COMPUTATIONAL STRUCTURES

LABORATORY MANUAL

B. TECH (IV YEAR – I SEM)

(2018-19)

Department of Aeronautical Engineering

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

Recognized under 2(f) and 12 (B) of UGC ACT 1956

Affiliated to JNTUH, Hyderabad, Approved by AICTE - Accredited by NBA & NAAC – 'A' Grade - ISO 9001:2015 Certified) Maisammaguda, Dhulapally (Post Via. Hakimpet), Secunderabad – 500100, Telangana State, India

LIST OF EXPERIMENTS

INTRODUCTION

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

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

1. **Pre-processing: defining the problem**; the major steps in pre-processing are given below:
    - Define key points/lines/areas/volumes
    - Define element type and material/geometric properties
    - Mesh lines/areas/volumes as required.

    The amount of detail required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axi-symmetric, 3D).

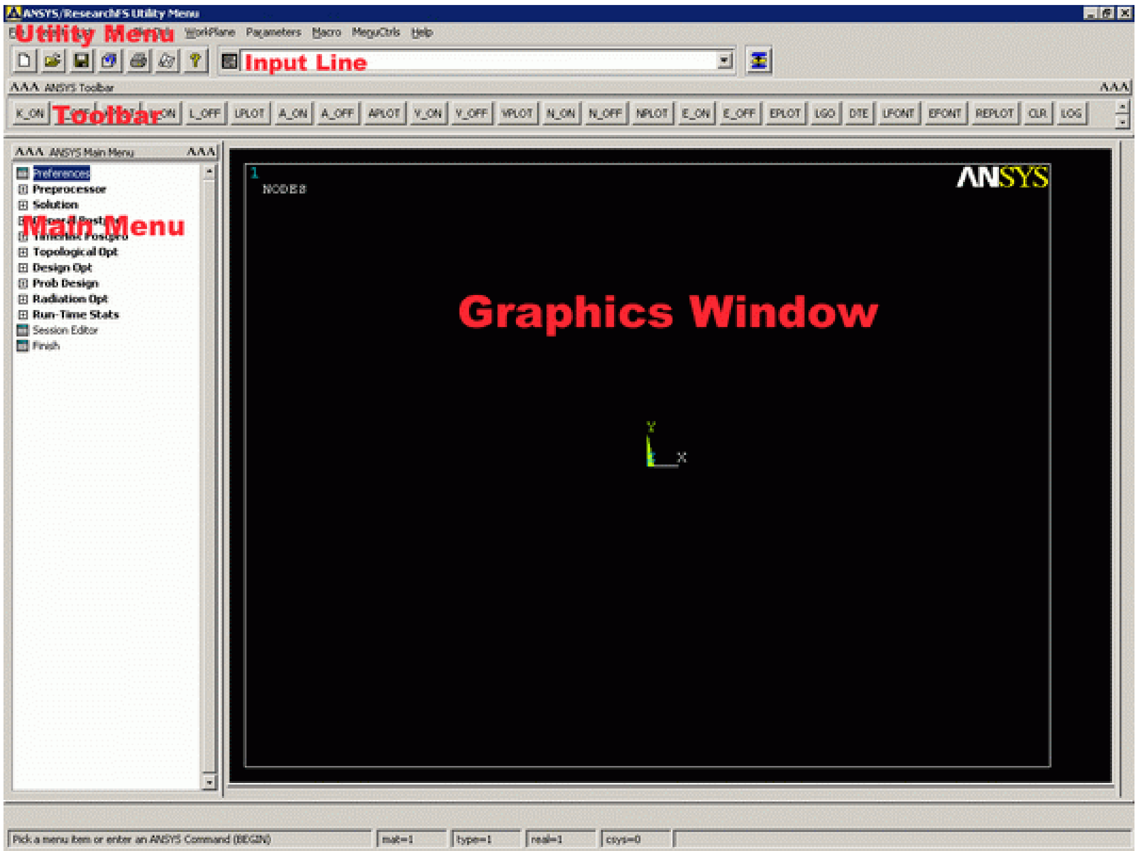
2. **Solution: assigning loads, constraints and solving**; here we specify the loads (point or pressure), constraints (translational and rotational) and finally solve the resulting set of equations.

3. **Postprocessing: further processing and viewing of the results;** in this stage one may wish to see:
    - Lists of nodal displacements
    - Element forces and displacements
    - Deflection plots
    - Stress contour diagrams

1. ANSYS 13.0 Environment

The ANSYS Environment for ANSYS 13.0 contains 2 windows: the Main Window and an Output Window. Note that this is somewhat different from the previous version of ANSYS which made use of 6 different windows.

1. Main Window



a. **Utility Menu**

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

b. **Input Window**

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

c. **Toolbar**

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

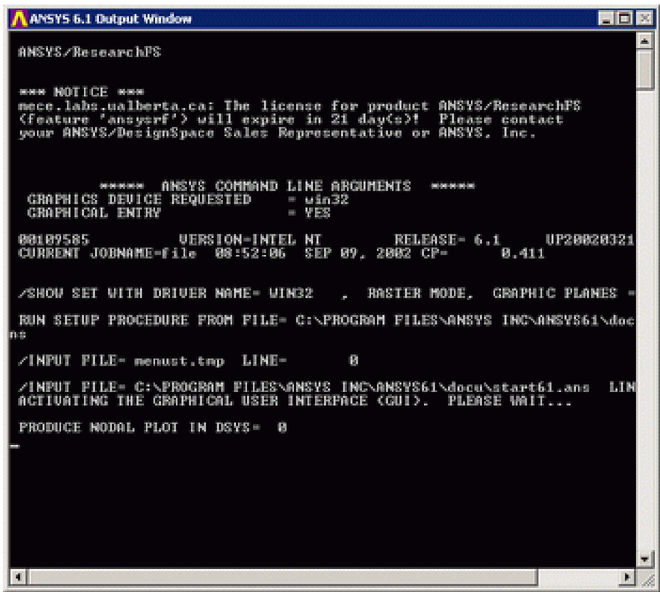
d. **Main Menu**

The Main Menu contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor, design optimizer. It is from this menu that the vast majority of modelling commands are issued. This is where you will note the greatest change between previous versions of ANSYS and version 7.0. However, while the versions appear different, the menu structure has not changed.

e. **Graphics Window**

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

2. **Output Window**



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

2. **ANSYS FILES**

INTRODUCTION

A large number of files are created when you run ANSYS. If you started ANSYS without specifying a jobname, the name of all the files created will be FILE.* where the * represents various extensions described below. If you specified a jobname, say Frame, then the created files will all have the file prefix, Frame again with various extensions:

frame.db

Database file (binary). This file stores the geometry, boundary conditions and any solutions.

frame.dbb

Backup of the database file (binary).

frame.err

Error file (text). Listing of all error and warning messages.

frame.out

Output of all ANSYS operations (text). This is what normally scrolls in the output window during an ANSYS session.

frame.log

Logfile or listing of ANSYS commands (text). Listing of all equivalent ANSYS command line commands used during the current session.

etc...

Depending on the operations carried out, other files may have been written. These files may contain results, etc.

3. Plotting ANSYS Results to a File

PLOTTING OF FIGURES

There are two major routes to get hardcopies from ANSYS. The first is a quick a raster-based screen dump, while the second is a scalable vector plot.

1.0 Quick Image Save

When you want to quickly save an image of the entire screen or the current 'Graphics window', select:

- 'Utility menu bar'/'PlotCtrls'/'Hard Copy ...'.
- In the window that appears, you will normally want to select 'Graphics window', 'Monochrome', 'Reverse Video', 'Landscape' and 'Save to:'.
- Then enter the file name of your choice.
- Press 'OK'

This raster image file may now be printed on a PostScript printer or included in a document.

Display and Conversion

The plot file that has been saved is stored in a proprietary file format that must be converted into a more common graphic file format like PostScript, or HPGL for example. This is performed by running a separate program called `display`. To do this, you have a couple of options:

1. Select `display` from the ANSYS launcher menu (if you started ANSYS that way)
2. Shut down ANSYS or open up a new terminal window and then type `display` at the Unix prompt.

Either way, a large graphics window will appear. Decrease the size of this window, because it most likely covers the window in which you will enter the `display` plotting commands. Load your plot file with the following command:

file,frame,pic

if your plot file is 'plots.pic'. Note that although the file is 'plots.pic' (with a period), Display wants 'plots,pic'(with a comma). You can display your plots to the graphics window by issuing the command like

plot,n

where n is plot number. If you plotted 5 images to this file in ANSYS, then n could be any number from 1 to 5.

Now that the plots have been read in, they may be saved to printer files of various formats:

1. **Colour PostScript:** To save the images to a colour postscript file, enter the following commands in `display`:
2. pscr,color,2
3. /show,pscr
4. plot,n

   Where n is the plot number, as above. You can plot as many images as you want to postscript files in this manner. For subsequent plots, you only require the `plot,n` command as the other options have now been set. Each image is plotted to a postscript file such as pscrxx.grph, where xx is a number, starting at 00.

   Note: when you import a postscript file into a word processor, the postscript image will appear as blank box. The printer information is still present, but it can only be viewed when it's printed out to a postscript printer.

   Printing it out: Now that you've got your color postscript file, what are you going to do with it? Take a look here for instructions on colour postscript printing at a couple of sites on campus where you can have your beautiful stress plot plotted to paper, overheads or even posters!

5. **Black & White PostScript:** The above mentioned colour postscript files can get very large in size and may not even print out on the postscript printer in the lab because it takes so long to transfer the files to the printer and process them. A way around this is to print them out in a black and white postscript format instead of colour; besides the colour specifications don't do any good for the black and white lab printer anyways. To do this, you set the postscript color option to '3', i.e. and then issue the other commands as before
6. pscr,color,3
7. /show,pscr
8. plot,n

4. MECHANICAL APDL DOCUMENTATION DESCRIPTIONS

The manuals listed below form the ANSYS product documentation set. They include descriptions of the procedures, commands, elements, and theoretical details needed to use ANSYS. A brief description of each manual follows.

Advanced Analysis Techniques Guide: Discusses techniques commonly used for complex analyses or by experienced ANSYS users, including design optimization, manual rezoning, cyclic symmetry, rotating structures, submodeling, substructuring, component mode synthesis, and cross sections.

ANSYS Connection User's Guide: Gives instructions for using the ANSYS Connection products, which help you import parts and models into ANSYS.

ANSYS Parametric Design Language Guide: Describes features of the ANSYS Parametric Design Language (APDL), including parameters, array parameters, macros, and ways to interface with the ANSYS GUI. Explains how to automate common tasks or to build your model in terms of parameters. Includes a command reference for all APDL-related commands.

Basic Analysis Guide: Describes general tasks that apply to any type of analysis, including applying loads to a model, obtaining a solution, and using the ANSYS program's graphics capabilities to review results.

Command Reference: Describes all ANSYS commands, in alphabetical order. It is the definitive reference for correct command usage, providing associated menu paths, product applicability, and usage notes.

Contact Technology Guide: Describes how to perform contact analyses (surface-to-surface, node-to-surface, node-to-node) and describes other contact-related features such as multipoint constraints and spot welds.

Coupled-Field Analysis Guide: Explains how to perform analyses that involve an interaction between two or more fields of engineering.

Distributed ANSYS Guide: Explains how to configure a distributed processing environment and proceed with a distributed analysis.

Element Reference: Describes all ANSYS element, in numerical order. It is the primary reference for correct element type input and output, providing comprehensive descriptions for every option of every element. Includes a pictorial catalog of the characteristics of each ANSYS element.

Modeling and Meshing Guide: Explains how to build a finite element model and mesh it.

Multibody Analysis Guide: Describes how to perform a multibody simulation to analyze the dynamic behavior of a system of interconnected bodies comprised of flexible and/or rigid components.

Operations Guide: Describes basic ANSYS operations such as starting, stopping, interactive or batch operation, using help, and use of the graphical user interface (GUI).

Performance Guide: Describes factors that impact the performance of ANSYS on current hardware systems and provides information on how to optimize performance for different ANSYS analysis types and equation solvers.

Rotordynamic Analysis Guide: Describes how to perform analysis of vibrational behavior in axially symmetric rotating structures, such as gas turbine engines, motors, and disk drives.

Structural Analysis Guide: Describes how to perform the following structural analyses: static, modal, harmonic, transient, spectrum, buckling, nonlinear, material curve fitting, gasket joint simulation, fracture, composite, fatigue, p-method, beam, and shell.

Theory Reference for the Mechanical APDL and Mechanical Applications: Provides the theoretical basis for calculations in the ANSYS program, such as elements, solvers and results formulations, material models, and analysis methods. By understanding the underlying theory, you can make better use of ANSYS capabilities while being aware of assumptions and limitations.

Thermal Analysis Guide: Describes how to do steady-state or transient thermal analyses.

EXPERIMENT: 1

TWO DIMENSIONAL STATIC LINEAR ANALYSIS OF A CANTILEVER BEAM

- **Experiment as given in the JNTUH curriculum.**
- **BENDING OF UNIFORM CANTILEVER BEAM**

AIM: To determine the stresses acting on a cantilever beam with a point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.

- Young's modulus = 2e5
- Poisson's ratio = 0.3
- Length of the beam = 2m = 2000mm
- Breadth of the beam = 10 cm = 100mm
- Height of the beam = 50mm

PROCEDURE:

PRE PROCESSING

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 → BEAM – 2D Elastic 3 → Apply → Close

Material properties → material models → Structural → Linear → Elastic → Isotropic

EX = 2e5; PRXY = 0.3

STEP 3: From the main menu select Pre-processor

Sections → Beam → Common Sections → Select subtype as Rectangular section → Enter B = 100, H = 50 → Apply → Preview

Real constants → Add → Add → Ok → Geometric Properties → Area = 5000, I_{zz} = 4170000, Height = 50 → Ok → Close

STEP 4: From the main menu select Pre-processor → **Modelling**

- Create the key points in the Workspace

Create → Key points → in active CS

X	0	2000
Y	0	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points
  Create → Lines → Lines → Straight Line → Click on Key points to generate lines

  Select Plot controls from menu bar → Capture image → file save as and save your file

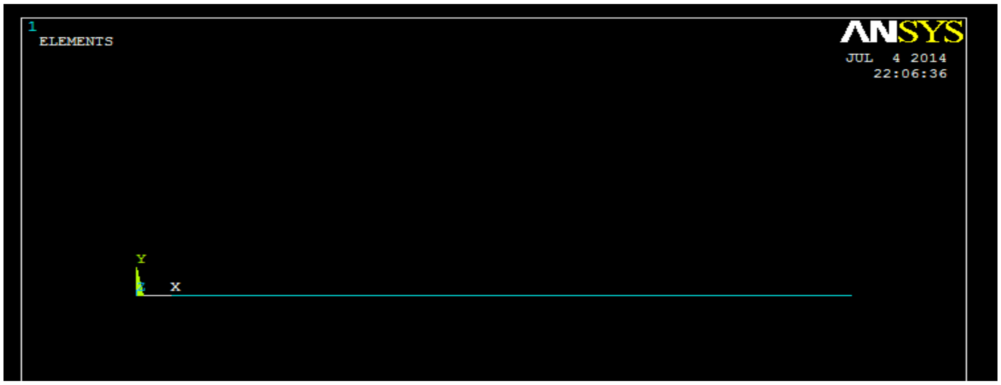


Figure: Model

STEP 4: Meshing the Geometry

From the main menu select **Meshing**

Meshing → Size controls → Manual size → Lines → All lines – Number of element divisions = **20** → Click OK

Meshing → Mesh → Lines – pick all

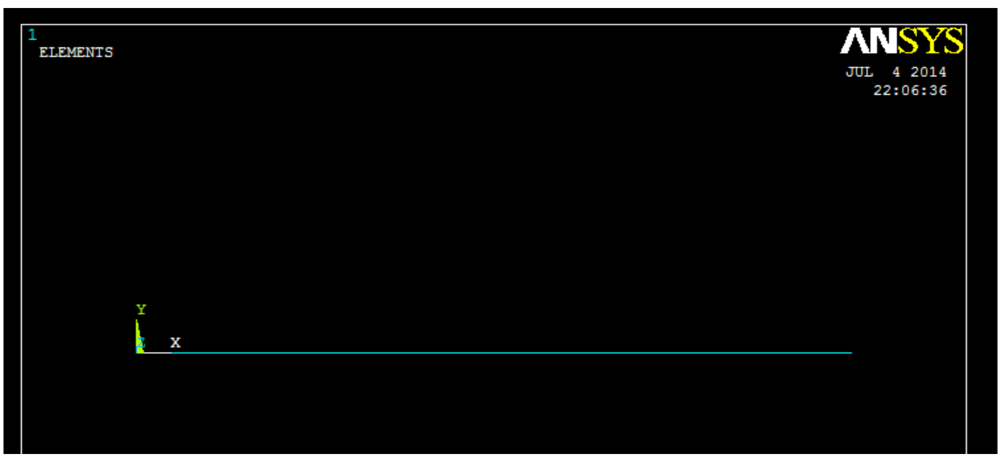


Figure: Meshed Model

<u>SOLUTION PHASE: ASSIGNING LOADS AND SOLVING</u>

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

Solution → Analysis type → new analysis – Static

STEP 6: Defining loads at the Key points

Solution → Define Loads → Apply → Structural → Displacement → On key points

Left end – ALL DOF arrested

Solution → Define loads → Apply → Structural → Force/moment → On key Points

Right end – Apply a load of FY = -1000N

Select Plot controls from menu bar → Capture image → file save as and save your file

Figure: Model with boundary conditions

STEP 7: Solving the system

Solution →Solve → Current LS

<u>POSTPROCESSING: VIEWING THE RESULTS</u>

1. **Deformation**

   From the main menu select **General post processing**

   General post processing → Plot Results → Deformed Shape

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

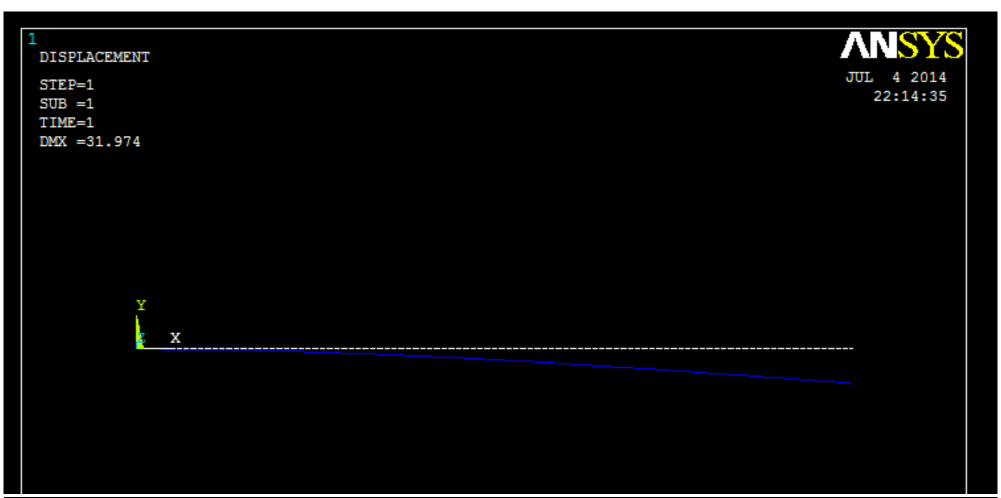


Figure: Deformed and undeformed Model

- **Nodal solution**

  From the Utility menu select PLOT

  PLOT →Results →Contour plot → Nodal solution – DOF solution – Y component of displacement – OK

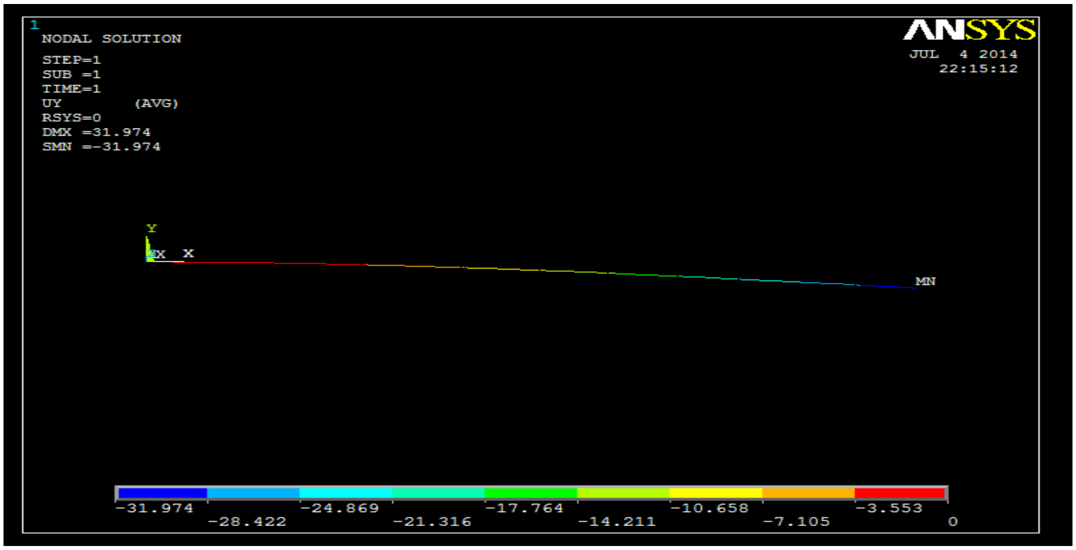


Figure: Y-Component displacement of the Model

<u>RESULT:</u>

Case: 1:- To determine the stresses acting on a cantilever beam with a point load of -10000 N acting at one of its ends and perpendicular to the axis of the beam.

1. DMX = 31.974

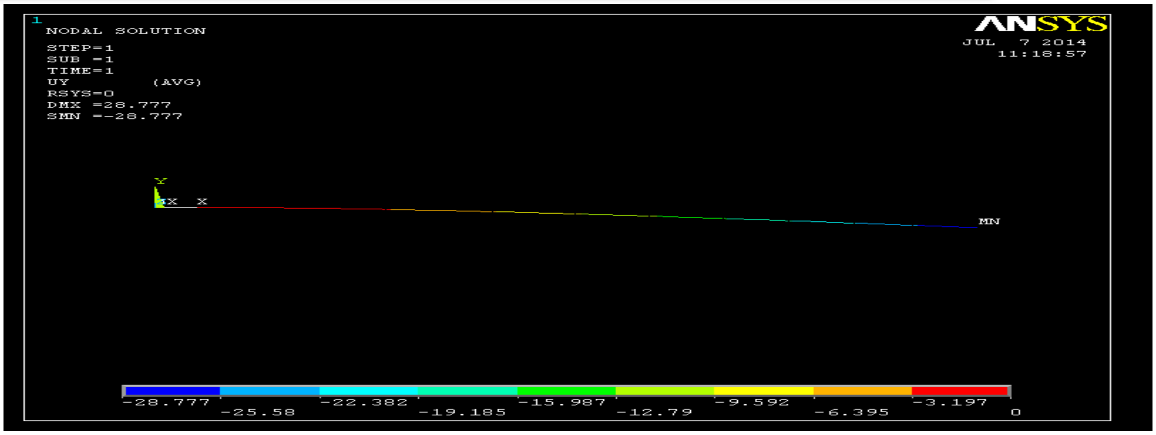
   SMN = -31.974

PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To determine the stresses acting on a cantilever beam with a point load of -9000 N acting at one of its ends and perpendicular to the axis of the beam.

1. DMX = 28.777

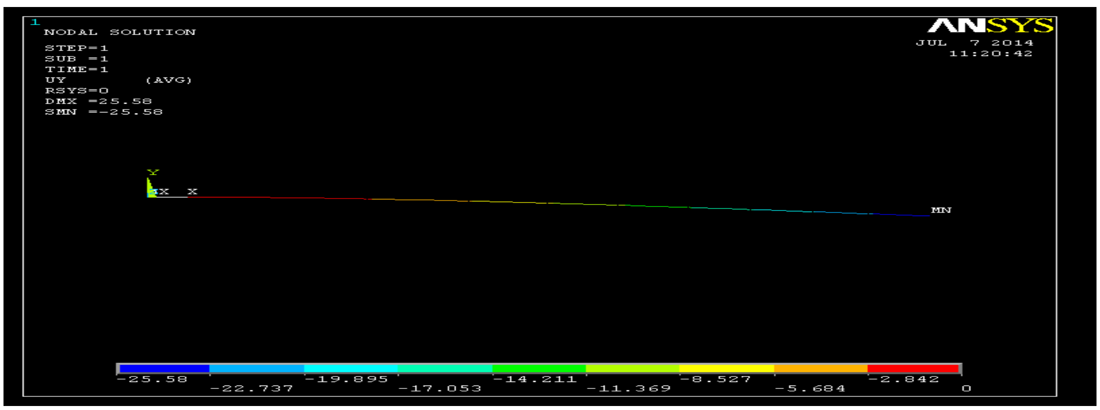
   SMN = -28.777



Case: 3:- To determine the stresses acting on a cantilever beam with a point load of -8000 N acting at one of its ends and perpendicular to the axis of the beam.

1. DMX = 25.58

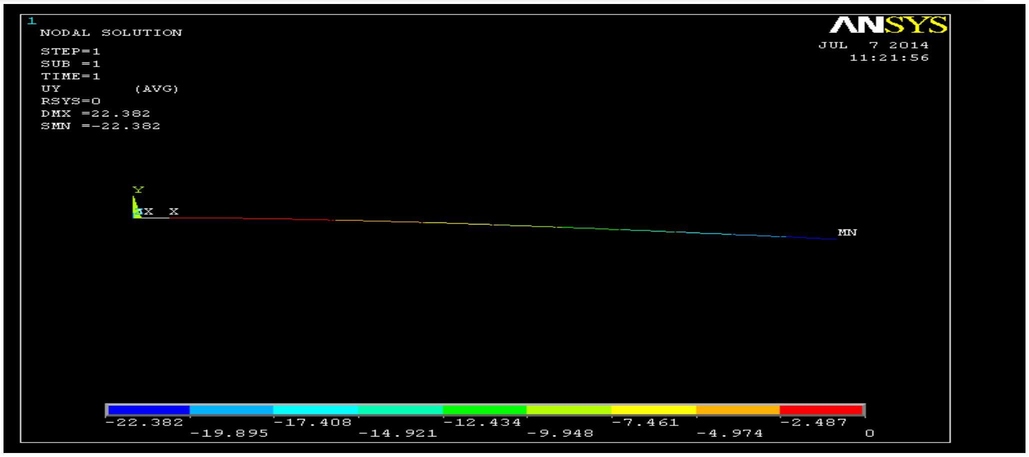
   SMN = -25.58



Case: 4:- To determine the stresses acting on a cantilever beam with a point load of -7000 N acting at one of its ends and perpendicular to the axis of the beam.

