COMPUTATIONAL STRUCTURAL ANALYSIS LABORARTORY

LAB MANUAL

Course Code	**:**	**AAE111**
Regulations	**:**	**IARE – R16**
Semester	**:**	**VII**
Branch	**:**	**AE**

Prepared by

Ms. Y Shwetha
Assistant Professor, AE

Mr. G S D Madhav
Assistant Professor, AE

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous) Dundigal, Hyderabad - 500 043

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad – 500043

Program Outcomes	
PO1	**Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO2	**Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO3	**Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO4	**Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO5	**Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO6	**The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
PO7	**Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO8	**Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO9	**Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
PO10	**Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO11	**Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
PO12	**Life-long learning**: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.
Program Specific Outcomes (AE)	
PSO1	**Professional skills:** Able to utilize the knowledge of aeronautical/aerospace engineering in innovative, dynamic and challenging environment for design and development of new products
PSO2	**Problem-solving Skills**: Imparted through simulation language skills and general purpose CAE packages to solve practical, design and analysis problems of components to complete the challenge of airworthiness for flight vehicles.
PSO3	**Practical implementation and testing skills:** Providing different types of in house and training and industry practice to fabricate and test and develop the products with more innovative technologies
PSO4	**Successful career and entrepreneurship:** To prepare the students with broad aerospace knowledge to design and develop systems and subsystems of aerospace and allied systems and become technocrats.

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad – 500043

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES			
S No	**Experiment**	**Program Outcome Attained**	**Program Specific Outcomes Attained**
1	Introduction to computational structures	PO1, PO2, PO3	PSO1, PSO3
2	Introduction to ANSYS APDL	PO1, PO2, PO3	PSO1, PSO3
3	APDL user interface, preprocessing, post processing & modeling.	PO1, PO2, PO3	PSO1, PSO3
4	Static Analysis Of 2d & 3d Truss At Various Loading Conditions.	PO2, PO3	PSO1, PSO3
5	Static analysis of different beams under various loading conditions.	PO3, PO4	PSO1, PSO3
6	Static analysis of plate with a cutout for various materials.	PO2, PO3	PSO1, PSO3
7	Modal & transient analysis of beams for various materials.	PO2, PO3	PSO1, PSO3
8	Non-linear analysis of the mild steel beam for large deflection.	PO2, PO3	PSO1, PSO3
9	Harmonic analysis of the spring mass system.	PO3, PO4	PSO1, PSO3
10	Static analysis of aircraft wing structure.	PO3, PO4	PSO1, PSO3
11	Static and modal analysis of main landing gear.	PO2, PO3	PSO1, PSO3
12	Static analysis of composite beam and plate.	PO2, PO3	PSO1, PSO3

COMPUTATIONAL STRUCTURAL ANALYSIS LABORATORY

VII Semester: AE

Course Code	Category	Hours / Week			Credits	Maximum Marks		
AAE111	Core	L	T	P	C	CIA	SEE	Total
		0	0	3	1.5	30	70	100
Contact Classes: Nil	**Tutorial Classes: Nil**	**Practical Classes: 36**				**Total Classes: 36**		

COURSE OBJECTIVES:

The course should enable the students to:

I. Apply the basic principles learnt from pre-requisites subjects to solve the structural problem.
II. Adopt any computational structural analysis software and learns how to perform analysis.
III. Analyze structural problems related to Aerospace industry.
IV. Interpret the results and how to apply them on the real-life structure.

LIST OF EXPERIMENTS

Week-l	**INTRODUCTION AND BASIC FUCTIONS**

a. Starting up of ANSYS/Nastran
b. Description of user interface

Week-2	**STATIC ANALYSIS: TRUSS AND FRAME STRUCTURES**

a. 2-D truss structures
b. 3-D truss structures

Week-3	**STATIC ANALYSIS: BEAMS**

a. Straight beams
b. Tapered beams

Week-4	**STATIC ANALYSIS: TWO DIMENSIONAL PROBLEMS**

a. 2-D structure with various loadings
b. 2-D structures with different materials
c. Plate with hole

Week-5	**DYNAMIC ANALYSIS: MODAL AND TRANSIENT ANALYSES**

a. Modal analysis
b. Transient Response (spring-mass system)

Week-6	**THERMAL ANALYSIS**

a. Bars and beams
b. 2D structures

Week-7	**NON LINEAR ANALYSIS**

a. Nonlinear behavior (Large deflections)
b. Nonlinear behavior (Materials)

Week-8	**HARMONIC RESPONSE ANALYSIS**
a. Random Vibration Analysis of a Deep Simply-Supported Beam b. Harmonic Response of a Spring-Mass System	
Week-9	**ANALYSIS OF AIRCARFT STRUCTURE: WING**
a. Static analysis of Aircraft wing structure b. Modal analysis of aircraft wing structure	
Week-10	**ANALYSIS OF AIRCARFT STRUCTURE:FUSELAGE**
a. Static analysis of Aircraft Semi monoque fuselage structure b. Modal analysis of aircraft Semi monoque fuselage structure	
Week-11	**ANALYSIS OF AIRCARFT STRUCTURE:LANDING GEAR**
a. Static analysis of main landing gear b. Modal analysis of main landing gear	
Week-12	**ANALYSIS OF COMPOSITE STRUCTURES**
a. Static analysis of composite bar and beam b. Static analysis of composite plate	
Text Books:	
1. Huei-Huang Lee, "Finite Element Simulations with ANSYS Workbench 16", SDC publications, 2^{nd} Edition, 2016. 2. Anderson, William J "MSC/Nastran: Interactive Training Program" Wiley 1^{st} Edition 2015. "ANSYS Mechanical APDL Basic Analysis Guide", ANSYS,Inc Release 16.0.	
Web References:	
1. https://www.ansys.com/services/learning-hub 2. https://caeai.com/ansys-software-support/ansys-software/mechanical-simulation-software	

WEEK – 1

INTRODUCTION TO ANSYS

OBJECTIVES:

a. To model of 2D truss and apply proper element type.
b. To perform the static analysis of 2D truss.
c. To study the behavior of truss under the applied loading condition.

RESOURCE:

ANSYS 16.0 Academic

PERFORMING 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.

A typical ANSYS analysis has three distinct steps:

- Build the model.
- Apply loads and obtain the solution.
- Review the results.

BUILDING A MODEL

Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a job name and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.

SPECIFYING A JOB NAME AND ANALYSIS TITLE

This task is not required for an analysis, but is recommended.

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. By using a job name for each analysis, you insure that no files are overwritten. If you do not specify a job name, all files receive the name *FILE* or *file*, depending on the operating system.

Command(s): /FILNAME

GUI: Utility Menu>File>Change Job name

DEFINING ELEMENT TYPES

The ANSYS element library contains more than 100 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

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.

DEFINING MATERIAL PROPERTIES

Most element types require material properties. Depending on the application, material properties may be:

- Linear or nonlinear
- Isotropic, orthotropic, or anisotropic
- Constant temperature or temperature-dependent.

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.

APPLYING LOADS

The word loads as used in this manual 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 (key points, lines, and areas) or the finite element model (nodes and elements). Two important load-related terms you need to know are load step and sub-step. A load step is simply a configuration of loads for which you obtain a solution. In a structural analysis, for example, you may apply wind loads in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

PRE LAB VIVA QUESTIONS:

WEEK – 2

STATIC ANALYSIS OF TWO DIMENSIONAL TRUSS

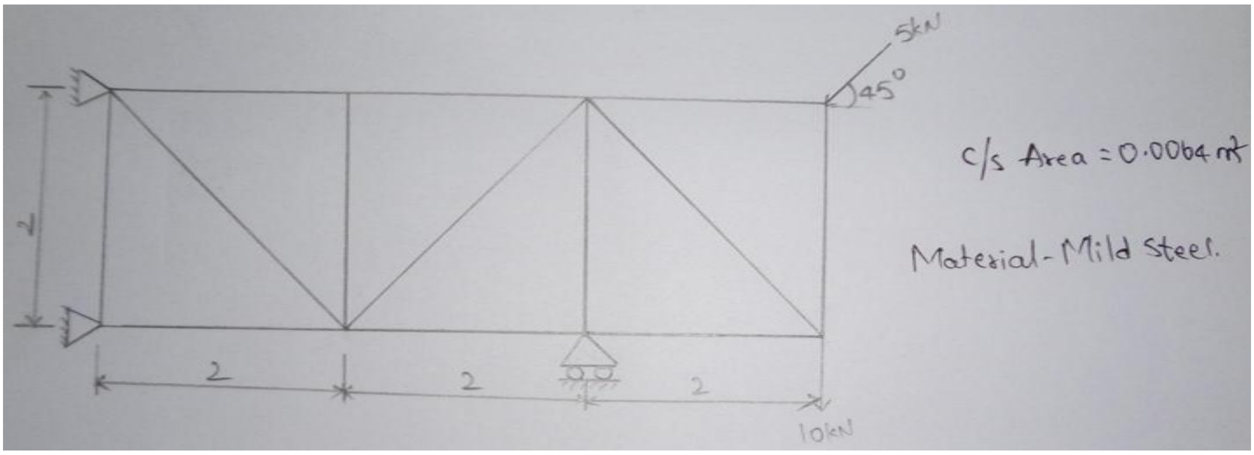
OBJECTIVES:

a. To model of 2D truss and apply proper element type.
b. To perform the static analysis of 2D truss.
c. To study the behavior of truss under the applied loading condition.

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:



PROCEDURE:

Preferences: Structural

Preprocessor:

- Element → Add → Link → 2D Spar 1 → Ok
- Real constants → Add Set1 → Cross section area = 0.0064 m^2 → Ok
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2.5e11 → PRXY = 0.28 → Ok
- Modeling → Create → nodes → Inactive CS → (0,0);(2,0);(4,0);(6,0); (0,2);(2,2);(4,2);(6,2) → Ok → Elements → Auto numbered → Thru nodes → Join nodes with lines → Ok

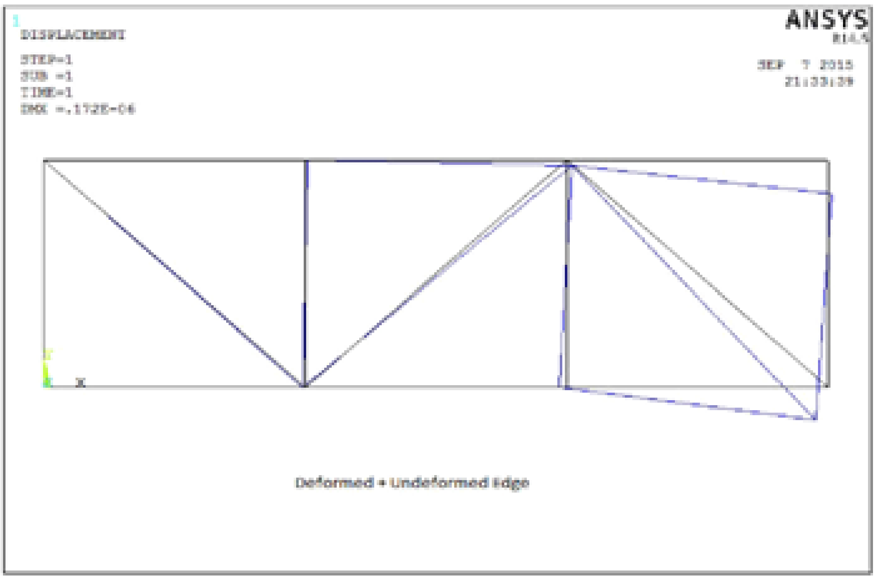
Solution:

- Analysis Type → New analysis → Static → Ok
- Define Loads → Apply → Structural → Displacement → On nodes → Ok → All DOF Constrained → Ok → UY DOF constrained → Ok →Force/Moments → On nodes → FY = -10 → apply → FY = -3.5 → apply → FX = -3.5 → Ok
- Solve → Current LS → Ok

