SRM VALLIAMMAI ENGINEERING COLLEGE

(An Autonomous Institution)

SRM NAGAR, KATTANKULATHUR – 603 203

ME3745- SIMULATION AND ANALYSIS LABORATORY MANUAL

(7th Semester B.E Mechanical Engineering Students for the Academic Year 2026-2027)

R2023

Prepared By

Faculty Members

Department of Mechanical Engineering

(Private Circulation Only)

SRM VALLIAMMAI ENGINEERING COLLEGE, SRM NAGAR,

KATTANKULATHUR-603203

Department of Mechanical Engineering

Instructions to the students

The following instructions must be followed by the students in their laboratory classes.

1. Students are expected to be punctual to the lab classes. If they are late, they will be considered absent for that particular session.
2. Students should strictly maintain the dress code (i.e.) they have to wear the lab uniform with black shoes.
3. Students must bring their observation note, record note (completed with previous experiment) and the calculator to every lab class without fail.
4. Students are advised to come with full preparation for their lab sessions by
    a. Reading the detailed procedure of the experiment from the laboratory manual.
    b. Completion of observation note book (i.e.) Aim, Apparatus required, Formula (with description), least count calculation, diagrams and the tabular column should be written in the observation note before entering into the laboratory.
5. Data enter in the observation note book must be by pen only.
6. Students must get attestations immediately for their observed readings.
7. Students are advised to get their results evaluated in the observation note book on the same day of that experiment.
8. Class assessment marks for each experiment is based only on their performance in the laboratory.
9. Record note has to be completed then and there and get corrected when the students are coming or the next lab class.
10. Students must strictly maintain silence during lab classes.
11. If any of the students is absent for the lab class for genuine reasons, he/she will be permitted to do the experiment during the repetition class only.
12. Students are advised to perform their experiments under safety care.
13. If any student is found causing damage to the lab equipments, he/she shall replace the same with a new.

Department of Mechanical Engineering

VII Semester Mechanical Engineering

ME3745- SIMULATION AND ANALYSIS LABORATORY

SYLLABUS

Course Objective: The main learning objective of this course is to prepare the students for:

1. Analyzing the force and stress in mechanical components.
2. Analyzing deflection in mechanical components.
3. Analyzing thermal stress of mechanical components.
4. Analyzing heat transfer in mechanical components..
5. Analyzing the vibration of mechanical components.

LIST OF EXPERIMENTS

1. Study of Basics in ANSYS
2. Stress analysis of a plate with a circular hole
3. Stress analysis of rectangular L bracket
4. Stress analysis of cantilever beam
5. Stress analysis of simply supported beam
6. Stress analysis of fixed beam
7. Stress analysis of an axi-symmetric component
8. Thermal stress analysis of a 2D component
9. Conductive heat transfer analysis of a 2D component
10. Convective heat transfer analysis of a 2D component
11. Mode frequency analysis of cantilever beam
12. Mode frequency analysis of simply supported beam
13. Harmonic analysis of a 2D component
14. Stress analysis of a truss

Course Outcomes: Upon completion of this course, the students will be able to:

1. Find out the effect of force and impact of stress on the mechanical components.
2. Calculate the deflection occuring on the mechanical components.
3. Get a detailed understanding of the thermal stress creation and its mechanism of spreading in mechanical components.
4. Gain knowledge regarding the mechanism of heat transfer in mechanical components.
5. Find out the vibration effects on mechanical components.

Department of Mechanical Engineering

VII Semester Mechanical Engineering
ME3745- SIMULATION AND ANALYSIS LABORATORY

Course Information Sheet

COURSE DESCRIPTION:

ME3745 Simulation And Analysis Laboratory is the seventh semester practical subject covering stress, thermal, harmonic analysis of mechanical components. This laboratory designed for fourth year Mechanical students with two credits.

PROGRAM SPECIFIC OUTCOMES (PSO):

1. **PSO1-** Ability to analyze and solve problems related to mechanical engineering
2. **PSO2-** Can be able to apply thermal, fluid, design, electrical & electronics engineering principles to mechanical engineering applications.
3. **PSO3-** Would be able to apply the tools and techniques of quality management of manufacturing service processes.
4. **PSO4-** Ability to use the concept of process planning and cost estimation to manufacture different products.

CONTRIBUTION OF COURSE TO MEETING THE REQUIREMENTS:

Mathematics and Basic Sciences	Engineering Topics	General Education	Engineering Design
✔	-	-	-

PROGRAM OUTCOMES (PO):

PO	Description	
A	Graduates will demonstrate basic knowledge in mathematics, science and engineering.	✔
B	Graduates will demonstrate the ability to design and conduct experiments, interpret and analyze data, and report results.	✔
C	Graduates will demonstrate the ability to design a mechanical system or a thermal	-

	system or a process that meets desired specifications and requirements.	
D	Graduates will demonstrate the ability to function on engineering and science laboratory teams, as well as on multidisciplinary design teams.	✔
E	Graduates will demonstrate the ability to identify, formulate and solve mechanical engineering problems.	-
F	Graduates will demonstrate an understanding of their professional and ethical responsibilities.	-
G	Graduates will be able to communicate effectively in both verbal and written forms.	✔
H	Graduates will have the confidence to apply engineering solutions in global and societal contexts.	-
I	Graduates should be capable of self-education and clearly understand the value of lifelong learning.	✔
J	Graduates will be broadly educated and will have an understanding of the impact of engineering on society and demonstrate awareness of contemporary issues.	-
K	Graduates will be familiar with modern engineering software tools and equipment to analyze mechanical engineering problems.	✔
L	Competent enough to succeed in National level competitive examinations in the field of Mechanical Engineering.	-
M	Ability to be a successful Entrepreneur in the fields of Mechanical, Automobile, Mechatronics Engineering and Industrial Automation.	✔

co	**PO**												**PSO**			
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**1**	**2**	**3**	**4**
1	3	3			3			3				3	3			
2	3	3			3			3				3	3			
3	3	3			3			3				3	3			
4	3	3			3			3				3	3			
5	3	3			3			3				3	3			

ME 3745 - SIMULATION AND ANALYSIS LABORATORY
CONTENTS

Sl. No.	Name of the experiments	Page No.
	ANALYSIS	
1	Study of Basics in ANSYS	8
2	Stress analysis of a plate with a circular hole	14
3	Stress analysis of rectangular L bracket	16
4	Stress analysis of cantilever beam	19
5	Stress analysis of simply supported beam	22
6	Stress analysis of fixed beam	25
7	Stress analysis of an axi-symmetric component	28
8	Thermal stress analysis of a 2D component	31
9	Conductive heat transfer analysis of a 2D component	33
10	Convective heat transfer analysis of a 2D component	35
11	Mode frequency analysis of cantilever beam	37
12	Mode frequency analysis of simply supported beam	39
13	Harmonic analysis of a 2D component	41
14	Stress analysis of a truss	43

EXP. NO.01 STUDY OF BASICS IN ANSYS

Aim:

To study about the basic procedure to perform the analysis in ANSYS

Performing a typical ANSYS analysis:

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines. The next few sections of this chapter cover general steps that are common to most analyses.

A typical ANSYS analysis has three distinct steps:

1. Build the model.
2. Apply loads and obtain the solution.
3. Review the results.

Build the model:

1. Defining the Job name:

The job name is a name that identifies the ANSYS job. When you define a job name for an analysis, the job name becomes the first part of the name of all files the analysis creates. (The extension or suffix for these files' names is a file identifier such as .DB.) By using a job name for each analysis, you ensure that no files are overwritten.

2. Defining an Analysis Title:

The TITLE command (Utility Menu> File> Change Title), defines a title for the analysis. ANSYS includes the title on all graphics displays and on the solution output. You can issue the /STITLE command to add subtitles; these will appear in the output, but not in graphics displays.

3. Defining Units:

The ANSYS program does not assume a system of units for your analysis. Except in magnetic field analyses, you can use any system of units so long as you make sure that you use that system for all the data you enter. (Units must be consistent for all input data.)

