



SRI SHANMUGHA COLLEGE OF ENGINEERING AND TECHNOLOGY

(Approved by AICTE, Affiliated to Anna University and Accredited by NAAC & NBA (ECE))

Pullipalayam, Morur (P.O), Sankari (T.k), Salem (D.T) – 637 304

DEPARTMENT OF MECHANICAL ENGINEERING



ME8711- Simulation and Analysis Laboratory

Vision and Mission of the institute

VISION

To be an Institute of repute in the field of Engineering and Technology by implementing the best educational practices akin to global standards for fostering domain knowledge and developing research attitude among students to make them globally competent

MISSION

M1: Achieving excellence in Teaching Learning process using state of the art resources.

M2: Extending opportunity to upgrade faculty knowledge and skills.

M3: Implementing best student training practices for requirements of Industrial scenario of the State.

M4: Motivating faculty and students in research activity for real-time application.

Vision and Mission of the Department

VISION

To prepare competent mechanical engineers capable of working in an interdisciplinary environment contributing to society through innovation, leadership and entrepreneurship

MISSION

M1: To offer quality education which enables them in professional practice and career

M2: To provide learning opportunities in the state-of-the-art research facilities to create, interpret, apply and disseminate knowledge in their profession

M3: To prepare the students as professional engineers in the society with an awareness of environmental and ethical values

PROGRAMOUTCOMES (POs):

P01 Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals and an engineering specialization to the solution of complex engineering problems.

P02 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.

P03 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, safety, cultural, societal and environmental considerations.

P04 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.

P05 Modern tool usage: Create, select, apply appropriate techniques, resources, modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

P06 The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal, cultural issues and the consequent responsibilities relevant to the professional engineering practice.

P07 Environment and sustainability: Understand the impact of the professional engineering solutions in societal, environmental contexts, demonstrate the knowledge and need for sustainable development.

P08 Ethics: Apply ethical principles, commit to professional ethics, responsibilities and norms of the engineering practice.

P09 Individual and team work: Function effectively as an individual, as a member or leader in diverse teams and in multidisciplinary settings.

P010 Communication: Communicate effectively on complex engineering activities with the engineering community with society at large being able to comprehend, write effective reports, design documentation, make effective presentations and receive clear instructions.

P011 Project management and finance: Demonstrate knowledge, 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.

P012 Life-long learning: Recognize the need, ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

PSO1 Manufacturing: Modelling, Simulation and Analysis in the field of Manufacturing.

PSO2 Design: Develop and implement new ideas on product design with help of modern CAD tools.

PROGRAM EDUCATIONAL OBJECTIVES (PEOs)

PEO1: To prepare students to take up career in Industry, Academia as well as in Public services.

PEO2: To provide core domain and interpersonal skills to design & develop mechanical systems for Interdisciplinary applications following ethical code.

PEO3: To develop qualities to progress in entrepreneurship and research activities.

COURSE OUTCOMES:

Upon the completion of this course the students will be able to

C417.1	Perform force and stress analysis using trusses cables, stress and deflection analysis in beam with different support condition
C417.2	Perform different stress analysis on flat plates simple shells and axi-symmetric components
C417.3	Perform thermal stress and heat transfer analysis on plates and cylindrical shells
C417.4	Perform vibration and model analysis on spring masses and beams
C417.5	Perform simulation on MATLAB dealing with matrices, graphing functions of one and two variables

CO-PO MAPPING MATRIX:

Course Outcomes	Program Outcomes													
	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
C417.1	3	3	3	2	2	1		1	2			2	1	3
C417.2	3	3	3	2	2	1		1	2			2	1	3
C417.3	3	2	2	2	2	1		1	2			2	2	2
C417.4	3	3	3	2	2	1		1	2	2		2	1	3
C417.5	3	1	2	1	2	1		1	1			2	1	2
C417	3.0	2.4	2.6	1.8	2.0	1.0		1.0	1.8	2.0		2.0	1.20	2.60

LIST OF EXPERIMENTS

EXP. NO.	K Level	TITLE	Relevance to COs	Page No
1	K2	Stress analysis of a plate with a circular hole	C02	
2	K4	Stress analysis of rectangular l bracket	C02	
3	K4	Stress analysis of an axi-symmetric component	C02	
4	K4	Stress analysis of cantilever beam	C01	
5	K4	Stress analysis of simply supported beam	C01	
6	K4	Stress analysis of fixed end beam	C01	
7	K4	Mode frequency analysis of a 2D component	C04	
8	K4	Mode frequency analysis of cantilever beam	C04	
9	K4	Mode frequency analysis of simply supported beam	C04	
10	K4	Mode frequency analysis of fixed end beam	C04	
11	K4	Harmonic analysis of a 2D component	C04	
12	K4	Thermal stress analysis of a 2D component	C03	
13	K4	Conductive heat transfer analysis of a 2D Component	C03	
14	K4	Convective heat transfer analysis of a 2D Component	C03	
15	K4	Simulation of cam and follower mechanism	C05	
Content Beyond the Syllabus				
1	K4	Stress analysis of a bicycle space frame	C01	

INTRODUCTION:

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.

1. Pre-processing: defining the problem; the major steps in pre-processing are given below:

- Define keypoints/lines/areas/volumes
- Define element type and material/geometric properties
- Mesh lines/areas/volumes as required

The amount of detail required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axi-symmetric, 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:

- Lists of nodal displacements
- Element forces and moments
- Deflection plots
- Stress contour diagrams

ANSYS ENVIRONMENT:

The ANSYS Environment 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:

- **Utility Menu**

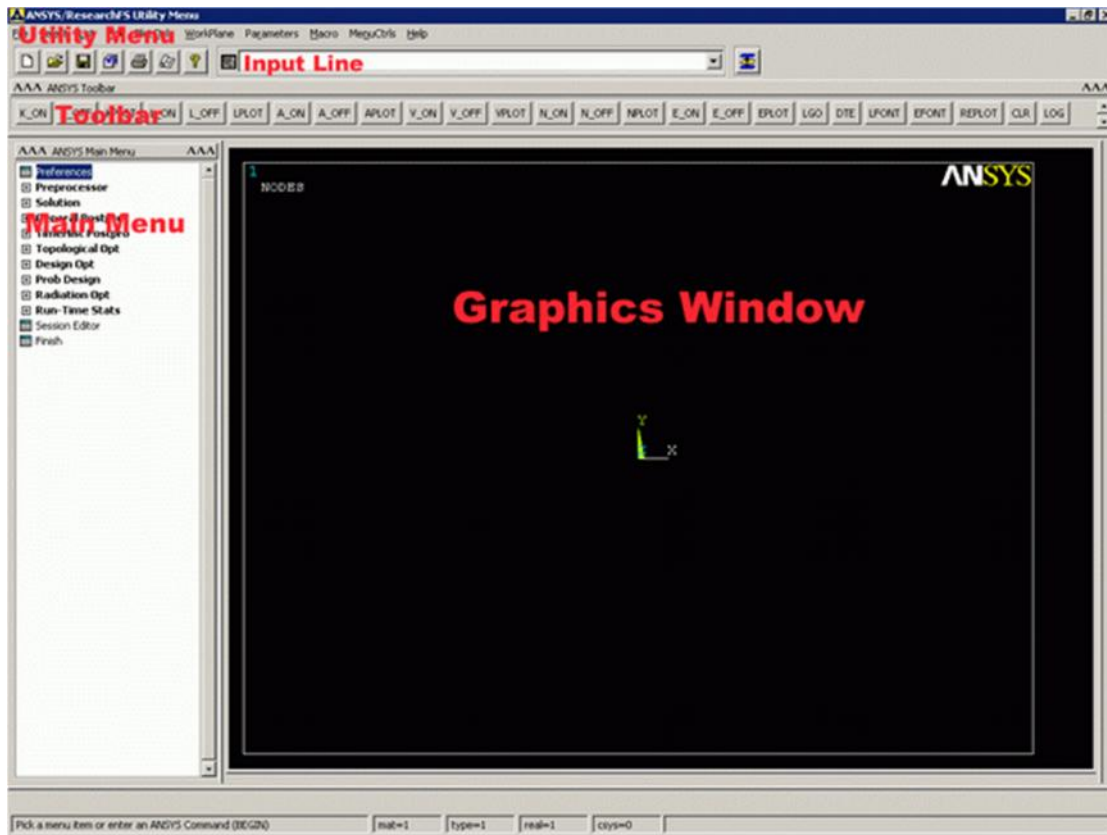
The Utility Menu contains functions that are available throughout the ANSYS session, such as file controls, selections, graphic controls and parameters.

- **Input Window**

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

- **Toolbar**

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



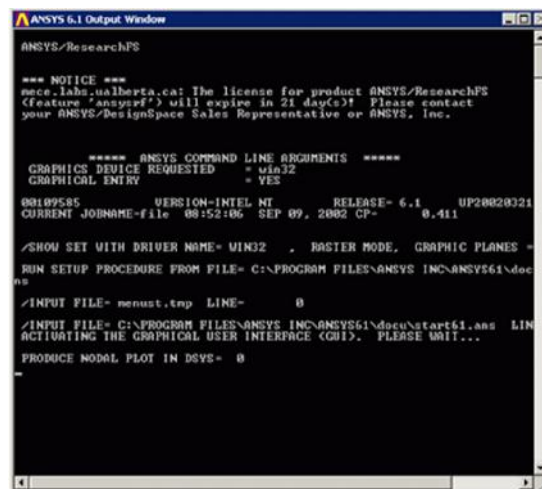
- **Main Menu**

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 modelling commands are issued. This is where you will note the greatest change between previous versions of ANSYS and version 7.0. However, while the versions appear different, the menu structure has not changed.

- **Graphics 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 be put to the front if necessary.

ANSYS INTERFACE:

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:

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 Problem**

The question that always arises is: How small do I need to make the elements before I can trust the solution?

- **What to do about it...**

In general there are no real firm answers on this. It will be necessary to conduct convergence tests! By this we mean that you begin with a mesh discretization and then observe and record the solution. Now repeat the problem with a finer mesh (i.e. more elements) and then compare the results with the previous test. If the results are nearly similar, then the first mesh is probably good enough for that particular geometry, loading and constraints. If the results differ by a large amount however, it will be necessary to try a finer mesh yet.

