

**MANUAL**  
**For**  
**CAE Laboratory**  
**B.E Mechanical & Production**



**Mechanical Engineering Department**  
**Muffakham Jah College of Engineering and Technology**  
**Banjara Hills, Road No.3, Hyderabad-500034**

**PREPARED BY**

Mr. S. Irfan Sadaq  
Assistant professor, MED, MJCET

Dr. Ishrat M. M  
Professor, MED, MJCET

ME 433

### CAE LAB

Instruction	3 Periods per week
Duration of University Examination	3 Hours
University Examination	50 Marks
Sessional	25 Marks

1. Analysis of Plane Truss & Spatial Truss with various cross sections and materials to determine member forces, member strains & stresses, joint deflections under static, thermal and combined loading.
2. 2D & 3D beam analysis with different sections, different materials for different loads (forces and moments) with different end supports.
3. Static analysis of plate with a hole to determine the deformations, the Stresses to study the failure behavior and SCF.
4. Plane stress, plane strain and axisymmetric loading on the in plane members with in plane loading to study the stresses and strains.
5. Static analysis of connecting rod with tetrahedron and brick elements.
6. Static Analysis of flat and curved shell due to internal pressure and moments to estimate the strains, stresses and reactions forces and moments with different boundary conditions.
7. Buckling analysis of plates, shells and beams to estimate BF and modes.
8. Modal analysis of beams, plates and shells for natural frequencies and mode shapes.
9. Harmonic analysis of a Shaft subjected to periodic force and transient analysis of plate subjected to stepped and ramped loading with varying time.
10. Steady state heat transfer Analysis Cross section of chimney and transient heat transfer analysis of solidification of castings.
11. Non linear analysis of cantilever beam with non-linear materials at tip moment and post buckling analysis of shells for critical loads.
12. Coupled field analysis.
13. Flow analysis of pipe with different fluids/gasses/air for velocity and pressure gradients.
14. CFD analysis of aerofoil design.
15. CFD analysis of ducts/impeller/fan.
16. Use of MATLAB for finding B matrix, stiffness matrix and loading matrices of beam/in plane/solid elements and interfacing with CAE software's

Note: 1. Any of FEA software ANSYS/ABAQUS/NASTRAN/NISA/CAEFEM/ADINA may be used

2. Any 12 experiments to be conducted

## **LIST OF EXPERIMENTS**

1. Analysis of Plane Truss & Spatial Truss with various cross sections and materials to determine member forces, member strains & stresses, joint deflections under static, thermal and combined loading.
2. 2D & 3D beam analysis with different sections, different materials for different loads (forces and moments) with different end supports.
3. Static analysis of plate with a hole to determine the deformations, the Stresses to study the failure behavior and SCF.
4. Plane stress, plane strain and axisymmetric loading on the in plane members with in plane loading to study the stresses and strains.
5. Static analysis of connecting rod with tetrahedron and brick elements.
6. Static Analysis of flat and curved shell due to internal pressure and moments to estimate the strains, stresses and reactions forces and moments with different boundary conditions.
7. Buckling analysis of plates, shells and beams to estimate BF and modes.
8. Modal analysis of beams, plates and shells for natural frequencies and mode shapes.
9. Harmonic analysis of a Shaft subjected to periodic force and transient analysis of plate subjected to stepped and ramped loading with varying time.
10. Steady state heat transfer Analysis Cross section of chimney and transient heat transfer analysis of solidification of castings.
11. Non linear analysis of cantilever beam with non-linear materials at tip moment and post buckling analysis of shells for critical loads.
12. CFD analysis of aerofoil design.

## CONTENTS

<b>Ex. No.</b>	<b>Title of the exercise</b>	<b>PG. NO.</b>
1.	Introduction to Computer Aided Engineering	
2.	Analysis of Plane Truss	
3.	2D & 3D beam analysis	
4.	Static analysis of plate with a hole	
5.	Static analysis of connecting rod	
6.	Buckling analysis of plates, shells and beams	
7.	Modal analysis	
8.	Harmonic analysis	
9	Steady state heat transfer Analysis	
10	CFD analysis of aerofoil design	

# **INTRODUCTION**

## **(Computer Aided Engineering)**

This module aims to give students an introduction to CAE to enable them to tackle a range of problems using appropriate software. An introduction to some numerical techniques is provided, the use of software is developed and critical appraisal of results is emphasized. For stress, thermal and many other types of analysis, only the simplest of configurations can be solved by exact analytic methods. For the vast majority of engineering components some alternative method is needed and the usual approach involves a method of approximation requiring some numerical techniques to solve, which in turn requires the use of a digital computer.

### **NUMERICAL TECHNIQUES**

#### **Stress, vibration and thermal analysis.**

There are 3 main methods of numerical approximation:

- Finite difference (**FD**) method
- Finite element analysis (**FEA**)
- Boundary element method (**BEM**)

These methods all have some advantages and disadvantages. FEA is a variant of some classical approximation methods, whereas the FD method is an older procedure that has been used in one form or another almost since the inception of differential calculus. FEA was first adapted for stress analysis problem solving by Southwell. Ever since FEA were first used, there have been arguments about their relative merits versus the longer established FD method.

#### **FEA**

FEA involves mathematical approximations associated with the displacement function of a particular element and geometric approximations to the component shape. An advantage of FEA is that it can be used to analyze virtually any structural problem in a routine manner - hence non homogeneous material can be accommodated as the assembly of elements with different properties is straightforward.

#### **FD Method**

The FD method uses a finite difference approximation to the (partial) differential equations that describe the behavior of the system. A significant difficulty in solving the equations is

that with fine meshes, needed for adequate accuracy, there may be instability in determining the solution. Discontinuous interfaces, such as abrupt changes in material properties or geometry require special treatment. FD tends to be used more for Fluids problems than for Mechanics problems.

## **BEM**

The BEM method reduces the dimensionality of the problem by 1, so the overall size of the computation the smaller than in FEA or FD, but the matrices produced are assymetric and densely packed and may require just as much computational effort to solve as when using FEA or FD. The method is particularly suited to problems with high stress gradients such as fracture mechanics. Boundaries that are nominally at 'infinity' may readily by handled whereas such boundaries increase the size of FEA and FD problems. The underlying mathematics is however more complex than the other two methods.

## **ANSYS**

### **FINITE ELEMENT ANALYSIS**

Finite element analysis (**FEA**) is a computer simulation technique used in engineering analysis it uses a numerical technique called the finite element method (FEA). There are many finite element software packages, both free and proprietary development of the finite element method in structural mechanics is usually based on an energy principle such as the virtual work principle or the minimum total potential energy principle.

In general, there are three phases in any computer-aided engineering task:

- Pre-processing- Defining the finite element model and environmental factors to be applied to it.
- Analysis solver- Solution of finite element model.
- Post-processing of results using visualization tools.

### **PRE-PROCESSING**

The first step in using FEA, pre-processing, is constructing a finite element model of the structure to be analyzed. The input of a topological description of he structure's geometric features is required in most FEA packages. This can be in either 1D, 2D or 3D form, modeled by line, shape, or surface representation, respectively, although nowadays 3D models are predominantly used. The primary objective of the model is to realistically replicate the important parameters and features of the real model. The simplest mechanism to achieve modeling similarity in structural analysis is to utilize pre-existing digital blueprints, design files, CAD models, and/or data by importing that into an FEA

environment. Once the finite element geometric model has been created, a meshing procedure is used to define and break-up the model into small elements. In general, a finite element model is defined by a mesh network, which is made up of the geometric arrangement of elements and nodes.

Nodes represent points at which features such as displacements are calculated. FEA packages use node numbers to serve as an identification tool in viewing solutions in structures such as deflections. Elements are bounded by sets of nodes, and define localized mass and stiffness properties of the model. Elements are also defined by mesh numbers, which allow references to be made to corresponding deflections or stresses at specific model locations.

### **ANALYSIS (COMPUTATION OF SOLUTION)**

The next stage of FEA process is analysis. The FEM conducts a series of computational procedures involving applied forces, and the properties of the elements which produce a model solution. Such a structural analysis allows the determination of effects such as deformations, strains, and stresses which are caused by applied structural loads such as force, pressure and gravity.

### **POST-PROCESSING (VISUALIZATION)**

These results can then be studied using visualization tools within the FEA environment to view and to fully identify implications of the analysis. Numerical and graphical tools allow the precise location of data such as stresses and deflections to be identified.

## INTRODUCTION TO ANSYS

Ansys is a general purpose finite element modeling package for numerically solving a wide variety of mechanical, electrical problems.

These problems include:

Static/Dynamic structural analysis (Both linear and nonlinear)

1. Fluid analysis
  - laminar flow
  - turbulent flow
2. Acoustic analysis
3. Electromagnetic analysis
4. Model analysis
5. Thermal analysis
  - conduction
  - convection
  - radiation
6. Transient thermal analysis
7. Buckling analysis
8. Spectrum analysis
9. Harmonic analysis

**Static analysis:** In this type of problem we determine the elastic data deflections and stresses at critical points due to a system to external forces acting on structure nodal.

**Modal analysis:** In this type of problem we determine the vibration characteristics.

**Harmonic analysis:** We will determine the response of structure to harmonically time varying loads.

**Buckling analysis:** We determine the buckling loads and also buckling shape.

**Thermal analysis:** In this we determine low thermal stresses are there in given structure.

**Fluid analysis:** In this we can see how a comprehensive got that fluid flows through a given number under given condition.

### **Structural Static Analysis**

A static analysis calculates the effects of *steady* loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Static analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.



The types of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)
- Fluences (for nuclear swelling)

### **Modal Analysis**

Use modal analysis to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis.

You use modal analysis to determine the natural frequencies and mode shapes of a structure. The

Natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required if you want to do a spectrum analysis or a mode superposition harmonic or transient analysis. You can do modal analysis on a prestressed structure, such as a spinning turbine blade. Another useful feature is modal cyclic symmetry, which allows you to review the mode shapes of a cyclically symmetric structure by modeling just a sector of it. Modal analysis in the ANSYS family of products is a linear analysis. Any nonlinearity, such as plasticity and contact (gap) elements, are ignored even if they are defined. You can choose from several mode extractions methods: Block Lanczos, Supernode, PCG Lanczos, reduced, asymmetric, damped, and QR damped.

The damped and QR damped methods allow you to include damping in the structure. The QR Damped method also allows for unsymmetrical damping and stiffness matrices.

### **Harmonic Response Analysis**

Any sustained cyclic *load* will produce a sustained cyclic *response* (a *harmonic* response) in a Structural system. Harmonic response analysis gives you the ability to predict the sustained dynamic behavior of your structures, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

Harmonic response analysis is a technique used to determine the steady-state response of a linear

Structure to loads that vary sinusoidally (harmonically) with time. The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency.

"Peak" responses are then identified on the graph and stresses reviewed at those peak frequencies. This analysis technique calculates only the steady-state, forced vibrations of a

structure. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic response analysis.

## **Buckling Analysis**

Buckling analysis is a technique used to determine *buckling loads* (critical loads at which a structure becomes unstable) and buckled mode shapes (the characteristic shape associated with a structure's buckled response).

Two techniques are available in the ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural, and ANSYS Professional programs for predicting the buckling load and buckling mode shape of a structure: **Nonlinear** buckling analysis, and **Eigenvalue** (or linear) buckling analysis. Because the two methods can yield dramatically different results, it is necessary to first understand the differences between them.

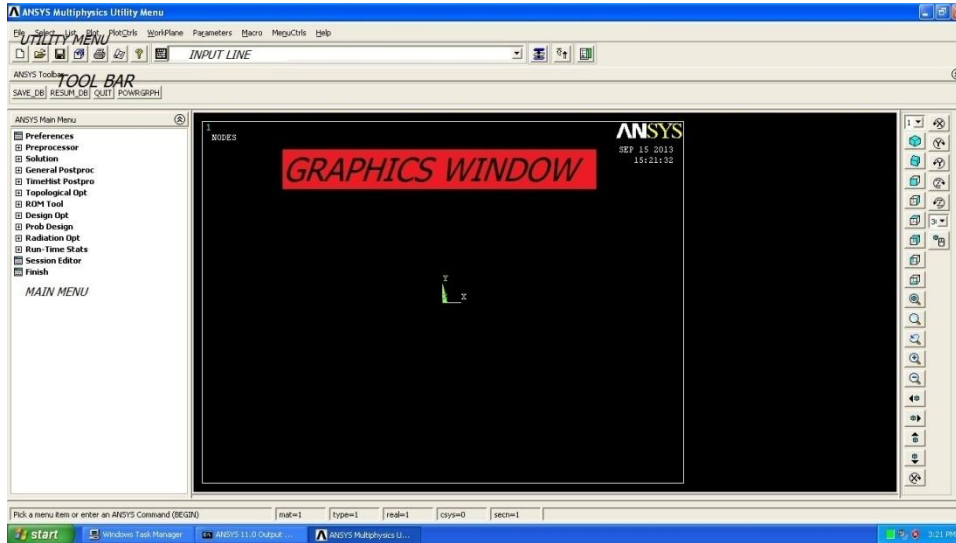
- ***Nonlinear buckling analysis*** is usually the more accurate approach and is therefore recommended for design or evaluation of actual structures. This technique employs a nonlinear static analysis with gradually increasing loads to seek the load level at which your structure becomes unstable.

Using the nonlinear technique, your model can include features such as initial imperfections, plastic behavior, gaps, and large-deflection response. In addition, using deflection-controlled loading, you can even track the post-buckled performance of your structure (which can be useful in cases where the structure buckles into a stable configuration, such as "snap-through" buckling of a shallow dome).

- ***Eigenvalue buckling analysis*** predicts the theoretical buckling strength (the bifurcation point) of an ideal linear elastic structure. This method corresponds to the textbook approach to elastic buckling analysis: for instance, an eigenvalue buckling analysis of a column will match the classical Euler solution.

However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. Thus, eigenvalue buckling analysis often yields unconservative results, and should generally not be used in actual day-to-day engineering analyses.

# 1. Main Window



Within the Main Window there are following 5 divisions:

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

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

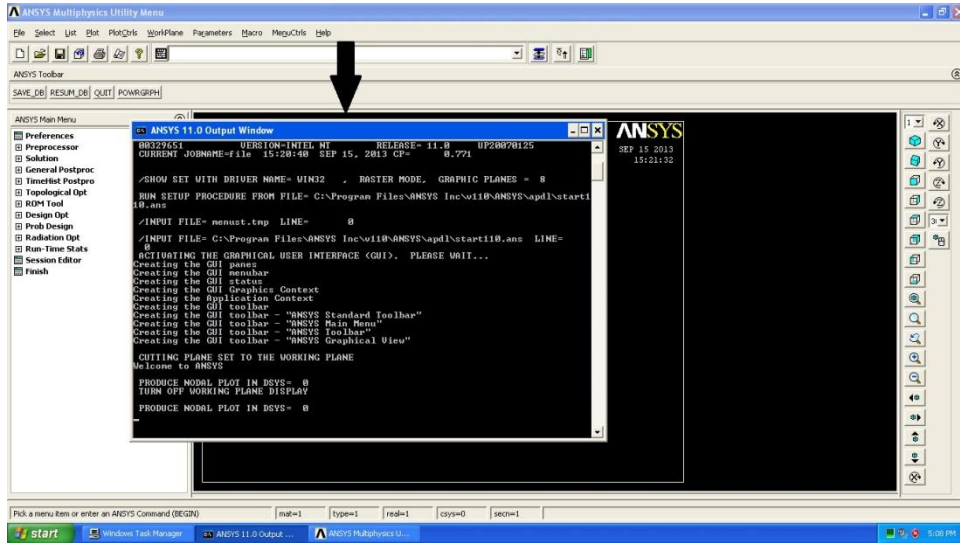
**Toolbar:** The Toolbar contains push buttons that execute commonly used ANSYS commands. More push buttons can be added if desired.

**Main Menu:** The Main Menu contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor, design optimizer. It is from this menu that the vast majority of modeling commands are issued. This is where you will note the greatest change between previous versions of ANSYS and version 11.0. However, while the versions appear different, the menu structure has not changed.

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

**Output Window:** Output Window displays text output from the program, such as listing of data, etc. It is usually positioned behind the Graphics Window and can be put to the front if necessary.

## D

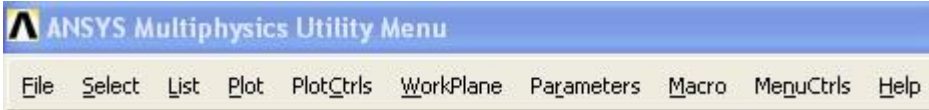


### Steps involved in problem:

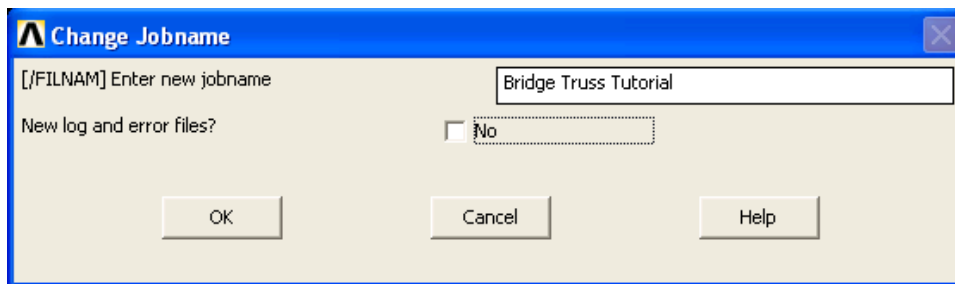
**Preprocessing:** Defining the Problem

**1. Give a Title** (such as 'Bridge Truss Tutorial').

In the Utility menu bar select **File > Change Title:**



The following window will appear:



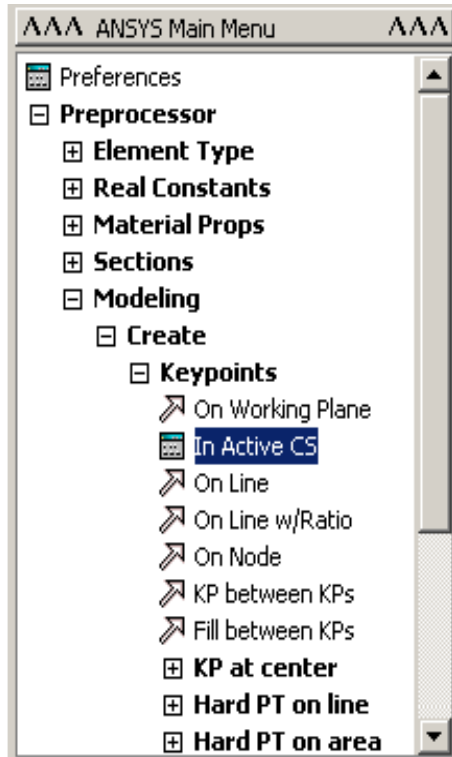
Enter the title and click 'OK'. This title will appear in the bottom left corner of the 'Graphics' Window once you begin. Note: to get the title to appear immediately, select **Utility Menu > Plot > Replot**.

**2. Enter Keypoints**

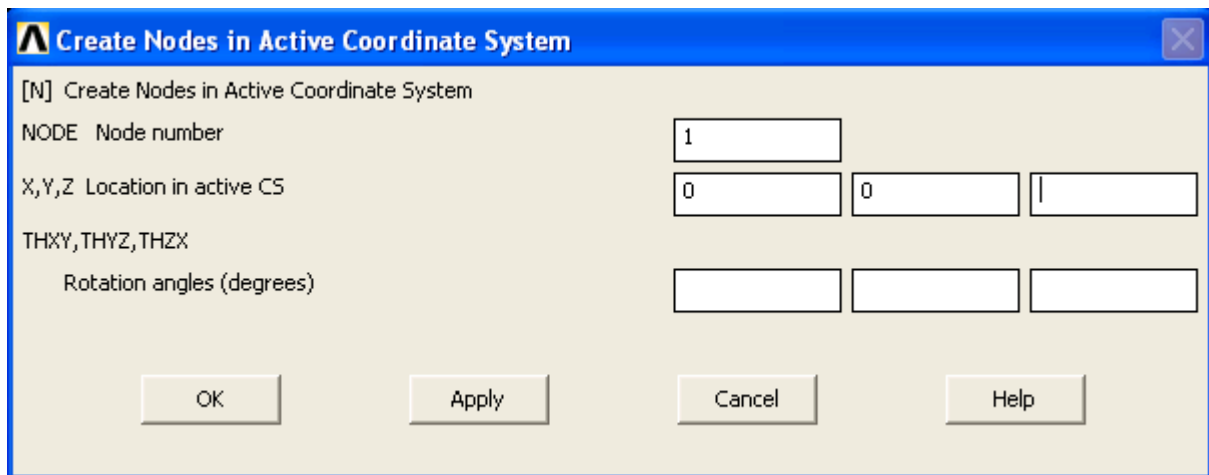
The overall geometry is defined in ANSYS using keypoints which specify various principal coordinates to define the body.

From the 'ANSYS Main Menu' select:

**Preprocessor > Modeling > Create > Keypoints > In Active CS**



The following window will then appear:



To define the first keypoint which has the coordinates  $x = 0$  and  $y = 0$ :

Enter keypoint number 1 in the appropriate box, and enter the x, y coordinates: 0, 0 in their appropriate boxes (as shown above).

Click 'Apply' to accept what you have typed.

Enter the remaining keypoints using the same method.

**Note:** When entering the final data point, click on 'OK' to indicate that you are finished entering keypoints. If you first press 'Apply' and then 'OK' for the final keypoint, you will have defined it twice! If you did press 'Apply' for the final point, simply press 'Cancel' to close this dialog box.

### **Units**

Note the units of measure (i.e. mm) were not specified. It is the responsibility of the user to ensure that a consistent set of units are used for the problem; thus making any conversions where necessary.

### **Correcting Mistakes**

When defining keypoints, lines, areas, volumes, elements, constraints and loads you are bound to make mistakes. Fortunately these are easily corrected so that you don't need to begin from scratch every time an error is made! Every 'Create' menu for generating these various entities also has a corresponding 'Delete' menu for fixing things up.

### **3. Form Lines**

The keypoints must now be connected. We will use the mouse to select the keypoints to form the lines.

