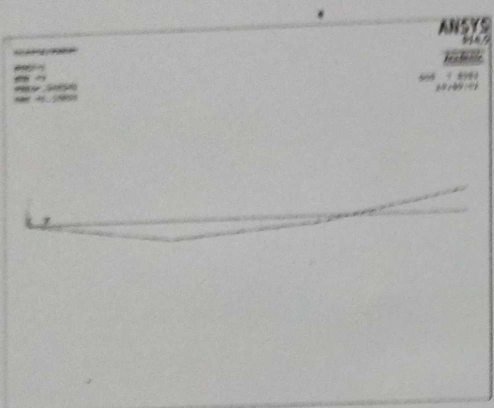
$PRXY = 0.27$

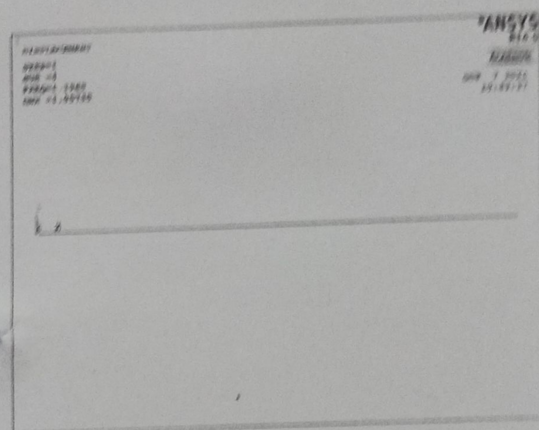
$\rho = 7830 \ kg/m^3$

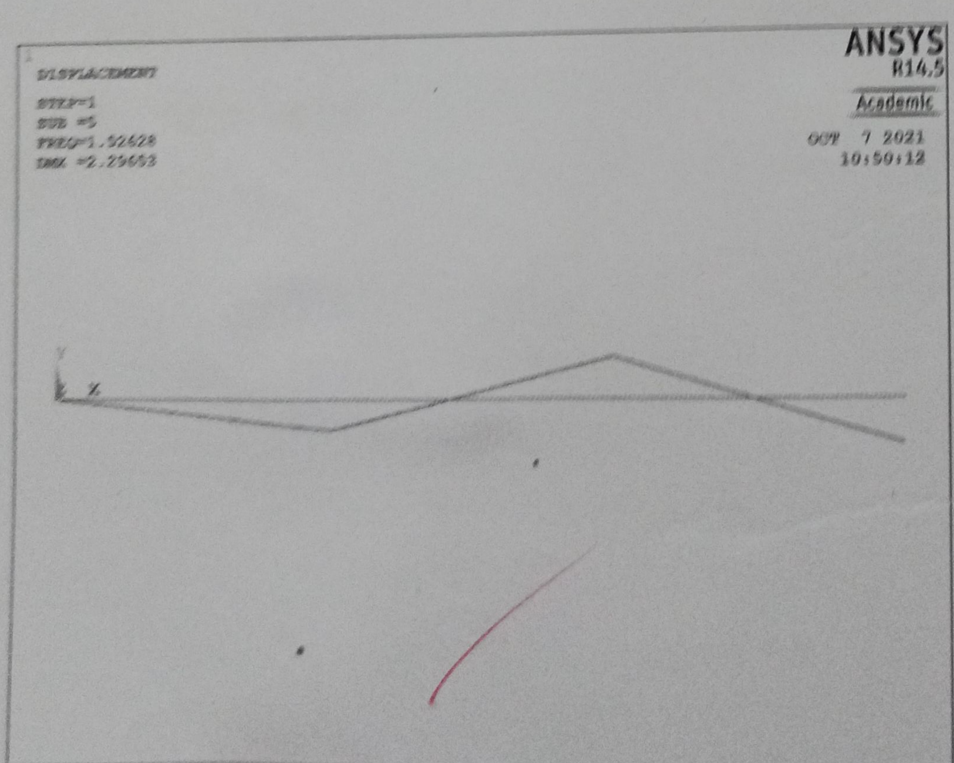
MODAL ANALYSIS OF SIMPLY SUPPORTED BEAM











Exp. No.: 11
Date: 27/10/21

Page No.: 45

modal analysis of simply supported beam.