- **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.

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 in case 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:

- 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.
- Or, start ANSYS and select Utility Menu -> File -> Resume from... and select your job from the list that appears.

GENERAL STEPS:

Step 1: Ansys Utility Menu

File – clear and start new – do not read file – ok

File – change job name – enter new job name – xxxx – ok

File – change title – enter new title – yyy – ok

Step 2: Ansys Main Menu – Preferences

Select – STRUCTURAL - ok

Step 3: Preprocessor

Element type – select type of element from the table and the required options

Real constants – give the details such as thickness, areas, moment of inertia, etc. required depending on the nature of the problem.

Material Properties – give the details such as Young's modulus, Poisson's ratio etc. depending on the nature of the problem.

Step 4: Modeling – create the required geometry such as nodes elements, area, volume by using the appropriate options.

Step 5: Generate – Elements/ nodes using Mesh Tool if necessary (in 2D and 3D problems)

Step 6: Apply boundary conditions/loads such as DOF constraints, Force/Momentum, Pressure etc.

Step 7: Solution – Solve the problem

Step 8: General Post Processor – plot / list the required results.

Step 9: Plot ctrl's – animate – deformed shape – def+undeformed-ok

Step 10: to save the solution

ansys tool bar- save model

Ex. No: 1	Stress Analysis of a Plate with a Circular Hole
Date:	

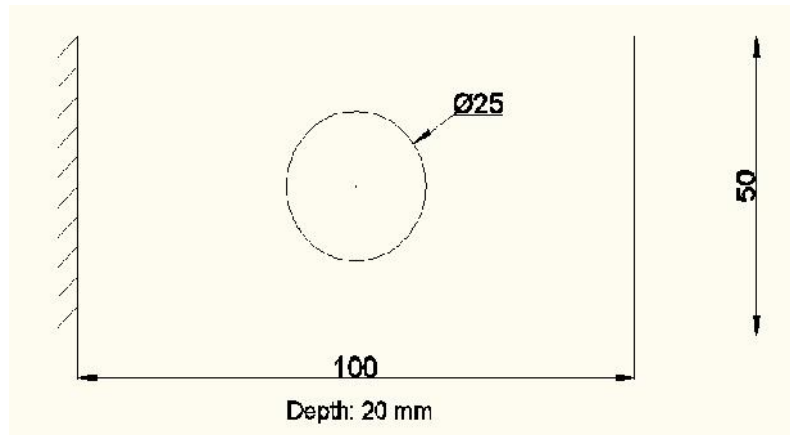
AIM:

To conduct the Stress Analysis on a Plate with a Circular Hole of given dimension using Ansys Workbench Software.

EQUIPMENT'S REQUIRED:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

DIAGRAM:



PROCEDURE:

1. File – Save – (Set New Folder & File Name) – Save
2. Pick & Drop Static Structural in Project Schematic Window.
3. Double Click Geometry – (Design Modeller window will open) – Set Length Unit as Millimeter – OK
4. Using Sketching under Tree Outline, draw the sketch in Graphics window as per the given diagram.
5. Extrude for desired depth and click generate.
6. Now go to main window and double click Model – It will open the Static Structural Analysis Window.
7. Under outline, right click Static Structural – Insert – Fixed Support – Set Fixed Support in required condition and then Set Force towards downward direction in the required point.
8. Solve the problem.
9. Add Total Deformation and Equivalent Stress under Solution and again solve the problem.
10. Save the File and also Image of final analysis result.

Plate with circular hole-Displacement

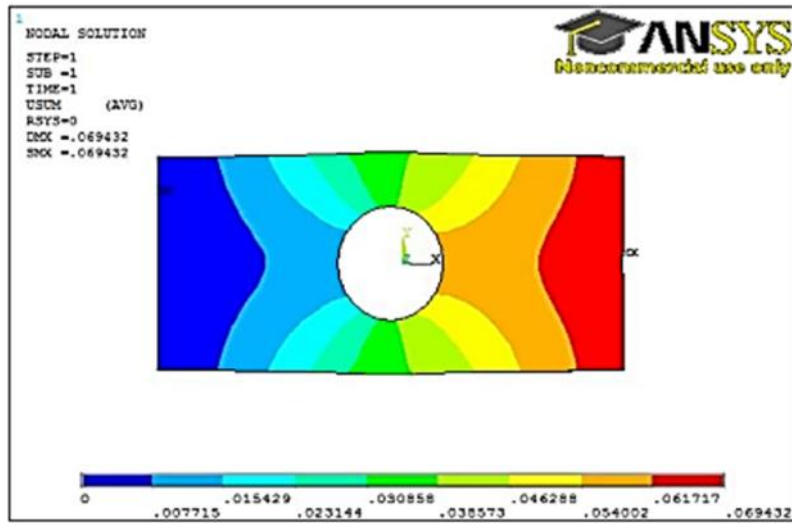
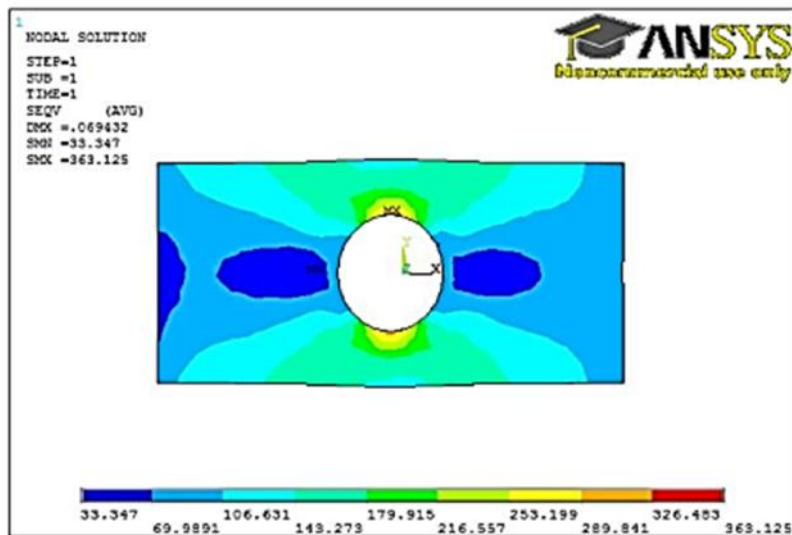


Plate with circular hole-Stress Distribution



RESULT:

Thus the Stress Analysis of Plate with a Circular Hole of given dimension was conducted using Ansys Workbench Software.

Total Deformation =

Equivalent Stress (Von Mises) =

Ex. No: 2	Stress Analysis of Rectangular L Bracket
Date:	

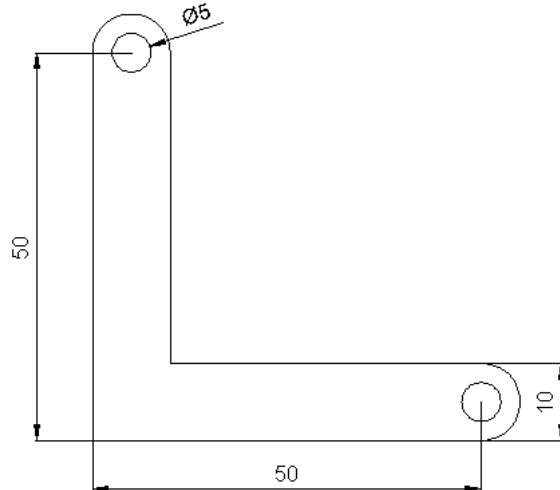
AIM:

To conduct the Stress Analysis of Rectangular L Bracket of given dimension using Ansys Workbench Software.

EQUIPMENT'S REQUIRED:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

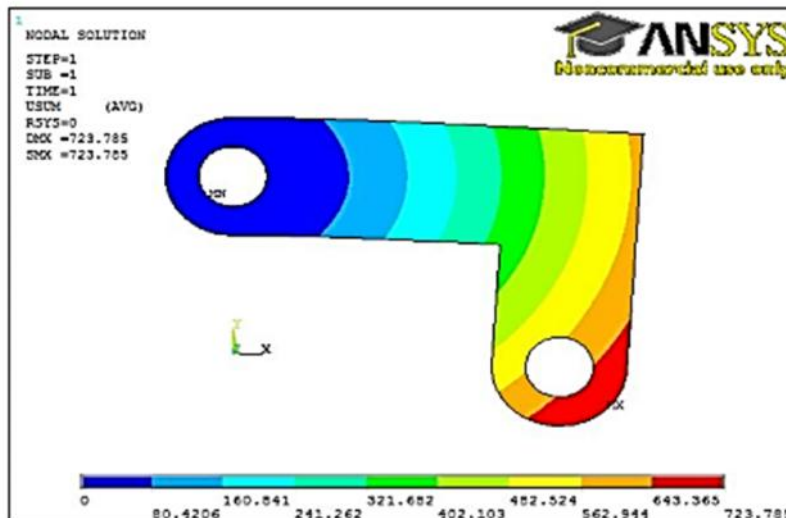
DIAGRAM:



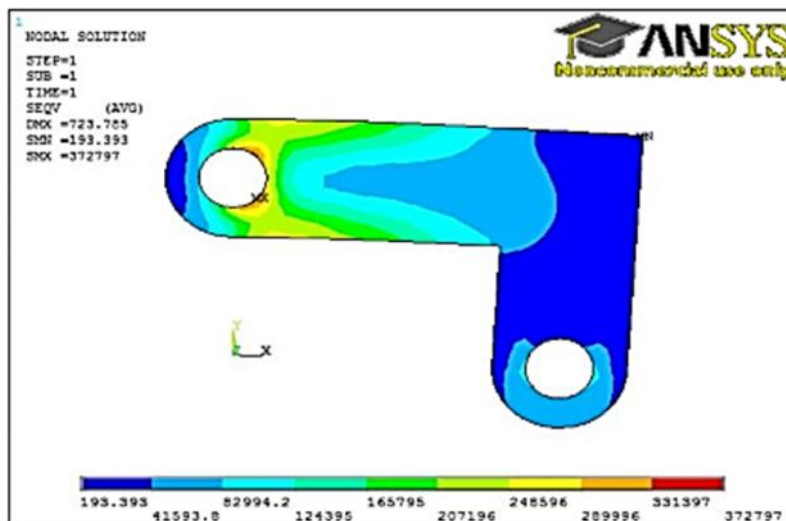
PROCEDURE:

1. File – Save – (Set New Folder & File Name) – Save
2. Pick & Drop Static Structural in Project Schematic Window.
3. Double Click Geometry – (Design Modeller window will open) – Set Length Unit as Millimeter – OK
4. Using Sketching under Tree Outline, draw the sketch in Graphics window as per the given diagram.
5. Extrude for desired depth and click generate.
6. Now go to main window and double click Model – It will open the Static Structural Analysis Window.
7. Under outline, right click Static Structural – Insert – Fixed Support – Set Fixed Support in required condition and then Set Force towards downward direction in the required point.
8. Solve the problem.
9. Add Total Deformation and Equivalent Stress under Solution and again solve the problem.
10. Save the File and also Image of final analysis result.