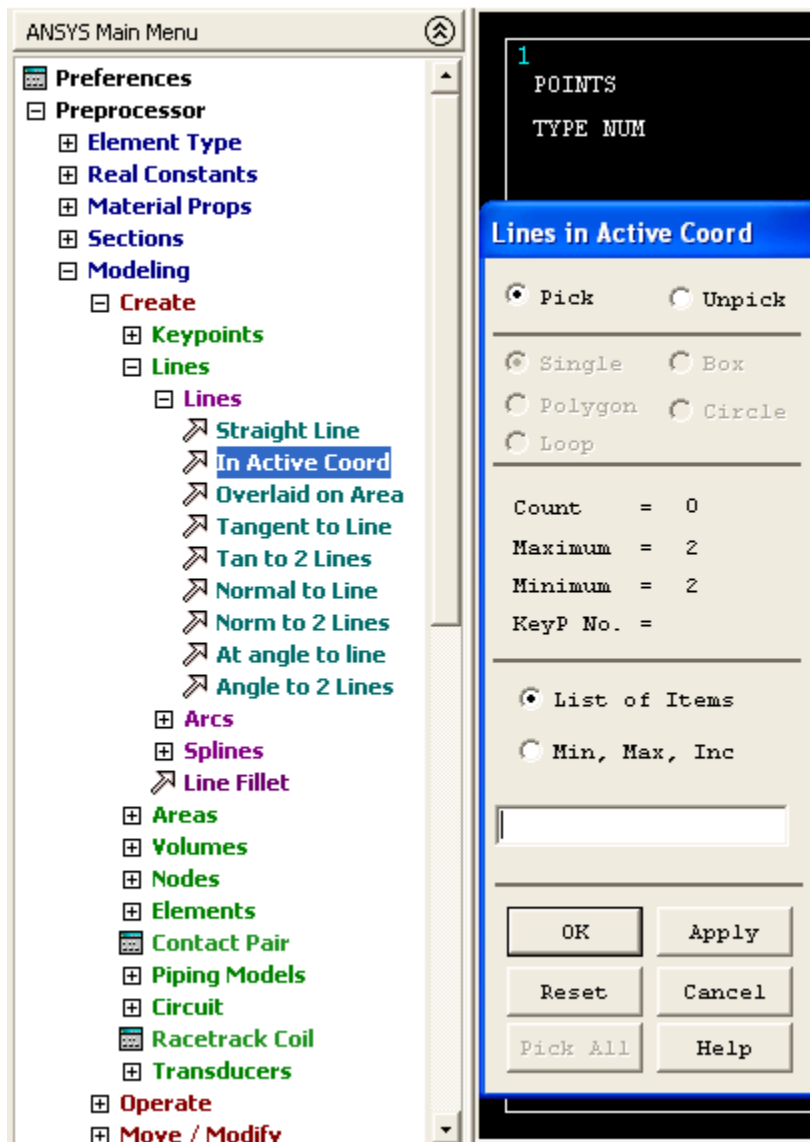
In the main menu select: **Preprocessor > Modeling > Create > Lines > Lines > In Active Coordinates**. The following window will then appear:

Use the mouse to pick keypoint #1 (i.e. click on it). It will now be marked by a small yellow box.

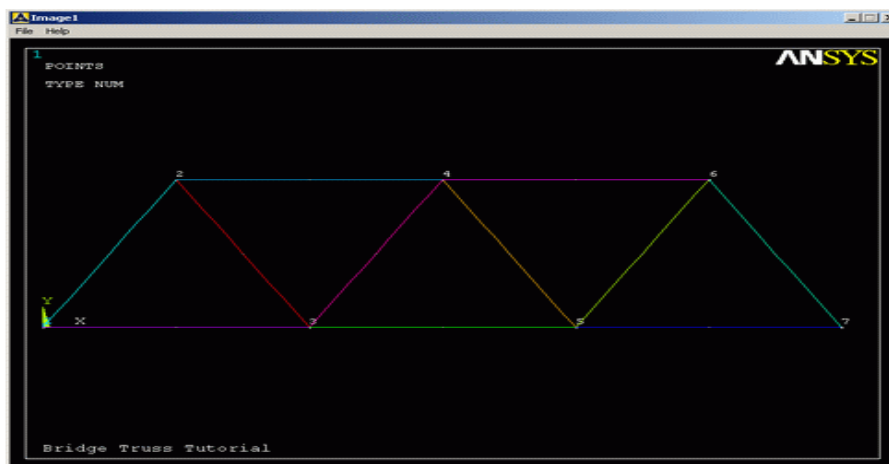
Now move the mouse toward keypoint #2. A line will now show on the screen joining these two points. Left click and a permanent line will appear.

Connect the remaining keypoints using the same method.

When you're done, click on 'OK' in the 'Lines in Active Coordinates' window, minimize the 'Lines' menu and the 'Create' menu. Your ANSYS Graphics window should look similar to the following:



FOR EXAMPLE:



## Disappearing Lines

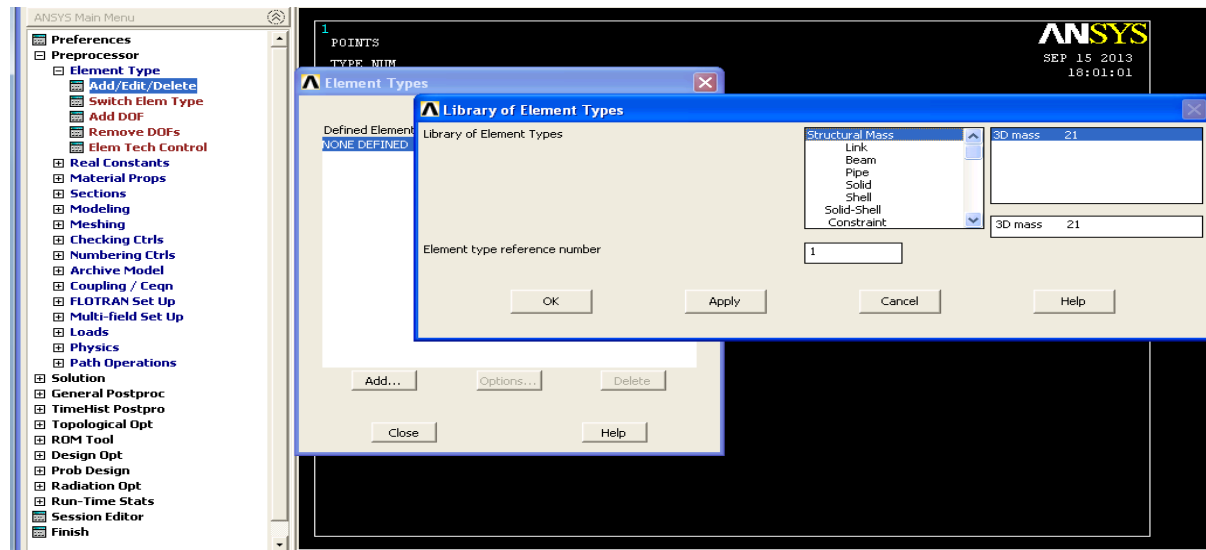
Please note that any lines you have created may 'disappear' throughout your analysis. However, they have most likely NOT been deleted. If this occurs at any time from the Utility Menu select: Plot > Lines

### 4. Define the Type of Element:

It is now necessary to create elements. This is called 'meshing'. ANSYS first needs to know what kind of elements to use for our problem:

From the Preprocessor Menu, select: **Element Type > Add/Edit/Delete**. The following window will then appear.

Click on the 'Add...' button. The following window will appear:



For this example, we will use the **2D spar element** as selected in the above figure. Select the element shown and click 'OK'. You should see '**Type 1 LINK1**' in the 'Element Types' window.

Click on 'Close' in the 'Element Types' dialog box.

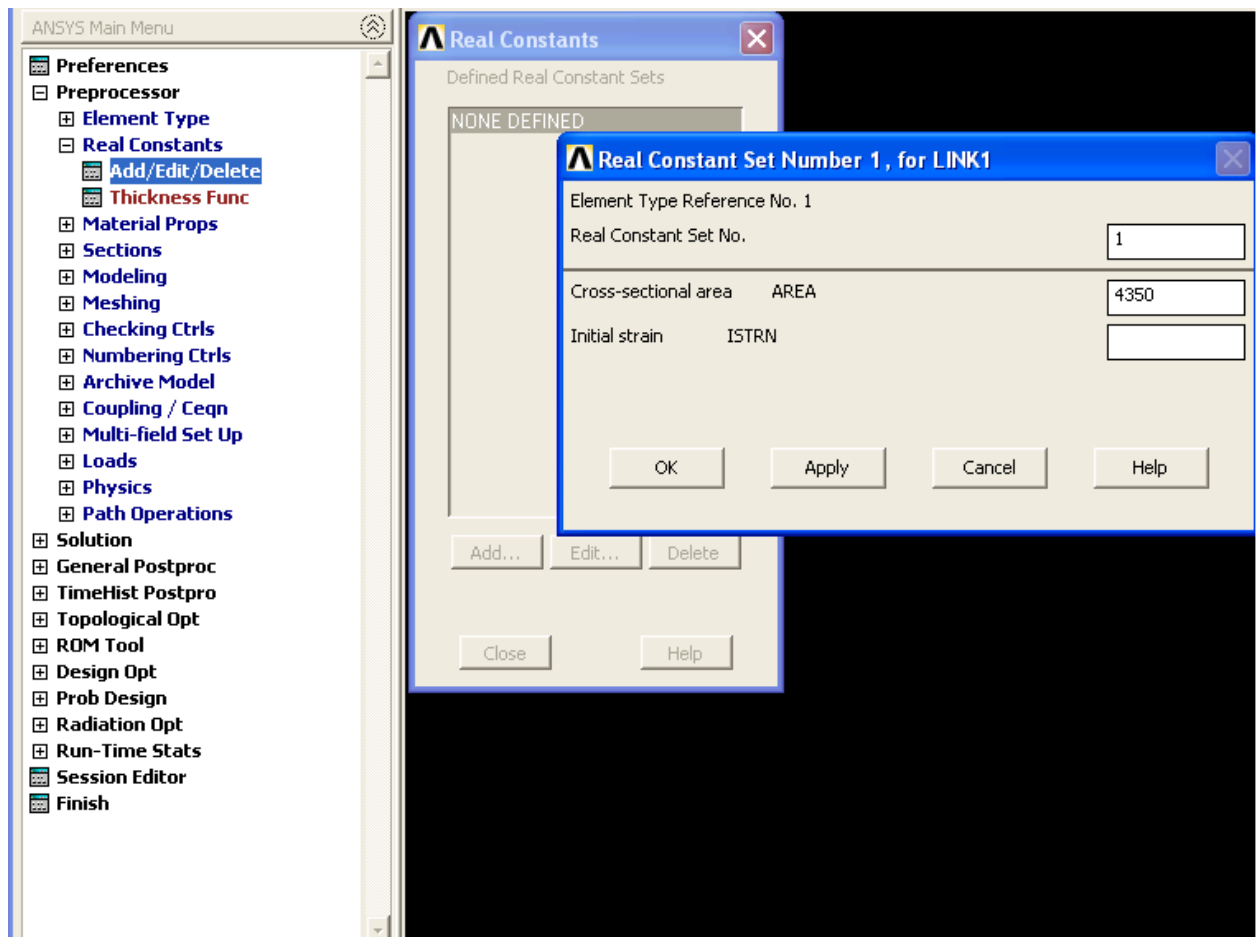
### 5. Define Geometric Properties

We now need to specify geometric properties for our elements:

In the Preprocessor menu, select **Real Constants > Add/Edit/Delete**.

Click **Add...** and select '**Type 1 LINK1**' (actually it is already selected). Click on 'OK'. The following window will appear:





As shown in the window above, enter the cross-sectional area (4350mm):

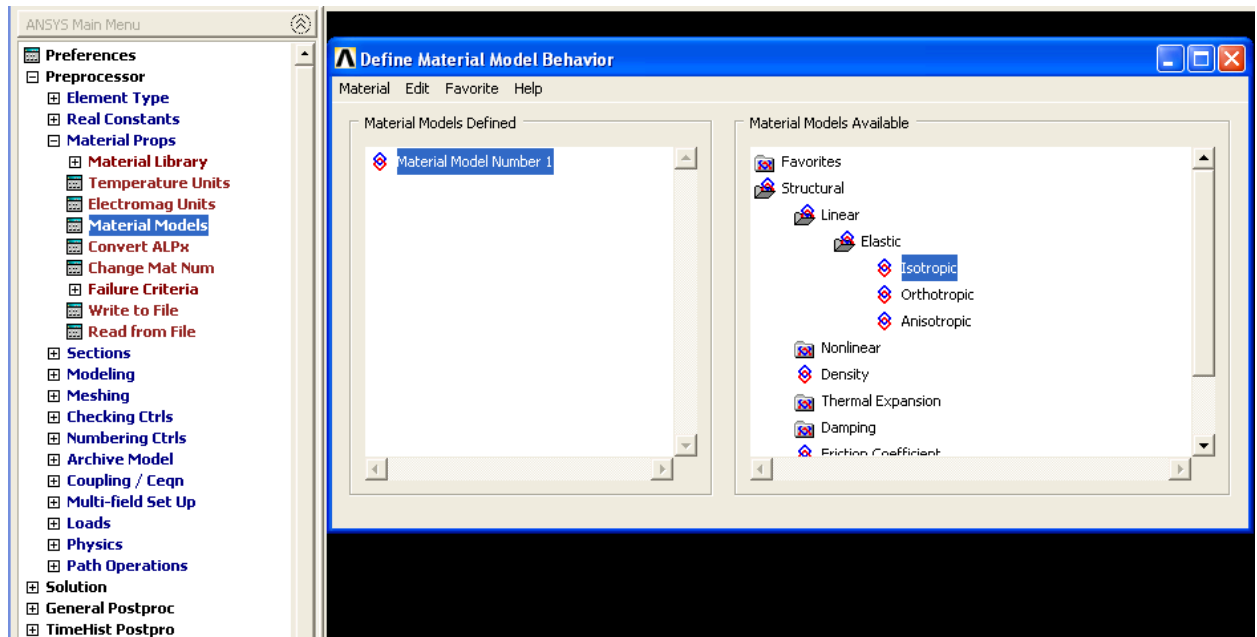
Click on 'OK'.

'Set 1' now appears in the dialog box. Click on 'Close' in the 'Real Constants' window.

## 6. Element Material Properties:

You then need to specify material properties:

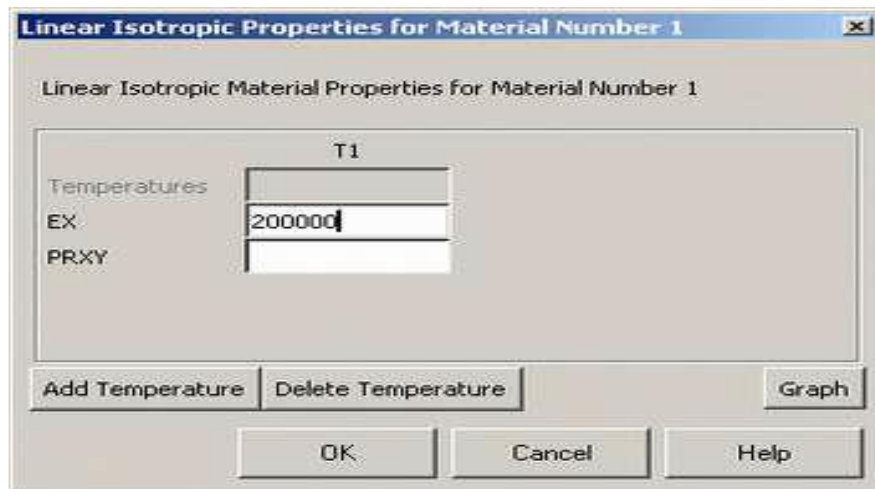
In the 'Preprocessor' menu select **Material Props > Material Models**



Double click on **Structural > Linear > Elastic > Isotropic**

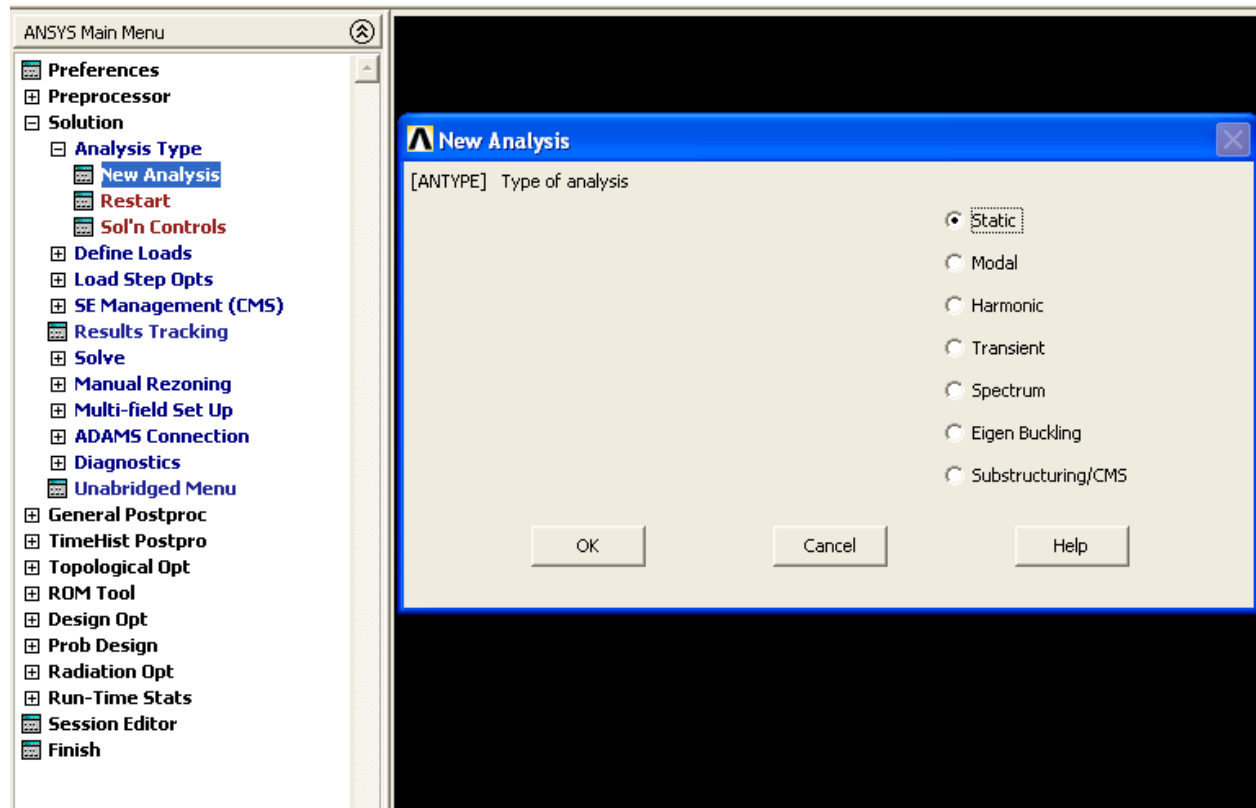
We are going to give the properties of Steel. Enter the following field:

EX = 200000



## 1. Define Analysis Type

First you must tell ANSYS how you want it to solve this problem:



From the Solution Menu, select **Analysis Type > New Analysis**.

Ensure that 'Static' is selected; i.e. you are going to do a static analysis on the truss as opposed to a dynamic analysis, for example.

Click 'OK'.

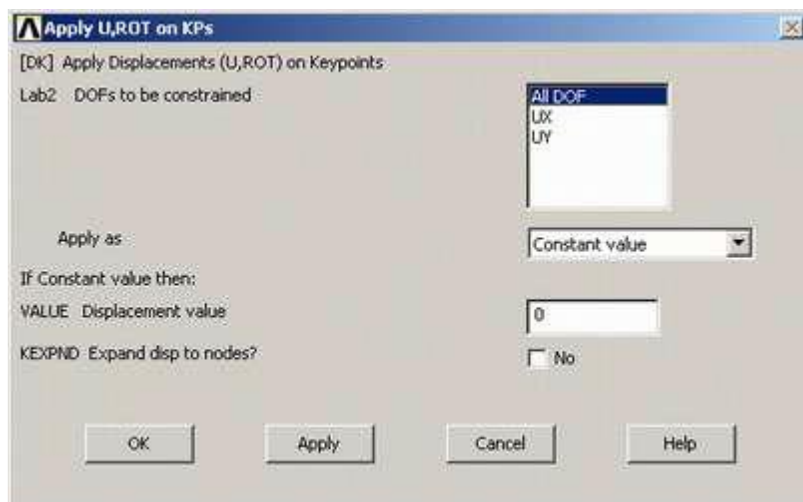
## 2. Apply Constraints

It is necessary to apply constraints to the model otherwise the model is not tied down or grounded and a singular solution will result. In mechanical structures, these constraints will typically be fixed, pinned and roller-type connections. As shown above, the left end of the truss bridge is pinned while the right end has a roller connection.

In the Solution menu, select **Define Loads > Apply > Structural > Displacement > On Keypoints**



Select the left end of the bridge (Keypoint 1) by clicking on it in the Graphics Window and click on 'OK'



This location is fixed which means that all translational and rotational degrees of freedom (DOFs) are constrained. Therefore, select '**All DOF**' by clicking on it and click 'OK'. You will see some blue triangles in the graphics window indicating the displacement constraints.

### 3. Apply Loads

As shown in the diagram, there are four downward loads of 280kN, 210kN, 280kN, and 360kN at keypoints 1, 3, 5, and 7 respectively.

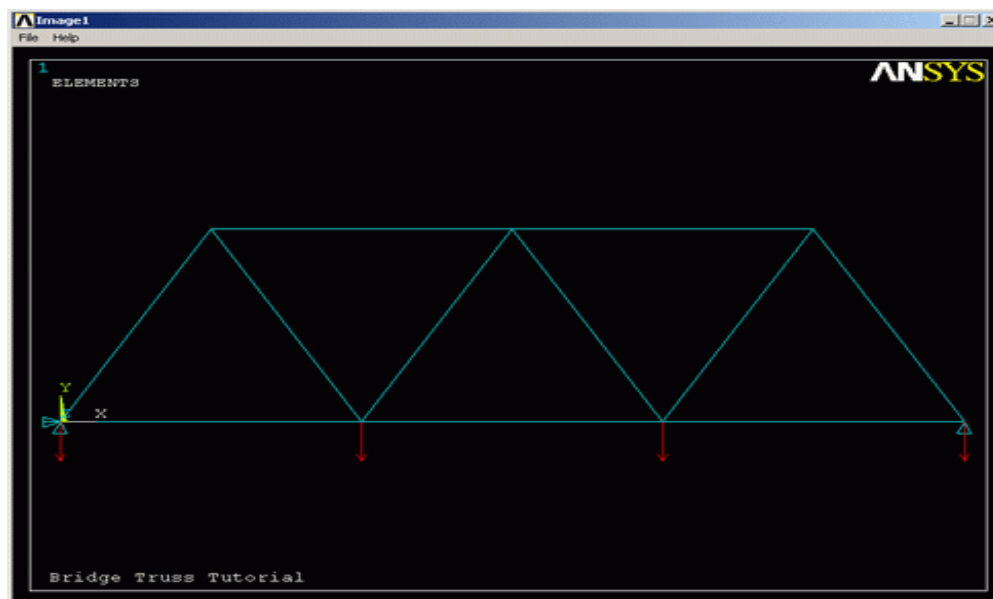
**Select Define Loads > Apply > Structural > Force/Moment > on Keypoints.**



Select FY in the 'Direction of force/mm'. This indicates that we will be applying the load in the 'y' direction

Enter a value of -280000 in the 'Force/moment value' box and click 'OK'. Note that we are using units of N here; this is consistent with the previous values input.

