

ANSYS TUTORIAL – 2-D Frame Analysis

ANSYS Release 7.0

Dr. A.-V. Phan, University of South Alabama

1 Problem Description

Reference: ‘A first Course in the Finite Element Method’ by Daryl L. Logan, 3rd Edition, p. 208.

The bar element 2 is used to stiffen the cantilever beam 1, as shown in Fig. 1(a). All members are made of steel (Young’s modulus $E = 210$ GPa). The bar has a cross-sectional area of $A = 1 \times 10^{-3}$ m². The cross-sectional area, principal moment of inertia and length for the beam are $A = 2 \times 10^{-3}$ m², $I_z = 5 \times 10^{-5}$ m⁴ and $L = 3$ m, respectively. Determine the nodal displacements, element forces and stress in each element.

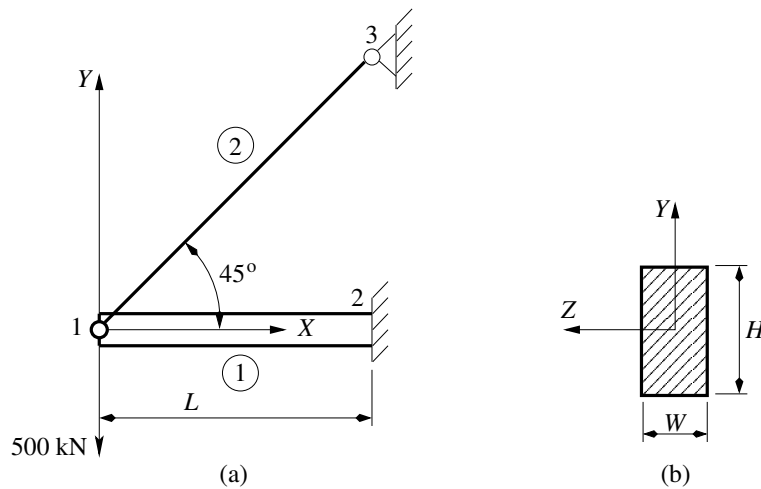


Figure 1: A stiffened beam

2 Listing Stress Results in ANSYS

- To inquire the axial stress in a bar element, use ‘LS, 1’.
- For a two-node beam element, ‘NMISC, 1’ and ‘NMISC, 3’ are associated with the maximum bending stresses at the starting node (I) and ending node (J), respectively; and ‘NMISC, 2’ and ‘NMISC, 4’ the minimum bending stresses at nodes I and J , respectively (see Fig. 2)

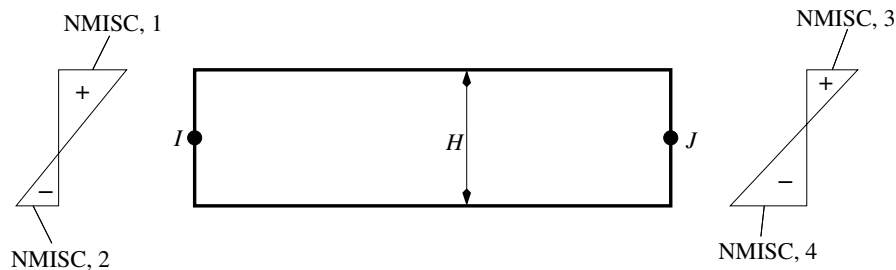


Figure 2: Variables NMISC for a beam element

3 Data Preparation

- Units to be used: Length (m), Force (N)
- Beam's height: To determine the stress distribution in the beam, it is necessary to know the height of the beam. Assume that the beam has a rectangular cross section of width W and height H as shown in Fig. 1(b). From