Rectangular L Bracket-Displacement



Rectangular L Bracket-Stress Distribution



RESULT:

Thus the Stress Analysis of Rectangular L Bracket of given dimension was conducted using Ansys Workbench Software.

Total Deformation =

Equivalent Stress (Von Mises) =

Ex. No: 3	Stress Analysis of an Axi-Symmetric Component
Date:	

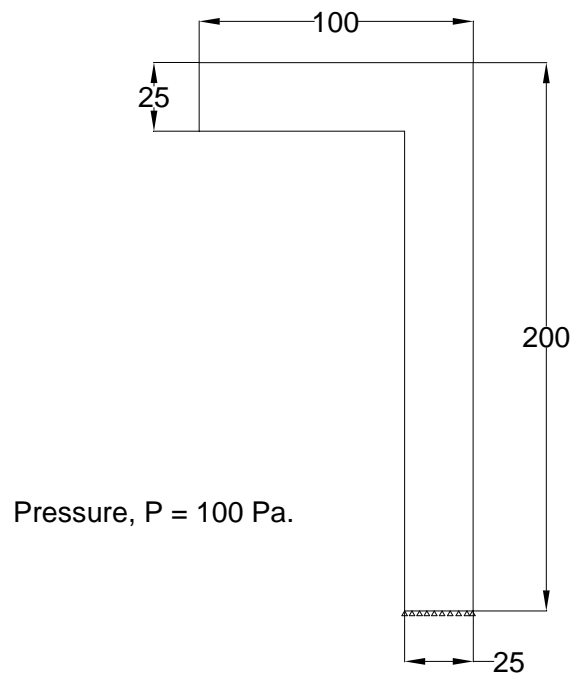
AIM:

To conduct the Stress Analysis on a Axi-Symmetric Component of given dimension using Ansys Software.

EQUIPMENT'S REQUIRED:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

DIAGRAM:



PROCEDURE:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as AXISYM)

File – Change Title – (Enter new title as Stress Analysis of an Axi-Symmetric Component)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Solid – Axisym 4node 272 – OK – Options – Set, Number of Node Planes K2 = 1

Axisymmetric – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Material – Exit
Preprocessor – Sections – Axis – Add – Add Axis Section With ID = 1 – OK – Section Name = 1 – V5 = 25 – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Areas – Rectangle – By 2 Corners – Set the values as (WP X = 75, WP Y = 0, Width = 25, Height = 200) – Apply – Set the values as (WP X = 0, WP Y = 175, Width = 75, Height = 25) – OK
Preprocessor – Modeling – Operate – Booleans – Add – Areas – (Pick the two rectangle) – OK
Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Object) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Constraints & Load)

Solution – Define Loads – Apply – Structural – Displacement – On Lines – (Pick the Bottom Horizontal Line of Width 25) – OK – ALL DOF – OK – On Lines – (Pick the Left Vertical Line Width 25) – OK – UX – OK
Solution – Define Loads – Apply – Structural – Pressure – On Lines – (Pick the Inner Vertical Line and Horizontal Line which inside the vessel) – OK – (Set Load PRES Value = 500) – OK

Step 6: Solution

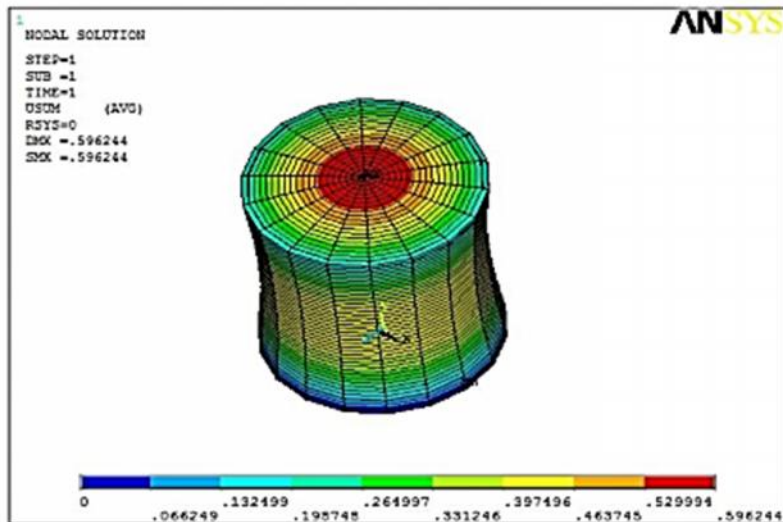
Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

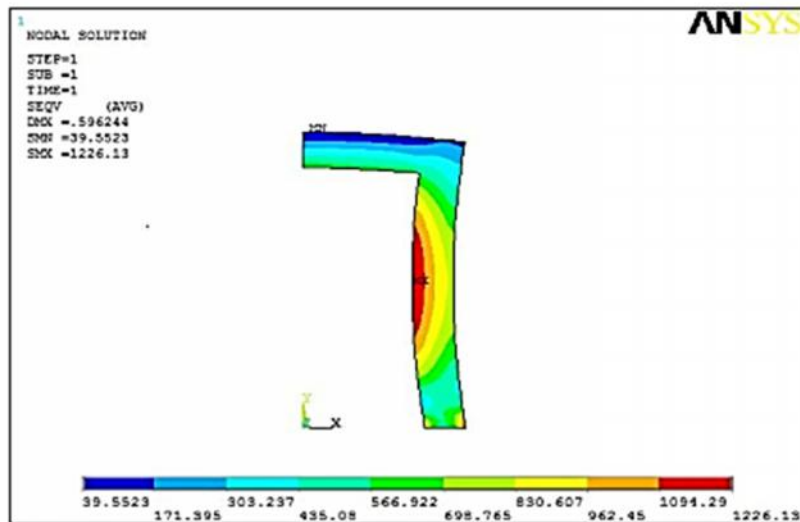
General Postproc – Plot Result – Contour Plot – Nodal Solu – Stress > von Mises Stress – OK
PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)
PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

Step 8: Finally note down the result include maximum deflection and maximum stress.

Axisymmetric component-Displacement



Axisymmetric component-Stress Distribution



RESULT:

Thus the Stress Analysis on a Axi-Symmetric Component of given dimension using Ansys Software.

Maximum Stress =

Maximum Displacement =

Ex. No: 4	Stress Analysis of Cantilever Beam
Date:	

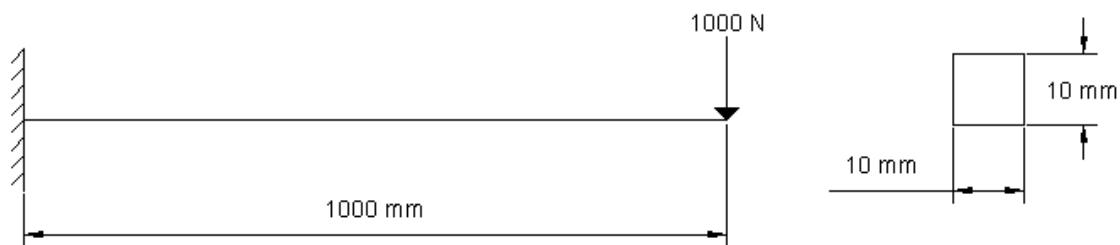
Aim:

To conduct the Stress Analysis for Cantilever Beam of given dimension using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴
Load	= 1000 N

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as CLBEAM)

File – Change Title – (Enter new title as Stress Analysis of Cantilever Beam)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Beam – 2 Node 188 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Beam) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Constraints & Load)

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side) – OK – ALL DOF – OK

Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – (Pick the Keypoint 2 which is in right side) – OK – (Choose Direction FY and Set Force/Moment Value = -100) – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

General Postproc – Plot Result – Contour Plot – Nodal Solu – Stress > von Mises Stress – OK

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

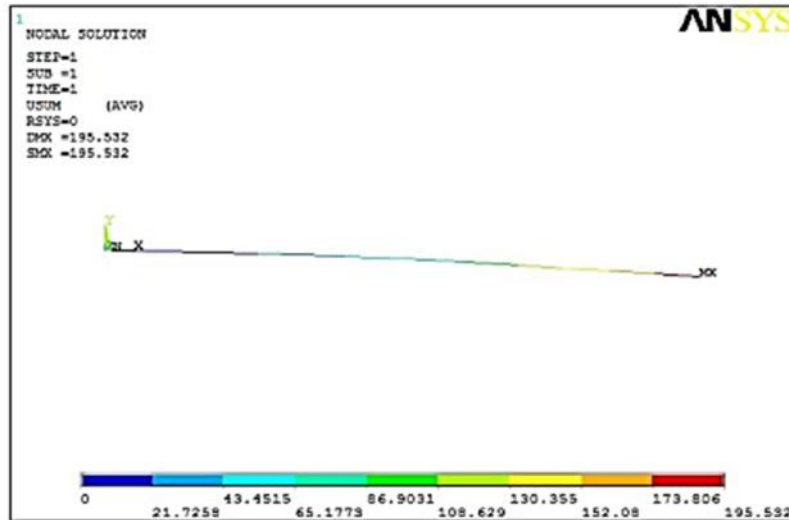
General Postproc – Plot Result – Contour Plot – Nodal Solu – DOF Solution > Displacement Vector sum – OK

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

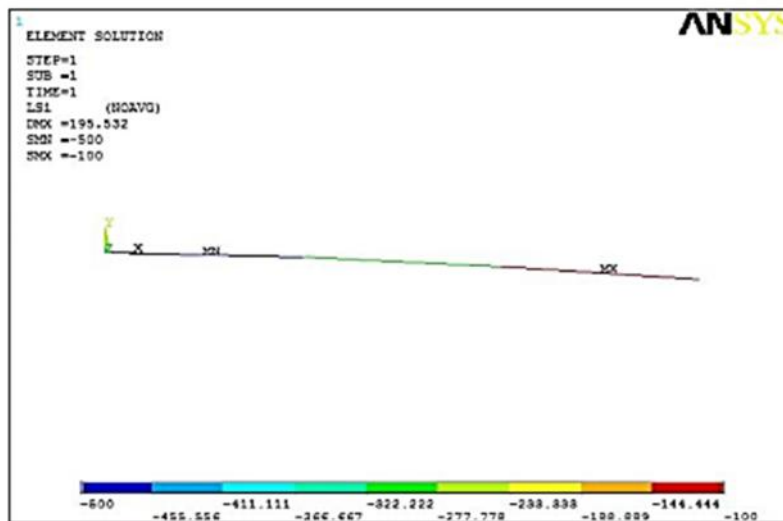
PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

Step 8: Finally note down the result include maximum deflection and maximum stress.