1. DMX = 22.382

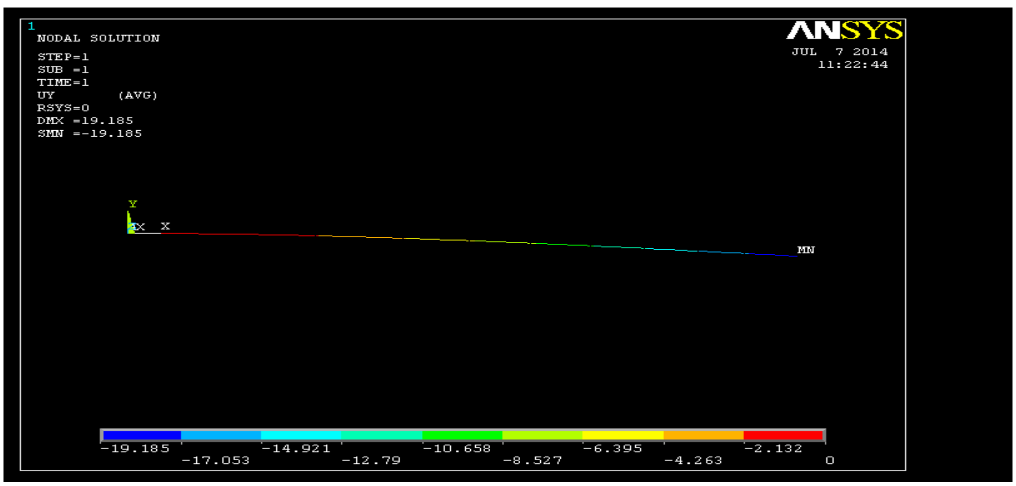
   SMN = -22.382



Case: 5:- To determine the stresses acting on a cantilever beam with a point load of -6000 N acting at one of its ends and perpendicular to the axis of the beam.

1. DMX = 19.185

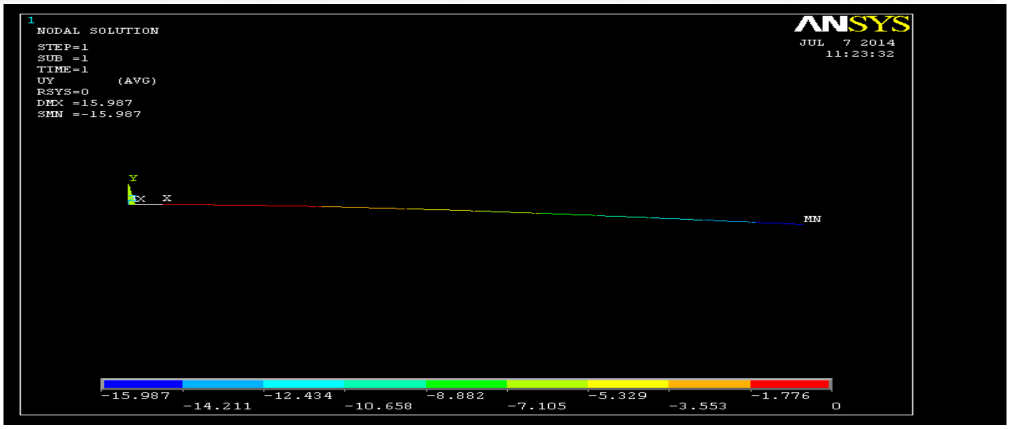
   SMN = -19.185



Case: 6:- To determine the stresses acting on a cantilever beam with a point load of -5000 N acting at one of its ends and perpendicular to the axis of the beam.

1. DMX = 15.988

   SMN = -15.988



EXERCISE PROBLEM

1) To perform static analysis on cantilever beam for different loadings with following specifications

Length of the beam (L) = 1m

Breadth of beam (B) = 0.01m

Width of the beam (W) = 0.1m

Loading conditions: 1KN, 10KN, 100KN (concentrated load)

State the observations on behavior of the deflection of beam

VIVA QUESTIONS

1. If a cantilever beam has a uniformly distributed load, will the bending moment diagram be quadratic or cubic?
2. Name the element type used for beams?
3. Define Analysis and its Purpose?
4. What are the modules in Ansys Programming?
5. What are the Real Constants & Material Properties in Ansys? Explain?

EXPERIMENT: 2

COMPRESSIVE STRENGTH OF RECTANGULAR STIFFENED PLANE PANEL OF UNIFORM CROSS-SECTION

- **Experiment as given in the JNTUH curriculum.**
- **Compressive strength of rectangular stiffened plane panel.**

AIM: To analyze the compressive strength of rectangular stiffened plane panel of uniform cross-section which is subjected to a pressure of 12000 Pa.

APPARATUS: Ansys 13.0

GIVEN DATA:

- Young's modulus = 2e11
- Thickness I=1.2, J=1.2
- Poisson's ratio = 0.27
- Density = 7850kg/m^3

PROCEDURE:

PRE PROCESSING

STEP 1: From the Main menu select preferences

Select structural→ h-method and press OK

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

Element type → Add / edit/Delete → Add → select shell→elastic 4 node 63→ok

Real constants → Add →Add→select type1 shell→ok

Thickness→I=1.2, J=1.2→ok

Material properties → material models → Structural → Linear → Elastic → Isotropic

EX = 2e11; PRXY = 0.27; Density = 7850

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

- Create the key points in the Workspace

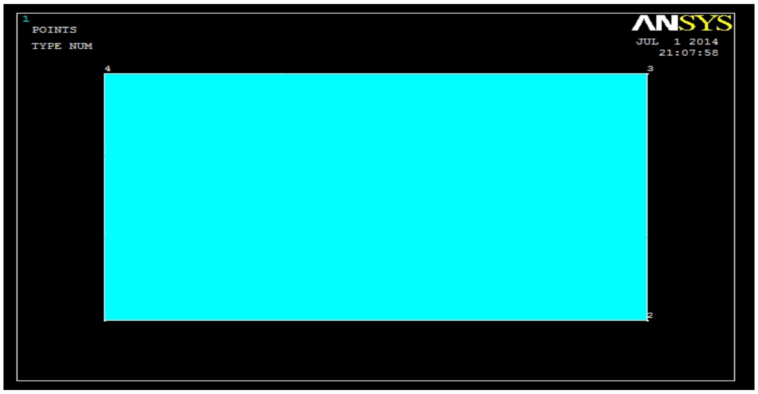
Create → Key points → In active CS

X	Y	Z
0	0	0
6	0	0
6	4	0
0	4	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points
  Create → Lines → Straight Line → select 1-2, 2-3, 3-4, 4-1 Key points to generate lines

STEP 4: Modeling → create → Areas arbitrary by lines → select all four lines → ok



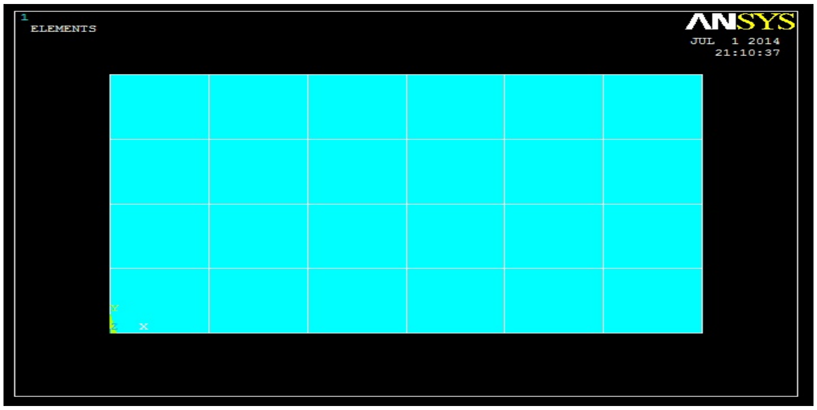
STEP 5: Meshing the Geometry

From the main menu select **Meshing**

Meshing → mesh attributes → all areas → select the area → shell → ok

Meshing → Size controls → Manual size → by areas → all areas → Number of element edge length = 1 → Click ok

Meshing → Mesh → areas → mapped → 3 or 4 sided → select area → ok



<u>SOLUTION PHASE: ASSIGNING LOADS AND SOLVING</u>

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

Solution → Analysis type → new analysis → Static

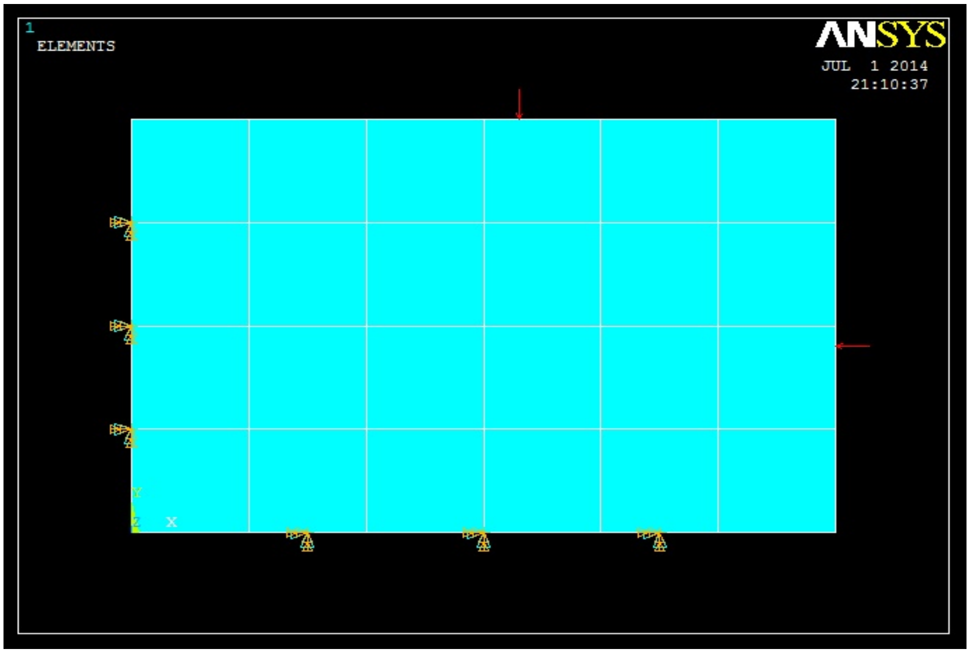
STEP 6: Defining loads

Loads → define loads → Apply → Structural → Displacement → On lines → select line 1-2 & 1-4 → ok

Select → ALL DOF arrested

Define loads → Apply → Structural → Pressure → select on lines 2-3 & 3-4 → ok

Enter pressure = 12000 → ok



STEP 7: Solving the system

Solution →Solve → Current LS

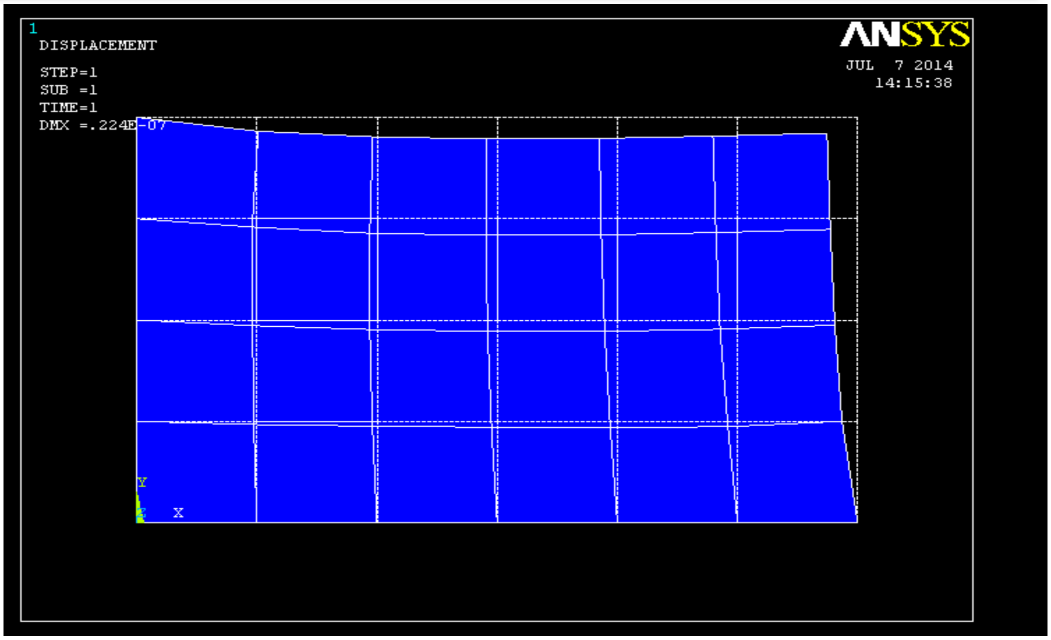
POSTPROCESSING: VIEWING THE RESULTS

2. **Deformation**

   From the main menu select **General post processing**

   General post processing → Plot Results → Deformed Shape

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



- **Nodal solution**

From the Utility menu select PLOT

PLOT →Results →Contour plot → Nodal solution

Result → DOF solution → Y component of displacement → OK

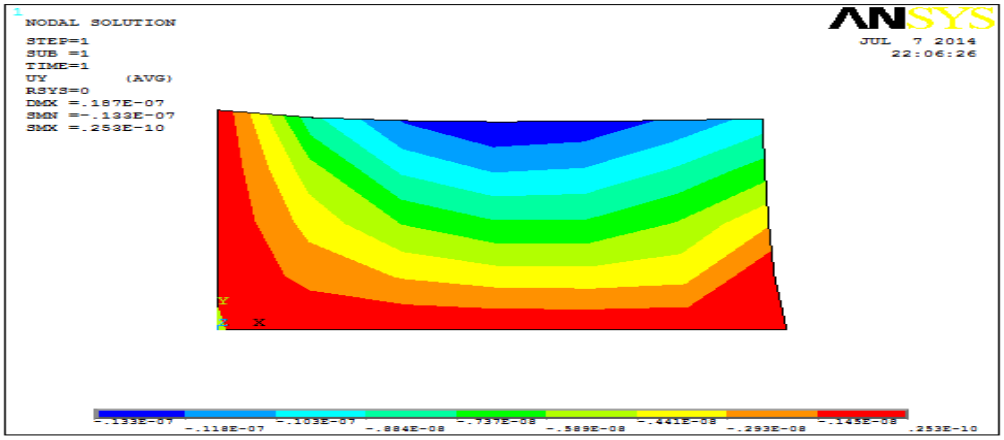
Result → stress → Von mises stress

RESULT:

Case: 1:- To determine the stresses acting on a rectangular plane with a pressure load of 12000 N acting on the lines 2 & 3.

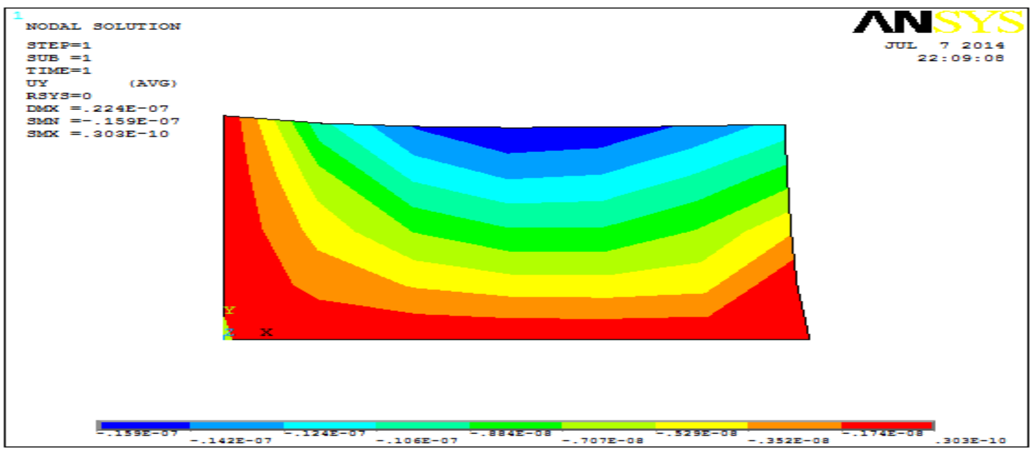
DMX = 0.187e-07

SMX = 939.279



PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

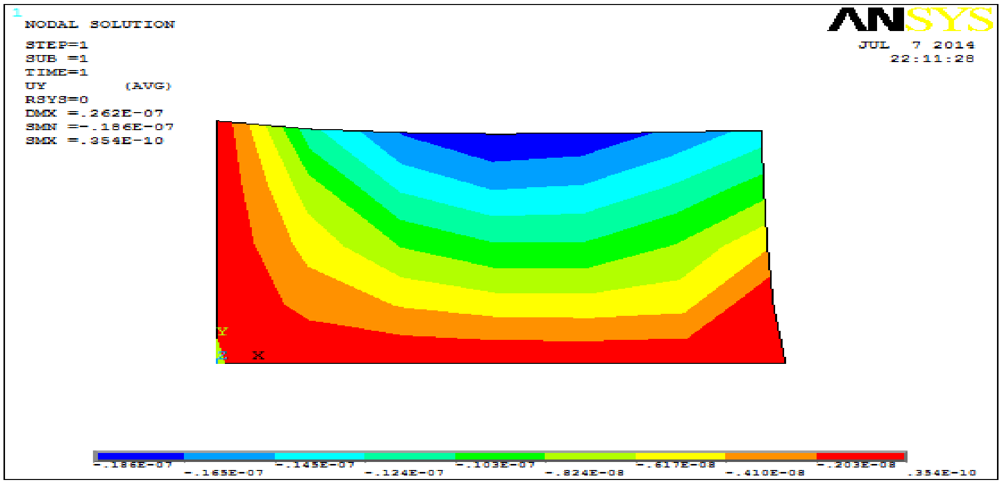
Case: 2:- To determine the stresses acting on a rectangular plane with a pressure load of 11000 N acting on the lines 2 & 3



DMX = .224e-07

SMX = 1127

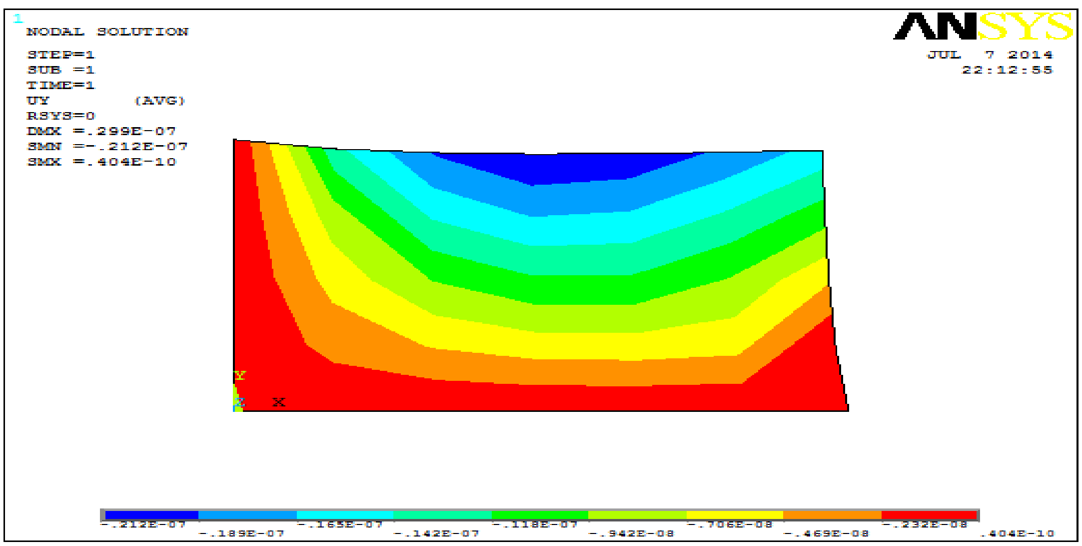
Case: 3:- To determine the stresses acting on a rectangular plane with a pressure load of 10000 N acting on the lines 2 & 3



DMX = 0.224e-06

SMX = 4747

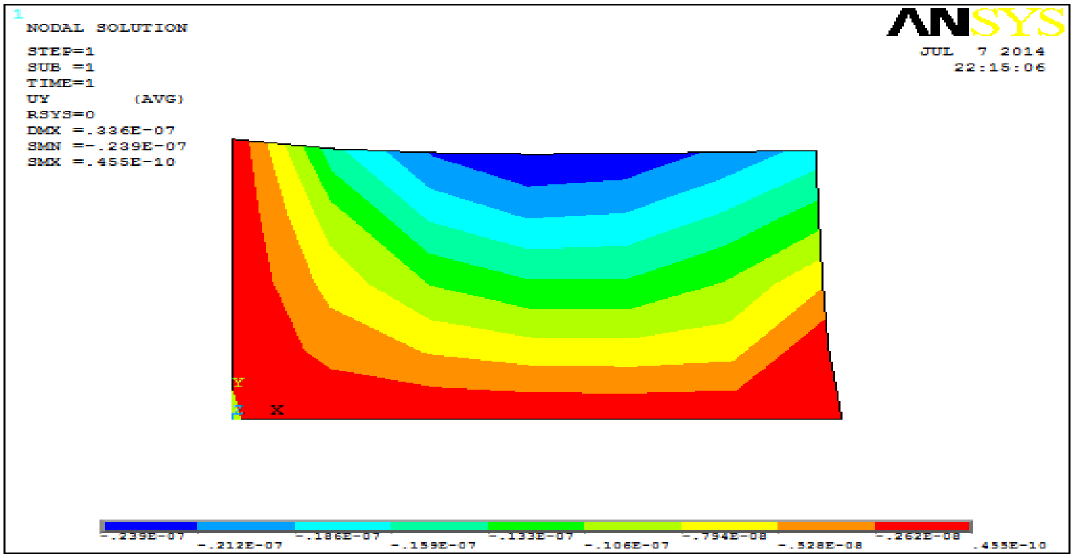
Case: 4:- To determine the stresses acting on a rectangular plane with a pressure load of 13000 N acting on the lines 2 & 3



DMX = 0.224e-06

SMX = 1127

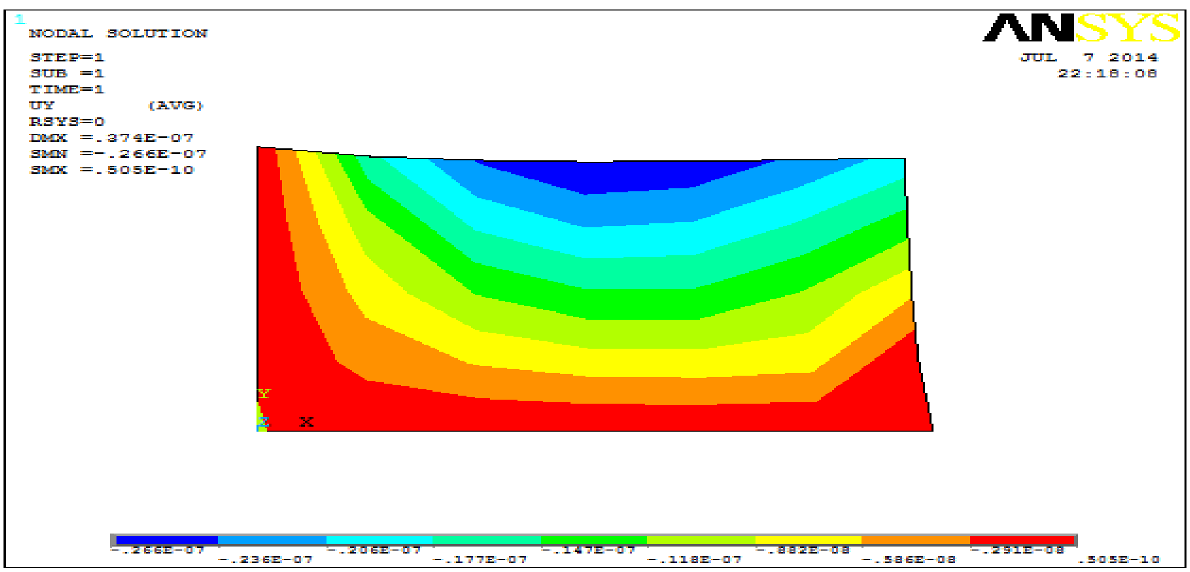
Case: 5:- To determine the stresses acting on a rectangular plane with a pressure load of 14000 N acting on the lines 2 & 3

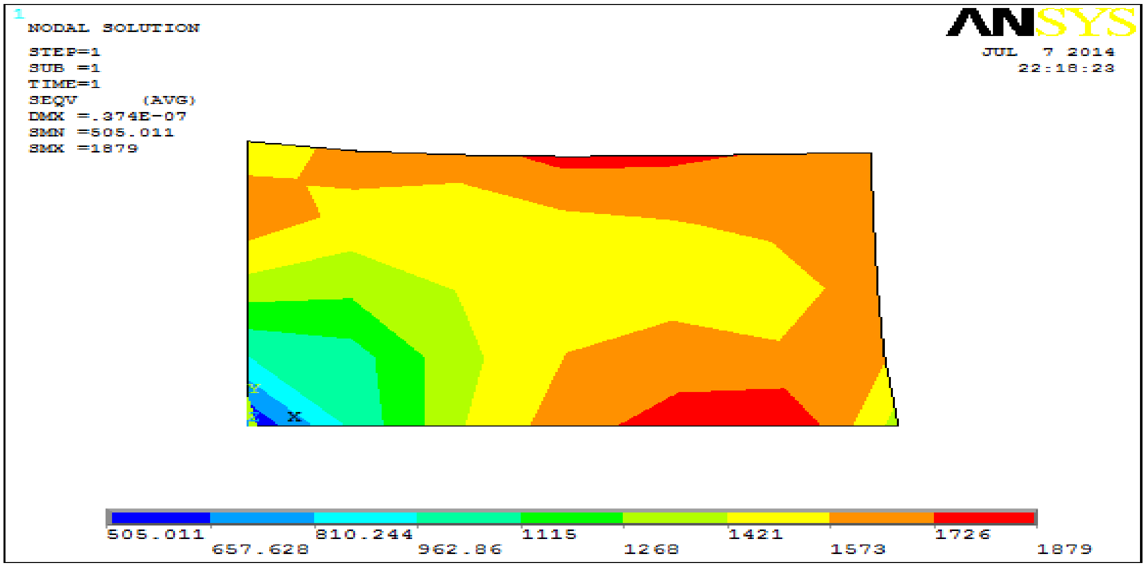


DMX = 0.224e-06

SMX = 1127

Case: 6:- To determine the stresses acting on a rectangular plane with a pressure load of 15000 N acting on the lines 2 & 3





DMX = 0.224e-06

SMX = 1127

EXERCISE PROBLEM

1) To perform static analysis on cantilever beam for different loadings with following specifications

Length of the beam (L) = 1m

Breadth of beam (B) = 0.01m

Width of the beam (W) = 0.1m

Loading conditions: 1KN, 10KN, 100KN

Consider the load to be Uniformly Distributed Load and Explain the behavior of the deflection of beam?