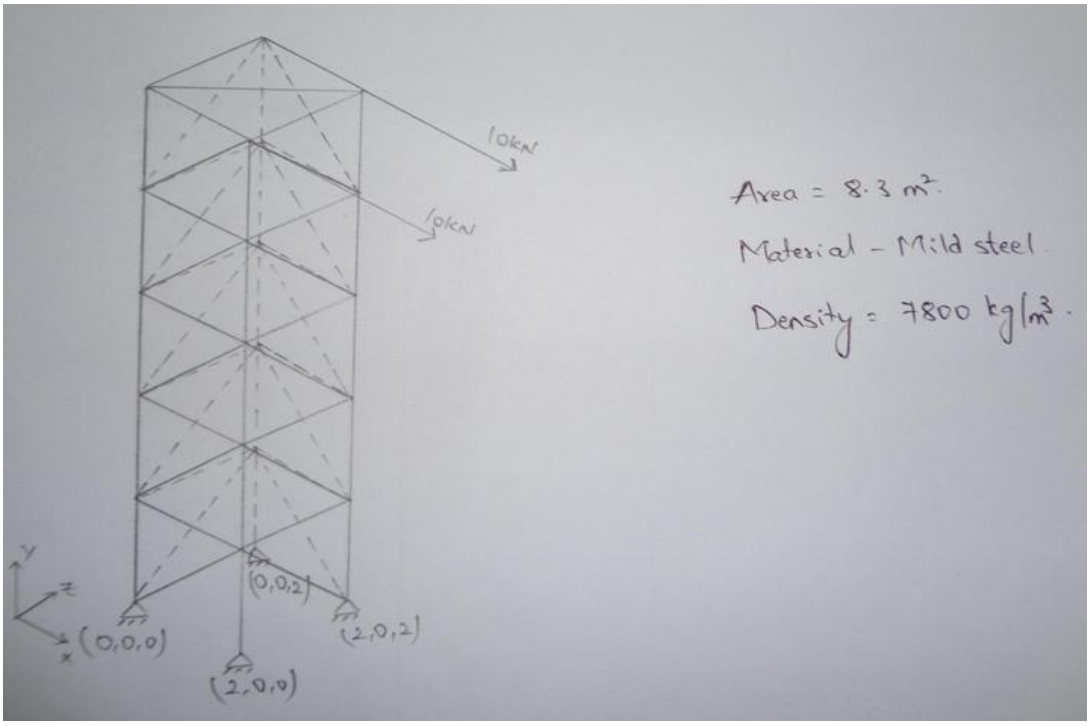
General Post Proc :

- Plot results → Deformed Shapes → Deformed + Un deformed

RESULTS:



Problem Description:



PROCEDURE:
Preferences: Structural

Preprocessor:

- Element → Add → Link → Spar 8 → Ok
- Real constants → Add Set1 → Cross section area = 8.3 m^2 → Ok
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2.5e11 → PRXY = 0.28 → Ok

- Modeling → Create → nodes → Inactive CS → (0,0,0);(2,0,0);(2,0,2);(0,0,2) → Ok → Copy → nodes → Copy → Select Nodes → y=2 → Ok
- Copy → nodes → Select 2nd set → y=2 → Ok → Copy → nodes → Select 3rd set → y=2 → Ok →Copy → nodes → Select 4th set → y=2 → Ok

- Modeling → Elements → Auto numbered → Thru nodes → Join nodes with lines → Ok

Solution:

- Analysis Type → New analysis → Static Ok

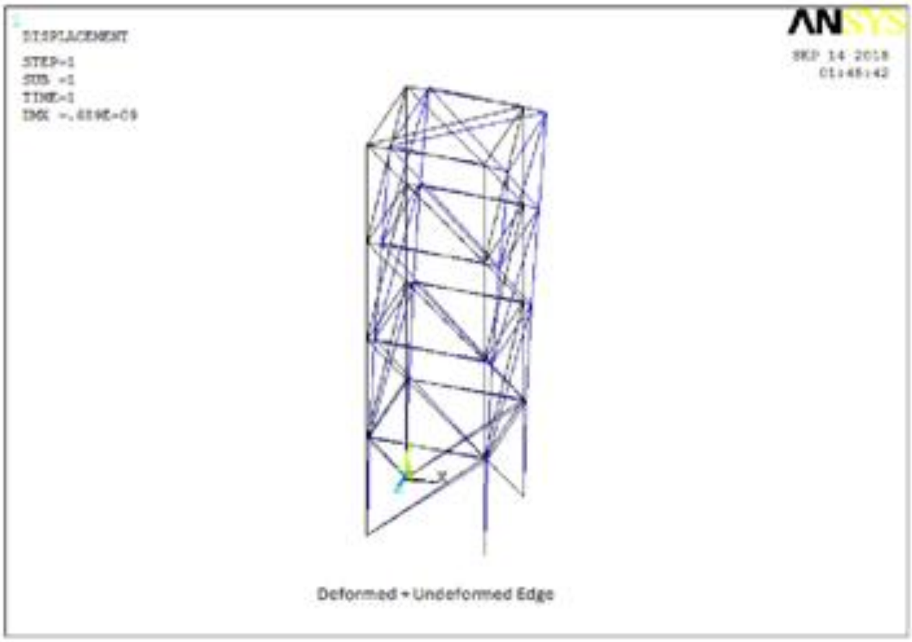
- Define Loads → Apply→ Structural → Displacement → On nodes → Ok → All DOF Constrained→ Ok →Force/Moments→ On nodes FX = 10 apply→ Ok

- Solve Current LS Ok

General Post Proc:

- Plot results Deformed Shapes Deformed + Un deformed

RESULTS:

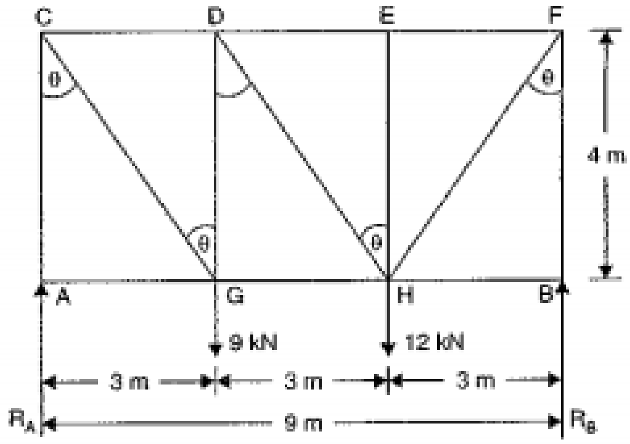


PRE LAB VIVA QUESTIONS:

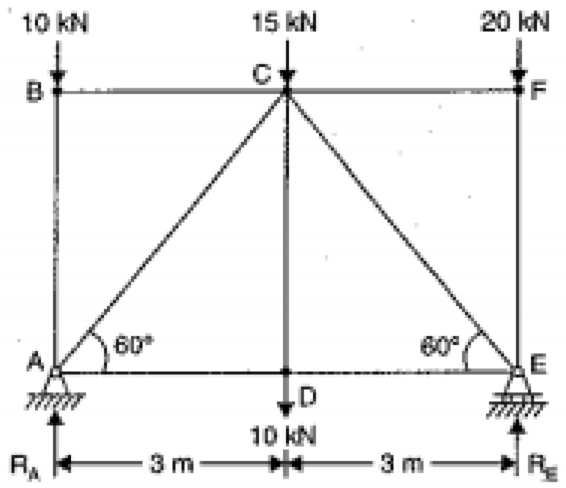
a. What is Truss?
b. Which type of loads does the truss take?
c. What are methods to solve the truss analytically?
d. Differentiate hinge supports and roller supports.

LAB ASSIGNMENT:

e. A truss of span 9m is loaded as shown in figure. Find the deflection of truss.



f. A truss is as shown in figure. Find the forces in all members of the truss and indicate it is in tension or compression.



POST LAB VIVA QUESTIONS:

a. How do you select the element type in static analysis?
b. Define elements and nodes?
c. What type of analysis is done to solve the problem?

WEEK – 3

STATIC ANALYSIS OF BEAMS

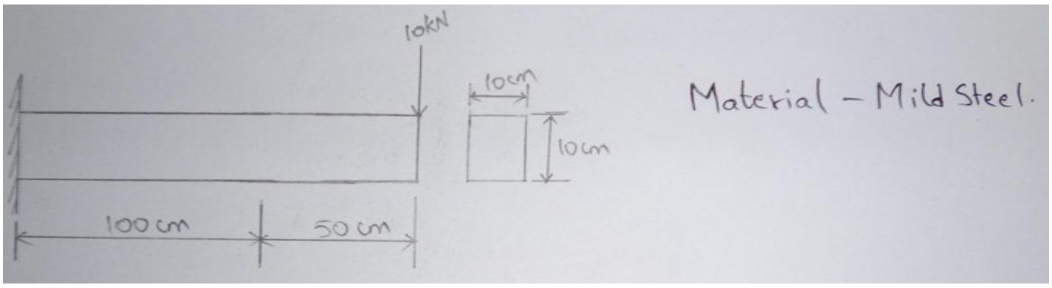
OBJECTIVE:

a. Understand the ANSYS APDL working plane.
b. Write a Python script to print the Fibonacci series using functions.
c. Write a Python script to find the GCD of two numbers.

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTON:



PROCEDURE:

Preferences : Structural

Preprocessor :

- Element → Add → Beam3
- Real constants → Add Set1 → Cross section area = 100 cm^2 → Izz = 833.33 cm^4 → Height = 10 cm → Ok → Close
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2.5e7 → PRXY = 0.28 → Ok
- Modeling → Create → Key points → Inactive CS → (0,0,0);(100,0,0);(150,0,0) → Ok → Lines → Straight Lines → Ok
- Meshing → Size Controls → Manual Sizing → Lines → Picked Lines → No. of elements = 20 → Ok → Mesh → Lines → Ok

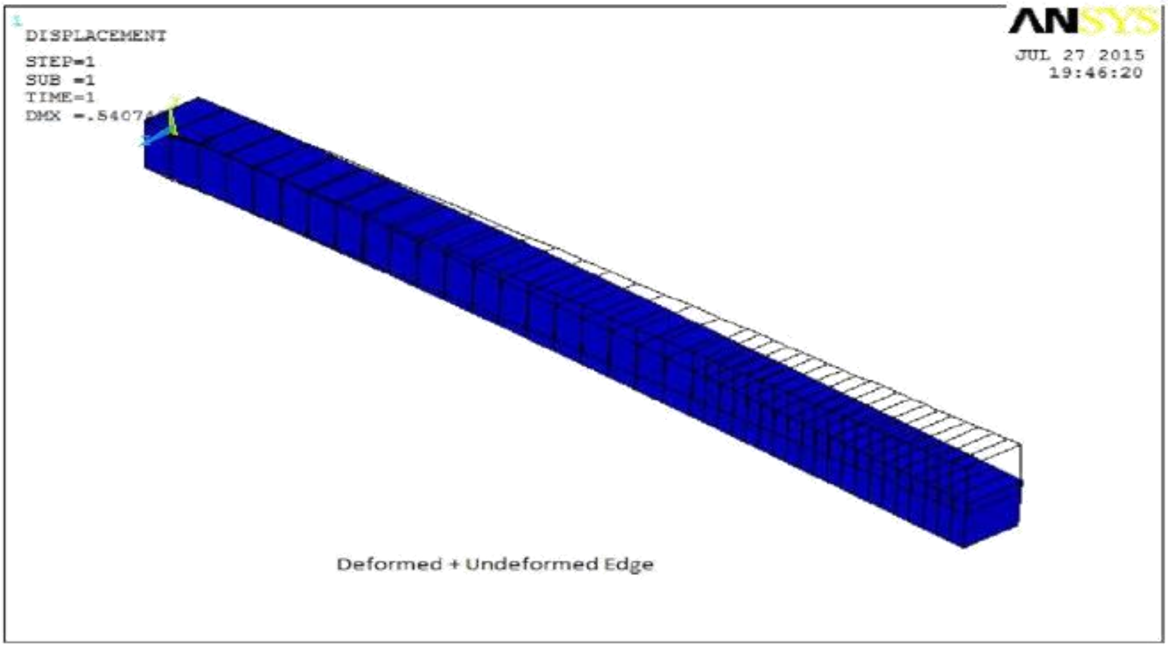
Solution :