Stress analysis-Cantilever beam-Displacement



Stress analysis-Cantilever beam-Stress Distribution



RESULT:

Thus the Stress Analysis for Cantilever Beam of given dimension was conducted using Ansys Software.

Maximum Stress =

Maximum Displacement =

Ex. No: 5	Stress Analysis of Simply Supported Beam
Date:	

Aim:

To conduct the Stress Analysis for Simply Supported Beam of given dimension using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴
Load	= 1000 N

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as SSBEAM)

File – Change Title – (Enter new title as Stress Analysis of Simply Supported Beam)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Beam – 2 Node 188 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 500, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 3, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – (Pick Keypoint 2 & 3) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Entire Beam) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Constraints & Load)

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side and Keypoint 3 which is right side) – OK – ALL DOF – OK

Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – (Pick the Keypoint 2 which is in the middle) – OK – (Choose Direction FY and Set Force/Moment Value = -1000) – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

General Postproc – Plot Result – Contour Plot – Nodal Solu – Stress > von Mises Stress – OK

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

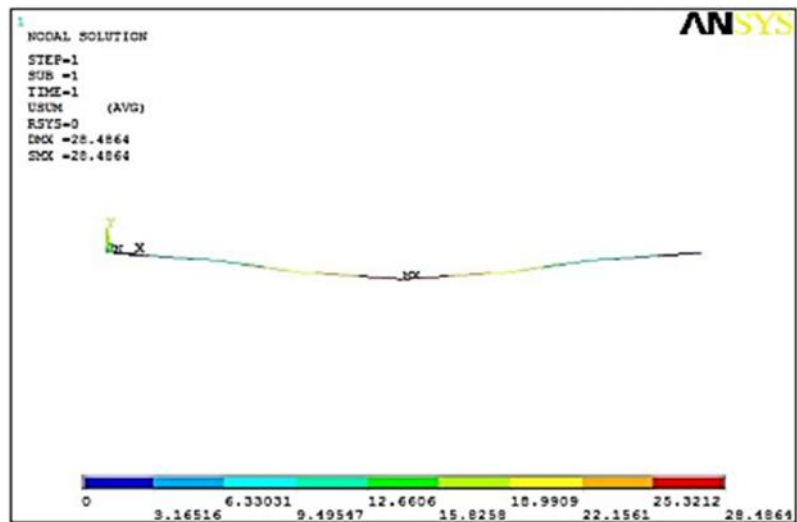
General Postproc – Plot Result – Contour Plot – Nodal Solu – DOF Solution > Displacement Vector sum – OK

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

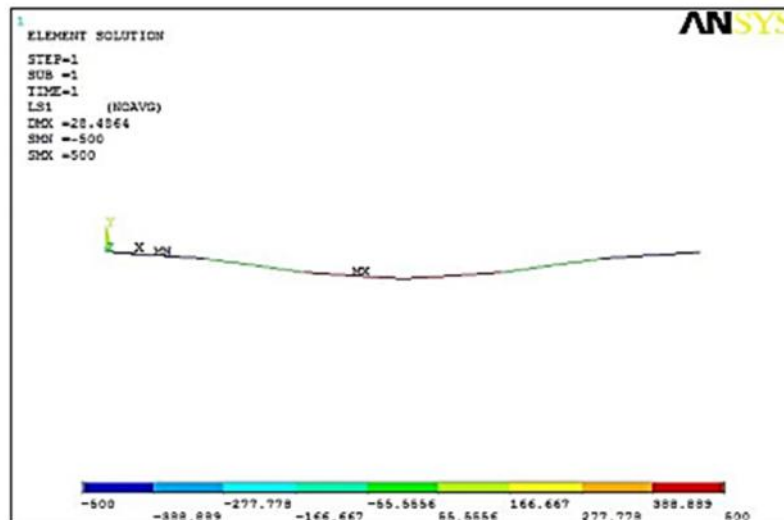
PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

Step 8: Finally note down the result include maximum deflection and maximum stress.

Stress analysis-Simply Supported Beam-Displacement



Stress analysis-Simply Supported Beam-Stress Distribution



RESULT:

Thus the Stress Analysis for Simply Supported Beam of given dimension was conducted using Ansys Software.

Maximum Stress =

Maximum Displacement =

Ex. No: 6	Stress Analysis of Fixed End Beam
Date:	

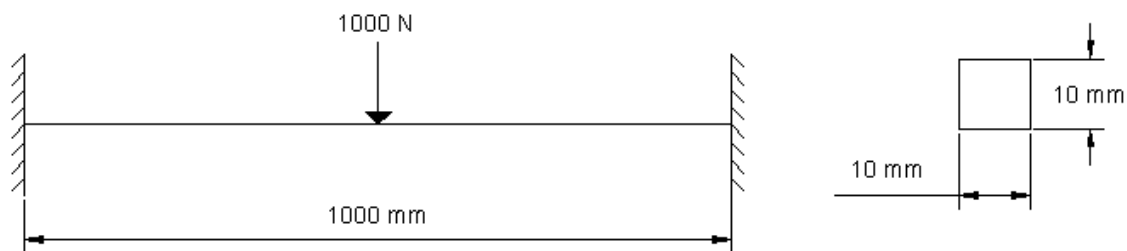
Aim:

To conduct the Stress Analysis for Fixed End Beam of given dimension using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴
Load	= 1000 N

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as FEBEAM)

File – Change Title – (Enter new title as Stress Analysis of Fixed End Beam)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Beam – 2 Node 188 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 500, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 3, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – (Pick Keypoint 2 & 3) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Entire Beam) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Constraints & Load)

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side and Keypoint 3 which is right side) – OK – ALL DOF – OK

Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – (Pick the Keypoint 2 which is in the middle) – OK – (Choose Direction FY and Set Force/Moment Value = -1000) – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

General Postproc – Plot Result – Contour Plot – Nodal Solu – Stress > von Mises Stress – OK

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

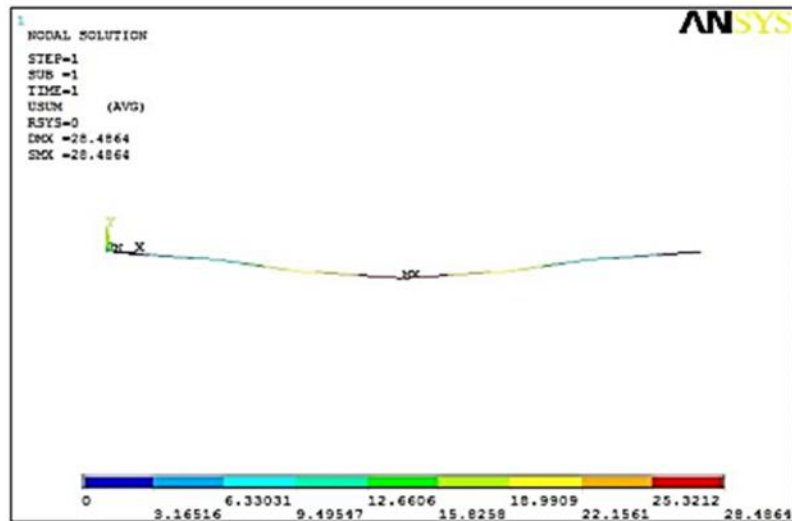
General Postproc – Plot Result – Contour Plot – Nodal Solu – DOF Solution > Displacement Vector sum – OK

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

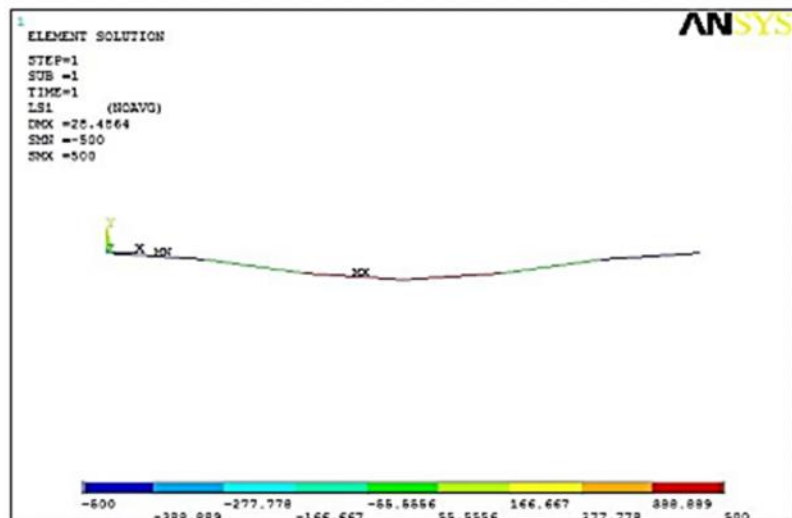
PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

Step 8: Finally note down the result include maximum deflection and maximum stress.

Stress analysis-Fixed Beam-Displacement



Stress analysis-Fixed Beam-Stress Distribution



RESULT:

Thus the Stress Analysis for Fixed End Beam of given dimension was conducted using Ansys Software.