VIVA QUESTIONS

1. What do you mean by degrees of freedom?
2. Define key points, lines, nodes, elements?
3. Can meshing is done after elements are created?
4. Types of co-ordinate systems?
5. What is symmetry and types of symmetry?

EXPERIMENT: 3(A)

a) SHEAR OF STIFFENED THIN WALLED OPEN SECTION BEAM

- **Experiment as given in the JNTUH curriculum.**
- **Shear of stiffened thin walled open section**

AIM: To analyze shear of stiffened thin walled open section beam which is subjected to a pressure of 50 MPa.

APPARATUS: Ansys 13.0

GIVEN DATA:

- Young's modulus = 0.7e11
- Thickness I = 1.3, J = 1.3
- Poisson's ratio = 0.3
- Density = 2700 kg/m^3

PROCEDURE:

PRE PROCESSING

STEP 1: From the Main menu select preferences

Select structural→ h-method and press OK

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

Element type → Add / edit/Delete → Add → select shell→elastic 4 node 63→apply→solid→quad 4 node 182→ok

Real constants → Add →Add→select type1 shell→ok → enter

Thickness→I =1.3, J=1.3 → ok → close

Material properties → material models → Structural → Linear → Elastic → Isotropic

EX = 0.7e11; PRXY = 0.3 & Density = 2700 → ok → close

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

- Create the key points in the Workspace

Create → Key points → In active CS

X	Y	Z
0	0	0
2	0	0
2	0.2	0
0.2	0.2	0

0.2	1.8	0
0.5	1.8	0
0.5	2	0
0	2	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points
  Create → Lines → Straight Line → Select 1-2, 2-3, 3-4, 4-5, 5-6, 6-7, 7-8, 8-1 Key points to generate lines

STEP 4: Modeling → create → Areas → arbitrary by lines → select all four lines → ok

Modeling → operate → extrude → areas → along normal → select the area → ok → enter the extrude length as 0.5

Select Plot controls from menu bar → Capture image → file save as and save your file

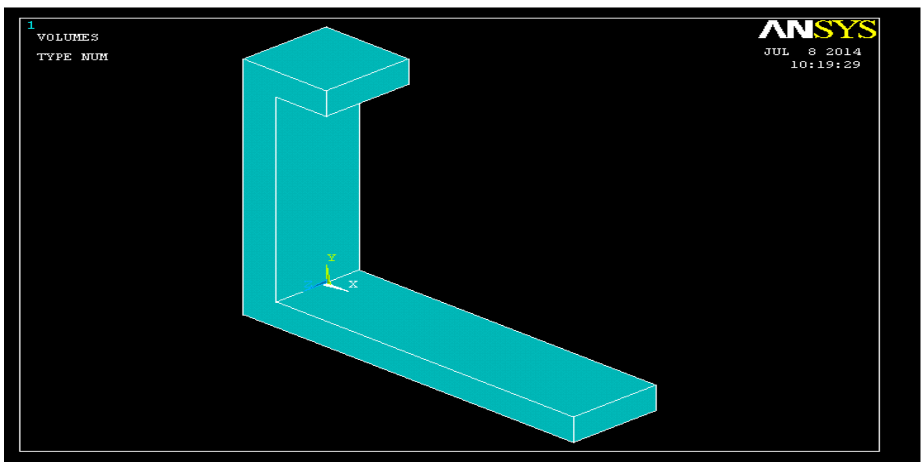


Figure: Open section beam model

STEP 5: Meshing the Geometry

From the main menu select **Meshing**

Meshing → mesh attributes → all areas → select the element type → no shell → ok

Select All volumes →select the element type number →plane →ok

Meshing → Size controls → Manual size → by areas → all areas → Number of element edge length = 0.025 → Click ok

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

Select Plot controls from menu bar → Capture image → file save as and save your file

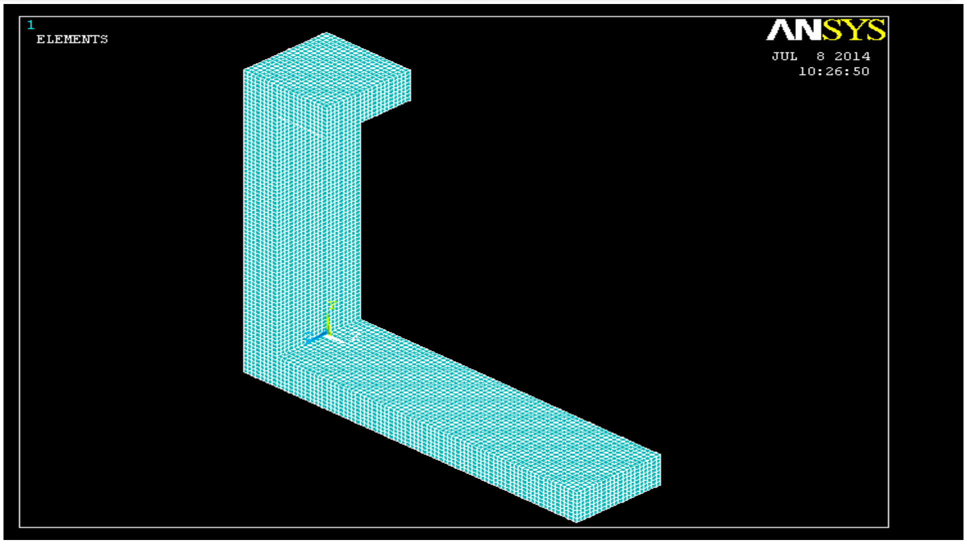


Figure: Open section beam meshed model

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

STEP 5: Defining loads

Loads → define loads → Apply → Structural → Displacement → On areas → select the bottom edge → ok → all DOF → ok

Select → ALL DOF arrested

Define loads → Apply → Structural → Pressure → on areas → select box type (instead of single) → select the top flange → ok

Enter pressure = 12000 → ok

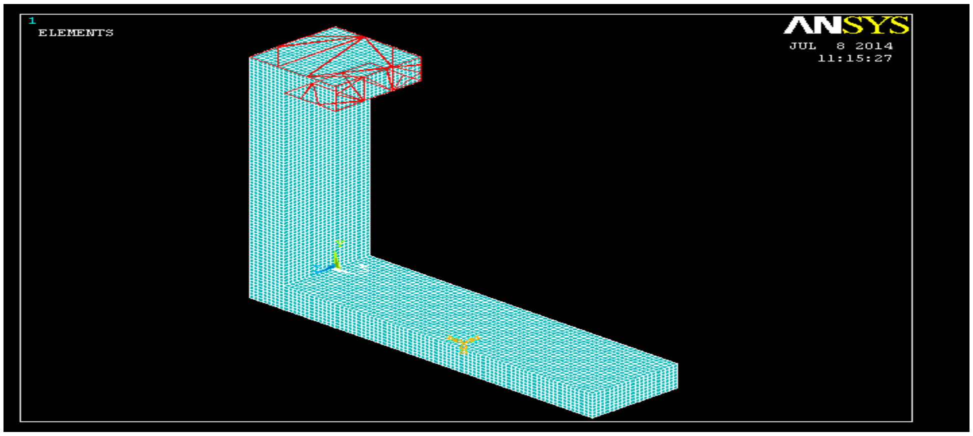


Figure: Boundary and operating conditions model

STEP 6: Solving the system

Solution →Solve → Current LS

POSTPROCESSING: VIEWING THE RESULTS

1. **Deformation**

From the main menu select **General post processing**

General post processing → Plot Results → Deformed Shape

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

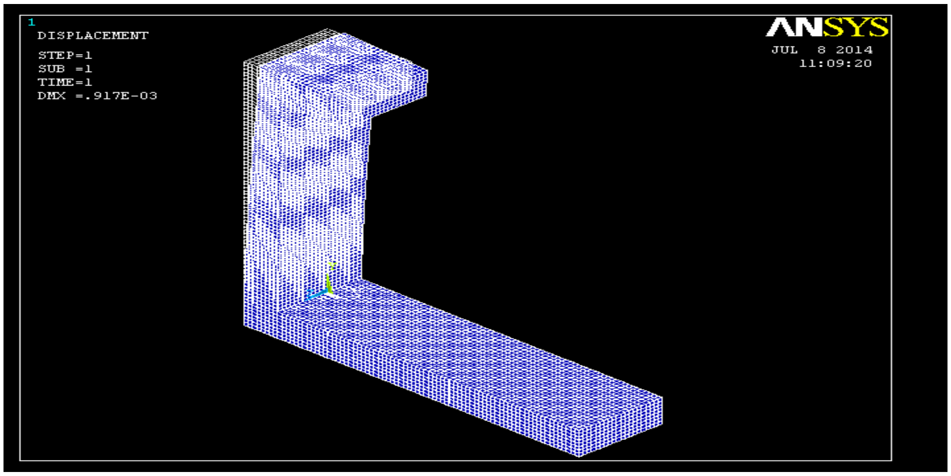


Figure: Deformed and undeformed model

Nodal solution

From the Utility menu select PLOT

PLOT →Results →Contour plot → Nodal solution

2. Select DOF solution → X component of displacement → OK

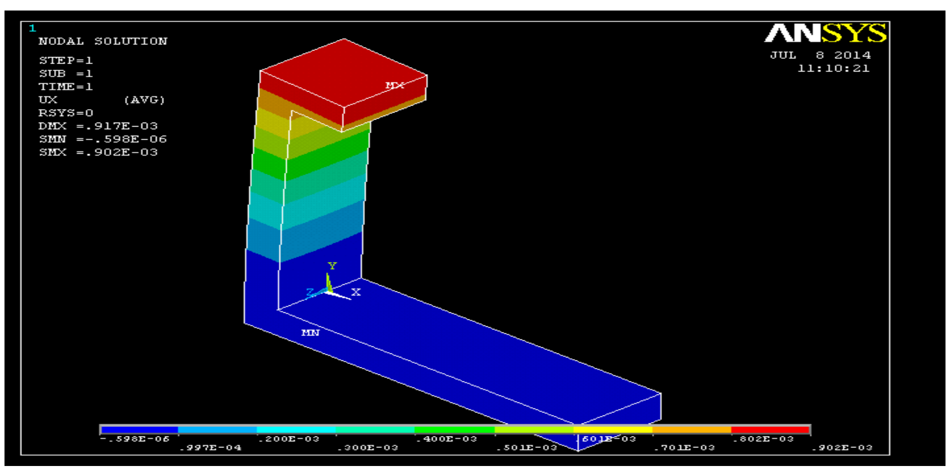


Figure: X-Component of displacement model

3. Select stress → XY shear stress

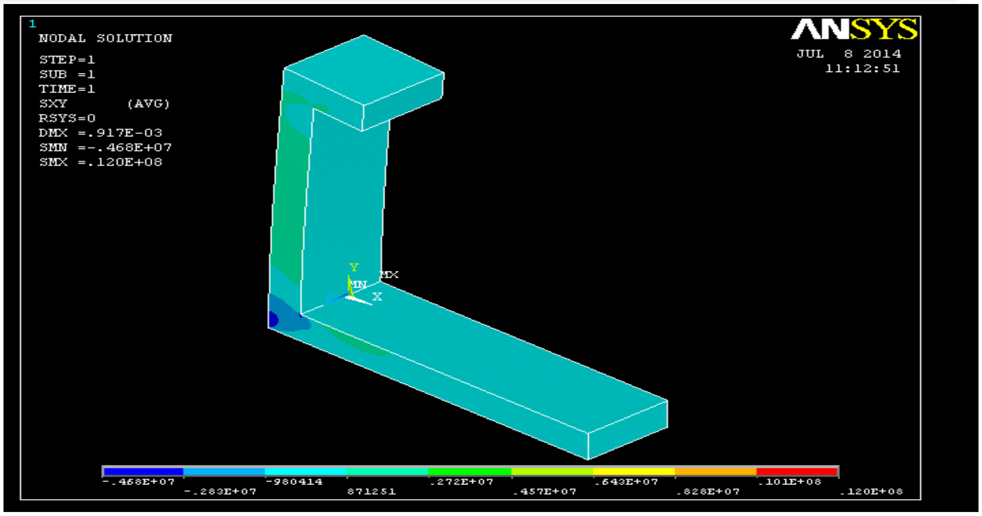


Figure: XY shear stress model

4. Select stress → Von mises stress

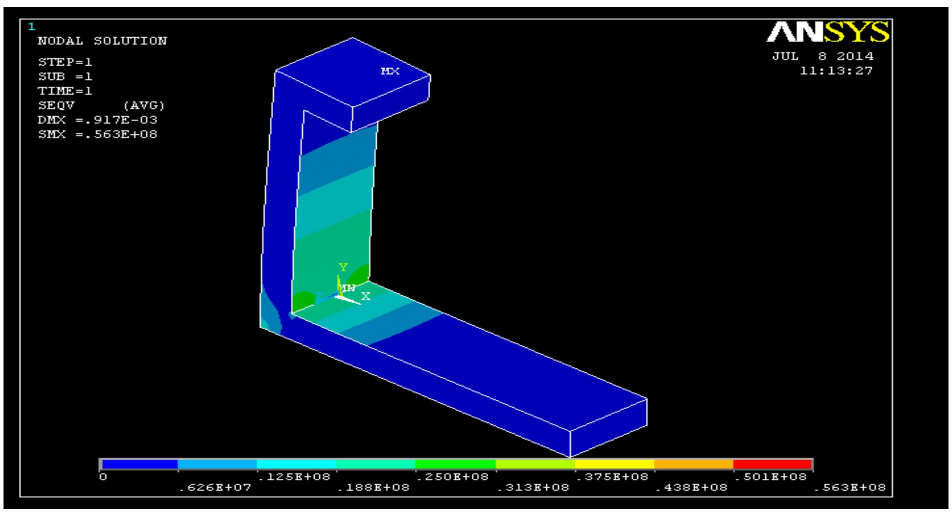


Figure: Von mises stress model

RESULT:

Case: 1:- To analyze shear of stiffened thin walled open section beam which is subjected to a pressure of 50 MPa.

1. DMX = 0.917E-03
2. DMX = 0.917E-03

   SMN = 0.598E-06

   SMX = 0.902E-03
3. DMX = 0.917E-03

   SMN = 0.468E+07

SMX = 0.120E+08

4. DMX = 0.917E-03

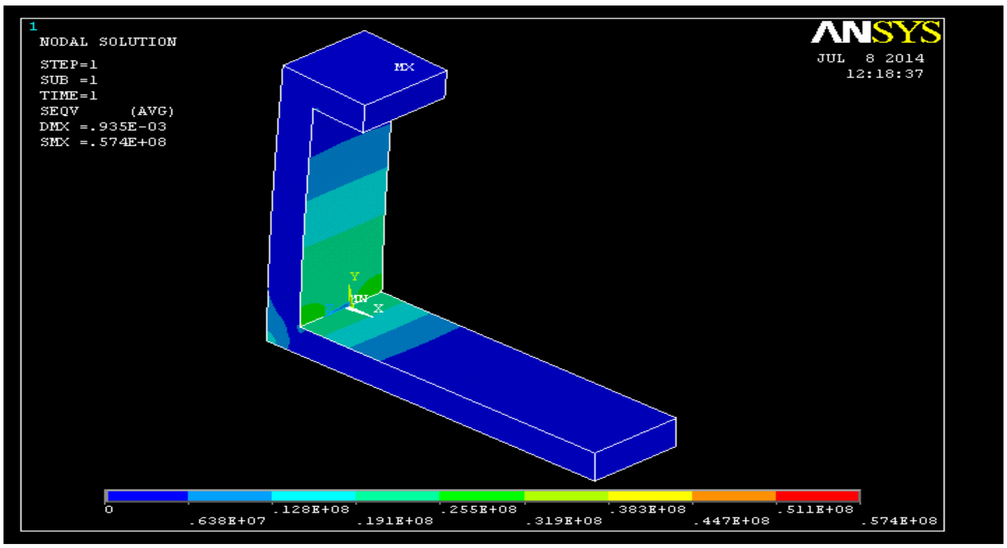
   SMX = 0.563E+08

PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To analyze shear of stiffened thin walled open section beam which is subjected to a pressure of 51 MPa.

1. DMX = 0.935E-03

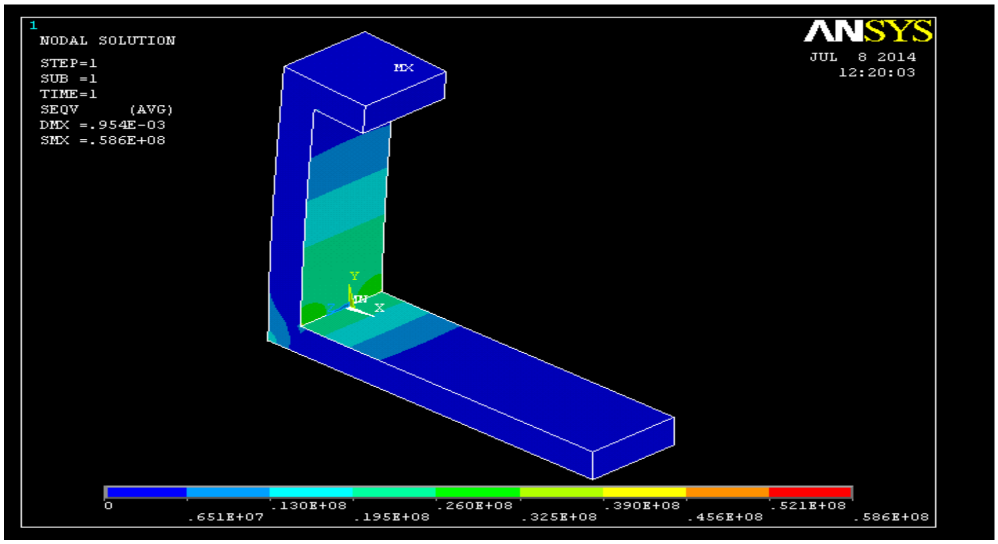
   SMX = 0.574E+08



Case: 3:- To analyze shear of stiffened thin walled open section beam which is subjected to a pressure of 52 MPa.

1. DMX = 0.954E-03

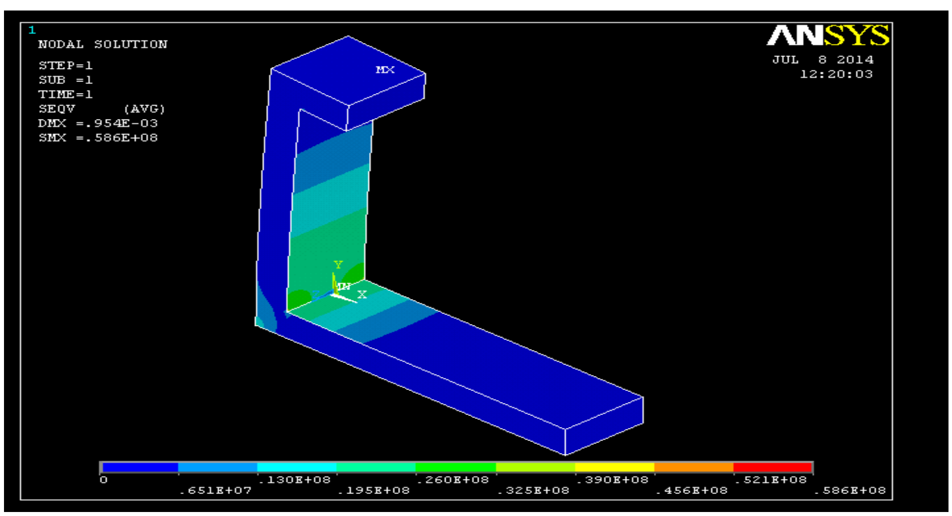
   SMX = 0.586E+08



Case: 4:- To analyze shear of stiffened thin walled open section beam which is subjected to a pressure of 53 MPa.

1. DMX = 0.972E-03

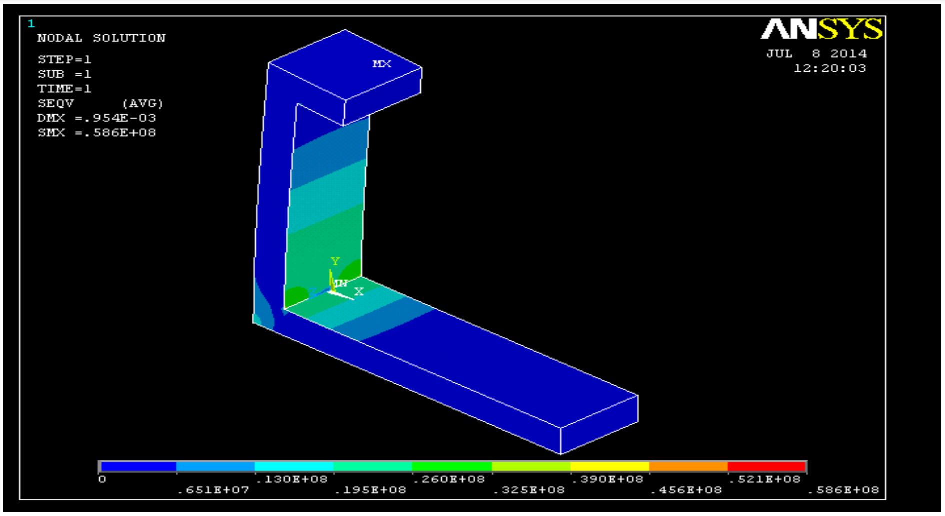
   SMX = 0.597E+08



Case: 5:- To analyze shear of stiffened thin walled open section beam which is subjected to a pressure of 54 MPa.

1. DMX = 0.990E-03

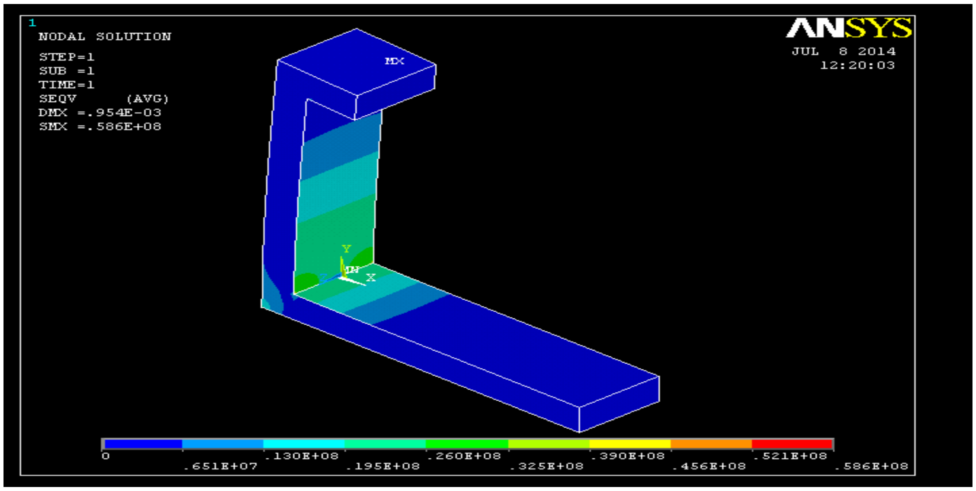
   SMX = 0.608E+08



Case: 6:- To analyze shear of stiffened thin walled open section beam which is subjected to a pressure of 55 MPa.

1. DMX = 1.08E-03

   SMX = 0.620E+08



EXPERIMENT: 3 (B)

TORSIONAL STRENGTH OF A THIN WALLED OPEN SECTION BEAM

- **Experiment as given in the JNTUH curriculum.**
- **TORSION OF STIFFENED THIN WALLED OPEN SECTION**

AIM: To analyze Torsion of stiffened thin walled open section beam which is subjected to a pressure of 20 MPa.