4. Defining Element Types:

The ANSYS element library contains more than 150 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID96, etc.

The following element categories are available:

COMBINation	PIPE
CONTACt	PLANE
FLUID	PRETS (Pretension)
HF (High Frequency)	SHELL
HYPERelastic	SOLID
INFINite	SOURCe
INTERface	SURFace
LINK	TARGEt
MASS	TRANSducer
MATRIX	USER

VISCOelastic (or viscoplastic)

The element type determines, among other things:

The degree-of-freedom set (which in turn implies the discipline - structural, thermal, magnetic, electric, quadrilateral, brick, etc.) Whether the element lies in 2-D or 3-D space.

4.Defining Element Real Constants:

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. For example, real constants for BEAM3, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS). Not all element types require real constants, and different elements of the same type may have different real constant values.

6. Defining Material Properties:

Most element types require material properties. Depending on the application, material properties can be linear (see linear material properties) or nonlinear (see nonlinear material properties).

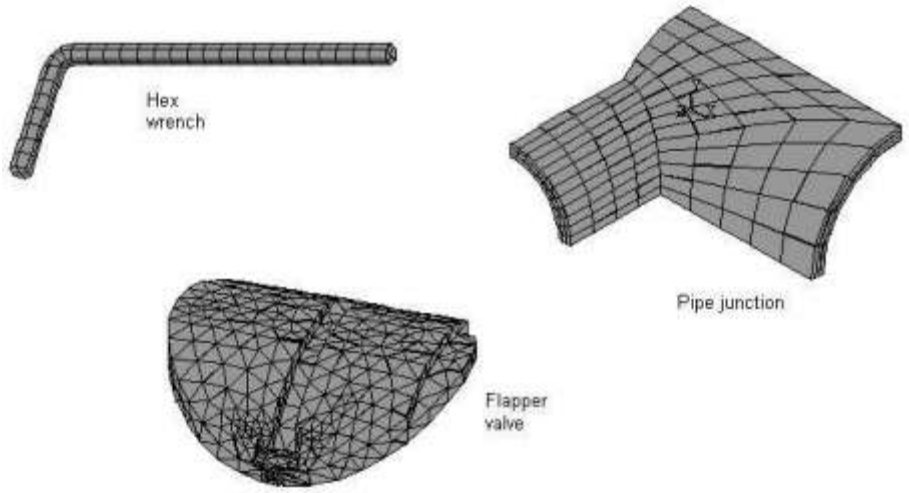
As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the material table. Within one analysis, you may have multiple material property sets (to correspond with multiple materials used in the model). ANSYS identifies each set with a unique reference number.

7. Creating the model geometry:

Once you have defined material properties, the next step in an analysis is generating a finite element model

i.e., nodes and elements that adequately describes the model geometry. The graphic below shows some sample finite element models.

There are two methods to create the finite element model; solid modeling and direct generation. With solid modeling, you describe the geometric shape of your model, then instruct the ANSYS program to automatically mesh the geometry with nodes and elements. You can control the size and shape in the elements that the program creates. With direct generation, you "manually" define the location of each node and the connectivity of each element.



Sample finite element models

Apply loads and obtain the solution:

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7 preprocessor.

1. Defining the analysis types and analysis options

You choose the analysis type based on the loading conditions and the response you wish to calculate. For example, if natural frequencies and mode shapes are to be calculated, you would choose a modal analysis. You can perform the following analysis types in the ANSYS program; static (or steady-state), transient, harmonic, modal, spectrum, buckling, and sub structuring.

Not all analysis types are valid for all disciplines. Modal analysis, for example, is not valid for a thermal model. The analysis guide manuals in the ANSYS documentation set describe the analysis types available for each

discipline and the procedures to do those analyses.

Analysis options allow you to customize the analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton-Raphson options.

2. Applying Loads

The word loads as used in ANSYS documentation includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads. Loads in the ANSYS program are divided into six categories:

DOF constraints

Forces

Surface loads

Body loads

Inertia loads

Coupled-field loads

You can apply most of these loads either on the solid model (keypoints, lines, and areas) or the finite element model (nodes and elements).

3. Specifying load step options

Load step options are options that you can change from load step to load step, such as number of sub steps, time at the end of a load step, and output controls. Depending on the type of analysis you are doing, load step options may or may not be required. The analysis procedures in the analysis guide manuals describe the appropriate load step options as necessary.

4. Initiating the solution:

To initiate solution calculations, use either of the following

Command(s): SOLVE

Main Menu> Solution>solve>current LS

Main Menu> Solution>solution method

When you issue this command, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file (Jobname.RST, Jobname.RTH, Jobname.RMG, or Jobname.RFL) and also to the database.

The only difference is that only one set of results can reside in the database at one time, while you can write all sets of results (for all substeps) to the results file.

Review the Results:

Once the solution has been calculated, you can use the ANSYS postprocessors to review the results. Two postprocessors are available: POST1 and POST26.

You use POST1, the general postprocessor, to review results at one substep (time step) over the entire model or selected portion of the model. The command to enter POST1 is /POST1 (Main Menu> General Post proc), valid only at the beginning level. You can obtain contour displays, deformed shapes, and tabular listings to review and interpret the results of the analysis. POST1 offers many other capabilities, including error estimation, load case combinations, calculations among results data, and path operations.

You use POST26, the time history postprocessor, to review results at specific points in the model over all time steps. The command to enter POST26 is /POST26 (Main Menu>Time Hist Post pro), valid only at the beginning level. You can obtain graph plots of results data versus time (or frequency) and tabular listings. Other POST26 capabilities include arithmetic calculations and complex algebra.

Specific Capabilities of ANSYS Structural Analysis:

Structural analysis is probably the most the common application of the finite element method such as piston, machine parts and tools.

Static Analysis:

It is the used to determine displacement, stress etc. under static loading conditions. ANSYS can compute linear and non-linear types (e.g. the large strain hyper elasticity and creep problems).

Transient Dynamic Analysis:

It is used to determine the response of a structure to time varying loads.

Buckling Analysis:

It is used to calculate buckling load and to determine the shape of the component after applying the buckling load. Both linear buckling and non - linear buckling analysis are possible.

Thermal Analysis:

The steady state analysis of any solid under thermal boundary conditions calculates the effect of steady thermal load on a system (or) component that includes the following.

1. Convection.
2. Radiation.
3. Heat flow rates.
4. Heat fluxes.
5. Heat generation rates.
6. Constant temperature boundaries.

Fluid Flow:

The ANSYS CFD offers comprehensive tools for analysis of two-dimensional and three dimensional fluid flow fields.

Magnetic:

Magnetic analysis is done using ANSYS / Electromagnetic program. It can calculate the magnetic field in device such as power generators, electric motor etc. Interest in magnetic analysis is finding magnetic flux, magnetic density, power loss and magnetic forces.

Acoustic / Vibrations:

Ansys is capable of modeling and analyzing vibration system. Acoustic is the study of the generation, absorption and reflection of pressure waves in a fluid application.

Few examples of acoustic applications are

1. Design of concert house, where an even distribution of sound pressure is possible
2. Noise cancellation in automobile
3. Underground water acoustics
4. Noise minimization in machine shop
5. Geophysical exploration

Coupled fields:

A coupled field analysis is an analysis that takes into account the interaction between two (or) more fields of engineering analysis. Pressure vessels, Induction heating and micro electro mechanical systems are few examples.