Maximum Stress =

Maximum Displacement =

Ex. No: 7	Mode Frequency Analysis of 2D Component
Date:	

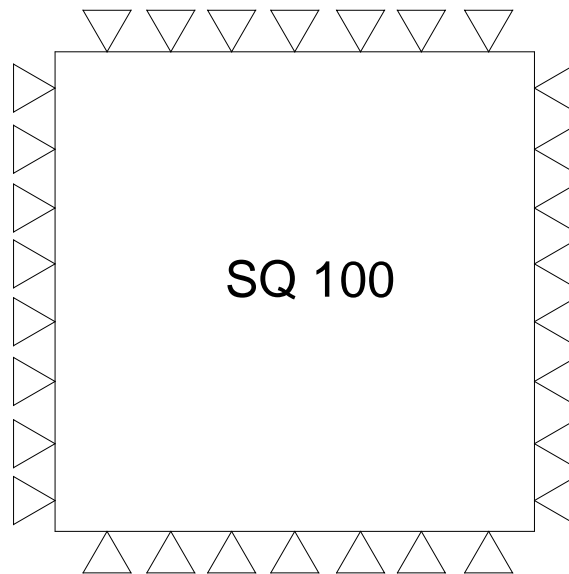
Aim:

To conduct the Mode Frequency Analysis for 2D Component of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as MODE)

File – Change Title – (Enter new title as Mode Frequency Analysis of 2D Component)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Solid – Quad 4 Node 182 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic –

Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Structural – Density – (DENS = 7830)

– OK – Material – Exit

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Areas – Rectangle – By 2 Corners – (Set Value: X = 0, Y = 0, Width = 100, Height = 100) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Object) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Analysis Type, Constraints & Load)

Solution – Analysis Type – New Analysis – Modal – OK

Solution – Analysis Type – Analysis Option – (No. of modes to extract = 5) – (NMODE No. of modes to expand = 5) – OK – (Block Lanczos Method Box Will open – (Start Freq = 0 & End Frequency = 100) – OK

Solution – Define Loads – Apply – Structural – Displacement – On Lines – (Pick all the Border Lines) – OK – ALL DOF – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

General Postproc – Result Summary – (Note down this result occur on the SET, LIST Command window in a tabular column as final result) – File – Close

Change the Diagram into Isometric View

General Postproc – Read Result – (Click First Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=1, FREQ, DMX.)

General Postproc – Read Result – (Click Next Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=2, FREQ, DMX. Now do the same above procedure for STEP=1, SUB=3, STEP=1, SUB=4, and STEP=1, SUB=5),

(For all the Above five result, take snapshot using this options.

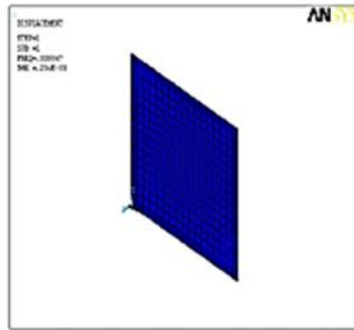
PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save))

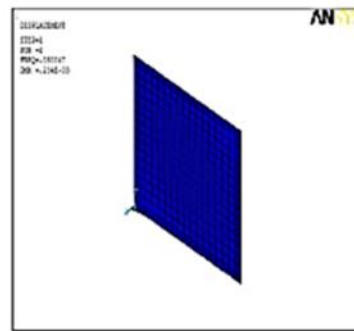
Step 8: Finally note down the result include maximum deflection.

Mode frequency analysis of 2D Component

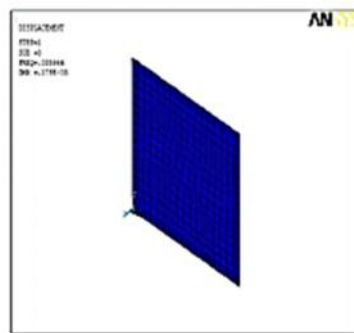
Mode frequency-1



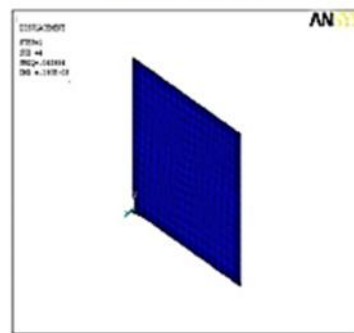
Mode frequency-2



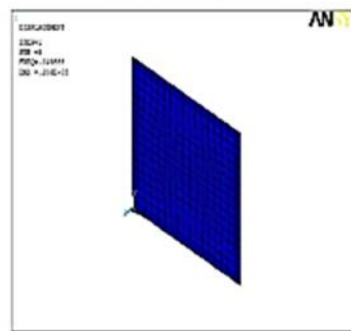
Mode frequency-3



Mode frequency-4



Mode frequency-5



RESULT:

Thus the Mode Frequency Analysis for 2D Component of given dimension was conducted using Ansys Software.

SET	TIME/FREQ	LOAD STEP	SUB STEP
1			
2			
3			
4			
5			

Ex. No: 8	Mode Frequency Analysis of Cantilever Beam
Date:	

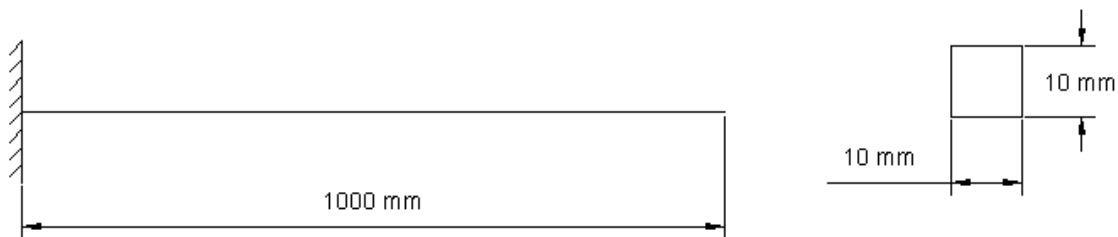
Aim:

To conduct the Mode Frequency Analysis for Cantilever Beam of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Density	= 7830 kg/m ³
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as MODECLBEAM)

File – Change Title – (Enter new title as Mode Frequency Analysis of Cantilever Beam)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Beam – 2 Node 188 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Structural – Density – (DENS = 0.00783) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Beam) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Analysis Type, Constraints & Load)

Solution – Analysis Type – New Analysis – Modal – OK

Solution – Analysis Type – Analysis Option – (Pick Subspace) – (No. of modes to extract = 5) – (NMODE No. of modes to expand = 5) – OK – (Block Lanczos Method Box Will open – (Start Freq = 0 & End Frequency = 100) – OK

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side) – OK – ALL DOF – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

General Postproc – Result Summary – (Note down this result occur on the SET, LIST Command window in a tabular column as final result) – File – Close

General Postproc – Read Result – (Click First Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=1, FREQ, DMX.)

General Postproc – Read Result – (Click Next Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=2, FREQ, DMX. Now do the same above procedure for STEP=1, SUB=3, STEP=1, SUB=4, and STEP=1, SUB=5),

(For all the Above result, take snapshot using this options.

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save))

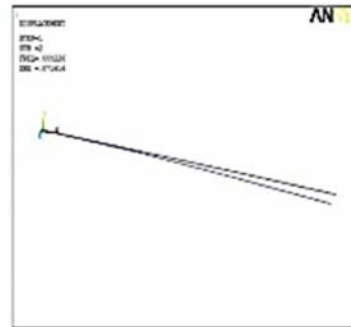
Step 8: Finally note down the result include maximum deflection and maximum stress.

Mode frequency analysis of Cantilever Beam

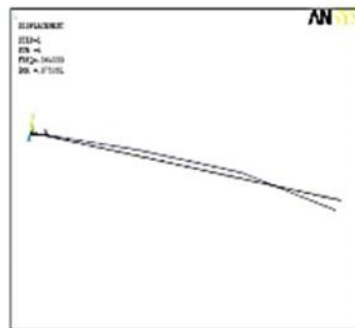
Mode frequency-1



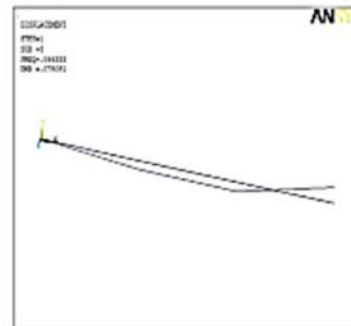
Mode frequency-2



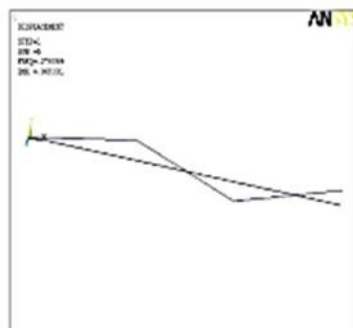
Mode frequency-3



Mode frequency-4



Mode frequency-5



RESULT:

Thus the Mode Frequency Analysis for Cantilever Beam of given dimension was conducted using Ansys Software.

SET	TIME/FREQ	LOAD STEP	SUB STEP
1			
2			
3			
4			

Ex. No: 9	Mode Frequency Analysis of Simply Supported Beam
Date:	

Aim:

To conduct the Mode Frequency Analysis for Simply Supported Beam of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Density	= 7830 kg/m ³
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as MODESSBEAM)

File – Change Title – (Enter new title as Mode Frequency Analysis of Simply Supported Beam)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Beam – 2 Node 188 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Structural – Density – (DENS = 0.00783) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 500, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 3, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – (Pick Keypoint 2 & 3) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Entire Beam) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Analysis Type, Constraints & Load)

Solution – Analysis Type – New Analysis – Modal – OK

Solution – Analysis Type – Analysis Option – (Pick Subspace) – (No. of modes to extract = 5) – (NMODE No. of modes to expand = 5) – OK – (Block Lanczos Method Box Will open – (Start Freq = 0 & End Frequency = 100) – OK

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side and Keypoint 3 which is right side) – OK – UY – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

General Postproc – Result Summary – (Note down this result occur on the SET, LIST Command window in a tabular column as final result) – File – Close

General Postproc – Read Result – (Click First Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=1, FREQ, DMX.)

General Postproc – Read Result – (Click Next Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=2, FREQ, DMX. Now do the same above procedure for STEP=1, SUB=3, STEP=1, SUB=4, and STEP=1, SUB=5),

(For all the Above result, take snapshot using this options.