APPARATUS: Ansys 13.0

GIVEN DATA:

- Young's modulus = 0.7e11
- Thickness I = 1.3, J = 1.3
- Poisson's ratio = 0.3
- Density = 2700 kg/m^3

PROCEDURE:

PRE PROCESSING

STEP 1: From the Main menu select preferences

Select structural→ h-method and press OK

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

Element type → Add / edit/Delete → Add → select shell→elastic 4 node 63→apply→solid→quad 4 node 182→ok

Real constants → Add →Add→select type1 shell→ok → enter

Thickness→I =1.3, J=1.3 → ok → close

Material properties → material models → Structural → Linear → Elastic → Isotropic

EX = 0.7e11; PRXY = 0.3 & Density = 2700 → ok → close

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

- Create the key points in the Workspace

Create → Key points → In active CS

X	Y	Z
0	0	0
2	0	0

2	0.2	0
0.2	0.2	0
0.2	1.8	0
0.5	1.8	0
0.5	2	0
0	2	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points
  Create → Lines → Straight Line → Select 1-2, 2-3, 3-4, 4-5, 5-6, 6-7, 7-8, 8-1 Key points to generate lines

STEP 4: Modeling → create → Areas → arbitrary by lines → select all four lines → ok

Modeling → operate → extrude → areas → along normal → select the area → ok → enter the extrude length as 0.5

Select Plot controls from menu bar → Capture image → file save as and save your file

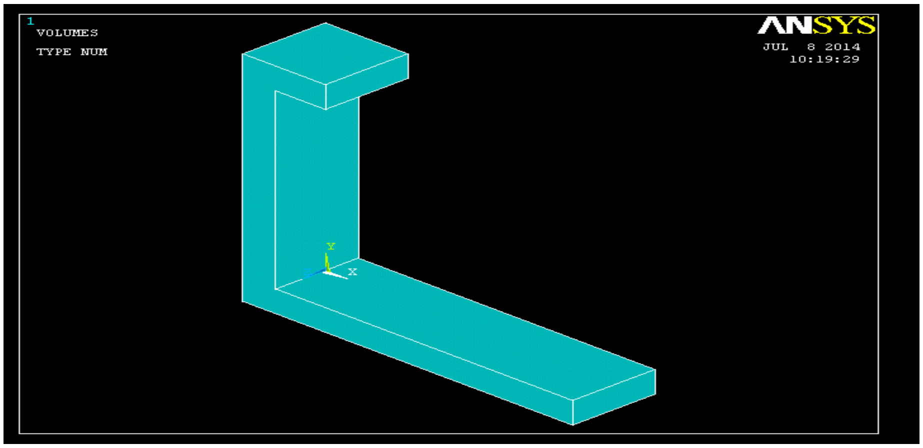


Figure: Open section beam model

STEP 5: Meshing the Geometry

From the main menu select **Meshing**

Meshing → mesh attributes → all areas → select the element type → no shell → ok

Select all volumes →select the element type number →plane →ok

Meshing → Size controls → Manual size → by areas → all areas → Number of element edge length = 0.025 → Click ok

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

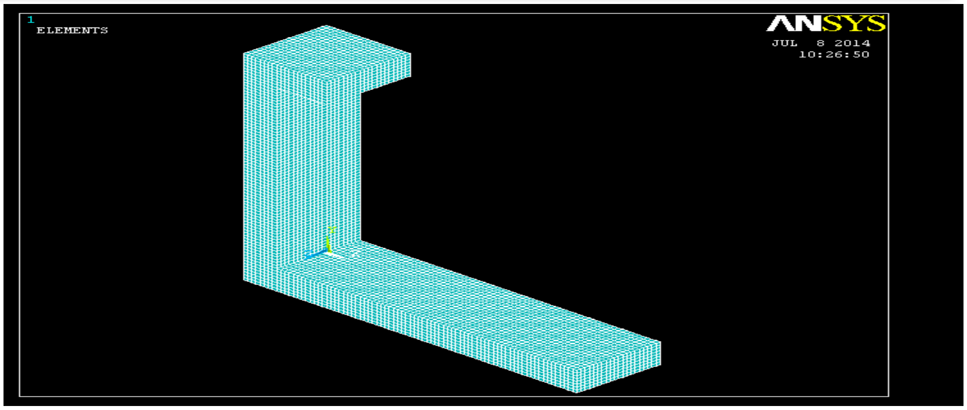


Figure: Open section beam meshed model

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

STEP 5: Defining loads

Loads → define loads → Apply → Structural → Displacement → On areas → select the front C/S area and select the bottom flange free end area → ok → all DOF → ok

Select → ALL DOF arrested

Define loads → Apply → Structural → Pressure → on areas → select the frontal area of web and free end area of top flange (20Mpa) → ok

Define loads → Apply → Structural → Pressure → on areas → select the back end area of web (-20 MPa) → ok

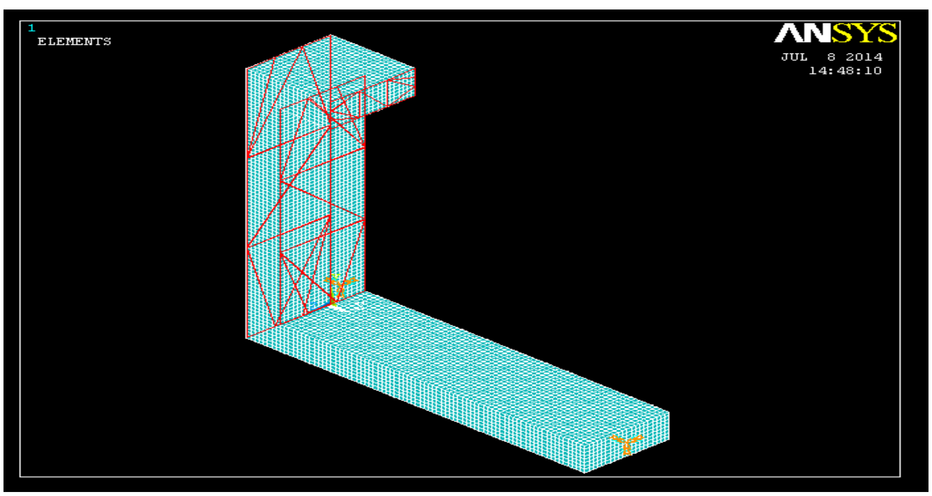


Figure: Boundary and operating conditions model

STEP 6: Solving the system

Solution →Solve → Current LS

POSTPROCESSING: VIEWING THE RESULTS

1. **Deformation**

   From the main menu select **General post processing**

   General post processing → Plot Results → Deformed Shape

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

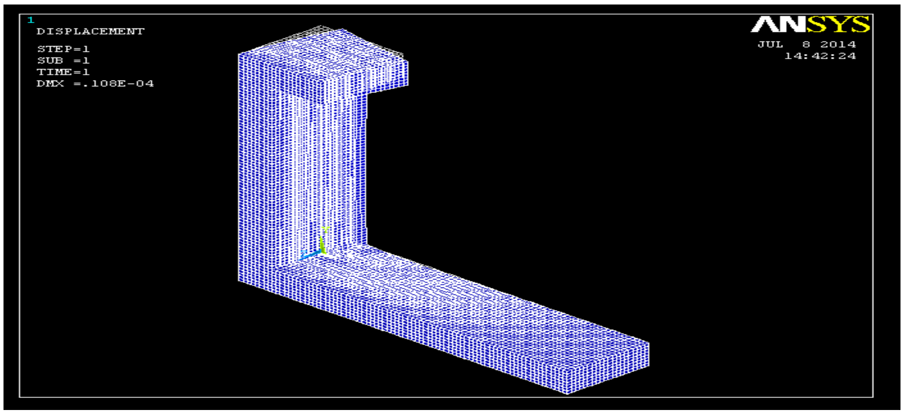


Figure: Deformed and undeformed model

Nodal solution

From the Utility menu select PLOT

PLOT →Results →Contour plot → Nodal solution

2.Select DOF solution → Y component of rotation → OK

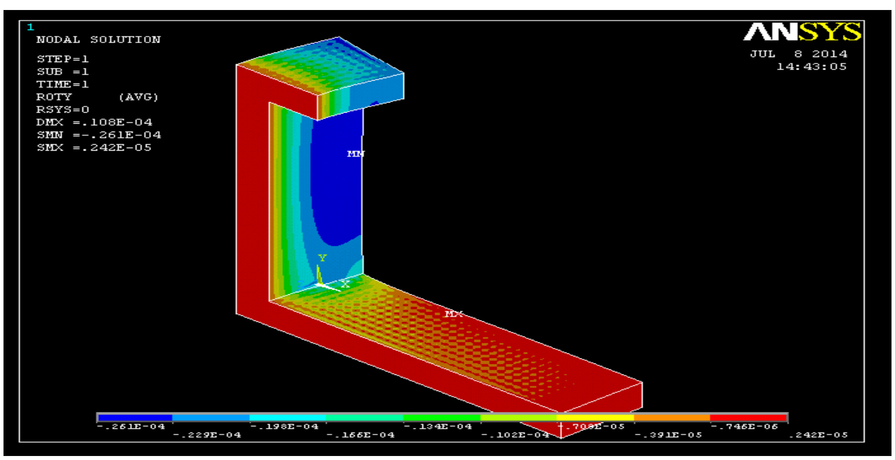


Figure: Y- component of rotation model

3.Select DOF solution → X component of displacement → OK

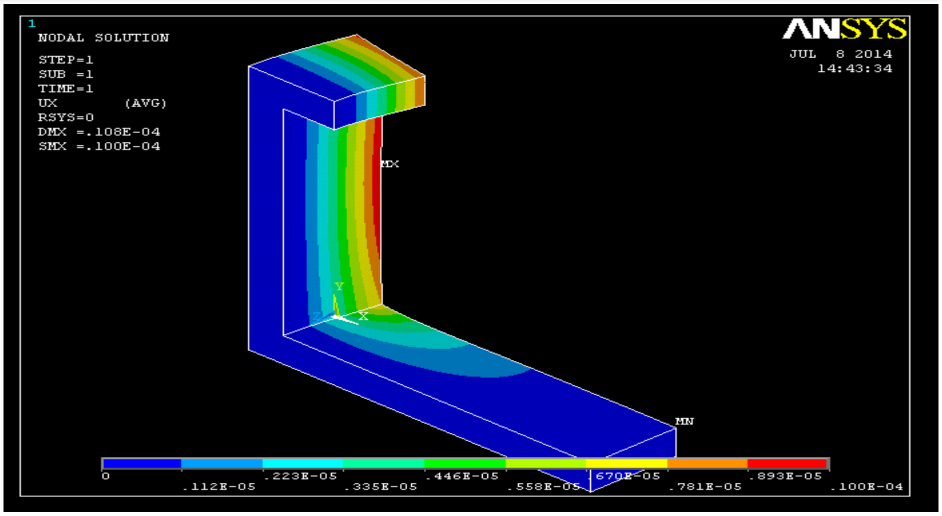


Figure: X- component of displacement model

4. Select stress → YZ shear stress

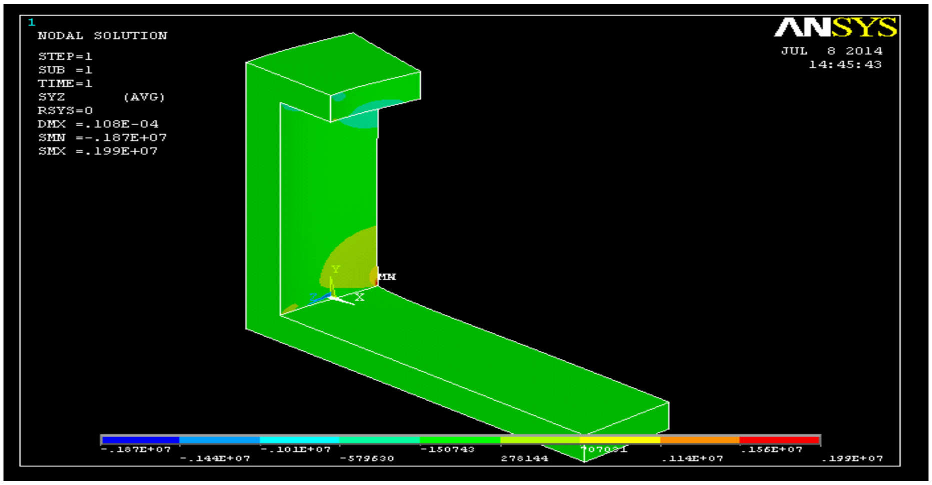


Figure: YZ shear stress model

5. Select stress → Von mises stress

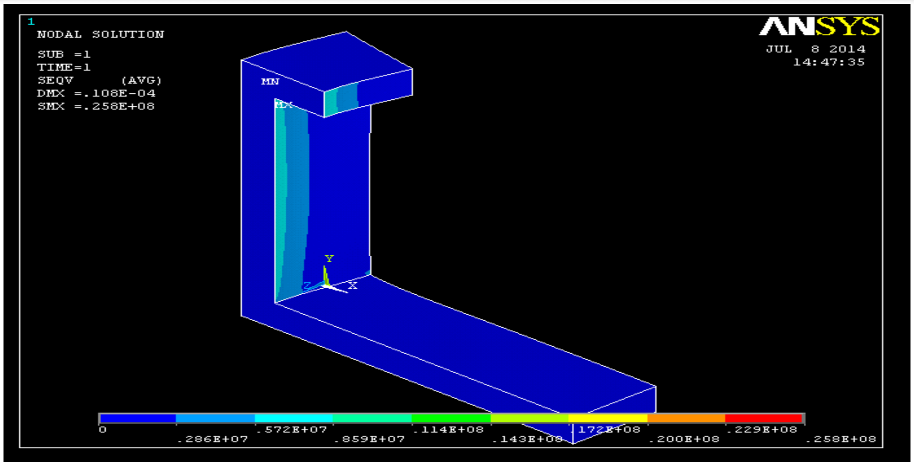


Figure: Von mises stress model

RESULT:

Case: 1:- To analyze Torsion of stiffened thin walled open section beam which is subjected to a pressure of 20 MPa.

1. DMX = 0.108E-04
2. DMX = 0.108E-04

   SMN = -0.261E-04

   SMX = 0.242E-05
3. DMX = 0.108E-04

   SMX = 0.100E-04
4. DMX = 0.108E-04

   SMN = 0.187E+07

   SMX = 0.199E+07
5. DMX = 0.108E-04

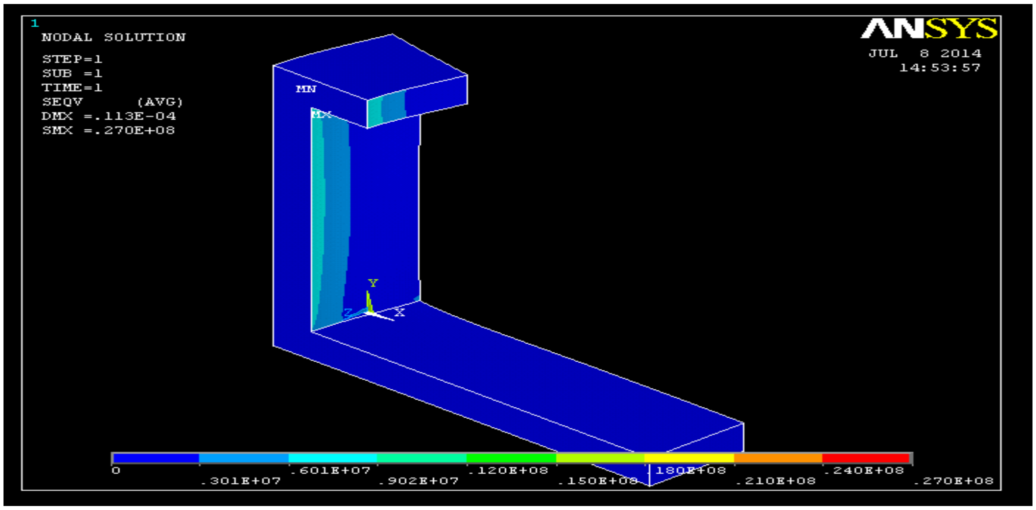
   SMX = 0.258E+08

PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To analyze Torsion of stiffened thin walled open section beam which is subjected to a pressure of 21 MPa.

1. DMX = 0.113E-04

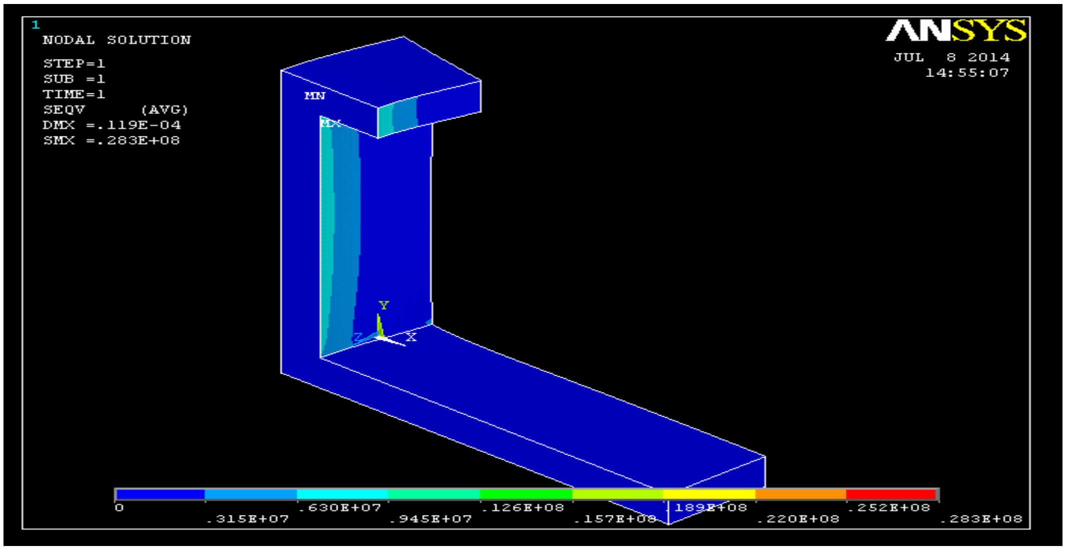
   SMX = 0.270E+08



Case: 3:- To analyze Torsion of stiffened thin walled open section beam which is subjected to a pressure of 22 MPa.

1. DMX = 0.119E-04

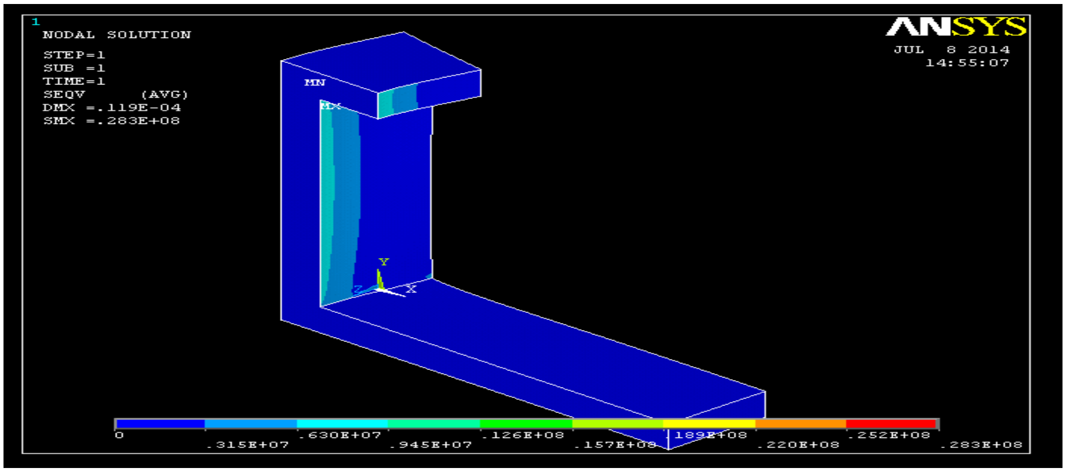
   SMX = 0.283E+08



Case: 4:- To analyze Torsion of stiffened thin walled open section beam which is subjected to a pressure of 23 MPa.

1. DMX = 0.124E-04

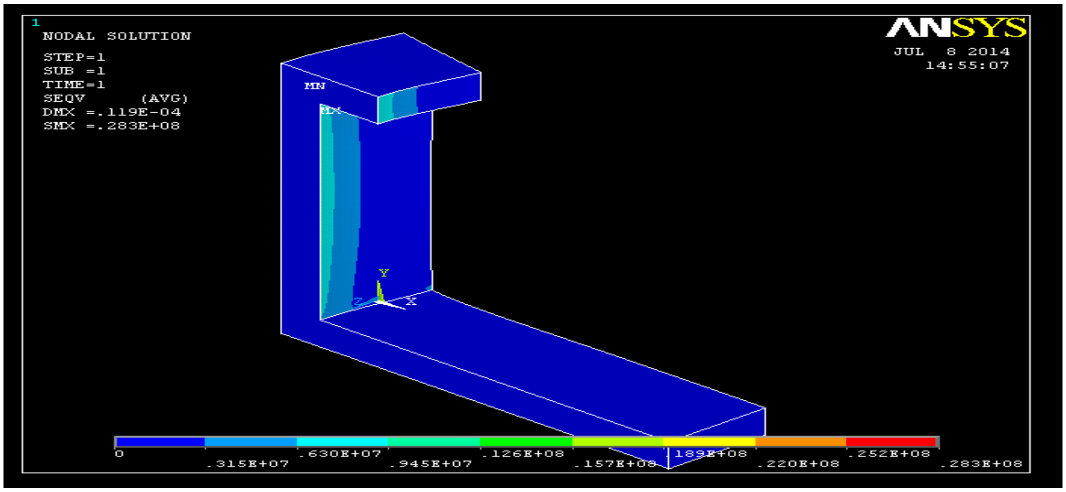
   SMX = 0.295E+08



Case: 5:- To analyze Torsion of stiffened thin walled open section beam which is subjected to a pressure of 24 MPa.

1. DMX = 0.130E-04

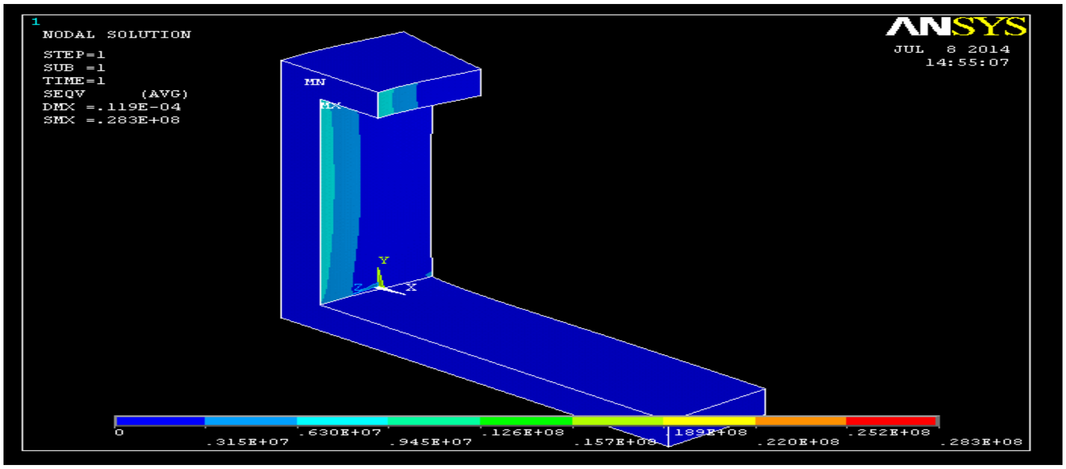
   SMX = 0.308E+08



Case: 6:- To analyze Torsion of stiffened thin walled open section beam which is subjected to a pressure of 25 MPa.

1. DMX = 0.135E-04

   SMX = 0.321E+08



EXERCISE PROBLEM

1) Find the compressive strength of a tapered stiffened panel with larger dimension as 0.5m and smaller dimension as 0.1m with distance between these dimensions are 1m.

2)Apply a UDL on a tapered cantilever beam to find out the deflection at free end, for different materials, say aluminum and Mild steel. State the observations.

VIVA QUESTIONS

1. Define shear flow.
2. Define Torsion.
3. Write down the torsion equation.
4. Define von mises stress.
5. Define elastic constants.

EXPERIMENT: 3 (C)

SHEAR FORCE OF STIFFENED THIN WALLED CLOSED SECTION BEAM

- **Experiment as given in the JNTUH curriculum.**
- **Shear of stiffened thin walled closed section**

AIM: To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 50 MPa.

APPARATUS: Ansys 13.0

GIVEN DATA:

- Young's modulus = 0.7e11
- Thickness I = 1.3, J = 1.3
- Poisson's ratio = 0.3
- Density = 2700 kg/m^3

PROCEDURE:

PRE PROCESSING

STEP 1: From the Main menu select preferences

Select structural→ h-method and press OK

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

Element type → Add / edit/Delete → Add → select shell→elastic 4 node 63→apply→solid→quad 4 node 182→ok

Real constants → Add →Add→select type1 shell→ok → enter

Thickness→I =1.3, J=1.3 → ok → close

Material properties → material models → Structural → Linear → Elastic → Isotropic

EX = 0.7e11; PRXY = 0.3 & Density = 2700 → ok → close

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

- Create the key points in the Workspace

Create → Key points → In active CS

X	Y	Z
0	0	0
1	0	0

0.2	0.2	0
0.8	0.2	0
0.2	1.8	0
0.8	1.8	0
0	2	0
1	2	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points
  Create → Lines → Straight Line → Select 1-2, 2-8, 8-7, 7-1, 3-4, 4-6, 6-5, 5-3 Key points to generate lines

STEP 4: Modeling → create → Areas → arbitrary by lines → select all lines → ok

Modeling → operate → extrude → areas → along normal → select the area → ok → enter the extrude length as 0.75

Select Plot controls from menu bar → Capture image → file save as and save your file

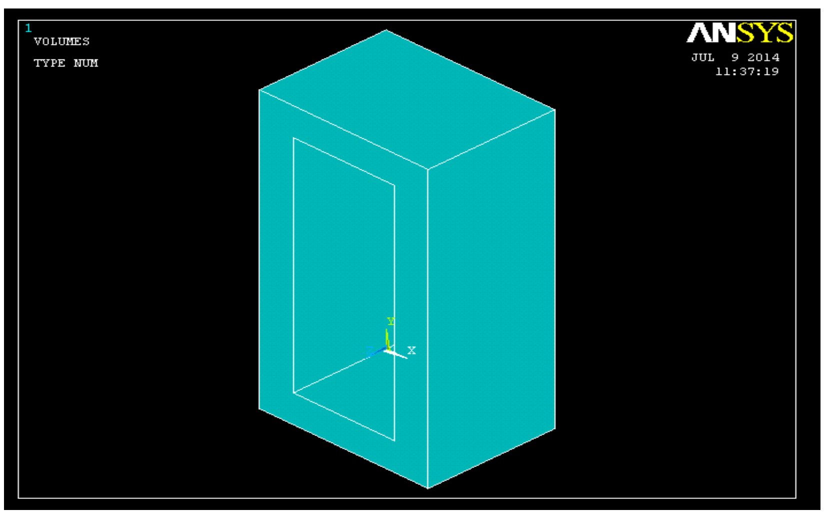


Figure: Closed section beam model

STEP 5: Meshing the Geometry

From the main menu select **Meshing**

Meshing → mesh attributes → all areas → select the element type → no shell → ok

Select all volumes →select the element type number →plane →ok

Meshing → Size controls → Manual size → by areas → all areas → Number of element edge length = 0.025 → Click ok

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

Select Plot controls from menu bar → Capture image → file save as and save your file

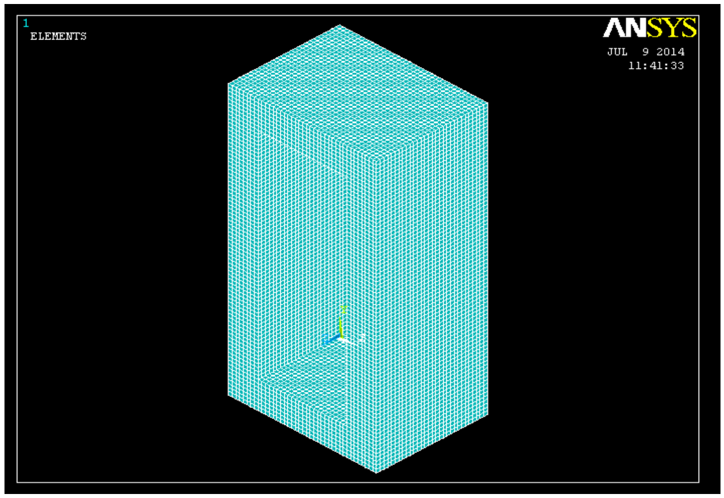


Figure: Closed section beam meshed model

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

STEP 5: Defining loads

Loads → define loads → Apply → Structural → Displacement → On areas → select the bottom edge → ok → all DOF → ok

Select → ALL DOF arrested

Define loads → Apply → Structural → Pressure → on areas → select surface of web → ok