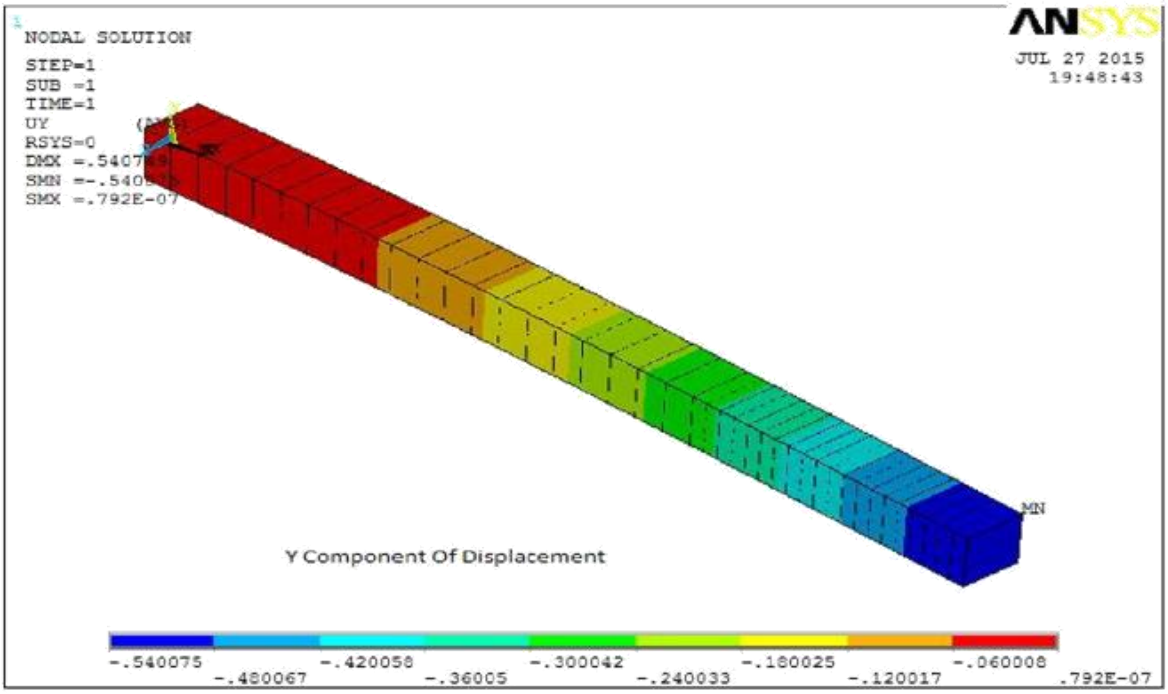
- Analysis Type → New analysis → Static → Ok
- Define Loads → Apply → Structural → Displacement → On Key Points → Ok → All DOF Constrained → Ok → Force/Moments → On Key Points → FY = - 10000 → Ok
- Solve → Current LS → Ok

General Post Proc :

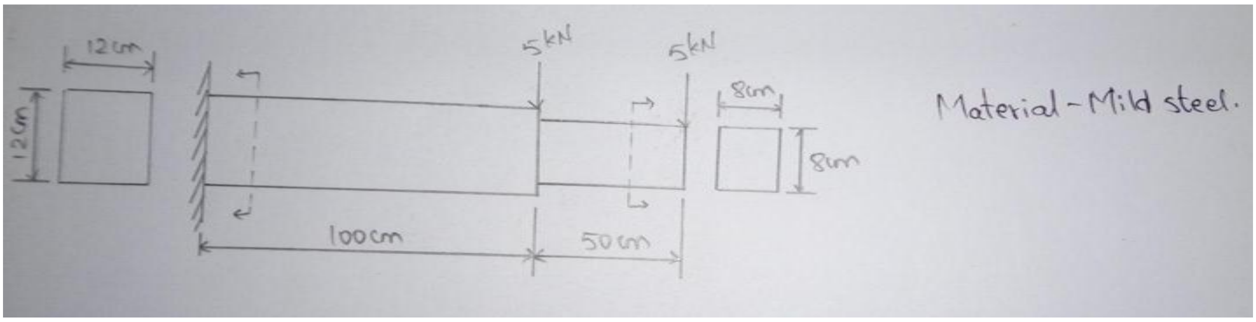
- Plot results → Deformed Shapes → Deformed + Un deformed
- Contour Plots → Nodal Solutions → DOF Solution → Y- Component Displacement Ok

RESULTS:





PROBLEM DESCRIPTION:



PROCEDURE:

Preferences : Structural

Preprocessor :

- Element → Add → Beam3
- Real constants → Add Set1 → Cross section area = 144 cm^2 → Izz = 1728 cm^4 → Height = 12 cm → Ok → Add Set2 → Cross section area = 64 cm^2 → Izz = 341.33 cm^4 → Height = 8 cm → Ok → Close
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2.5e7 → PRXY = 0.28 → Ok
- Modeling → Create → Key points → Inactive CS → (0,0,0);(100,0,0);(150,0,0) →Ok → Lines → Straight Lines → Ok

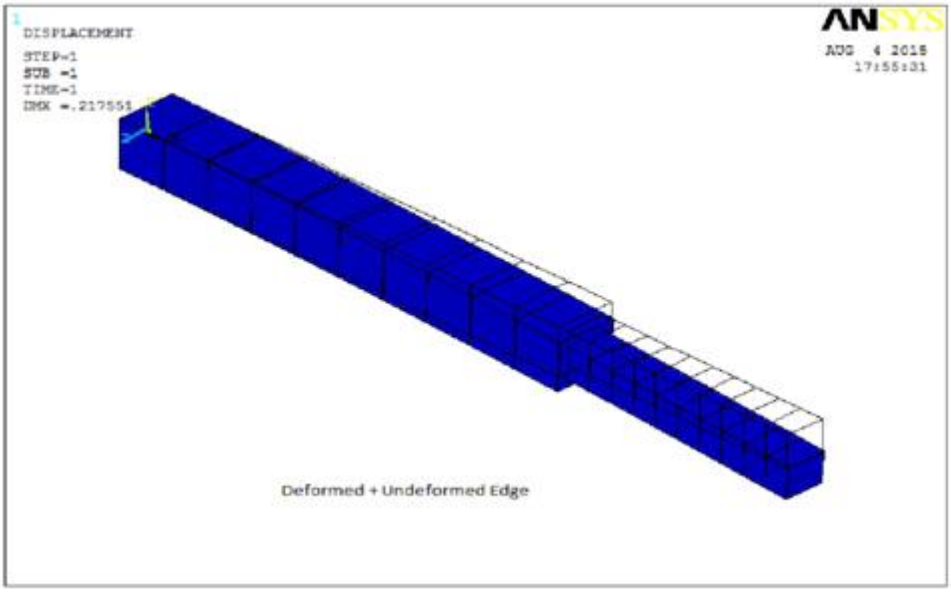
→ Meshing → Mesh Attributes → Picked Lines → Ok → Real Constant set No.1 → Ok → Picked Lines → Ok → Real Constant set No.2 → Ok → Size Controls → Manual Sizing → Lines → Picked Lines → No. of elements = 10 → Ok → Mesh →Lines → Ok

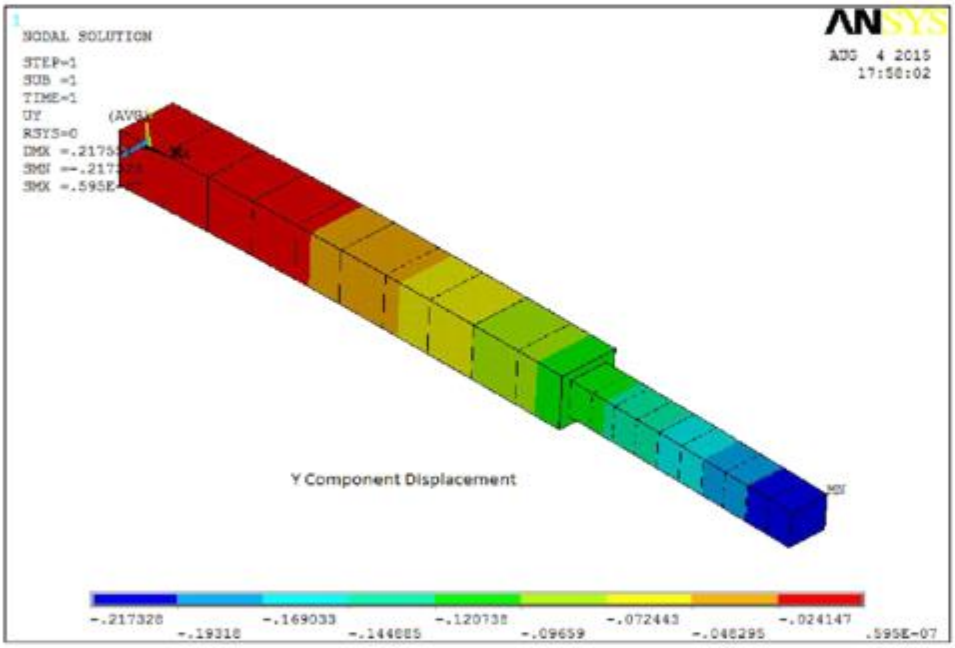
Solution:

- Analysis Type → New analysis → Static → Ok
- Define Loads → Apply → Structural → Displacement → On Key Points → Ok → All DOF Constrained → Ok → Force/Moments → On Key Points → FY = - 5000 → Ok
- Solve → Current LS → Ok

General Post Proc:

- Plot results → Deformed Shapes → Deformed + Un deformed
- Contour Plots → Nodal Solutions → DOF Solution → Y- Component Displacement → Ok



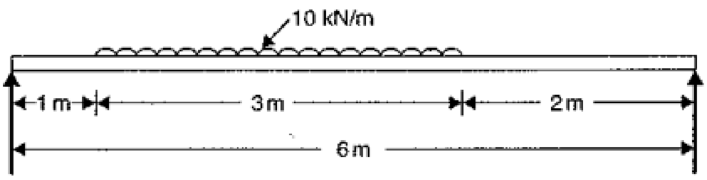


PRE LAB VIVA QUESTIONS:

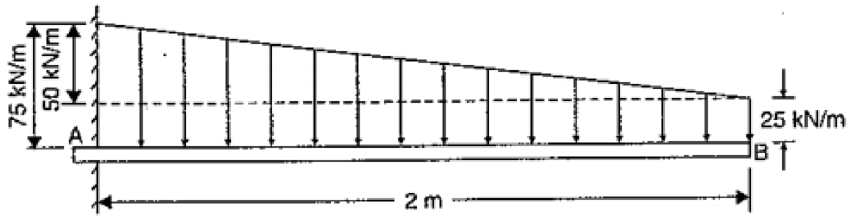
a. What is beam?
b. Which type of loads does the beam can carry?
c. What are the different type of beams?
d. What is slope?
e. Define bending moment.

LAB ASSIGNMENT:

d. A beam of length 6m is simply supported at its ends. It carries a uniformly distributed load of 10kN/m as shown in the figure. Determine the deflection of the beam at its mid-point and also the position and maximum deflection.



e. A cantilever of length 2m carries a uniformly distributed load of 2m carries a uniformly varying load of 25kN/m at the free end to 75 kN/m at the fixed end. If $E=1\times10^5 N/mm^2$ and $I=10^6$ mm^4, determine the deflection of the cantilever at the free end.



POST LAB VIVA QUESTIONS:

f. How many degrees of freedom does the support have for SSB?

g. What kind of mesh technique is used for meshing the beam?

h. Differentiate uniformly distributed load and uniformly varying load.

WEEK – 4

STATIC ANALYSIS OF TWO DIMENSIONAL STRUCTURES

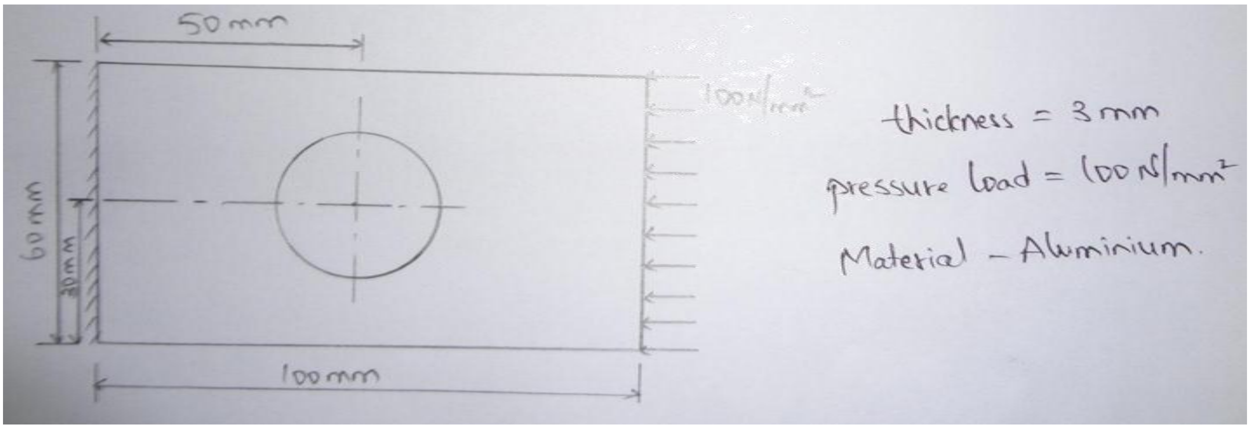
OBJECTIVES:

a. To model the 2dimensional structure in ANSYS software.
b. To perform the static analysis on 2 dimensional structure and study its behavior.