$$A = WH \text{ and } I_z = \frac{WH^3}{12}$$

it is found that

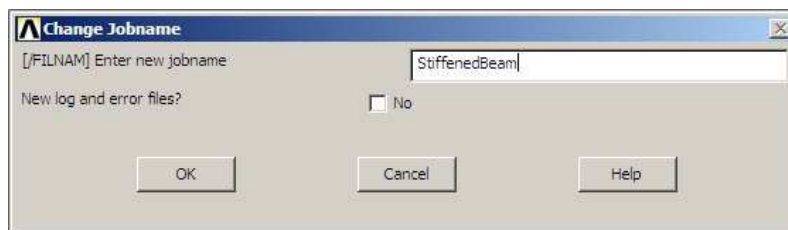
$$H = \sqrt{\frac{12 I_z}{A}} = \sqrt{\frac{(12)(5 \times 10^{-5})}{2 \times 10^{-3}}} = 0.548 \text{ m}$$

4 Preprocessing

1. Give the Job a Name

Utility Menu > File > Change Jobname ...

The following window comes up. Enter a name, e.g. 'StiffenedBeam', and click on **OK**.

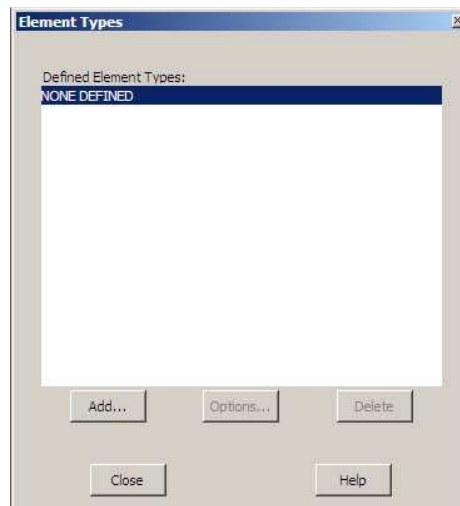


2. Define Element Types

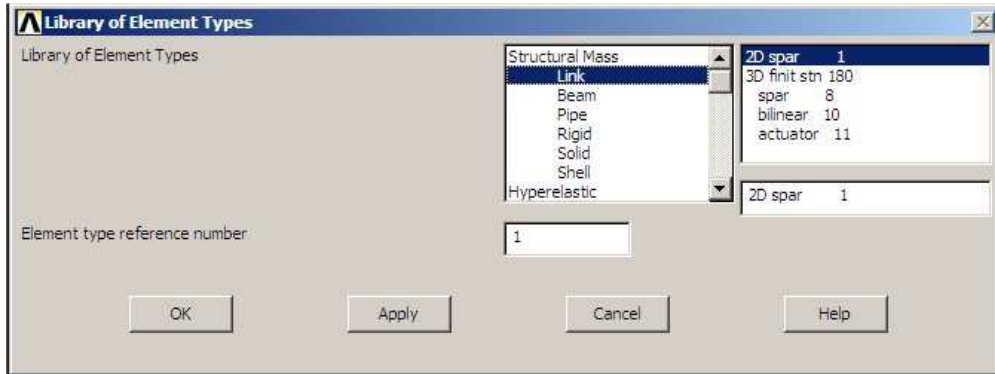
- (a) Bar/Truss Element

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

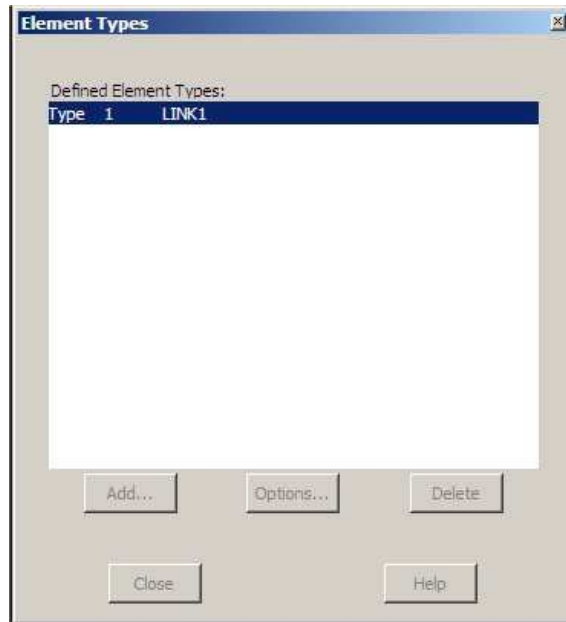
- This brings up the 'Element Types' window. Click on the **Add...** button.