Enter pressure = 50 MPa → ok

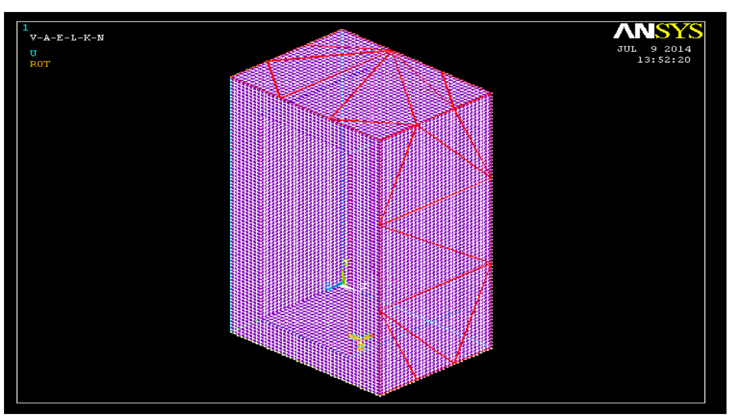


Figure: Boundary and operating conditions model

STEP 6: Solving the system

Solution →Solve → Current LS

POSTPROCESSING: VIEWING THE RESULTS

1. Deformation

From the main menu select **General post processing**

General post processing → Plot Results → Deformed Shape

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

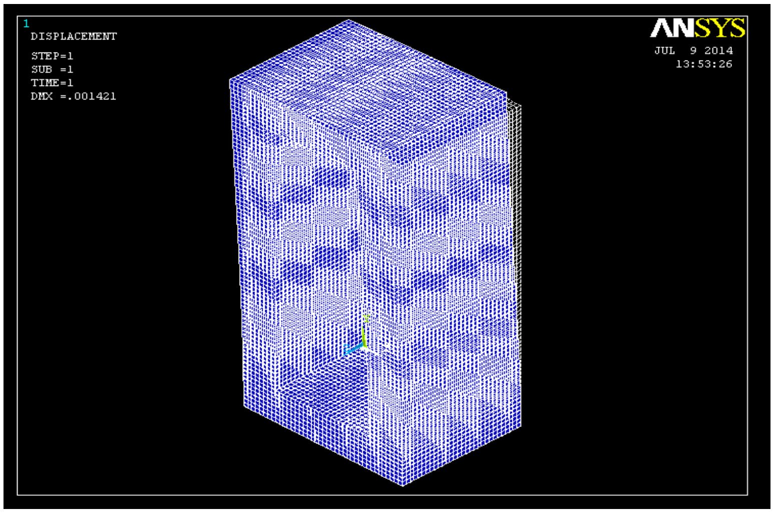


Figure: Deformed and undeformed model

Nodal solution

From the Utility menu select PLOT

PLOT →Results →Contour plot → Nodal solution

2. Select DOF solution → Y component of displacement → OK

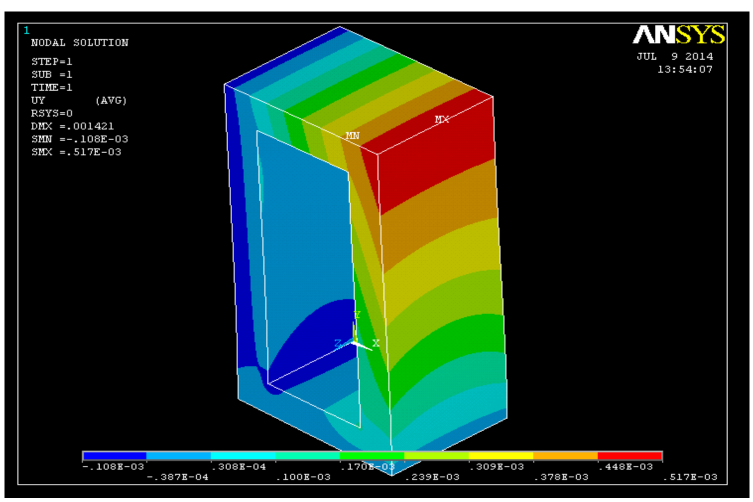


Figure: Y-Component of displacement model

3. Select stress → XZ shear stress

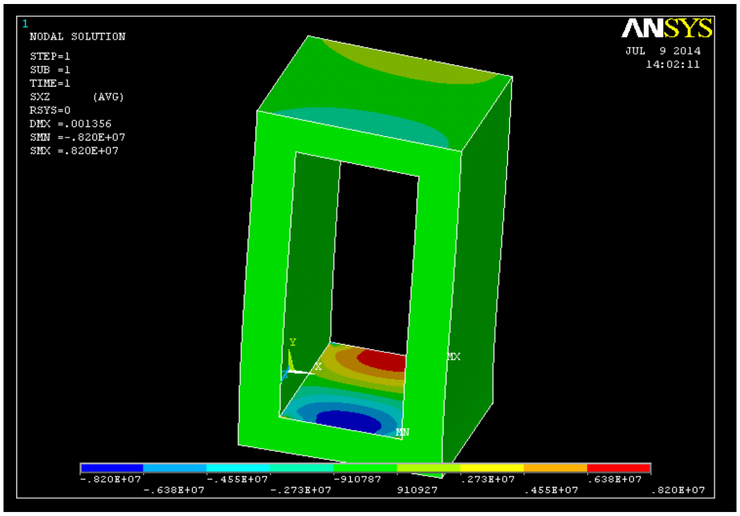


Figure: XZ shear stress model

4. Select stress → Von mises stress

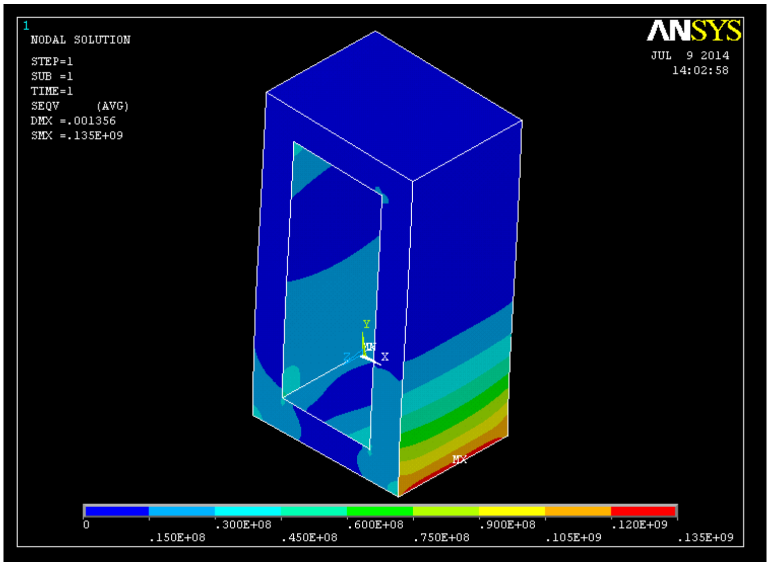


Figure: Von mises stress model

<u>RESULT:</u>

Case: 1:- To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 50 MPa.

1. DMX = 0.001421
2. DMX = 0.001421

SMN = -0.108E-06

SMX = 0.517E-03

3. DMX = 0.001421

   SMN = -0.820E+07

   SMX = 0.820E+07

4. DMX = 0.001421

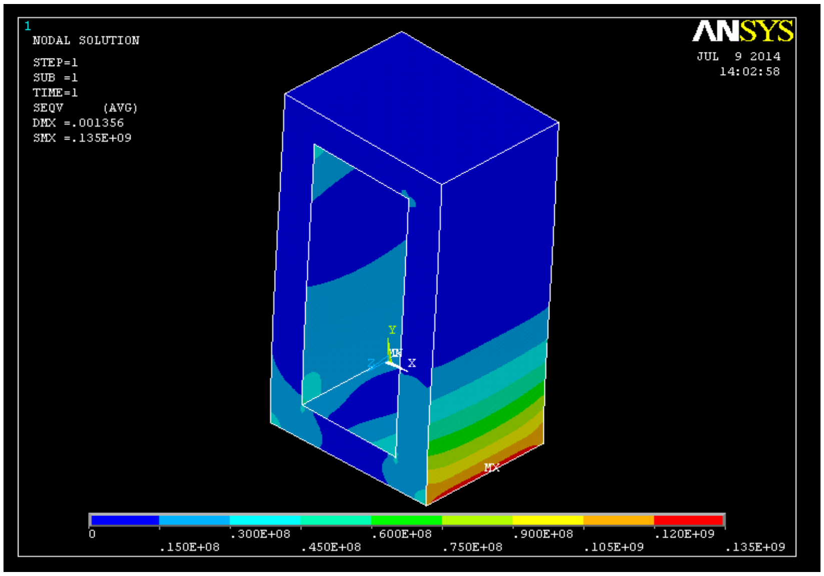
   SMX = 0.135E+09

PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 51 MPa.

DMX = 0.001383

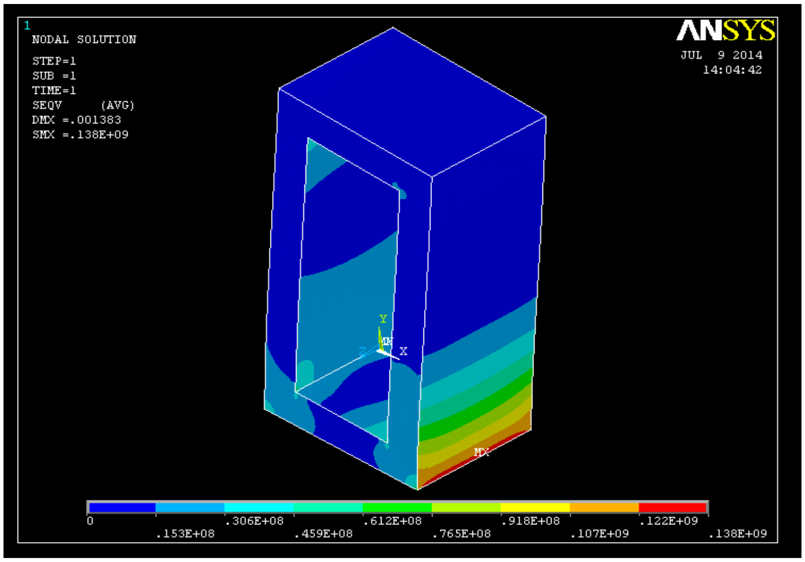
SMX = 0.138E+09



Case: 3:- To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 52 MPa.

DMX = 0.00141

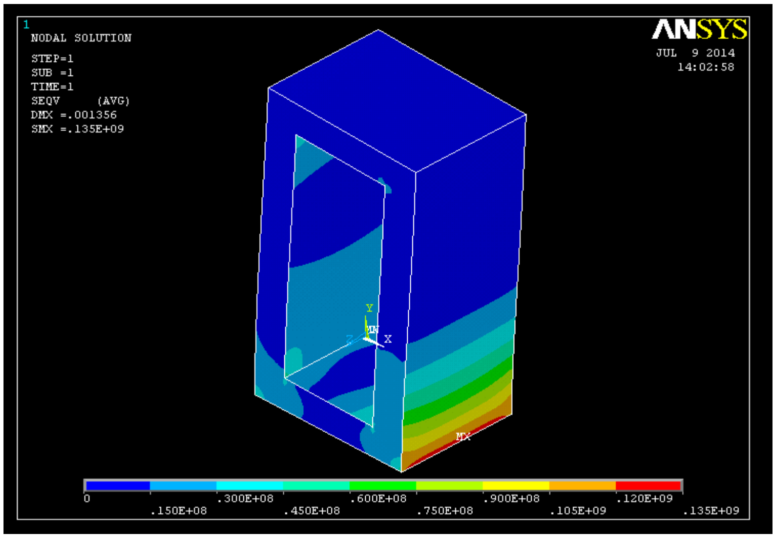
SMX = 0.140E+09



Case: 4:- To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 53 MPa.

DMX = 0.001437

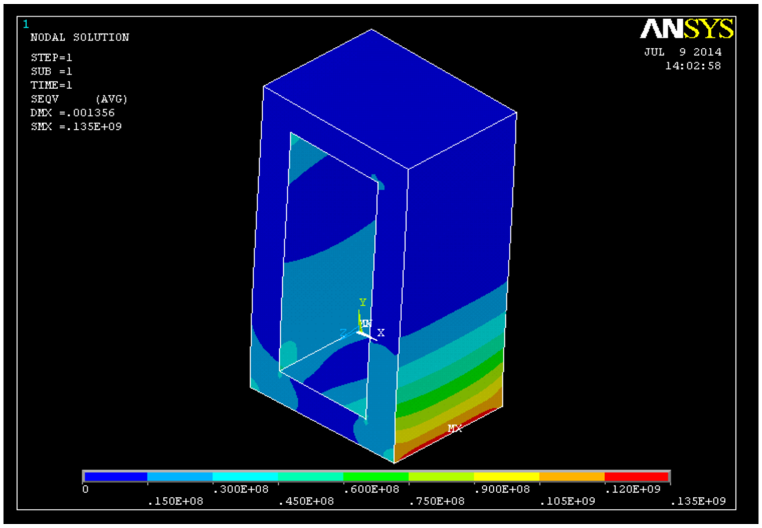
SMX = 0.143E+09



Case: 5:- To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 54 MPa.

DMX = 0.001464

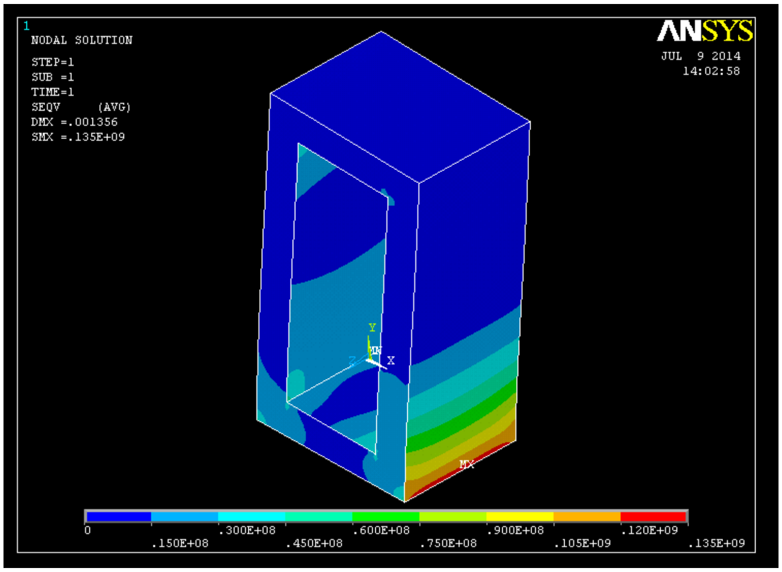
SMX = 0.145E+09



Case: 6:- To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 55 MPa.

DMX = 0.001491

SMX = 0.147E+09



EXERCISE PROBLEMS

1) Find the Shear force of a thin walled open section beam for the following

1. Z Section
2. T section

Consider flange dimension as 30cms and web dimension as 30cms, and shear force of 30MPa consider the material as Aluminum and Steel. State which one will be more efficient to install.

EXPERIMENT: 3(D)

TORSIONAL STRENGTH OF A THIN WALLED CLOSED SECTION BEAM

- **Experiment as given in the JNTUH curriculum.**
- **Shear of stiffened thin walled closed section**

AIM: To analyze shear of stiffened thin walled closed section beam which is subjected to a pressure of 20 MPa.

APPARATUS: Ansys 13.0

GIVEN DATA:

- Young's modulus = 0.7e11
- Thickness I = 1.3, J = 1.3
- Poisson's ratio = 0.3
- Density = 2700 kg/m^3

PROCEDURE:

PRE PROCESSING

STEP 1: From the Main menu select preferences

Select structural→ h-method and press OK

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

Element type → Add / edit/Delete → Add → select shell→elastic 4 node 63→apply→solid→quad 4 node 182→ok

Real constants → Add →Add→select type1 shell→ok → enter

Thickness→I =1.3, J=1.3 → ok → close

Material properties → material models → Structural → Linear → Elastic → Isotropic

EX = 0.7e11; PRXY = 0.3 & Density = 2700 → ok → close

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

- Create the key points in the Workspace

Create → Key points → In active CS

X	Y	Z
0	0	0
1	0	0
0.2	0.2	0

0.8	0.2	0
0.2	1.8	0
0.8	1.8	0
0	2	0
1	2	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points
  Create → Lines → Straight Line → Select 1-2, 2-8, 8-7, 7-1, 3-4, 4-6, 6-5, 5-3 Key points to generate lines

STEP 4: Modeling → create → Areas → arbitrary by lines → select all lines → ok

Modeling → operate → extrude → areas → along normal → select the area → ok → enter the extrude length as 0.75

Select Plot controls from menu bar → Capture image → file save as and save your file

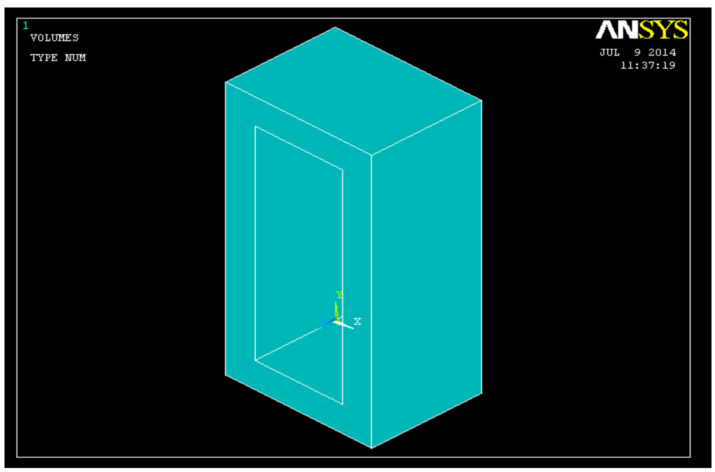


Figure: Closed section beam model

STEP 5: Meshing the Geometry

From the main menu select **Meshing**

Meshing → mesh attributes → all areas → select the element type → no shell → ok

Select all volumes →select the element type number →plane →ok

Meshing → Size controls → Manual size → by areas → all areas → Number of element edge length = 0.025 → Click ok

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

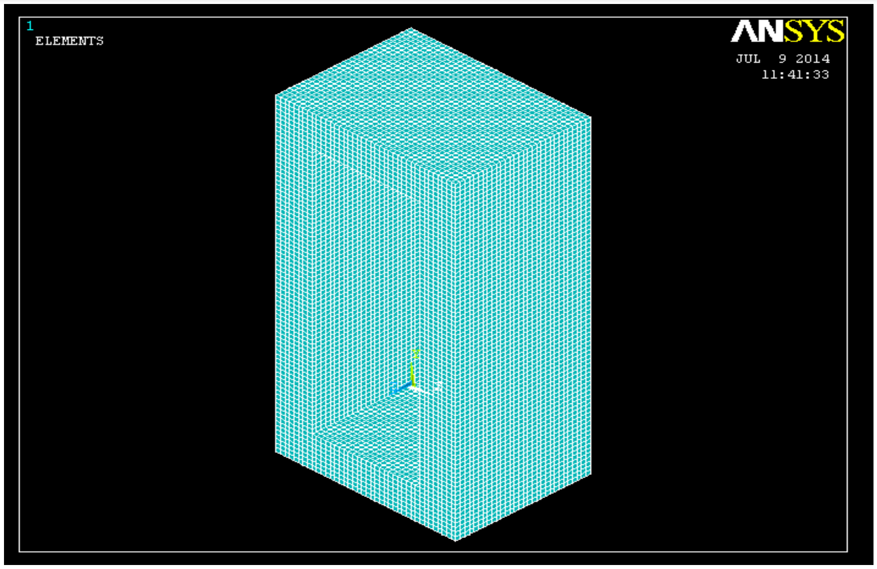


Figure: Closed section beam meshed model

<u>SOLUTION PHASE: ASSIGNING LOADS AND SOLVING</u>

STEP 5: Defining loads

Loads → define loads → Apply → Structural → Displacement → On areas → select the bottom edge and end C/S area of beam → ok → all DOF → ok

Select → ALL DOF arrested

Define loads → Apply → Structural → Pressure → on areas → select the extreme right of web (20Mpa) → ok

Define loads → Apply → Structural → Pressure → on areas → select the extreme left of web (-20 MPa) → ok

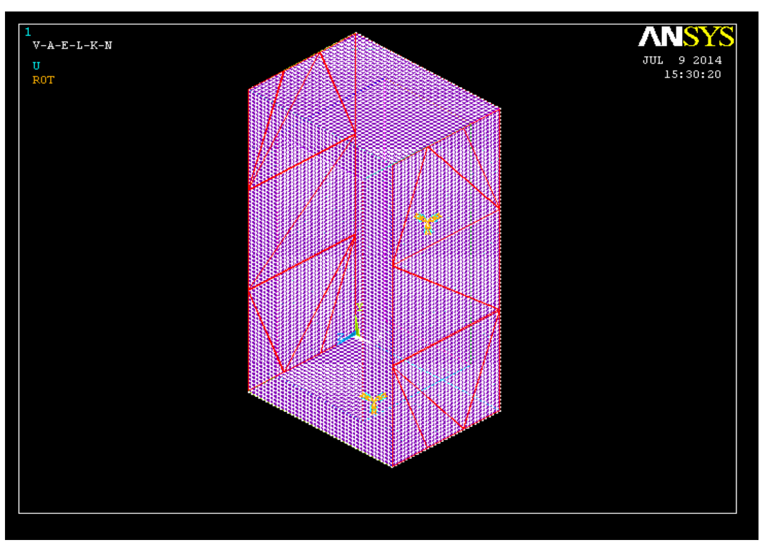


Figure: Boundary and operating condition model

STEP 6: Solving the system

Solution →Solve → Current LS

<u>POSTPROCESSING: VIEWING THE RESULTS</u>

1. **Deformation**

From the main menu select **General post processing**

General post processing → Plot Results → Deformed Shape

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

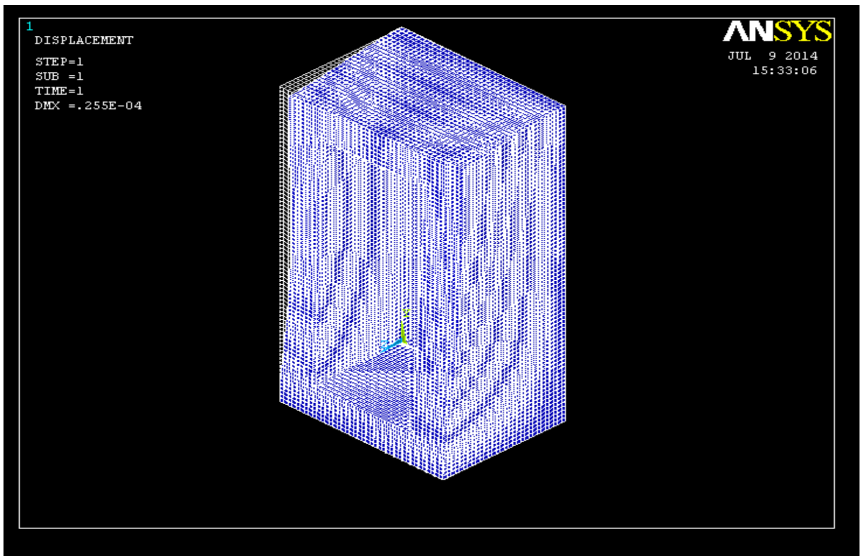


Figure: Deformed and undeformed model

Nodal solution

From the Utility menu select PLOT

PLOT →Results →Contour plot → Nodal solution

2. Select DOF solution → Y component of rotation → OK

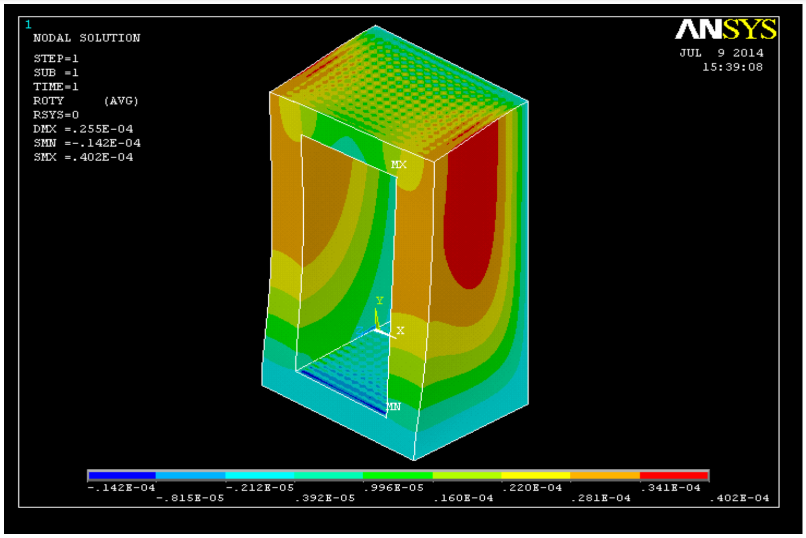


Figure: Y- component of rotation model

3. Select DOF solution → X component of displacement → OK

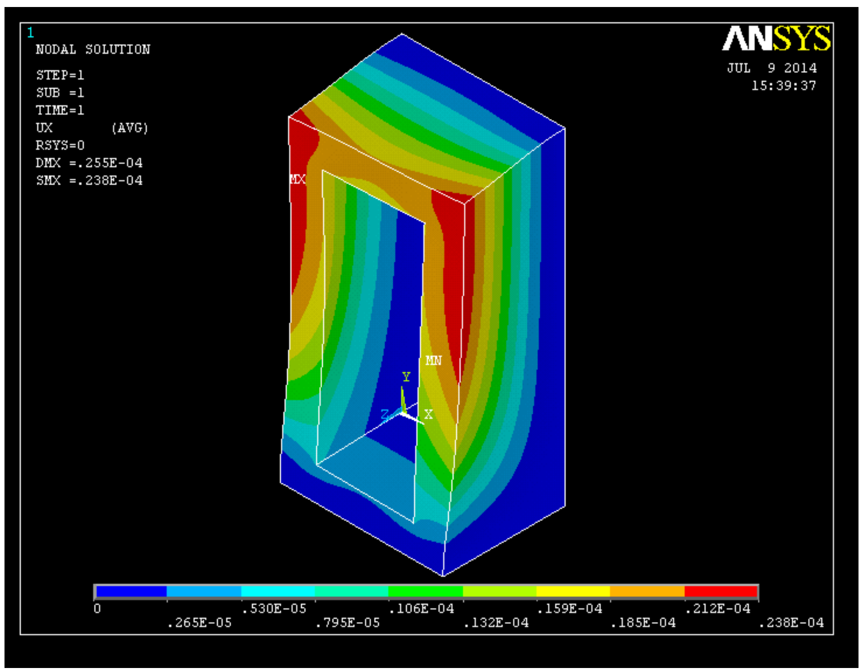


Figure: X- component of displacement model

4. Select stress → YZ shear stress

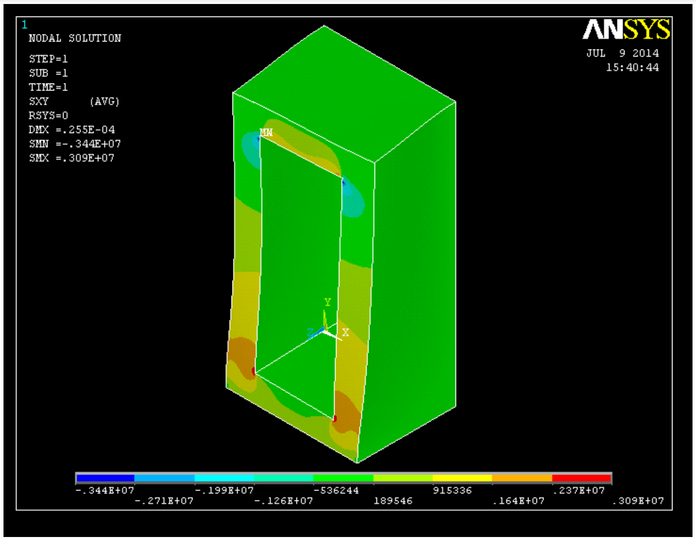


Figure: YZ shear stress model

5. Select stress → Von mises stress

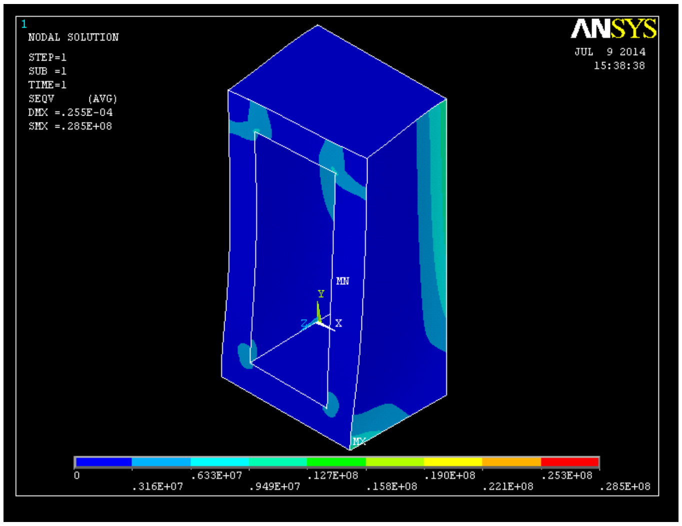


Figure: Von mises stress model

RESULT:

Case: 1:- To analyze Torsion of stiffened thin walled closed section beam which is subjected to a pressure of 20 MPa.