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:



PROCEDURE:

Preferences: Structural

Preprocessor:

- Element → Add → Solid → Quad 4 node 42 → Ok → Options → Element Behavior = Plane stress w/thk → Ok
- Real constants → Add Set1 → THK = 3 mm → Ok → Close
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 0.7e5 → PRXY = 0.32 → Ok
- Modeling → Create → Areas → Rectangle → By Centre and Corner → PX = 0,PY = 0, Width = 50, Height = 30 → Ok → Circle → Two end pts →(-10,0),(10,0) → Ok
- Operate → Booleans → Subtract → Select Area → Ok
- Meshing → Size Controls → Manual Sizing → Areas → All Areas → Element Edge Length = 5 → Ok → Mesh → Areas → Free → Ok

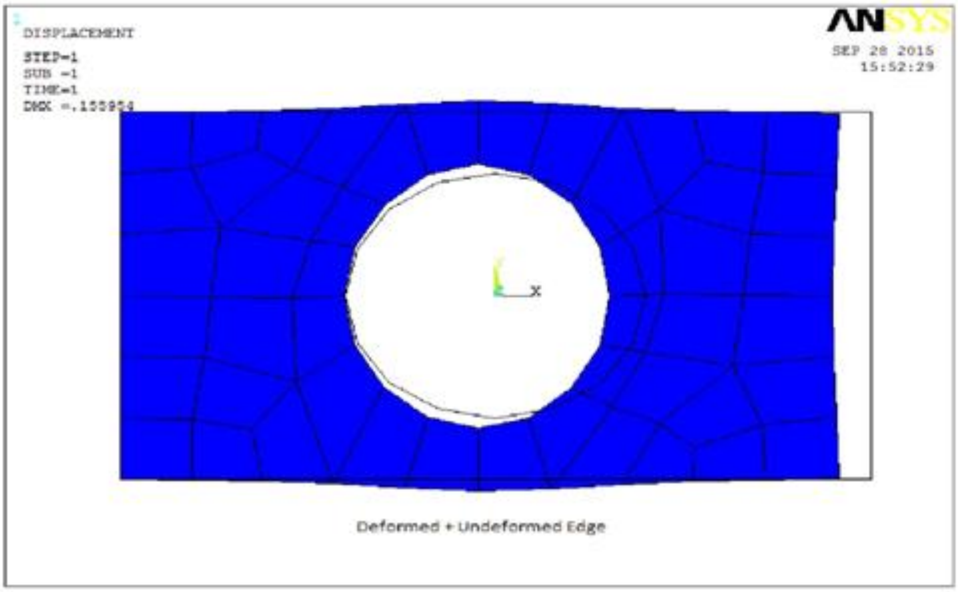
Solution:

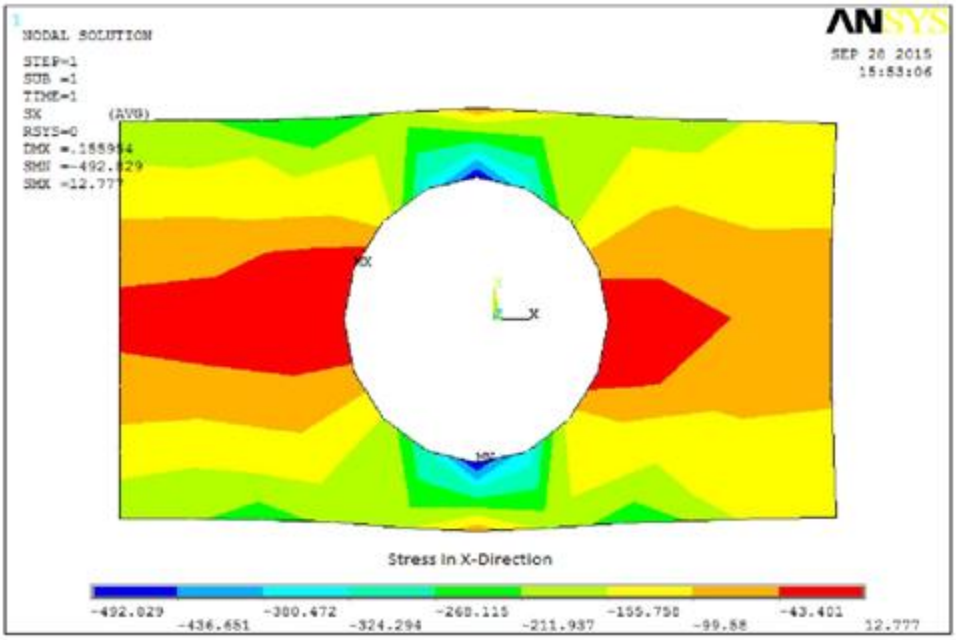
- Analysis Type → New analysis → Static → Ok
- Sol'n Controls → Automatic Time Stepping = On → No. of Sub-steps = 5 → Max. No. of Sub-steps = 1000 → Min. No of Sub-steps = 1 → Ok
- Define Loads → Apply → Structural → Displacement → On lines → Ok → All DOF Constrained → Ok → Pressure → On lines → 100 N/(mm^2) → Ok
- Solve → Current LS → Ok

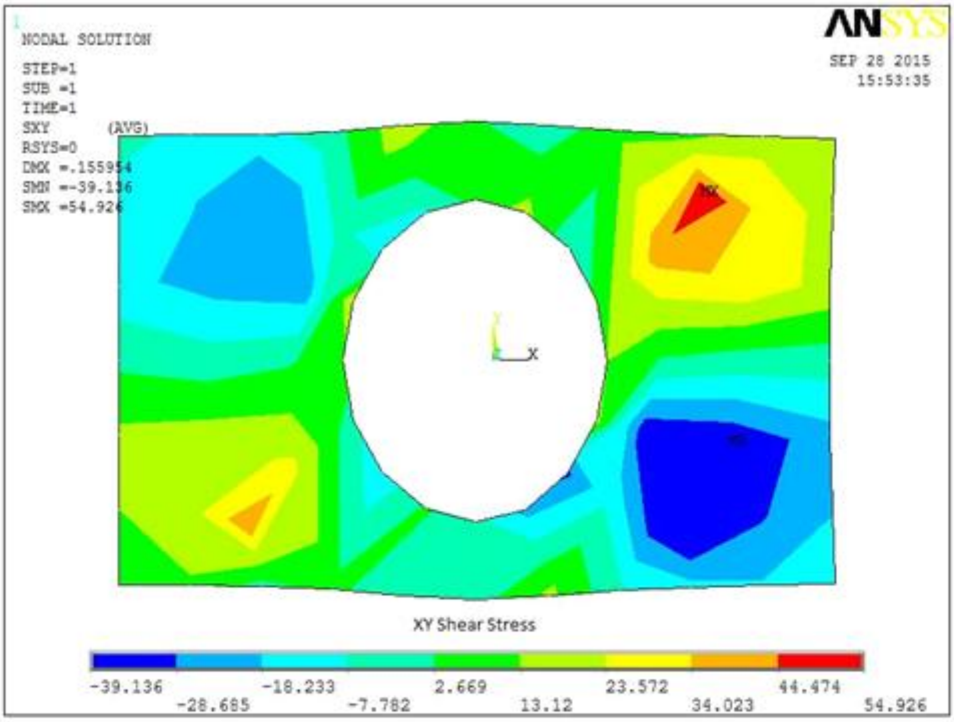
General Post Proc:

- Plot results → Deformed Shapes → Deformed + Un deformed
- Contour Plots → Nodal Solution → Stress → X Comp. → Ok
- Contour Plots → Nodal Solution → Stress → XY Shear → Ok
- Contour Plots → Nodal Solution → Stress → Von Mises Stress → Ok

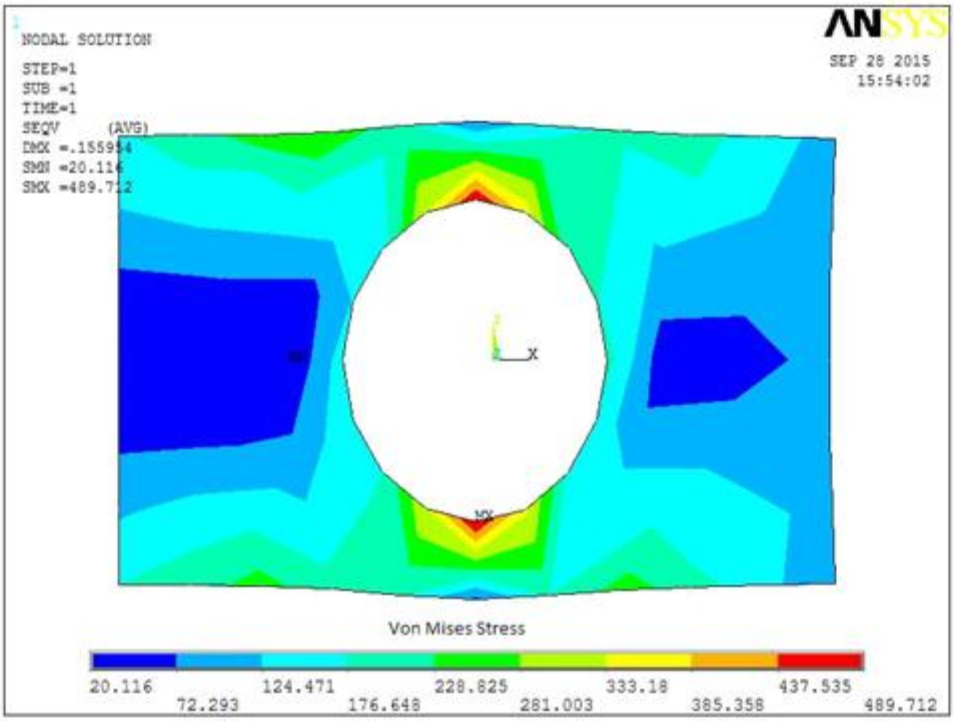
Results:







NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SXY (AVG)
RSYS=0
DMX =.155954
SMN =-39.136
SMX =54.926
ANSYS
SEP 28 2015
15:53:35
XY Shear Stress
-39.136
-28.685
-18.233
-7.782
2.669
13.12
23.572
34.023
44.474
54.926



NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (AVG)
DMX =.155954
SMN =20.116
SMX =489.712
ANSYS
SEP 28 2015
15:54:02
Von Mises Stress
20.116
72.293
124.471
176.648
228.825
281.003
333.18
385.358
437.535
489.712

PRE LAB VIVA QUESTIONS:

a. Define stress concentration.

b. What are different types of loads acting on a plate?

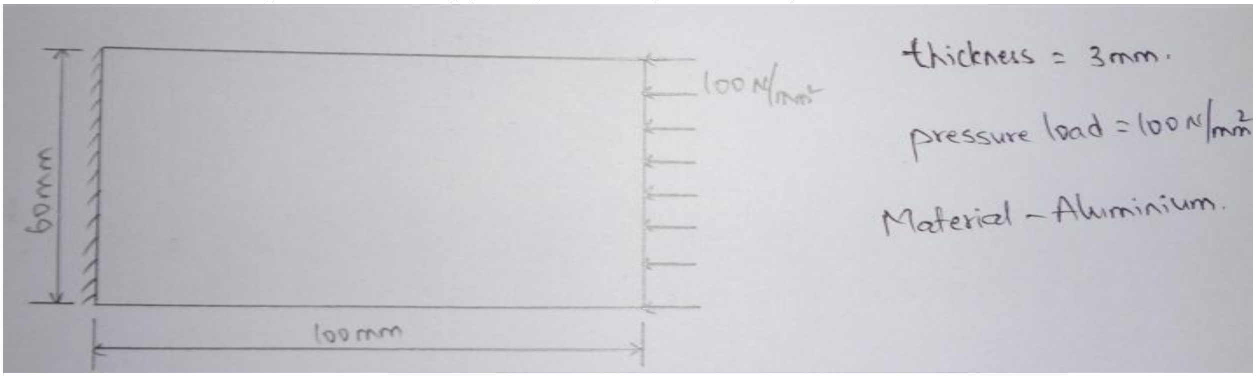
c. Explain pure bending of thin plates.

d. How does the bending moment vary in a plate with simply supported edge?

e. Name different two dimensional elements.