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save))

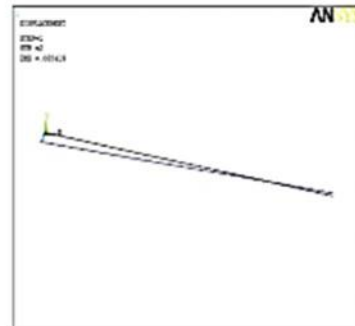
Step 8: Finally note down the result include maximum deflection.

Mode frequency analysis of simply supported beam

Mode frequency-1



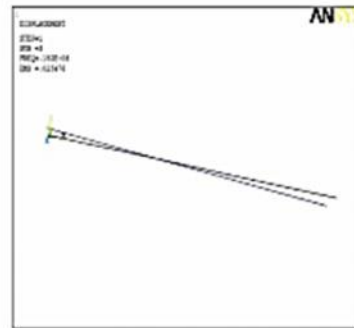
Mode frequency-2



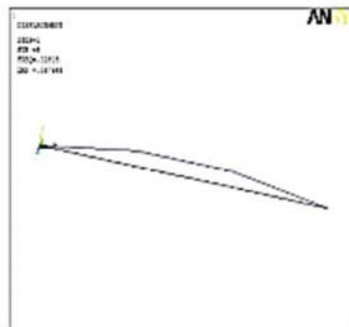
Mode frequency-3



Mode frequency-4



Mode frequency-5



RESULT:

Thus the Mode Frequency Analysis for Cantilever Beam of given dimension was conducted using Ansys Software.

SET	TIME/FREQ	LOAD STEP	SUB STEP
1			
2			
3			
4			

Ex. No: 10	Mode Frequency Analysis of Fixed End Beam
Date:	

Aim:

To conduct the Mode Frequency Analysis for Fixed End Beam of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Density	= 7830 kg/m ³
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as MODEFEBEAM)

File – Change Title – (Enter new title as Mode Frequency Analysis of Fixed End Beam)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Beam – 2 Node 188 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Structural – Density – (DENS = 0.00783) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 500, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 3, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – (Pick Keypoint 2 & 3) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Entire Beam) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Analysis Type, Constraints & Load)

Solution – Analysis Type – New Analysis – Modal – OK

Solution – Analysis Type – Analysis Option – (Pick Subspace) – (No. of modes to extract = 5) – (NMODE No. of modes to expand = 5) – OK – (Block Lanczos Method Box Will open – (Start Freq = 0 & End Frequency = 100) – OK

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side and Keypoint 3 which is right side) – OK – ALL DOF – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

General Postproc – Result Summary – (Note down this result occur on the SET, LIST Command window in a tabular column as final result) – File – Close

General Postproc – Read Result – (Click First Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=1, FREQ, DMX.)

General Postproc – Read Result – (Click Next Set)

General Postproc – Plot Results – Deformed Shape – (Pick Def + undeformed) – OK
(Result will be displayed as STEP=1, SUB=2, FREQ, DMX. Now do the same above procedure for STEP=1, SUB=3, STEP=1, SUB=4, and STEP=1, SUB=5),

(For all the Above result, take snapshot using this options.

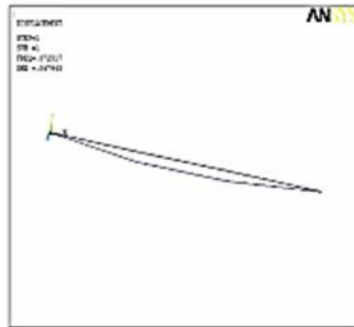
PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save))

Step 8: Finally note down the result include maximum deflection.

Mode frequency analysis of fixed beam

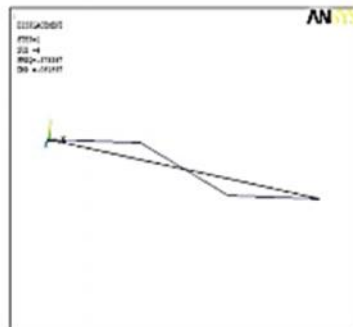
Mode frequency-1



Mode frequency-2



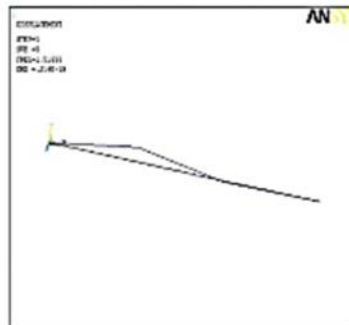
Mode frequency-3



Mode frequency-4



Mode frequency-5



RESULT:

Thus the Mode Frequency Analysis for Cantilever Beam of given dimension was conducted using Ansys Software.

SET	TIME/FREQ	LOAD STEP	SUB STEP
1			
2			
3			
4			

Ex. No: 11	Harmonic Analysis of a 2D Component
Date:	

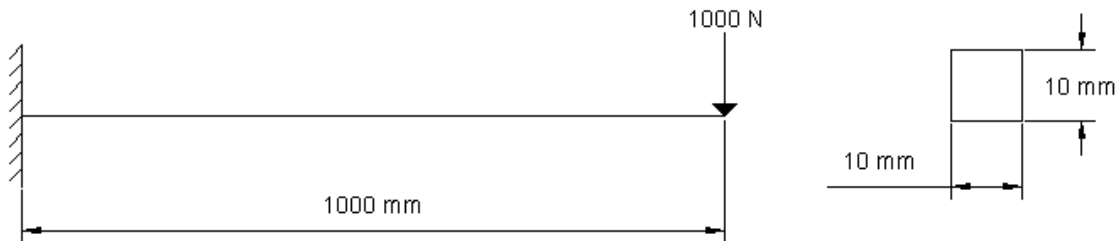
Aim:

To conduct the Harmonic Analysis on 2D Component of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Density	= 0.00783 kg/mm ³
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴
Load	= 1000 N
Frequency Range	= 0 – 100 Hz.

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as HARMONIC)

File – Change Title – (Enter new title as Harmonic Analysis of a 2D Component)

Step 2: Ansys Main Menu – Preference

Preference – Structural - OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Beam – 2 Node 188 – OK – Close

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Structural – Density – (DENS = 0.00783) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Beam) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Constraints & Load)

Solution – Analysis Type – New Analysis – Harmonic – OK

Solution – Analysis Options – OK – OK

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side) – OK – ALL DOF – OK

Solution – Define Loads – Apply – Structural – Force/Moment – On Keypoints – (Pick the Keypoint 2 which is in right side) – OK – (Choose Direction FY and Set Force/Moment Value = -1000) – OK

Solution – Load Step Opts – Time/Frequenc – Freq and Substps – (Harmonic freq range = 0 to 100) – (Number of Substeps = 20) – (Pick Stepped) – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: Post Processor (Viewing the Result)

TimeHistPostpro – (Time History Variables window will appear) – (Pick the green '+' sign in the upper left corner) – (In Add Time-History Variable Window, choose Nodal Solution > DOF Solution > Y-Component of displacement) – (Pick the Keypoint 2 which is in right side where the load was applied)

In Time History Variables window, click the 'List' button, 3 buttons to the right of '+'
PRVAR Command window will appear. Close the window.

In Time History Variables window, click the 'Plot' button, 2 buttons to the right of '+'
(Now the graph should be plotted in the Ansys Main Window)

Close the Time History Variables window.

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

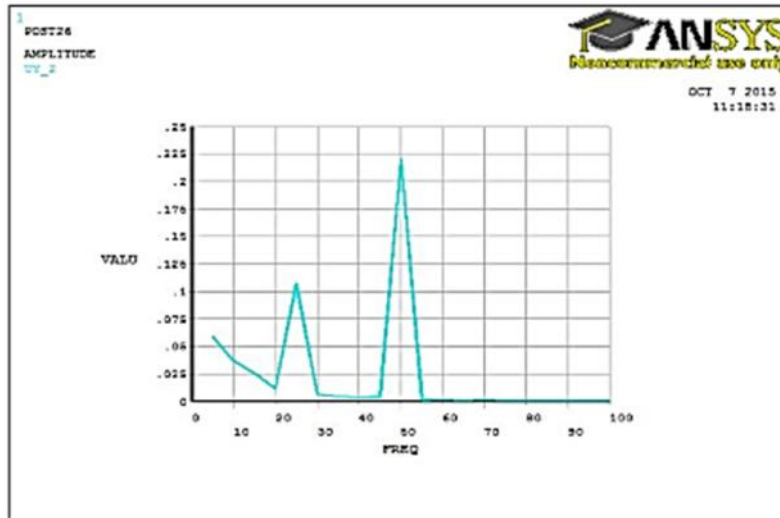
Utility Menu – PlotCtrls – Style – Graphs – Modify Axes – (Axes Modifications for Graph Plots window will appear) – (Change the Value of 'LOGY Y-axis scale' to 'Logarithmic' – OK

Utility Menu – Plot – Replot

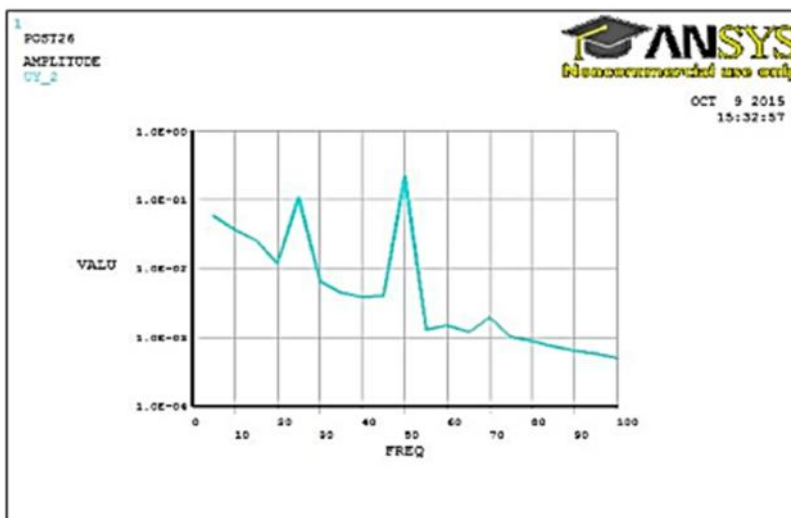
PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

Frequency Vs Amplitude-Linear scale



Frequency Vs Amplitude-Logarithmic scale



RESULT:

Thus the Harmonic Analysis of a 2D component of given dimension was conducted using Ansys Software.