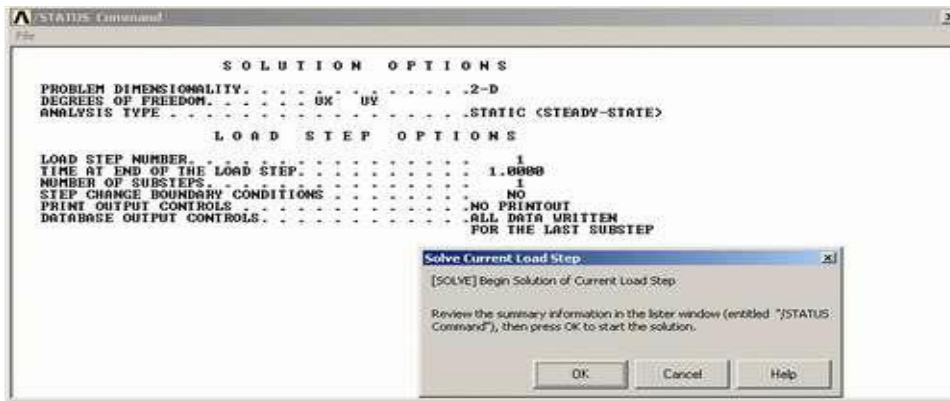
The force will appear in the graphics window as a red arrow. Apply the remaining loads in the same manner. The applied loads and constraints should now appear as shown below



#### 4. Solving the System:

We now tell ANSYS to find the solution:

In the 'Solution' menu select **Solve > Current LS**. This indicates that we desire the solution under the current Load Step (LS).



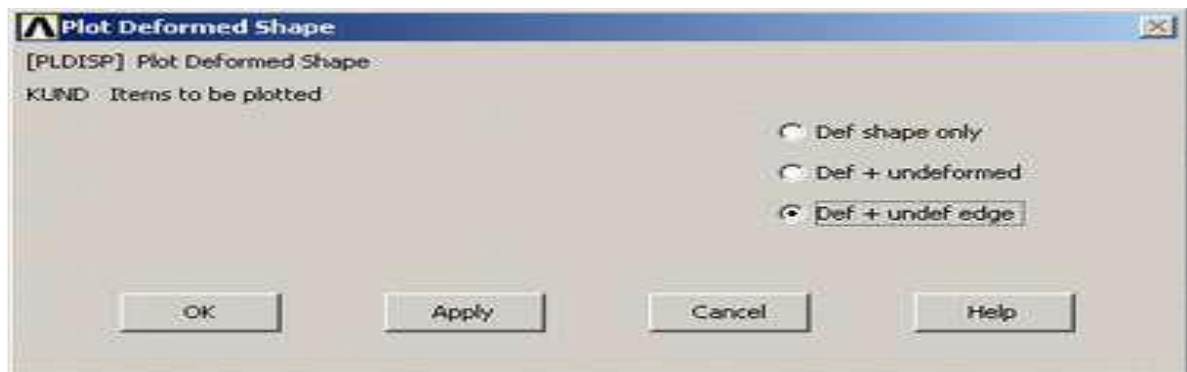
The above windows will appear. Ensure that your solution options are the same as shown above and click 'OK'.

Once the solution is done the following window will pop up. Click 'Close' and close the /STATUS Command Window.

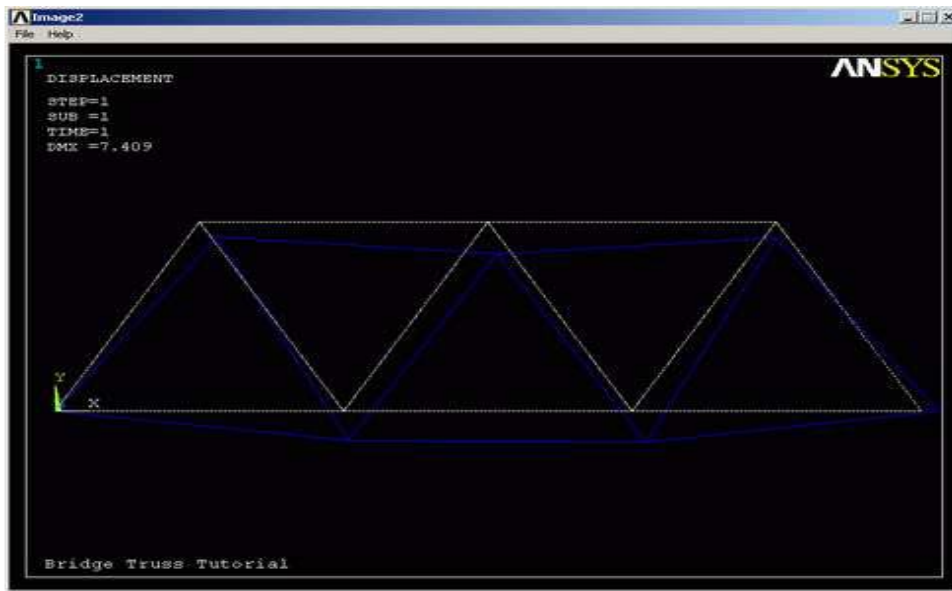


## Deformation

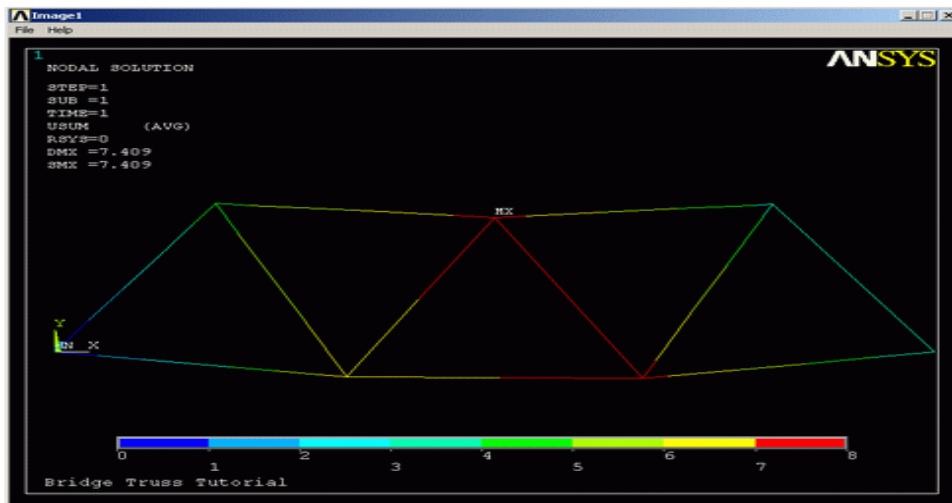
In the General Postproc menu, select **Plot Results > Deformed Shape**. The following window will appear.



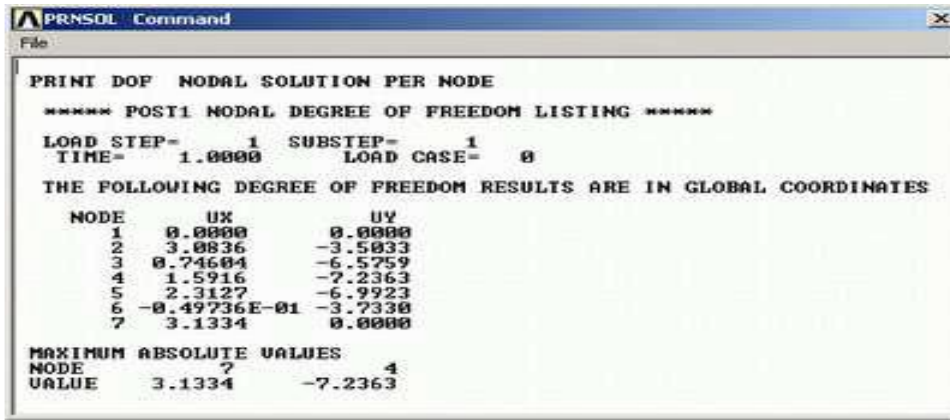
Select '**Def + undef edge**' and click 'OK' to view both the deformed and the undeformed object.



You should obtain the following.



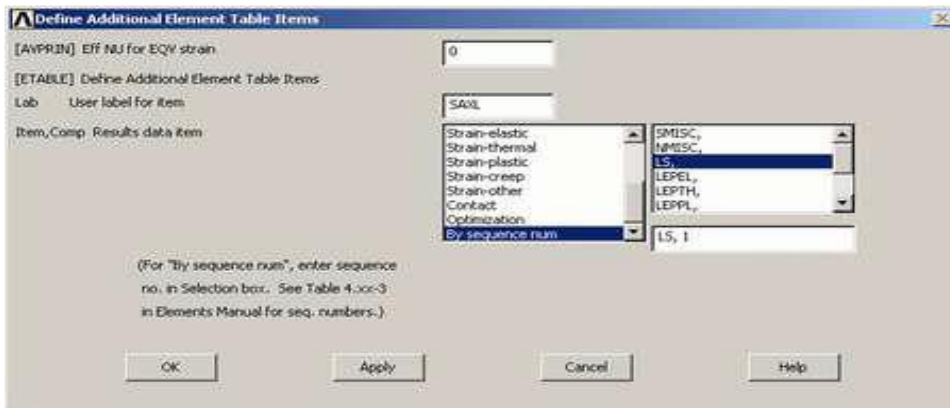
The deflection can also be obtained as a list as shown below. **General Postproc > List Results > Nodal Solution** select 'DOF Solution' and 'ALL DOFs' from the lists in the 'List Nodal Solution' window and click 'OK'. This means that we want to see a listing of all degrees of freedom from the solution.



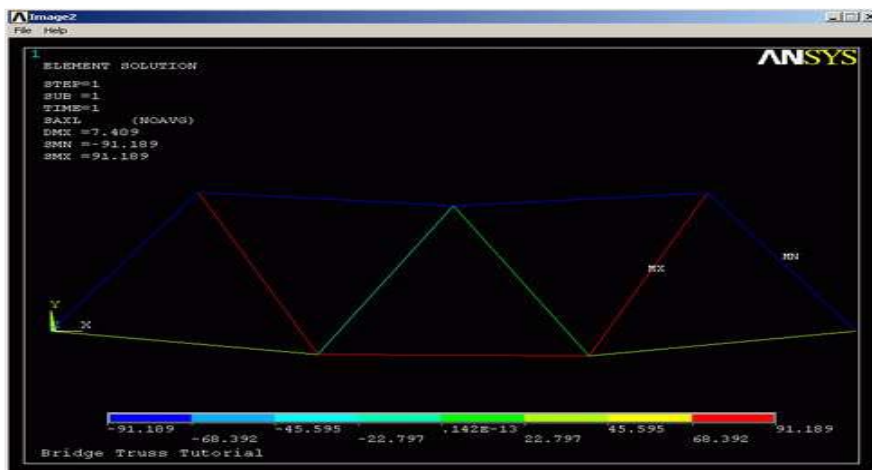
## Axial Stress

For line elements (ie links, beams, spars, and pipes) you will often need to use the Element Table to gain access to derived data (ie stresses, strains). For this example we should obtain axial stress The Element Table is different for each element, therefore, using the item 'LS,1'.

From the General Postprocessor menu select Element Table > Define Table. Click on 'Add'



## Utility Menu > PlotCtrls> Style > Contours





## BARS

- Start → Programs → Ansys → Run interactive → Ok
- File → Clear & Start new → Do not read file → Ok
- File → Change job name → Bar1 → Ok
- File → Change title → Structural Analysis → Ok
- Preferences → Structural and h-method → Ok
- Preprocessor → Real Constant → Add → Link1 → Ok → Cross section area and initial strain → Ok
- Preprocessor → Material Properties → Isotropic1 → Ok → Young's modulus and Poisson ratio → Ok
- Preprocessor → Create → Key points → Inactive CS → Pick all → Ok
- Preprocessor → Size controls → All lines/Picked lines → No of element division → Ok
- Preprocessor → Mesh lines → Pick all → Ok
- Plot controls → Numbering → Node numbers on → Ok
- Solution → Apply → Displacement → On nodes → Select node → Ok
- Solution → Apply → Force → On nodes → Ok
- Solution → Solve → Current LS → Ok
- General postprocessor → Element table → Define table → Add → By sequence num → SMISC 1 2 3 4 → Apply → NMISC 2 4 8 12 → Ok
- General postprocessor → Plot element table → Item to be plotted → SMISC2 SMISC8 → Ok
- General postprocessor → Plot element table → Item to be plotted → SMISC6 SMISC12 → Ok
- General postprocessor → Plot element table → Item to be plotted → NMISC1 NMISC3 → Ok
- General postprocessor → Plot element table → Item to be plotted → NMISC2 NMISC4 → Ok
- General postprocessor → List results → Nodal solution → All DOF → Ok
- Standard tool bar → File → Save as save database to filename. jpeg → Ok
- Standard tool bar → Exit → Quit-no save → Ok

## TRUSSES

- Start → Programs → Ansys → Run interactive → Ok
- File → Clear & Start new → Do not read file → Ok
- File → Change job name → truss1 → Ok
- File → Change title → Structural analysis → Ok
- Preferences → Structural and h-method → Ok
- Preprocessor → Real constants → Add → Link1 → Ok → Cross-sectional area and initial strain → Ok
- Preprocessor → Material properties → Isotropic1 → Ok → Young's modulus and Poisson ratio → Ok
- Preprocessor → Create → Key points → Inactive CS → Give co-ordinate → Apply and for last point → Ok
- Preprocessor → Create → Lines → Inactive CS → Pick all → Ok
- Preprocessor → Size controls → All lines/Picked lines → No of element division → Ok
- Preprocessor → Mesh lines → Pick all → Ok
- Plot Controls → Numbering → Node numbers on → Ok
- Solution → Apply → Displacement → On Nodes → Select node → Ok
- Solution → Apply → Force → On nodes → Ok
- Solution → Solve → Current Ls → Ok
- General postprocessor → Element table → Define Table → Add → By sequence num → SMISC 1 2 3 4 → Apply → NMISC 2 4 8 12 → Ok
- General postprocessor → Plot element table → Item to be plotted → SMISC2 SMISC8 → Ok
- General postprocessor → Plot element table → Item to be plotted → SMISC6 SMISC12 → Ok
- General postprocessor → Plot element table → Item to be plotted → NMISC1 NMISC3 → Ok
- General postprocessor → Plot element table → Item to be plotted → NMISC2 NMISC4 → Ok

- General postprocessor → List results → Nodal solution → All DOF → Ok
- Standard tool bar → File → Save as save database to filename. jpeg → Ok
- Standard tool bar → Exit → Quit-no save → Ok

## BEAMS

- Start → Programs → Ansys → Run interactive → Ok
- File → Clear & Start new → Do not read file → Ok
- File → Change job name → truss1 → Ok
- File → Change title → Structural analysis → Ok
- Preferences → Structural and h-method → Ok
- Preprocessor → Real Constant → Add → Link1 → Ok
- Preprocessor → Material Properties → Isotropic1 → Ok → Young's modulus and Poisson ratio → Ok
- Preprocessor → Create → Areas → Rectangle → By 2 corners → Width: Height: → Ok
- Preprocessor → Size controls → All lines/Picked lines → No of element division → Ok
- Preprocessor → Mesh lines → Pick all → Ok
- Plot controls → Numbering → Node numbers on → Ok
- Solution → Apply → Displacement → On nodes → Select node → Ok
- Solution → Apply → Force → On nodes → Ok
- Solution → Solve → Current LS → Ok
- General postprocessor → Element table → Define table → Add → By sequence num → SMISC 1 2 3 4 → Apply → NMISC 2 4 8 12 → Ok
- General postprocessor → Plot element table → Item to be plotted → SMISC2 SMISC8 → Ok
- General postprocessor → Plot element table → Item to be plotted → SMISC6 SMISC12 → Ok
- General postprocessor → Plot element table → Item to be plotted → NMISC1 NMISC3 → Ok

- General postprocessor → Plot element table → Item to be plotted → NMISC2  
NMISC4 → Ok
- General postprocessor → List results → Nodal solution → All DOF → Ok
- Standard tool bar → File → Save as save database to filename. jpeg → Ok
- Standard tool bar → Exit → Quit-no save → Ok

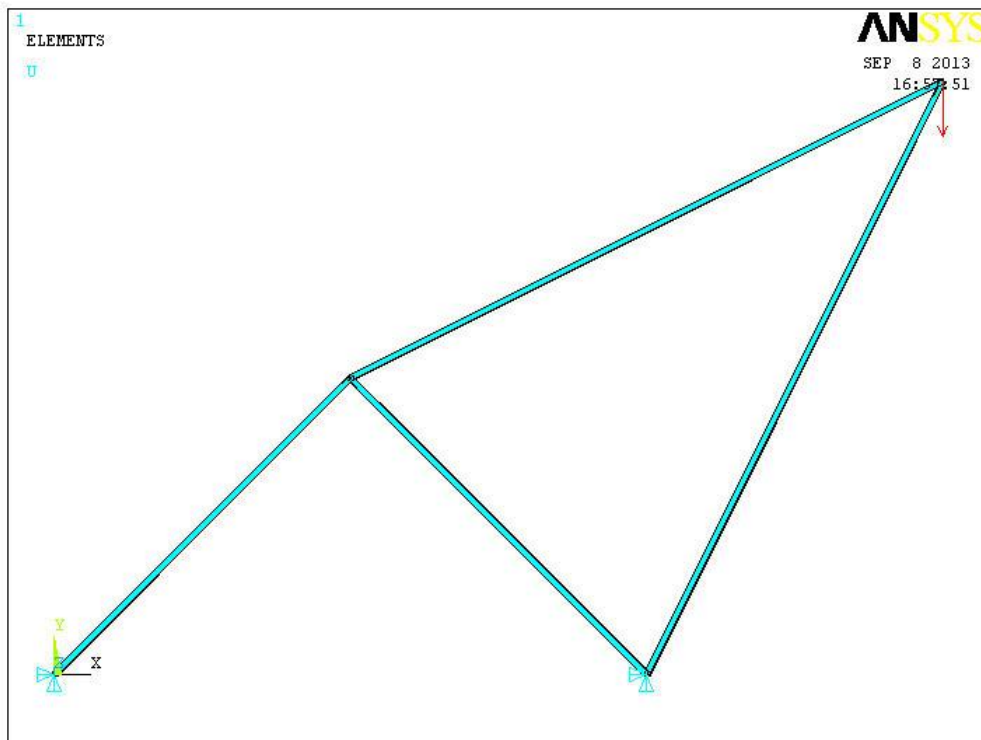
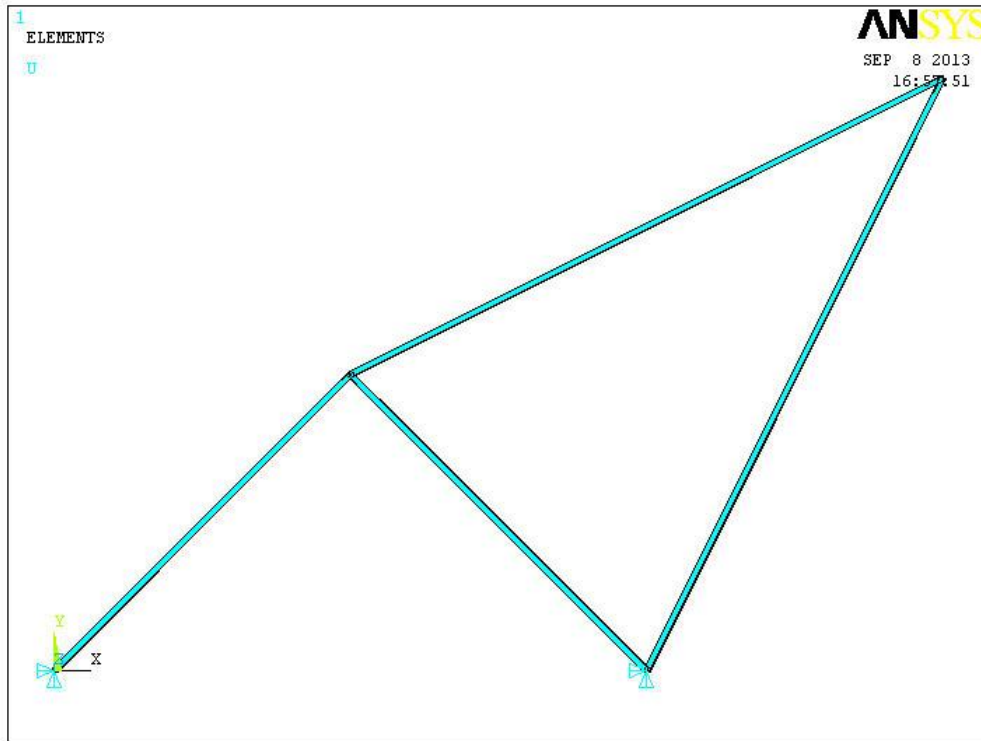
## **THERMAL**

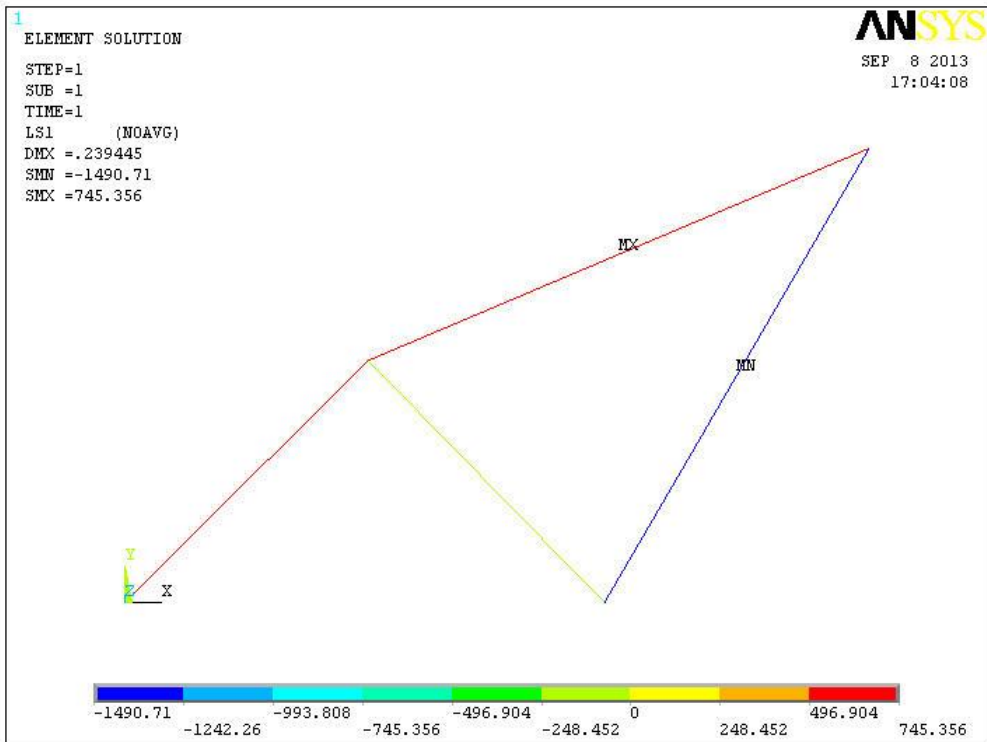
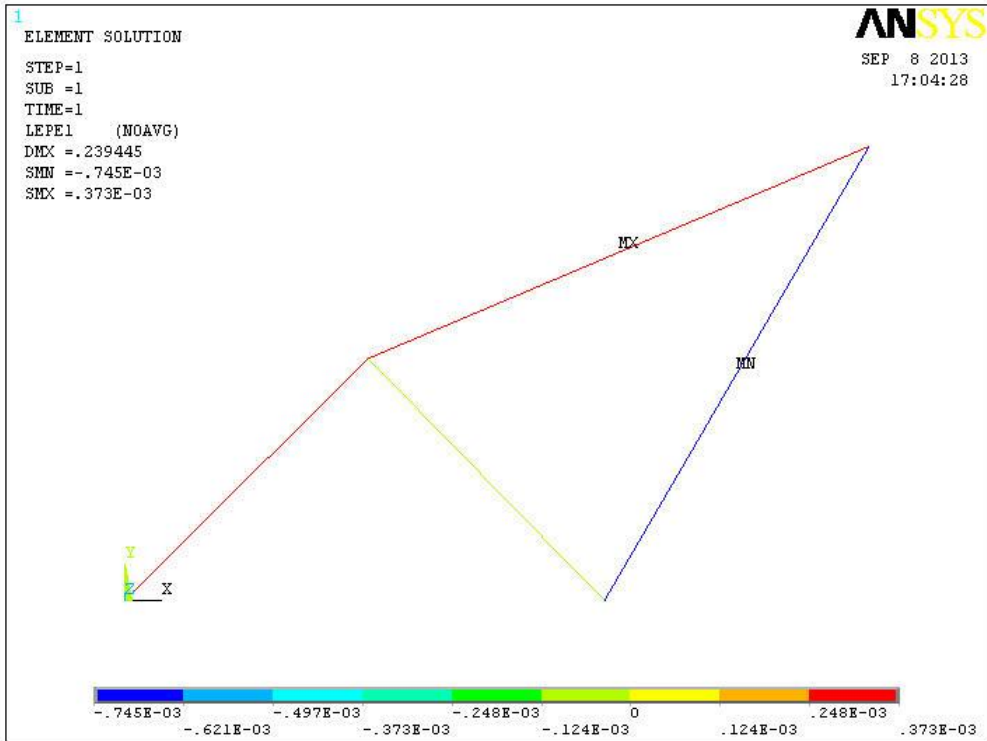
- Start → Programs → Ansys → Run interactive → Ok
- File → Clear & Start new → Do not read file → Ok
- File → Change job name → truss1 → Ok
- File → Change title → Structural analysis → Ok
- Preferences → Thermal and h-method → Ok
  