LAB ASSIGNMENT

a. Determine the stresses developed in following plate performing static analysis



POST LAB VIVA QUESTIONS:

a. Define stress concentration.

b. What are Boolean operations?

c. What are different types of loads acting on a plate?

d. What are Von-misses stresses?

e. Explain the elemental properties of Quad 4 node 42.

WEEK – 5

MODAL ANALYSIS OF A BEAM

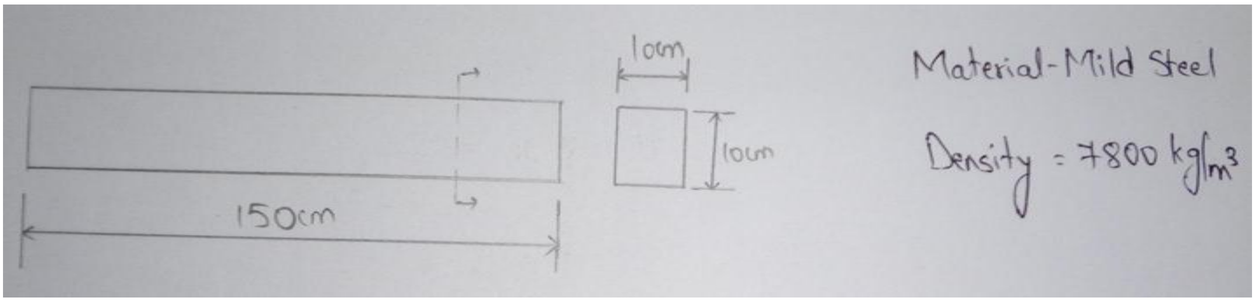
OBJECTIVES:

a. To learn how to perform modal analysis of a beam using Ansys software.
b. To determine the natural frequency of the beam for free-free condition.

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:



PROCEDURE:

Preferences: Structural

Preprocessor:

- Element → Add → Beam3
- Real constants → Add Set1 → Cross section area = 100 cm^2 → Izz = 833.33 cm^4 → Height = 10 cm → Ok → Close
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2.5e7 → PRXY = 0.28 → Density = 7800kg/(m^3) →Ok
- Modeling → Create → Key points → Inactive CS → (0,0,0);(100,0,0);(150,0,0) → Ok → Lines →Straight Lines → Ok
- Meshing → Size Controls → Manual Sizing → Lines → Picked Lines → No. of elements = 10 → Ok → Mesh → Lines → Ok

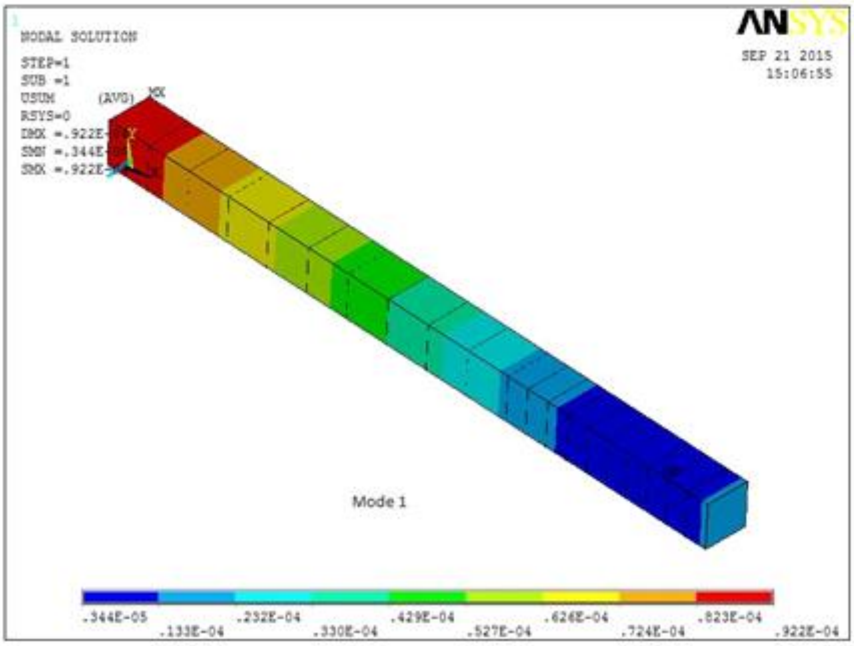
Solution:

- Analysis Type → New analysis → Modal → Ok
- Analysis Option → Block Lancoz → No. of Modes to extract = 5 → No. of Modes to expand = 5 → Ok
- Solve → Current LS → Ok

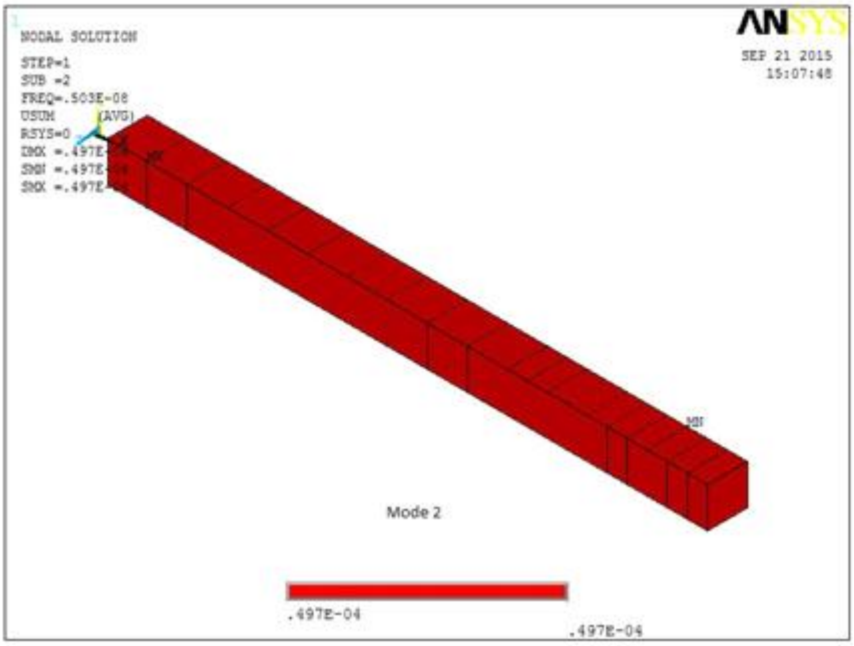
General Post Proc:

- Plot results → Deformed Shapes → Deformed + Un deformed

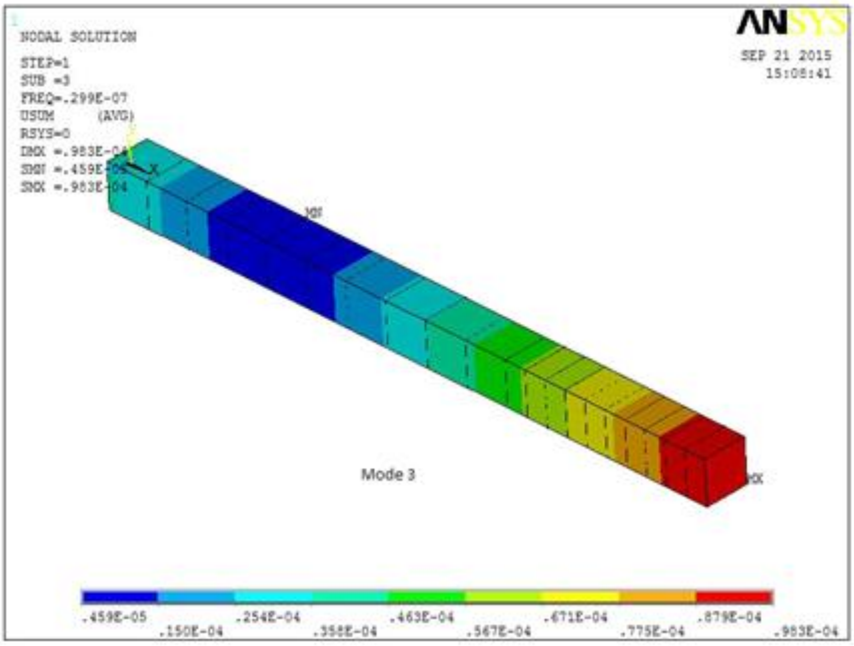
Results:

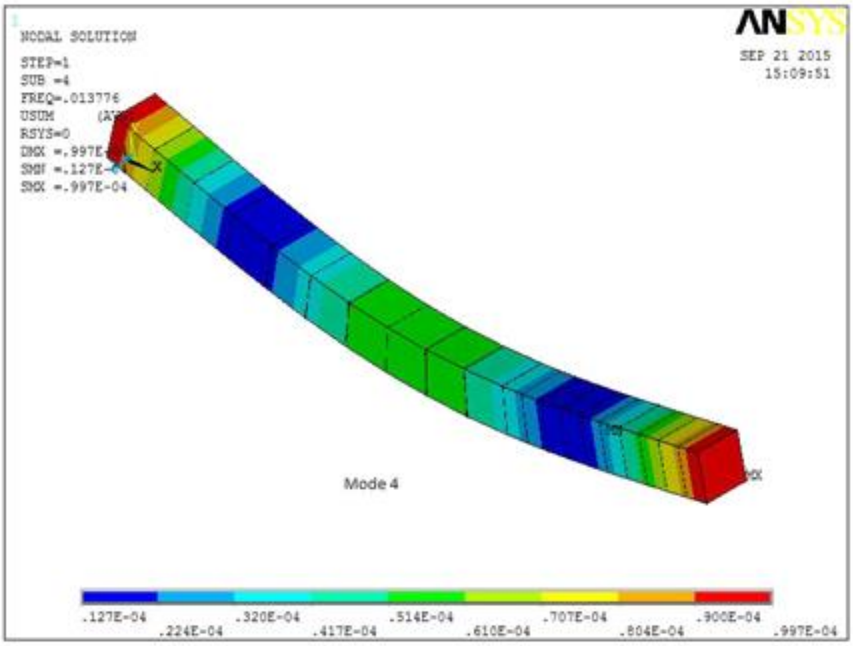


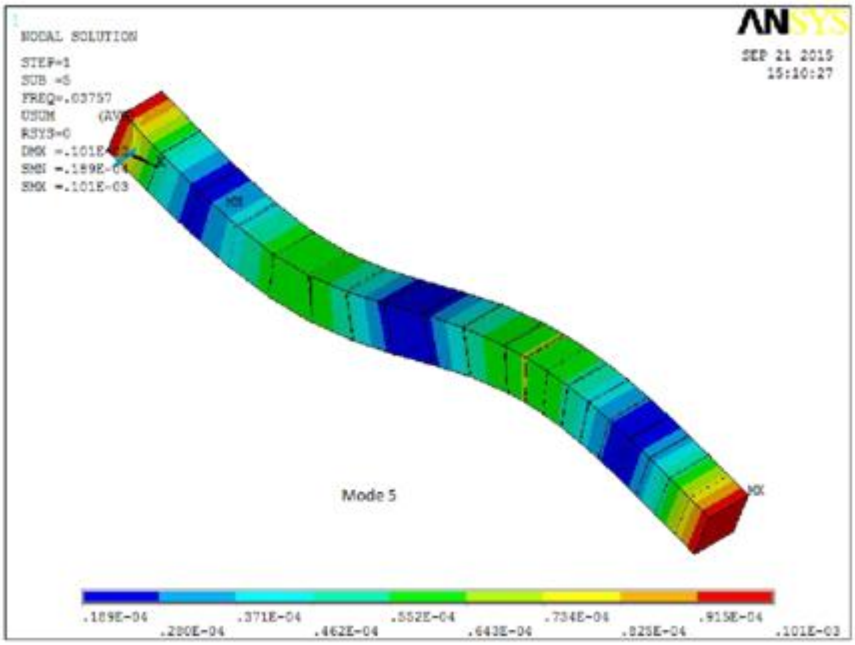
NODAL SOLUTION
STEP=1
SUB =1
USUM (AVG)
RSYS=0
SEP 21 2015
15:06:55
Mode 1
.344E-05
.133E-04
.232E-04
.330E-04
.429E-04
.527E-04
.626E-04
.724E-04
.823E-04
.922E-04



NODAL SOLUTION
STEP=1
SUB =2
FREQ=.503E-08
USUM (AVG)
RSYS=0
SEP 21 2015
15:07:48
Mode 2
.497E-04
.497E-04


NODAL SOLUTION
STEP=1
SUB =3
FREQ=.299E-07
USUM (AVG)
RSYS=0
DMX =.983E-04
SMN =.459E-05
SMX =.983E-04
ANSYS
SEP 21 2015
15:08:41
Mode 3
.459E-05
.150E-04
.254E-04
.358E-04
.463E-04
.567E-04
.671E-04
.775E-04
.879E-04
.983E-04


NODAL SOLUTION
STEP=1
SUB =4
FREQ=.013776
USUM
RSYS=0
SMX =.997E-04
ANSYS
SEP 21 2015
15:09:51
Mode 4
.127E-04
.224E-04
.320E-04
.417E-04
.514E-04
.610E-04
.707E-04
.804E-04
.900E-04
.997E-04



PRE-LAB VIVA QUESTIONS:

a. Define the term natural frequency.