Aim:-

To analysis the modal frequency in the given simply supported beam using Ansys software

Software used:

ANSYS 14.5

Procedure:-

Step 1 : utility menu -> change directory -> create folder -> change Job name -> fixed modal

Step 2 :- main menu -> preference -> structural -> ok.

Step 3 :- preference -> material prop -> structural -> elastic -> isotropic -> Enter Ex: 206800 eb -> P=0.27 -> P=7830

Step 4 :- preprocessor -> section -> beam -> create -> key point -> in ansys -> enter value (0,0,0) (1,0,0)

Step 5 :- main menu -> preprocessor -> line straight line -> pick point.

Step 6 :- preprocessor -> meshing -> size control -> manual size. mesh.

Step 7 :- main menu -> preprocessor -> Load -> Analysis -> Apply - structural -> displacement -> second poin -> click ux, uz, rot x, roty, rotz

Step 8 :- General post processor -> Analysis option -> overs

Exp. No.:

Date :

Step 9 :- General post processor -> read result -> first set -> plot solution -> deformed shape -> def + undeformed -> ok

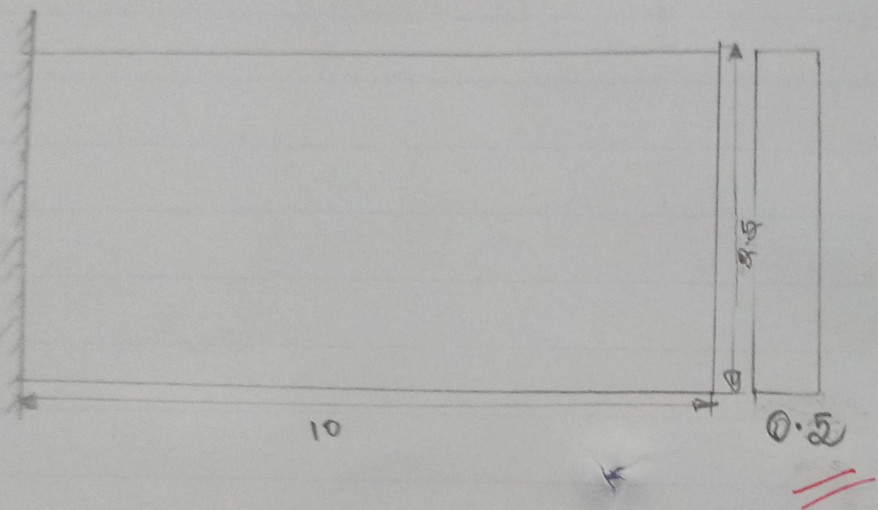
Step 10 :- next set -> plot results -> deformed shape -> def + undeformed.

PRACTICAL EVALUATION	
Observation (10)	8
Calculation / Result (10)	8
Viva (10)	8
Total Marks (30)	24
Signature of Faculty	PSL 30/10/2

Result :-

Thus the modal analysis on a simply supported beam was analysed by using ansys software.