- Preprocessor → Element type → Add → Solid → Thermal mass → Ok
- Preprocessor → Material Properties → Thermal conductivity → Isotropic1 → Ok →  
Young's modulus and Poisson ratio → Ok
- Preprocessor → Create → Areas → Rectangle → By 2 corners → Width: Height: →  
Ok
- Preprocessor → Size controls → Manual size → Areas → All areas → Edge length  
→ Ok
- Preprocessor → Mesh → Areas → Mapped → 3 or 4 sided → Select area → Ok
- Plot controls → Numbering → Node numbers on → Ok
- Solution → Apply → Thermal → Temperature → On nodes → Pick lines → Ok →  
Temp: → Ok
- Solution → Solve → Current LS → Ok
- General postprocessor → Plot results → Contour Plot → Nodal solution → All DOF  
→ Ok
- General postprocessor → List results → Nodal solution → Thermal gradient vector  
sum → Ok
- General postprocessor → List results → Nodal solution → Thermal flux → Thermal  
flux sum → Ok

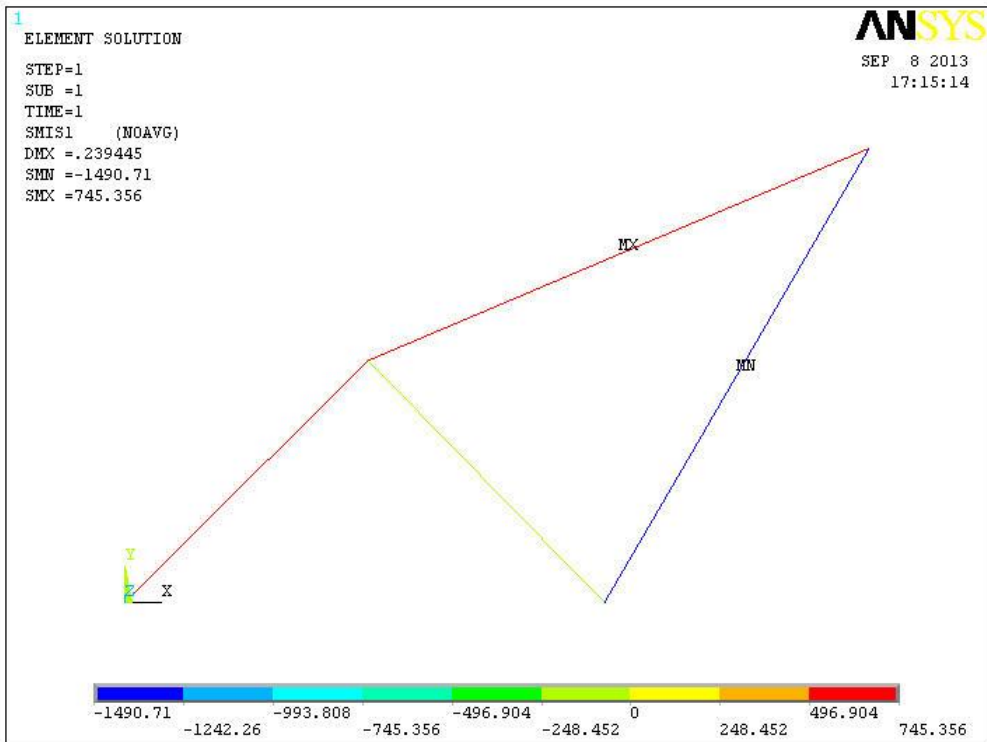
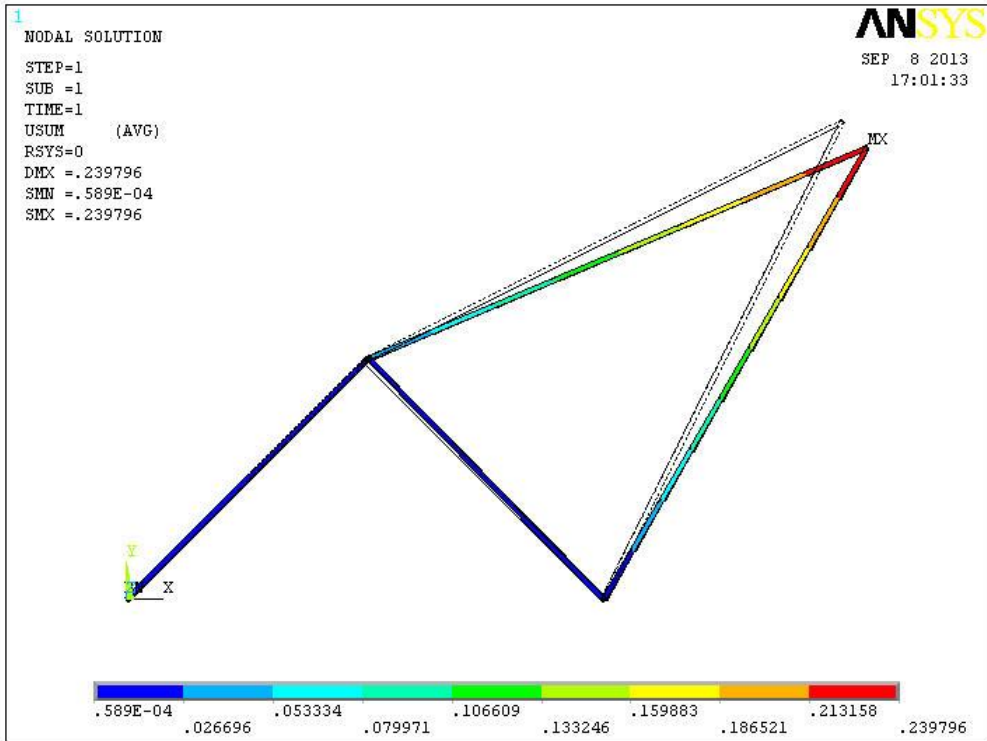
- Standard tool bar → File → Save as save database to filename. jpeg → Ok
- Standard tool bar → Exit → Quit-no save → Ok

## EXERCISE-1

**Problem 1:** Determine the nodal deflections, reaction forces, and stress for the truss system shown below









## Solution

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.23570E-01	0.11785E-01	0.0000	0.26352E-01
3	0.0000	0.0000	0.0000	0.0000
4	0.16351	-0.17492	0.0000	0.23945

### MAXIMUM ABSOLUTE VALUES

NODE	4	4	0	4
VALUE	0.16351	-0.17492	0.0000	0.23945

\*\*\*\*\* POST1 ELEMENT TABLE LISTING \*\*\*\*\*

STAT	CURRENT	CURRENT
ELEM	LS1	LEPE1
1	707.11	0.35355E-03
2	-235.70	-0.11785E-03
3	745.36	0.37268E-03
4	-1490.7	-0.74536E-03

### MINIMUM VALUES

ELEM	4	4
VALUE	-1490.7	-0.74536E-03

### MAXIMUM VALUES

ELEM	3	3
VALUE	745.36	0.37268E-03

\*\*\*\*\* 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
1	-500.00	-500.00
3	500.00	1500.0

### TOTAL VALUES

VALUE	0.11369E-12	1000.0
-------	-------------	--------

ELEM	SMIS1
1	707.11

2 -235.70  
3 745.36  
4 -1490.7

MINIMUM VALUES

ELEM 4

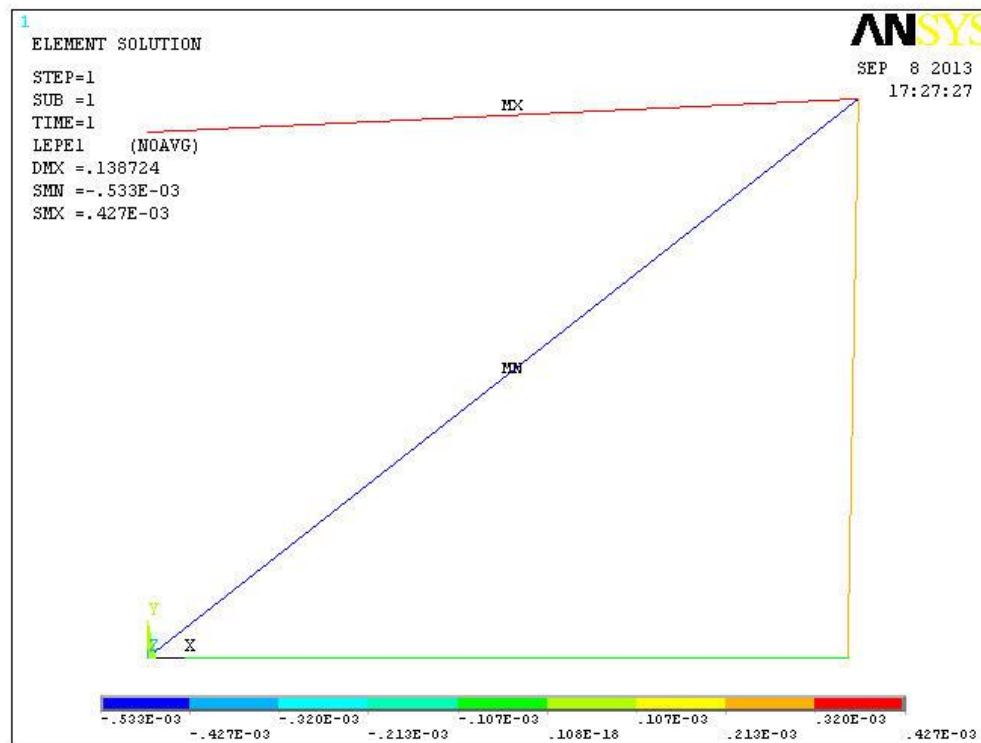
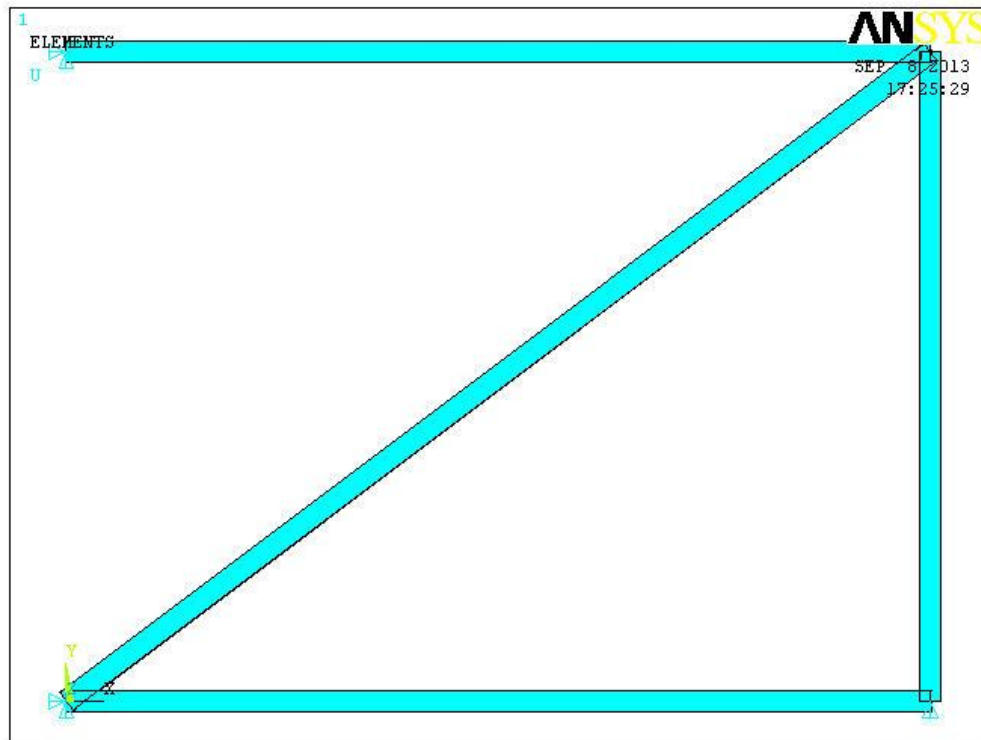
VALUE -1490.7

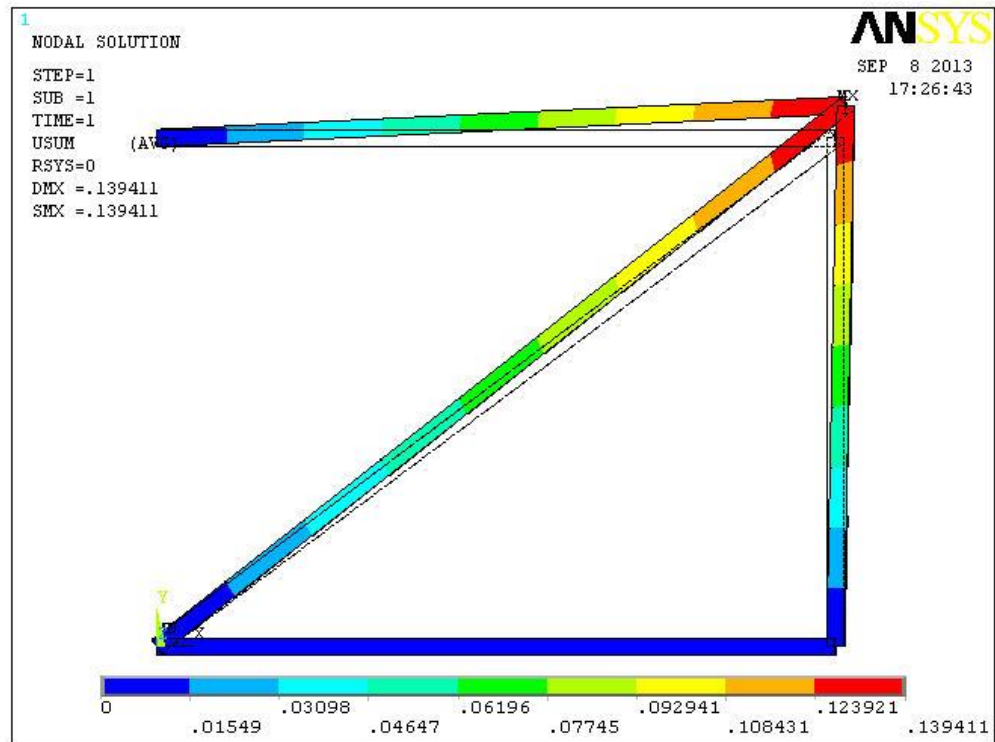
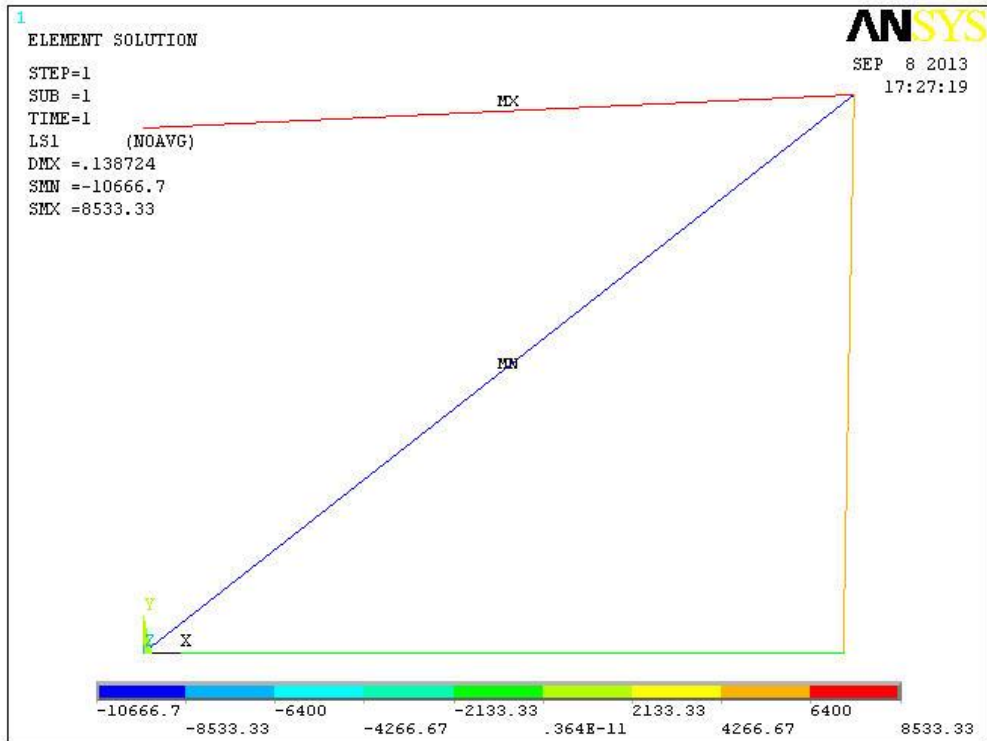
MAXIMUM VALUES

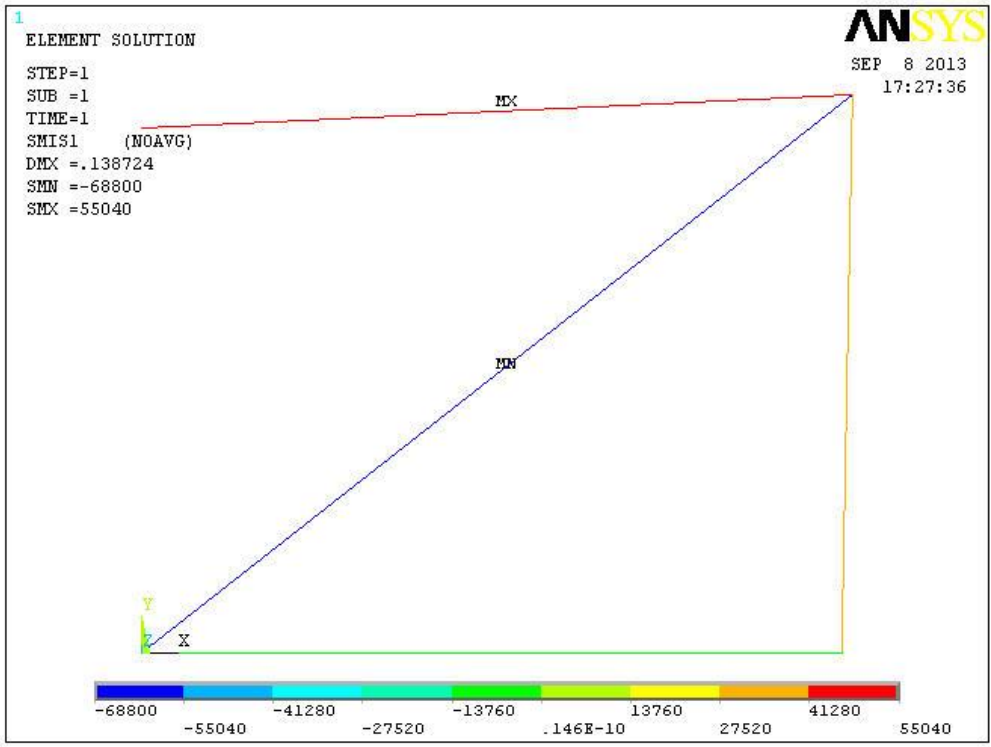
ELEM 3

VALUE 745.36

**Problem 2:** Determine the nodal deflections, reaction forces, and stress for the truss system shown below







## Solution

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000	0.0000
3	0.42667E-01	0.13200	0.0000	0.13872
4	0.0000	0.0000	0.0000	0.0000

### MAXIMUM ABSOLUTE VALUES

NODE	3	3	0	3
VALUE	0.42667E-01	0.13200	0.0000	0.13872

\*\*\*\*\* POST1 ELEMENT TABLE LISTING \*\*\*\*\*

STAT	CURRENT	CURRENT	CURRENT
ELEM	LS1	LEPE1	SMIS1
1	0.0000	0.0000	0.0000
2	6400.0	0.32000E-03	41280.
3	-10667.	-0.53333E-03	-68800.
4	8533.3	0.42667E-03	55040.

### MINIMUM VALUES

ELEM	3	3	3
VALUE	-10667.	-0.53333E-03	-68800.

### MAXIMUM VALUES

ELEM	4	4	4
VALUE	8533.3	0.42667E-03	55040.