1. DMX = 0.255E-04
2. DMX = 0.255E-04

   SMN = -0.261E-04

   SMX = 0.242E-05
3. DMX = 0.255E-04

SMX = 0.238E-04

4. DMX = 0.255E-04

   SMN = -0.344E+07

   SMX = 0.309E+07

5. DMX = 0.255E-04

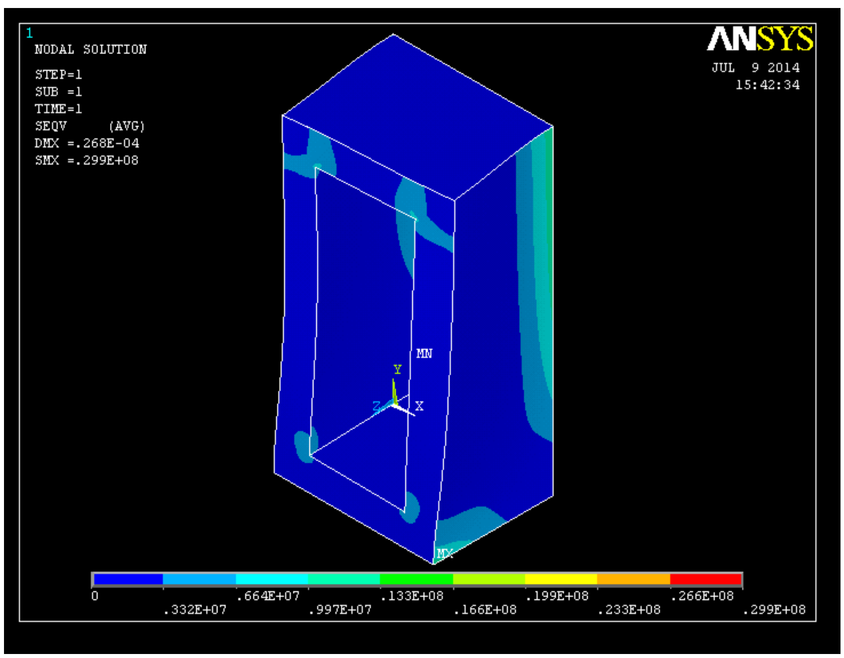
   SMX = 0.285E+08

PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To analyze Torsion of stiffened thin walled closed section beam which is subjected to a pressure of 21 MPa.

DMX = 0.268E-04

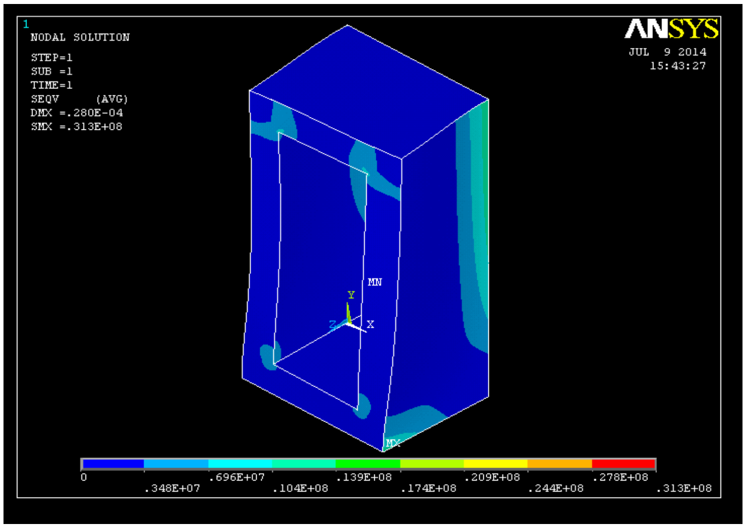
SMX = 0.299E+08



Case: 3:- To analyze Torsion of stiffened thin walled closed section beam which is subjected to a pressure of 22 MPa.

DMX = 0.280E-04

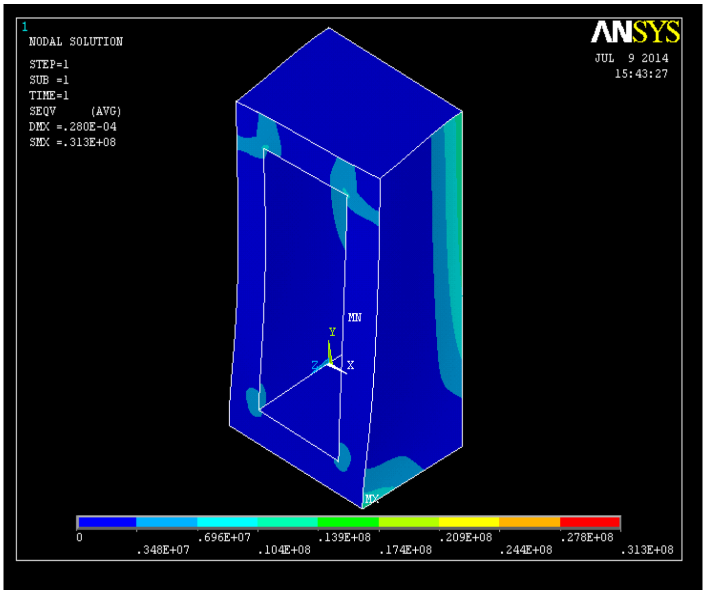
SMX = 0.313E+08



Case: 4:- To analyze Torsion of stiffened thin walled closed section beam which is subjected to a pressure of 23 MPa.

DMX = 0.293E-04

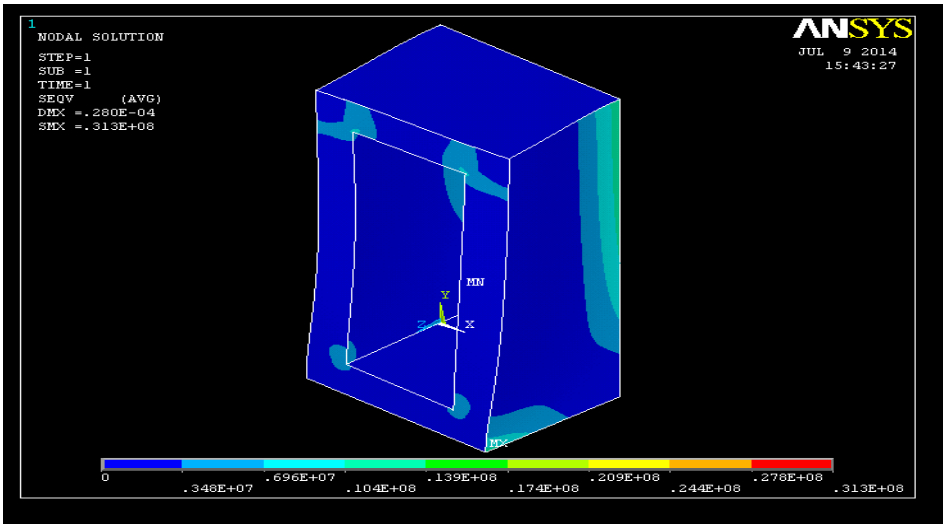
SMX = 0.327E+08



Case: 5:- To analyze Torsion of stiffened thin walled closed section beam which is subjected to a pressure of 24 MPa.

DMX = 0.305E-04

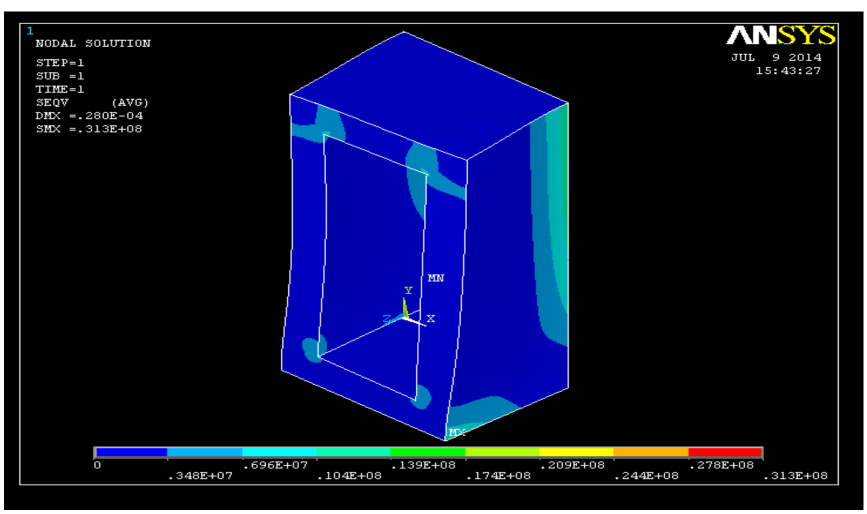
SMX = 0.341E+08



Case: 6:- To analyze Torsion of stiffened thin walled closed section beam which is subjected to a pressure of 25 MPa.

DMX = 0.318E-04

SMX = 0.355E+08



EXERCISE PROBLEM

Find the Torsional strength of a thin walled open section beam for the following

1. Z Section
2. T section

Consider flange dimension as 30cms and web dimension as 30cms, and pressure of 30MPa consider the material as Aluminum and Steel. State which one will be more efficient to install.

VIVA QUESTIONS

1. Define stiffness.
2. What are Boolean operations?
3. Define truss?
4. Name all the types of elements used in Ansys with example?
5. What is Poisson's ratio and give the steps for obtaining Poisson's ratio value.

EXPERIMENT: 4

2-D STATIC LINEAR ANALYSIS OF A TRUSS STRUCTURE

- **Experiment as given in the JNTUH curriculum.**
- **Statically indeterminate truss.**

AIM: To determine the nodal deflections, reaction forces, and stress of the indeterminate truss system when it is subjected to a load of 8000 N. (E = 200GPa, A = 3250mm^2)

PROCEDURE:

PREPROCESSING

STEP 1: From the Main menu select preferences

Select structural and press OK

STEP 2: From the main menu select **Preprocessor**

Element type → Add / edit/Delete → Add → Link – 2D spar 8 → ok →close

Real constants → Add → Geometric Properties → Area = 3250

Material properties → material models → Structural → Linear → Elastic → Isotropic

EX = 2e5; PRXY = 0.3

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

- Create the key points in the Workspace

Pre-processor → Modeling → Create → Nodes → In active CS

X	Y	Z
0	0	0
5	0	0
10	0	0
15	0	0
2.5	2.5	0
7.5	2.5	0
12.5	2.5	0

Click APPLY to all the points and for the last point click OK

- Create LINES using the Elements

Pre-processor → Modeling → Create → Elements → Auto numbered → through nodes → select node 1&2 → apply →2&3 → apply→3&4 → apply→1&5 → apply→5&2

→ apply →2&6 → apply→6&3→ apply → 3&7 → apply → 7&4 → apply → 5&6 → apply→ 6&7 →ok → close

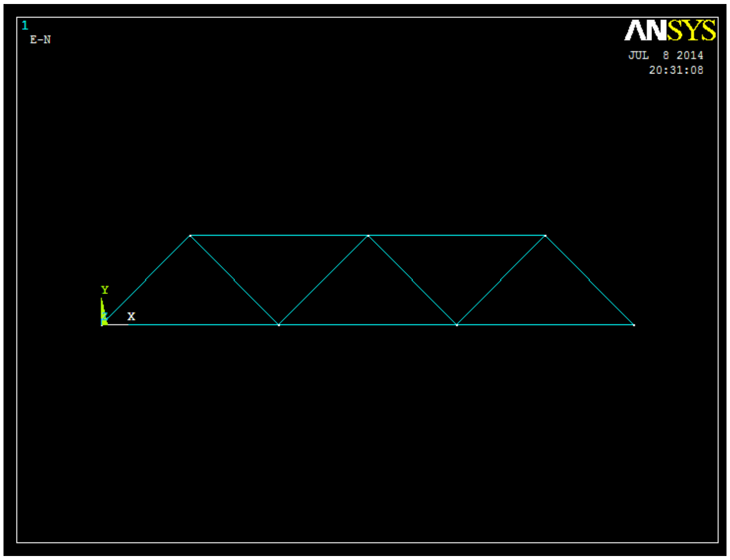


Figure: Model

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

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

Solution → Analysis type → new analysis – Static

STEP 6: Defining loads at the Key points

Solution → Define Loads → Apply → Structural → Displacement → On nodes → select node 1&4 →ok → select All DOF → ok

Left end – ALL DOF arrested

Solution → Define loads → Apply → Structural → Force/moment → On nodes

Select node 2&3 → ok FY direction → Give force value as -8000N → ok → close

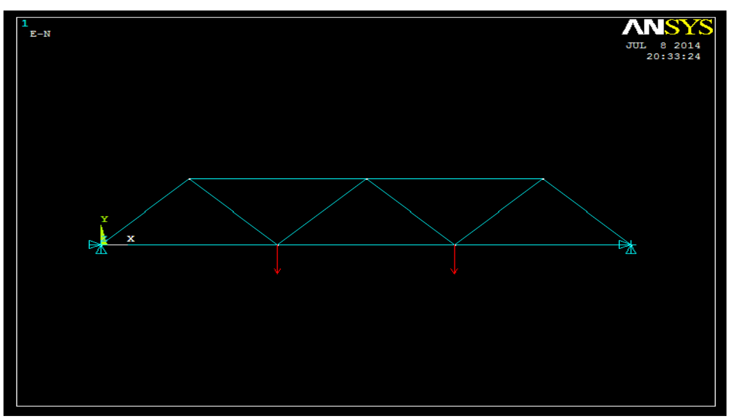


Figure: Model with boundary conditions

STEP 7: Solving the system

Solution →Solve → Current LS

POSTPROCESSING: VIEWING THE RESULTS

1. Deformation

From the main menu select **General post processing**

General post processing → Plot Results → Deformed Shape

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

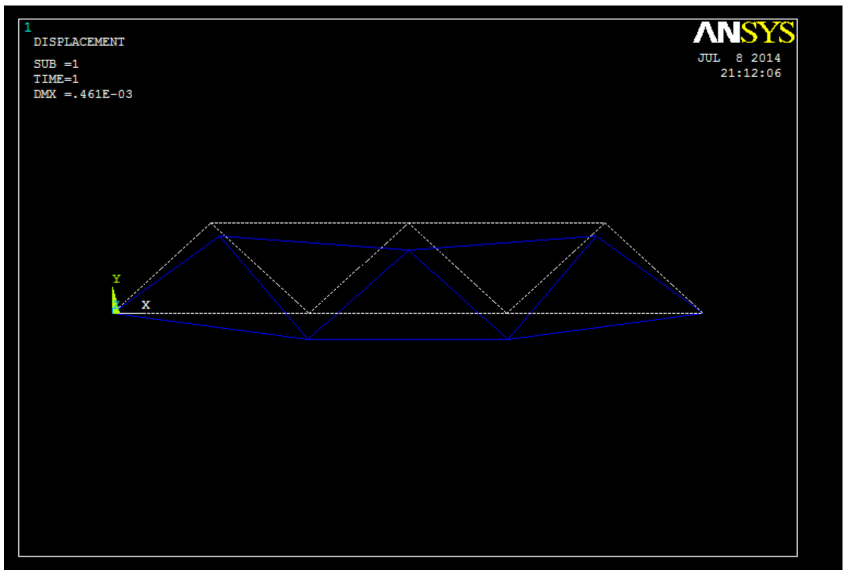


Figure: Deformed and undeformed Model

Nodal solution

From the Utility menu select PLOT

PLOT →Results →Contour plot → Nodal solution → DOF solution → Y component of displacement → OK

RESULT:

Case: 1:- To Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm2). At load -8000N

1. DMX = .461E-03

   SMN = -.461E-03

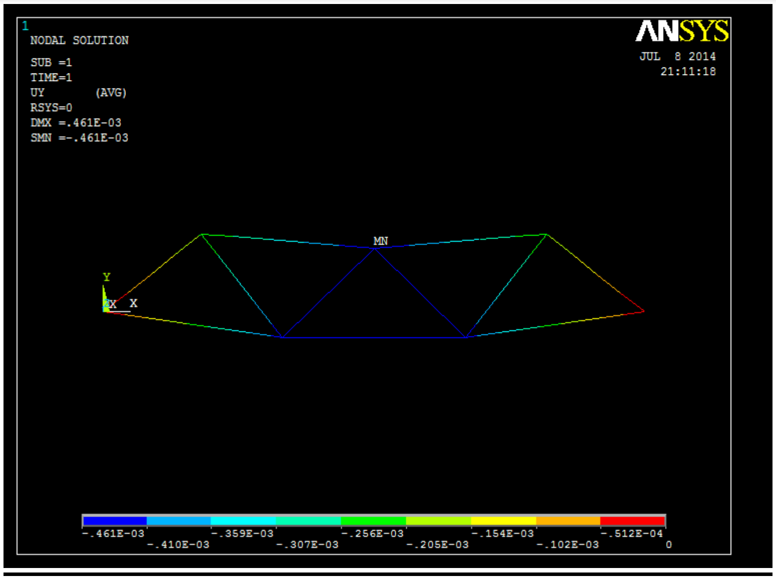


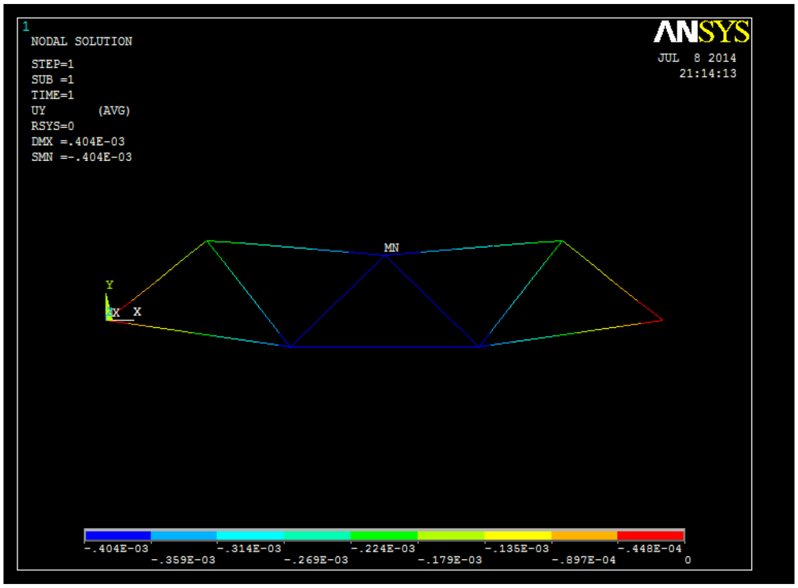
Figure: Y-Component displacement of the Model

PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm2). At load -7000N

2. DMX = .404E-03

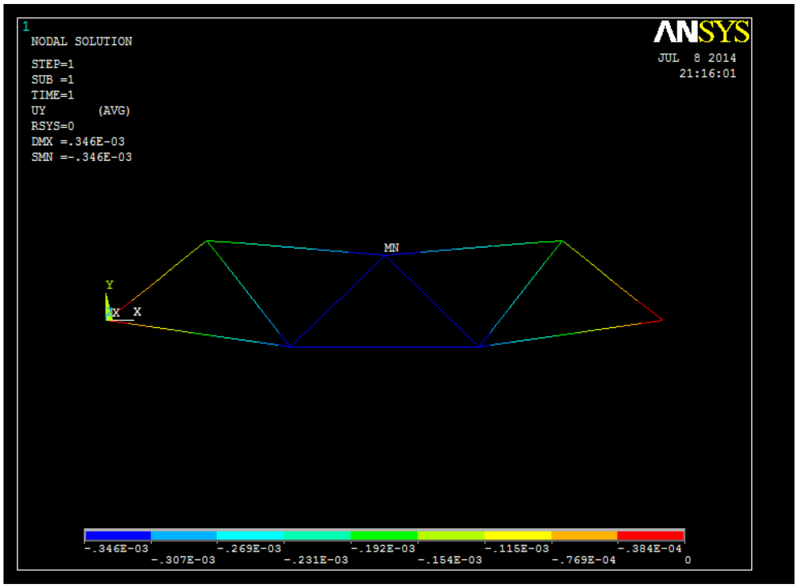
   SMN = -.404E-03



Case: 3:- To Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm2). At load -6000N

3. DMX = .346E-03

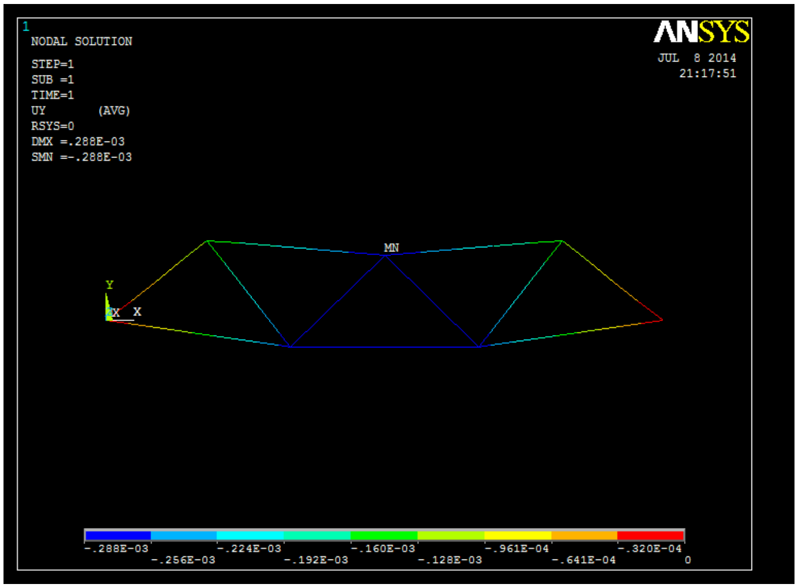
   SMN = -.346E-03



Case: 4:- To Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm2). At load -5000N

4. DMX = .288E-03

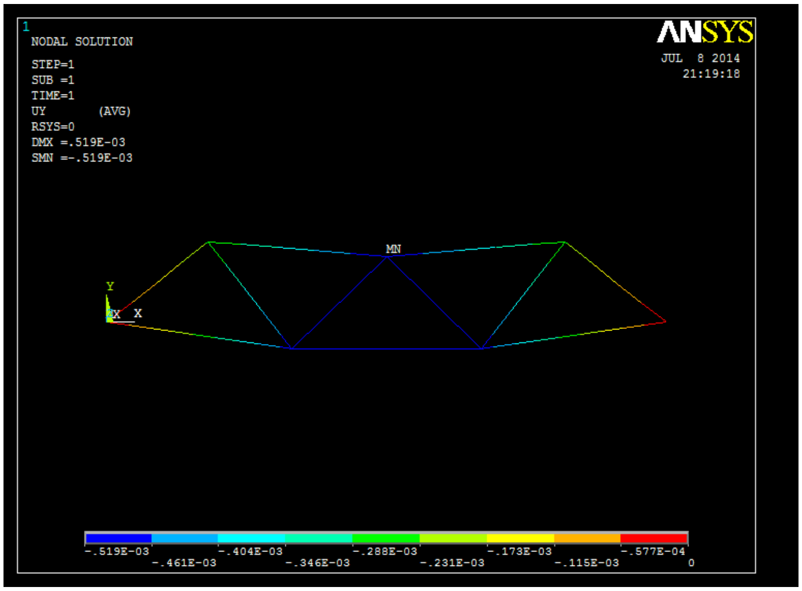
   SMN = -.288E-03



Case: 5:- To Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm2). At load -9000N

5. DMX = .519E-03

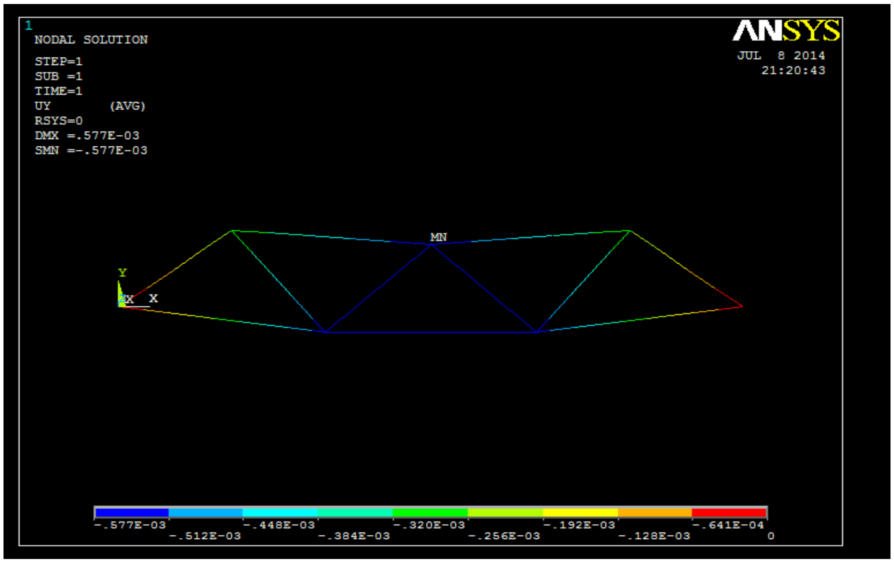
   SMN = -.519E-03



Case: 6:- To Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm2). At load -10000N

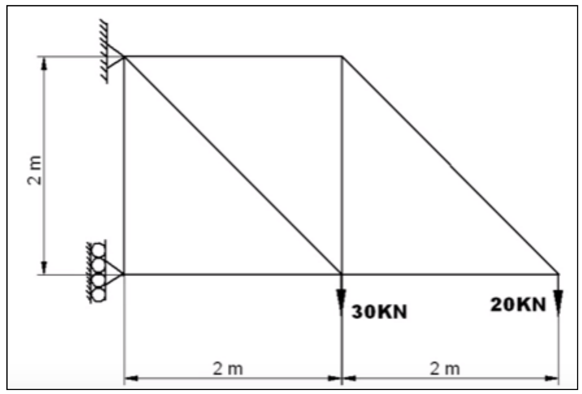
6. DMX = .577E-03

   SMN = -.577E-03



EXERCISE PROBLEM

1.Find the forces and stresses in the members of the truss.