EXP. NO.02 STRESS ANALYSIS OF A PLATE WITH A CIRCULAR HOLE

Aim:

To conduct the stress analysis in a plate with a circular hole using ANSYS software

System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Pre Processing
2. Solution
3. Post Processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

Preprocessing:

1. Preference - Structural- h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, 8 node 82 – ok – Option – choose Plane stress w/thk - close
3. Real constants - Add/Edit/Delete – Add – ok – THK 0.5 – ok - close
4. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX 2e5, PRXY 0.3 - ok
5. Modeling – Create – Areas – Rectangle - by 2 corner - X=0, Y=0, Width=100, Height=50 - ok- Circle - Solid circle - X=50, Y=25, Radius=10 - ok-operate –Booleans – Subtract – Areas - Select the larger area (rectangle) – ok – Select Circle – Next –ok
6. Meshing - Mesh Tool – Area – Set - Select the object – ok - Element edge length2/3/4/5– ok - Mesh Tool - Select TRI or QUAD - Free/Mapped – Mesh - Select theobject –ok

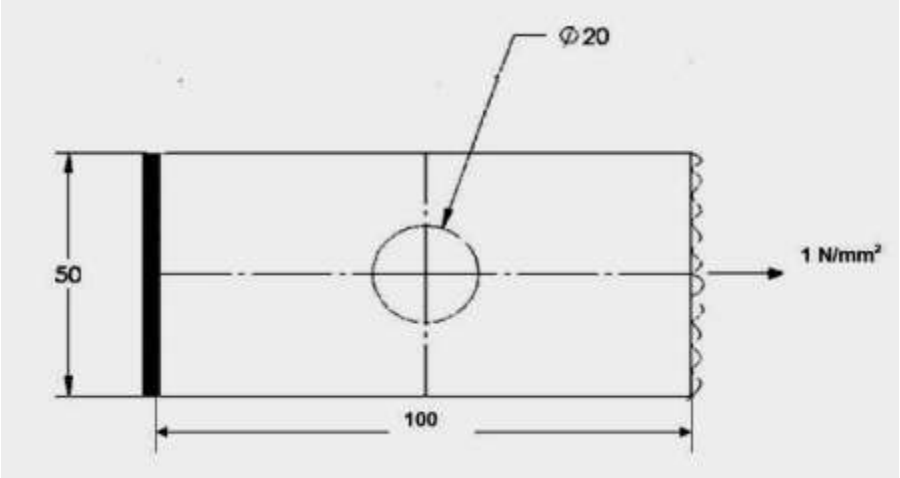
Solution:

7 Solution – Define Loads – Apply – Structural – Displacement - On lines - Select the boundary where is going to be arrested – ok - All DOF - ok. Pressure - On lines - Select the load applying area – ok - Load PRES valve = 1N/mm – ok

8. Solve – Current LS – ok– Solution is done – close

Post processing:

9. General post proc - Plot Result - Contour plot - Nodal Solution – Stress –Von mises stress – ok
10. Plot control – Animates - Mode Shape – Stress - Von mises – ok
11. Plot control – Animate - Save Animation - Select the proper location to save the file (E drive-user) – ok
12. File – Report Generator – Choose Append – ok – Image Capture – ok - close



Young"s Modulus : 2×10^{5} N/mm ^{2}

Poisson"s ratio : 0.3

Result:

Thus the stress analysis of rectangular plate with a circular hole is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse a rectangular plate with a circular hole using the ANSYS software for structural applications.

Application:

1. Air Craft Industry
2. Automobile Industry

EXP. NO.03 STRESS ANALYSIS OF A RECTANGULAR L - BRACKET

Aim:

To conduct the stress analysis of a rectangular L section bracket using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

Preprocessing:

1. Preference - Structural- h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, 8 node 82 –ok– Option – Choose Planestress w/thk - close
3. Real constants - Add/Edit/Delete – Add – ok – THK 0.5 – ok - close
4. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX 2e5, PRXY 0.3 - ok
5. Modeling – Create – Key points - In active CS – enter the key point number and X, Y, Z location for 6 key points to form the rectangular L-bracket.
   Lines – lines - Straight line - Connect all key points to form as lines. Areas – Arbitrary - by lines - Select all lines - ok. Lines - Line fillet - Select the two lines where the fillet is going to be formed – ok – enter the Fillet radius=10- ok Areas –Arbitrary - through KPs - Select the key points of the fillet - ok
   Operate – Booleans – Add – Areas - Select the areas to be add (L Shape & fillet area) - ok. Create – Areas – Circle - Solid circle - Enter the co-ordinates, radius of the circles at the two ends(semicircles) -ok. Operate – Booleans – Add – Areas - Select the areas to be add (L Shape &two circles) - ok

Create – Areas – Circle - Solid circle – Enter the coordinates, radius of the two circles which are mentioned as holes - ok. Operate – Booleans – Subtract – Areas - Select the area of rectangle – ok - Select the two circles - ok

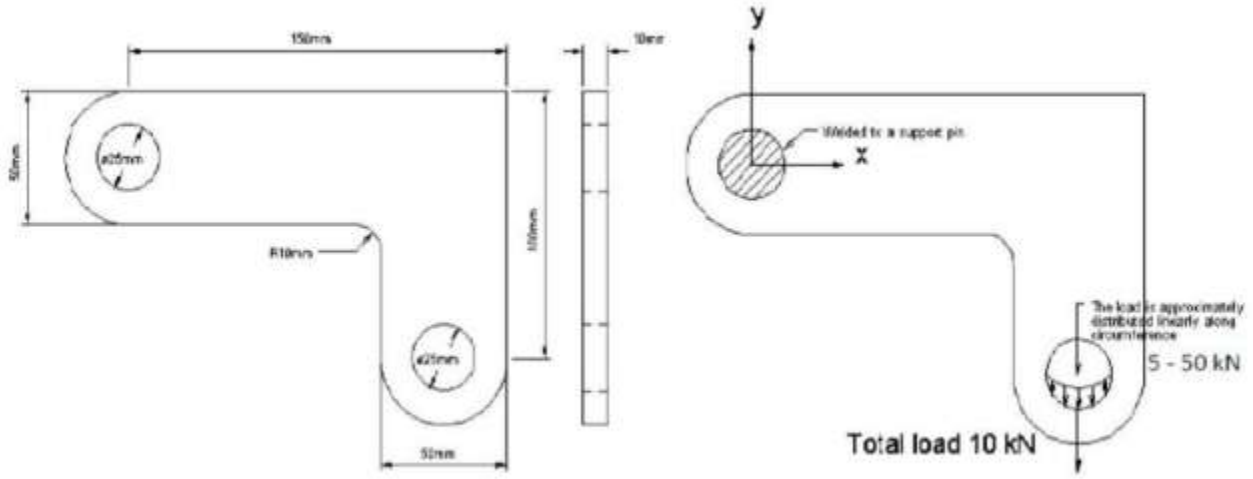
6. Meshing - Mesh Tool – Area – Set - Select the object – ok - Element edge length 2/3/4/5 – ok - Mesh Tool - Select TRI or QUAD - Free/Mapped – Mesh - Select the object - ok

Solution:

7. Solution – Define Loads – Apply – Structural – Displacement - On lines - Select the boundary where is goingto be arrested – ok - All DOF - ok.Pressure - On lines - Select the load applying area – ok - Load PRES valve = -10000 N (- Sign indicates thedirection of the force i.e. downwards) – ok
8. Solve – Current LS – ok– Solution is done – close

Post processing:

9. General post proc - Plot Result - Contour plot - Nodal Solution – Stress –Vonmises stress - ok
10. Plot control – Animates - Mode Shape – Stress - Von mises - ok
11. Plot control – Animate - Save Animation - Select the proper location to save the file (E drive-user) - ok
12. File – Report Generator – Choose Append – ok – Image Capture – ok – close



Young"s Modulus : 200 GPa

Poisson"s ratio : 0.3

Result:

Thus the stress analysis of rectangular L section bracket is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse a rectangular L section bracket using the ANSYS software for structural applications.