- The 'Library of Element Types' window appears. Highlight 'Link' and '2D spar 1' as shown. Click on **Apply**.

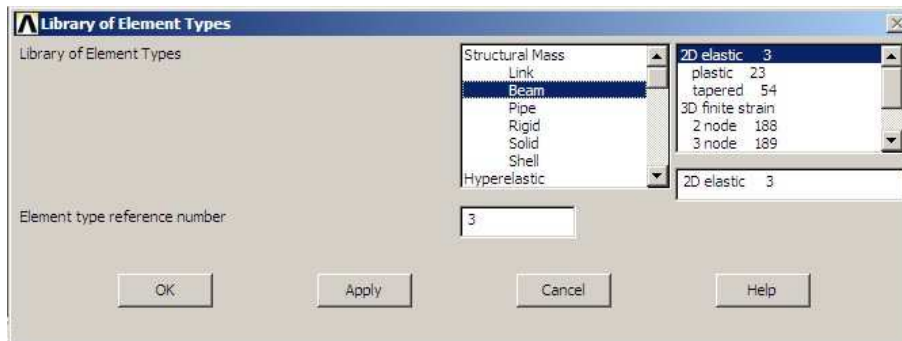


- You should see 'Type 1 LINK1' in the 'Element Types' window as follows:

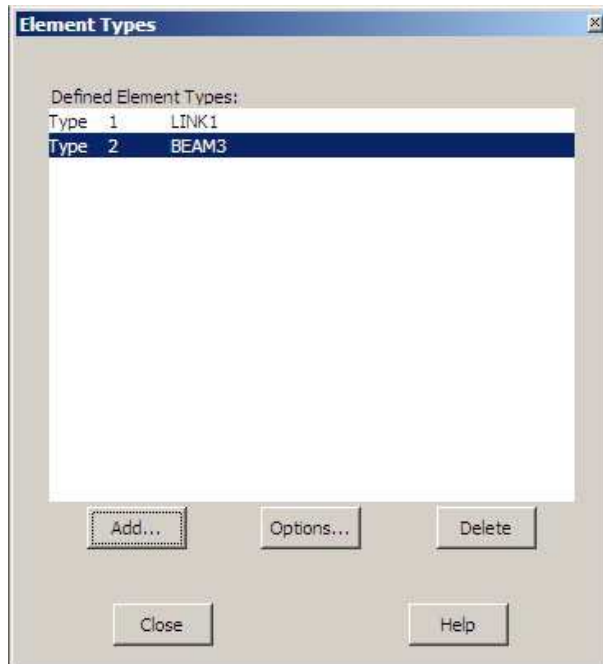


(b) Beam Element

- Highlight 'Beam' and '2D elastic 3' as shown. Click **OK**.



- The 'Element Types' window appears as shown. Click on the **Close** button.



3. Define Real Constants

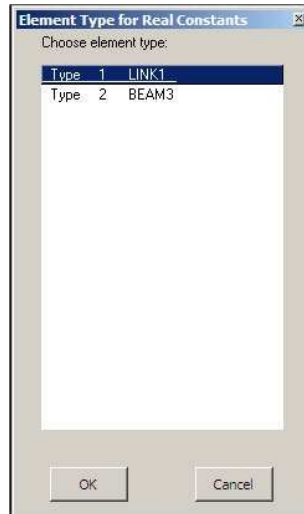
(a) Bar/Truss Element

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

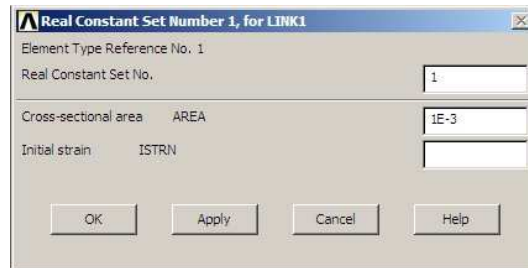
- This brings up the 'Real Constants' window. Click on the **Add...** button.