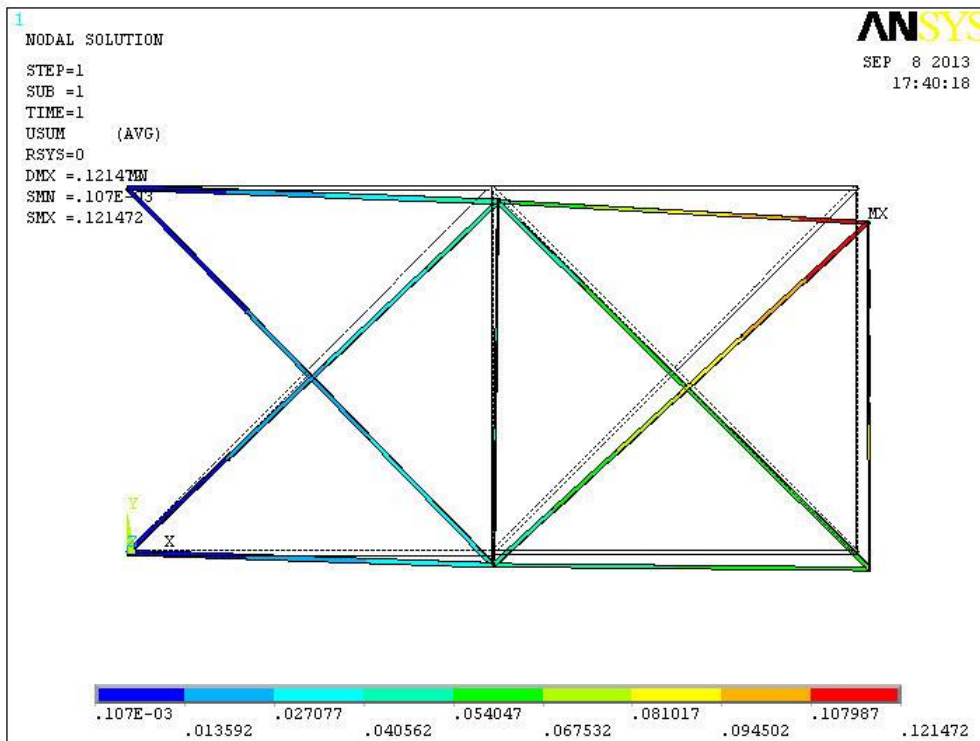
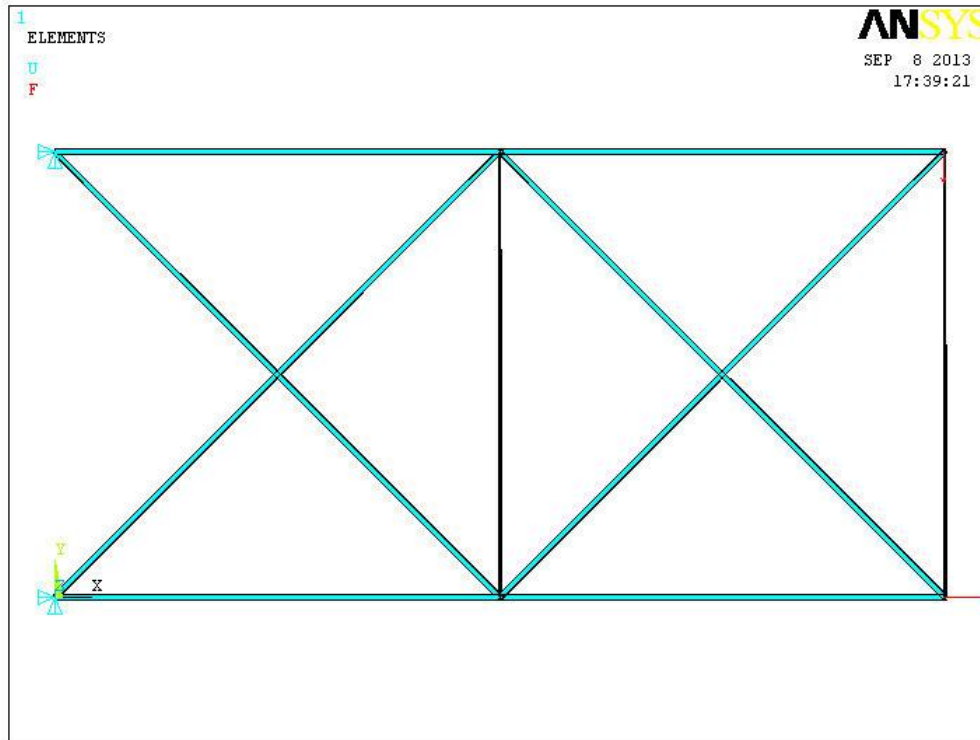
THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

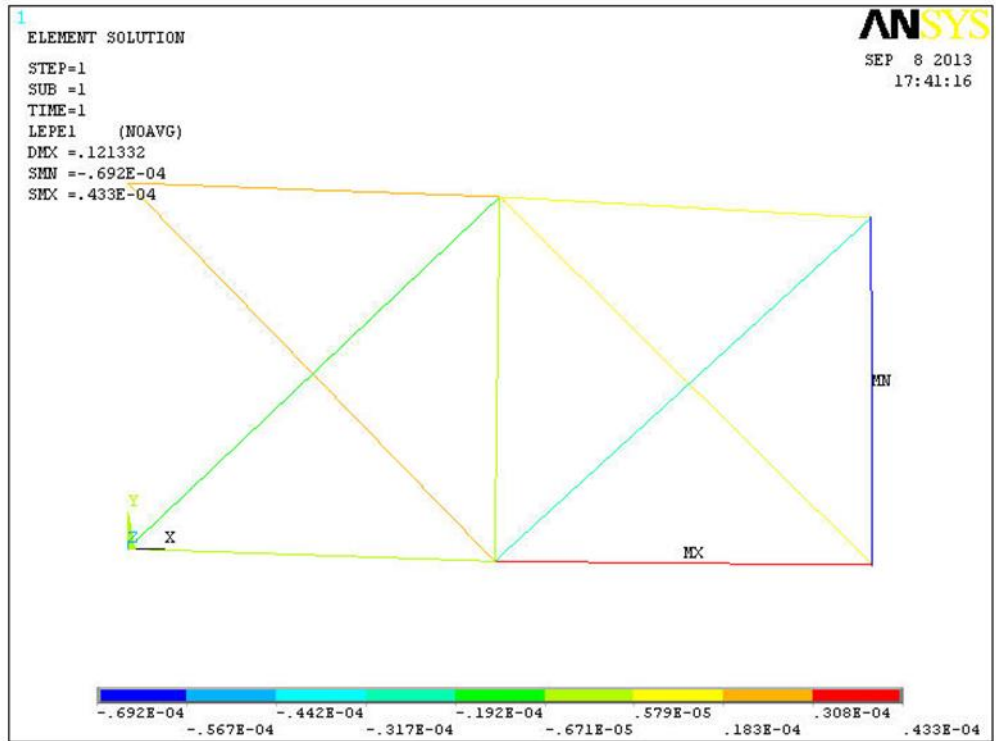
NODE	FX	FY	FZ
1	55040.	41280.	
2		-41280.	
4	-55040.	0.0000	

### TOTAL VALUES

VALUE	-0.72760E-11	-0.29104E-10	0.0000
-------	--------------	--------------	--------

**Problem 3:** Determine the nodal deflections, reaction forces, and stress for the truss system shown below





## Solution

\*\*\*\*\* POST1 ELEMENT TABLE LISTING \*\*\*\*\*

STAT CURRENT

ELEM	LEPE1	Current
1	0.28274E-05	
2	0.43277E-04	
3	-0.69190E-04	
4	0.18277E-04	
5	-0.56981E-05	
6	0.21577E-04	
7	-0.14965E-04	
8	0.24319E-04	
9	-0.22975E-04	
10	0.16308E-04	

MINIMUM VALUES

ELEM 3  
 VALUE -0.69190E-04

MAXIMUM VALUES

ELEM 2



VALUE 0.43277E-04

\*\*\*\*\* POST1 NODAL DEGREE OF FREEDOM LISTING \*\*\*\*\*

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

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL  
COORDINATE SYSTEM

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.25446E-02	-0.41229E-01	0.0000	0.41307E-01
3	0.41494E-01	-0.53638E-01	0.0000	0.67814E-01
4	0.35869E-01	-0.11591	0.0000	0.12133
5	0.19420E-01	-0.46357E-01	0.0000	0.50260E-01
6	0.0000	0.0000	0.0000	0.0000

MAXIMUM ABSOLUTE VALUES

NODE	3	4	0	4
VALUE	0.41494E-01	-0.11591	0.0000	0.12133

\*\*\*\*\* 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

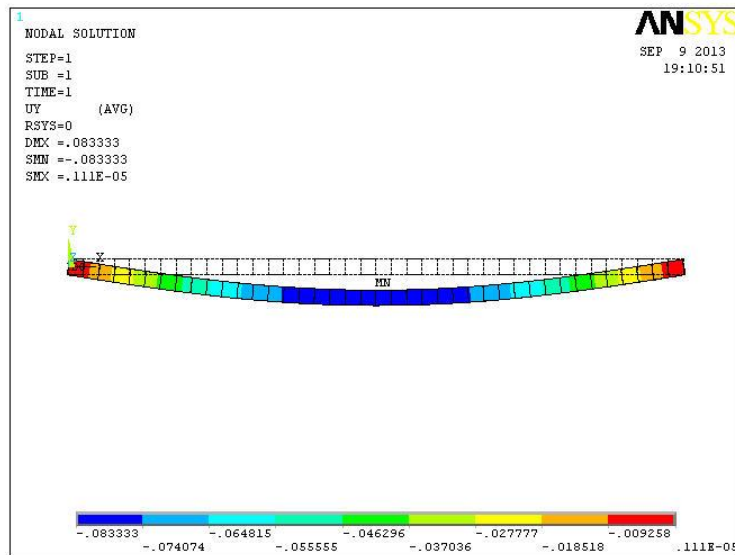
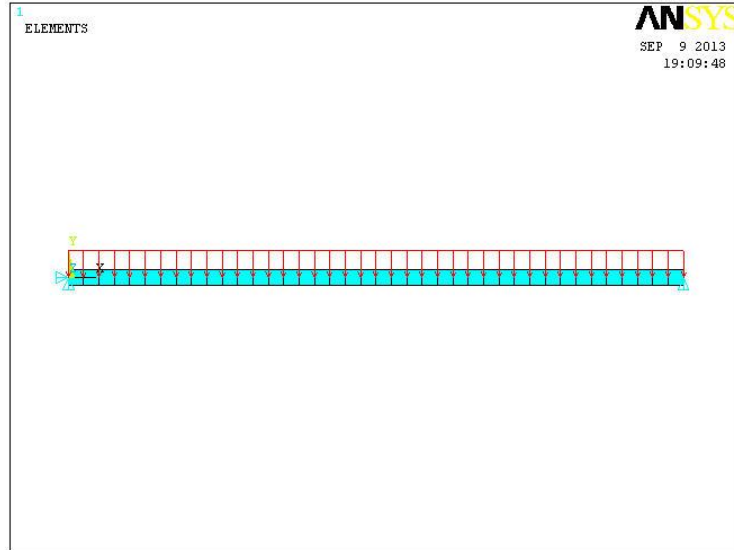
NODE	FX	FY
1	10000.	19048.
6	-0.10000E+06	30952.

TOTAL VALUES

VALUE	-90000.	50000.
-------	---------	--------

## EXERCISE-2

**Problem 1:** A distributed load of 1000 N/m (1 N/mm) will be applied to a solid steel beam with a rectangular cross section as shown in the figure below. The cross-section of the beam is 10mm x 10mm while the modulus of elasticity of the steel is 200GPa.



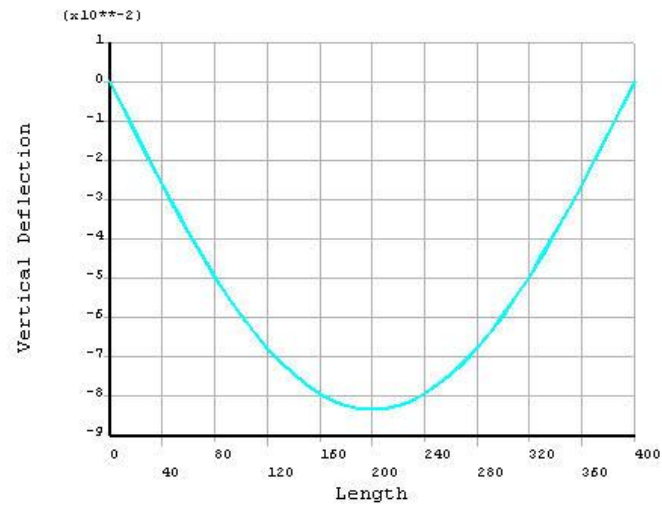
1

Graph(1,2)

COL 2

ANSYS

SEP 9 2013  
19:16:07



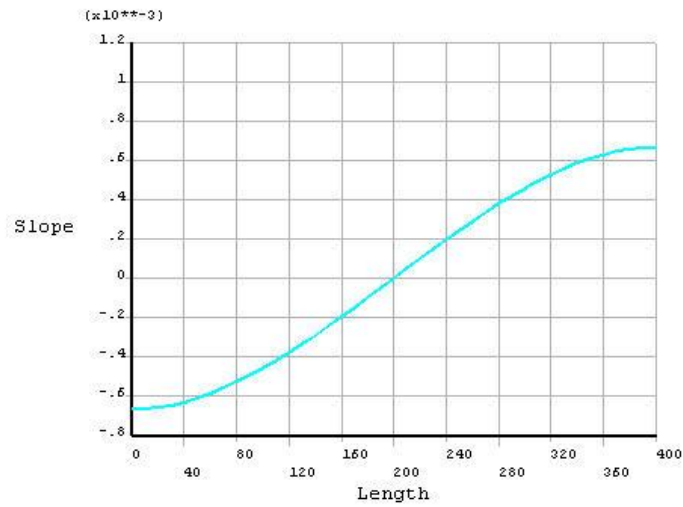
1

Graph(1,3)

COL 3

ANSYS

SEP 9 2013  
19:16:41



## Solution

\*\*\*\*\* POST1 NODAL DEGREE OF FREEDOM LISTING \*\*\*\*\*

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

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL  
COORDINATE SYSTEM

NODE	UX	UY	UZ	USUM
38	0.0000	-0.19783E-01	0.0000	0.19783E-01
39	0.0000	-0.13268E-01	0.0000	0.13268E-01
40	0.0000	-0.66584E-02	0.0000	0.66584E-02
41	0.0000	0.0000	0.0000	0.0000

MAXIMUM ABSOLUTE VALUES

NODE	0	21	0	21
VALUE	0.0000	-0.83333E-01	0.0000	0.83333E-01

\*\*\*\*\* POST1 NODAL DEGREE OF FREEDOM LISTING \*\*\*\*\*

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

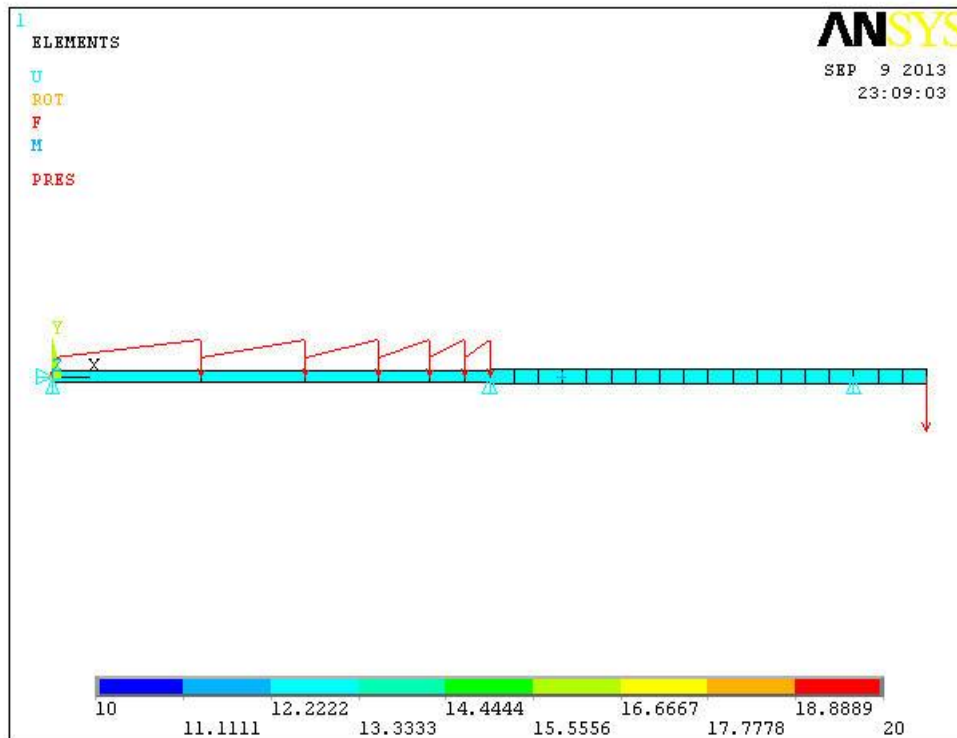
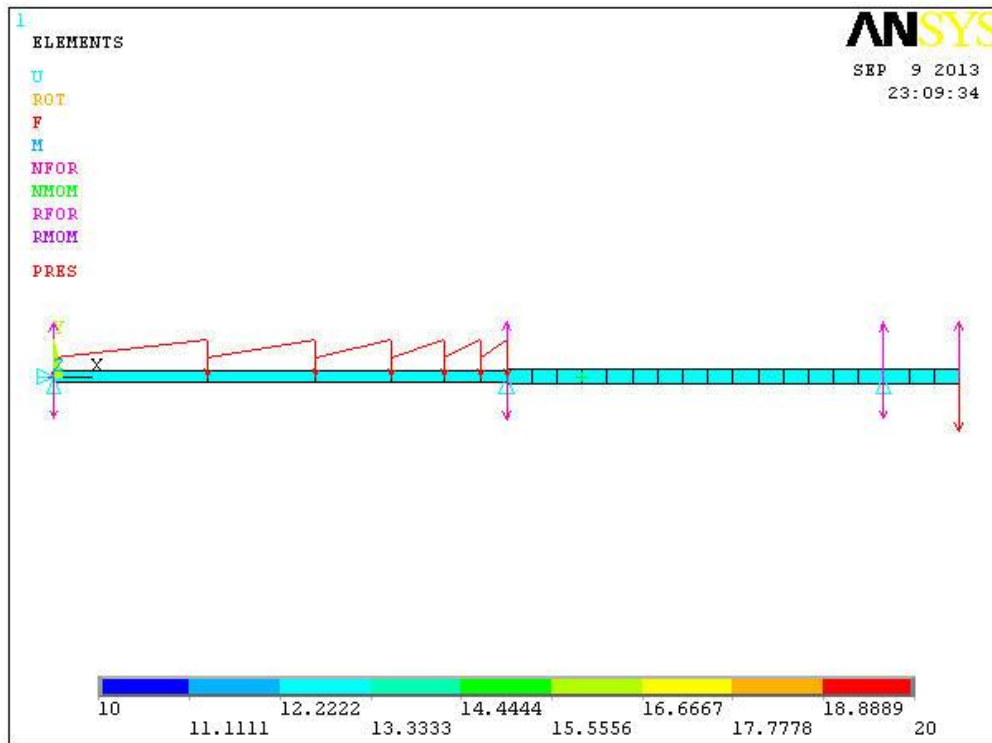
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL  
COORDINATE SYSTEM

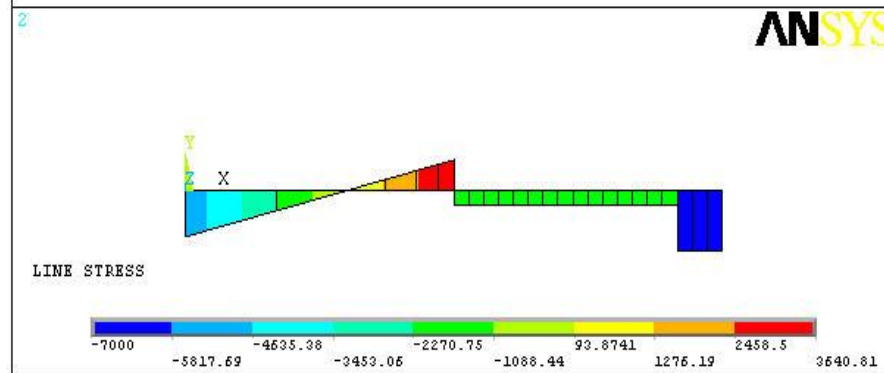
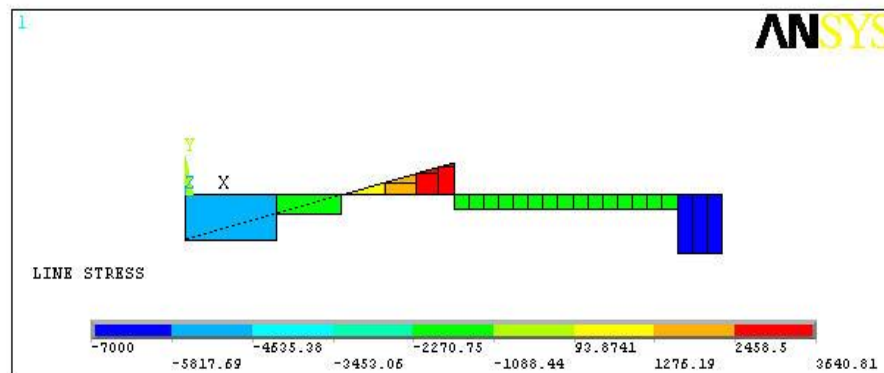
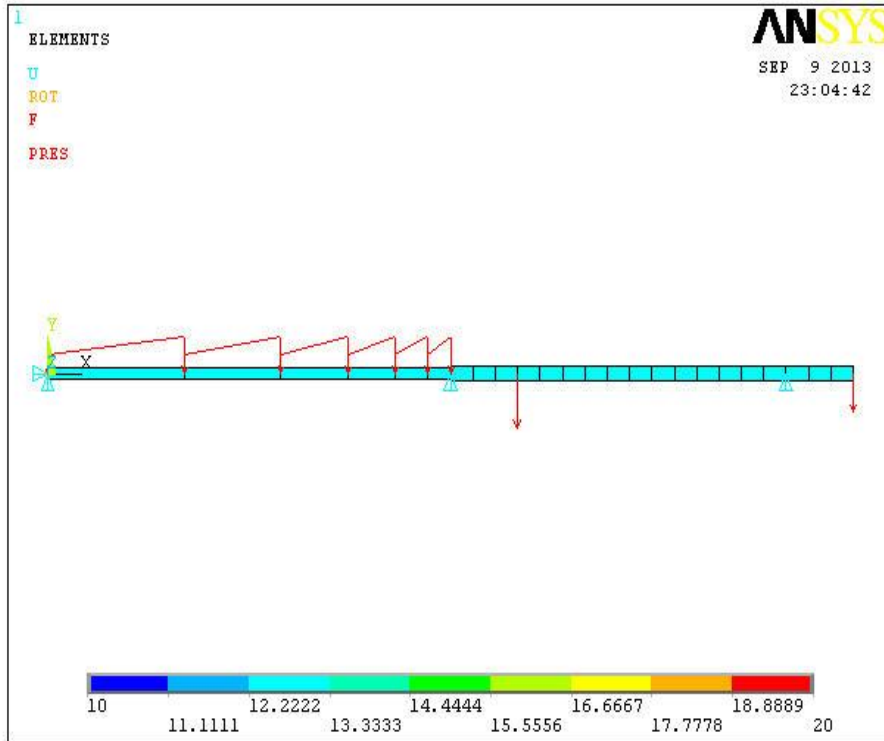
NODE	ROTZ
38	0.64529E-03
39	0.65700E-03
40	0.66421E-03
41	0.66667E-03