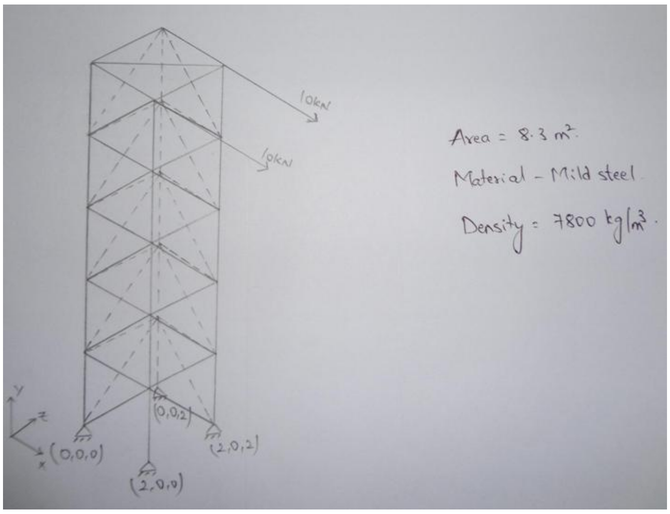
b. Classify various types of vibrations.

c. What is mathematical model?

d. What are different parts of basic spring-mass system?

LAB ASSIGNMENT:

e. Determine the natural frequencies of the following 3D truss.



POST-LAB VIVA QUESTIONS:

a. What are the h and p versions of finite element method?

b. What is difference between static and dynamic analysis?

c. What is CST element?

d. Explain about Block Lancoz method.

WEEK – 6

HARMONIC ANALYSIS

OBJECTIVES:

a. To perform the time variant analysis of a cantilever beam.
b. To interpret the results of the transient analysis and study the behavior of the beam.

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:

PROCEDURE:

Preferences: Structural

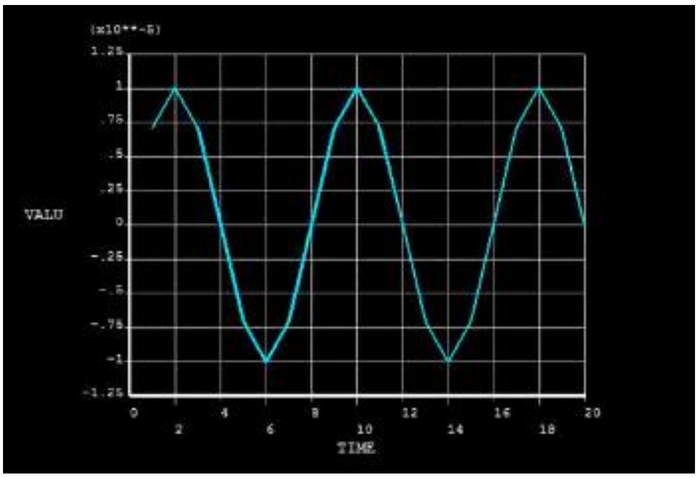
Preprocessor:

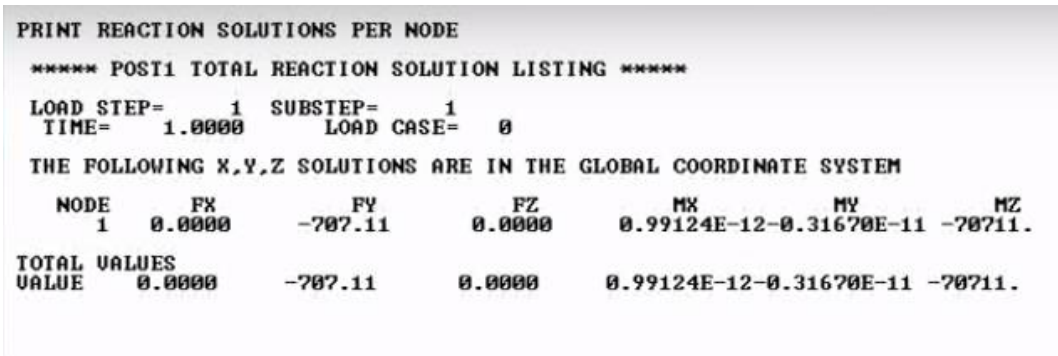
- Element → Add → Beam 3
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2e11 → PRXY = 0.33 → Density = 7850kg/(m^3) →Ok
- Modeling → Create → Key points → Inactive CS → (0,0,0);(100,0,0) → Ok → Lines → Straight Lines → Ok
- Meshing → Size Controls → Manual Sizing → Lines → Picked Lines → No. of elements = 20 → Ok → Mesh → Lines → Ok
- Select parameters → select functions define

Solution:

- Analysis Type → New analysis →Transient → Ok
- Parameters →functions→ define/edit→ type in result= 1000 k sin({pi}/4*{time})
- Select file→ file name= transient→ desktop→ save
- Parameters→ functions→ read from file→ open transient→ give table parameter name cantilever
- Select loads→ define loads→ apply→ structural→ displacement→ on keypoints→ all DOF→ keypoint 1→ ok
- Solve → Current LS → Ok

Results





```
PRINT REACTION SOLUTIONS PER NODE

 ***** POST1 TOTAL REACTION SOLUTION LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0

 THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

   NODE      FX          FY          FZ          MX          MY          MZ
      1   0.0000     -707.11      0.0000     0.99124E-12-0.31670E-11 -70711.

TOTAL VALUES
VALUE    0.0000     -707.11      0.0000     0.99124E-12-0.31670E-11 -70711.
```

PRE-LAB VIVA QUESTIONS:

a. What are damped vibrations?
b. What is difference between damped and undamped vibrations?
c. What happens to the response of an undamped system at resonance?
d. What is self excited vibration?
e. What assumptions are made about the motion of a forced vibration with non-viscous damping in finding the amplitude?
f. Why is viscous damping used in most cases rather than other types of damping?

LAB ASSIGNMENT:

a. Compute the total response of spring-mass system with the values given below and plot the response for 10 seconds

Parameter	Value
k	1000 N/m
m	10 kg
F_0	100 N
ω	8.16rad/s
x_0	0.01m

v_0	0.01m/s

POST-LAB VIVAQUESTIONS:

a. How do you find the response of a viscously damped system?

b. What assumptions are made about the motion of a forced vibration with non-viscous damping in finding the amplitude?

c. What happens to the response of an undamped system at resonance?

d. What is the frequency of the response of a viscously damped system when the external force is $F_0 \sin \omega t$? Is this response harmonic?

e. What assumptions are made about the motion of a forced vibration with non-viscous damping in finding the amplitude?

f. Why is viscous damping used in most cases rather than other types of damping?

WEEK – 7

ANALYSIS OF AIRCRAFT WING STRUCTURE

OBJECTIVES:

a. To perform the static structural analysis of a aircraft wing.
b. To study the shear stress distributions over the wing surface.

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:

- Perform the static analysis of aircraft wing of span 550m and taper ratio 0.4.

PROCEDURE:

Preferences: Structural

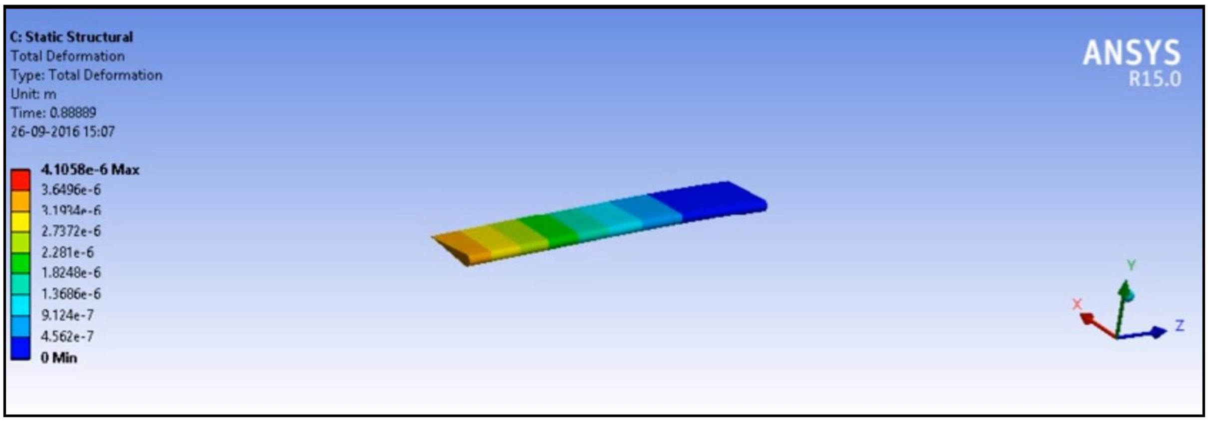
Preprocessor:

- Element → Add → Beam 3
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2e11 → PRXY = 0.33 → Density = 7850kg/(m^3) →Ok
- Modeling → Create → Key points → Inactive CS → (0,0,0);(100,0,0) → Ok → Lines → Areas → Ok
- Meshing → Size Controls → Manual Sizing → Lines → Picked Lines → No. of elements = 20 → Ok → Mesh → Lines → Ok
- Select parameters → select functions define

Solution:

- Analysis Type → New analysis →Transient → Ok
- Parameters →functions→ define/edit→ type in result= 1000 k sin({pi}/4*{time})
- Select file→ file name= transient→ desktop→ save
- Parameters→ functions→ read from file→ open transient→ give table parameter name cantilever
- Select loads→ define loads→ apply→ structural→ displacement→ on keypoints→ all DOF→ keypoint 1→ ok
- Solve → Current LS → Ok

Results



PRE-LAB VIVA QUESTIONS:

a. What is taper ratio?
b. What are the different types of wings? classify

LAB ASSIGNMENT:

a. Perform the aircraft wing modal analysis of span 300 m and taper ratio 0.5 and study its behavior.

POST-LAB VIVA QUESTIONS:

a. What are the types of meshing we have in Ansys workbench?
b. Which type of mesh is used to solve wing?

WEEK – 8
ANALYSIS OF AIRCRAFT WING STRUCTURE

OBJECTIVES:

a. To perform the static structural analysis of a aircraft wing.
b. To study the shear stress distributions over the wing surface.