VIVA QUESTIONS

1. Ansys needs the final element model(FEM) for its final solution.(T/F)
2. Element attributes must be set before meshing the solid model. (T/F)
3. In a plane strain, the strain in the direction of thickness is assumed to be zero.(T/F)
4. The ______ elements are used for in-plane bending problems.
5. Which one of the following elements is required to define the thickness as a real constant?
   a. Beam
   b. Shell
   c. Solid
   d. None

EXPERIMENT: 5

MODAL ANALYSIS OF UNIFORM CANTILEVER BEAM

- **Experiment as given in the JNTUH curriculum.**
- **Free vibration of uniform cantilever beam.**

Aim: Analyze the given uniform cantilever beam using Ansys and find out the variation in the frequencies for 5 mode shapes.

Apparatus: ANSYS Software 13.0

Given Data:

Young's Modulus: 2e5

Poisson's Ratio: 0.27

Length of the beam: 1000

Steps of Modeling:

Preferences ▶ Structural ▶ H- method ▶ OK

Preprocessor ▶ Element Type ▶ Add ▶ Add ▶

Beam ▶ 2D elastic 3 ▶ Apply▶ OK

Real constants ▶ add ▶ beam 3 ▶ Area = 1025

▶ I_{zz} = 450

▶ thickness = 6 & width 25 mm

Material Properties ▶ Material Models ▶ Structural ▶ Linear ▶ Elastic ▶ Isotropic ▶

EXX: 2e5

PRXY: .27

Density: 2870

Modeling ▶ Create ▶ Key points ▶ In Active CS

	X	Y	Z
1.	0	0	0
2.	1000	0	0

Pre-processor → Modelling → Create → Lines → Straight Line → Click on Key points to generate lines

Meshing the Geometry

From the main menu select **Meshing**

Meshing → Size controls → Manual size → Lines → All lines – Number of element divisions = 1 → Click OK

Meshing → Mesh → Lines – pick all

Defining loads

Loads ► Define Loads ► Apply ► Structural ► Displacement ► On nodes ► Select node 1 ► Select All DOF ► OK

Solution

Loads ► Analysis Type ► New Analysis ► Select Modal ► OK

Loads ► Analysis Option ► No.of Mode Shapes = 5 ► OK

Enter the Start Freq = 0

End Frequency = 0 ► OK

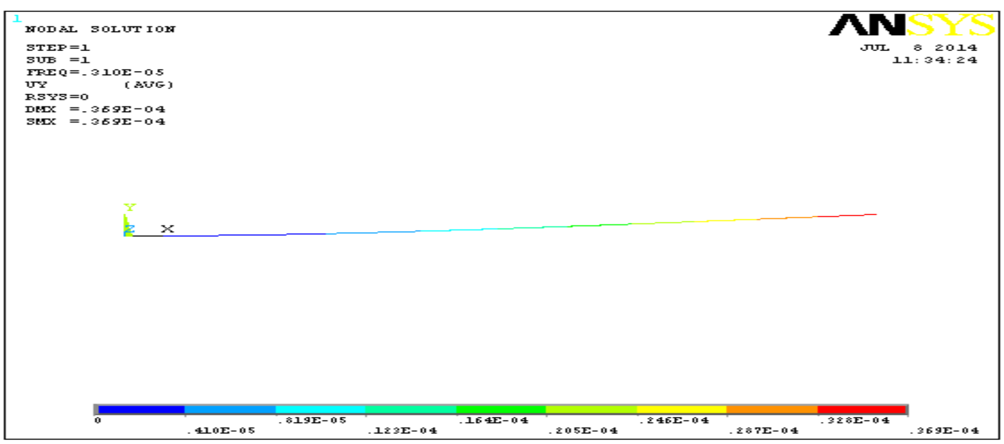
Solution ► Solve ► Current LS ► Warnings can be ignored ► Solution is Done

RESULTS:

General Post Processor ► Read Results ► by Pick

RESULT:

Case: 1:- To determine the 1st mode frequency acting on cantilever beam.

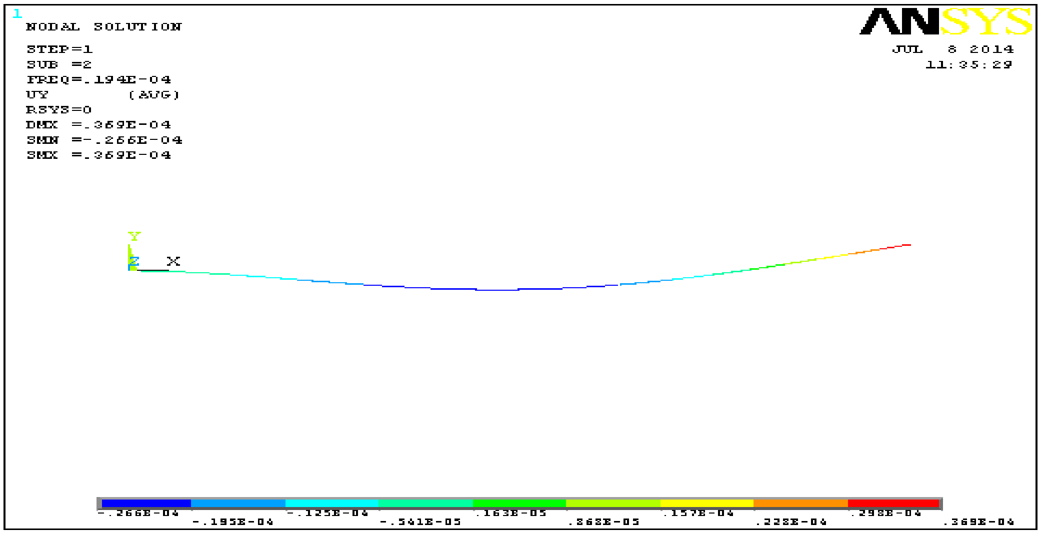


DMX: 0.369e-04

Frequency: 0.310e-05

PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

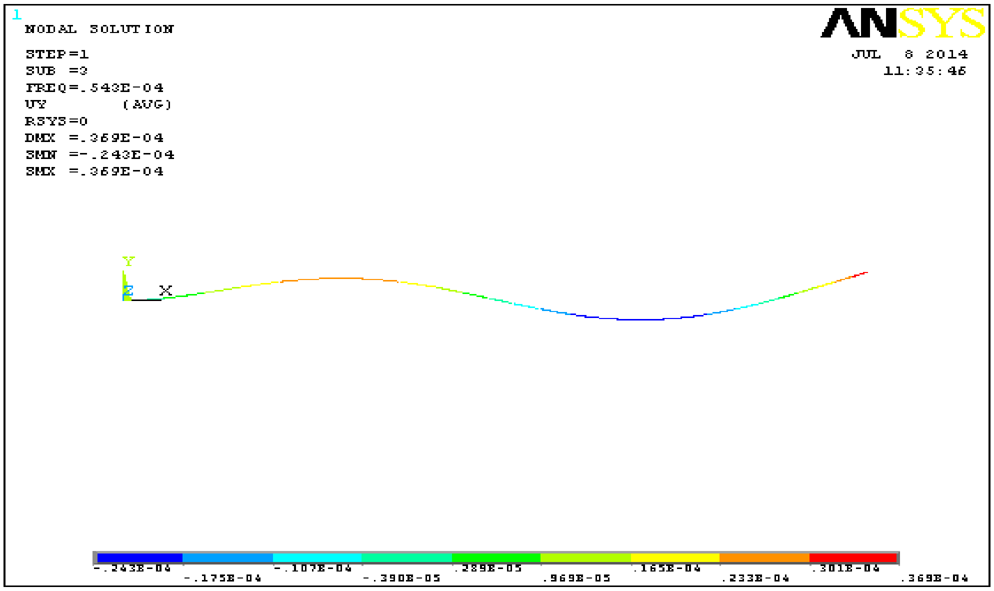
Case: 2:- To determine the 2nd mode frequency acting on cantilever beam.



DMX: 0.369e-04

Frequency: 0.194e-04

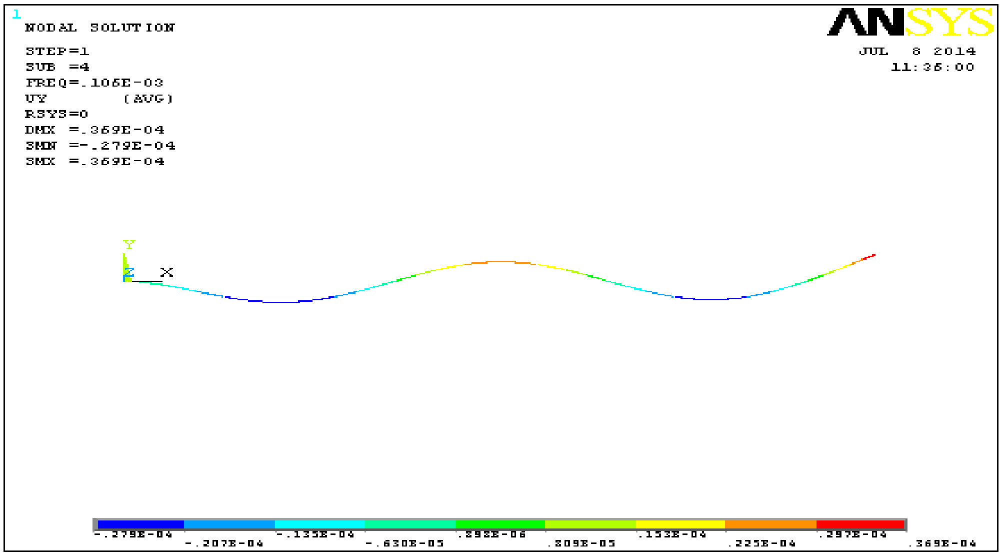
Case: 3:- To determine the 3rd mode frequency acting on cantilever beam.



DMX: 0.369e-04

Frequency: 0.543e-04

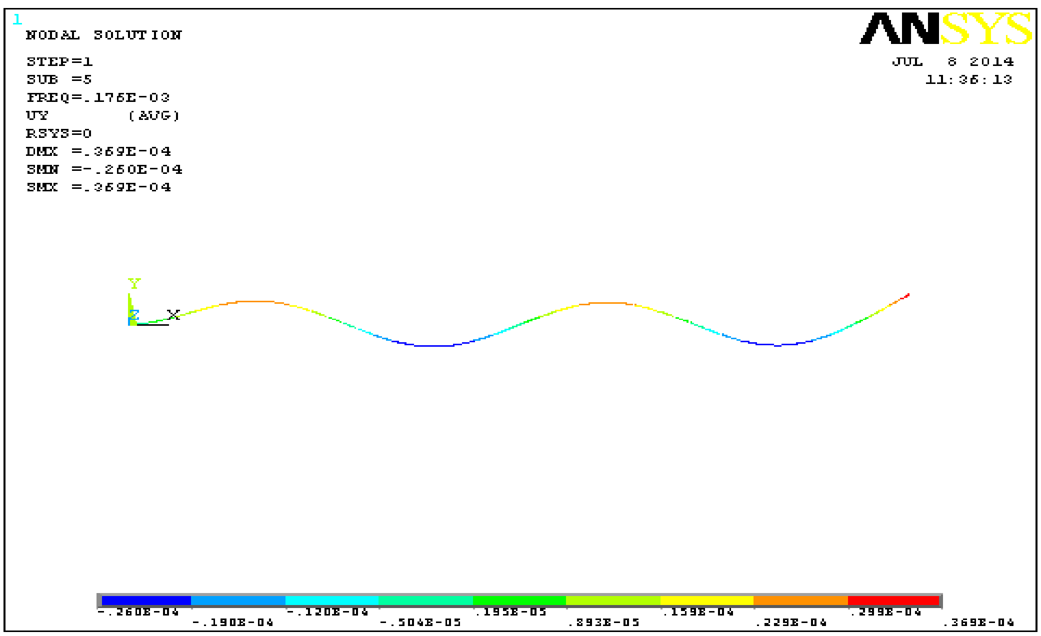
Case: 4:- To determine the 3rd mode frequency acting on cantilever beam.



DMX: 0.369e-04

Frequency: 0.106e-04

Case: 4:- To determine the 3rd mode frequency acting on cantilever beam.



DMX: 0.369e-04

Frequency: 0.176e-04

EXERCISE PROBLEM

Perform the modal analysis of the same cantilevered beam under different isotropic material and state your observation.

VIVA QUESTIONS

1. Name the types of meshing.
2. Explain the Main Steps involved in Ansys Programming.
3. What is Modal Analysis? Write the Steps involved in Modal Analysis.
4. How do you see the Animations of the Deformed Shapes in Ansys?
5. Write the Procedure for finding the SFD & BMD of a Link.

EXPERIMENT: 6

ANALYSIS OF A LANDING GEAR

- **Experiment as given in the JNTUH curriculum.**
- **3 dimensional landing gear trusses.**

Aim: Analyze the given landing gear structure with applied load of 10000N.

Apparatus: Ansys Software 13.0 Version

Given Data:

Angle (Strut):60 degrees

Poisson's Ratio=0.3

Steps of Modeling:

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

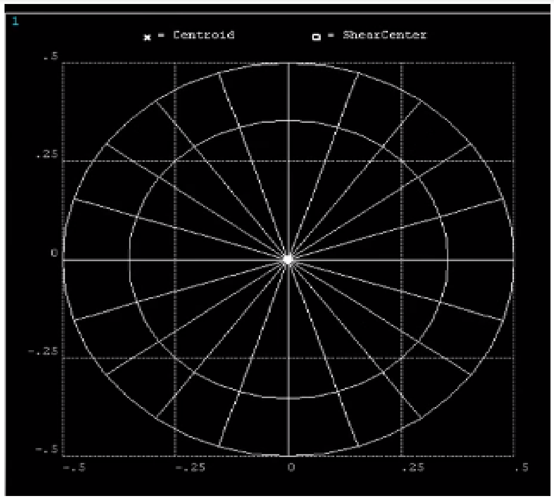
Poisson's Ratio (PRXY) = 0.3

Sections ►Beam ►Common Sections ►Subtype ► Select Solid Circle

R=0.5

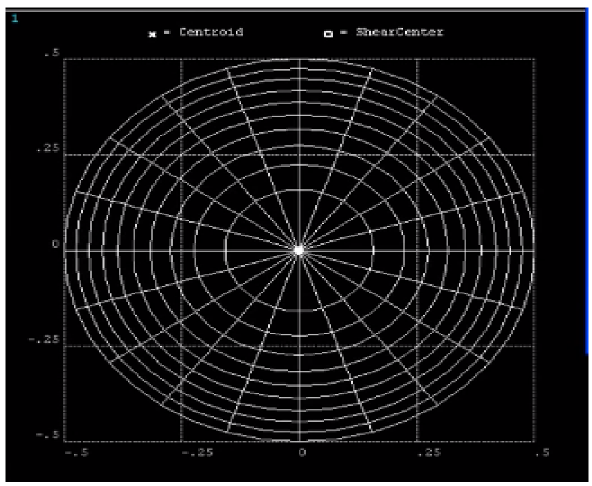
N=20

T=0, Meshview



T=10, Mesh view,OK

Right click,Replot.



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

Create the keypoints 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

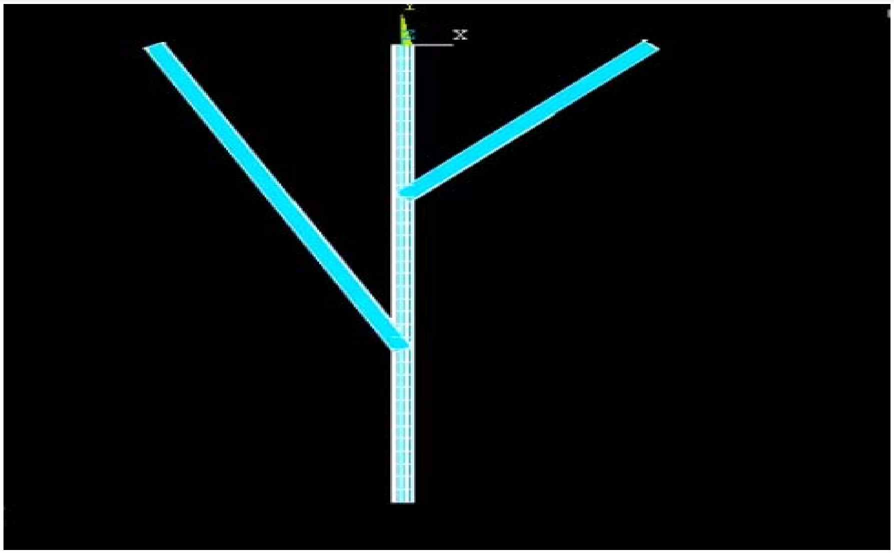
Lines►set►1&4 line click►4&5 line click►5&6 line click►ok

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,



Disable 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

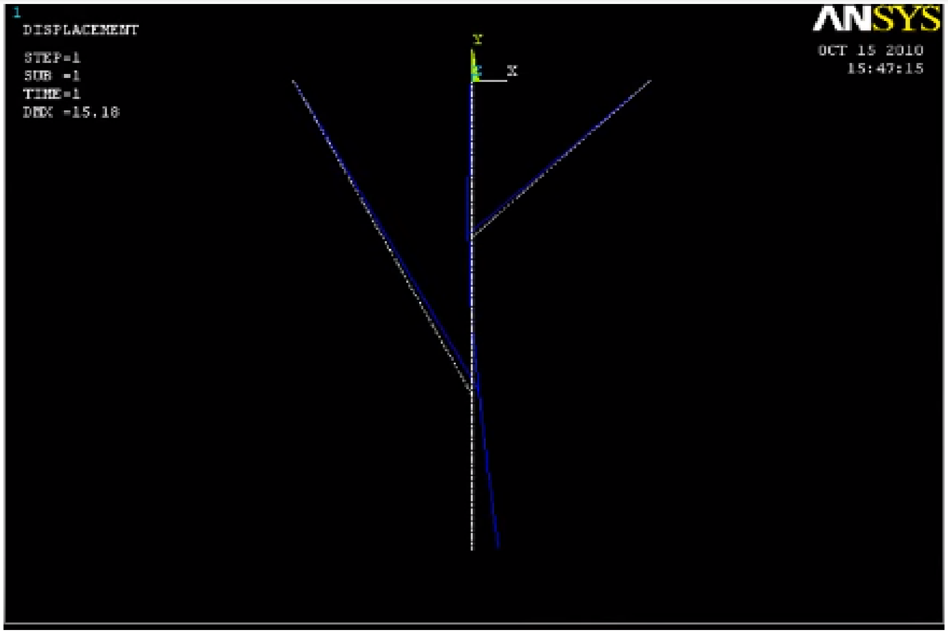
THXY >60>ok

Loads>Apply>Structural>Force/Moment>click On nodes 28/Key point 6>

Force/Moment value >10000

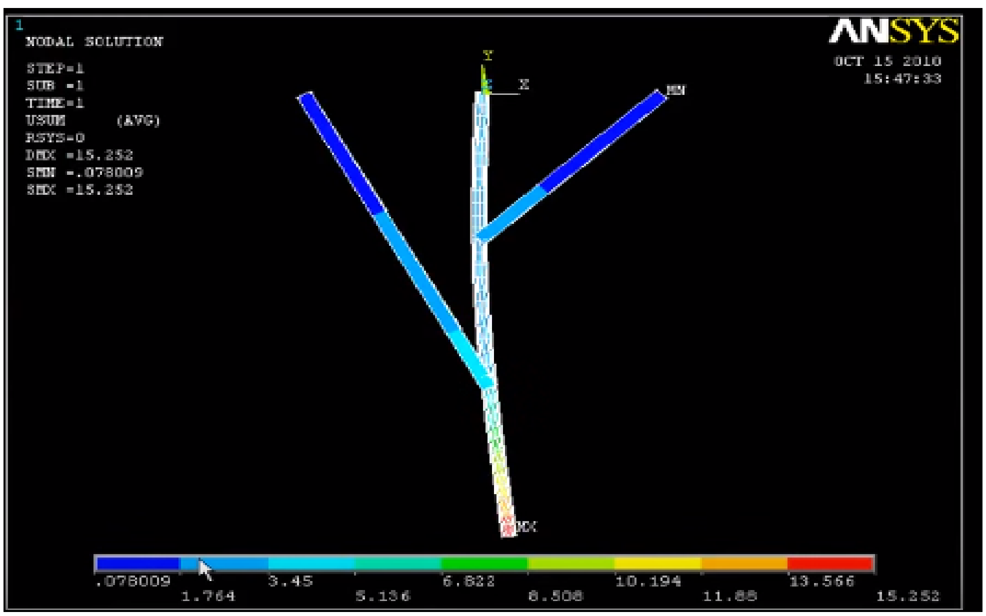
Solution ► Solve>General Post proc>List results>Rection solution>

Plot results:Defromed shape

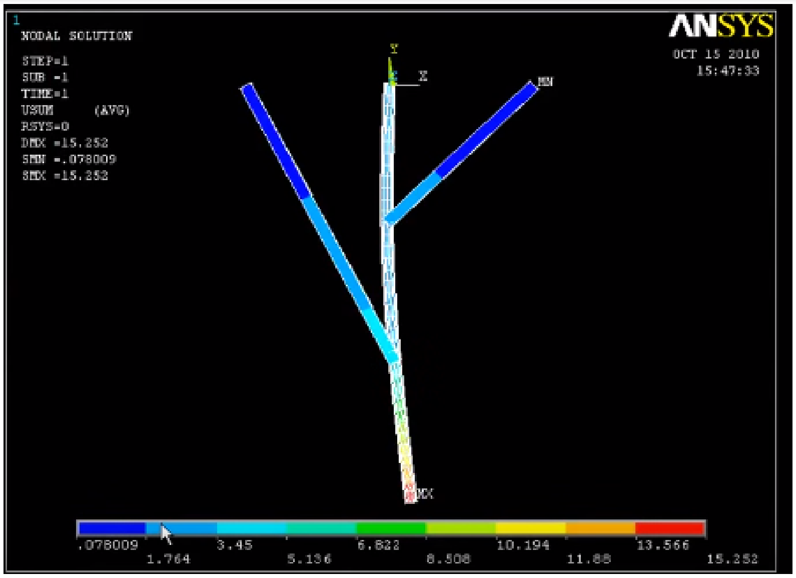


Plot results: >nodal solution>DOF solution>Displacement vector sum>ok

Plot ctrls> Click in the box against Display Element Type>ok

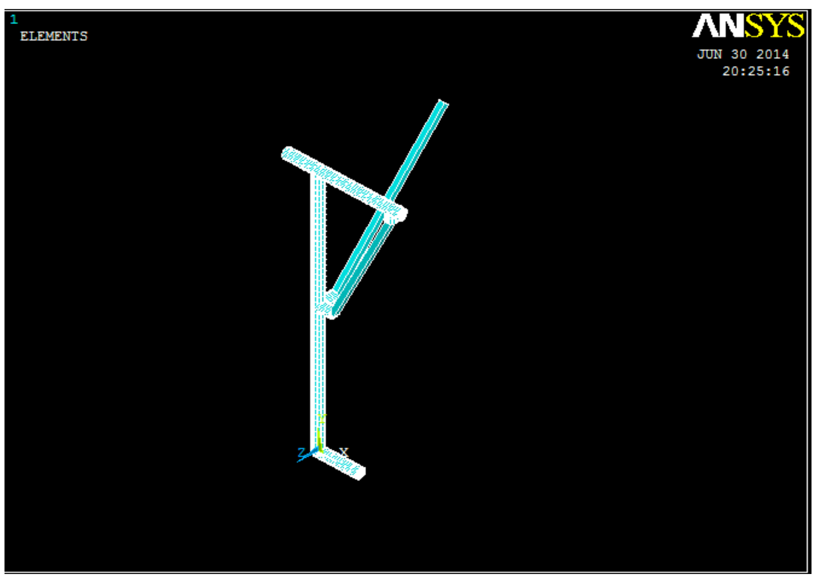


Result:



EXERCISE PROBLEM

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

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?

EXPERIMENT: 7

STATIC ANALYSIS OF TAPERED WING BOX

- **Experiment as given in the JNTUH curriculum.**
- **Tapered wing box beam.**

Aim: Analyze the given wing structure using Ansys and find out the variation in the Structure of the Wing.