Application:

1. Structural Application
2. Military Application

EXP. NO.04 STRESS ANALYSIS OF CANTILEVER BEAM

Aim:

To conduct the stress analysis in a cantilever beam using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

Preprocessing:

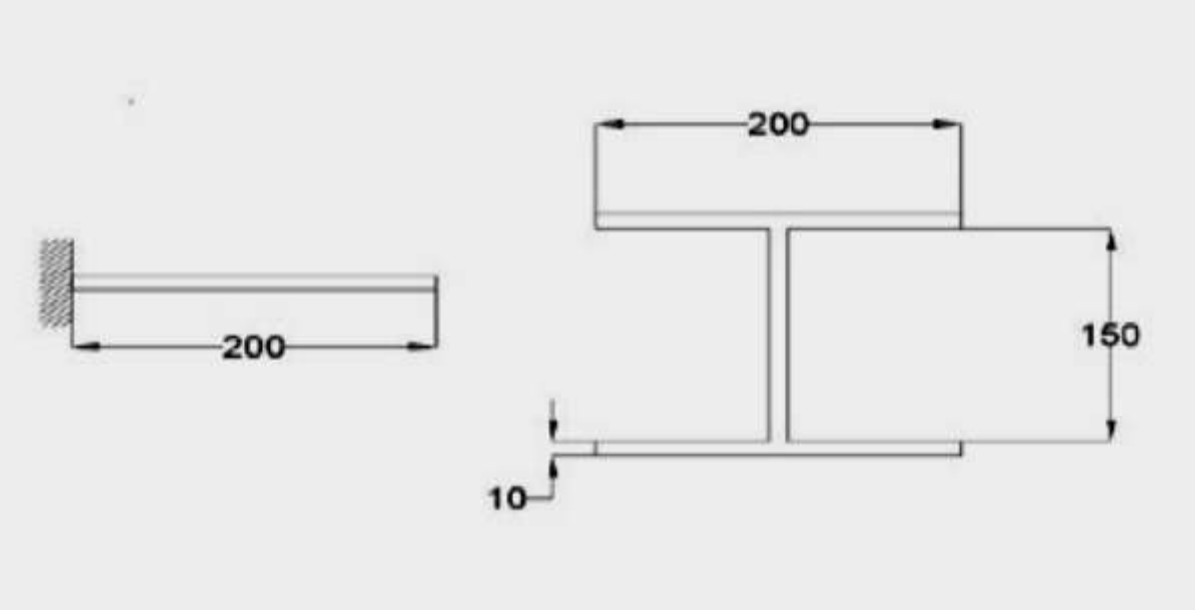
1. Preference - Structural- h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – ok – Options – ok - close
3. Sections – beam – Common sections – Select the correct section of the beam and input the of “w1, w2,w3” and “t1, t2, t3” – Preview – Note down the values of area, Iyy
4. Real constants - Add/Edit/Delete – Add – ok – Enter the values of area=5500, Izz=0.133e8, height=3 – ok -close
5. Material props - Material Models – Structural – Linear – Elastic – Isotropic - EX 2e5, PRXY 0.3 - ok
6. Modeling – Create – Key points – In active CS – Enter the values of CS of each key points – Apply – ok. Lines – Lines – Straight line – Pick the all points – ok.
7. Meshing – Mesh attributes – All lines – ok. Meshing – Size cntrls – Manual size –Lines – All lines – Enter the value of element edge length [or] Number of element divisions – ok. Mesh tool – Mesh – Pick all

Solution:

8. Solution – Define Loads – Apply – Structural – Displacement - On key points – Select the 1st key point – ALL DOF – ok. On key points – select the 2nd key point– UY – ok. Force/Moment – On key points – Select the key point – ok – direction of force/moment FY, Value = -1,000 (- sign indicates the direction of the force) – ok
9. Solve – Current LS – ok – Solution is done – close

Post processing:

10. General post proc – Element table – Define table – Add – By sequence num – SMISC, 6 – ok – SMISC,12 –ok – LS,2 – ok – LS,3 - ok – Close. Plot results – Contour plot – Nodal solution – DOF solution – Y component of displacement – ok. Contour plot – Line ElementRes – Node I SMIS 6, Node J SMISC, 12 – ok. Contour plot– Line element Res – Node I LS 2, Node J LS 3 – ok
11. File–Report Generator–Choose Append–ok–Image Capture–ok - close



Young‟s Modulus : 200 GPa

Poisson‟s ratio : 0.3

Result:

Thus the stress analysis of a cantilever beam is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse a cantilever beam using the ANSYS software.

Application:

1. Structural Application
2. Construction Application

Expt.No.05 STRESS ANALYSIS OF SIMPLY SUPPORTED BEAM

Aim:

To conduct the stress analysis in a simply supported beam using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

Preprocessing:

1. Preference - Structural – h method - ok
2. Preprocessor - Element type - Add/Edit/Delete - Add - beam 2D elastic 3 –Options – ok– close – real constant- Add/Edit/Delete- Add- area = 100, I_{zz} = 833.33 & height =10- ok
3. Preprocessor – Material Properties – Material Model – Structural – Linear – Elastic – Isotropic – EX 2e5, PRXY0.3 – ok
4. Preprocessor – Modeling – create – nodes – inactive CS

   Node 1 X=0 Y=0

   Node 2 X= 25 Y=0

   Node 3 X= 50 Y=0

   Node 4 X= 75 Y=0

   Node 5 X= 100 Y=0
5. List - nodes - coordinate only -ok
6. Preprocessor- modeling- create- elements- Auto numbered through nodes- select

Node 1 & 2

Node 2 & 3

Node 3 & 4

Node 4 & 5

Node 5 & 6 - ok

Solution:

7. Solution - define loads- apply- structural - displacement - on nodes - select node 1 & node 5 - apply - UY - displacement = 0 -ok
8. Solution - Force/moment - on nodes - node 3 - apply - FY = -100 -ok
9. Solution - solve - current LS -ok

Post Processing:

10. General post processor - plot result - deform shape - Deformed + Undeformed-ok
11. General post processor - element table - define table - add - user table for item

    Smax I > by sequence num> NMISC 1 > apply

    Smax J > by sequence num> NMISC 3 >apply

    Smin I > by sequence num> NMISC 2 > apply

    Smin J > by sequence num> NMISC 4 >ok

12. Plot result - line element result -Smax I-Smax J - first result -Evaluate table data – Smax I, Smax J, Smin I, Smin J -ok
13. General postprocessor - list result - nodal solution - DOF solution - UY-displacement result
14. General postprocessor - contour plot - line element res. –ok

Result:

Thus the stress analysis of a simply supported beam is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse a simply supported beam using the ANSYS software.

Application:

1. Structural Application
2. Construction Application

EXP. NO.06 STRESS ANALYSIS OF FIXED BEAM

Aim:

To conduct the stress analysis in a fixed beam using ANSYS software

System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

Preprocessing:

1. Preference - Structural – h method - ok
2. Preprocessor - Element type - Add/Edit/Delete - Add - beam 2D elastic 3 –Options – ok – close – real constant- Add/Edit/Delete- Add- area = 100, I_{zz} = 833.33 & height =10- ok
3. Preprocessor – Material Properties – Material Model – Structural – Linear – Elastic – Isotropic – EX 2e5, PRXY0.3 – ok
4. Preprocessor – Modeling – create – nodes – inactive CS

   Node 1 X=0 Y=0

   Node 2 X= 25 Y=0

   Node 3 X= 50 Y=0

   Node 4 X= 75 Y=0

   Node 5 X= 100 Y=0
5. List - nodes - coordinate only -ok
6. Preprocessor- modeling- create- elements- Auto numbered through nodes- select

Node 1 & 2

Node 2 & 3

Node 3 & 4

Node 4 & 5

Node 5 & 6 -ok

Solution:

7. Solution - define loads- apply- structural - displacement - on nodes - select node 1 & node 5 - apply - UY - displacement = 0 -ok
8. Solution - Force/moment - on nodes - node 3 - apply - FY = -100 -ok
9. Solution - solve - current LS -ok

Post processing:

10. General post processor - plot result - deform shape - Deformed + Undeformed -ok
11. General post processor - element table - define table - add - user table for item

    Smax I > by sequence num> NMISC 1 > apply

    Smax J > by sequence num> NMISC 3 >apply

    Smin I > by sequence num> NMISC 2 > apply

    Smin J > by sequence num> NMISC 4 >ok
12. Plot result - line element result - Smax I- Smax J - first result -Evaluate table data –Smax I, Smax J, Smin I, Smin J -ok

    General postprocessor - list result - nodal solution - DOF solution - UY - displacement result
13. General postprocessor - contour plot - line element res. -ok

Result:

Thus the stress analysis of a fixed beam is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse a fixed beam using the ANSYS software for structural applications.