Ex. No: 12	Thermal Stress Analysis of a 2D Component
Date:	

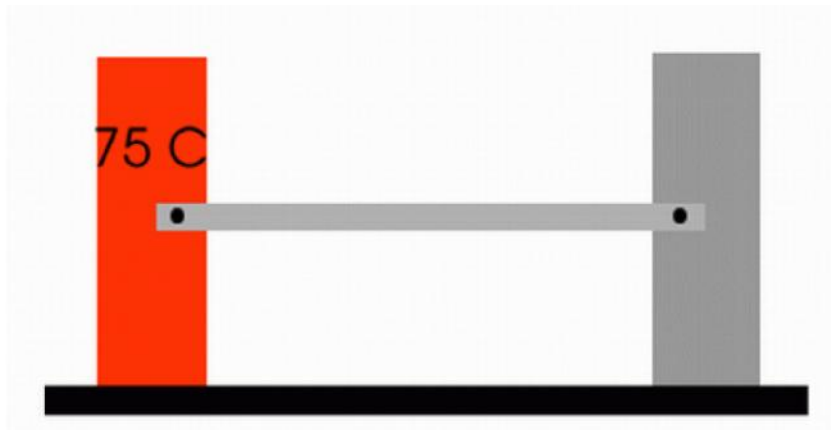
Aim:

To conduct the Thermal Stress Analysis on 2D Component of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Temperature Change	= 75°C = 348K
Thermal Conductivity	= 60.5 W/m°K
Thermal Expansion Coefficient	= 12E-6 /K
Young's Modulus (EX)	= 2E5 N/mm ²
Poisson's Ration (PRXY)	= 0.3
Cross Section Area	= 100 mm ²
Moment of Inertia (IZZ)	= 833.33 mm ⁴

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as THERMALSTRESS)

File – Change Title – (Enter new title as Thermal Stress Analysis of a 2D Component)

Step 2: Ansys Main Menu – Preference

Preference – Structural & Thermal – OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)

Add – Thermal Mass – Link – 3D Conduction 33 – OK – Close

Preprocessor – Material Props – Material Models – Thermal – Conductivity – Isotropic – (Enter KXX = 60) – OK – Material – Exit

Preprocessor – Sections – Beam – Common Sections – (Set Value: B = 10, H = 10) – OK

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Keypoints – In Active CS – (Set Value: Keypoint number = 1, X = 0, Y = 0, Z = 0) – Apply – (Set Value: Keypoint number = 2, X = 1000, Y = 0, Z = 0) – OK

Preprocessor – Modeling – Create – Lines – Lines – Straight Line – (Pick Keypoint 1 & 2) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Object) – OK – (Close the MeshTool Dialog Box)

Step 5: Preprocessor (Write Environment)

Preprocessor – Physics – Environment – Write – (In the window that appear, enter TITLE = THERMAL) – OK

Preprocessor – Physics – Environment – Clear – OK

Step 6: Preprocessor (Structural Environment – Define Physical Properties)

Preprocessor – Element Type – Switch Elem Type – (Select ‘Thermal to Struc’) – OK

Preprocessor – Material Props – Material Models – Structural – Linear – Elastic – Isotropic – (Enter EX = 2E5 & PRXY = 0.3) – OK – Structural – Thermal Expansion – Secant Coefficient – Isotropic – (Enter ALPX = 12E-6) – OK – Material – Exit

Preprocessor – Physics – Environment – Write – (In the window that appear, enter TITLE = STRUCT) – OK

Step 7: Solution

Solution – Analysis Type – New Analysis – Steady-State – OK

Preprocessor – Physics – Environment – Read – (Select ‘THERMAL’) – OK

Solution – Define Loads – Apply – Thermal – Temperature – On Keypoints – (Set the temperature of Keypoint 1, the left most point to 348 Kelvin)

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window) – Main Menu – Finish

Preprocessor – Physics – Environment – Read – (Select ‘STRUCT’) – OK

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 1 which is in left side) – OK – ALL DOF – OK

Solution – Define Loads – Apply – Structural – Displacement – On Keypoints – (Pick the Keypoint 2 which is in right side) – OK – UX – OK

Solution – Define Loads – Apply – Structural – Temperature – From ThermAnaly – (Enter the Name of result file as *file.rth*) – OK

Preprocessor – Loads – Define Loads – Settings – Reference Temp – (Set Reference Temperature to 273) – OK

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

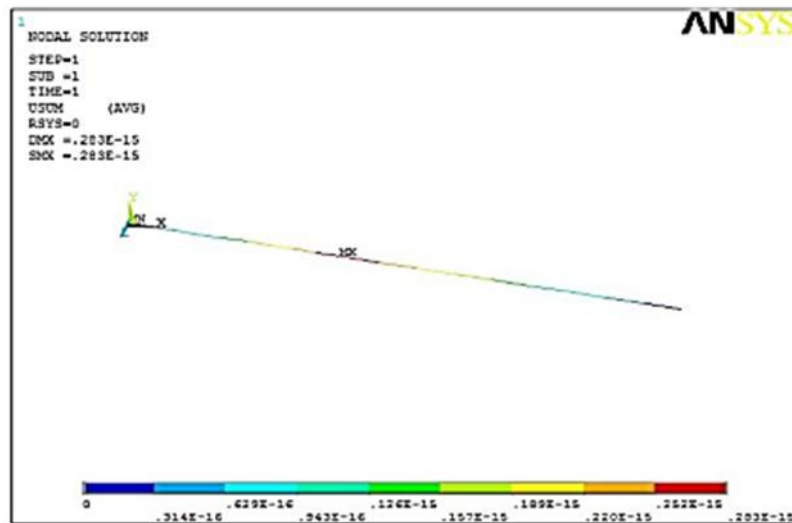
Step 8: General Post Processor

General Postproc – Element Table – Define Table – Add – (User label for item = COMPSTR) – (In Results data item, pick ‘By Sequence num> LS > LS, 1) – OK

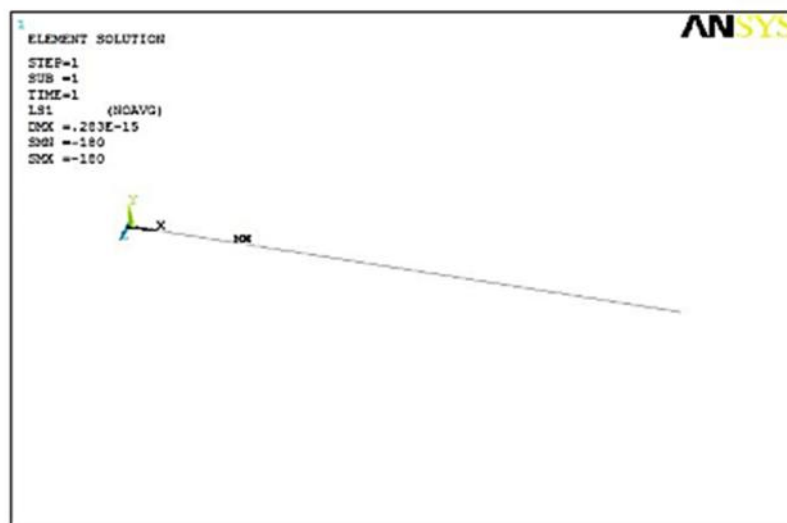
General Postproc – Element Table – List Elem Table – COMPSTR – OK

PRETAB Command window will appear. Note the stress in each element: -0.180e9 Pa or 180MPa in compression as expected.

Thermal stress analysis-Displacement



Thermal stress analysis-Stress Distribution



RESULT:

Thus the Thermal Stress Analysis of a 2D Component for given dimension was conducted using Ansys Software.

Ex. No: 13

Date:

Conductive Heat Transfer Analysis of a 2D Component

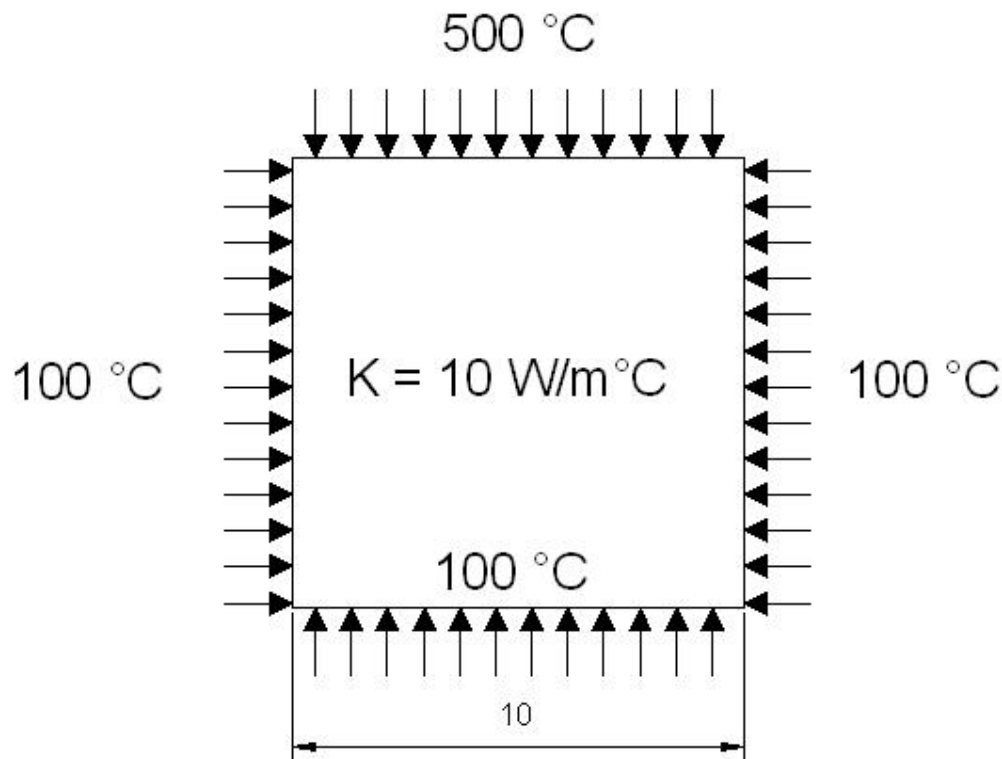
Aim:

To conduct the Conductive Heat Transfer Analysis on 2D Component of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Thermal Conductivity (KXX) = $10 \text{ W/m}^\circ\text{C}$

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)

File – Change Jobname – (Enter new jobname as CONDUCTIVE)

File – Change Title – (Enter new title as Conductive Heat Transfer Analysis of a 2D Component)

Step 2: Ansys Main Menu – Preference

Preference – Thermal – OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)
Add – Solid – Quad 4node 55 – OK – Close
Preprocessor – Material Props – Material Models – Thermal – Conductivity – Isotropic –
(EnterKXX = 10) – OK– Material – Exit

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Areas – Rectangle – By 2 Corners - (Set Value: WP X
= 0, WP Y = 0, Width = 10, Height = 10) – OK
Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select
the Object) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Temperature)

Solution – Analysis Type – New Analysis – Steady-State – OK
Solution – Define Loads – Apply – Thermal – Temperature – On Nodes – (Pick the Box
Option and pick Left, Bottom and Right side edge nodes) – OK – TEMP – (Load TEMP
Value = 100) – OK
Solution – Define Loads – Apply – Thermal – Temperature – On Nodes – (Pick the Box
Option and pick Top side edge nodes) – OK – TEMP – (Load TEMP Value = 500) – OK

Step 6: Solution

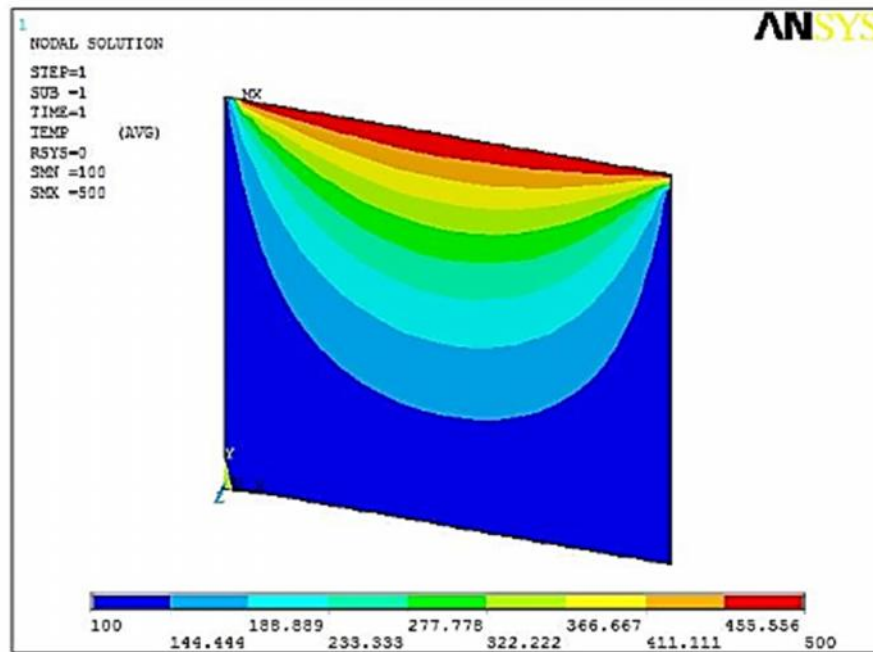
Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close
the Status Command Window)

Step 7: General Post Processor

General Postproc – Plot Result – Contour Plot – Nodal Solu – DOF Solution –
Temperature – OK
PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name
and Save)
PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

Step 8: Finally note down the result include maximum deflection and maximum stress.

Temperature Distribution-Conduction 2D Component



RESULT:

Thus the Conductive Heat Transfer Analysis of a 2D Component for given dimension was conducted using Ansys Software.

Ex. No: 14	Convective Heat Transfer Analysis of a 2D Component
Date:	

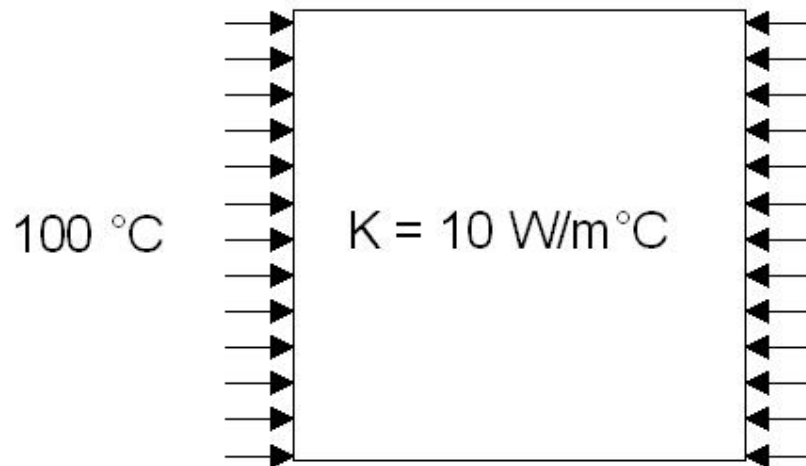
Aim:

To conduct the Convective Heat Transfer Analysis on 2D Component of given dimensions using Ansys Software.

Equipment's Required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Diagram:



Thermal Conductivity (KXX)	= 10 W/m°C
Film Coefficient	= 10
Bulk Temperature	= 20

Procedure:

Step 1: Ansys Utility Menu

File – Change Directory – (Set Directory to save all the working files)
 File – Change Jobname – (Enter new jobname as CONVECTIVE)
 File – Change Title – (Enter new title as Convective Heat Transfer Analysis of a 2D Component)

Step 2: Ansys Main Menu – Preference

Preference – Thermal – OK

Step 3: Preprocessor (Constant Values)

Preprocessor – Element Type – Add/Edit/Delete – (Element Type Dialog Box will open)
 Add – Solid – Quad 8node 278 – OK – Close
 Preprocessor – Material Props – Material Models – Thermal – Conductivity – Isotropic – (Enter KXX = 10) – OK – Material – Exit

Step 4: Preprocessor (Modelling)

Preprocessor – Modeling – Create – Volumes – Block – By 2 Corners & Z - (Set Value: WP X = 0, WP Y = 0, Width = 1, Height = 1, Depth = 2) – OK

Preprocessor – Meshing – MeshTool – (Pick Smart Size and set to Fine) – Mesh – (Select the Object) – OK – (Close the MeshTool Dialog Box)

Step 5: Solution (Temperature)

Solution – Define Loads – Apply – Thermal – Temperature – On Areas – (Pick the Left Rectangular Face of the Object) – OK – TEMP – (Load TEMP Value = 200) – OK

Solution – Define Loads – Apply – Thermal – Convection – On Areas - (Pick the Right Rectangular Face of the Object) – (Film Coefficient = 10, Bulk Temperature = 150) – OK

Step 6: Solution

Solution – Solve – Current LS – OK – (A Dialog box occur as Solution is Done) – (Close the Status Command Window)

Step 7: General Post Processor

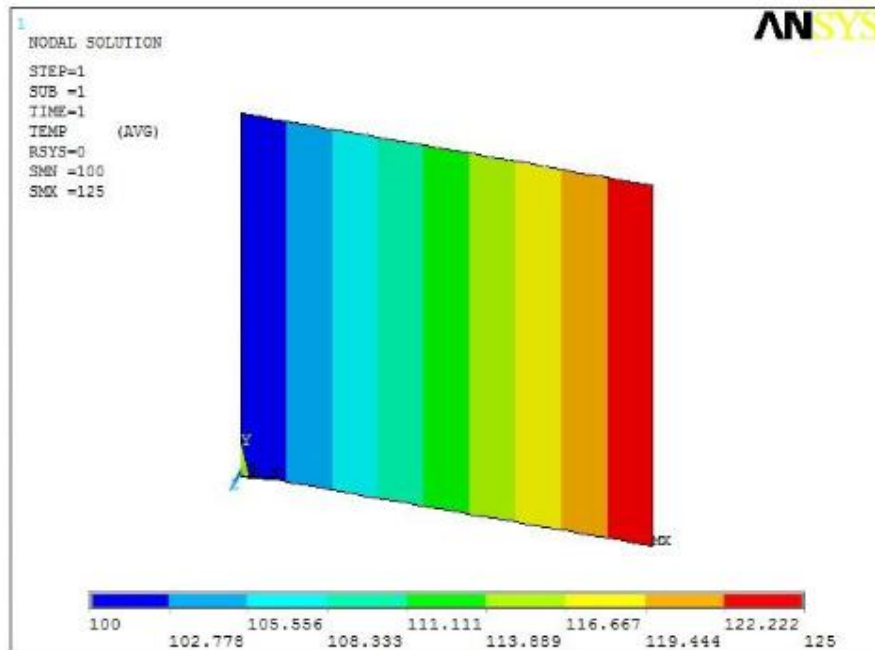
General Postproc – Plot Result – Contour Plot – Nodal Solu – DOF Solution – Nodal Temperature – OK

PlotCtrls – Capture Image – (Snapshot page will open) File – Save As – (Enter File Name and Save)

PlotCtrls – Write Metafile – Invert White/Black - (Enter File Name and Save)

Step 8: Finally note down the result include maximum and minimum temperature.

Temperature Distribution-Convection 2D Component



RESULT:

Thus the Convective Heat Transfer Analysis of a 2D Component for given dimension was conducted using Ansys Software.

Ex. No: 15	Simulation of cam follower mechanism
Date:	

Aim:

To simulate the cam follower mechanism by using MATLAB Software.

Equipment's required:

- Pentium Dual Core Processor
- Minimum 40GB HDD and 1GB RAM
- Ansys Version 14 With Workbench
- Laser Printer

Procedure:

- Open MATLAB – File – New – Script (Editor Window will open) – (Now type the program)
- (After complete typing the program) – Debug – Save file and Run – (Save file in the proper folder) – (Now the Result will appear in Figure 1 Window)
- File – Save – (Save the figure)
- Edit – Copy Figure – (Open MS Office and paste the figure) – Save the Result

Result:

Thus the Simulation of cam follower mechanism was conducted by using MATLAB Software.