model analysis 8 2D Plate



ALL DIMENSIONS ARE IN M

$E = 30e6\ N/m^2$

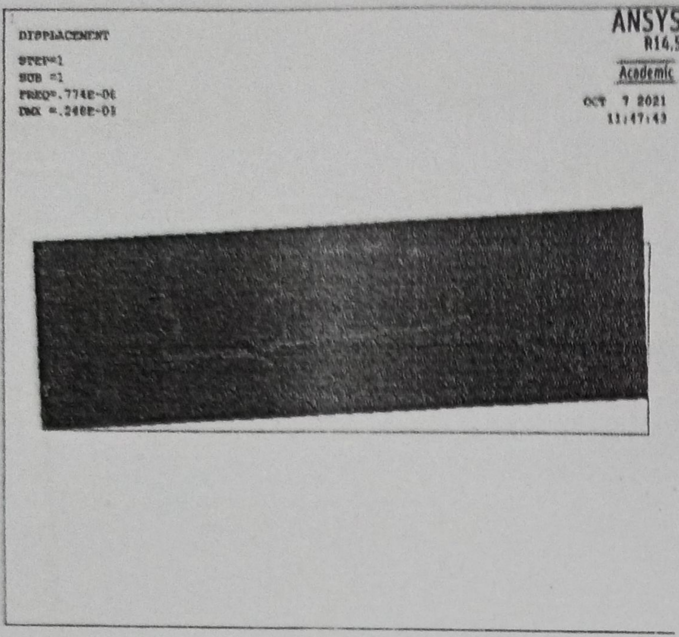
$x_1 = 0$ $x_2 = 10$

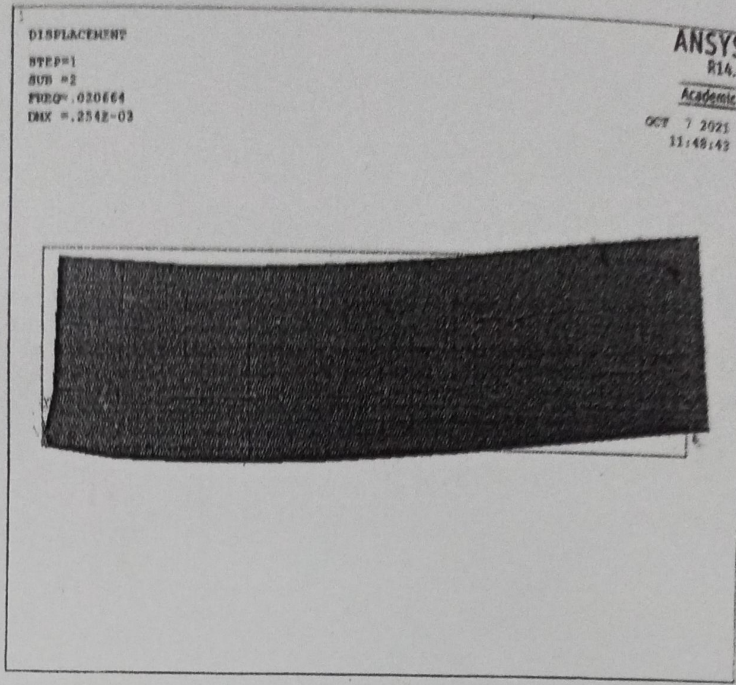
PRXY $= 0.82$

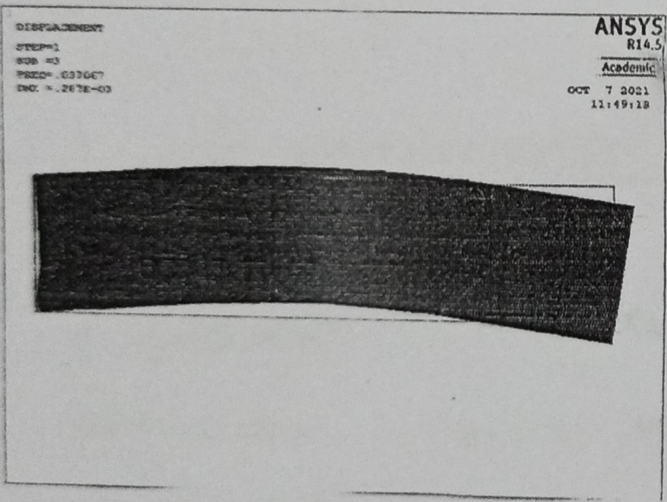
$y_1 = 0$ $y_2 = 2.5$

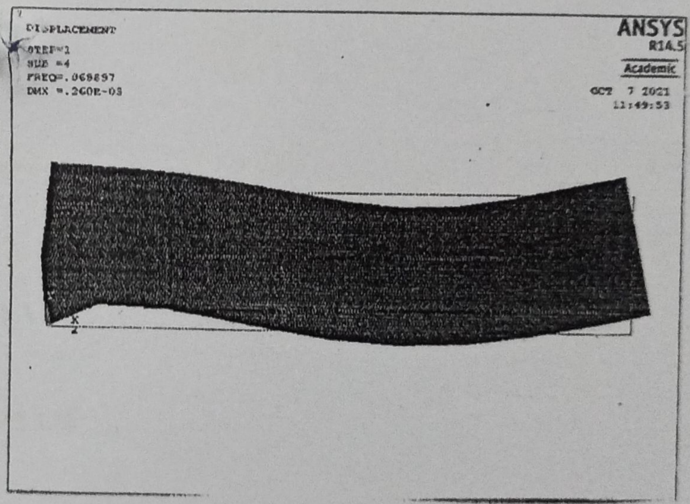
$P = 7830\ kg/m^3$

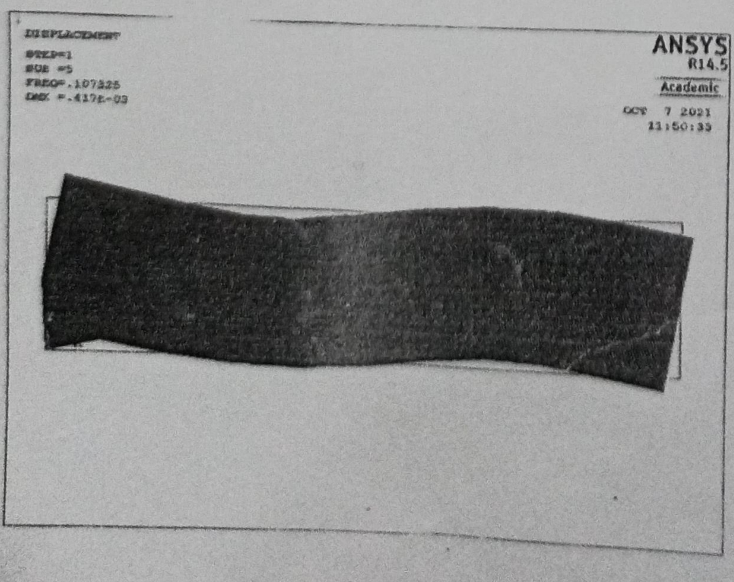
MODAL ANALYSIS OF 2D CANTILEVER PLATE











Exp. No.: 12
Date : 27/10/21

Page No.: 49

modal analysis of 2D plate with cantilever support

Aim:-

to analyse the modal ~~frequency~~ frequency in the 2D plate with cantilever with cantilever beam using ANSYS 14.5

Software used:-

ANSYS 14.5

Procedure:-

Step 1:- utility menu -> change -> create folder -> change Job name -> fixed modal.

Step 2:- main menu -> preference -> structural -> ok.

Step 3:- preference -> material prop -> structural Elastic -> Elastic -> isotropic -> Enter -> Ex = 30e6 Prxy: 0.27 D:7830

Step 4:- preprocessor -> select -> Area -> rectangle (x1,y2) (10,2.5) (x1,y1) (0,0)

Step 5:- meshing -> size control -> manual size -> Enter.

Step 6:- mesh -> Area -> free mesh.

Step 7:- solution -> Analysis type -> modal -> Analysis operation -> Enter

Step 8:- define load -> Apply -> displacement -> on key point -> Arrange to the wall day.

Step 9:- solution -> solve -> solution is done.