- The 'Element Type for Real Constants' window appears. Highlight 'Type 1 LINK1' as shown and click on **OK**.



- In the 'Real Constant Set Number 1, for LINK1' window that opens, enter '1E-3' for the 'AREA' field as shown,



- Click on **OK**. You should see the 'Real Constants' window as follows:



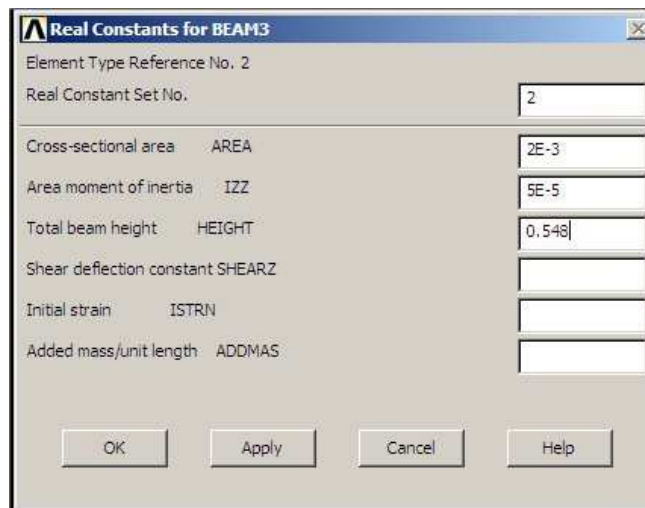
(b) Beam Element

- In the 'Real Constants' window, click on **Add**

- This brings up the ‘Element Type for Real Constants’ window as shown. Highlight ‘Type 2 BEAM3’ and click **OK**.



- In the ‘Real Constants for BEAM3’ window that appears, enter ‘2E-3’, ‘5E-5’ and ‘0.548’ for ‘AREA’, ‘IZZ’ and ‘HEIGHT’, respectively. Click on **OK**.