Application:

1. Structural Application
2. Construction Application

EXP. NO.07 STRESS ANALYSIS OF AN AXI-SYMMETRIC COMPONENT

Aim:

To obtain the stress distribution of an axisymmetric component

The model will be that of a closed tube made from steel. Point loads will be applied at the centre of the top and bottom plate.

System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Preprocessing:

1. Utility Menu - Change Job Name - Enter Job Name. Utility Menu - File - Change Title - Enter New Title
2. Preference - Structural –h method - ok
3. Preprocessor - Element type - Add/Edit/ delete - solid 8node 183 – options-axisymmetric
4. Preprocessor - Material Prop - Material Model - Structural - Linear - Elastic -Isotropic - EX = 2e5, PRXY = 0.3
5. Preprocessor –Modeling -create- Areas-Rectangle - By dimensions

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

6. Preprocessor - Modeling - operate - Booleans - Add - Areas - pick all –ok
7. Preprocessor - meshing - mesh tool - size control - Areas - Element edge length = 2 mm -ok- mesh - Areas – free- pick all.

Solution:

8. Solution - Analysis Type-New Analysis-Static

9. Solution - Define loads - Apply .Structural - displacement - symmetry BC - on lines.

(Pick the two edger on the left at X = 0)

10. Utility menu - select - Entities - select all

11. Utility menu - select - Entities - by location - Y = 50 -ok.

12. Solution - Define loads - Apply - Structural - Force/Moment - on key points - FY =100 - Pick the top left corner of the area -ok

13. Solution - Define Loads - apply - Structural - Force/moment - on key points - FY =-100- Pick the bottom left corner of the area -ok

14. Solution - Solve - Current LS

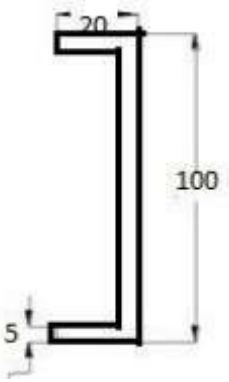
15. Utility Menu - select - Entities

16. Select nodes - by location - Y coordinates and type 45, 55 in the min., max. box, asshown below and click ok

Post processing:

17. General postprocessor - List results - Nodal solution - stress - components SCOMP

18. Utility menu - plot controls - style - Symmetry expansion - 2D Axisymmetric - ¾ expansion



Young‟s Modulus : 200 GPa

Poisson‟s ratio : 0.3

Result:

Thus the stress analysis of an axi-symmetric component done by using the ANSYS software.

Outcome:

From this experiment, students will be able to analyse an axi-symmetric component using the ANSYS software.

Application:

1. Structural Application
2. Automobile Application

EXP. NO.08 THERMAL STRESS ANALYSIS OF A 2DCOMPONENT

Aim:

To conduct the thermal stress analysis of a 2D component by using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run.

Preprocessing:

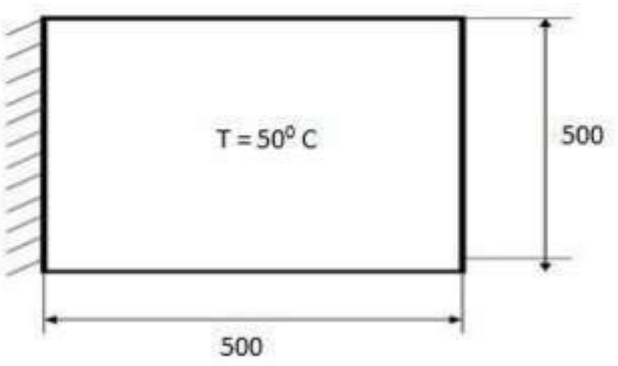
1. Preference – Thermal - h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 42 – ok – Options – plane strsw/thk – ok – Close
3. Real constants - Add/Edit/Delete – Add – ok – THK 100 – ok – Close
4. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 2e5,PRXY 0.3 – ok –Thermal expansion – Secant coefficient – Isotropic – ALPX 12e-6 – ok
5. Modeling – Create – Areas - Rectangle – by 2 corners – Enter the coordinate values, height, width-ok
6. Meshing – Mesh tool – Areas, set – select the object – ok – Element edge length 10 -ok – Mesh tool- Tri, free - mesh – Select the object

Solution:

7. Solution – Define Loads – Apply – Structural – Displacement - On lines – Select the boundary on the object –ok – Temperature – Uniform Temp – Enter the temp. Value 50 –ok.
8. Solve – Current LS – ok – Solution is done – close

Post processing:

9. General post proc – Plot results – Contour plot – Nodal solution – Stress – 1st principal stress – ok – Nodal solution – DOF Solution – Displacement vector sum - ok
10. File – Report Generator – Choose Append – ok – Image Capture – ok - close



Young‟s Modulus = 200 GPa

Poisson‟s ratio = 0.3

Thermal expansion coefficient = 12 ×10 $^{-6}$/ C

Result:

Thus the thermal stress analysis of a 2D component is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse the thermal stress of a 2D component using the ANSYS software for heat transfer applications.

Application:

1. Structural Application
2. Heat Exchange Application

EXP. NO.09 CONDUCTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

Aim:

To conduct the conductive heat transfer analysis of a 2D component by using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Preprocessing:

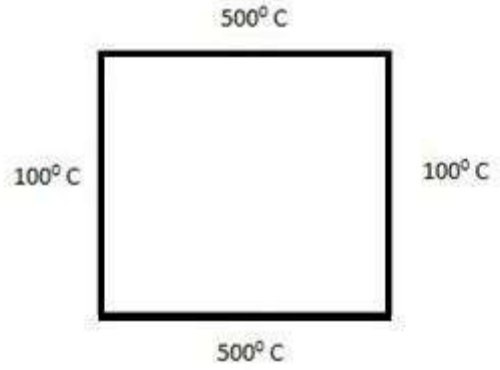
1. Preference – Thermal - h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 55 – ok – close – options – plane thickness – ok
3. Real constants - Add/Edit/Delete – Add – ok – THK 0.5 – ok – Close
4. Material props - Material Models –Thermal – Conductivity – Isotropic – KXX 10 – ok
5. Modeling – Create – Areas - Rectangle – by 2 corners – Enter the coordinate values, width - ok
6. Meshing – Mesh tool – Areas, set – select the object – ok – Element edge length 0.05 - ok – Mesh tool- Tri, free - mesh – Select the object –ok

Solution:

7. Solution – Define Loads – Apply – Thermal – Temperature - On lines – Select the right and left side of the object –ok – Temp. Value 100 – On lines – select the top and bottom of the object – ok –Temp 500 – ok
8. Solve – Current LS – ok – Solution is done – close

Post processing:

9. General post proc – Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature – ok
10. File–Report Generator–Choose Append–ok–Image Capture–ok - close



Thermal Conductivity of the material = 10 W/m ℃

Dimension of the object = 2m × 2m

Result:

Thus the conductive heat transfer analysis of a 2D component is done by using ANSYS software.

Outcome:

From this experiment, students will be able to analyse the conductive heat transfer of a 2D component using the ANSYS software.