Step 10:- General post preprocessor -> read result -> finish set -> plot result -> deformed shape, result

PRACTICAL EVALUATION	
Observation (10)	8
Calculation / Result (10)	8
Viva (10)	8
Total Marks (30)	24
Signature of Faculty	P8L 30/10

Result:

Thus the model analysis of the 3D plate with cantiliver beam ~~was~~ analysis by using ansys software

Completed

P.S.R. ENGINEERING COLLEGE

(Autonomous Institution, Affiliated to Anna University, Chennai)

Sevalpatti, Sivakasi – 626140

Department of Mechanical Engineering

ATTAINMENT VALUE OF LABORATORY OUTCOMES

161ME77 - ANALYSIS LABORATORY

Course Outcomes:

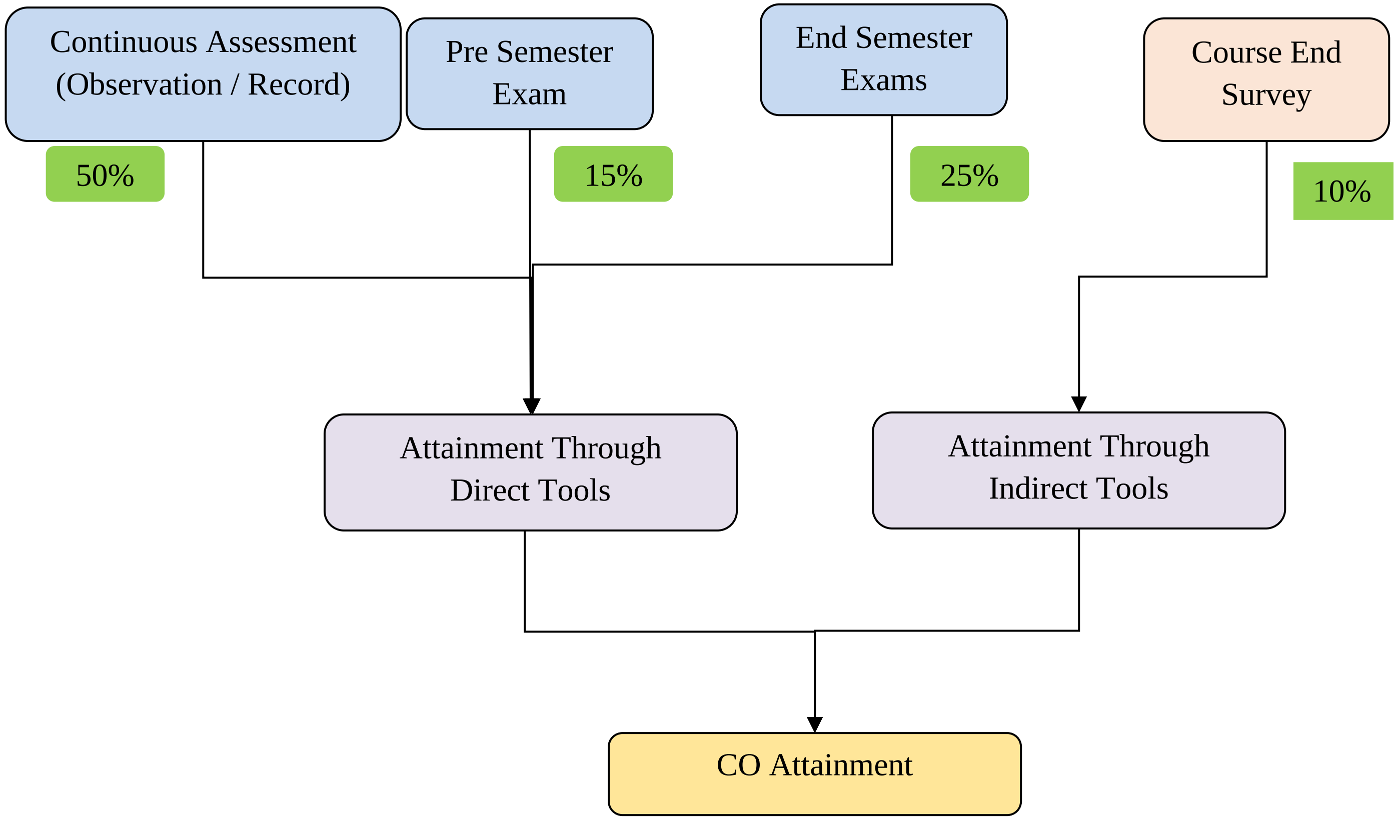
The students will be able to

CO1. Demonstrate the features of ANSYS software
CO2. Validate the stress analysis in beam problems with empirical formulas
CO3. Explicit the stress analysis of a plate with a circular hole and axi-symmetric component
CO4. Identify the need of mode frequency analysis in 2D component
CO5. Realize the Thermal analysis of a 2D component
CO6. Import any solid model to ANSYS for contact analysis