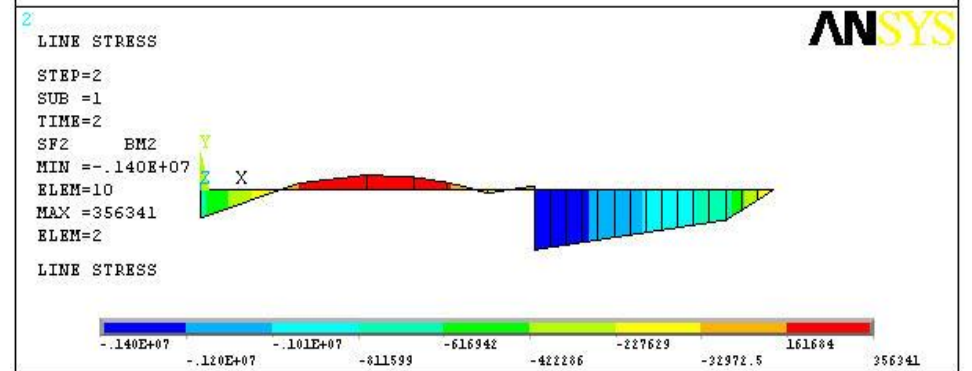
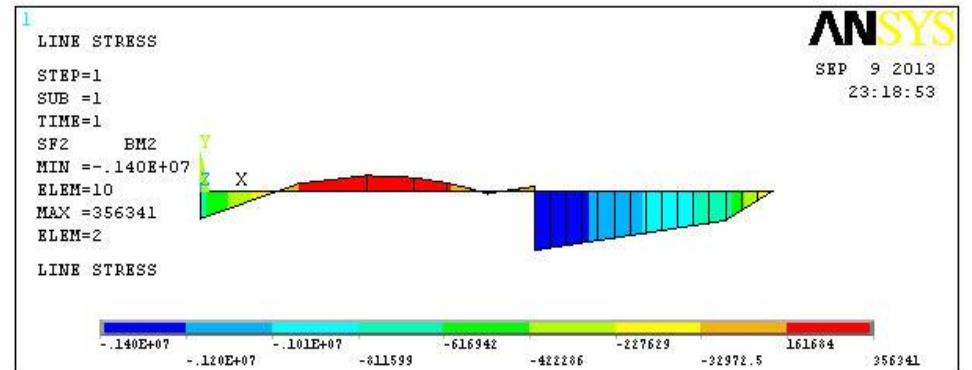
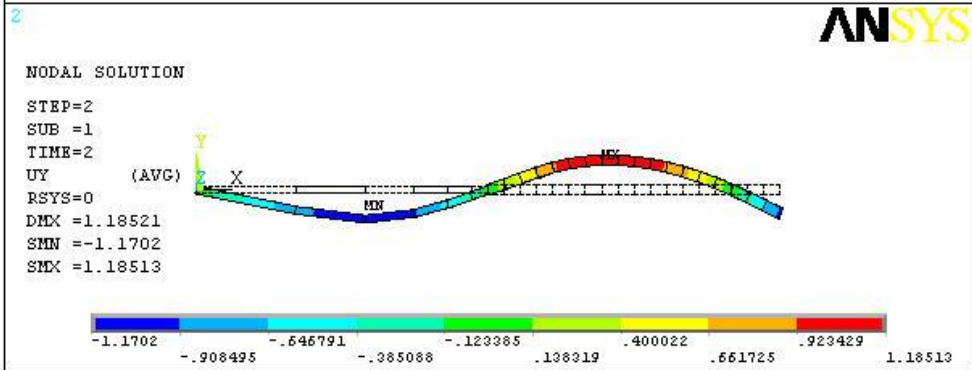
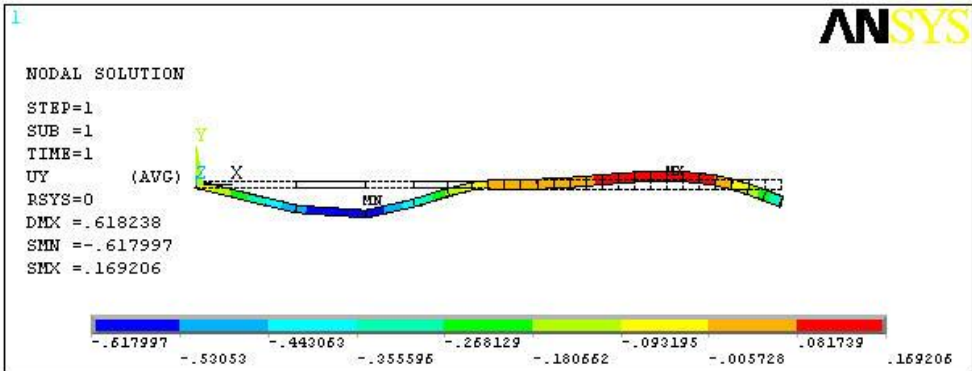
MAXIMUM ABSOLUTE VALUES

NODE	1
VALUE	-0.66667E-03

**Problem 2:** A distributed load of 1000 N/m (1 N/mm) will be applied to a solid steel beam with a rectangular cross section as shown in the figure below. The cross-section of the beam is 10mm x 10mm while the modulus of elasticity of the steel is 200GPa.







## Solution

### MAXIMUM ABSOLUTE VALUES

NODE 4  
VALUE -0.61798

### MAXIMUM ABSOLUTE VALUES

NODE 15  
VALUE 1.1851

### MINIMUM VALUES

ELEM	22	10	22	10
VALUE	-7000.0	-0.13956E+07	-7000.0	-0.13376E+07

### MAXIMUM VALUES

ELEM	6	3	6	2
VALUE	3129.5	0.35634E+06	3640.8	0.35634E+06

### MINIMUM VALUES

ELEM	22	10	22	10
VALUE	-7000.0	-0.13956E+07	-7000.0	-0.13376E+07

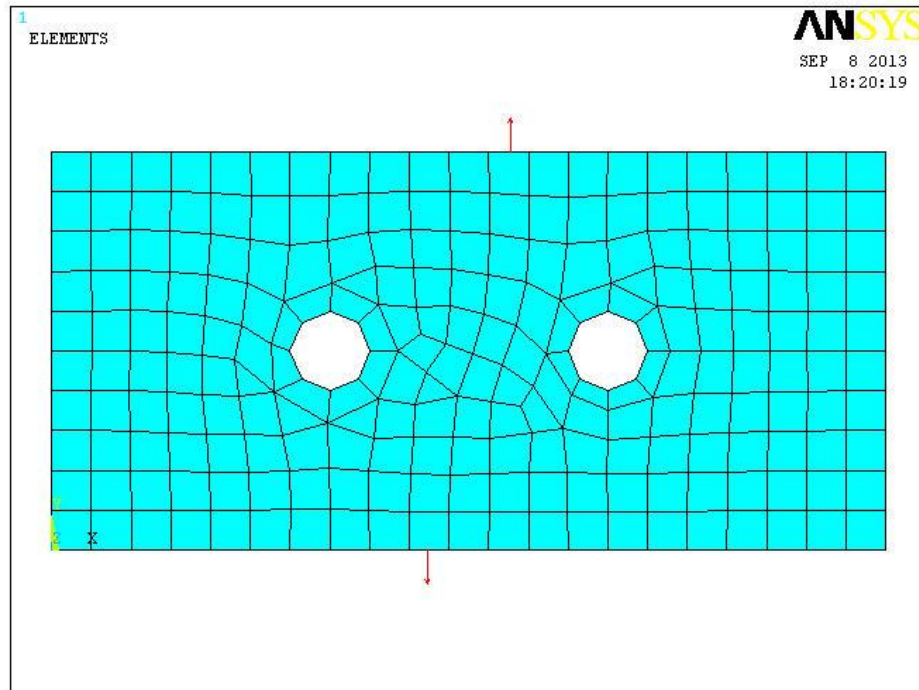
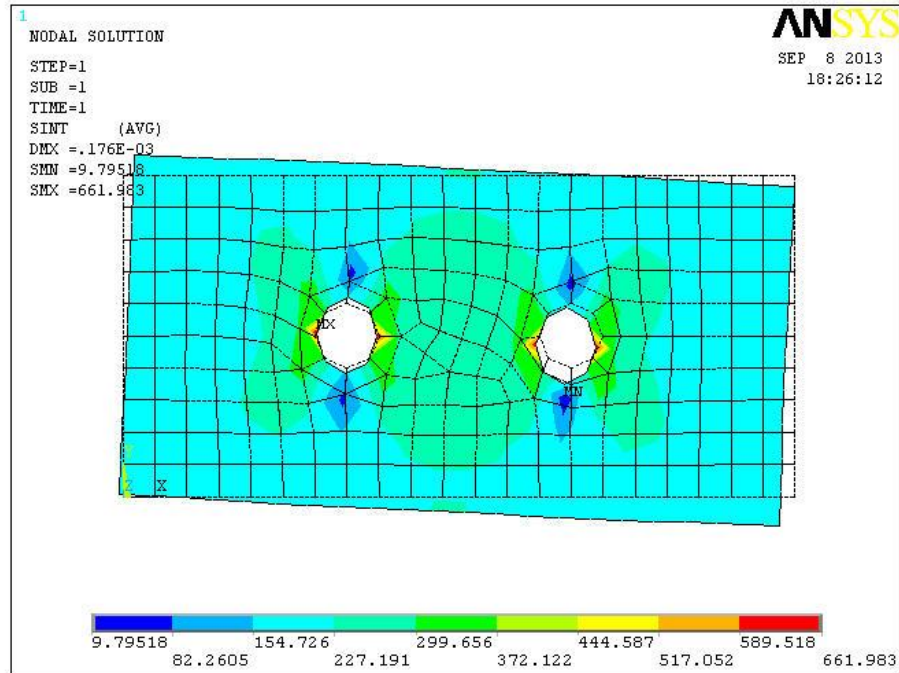
### MAXIMUM VALUES

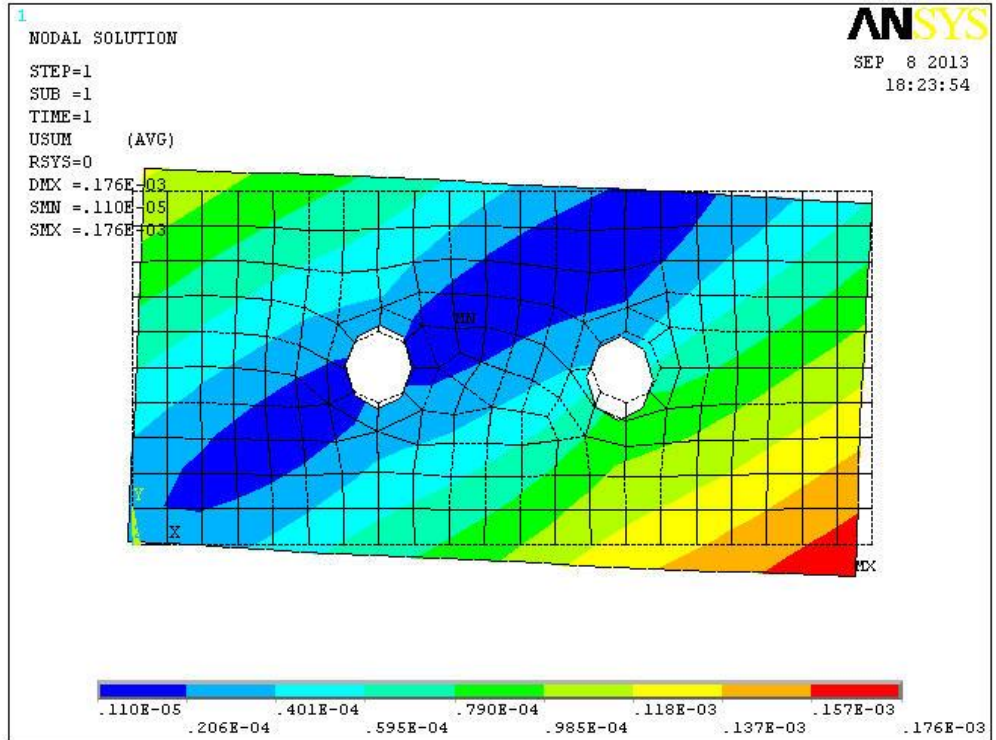
ELEM	6	3	6	2
VALUE	3129.5	0.35634E+06	3640.8	0.35634E+06



## EXERCISE-3

**Problem 1:** A rectangular plate with two circular holes, subjected to a uniform tensile stress of  $200\text{N/cm}^2$  as shown in figure. Thickness of plate is  $0.5\text{cm}$ ,  $E=2 \times 10^7 \text{ N/cm}^2$  poisson's ratio  $\nu=0.3$ . perform the static analysis and calculate the deformations.





## Solution

### MAXIMUM ABSOLUTE VALUES

NODE	2	2	0	2
VALUE	-0.84066E-04	-0.15510E-03	0.0000	0.17642E-03

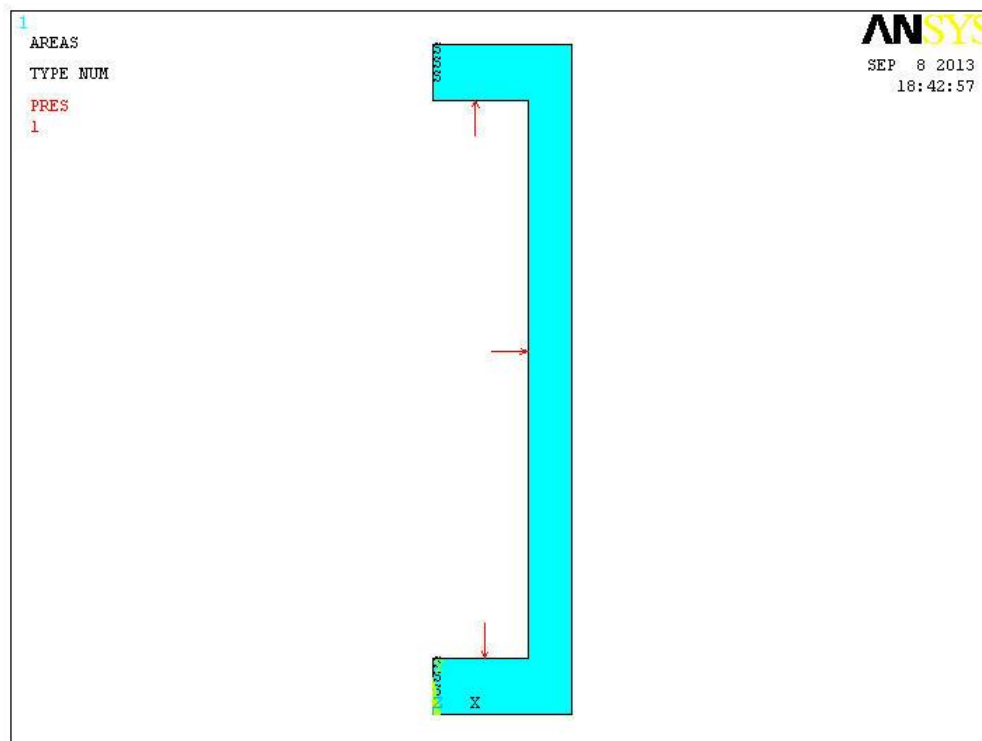
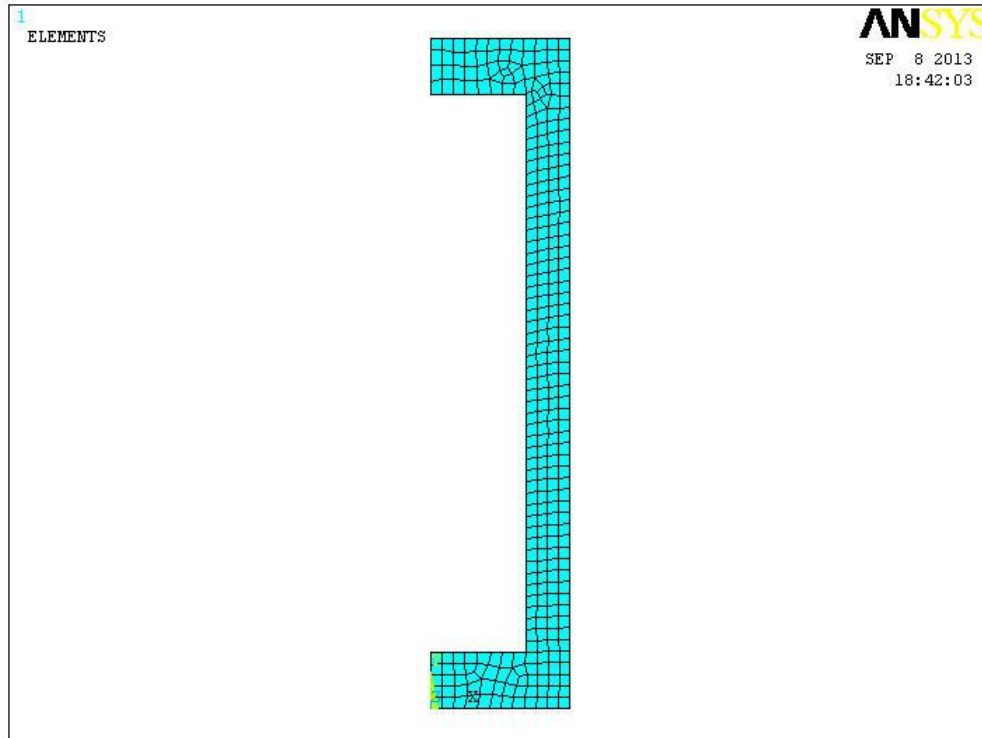
### MINIMUM VALUES

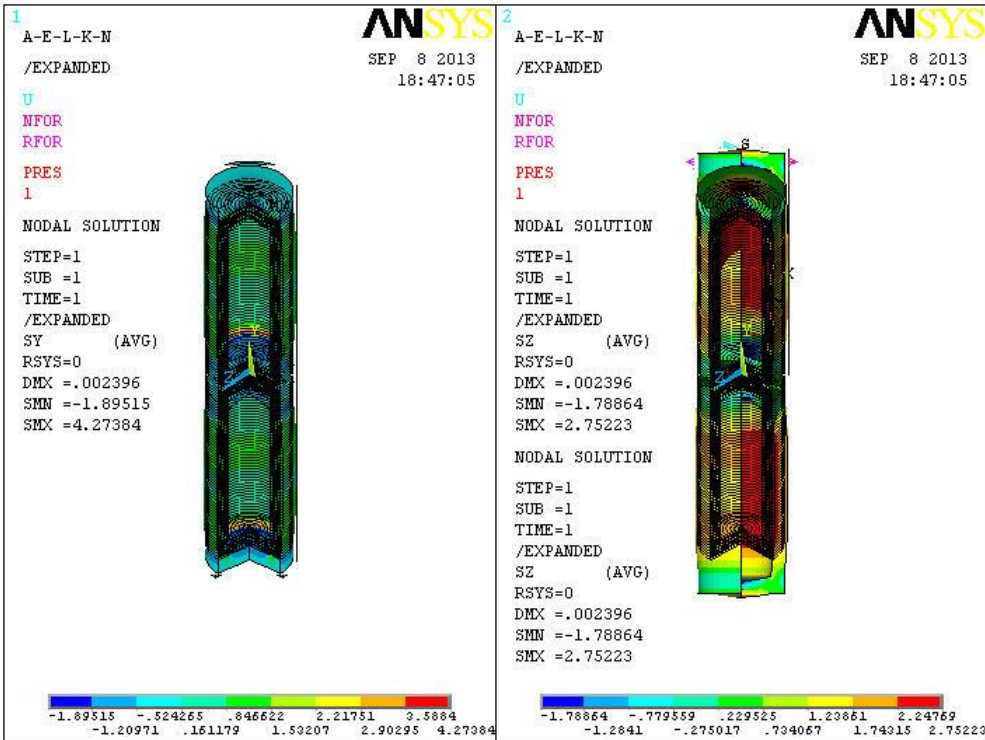
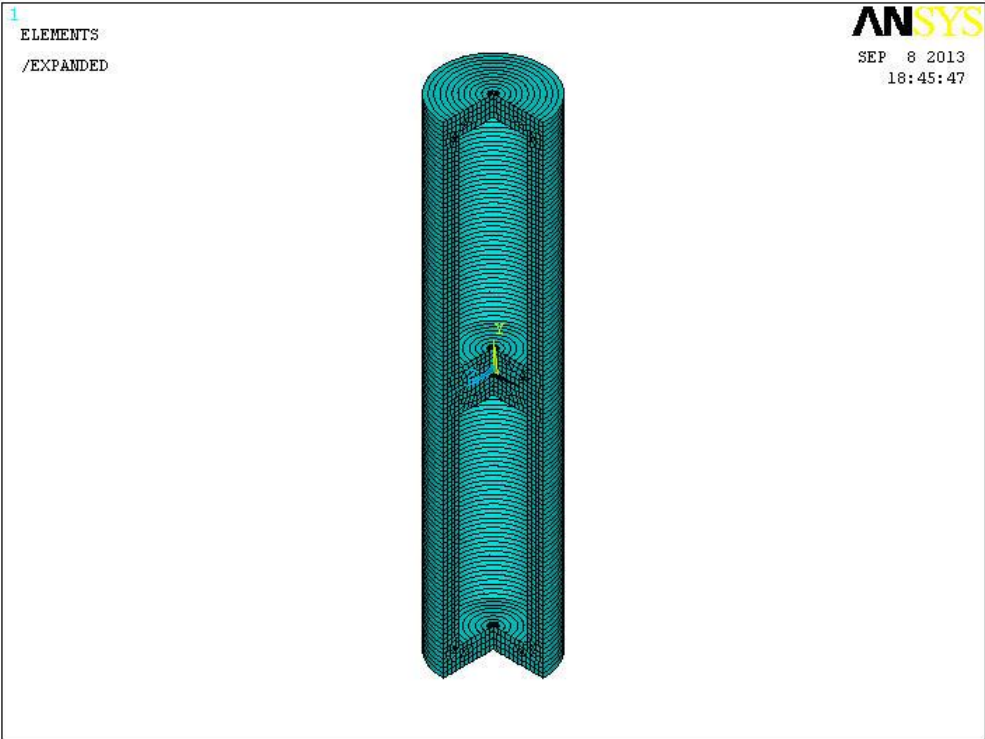
NODE	125	134	150	572	572
VALUE	0.0000	-55.062	-200.56	9.7952	8.4946

### MAXIMUM VALUES

NODE	126	552	1	126	126
VALUE	661.98	113.43	0.0000	661.98	659.65

**Problem 2:** A long cylindrical pressure vessel of mean diameter  $d$  and wall thickness  $t$  has closed ends and is subjected to an internal pressure  $P$ . Determine the axial stress  $\sigma_y$  and the hoop stress  $\sigma_z$  in the vessel at the mid thickness of the wall.