Application:

1. Boiler Application
2. Heat Transfer Application

EXP. NO.10 CONVECTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT

Aim:

To conduct the convective heat transfer analysis of a 2D component by using ANSYS software

System configuration:

Ram : 8 GB

Processor : Core 2 Quad / Core 2 Duo

Operating system : Windows 7

Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Preprocessing:

1. Preference – structural - h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 55 – ok – Close
3. Real constants - Add/Edit/Delete – Add – ok
4. Material props - Material Models –Thermal – Conductivity – Isotropic – KXX 16 – ok
5. Modeling – Create – Key points - In active CS – enter the key point number and X, Y, Z location for 8 key points to form the shape as mentioned in the drawing. Lines – lines - Straight line - Connect all the key points to form as lines. Areas – Arbitrary - by lines - Select all lines - ok.
6. Meshing – Mesh tool – Areas, set – select the object – ok – Element edge length 0.05 - ok – Mesh tool- Tri,free mesh – Select the object –ok

Solution:

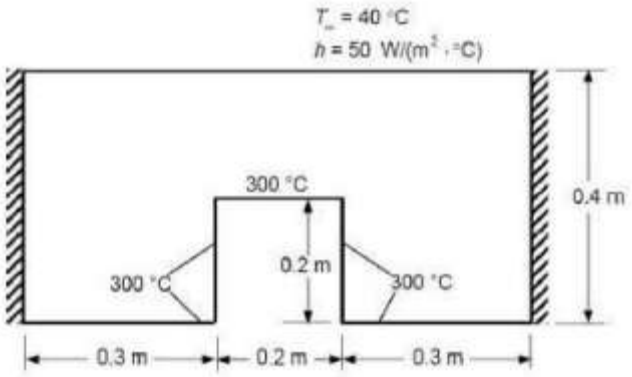
7. Solution – Define Loads – Apply – Thermal – Temperature - On lines – Select the lines–ok – Temp. Value 300 – ok – Convection – On lines – select the appropriate line – ok – Enter the values of film coefficient 50, bulk temperature 40 –ok
8. Solve – Current LS – ok – solution is done – Close

Post Processing:

9. General post proc – List results – Nodal Solution – DOF Solution – Nodal temperature – ok

10. Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature – ok
11. File – Report Generator – Choose Append – ok – Image Capture – ok - Close

.



Thermal Conductivity of the material = 16 W/m ^{0}

Result:

Thus the convective heat transfer analysis of a 2D component is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse the convective heat transfer of a 2D component using the ANSYS software.

Application:

1. Structural Application
2. Heat Exchanger Application

EXP. NO.11 MODE FREQUENCY ANALYSIS OF A CANTILEVER BEAM

Aim:

To conduct the Mode frequency analysis of a cantilever beam using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Preprocessing:

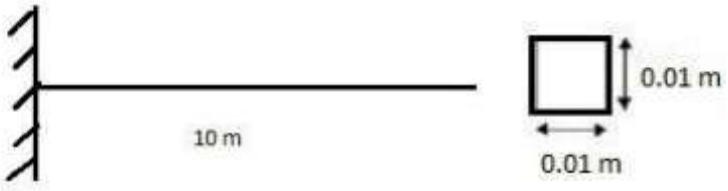
1. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – ok – close
2. Real constants - Add/Edit/Delete – Add – ok – Area 0.1e-3, Izz 0.833e-9, Height 0.01 – ok – close
3. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 2e9, PRXY 0.25 – ok – Density – DENS 7830 – ok
4. Modeling – Create – Key points – Inactive CS – Enter the coordinate values - ok. Lines -lines – Straight Line – Join the two key points – ok
5. Meshing – Size Cntrls – manual size – lines – all lines – Enter the value of no of element divisions 25 – ok. Mesh – Lines – Select the line – ok

Solution:

6. Solution – Define Loads – Apply – Structural – Displacement - On nodes – Select the node point –ok – All DOF – ok- Analysis type – New analysis – Modal – ok
7. Analysis type – Analysis options – Block Lanczos – enter the value no of modes to extract as 3 or 4 or 5 – ok – End Frequency 10000 – ok.
8. Solve – Current LS – ok – Solution is done – close

Post processing:

9. General post proc – Read results – First set - Plot results – Deformed shape – Choose Def+undeformed – ok.Read results – Next set - Plot results – Deformed shape – Choose Def+undeformed – ok and so on
10. File–Report Generator–Choose Append–ok–Image Capture–ok – Close (Capture all images)



Young"s Modulus = 200×10 N/m^2

Poisson"s ratio = 0.25

Weight Density = 7.83 ×10 kg/m^3

Result:

Thus the mode frequency analysis of a cantilever beam is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse the mode frequency of a cantilever beam using the ANSYS software.

Application:

1. Structural Application
2. Construction Application

EXP. NO.12 MODE FREQUENCY ANALYSIS OF A SIMPLYSUPPORTED BEAM

Aim:

To conduct the Mode frequency analysis of a simply supported beam using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Preprocessing:

1. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – ok – Close
2. Real constants - Add/Edit/Delete – Add – ok – Area 0.1e-3, Izz 0.833e-9, Height 0.01 – ok – close
3. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 2e9, PRXY 0.25 – ok – Density – DENS 7830 – ok
4. Modeling – Create – Key points – Inactive CS – Enter the coordinate values - ok. Lines -lines – Straight Line – Join the two key points – ok
5. Meshing – Size Cntrls – manual size – lines – all lines – Enter the value of no of element divisions 25 – ok. Mesh – Lines – Select the line – ok

Solution:

6. Solution – Define Loads – Apply – Structural – Displacement - On nodes – Select the first point and second point –ok – apply –UY- displacement = 0-ok. Analysis type – New analysis – Modal – ok
7. Analysis type – Analysis options – Block Lanczos – enter the value no of modes to extract as 3 or 4 or 5 – ok – End Frequency 10000 – ok
8. Solve – Current LS – ok – Solution is done – close

Post processing:

9. General post proc – Read results – First set - Plot results – Deformed shape – Choose Def+undeformed – ok.Read results – Next set - Plot results – Deformed shape – Choose Def+undeformed – ok and so on
10. File–Report Generator–Choose Append–ok–Image Capture–ok - Close.(Capture all images)

Result:

Thus the mode frequency analysis of a simply supported beam is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse the mode frequency of a simply supported beam using the ANSYS software.

Application:

1. Structural Application
2. Construction Application

EXP. NO.13 HARMONIC ANALYSIS OF A 2D COMPONENT

Aim:

To conduct the harmonic analysis of a 2D component by using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Preprocessing
2. Solution
3. Post processing

Preprocessing:

1. Preprocessor - Element type - Add/Edit/Delete – Add – Beam, 2D elastic 3 – ok – Close
2. Real constants - Add/Edit/Delete – Add – ok – Area 0.1e-3, Izz 0.833e-9, Height 0.01 – ok – close
3. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 206e9, PRXY 0.25 – ok – Density – DENS 7830 – ok
4. Modeling – Create – Key points – Inactive CS – Enter the coordinate values - ok. Lines – lines – Straight Line – Join the two key points – ok
5. Meshing – Size Cntrls – manual size – lines – all lines – Enter the value of no of element divisions 25 – ok. Mesh – Lines – Select the line – ok

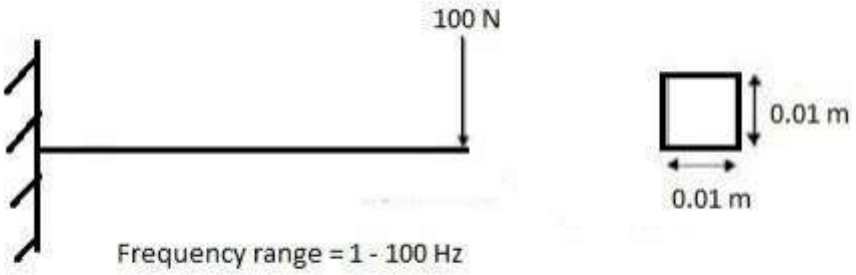
Solution:

6. Solution - Analysis type – New analysis – Harmonic – ok. Analysis type – Analysis options – Full, Real+ imaginary – ok– Use the default settings – ok
7. Solution – Define Loads – Apply – Structural – Displacement - On nodes – Select the node point –ok – All DOF – ok. Force/Moment – On Nodes – select the node 2 – ok – Direction of force/mom FY, Real part of force/mom -100 – ok. Load step Opts – Time/Frequency – Freq and Sub stps – Enter the values of Harmonic freq range 1-100, Number of sub steps 100, Stepped – ok

8. Solve – Current LS – ok – Solution is done – close

Post Processing:

9. Time Hist post pro – Variable Viewer – Click “Add” icon – Nodal Solution – DOF Solution –Y-Component of displacement – ok – Enter 2 – ok. Click “List data” icon and view theamplitude list. Click “Graph” icon and view the graph. To get a better view of the response, view the log scale of UY. Plot ctrls – Style – Graphs – Modify axes – Select Y axis scale as Logarithmic – ok. Plot – Replot – Now we can see the better view.
10. File–Report Generator–Choose Append–ok–Image Capture–ok - close



Young‟s Modulus = 206×10 N/m^2

Poisson‟s ratio = 0.25

Weight Density = 7.83 × 10 kg/m^3

Length of the beam = 1 m

Result:

Thus the harmonic analysis of 2D component is done by using the ANSYS Software.