Course Outcomes	Program Outcomes (POs)												Program Specific Outcomes (PSOs)			
	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	2	3	3						2	3		2	2			2
CO2	2	1	1		2				2	2		3	3	3	1	2
CO3	2	3	3		2					2		2	3	3	2	2
CO4	2	3	3		2					2		2	3	3	2	2
CO5	2	3	3		2				2	2		2	3	3	2	2
CO6	2	3	3		2				2	1		3	2	3	3	3

1: Slight (Low) 2: Moderate (Medium) 3: Substantial (High)

COURSE OUTCOMES ATTAINMENT – PRACTICAL COURSES



[Reference from Evaluation Manual]

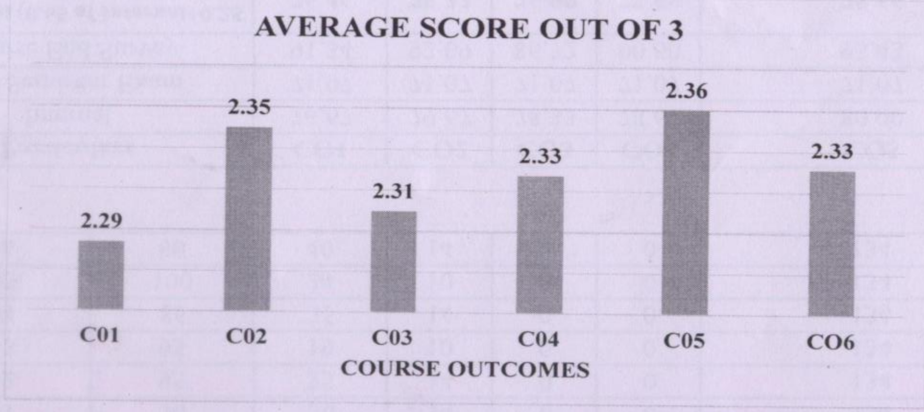
Evaluation of Course Outcomes				
Course Code & Name	: 161ME77 & Computer Aided Simulation & Analysis Laboratory			
Course Teacher	: Dr. P. Shenbaga Velu ASP/Mech & Mr.D.Sundarrajan AP/Mech			
Year / Semester	: IV/VII/ I & II			
Academic Year	:2020-21	Odd	Batch	2017-2021

Course End Survey

Course Outcomes	Marks obtained for Course Outcome					Total No of Students	Score	
	5	4	3	2	1		Net	100
CO1	90	30	14	0	0	134	612	**91.34**
CO2	95	25	14	0	0	134	617	**92.09**
CO3	95	19	10	0	0	134	581	**86.72**
CO4	85	35	14	0	0	134	607	**90.60**
CO5	100	24	10	0	0	134	626	**93.43**
CO6	80	40	14	0	0	134	602	**89.85**