# Solution

LOAD STEP= 1 SUBSTEP= 1  
TIME= 1.0000 LOAD CASE= 0  
NODAL RESULTS ARE FOR MATERIAL 1

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

NODE	SX	SY	SZ	SXY	SYZ	SXZ
1084	-0.35231	0.86158	2.0846	0.14558E-02	0.0000	0.0000
1085	-0.35234	0.86163	2.0841	-0.27677E-03	0.0000	0.0000
1086	-0.35194	0.86157	2.0848	-0.20478E-02	0.0000	0.0000
1087	0.58123	0.74798	0.54742	-0.16055	0.0000	0.0000

MINIMUM VALUES  
NODE 193 290 5 6 1 1  
VALUE -2.2292 -1.8952 -1.7886 -2.7841 0.0000 0.0000

MAXIMUM VALUES  
NODE 6 6 255 7 1 1  
VALUE 3.4745 4.2738 2.7522 2.6033 0.0000 0.0000

LOAD STEP= 1 SUBSTEP= 1  
TIME= 1.0000 LOAD CASE= 0  
NODAL RESULTS ARE FOR MATERIAL 1

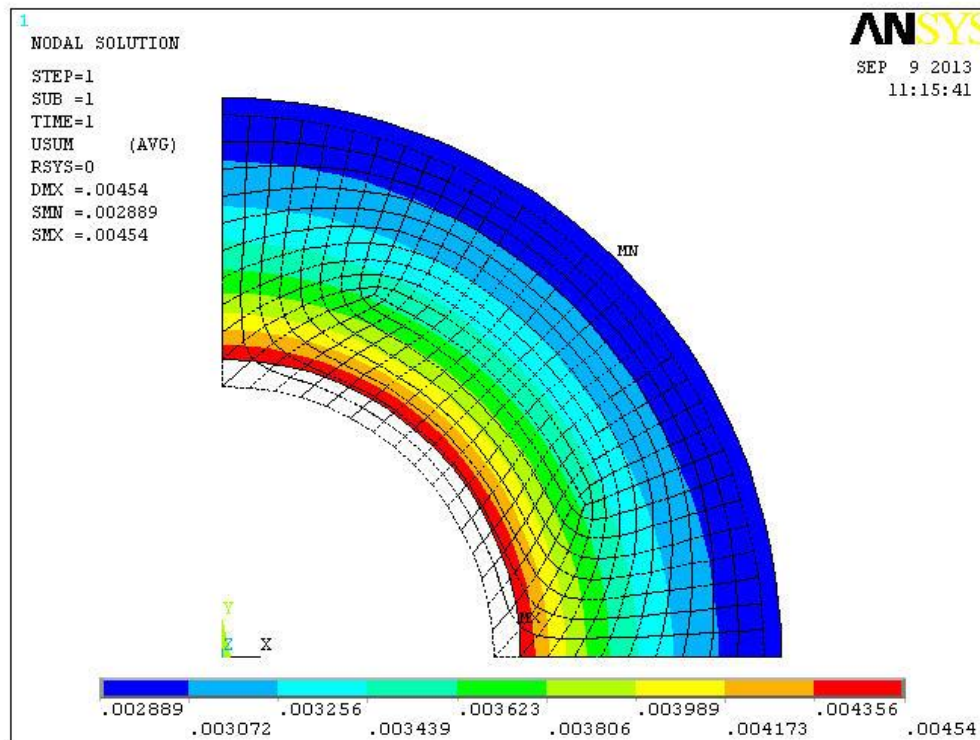
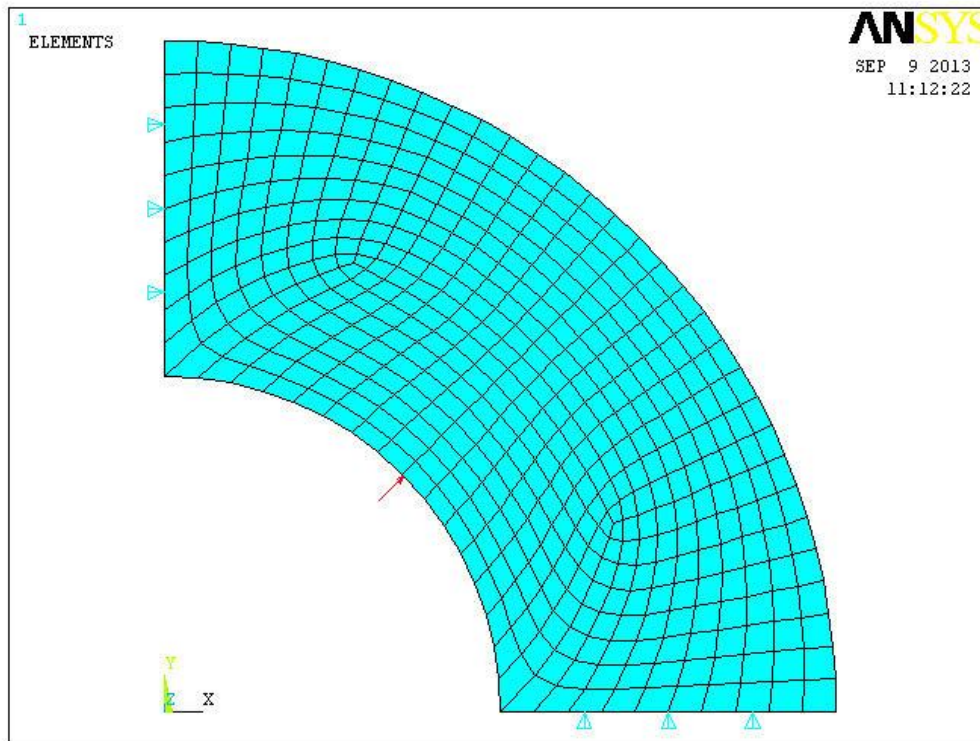
THE FOLLOWING X,Y,Z VALUES ARE IN GLOBAL COORDINATES

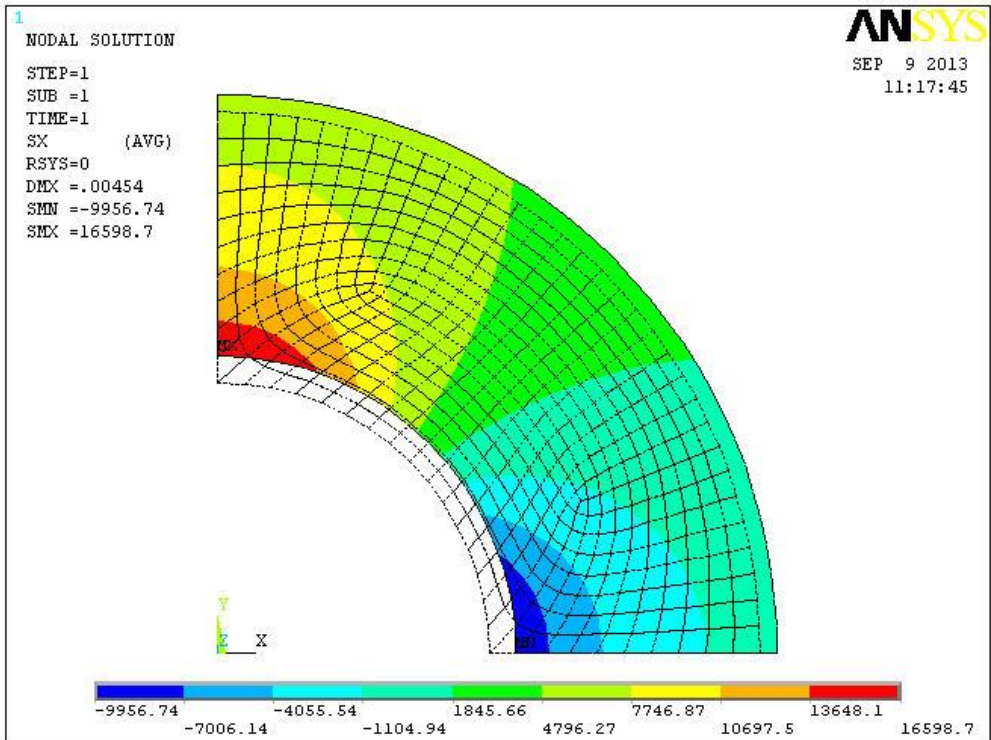
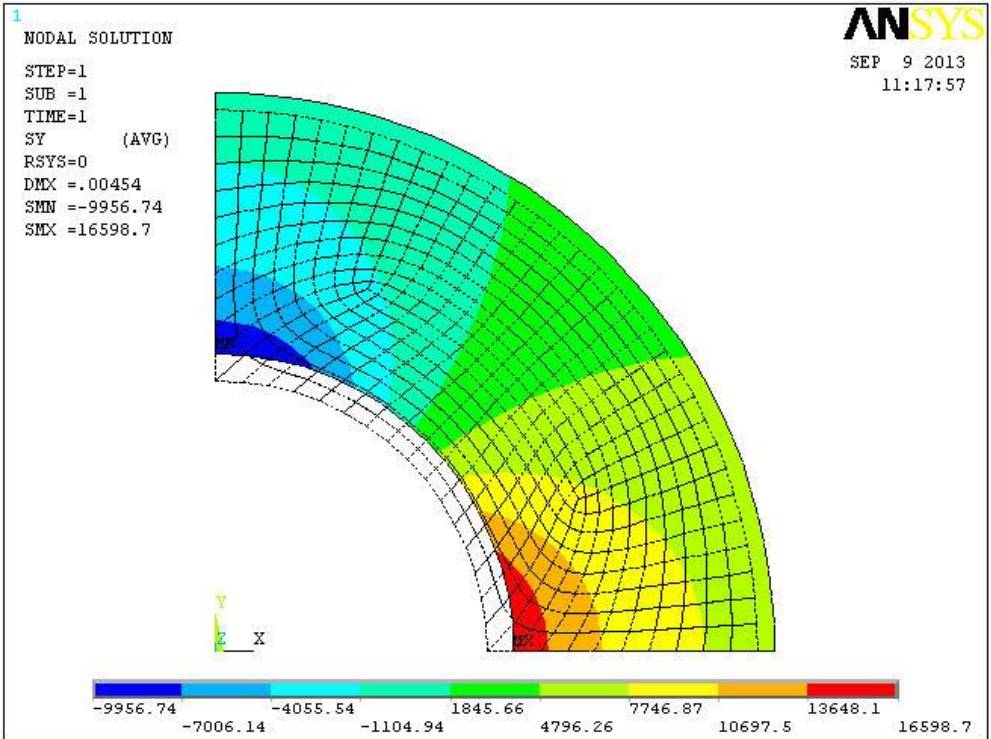
NODE	SX	SY	SZ	SXY	SYZ	SXZ
1084	-0.35231	0.86158	2.0846	0.14558E-02	0.0000	0.0000
1085	-0.35234	0.86163	2.0841	-0.27677E-03	0.0000	0.0000
1086	-0.35194	0.86157	2.0848	-0.20478E-02	0.0000	0.0000
1087	0.58123	0.74798	0.54742	-0.16055	0.0000	0.0000

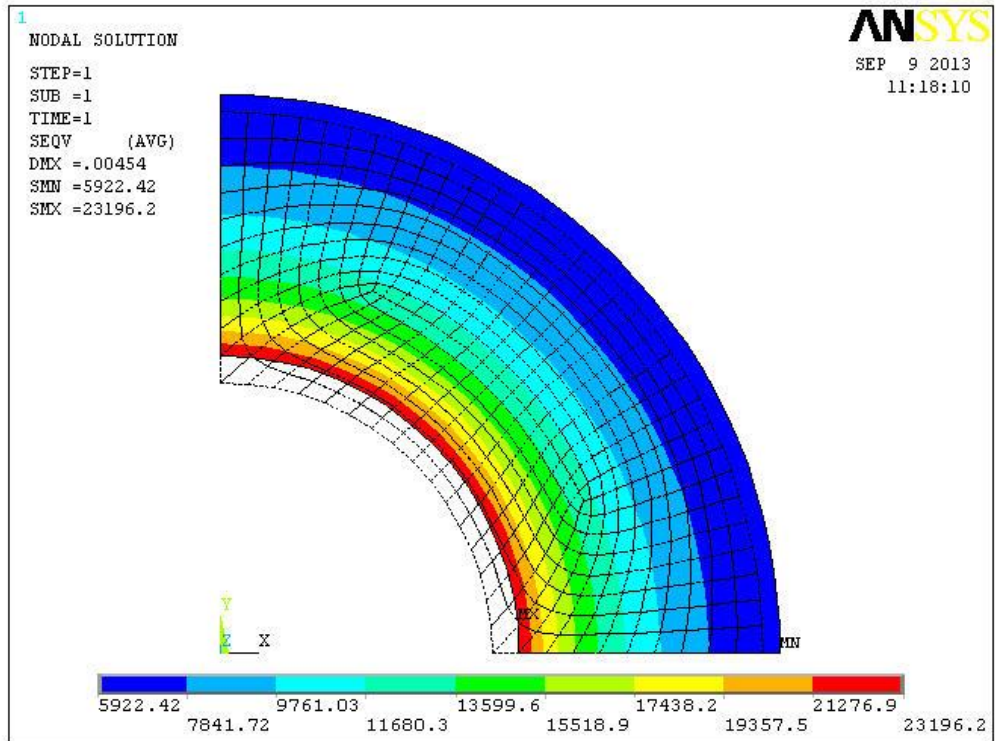
MINIMUM VALUES  
NODE 193 290 5 6 1 1  
VALUE -2.2292 -1.8952 -1.7886 -2.7841 0.0000 0.0000

MAXIMUM VALUES  
NODE 6 6 255 7 1 1  
VALUE 3.4745 4.2738 2.7522 2.6033 0.0000 0.0000

**Problem 3:** A long cylinder subjected to uniform internal pressure (plane strain condition).  $E=2.1 \times 10^7 \text{ N/cm}^2$ ,  $1/m=0.3$ . calculate the max deflection and radial stresses induced in the pressure vessel







## Solution

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

### MAXIMUM ABSOLUTE VALUES

NODE	86	66	0	88
VALUE	0.45396E-02	0.45396E-02	0.0000	0.45398E-02

### MINIMUM VALUES

NODE	86	66	66	102	1	1
VALUE	-9956.7	-9956.7	1992.6	-13357.	0.0000	0.0000

### MAXIMUM VALUES

NODE	66	86	84	1	1	1
VALUE	16599.	16599.	2005.1	-0.13221	0.0000	0.0000

LOAD STEP= 1 SUBSTEP= 1  
 TIME= 1.0000 LOAD CASE= 0  
 NODAL RESULTS ARE FOR MATERIAL 1

### MINIMUM VALUES



NODE	86	66	66	102	1	1	
VALUE	-9956.7	-9956.7	1992.6	-13357.	0.0000	0.0000	

MAXIMUM VALUES

NODE	66	86	84	1	1	1	
VALUE	16599.	16599.	2005.1	-0.13221	0.0000	0.0000	

LOAD STEP= 1 SUBSTEP= 1  
TIME= 1.0000 LOAD CASE= 0  
NODAL RESULTS ARE FOR MATERIAL 1

NODE	S1	S2	S3	SINT	SEQV
1403	6983.0	2000.0	-316.31	7299.3	6460.5
1405	7003.8	2000.0	-337.07	7340.8	6495.7

MINIMUM VALUES

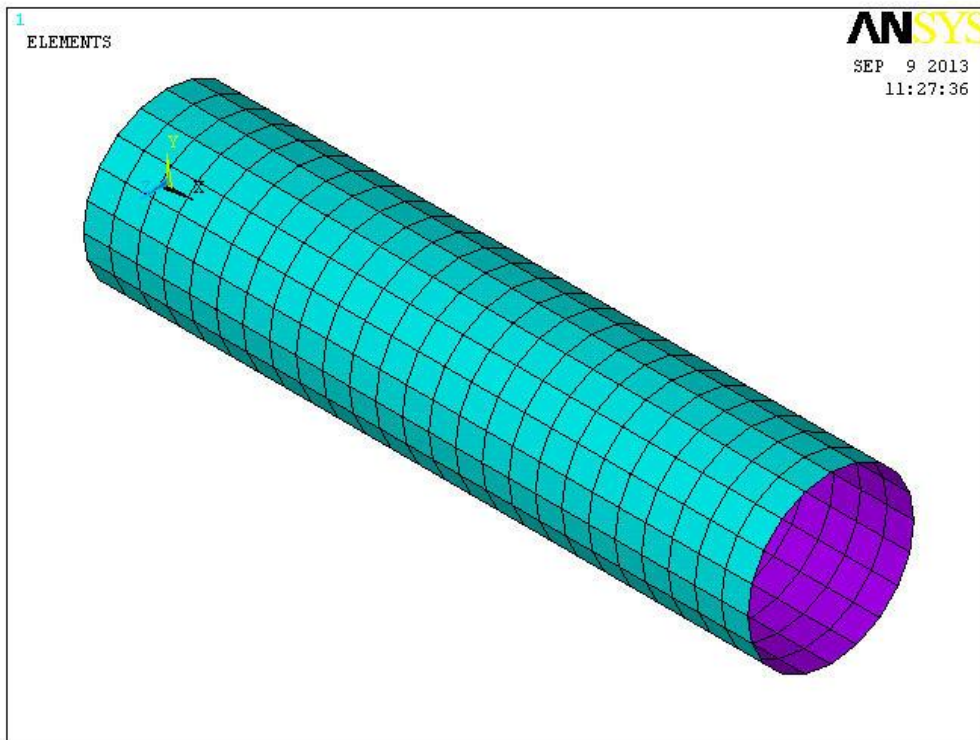
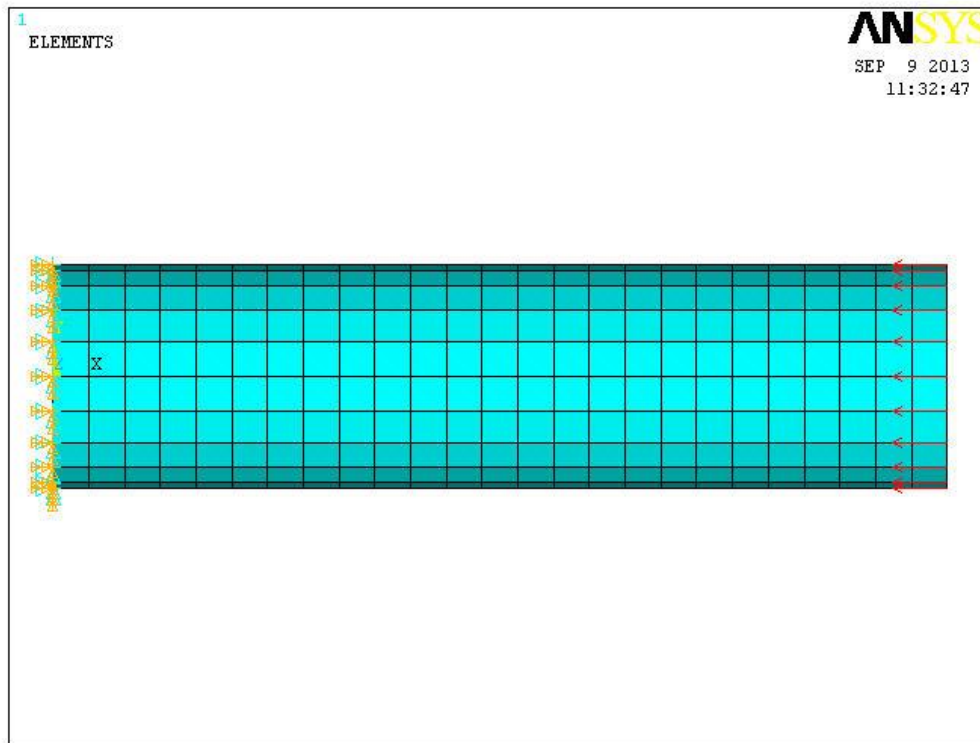
NODE	1	66	88	1	1	
VALUE	6664.9	1992.6	-10037.	6663.1	5922.4	

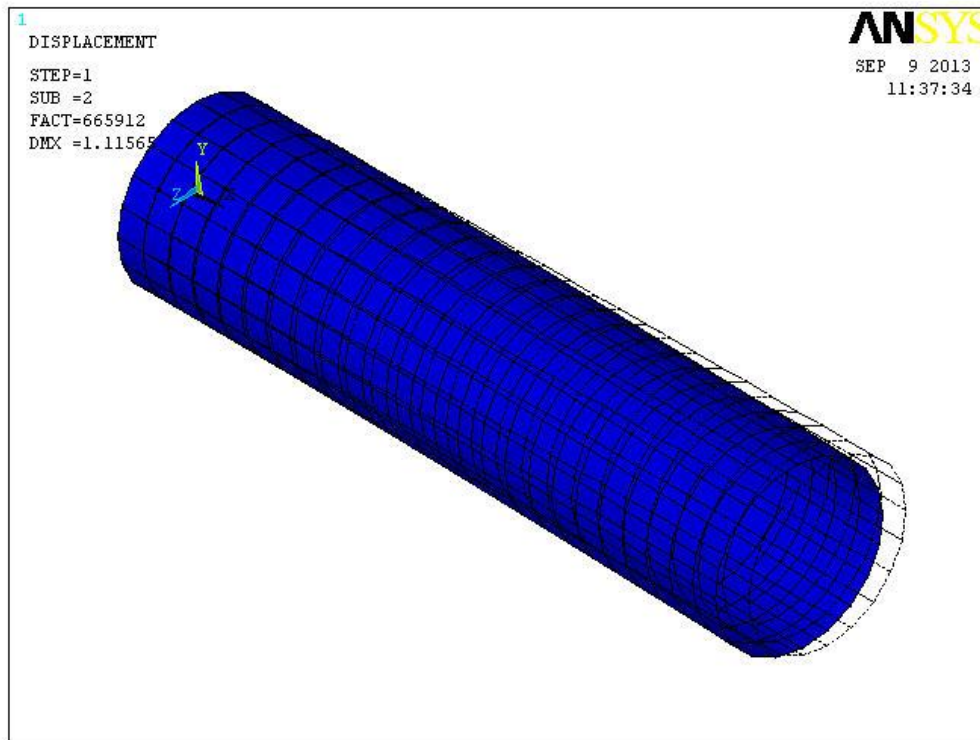
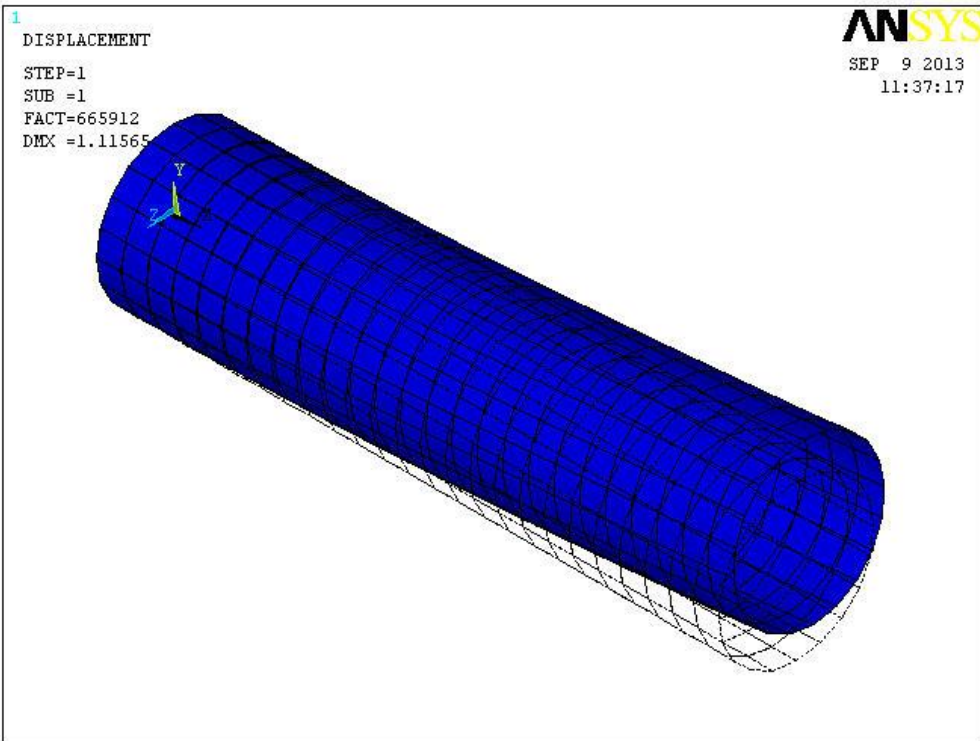
MAXIMUM VALUES