Outcome:

From this experiment, students will be able to analyse the harmonic analysis of 2D component using the ANSYS software.

Application:

1. Structural Application
2. Construction Application

EXP. NO.14 STRESS ANALYSIS OF A TRUSS

Aim:

To conduct the stress analysis of a truss by using ANSYS software

System configuration:

Ram : 8 GB
Processor : Core 2 Quad / Core 2 Duo
Operating system : Windows 7
Software : ANSYS (Version12.0/12.1)

Procedure:

Step 1: Ansys Utility Menu

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

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL - ok

Step 3: Preprocessor

Element type – Add/Edit/Delete – Add – Link – 2D spar 1 – ok – close.

Real constants – Add – ok – real constant set no – 1 – c/s area – 0.1 – ok – close.

Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 210e9 – ok – close.

Step 4: Preprocessor

Modeling – Create – Nodes – In Active CS – Apply (first node is created) – x,y,z location in CS – 4 (x value w.r.t first node) – apply (second node is created) – x,y,z location in CS – 4, 3 (x, y value w.r.t first node) – apply (third node is created) – 0, 3 (x, y value w.r.t first node) – ok (forth node is created).

Create – Elements – Elem Attributes – Material number – 1 – Real constant set number – 1 – ok Auto numbered – Thru Nodes – pick 1 & 2 – apply – pick 2 & 3 – apply – pick 3 & 1 – apply – pick 3 & 4 – ok (elements are created through nodes).

Step 5: Preprocessor

Loads – Define loads – apply – Structural – Displacement – on Nodes – pick node 1 & 4 – apply – DOFs to be constrained – All DOF – ok – on Nodes – pick node 2 – apply – DOFs to be constrained – UY – ok.

Loads – Define loads – apply – Structural – Force/Moment – on Nodes- pick node 2 – apply – direction of For/Mom – FX – Force/Moment value – 2000 (+ve value) – ok – Structural – Force/Moment – on Nodes- pick node 3 – apply – direction of For/Mom – FY – Force/Moment value – -2500 (-ve value) – ok.

Step 6: Solution

Solve – current LS – ok (Solution is done is displayed) – close.

Step 7: General Post Processor

Element table – Define table – Add – 'Results data item' – By Sequence num – LS – LS1 – ok.

Step 8: General Post Processor

Plot Results – Deformed Shape – def+undeformed – ok.

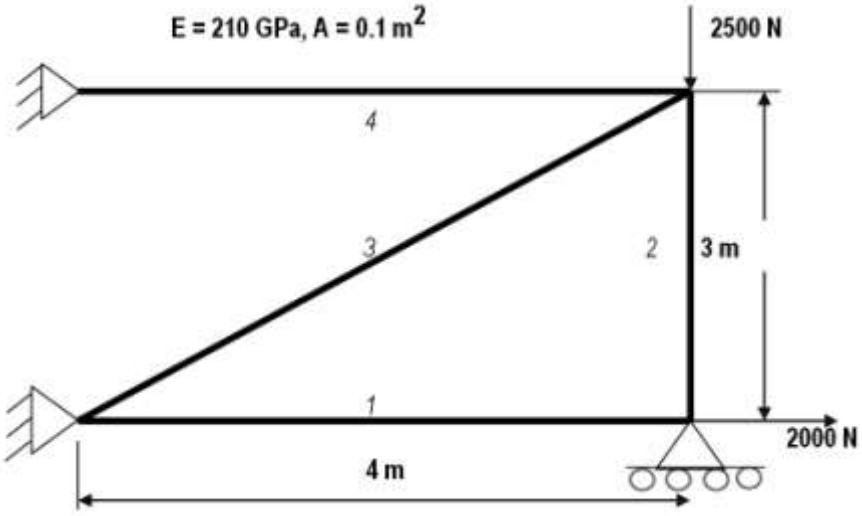
Plot results – contour plot – Line Element Results – Elem table item at node I – LS1 – Elem table item at node J – LS1 – ok (Line Stress diagram will be displayed).

Plot results – contour plot – Nodal solution – DOF solution – displacement vector sum – ok.

List Results – reaction solution – items to be listed – All items – ok (reaction forces will be displayed with the node numbers).

List Results – Nodal loads – items to be listed – All items – ok (Nodal loads will be displayed with the node numbers).

Step 9: PlotCtrls – Animate – Deformed shape – def+undeformed-ok



SIMULATION AND ANALYSIS LAB VIVA QUESTIONS

1) What are the different approximate solution methods.
Different approximate solution methods are
1.Functional approximation
2.Finite Difference method
3. Finite Element method

2) What do you mean by continuum.
Structure which is considered for analysis is called continuum.

3) Define term node
The element which is connected with another element at junction is called node.

4) Define term element
Descritised structure is called an element .

5) What is convergence
Process of achieving value to actual solution.

6) What are the types convergence?
p-convergence
h-convergence

7) What is p- convergence?
Convergence by increasing the elements

8) What is h convergence?
Convergence by increasing nodes

9) What is higher order elements?
The element which contain more no. of nodes are called higher order element .

10) Give example for higher order elements
Higher order elements are CST, Quadrilateral element.

11) What do you mean by compatible elements?
Elements, which are compatible with adjacent element, like no discontinuity, overlap or sudden slope.

12) What is geometric invariance?
The property in which the shape of the element will not change with change in local coordinates is called geometric invariance.

13) Why do we use Pascal‟s triangle in FEA?
If the displacement equation doesn‟t contain all the required terms then balancing is done by the Pascal‟s triangle.

14) What are the steps involved in FEA?
Steps involved in FEA:
1.Modelling
2.Descritization of structure

3.Derivation of elemental stiffness matrix.
4.Assembly of elemental equation
5.Applying boundary conditions
6.Computation of stress and strain
7.Interpretaion of results

15) What is stiffness matrix?
The matrix when contain parameters like E, A, displacement and applied force is stiffness matrix

16) How to obtain stiffness matrix?
Stiffness matrix can be obtained by applying condition of minimum potential energy to potential energy equation.

17) What is displacement function?
Displacement function is the assumed polynomial equation, which satisfies boundary conditions.

18) How to identify order of elements?
Order of elements depends on the no. of nodes.

19) Mention different types of elements.
Different types of elements are bar elements, beam element, truss element, shell element, axis symmetric.

20) Mention some application of FEA.
Mechanical, Aerospace, Civil, structure analysis, biomedical, geo -mechanic, electromagnetic.

21) What is connectivity?
Relation between the connected elements is connectivity.

22) What are the methods to improve problem solution?
Problem solution can be improved by increasing no. of elements or no. of nodes .

23) Define symmetry in matrix.
It is the square matrix in which the element of the row are same as that of element of column.

24) What is plane stress?
Stress acting on 2-D element.

25) What is plane strain?
Strain occurring in 2-D element.

26) Compare FEA with solid mechanics.
complicated irregular structures are difficult in solid mechanics but FEA it‟s easier with greater accuracy.