Particulars	CO1	CO2	CO3	CO4	CO5	CO6
Internal	76.67	79.67	78.33	78.67	80.00	79.00
End Semester Exam	71.07	71.07	71.07	71.07	71.07	71.07
Course End Survey	91.34	92.09	86.72	90.60	93.43	89.85
Attainment (0.65 of Internal+0.25 of ESE + 0.1 of CES)	**76.46**	**78.33**	**76.99**	**77.58**	**78.66**	**77.71**

Course Outcomes	C01	C02	C03	C04	C05	CO6
Average Score Out of 5	3.82	3.92	3.85	3.88	3.93	3.89
Average Score Out of 3	2.29	2.35	2.31	2.33	2.36	2.33



P.Sd 5/2/21
Signature of the Course Tutor

5/2/2021
Signature of the Course Co-ordinator/Moderator

5/2/2021
Head of the Department

161ME77- Computer Aided Simulation and Analysis Laboratory
CO- PO Mapping

Course Outcomes	Program Outcomes (POs)												Program Specific Outcomes (PSOs)			
	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	2	3	3						2	3		2	2			2
CO2	2	1	1		2				2	2		3	3	3	1	2
CO3	2	3	3		2					2		2	3	3	2	2
CO4	2	3	3		2					2		2	3	3	2	2
CO5	2	3	3		2				2	2		2	3	3	2	2
CO6	2	3	3		2				2	1		3	2	3	3	3

Internal CO-PO Mapping

CO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	76.67	76.67	76.67						76.67	76.67		76.67	76.67			76.67
CO2	79.67	79.67	79.67		79.67				79.67	79.67		79.67	79.67	79.67	79.67	79.67
CO3	78.33	78.33	78.33		78.33					78.33		78.33	78.33	78.33	78.33	78.33
CO4	78.67	78.67	78.67		78.67					78.67		78.67	78.67	78.67	78.67	78.67
CO5	80.00	80.00	80.00		80.00				80	80.00		80.00	80.00	80.00	80.00	80.00
CO6	79.00	79.00	79.00		79.00				79	79.00		79.00	79.00	79.00	79.00	79.00

External CO-PO Mapping

CO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	71.07	71.07	71.07						71.07	71.07		71.07	71.07			71.07
CO2	71.07	71.07	71.07		71.07				71.07	71.07		71.07	71.07	71.07	71.07	71.07
CO3	71.07	71.07	71.07		71.07					71.07		71.07	71.07	71.07	71.07	71.07
CO4	71.07	71.07	71.07		71.07					71.07		71.07	71.07	71.07	71.07	71.07
CO5	71.07	71.07	71.07		71.07				71.07	71.07		71.07	71.07	71.07	71.07	71.07
CO6	71.07	71.07	71.07		71.07				71.07	71.07		71.07	71.07	71.07	71.07	71.07

	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
Internal	78.7	78.6	78.6		79.1				78.8	78.5		78.8	78.8	79.1	79.1	78.7
External	71.1	71.1	71.1		71.1				71.1	71.1		71.1	71.1	71.1	71.1	71.1

5/2/21
Signature of the Course Tutor

5/2/2021
Signature of the Course Co-ordinator/Moderator

5/2/21
Programme Co-ordinator

5/2/21
Head of the Department

Evaluation of PO & PSO

Course Code & Name	: 161ME77 & Computer Aided Simulation & Analysis Laboratory
Year / Semester	: IV / VII
Direct Tool	: Program Outcomes (POs) & Program Specific Outcomes (PSOs)

Table 3. Average attainment score of Course Outcomes based on Program Outcomes (POs) & Program Specific Outcomes (PSOs)