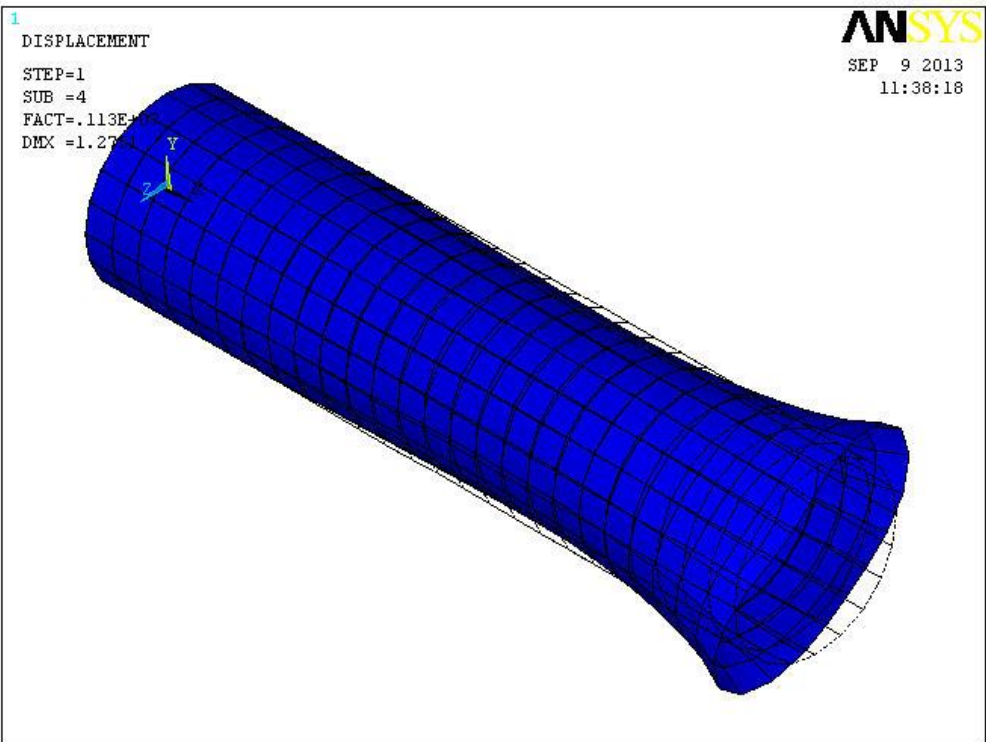
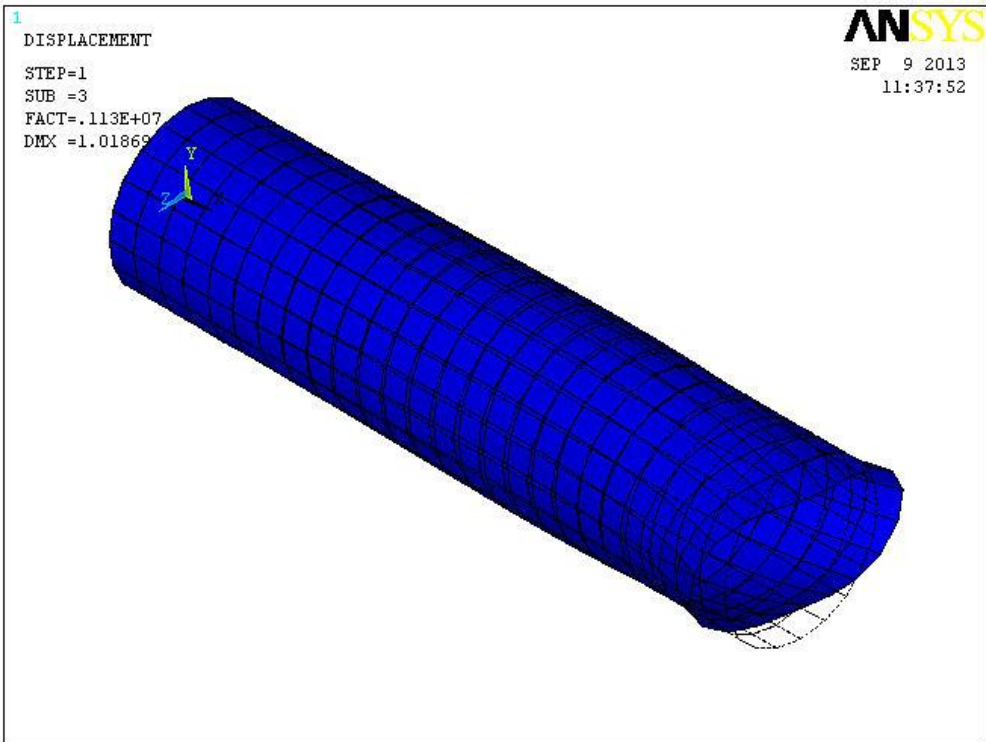
NODE	88	84	2	88	88	
VALUE	16704.	2005.1	1.8681	26740.	23196.	

## EXERCISE-4

### Problem 1: Buckling analysis of open cylindrical shell.





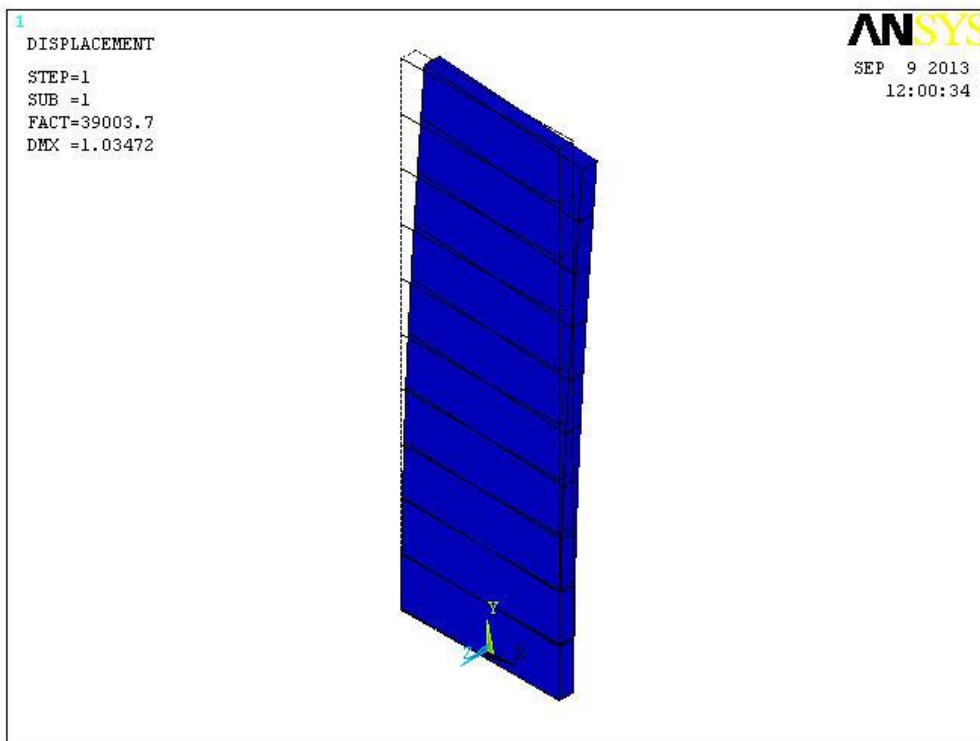
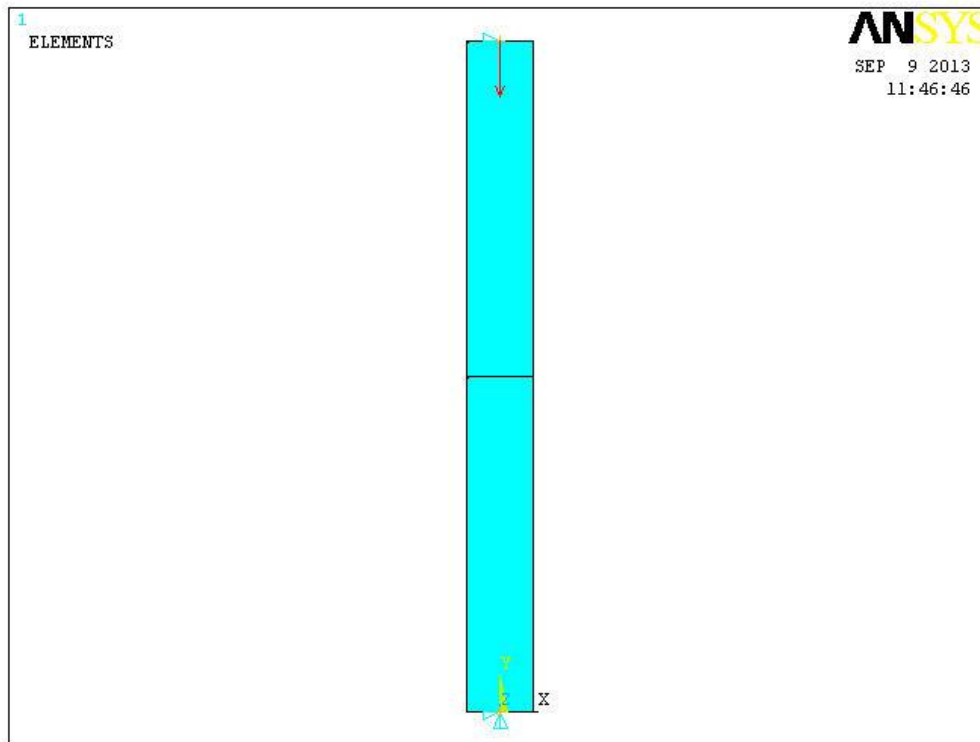


## Solution

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

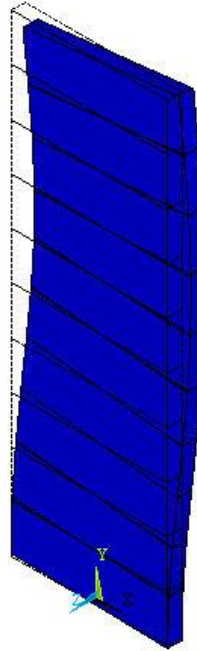
SET	TIME/FREQ	LOAD	STEP	SUBSTEP	CUMULATIVE
1	0.66591E+06	1	1	1	
2	0.66591E+06	1	2	2	
3	0.11254E+07	1	3	3	
4	0.11254E+07	1	4	4	

## Problem 2: Harmonic Analysis of a rectangular plate



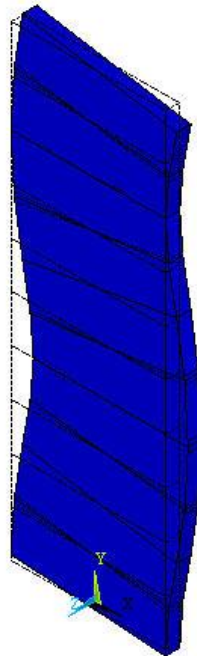
1  
DISPLACEMENT  
STEP=1  
SUB =2  
FACT=245010  
DMX =1.0116

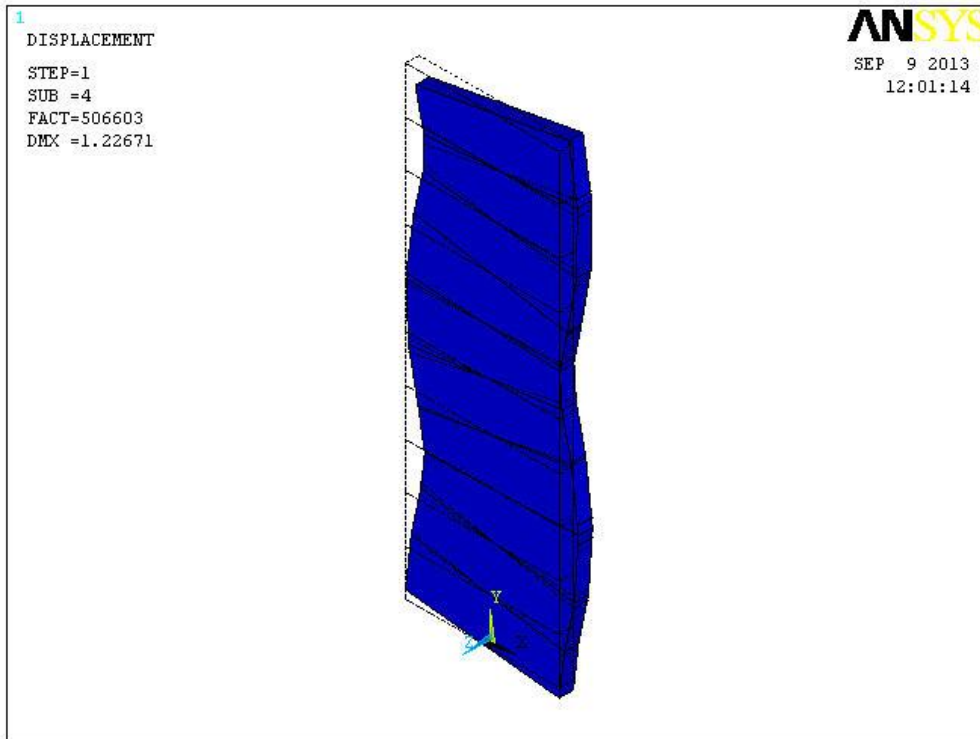
**ANSYS**  
SEP 9 2013  
12:00:44



1  
DISPLACEMENT  
STEP=1  
SUB =3  
FACT=414384  
DMX =1.04218

**ANSYS**  
SEP 9 2013  
12:00:56





## Solution

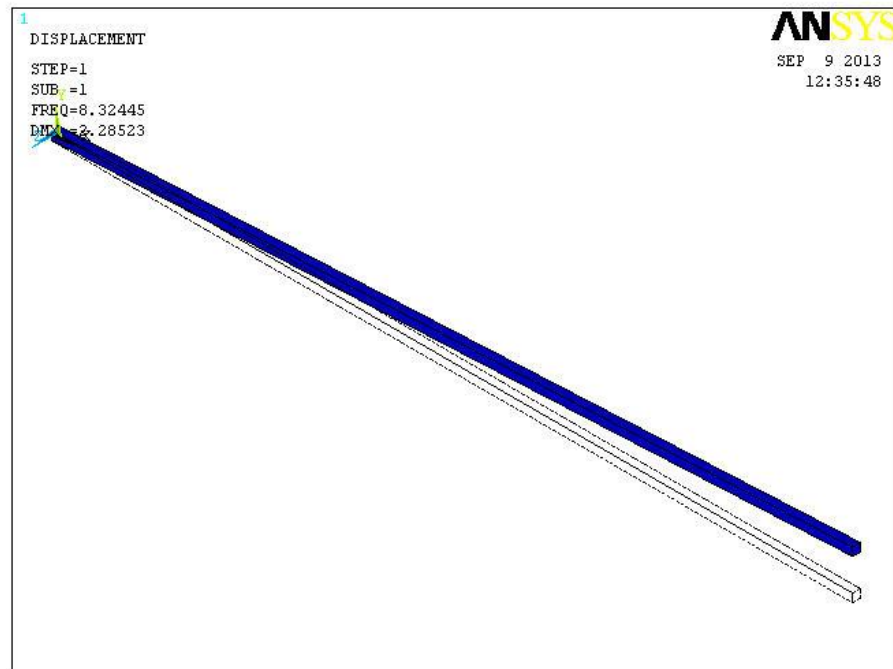
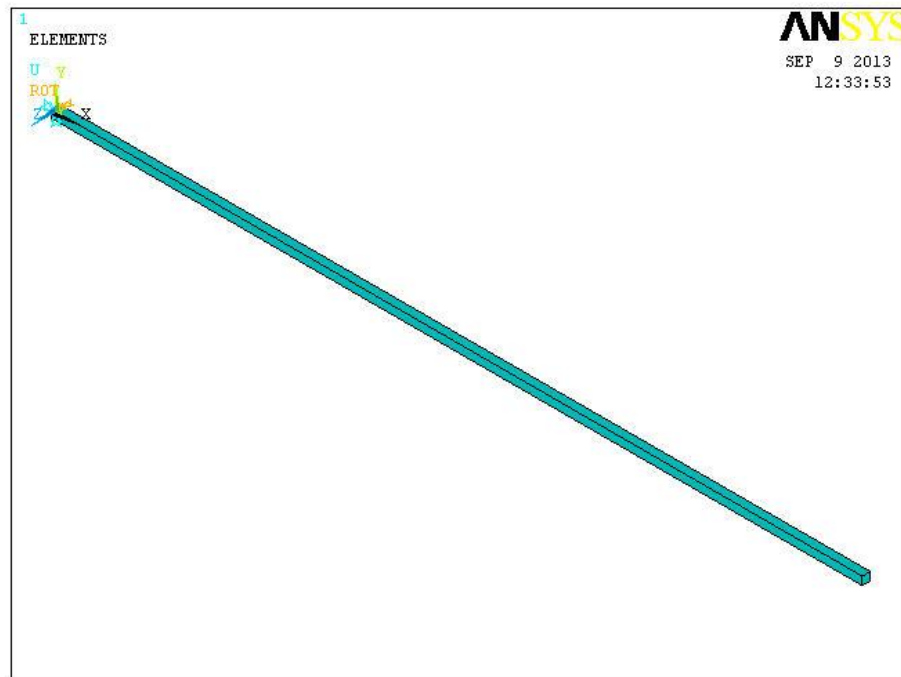
\*\*\*\*\* INDEX OF DATA SETS ON RESULTS FILE \*\*\*\*\*

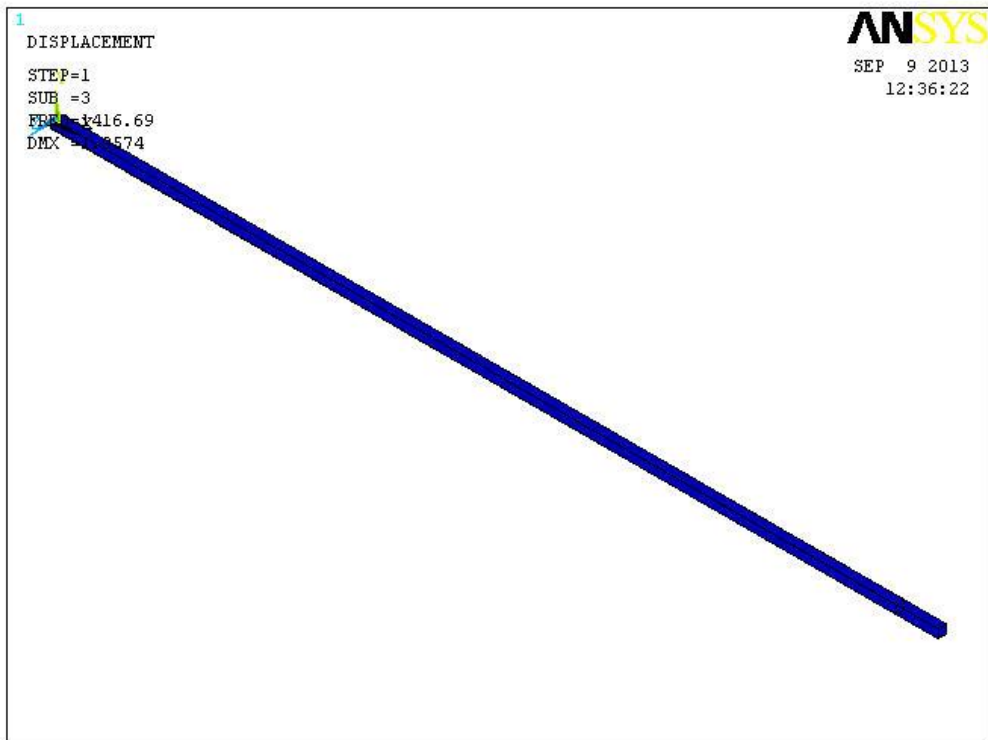
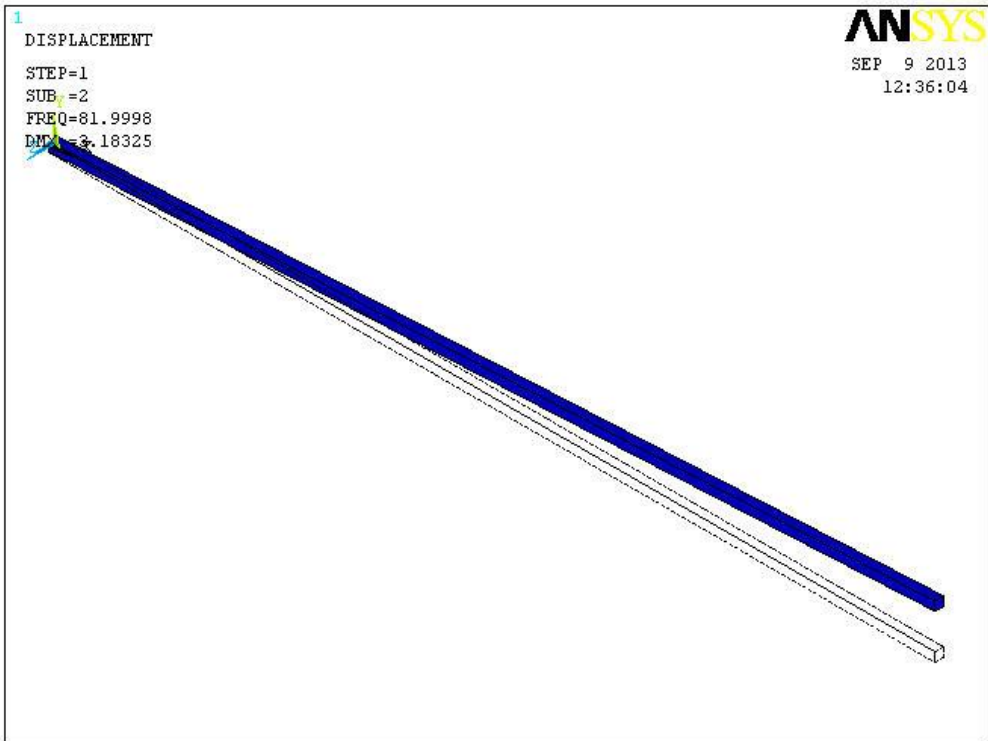
SET	TIME/FREQ	LOAD	STEP	SUBSTEP	CUMULATIVE
1	0.66591E+06	1	1	1	
2	0.66591E+06	1	2	2	
3	0.11254E+07	1	3	3	
4	0.11254E+07	1	4	4	



# EXERCISE-5

## Problem 1: Modal analysis of Beam





## Solution

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

SET	TIME/FREQ	LOAD	STEP	SUBSTEP	CUMULATIVE
1	8.3244	1	1	1	
2	82.000	1	2	1	
3	1416.7	1	3	1	

## Problem 2: Harmonic analysis of a Beam