27) What are the packages available for FEA?
packages available for FEA: ANSYS, I-DEAS,NASTRAN,ABAQUS, COSMOS,ALGOL,PATRAN.

28) Define potential energy.
Energy possessed by the body due to it‟s position

29) Define minimum potential energy.
for consertive system of all kinematically admissible displacement field those corresponding to equilibrium extremise the total potential energy if extreme condition is a minimum the equilibrium state is stable

30) Write potential energy equation for cantilever beam
= EI/2 (d2y/dx2) dx – pie

31) Mention 2 different methods to approach the model of physical system .
Discrete system , continuum

33) What is local coordinate?
local coordinate contains 1-D.O.F. at each node.

34) What is global co ordinate?
Global coordinate contains 2-D.O.F. at each node.

35) General assumption made in stress
Assumptions:
1.Truss elements are connected by fracture less pin.
2.Load is applied on the load.
3.Only two forces compressive and tensile are considered.

36)What is shape function?
It is mathematical polynomial, which gives displacement within the element.

37)What are two general natural coordinate?
Two general co-ordinates are (eeta) and zeeta.

38) Mention the range of natural co-ordinate.
Range of natural coordinates is – 0 to 1 and –1 to 1.

39) What is the number of shape function in CST?
Number of shape function in CST are three.

40) What is the number of shape function in quadrilateral?
Number of shape function in quadrilateral are 4.

41) Why we are using natural integration?
we are using natural integration to simplify the problem.

42) Explain one point formula.
1-point shape function contains one term f(x)dx=w.f(x)

43) Explain two point formula.
2-point shape function contains two term f(x).dx=w1f(x1) w2f(x2)

44) Why we are using polynomial equation in FEA?
Polynomial equation gives continuous solution & it is simple to solve problem.

45) What are the two important characteristics in stiffness matrix?
Characteristics of stiffness matrix is symmetric and bonded

47) Mention two schemes to represent band width.
2-Schemes are
Horizontal numbering
Vertical numbering

48)What are forces involved in work potential?
Forces involved are Body force, Traction force and Point force

49) What is isoparametric elements?
These are those the S.F. used to define variables of displacement equal to S.F. used to represent geometry.

50) What is orthotropic elements?
Material which has three orthogonal planes of symmetry said to be orthotropic elements. Only nine constants are required to describe constituent equation

51) What is anisotropic elements?
The material which doesn‟t contain any plane of symmetry

52) What is isotropic elements?
isotropic material is one in which every plane is plane of symmetry only two constants are enough to describe constituent equation

53) What is super parametric elements?
GSF > DSF (geometric shape function, displacement shape function
54) What is sub parametric elements
GSF < DSF

55) Different coordinates involved in chain rule.
Different coordinates involved in chain rule are normal , local and displacement coordinates .

56) What are the 2 different approaches to study elasticity ?
Strength of material , Theory of elasticity

57) Mention any two methods to solve continuum problems.
1. Raleighritz method
2. Galerkin method

58) List the properties of shape functions.
SFfor 1D bar element N1=0, N2= 0 at node 1
N1=0, N2= 1 at node 2 .
Diff. Of S.F. S.F are constant

59) Define truss.
Structural member which is subjected to either tensile or compression

60) What is weighted residual methods.
It‟s a method in FEA for accurate solution avoiding error (residue = error)

61) Different methods to solve weighed residual problem.
Point allocation, Sub domain , Galerkein,least square.

62)Explain the principle of virtual work.
If the force and displacement are unrelated by cause effect relation then the work is said to be virtual work.

63) Explain the principle of virtual displacement.
Actual displacement is considered without bothering amount of force is called virtual displacement .

64)What are different types of shape functions.
Diff types are Lagarangian and Hermite Shape function

65) Differentiate two types of shape functions.
Lagarangian shape function only for variable
Hermite shape function is for both variable and its derivative

66) Mention some advantages of FEA over solid mechanics.
1. Applied for complicated structure
2. Analysis is simple
3. More accurate solution

67) Define Young‟s Modulus and Poisson‟s Ratio.
E - It is the ratio of stress and strain
μ -It is the ratio of lateral strain and longitudinal strain

68) Mention different types of elastic constants.
Young‟s modulus, shear modulus, bulk modulus

69)Specify the terms required to solve FEA problem.
Meshing , properties of material, boundry condition and initial condition.

70) What are the assumptions made in linear static problems.
all displacement is small, material is isotrpic, linear, elastic solid with E and

71) Which is the most accepted form of numerical integration in FEM?
Gaussian Quadrature

72)List the different approaches to derive integral equation
Direct method , variation method, weighed residual method, energy method

73)What are the advantages of symmetrical matrix.
symmetrical matrix simplifies the calculation

74) What are the different types of errors in FEA?
Modeling error , Descritized and Numerical error

75) What are the advantages of isoparametric elements
Useful in modeling structure with curved edges
They are versatile & they are used in 2-D and 3-D elasticity problems

76). Define frontal method for finite element matrices.
In 3- dimensional problems, the size of the stiffness matrix increases rapidly even with the banded method of modeling. An alternative direct method that results in considerable saving in the use of computer memory is called frontal method.

77) What is the another name of the 3-dimensional frames
Space frames.

78). Define beam elements.
Beam elements are slender members that are used for supporting transverses loading.

79) Explain preprocessor steps.
Determining the Nodal coordinates, connectivity, boundary condition, material information

80) Explain processing steps.
Stiffness generation, modification, solutions to the equation resulting in evolution.

81) Explain post processing steps.
Deformation confirmation, mode shapes, temperature and stress distribution, interpretation

82) What are the difference b/w beams and plane frames.
It is similar to beams expect that axial loads &axial deformations are present. The elements also have different orientation.

83) Mention some common material properties.
Isotropic, orthrotropic, ductility, brittleness

84) What are the different types of analysis.
Thermal, structural (load), fluid, electromagnetic analysis

85) Define steady state analysis.
The analysis carried out at constant temperature.

86) What is the advantage of subjecting solids to axisymmetric loading.
Axisymmetric loading reduces the 3-D problem into 2-D problems because of total symmetry about the z-axis.

87) Define CST elements.
The constant strain triangle is that where the displacement inside an element is represented by 3-nodal displacement (3-shape functions).

88) How to generate the data files for larger problems in FEA. Using
MESHGEN program generates data files.

89) Define mesh plotting
It is the convenient way of reviewing the coordinate and connectivity data is by plotting it using computer.

90) Explain lumped mass matrices.
It is the total element mass in each direction is distributed equally to the nodes of the element, and the masses are associated with translational degrees of freedom.

91) Briefly explain steps involved in Lagrangian method

Formulation of potential energy function.

Assuming displacement function
Checking displacement function considering boundary condition
Substitute differential function in potential energy equation
Potential energy function is minimized
Unknown parameters are determined and substituted in assumed equation

92) Explain steps involved in Gallerkins method.
Formulate differential equation of the equilibrium
Assume trial function, which satisfies boundary conditions
Substitute displacement function in differential equation then assume the difference due to approx. function be „R" (residue)
Use gallerkin formula
Determine unknown terms and then substitute in differential function

93) Define jacobian matrix.
The matrix which is defined explicitly in terms of the local coordinate is known as jacobian(J).

94) Mention six components of stress.
3 linear stress along x, y, z direction
3 lateral stress along x, y, z direction

95) Mention six components of strain.
3 linear strain along x, y, z direction
3 lateral strain along x, y, z direction

96) Define Winkler foundations.
Large beams are supported on soil form a class of applications known as winkler foundations.

97) Define variational principle
The problem which specifies a scalar quantity potential energy is defined in an integral form.

98) How to solve the prismatic problems.
The coefficient of the ordinary differential equation are independent of one of the coordinate and the solution of the system can frequently carried out efficiently by standard analytical methods.

99) Define stress & strain.
Stress is the ratio of applied load to its area.
Strain is the ratio of change in length to its original length.

100) Mention the two distinct procedures available for obtaining the approximation in the integral forms.
Method weighed residuals. Method
of variation functional.