Attainment of POs & PSOs from a Course considering all the Direct tools

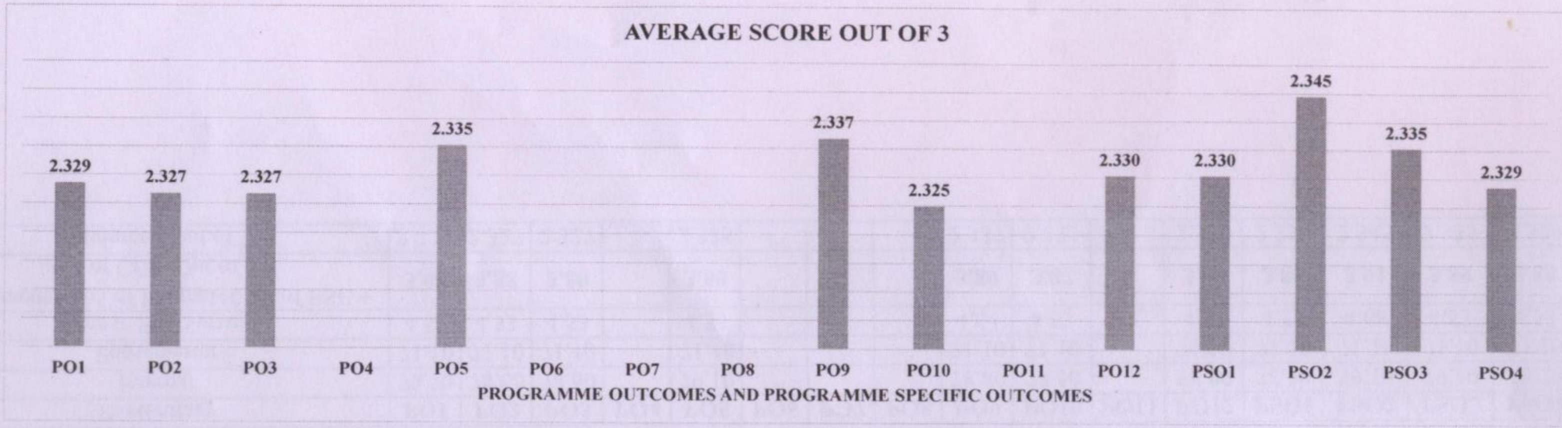
Course End Survey from CO Attainment

Course Outcomes	Survey Score
C01	91.34
C02	92.09
C03	86.72
C04	90.60
C05	93.43
CO6	89.85

CO- PO - PSO Mapping

CO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
C01	91.34	91.34	91.34						92.00	91.34		91.34	91.34			91.34
C02	92.09	92.09	92.09		92.09				96.00	92.09		92.09	92.09	96.00	92.09	92.09
C03	86.72	86.72	86.72		86.72					86.72		86.72	86.72	88.00	86.72	86.72
C04	90.60	90.60	90.60		90.60					90.60		90.60	90.60	88.00	90.60	90.60
C05	93.43	93.43	93.43		93.43				93.43	93.43		93.43	93.43	96.00	93.43	93.43
CO6	89.85	89.85	89.85		89.85				89.85	89.85		89.85	89.85	100.00	89.85	89.85
Score	**4.53**	**4.53**	**4.53**		**4.53**				**4.64**	**4.53**		**4.53**	**4.53**	**4.68**	**4.53**	**4.53**

Particulars	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
Internal	78.70	78.60	78.60		79.10				78.80	78.50		78.80	78.80	79.10	79.10	78.70
Endsemester	71.10	71.10	71.10		71.10				71.10	71.10		71.10	71.10	71.10	71.10	71.10
Course End Survey	4.53	4.53	4.53		4.53				4.64	4.53		4.53	4.53	4.68	4.53	4.53
Attainment (0.65 of Internal+0.25 of ESE + 0.1 of CES) Out of 5	**3.88**	**3.88**	**3.88**		**3.89**				**3.89**	**3.87**		**3.88**	**3.88**	**3.91**	**3.89**	**3.88**
Attainment out of 3	2.329	2.327	2.327		2.335				2.337	2.325		2.330	2.330	2.345	2.335	2.329



Signature of the Course Tutor	Signature of the Course Co-ordinator/Moderator	Programme Co-ordinator	Head of the Department
5/2/21	5/2/2021	5/2/21	5/2/2021