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:

- Perform the static analysis of aircraft wing of span 550m and taper ratio 0.4.**PROCEDURE:**

Preferences: Structural

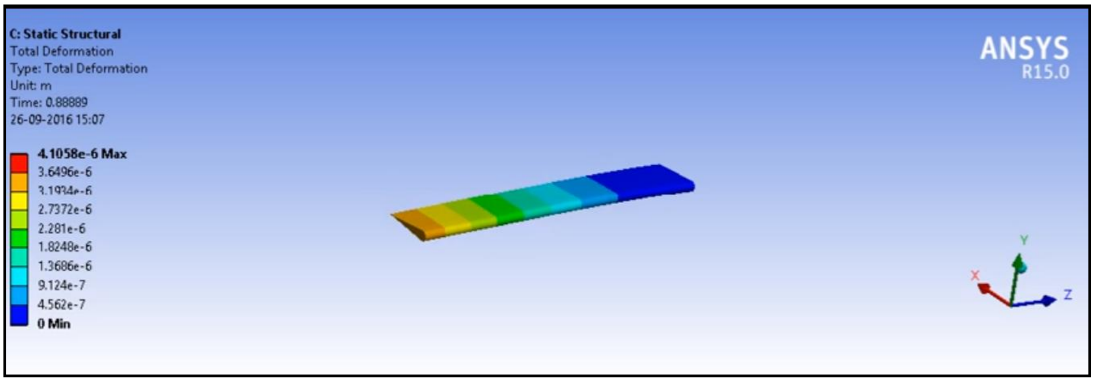
Preprocessor:

- Element → Add → Beam 3
- Material Properties → Material Models → Structural → Linear → Elastic → Isotropic → Ex = 2e11 → PRXY = 0.33 → Density = 7850kg/(m^3) →Ok
- Modeling → Create → Key points → Inactive CS → (0,0,0);(100,0,0) → Ok → Lines → Areas → Ok
- Meshing → Size Controls → Manual Sizing → Lines → Picked Lines → No. of elements = 20 → Ok → Mesh → Lines → Ok
- Select parameters → select functions define

Solution:

- Analysis Type → New analysis →Transient → Ok
- Parameters →functions→ define/edit→ type in result= 1000 k sin({pi}/4*{time})
- Select file→ file name= transient→ desktop→ save
- Parameters→ functions→ read from file→ open transient→ give table parameter name cantilever
- Select loads→ define loads→ apply→ structural→ displacement→ on keypoints→ all DOF→ keypoint 1→ ok
- Solve → Current LS → Ok

Results



PRE-LAB VIVA QUESTIONS:

a. What is taper ratio?
b. What are the different types of wings? classify

LAB ASSIGNMENT:

b. Perform the aircraft wing modal analysis of span 300 m and taper ratio 0.5 and study its behavior.

POST-LAB VIVA QUESTIONS:

a. What are the types of meshing we have in Ansys workbench?
b. Which type of mesh is used to solve wing?

WEEK – 9
ANALYSIS OF LANDING GEAR

OBJECTIVE:

RESOURCE:
ANSYS 16.0 Academic

PROBLEM DESCRIPTION:
- Analyze the landing gear structure with applied load of 10000N

PROCEDURE:
Preferences ► Structural ► H-Method ► OK

Preprocessor ► Element Type ► Add ► Add ► Select Link ► 2D spar 1 ► Apply Preprocessor ► Element Type ► Add ► Add ► Select Beam ► 2 Node 188 ► OK ► Close Real Constants ► Add ► Add ► Select Type Link 1 ► Click OK

Enter the cross sectional area =1 ► OK ► Close

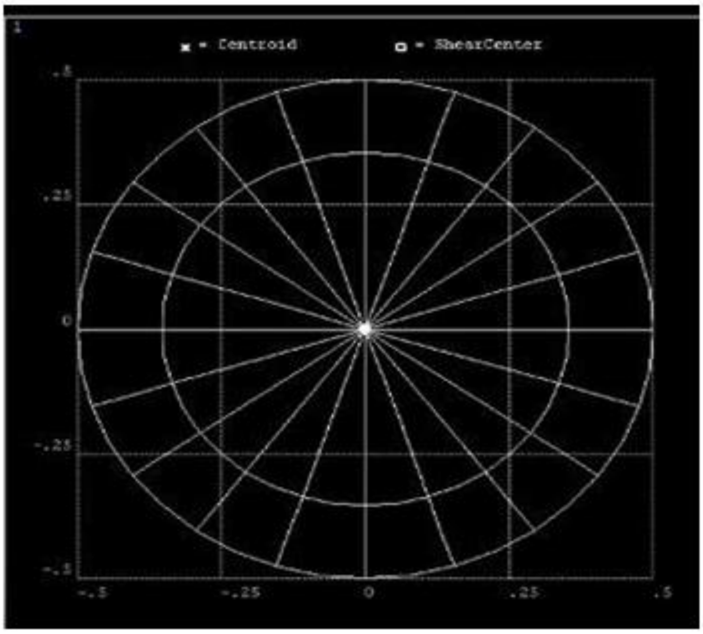
Material Properties ► Material Models ► Structural ► Linear ► Elastic ► Isotropic Enter the Young's Modulus (EXY) = 3e7

Poisson's Ratio (PRXY) = 0.3

Sections ►Beam ►Common Sections ►Subtype ► Select Solid Circle R=0.5

N=20

T=0, Mesh view



Preprocessor ► Modeling ► Create ► Key points ► In Active CS ►

Create the key points according to the table

KP no	X	Y	Z
1.	0	0	0
2.	-12	0	0
3.	12	0	0
4.	0	-12	0
5.	0	-12-12	0
6.	0	-12-12-12	0

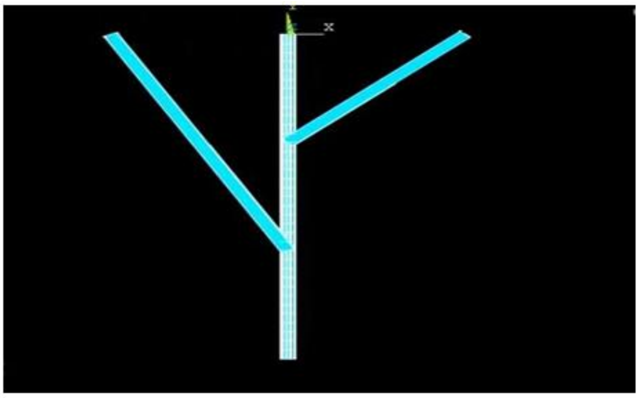
Modeling ► Create ► Lines ► Lines ► Straight Lines ►

Join the key points according to table

Line no	Join
1.	1 & 4
2.	4& 5
3.	5& 6
4.	2& 5
5.	3& 4

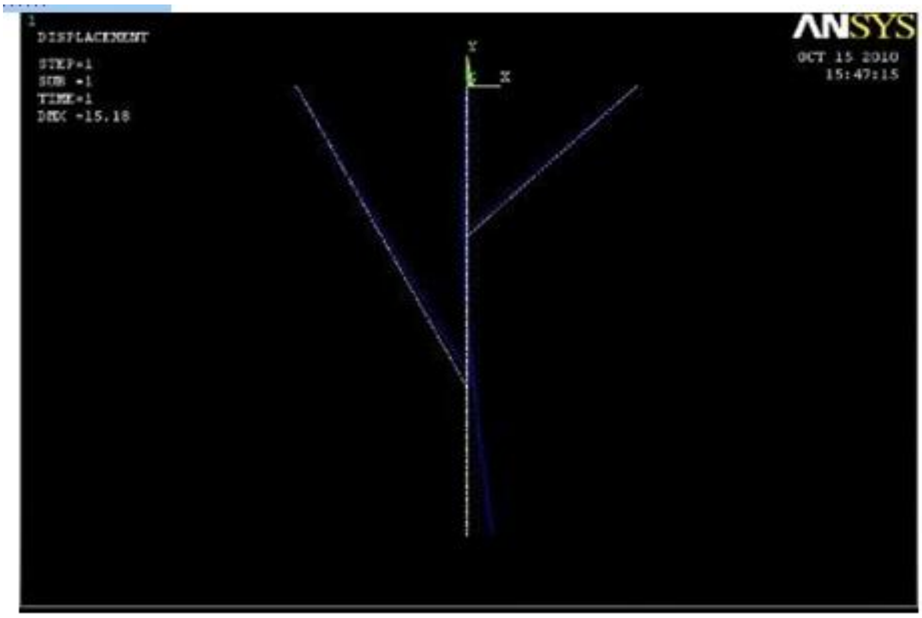
Preprocessor ► Meshing ► Mesh Attributes ► All lines ►
Select element type Beam 188, Ok Meshing ► Mesh tool► set ►Global
1Link1►Ok Lines►set►3&4 line click►2&5 line click►ok No of divisions 1►ok
Mesh Tool►Mesh►Mesh only strut►ok Meshing ► Mesh tool► set ►Global
2 Beam 188►Ok
Lines►set►1&4 line click►4&5 line click►5&6 line click►ok Element egde length►1►ok
Mesh Tool►Mesh►Mesh only Vertical line►ok

Main menu► plot Cntrls ► Style ► Size and Shape
Click in the box against Display Element Type,

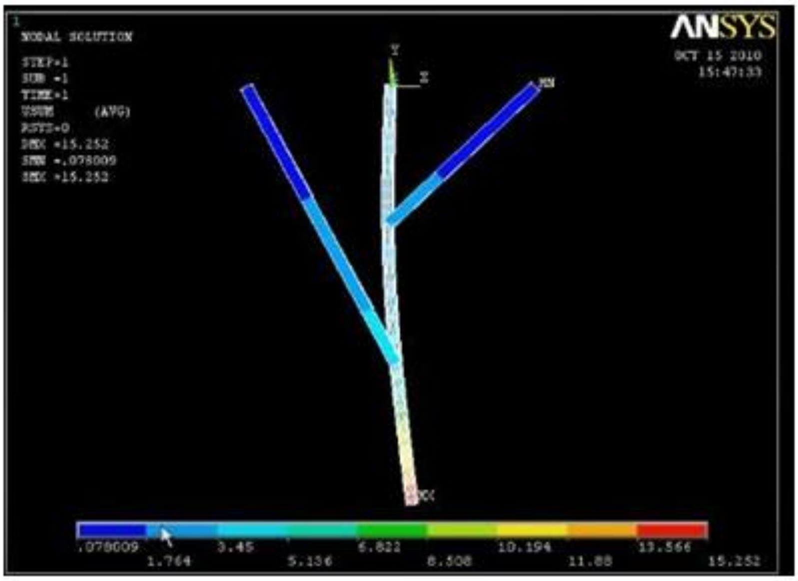


Solution ► Define Loads ► Apply ► Structural ► Displacement ► On key Points ► Select keypoints 2 & 3►select UX,UY,UZ,ROTX,ROTY►Ok
Select keypoints 2 & 3►select UX,UZ►Ok
Modeling >Create>Nodes>Rotate nodes CS>By angles>click 6th keypoint THXY >60>ok
Loads>Apply>Structural>Force/Moment>click On nodes 28/Key point 6> Force/Moment value >10000

Solution ► Solve>General Post proc>List results>Rection solution> Plot results:Defromed shape



Results:

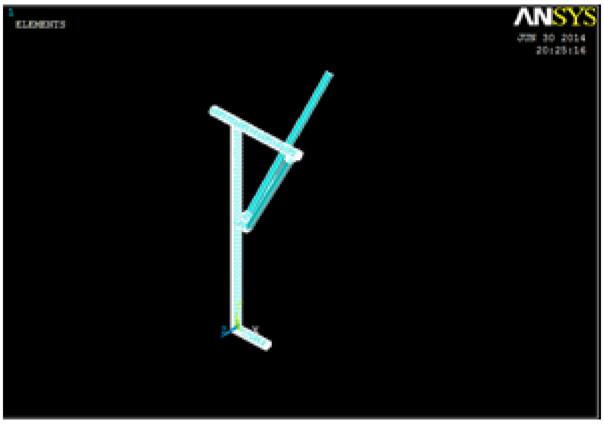


PRE-LAB VIVA QUESTIONS:

a. What do you mean by stack overflow?
b. What are the basic operations of a stack?
c. How to implement stack?

LAB ASSIGNMENT:

d. Analyze the given landing gear as shown in the below figure structure with applied load of 10000N



Angle (Strut):30 degrees

Poisson's Ratio=0.3

POST-LAB VIVA QUESTIONS:

1. Name the element type used for beams?
2. Define Analysis and its Purpose?
3. What are the modules in Ansys Programming?
4. What are the Real Constants & Material Properties in Ansys? Explain?

WEEK – 10
ANALYSIS OF FUSELAGE

OBJECTIVE:

a. To perform the variant analysis of a Aircraft fuselage.
b. To interpret the results of the analysis and study the behavior of the Strucutre

RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:

- To calculate the deformation of the aluminum fuselage section under the application of internal load of 100000 Pa.

PROCEDURE:

PREPROCESSING

STEP 1: From the Main menu select preferences Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type → Add / edit/Delete → Add → Solid – 10 node 92→ Apply
Add → Beam 2 Node 188 → Apply → Add → Shell →Elastic 4 node 63

Real Constants → Add → Select shell → give thickness (I) = 1→ ok → close.