Apparatus: ANSYS Software 13.0

Given Data:

Young's Modulus: 7e10

Poisson's Ratio: 0.3

Length of the Wing: 30

Steps of Modeling:

Preferences ► Structural ► H- method ► OK

Preprocessor ► Element Type ► Add ► Add ►

Solid ► Brick 8 Node 45 ► Apply

Beam ► 2 node 188 ► Apply ►

Shell ► elastic 4 node 63 ► Click OK

Real constants ► Add ► shell 63 ► I = 1.2, j = 1.7, k = 2.2

Material Properties ► Material Models ► Structural ► Linear ► Elastic ► Isotropic ►

EXX: 7e10

PRXY: 0.3

Density: 2700

Modeling ► Create ► Key points ► In Active CS

	X	Y	Z
1.	0	0	0
2.	8	4	0
3.	8	-4	0
4.	-6	3	0
5.	-6	-3	0

6.	6	3	30
7.	6	-3	30
8.	-4	-2	30
9.	-4	2	30

Modeling ▶ Create ▶ Lines ▶ straight lines ▶ 2, 3 - 2, 4 - 4, 5 - 5, 3 ▶ Apply

Lines ▶6, 8 - 6, 7 - 7, 9 - 8, 9 ▶ Apply

Lines ▶ 8, 4 & 9, 5

Lines ▶ 2, 6 & 7, 3 ▶ OK

Modeling ▶ Create ▶ Areas ▶ Arbitrary ▶ by lines ▶ Select Upper Lines of Both sides ▶ Left Line, Right Lines ▶ Click Apply ▶ Select Lower Lines of both the sides ▶ Left Line and Right Line Click Apply ▶ Click OK

Modeling ▶ Create ▶ Volumes ▶ Arbitrary ▶ by Areas ▶ Box Selection ▶ Select all the Areas ▶ Click OK ▶ Hence a Solid Volume is created

Meshing ▶ Mesh Attributes ▶ all lines ▶ Select beam 188 ▶ OK

Meshing ▶ Mesh Attributes ▶ All Areas ▶ Select shell 63 ▶ OK

Meshing ▶ Mesh Attributes ▶ All Volumes ▶ Select solid 45 ▶ OK

Meshing ▶ Size control ▶ manual size ▶ pick all lines ▶ Enter the Element Edge Length as 1 ▶ OK

Meshing ▶size control ▶ areas ▶ Box Selection ▶ Enter the Element Edge Length as 1 ▶ OK

Meshing ▶ Mesh ▶ Volumes ▶ free ▶ Select the box ▶ select full body ▶ OK

Loads ▶ Define Loads ▶ Apply ▶ Structural ▶ Displacement ▶ On Areas ▶ Select the Large Airfoil Area ▶ Click Apply ▶ Select All DOF ▶ OK

Loads ▶ Define Loads ▶ Apply ▶ Structural ▶ Pressure ▶ On Areas ▶ Select the upper and lower surface ▶ Click Apply ▶ Enter the Load Value =10000N & -10000N

Loads ▶ Analysis Type ▶ New Analysis ▶ Select Static ▶ OK

Solution ▶ Solve ▶ Current LS ▶ Warnings can be ignored ▶ Solution is done

RESULTS:

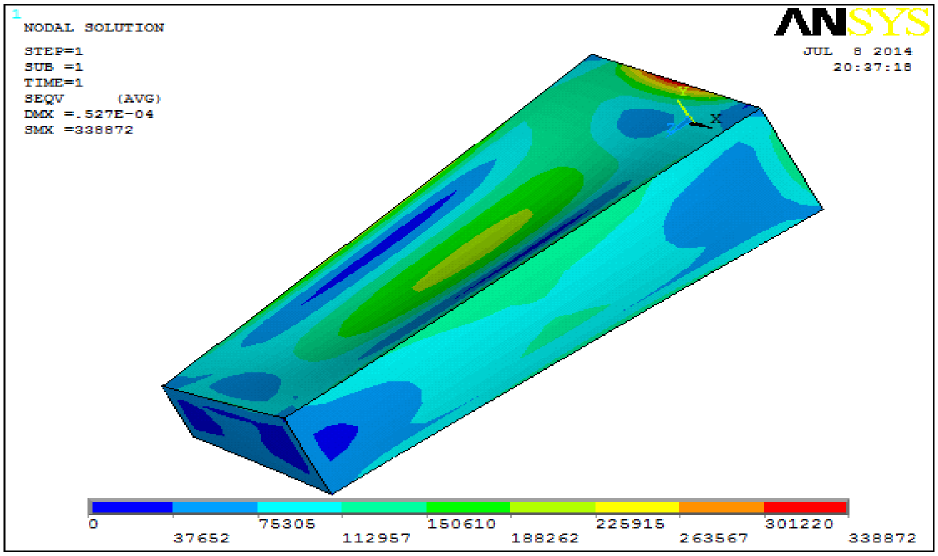
General Post Processor ▶ Plot Results ▶ Deformed Shape ▶ Deformed + Undeformed ▶ OK

General Post Processor ▶ Plot Results ▶ Contour Plot ▶ Nodal Solution ▶ DOF Solution

Case: 1:- To determine the stresses acting on a tapered wing with a pressure load of 10000 & -10000 N acting on the lines upper and lower surfaces.

Y Component of Displacement = 0.527e-04

Von Mises Stress = 338872

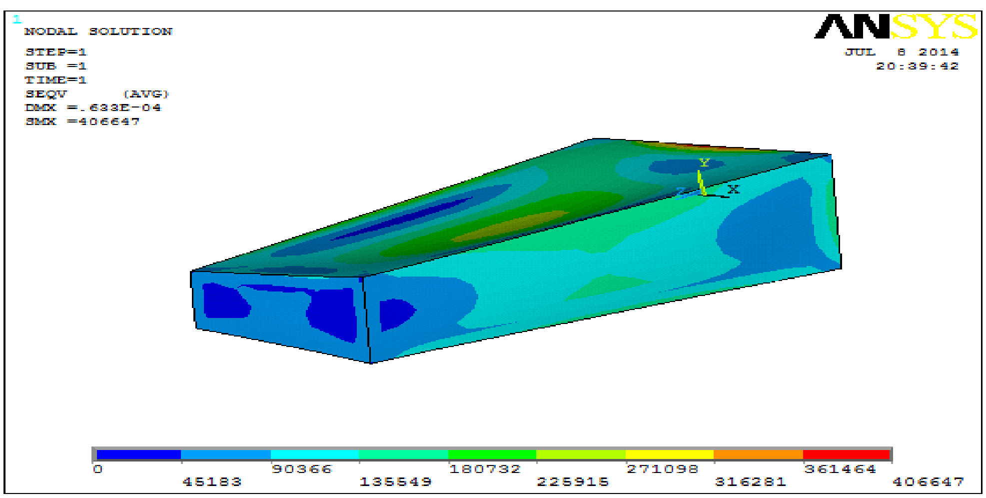


PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To determine the stresses acting on a tapered wing with a pressure load of 12000 & -12000 N acting on the lines upper and lower surfaces.

Y Component of Displacement = 0.633e-04

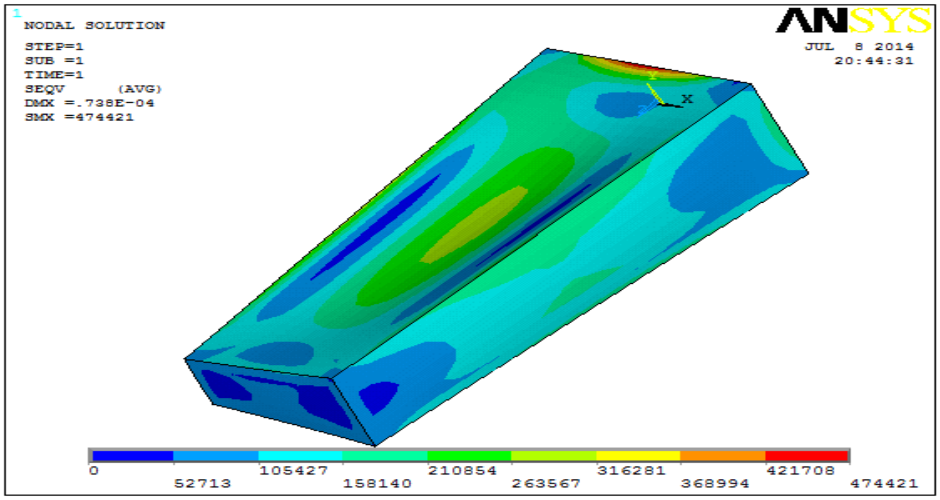
Von Mises Stress = 406647



Case: 3:- To determine the stresses acting on a tapered wing with a pressure load of 14000 & -14000 N acting on the lines upper and lower surfaces.

Y Component of Displacement = 0.738e-04

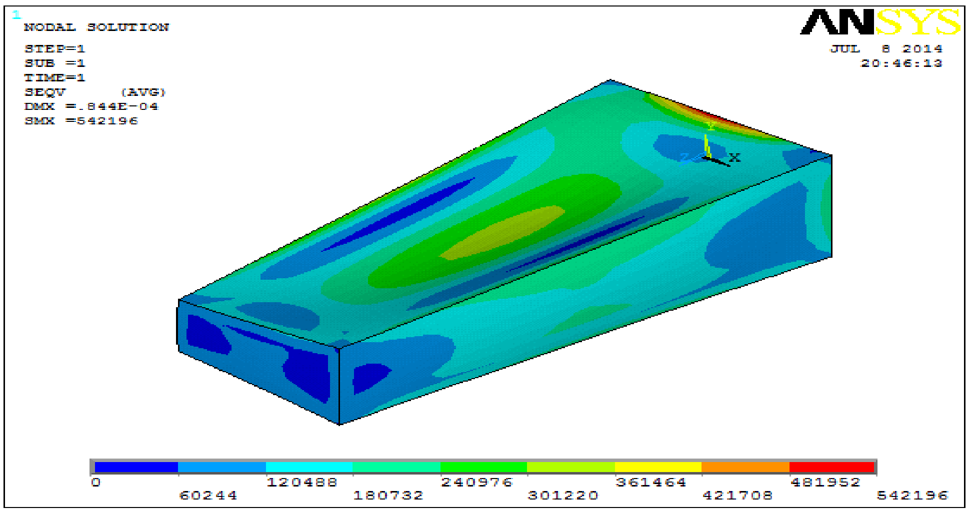
Von Mises Stress = 474421



Case:4:- To determine the stresses acting on a tapered wing with a pressure load of 16000 & -16000 N acting on the lines upper and lower surfaces.

Y Component of Displacement = 0.844e-04

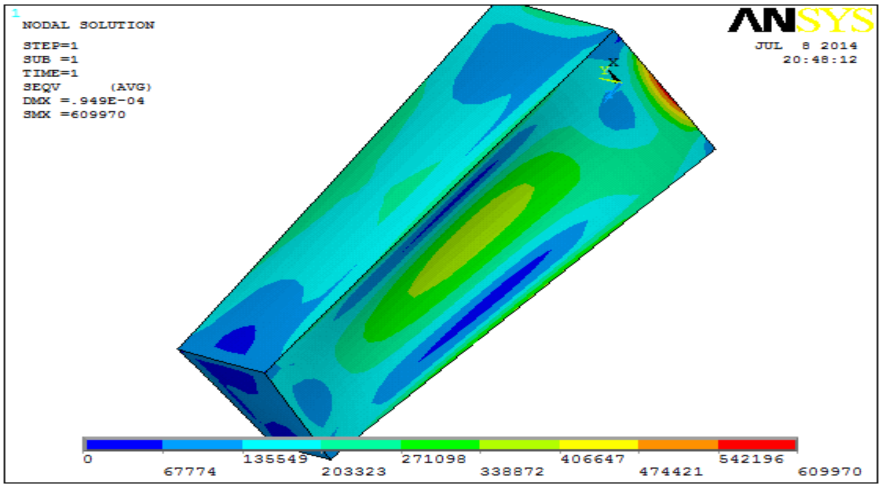
Von Mises Stress = 542196



Case: 5:- To determine the stresses acting on a tapered wing with a pressure load of 10000 & -10000 N acting on the lines upper and lower surfaces.

Y Component of Displacement = 0.949e-04

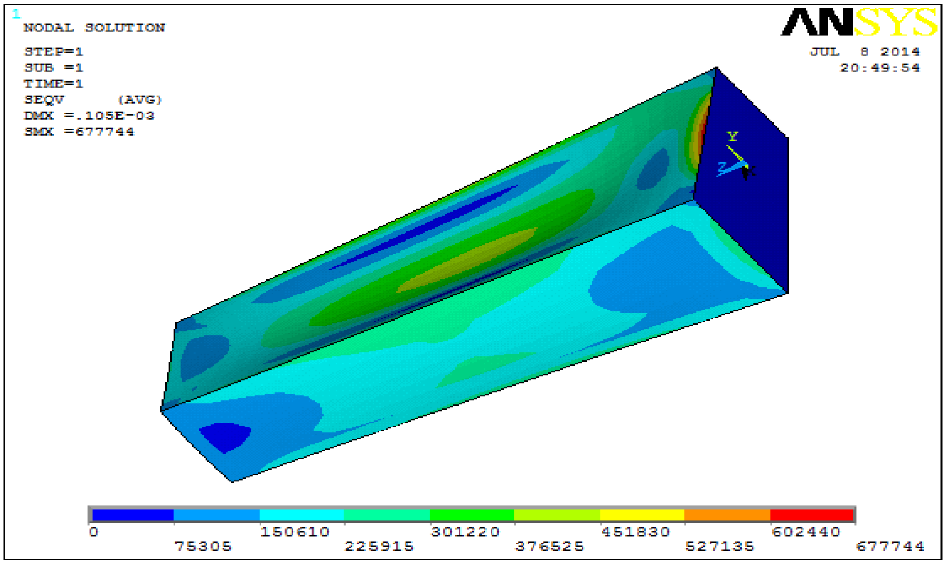
Von Mises Stress = 609970



Case: 6:- To determine the stresses acting on a tapered wing with a pressure load of 10000 & -10000 N acting on the lines upper and lower surfaces.

Y Component of Displacement = 0.105e-03

Von Mises Stress = 677744



EXERCISE PROBLEM

1. Modal Analysis of a airplane wing

VIVA QUESTIONS

1. The _____ analysis is used to calculate the vibration characteristics of a structure.
2. The SI unit of frequency is _________.
3. Ansys report is saved with the _______ file extension.
4. The images captured using the Ansys report generator are saved with a_______ file extension.
5. The maximum stress value should be less than the applied stress bound value. (T/F)

EXPERIMENT: 8

ANALYSIS OF A FUSELAGE

- **Experiment as given in the JNTUH curriculum.**
- **Fuselage bulkhead.**

AIM: - To Calculate the deformation of the aluminum fuselage section under the application of internal load of 100000 Pa.

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

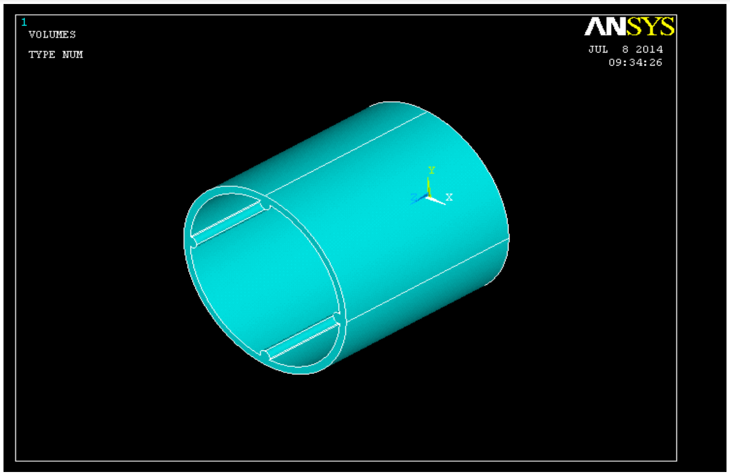
WP x = 0; X =0; Y = 2.25; Radius = 0.15 Apply

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

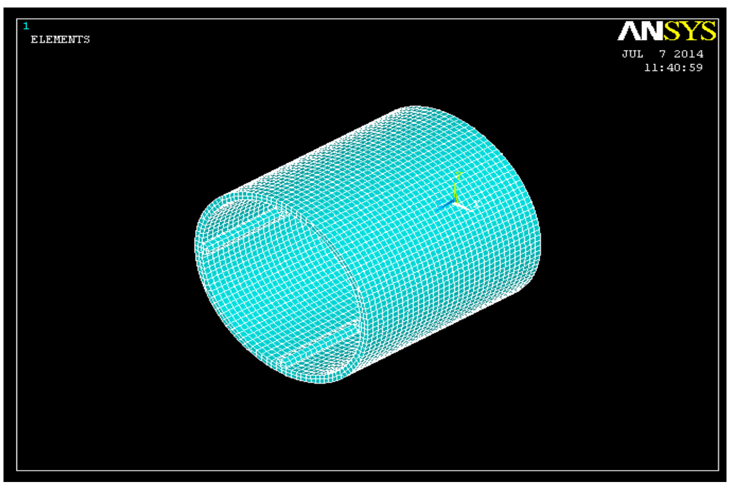


STEP 4: Meshing the Geometry

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 0.15 → 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

<u>RESULT:</u>

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

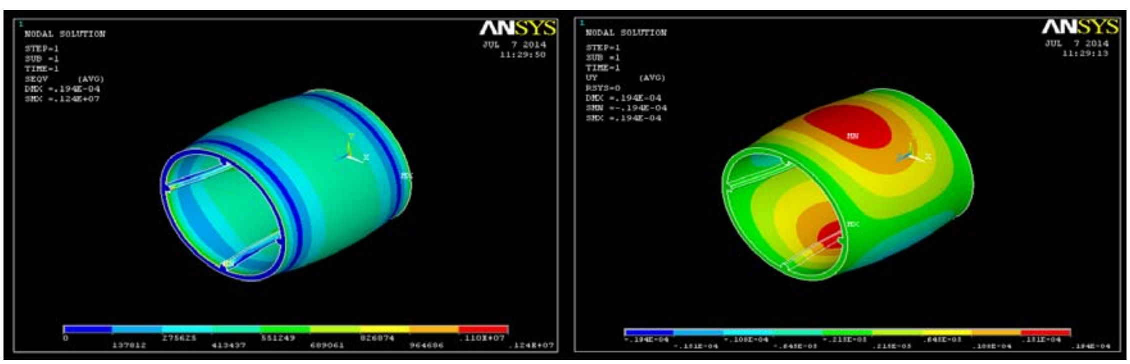
SMN = -.194E-04

SMX = .194E-04

VON MISSES STRESS

DMX = .194E-04

SMX = .124E+07



PROBLEM DEFINITIONS DIFFERENT FROM JNTU TOPICS

Case: 2:- To Calculate the deformation of the aluminum fuselage section under the application of internal load at 1.1e5.

Y COMPONENT OF DISPLACEMENT

DMX = .819E-05

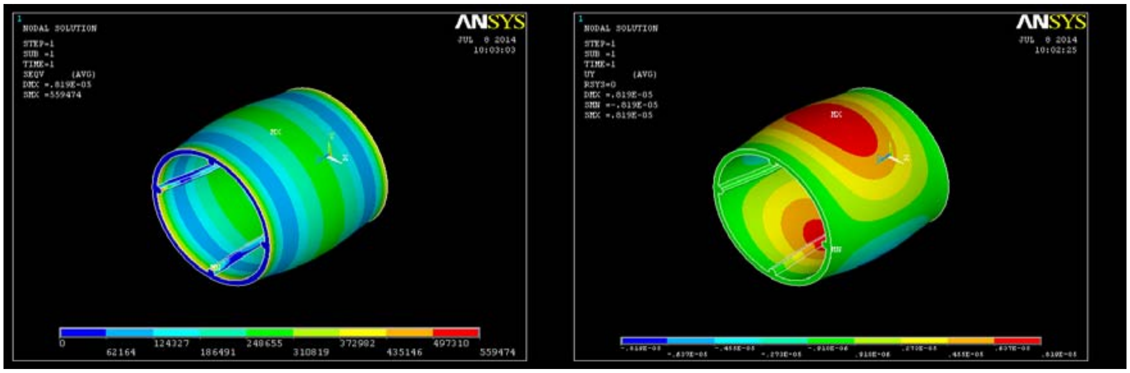
SMN = -.819E-05

SMX = .819E-05

VON MISSES STRESS

DMX = .819E-05

SMX = 559474



Case: 3:- To Calculate the deformation of the aluminum fuselage section under the application of internal load at 1.2e5.

Y COMPONENT OF DISPLACEMENT

DMX = .893E-05

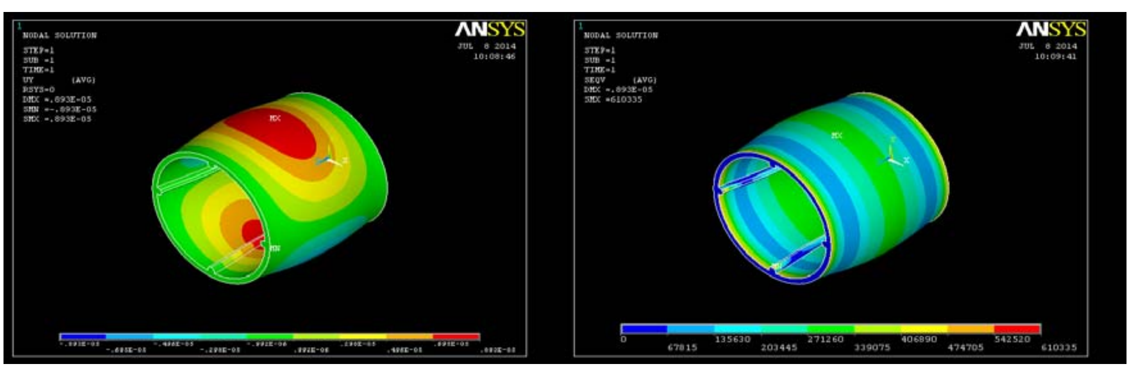
SMN = -.893E-05

SMX = .893E-05

VON MISSES STRESS

DMX = .893E-05

SMX = 610335



Case: 4:- To Calculate the deformation of the aluminum fuselage section under the application of internal load 0.9e5.

Y COMPONENT OF DISPLACEMENT

DMX = .670E-05

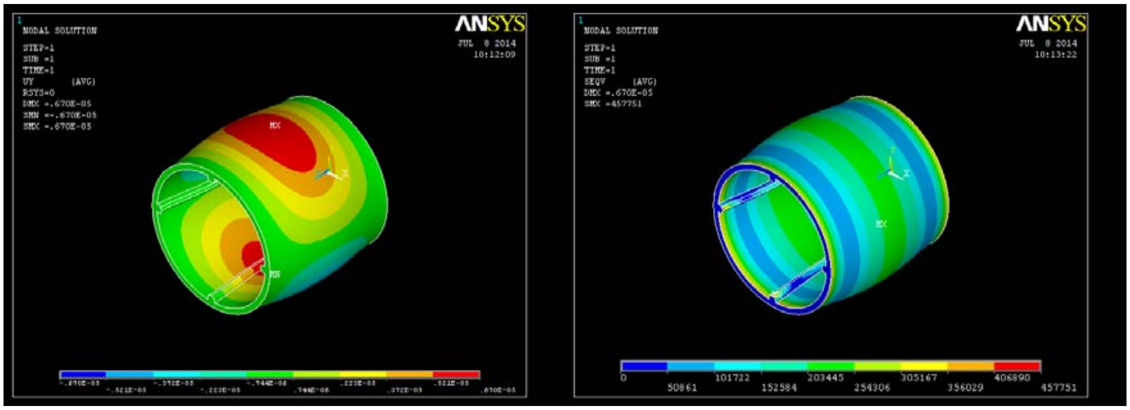
SMN = -.670E-05

SMX = .670E-05

VON MISSES STRESS

DMX = .670E-05

SMX = 457751



Case: 5:- To Calculate the deformation of the aluminum fuselage section under the application of internal load at 0.8e5.

Y COMPONENT OF DISPLACEMENT

DMX = .595E-05

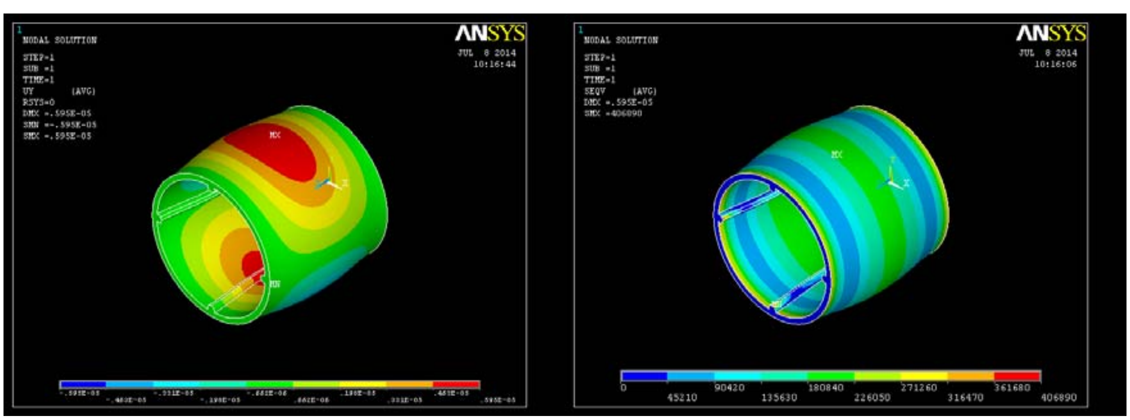
SMN = -.595E-05

SMX = .595E-05

VON MISSES STRESS

DMX = .595E-05

SMX = 406890



Case: 6:- To Calculate the deformation of the aluminum fuselage section under the application of internal load at 0.7e5.

Y COMPONENT OF DISPLACEMENT

DMX = .521E-05

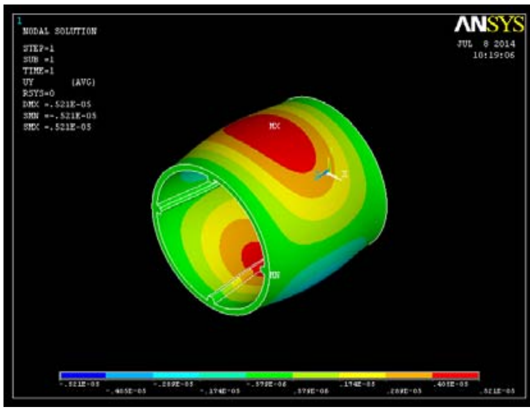
SMN = -.521E-05

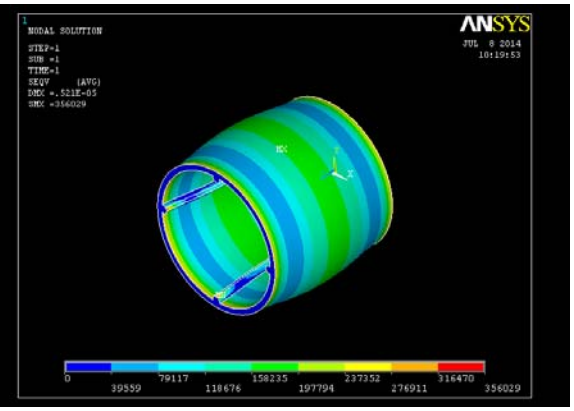
SMX = .521E-05

VON MISSES STRESS

DMX = .521E-05

SMX = 356029





EXERCISE PROBLEM

1. Static analysis of pressure vessel.

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.
5. Define Spectrum Analysis.