Material properties → material models → Structural → Linear → Elastic → Isotropic EX = 0.7e11; PRXY = 0.3; Density = 2700

STEP 3: From the main menu select **Pre-processor**

Pre-processor → modelling → Create → Areas → Circle → Annulus WP x = 0 ; WP y = 0; Rad – 1 = 2.5; Rad - 2 = 2.3 OK

Pre-processor → Modelling → Create → Circle → Solid –

WP x = 0; X = 2.25; Y = 0 Radius = 0.15

Apply WP x = 0; X = -2.25; Y = 0 Radius = 0.15
Apply WP x = 0; X =0; Y = 2.25; Radius = 0.15
Apply WP x = 0; X = 0; Y = -2.25 Radius = 0.15
OK

Pre-processor → Modelling → Operate → Booleans → Add → Areas – Pick all OK Pre-processor → Modelling → Operate → Extrude → Areas → By XYZ offset

X= 0; Y=0; Z = 5

STEP 4: Meshing the Geometry

Pre-processor →Meshing → Size controls → Manual Size → All Areas → give element edge length as 0.15 → ok

Meshing → Size controls → Manual Size → All lines → give element edge length as→ ok

Meshing → Mesh → areas → free → select box type instead of single → select the total volume → ok

SOLUTION PHASE*:*

STEP 5: From the ANSYS main menu open **Solution**

STEP 6: Loads → define loads → Apply → Structural → Displacement → On areas → select box type → select box (4 points at centre) → all DOF → ok Select → ALL DOF arrested

Define loads → Apply → Structural → Pressure → on areas → select the internal surface of the fuselage and give value (100000) → ok

STEP 7: Solving the system Solution →Solve → Current LS

POSTPROCESSING: VIEWING THE RESULTS

RESULT:

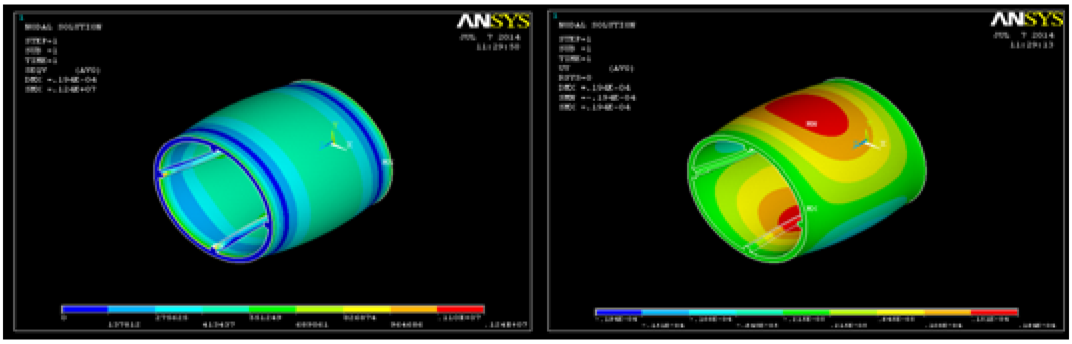
Case: 1:- To Calculate the deformation of the aluminum fuselage section under the application of internal load at 1e5.

Y COMPONENT OF DISPLACEMENT

DMX = .194E-04 SMN = -.194E-04 SMX = .194E-04

VON MISSES STRESS

DMX = .194E-04 SMX = .124E+07



PRE-LAB VIVA QUESTIONS:

a. What is conical lofting?
b. Why is the fuselage on an airliner circular-shaped?
c. Why do airliners have pressured bulk heads?

POST-LAB VIVA QUESTIONS:

1. Difference between interactive mode and batch mode.
2. What are different types of structural analysis used in ansys?
3. What are the different types of thin walled beams?
4. Define Harmonic analysis.

WEEK – 11

THERMAL ANALYSIS

OBJECTIVE:

a. To understand the thermal behavior of structural elements
b. To Intermit the results of conduction convection and radiation phenomena of objects

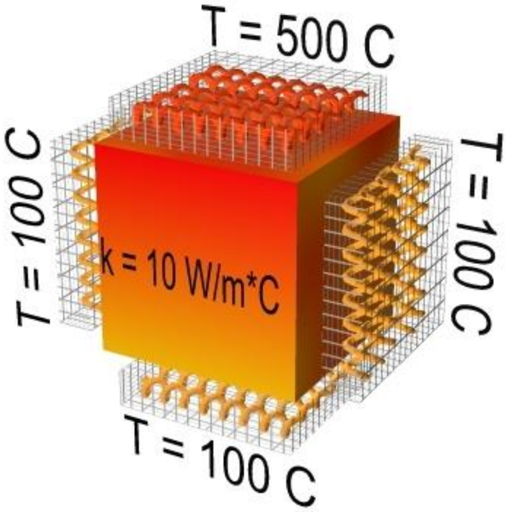
RESOURCE:

ANSYS 16.0 Academic

PROBLEM DESCRIPTION:

- The Simple Conduction Example is constrained as shown in the following figure.

Thermal conductivity (k) of the material is 10 W/m*C and the block is assumed to be infinitely long.



PROCEDURE:

Open preprocessor menu

ANSYS Main Menu > Preprocessor
```
/PREP7
```

Create geometry

Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners > X=0, Y=0, Width=1, Height=1
```
BLC4,0,0,1,1
```

Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete... > click 'Add' > Select Thermal Mass Solid, Quad 4Node 55
```
ET,1,PLANE55
```

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

Element Material Properties

Preprocessor > Material Props > Material Models > Thermal > Conductivity > Isotropic > KXX = 10 (Thermal conductivity)
```
MP,KXX,1,10
```

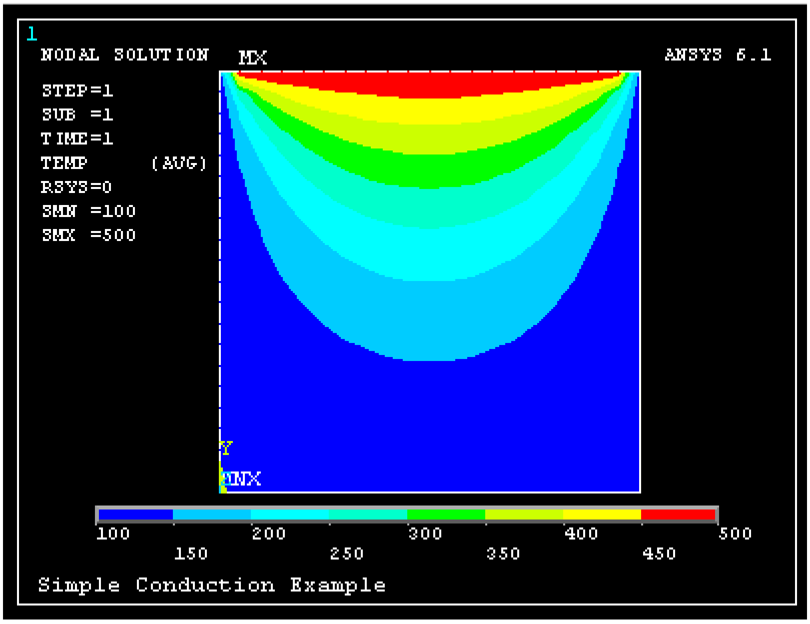
Mesh Size

Preprocessor > Meshing > Size Cntrls > ManualSize > Areas > All Areas > 0.05
`AESIZE,ALL,0.05`

Mesh

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

RESULTS:



PRE LAB VIVA QUESTIONS:

a. Define stress concentration.

b. What are different types of loads acting on a plate?

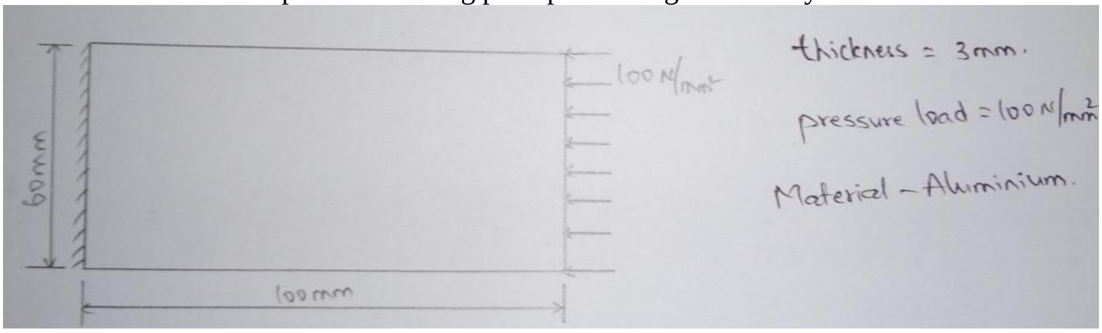
c. Explain pure bending of thin plates.

d. How does the bending moment vary in a plate with simply supported edge?

e. Name different two dimensional elements.

LAB ASSIGNMENT

a. Determine the stresses developed in following plate performing static analysis



POST LAB VIVA QUESTIONS:

b. Define stress concentration.
c. What are Boolean operations?
d. What are different types of loads acting on a plate?
e. What are Von-misses stresses?
f. Explain the elemental properties of Quad 4 node 42.

WEEK – 12

ANALYSIS OF COMPOSITE STRUCTURE

OBJECTIVE:

- To create a composite structure in ANSYS Workbench
- To perform the static analysis for various layers of composite beam.

RESOURCE:

ANSYS 16.0 Academic

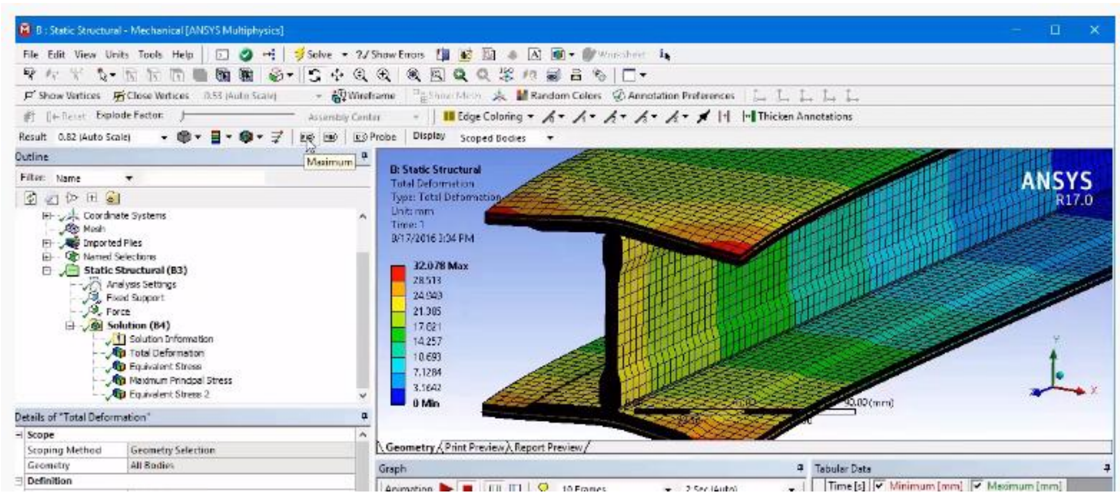
PROBLEM DESCRIPTION:

- Perform the static analysis of composite cantilever beam of length 2m and having cross section 10cm×10cm

PROCEDURE:

- Define draping for modeling groups
- Connecting ACP to Static Structural Tools in Ansys
- Doing Stress Analysis
- Solving the model
- Analyzing Data in static Structural
- Analyzing Data in Post-Processing

Results



PRE-LAB VIVA QUESTIONS:

a. Define tree traversal and mention types of traversal?
b. Define a tree?
c. Define height of a tree?
d. Define depth of a tree?
e. Define degree of a node?
f. Define Degree of a tree?
g. Define Terminal node or leaf node?
h. Define Non-terminal node?
i. Define Sibling?
j. Define Binary Tree?
k. Write the properties of Binary Tree?
l. Find the minimum and maximum height of a binary tree?

LAB ASSIGNMENT:

a. Formulate a program to create a Binary Tree of integers?

b. Write a recursive program, for traversing a binary tree in preorder, inorder and postorder?
c. Compose a non-recursive program, for traversing a binary tree in preorder, inorder and postorder?
d. Write a program to check balance property of a tree?

POST-LAB VIVA QUESTIONS:

a. Write the balance factor of a Binary Tree?
b. What is a spanning Tree?
c. Define a Complete Binary Tree?
d. List out the applications of Binary Tree?
e. Write the two approaches for Binary Tree Traversal?
f. Write the various operations performed in the binary search tree?
g. List out few of the Application of tree data-structure?
h. Define pre-order traversal.
i. Define post-order traversal.
j. Define in-order traversal.