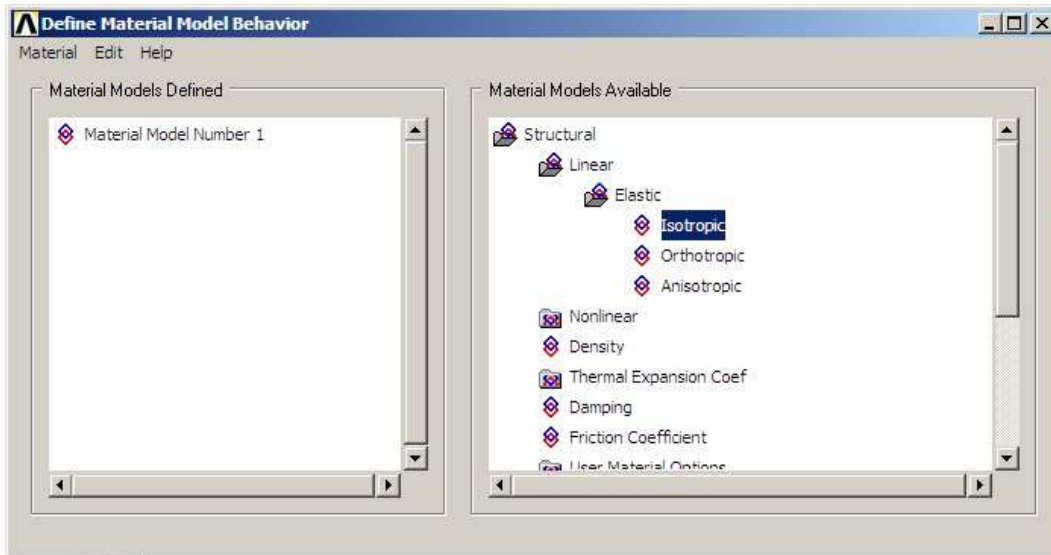


- Click on the **Close** button in the ‘Real Constants’ window.

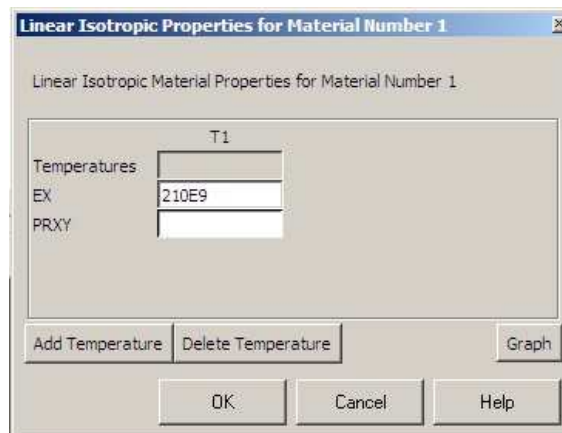
4. Define Material Properties

Main Menu > Preprocessor > Material Props > Material Models

- In the right side of the ‘Define Material Model Behavior’ window that opens, double click on ‘Structural’, then ‘Linear’, then ‘Elastic’, then finally ‘Isotropic’.



- The following window comes up. Enter in value for the Young's modulus of 210×10^9 Pa (EX = 210E9), then click **OK**.



- Click on **OK** in the 'Note' window that appears. Then close the 'Define Material Model Behavior' window.

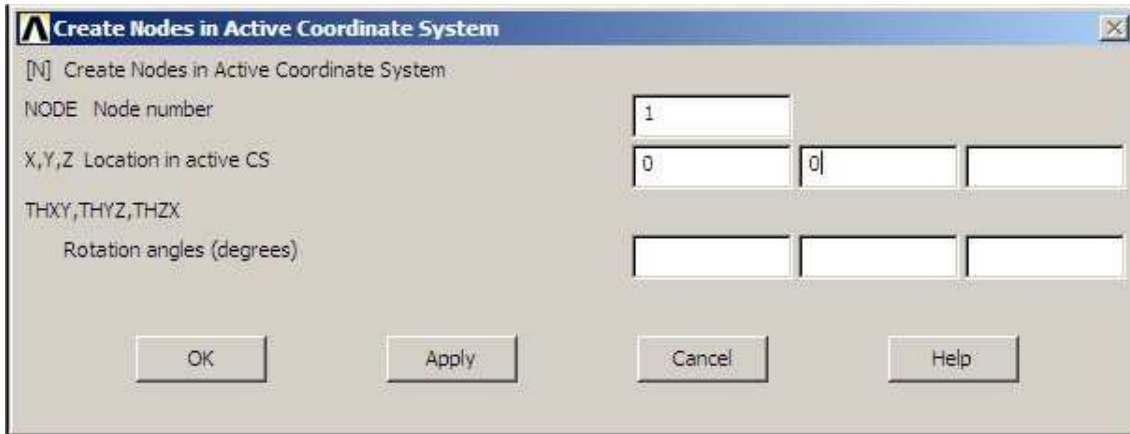
5. Define Nodes

We are going to create 3 nodes given in the following table:

Node #	X	Y
1	0	0
2	3	0
3	3	3

Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS

- To create node #1, enter the following data in the 'Create Nodes in Active Coordinate System' window that comes up. Click on **Apply**.



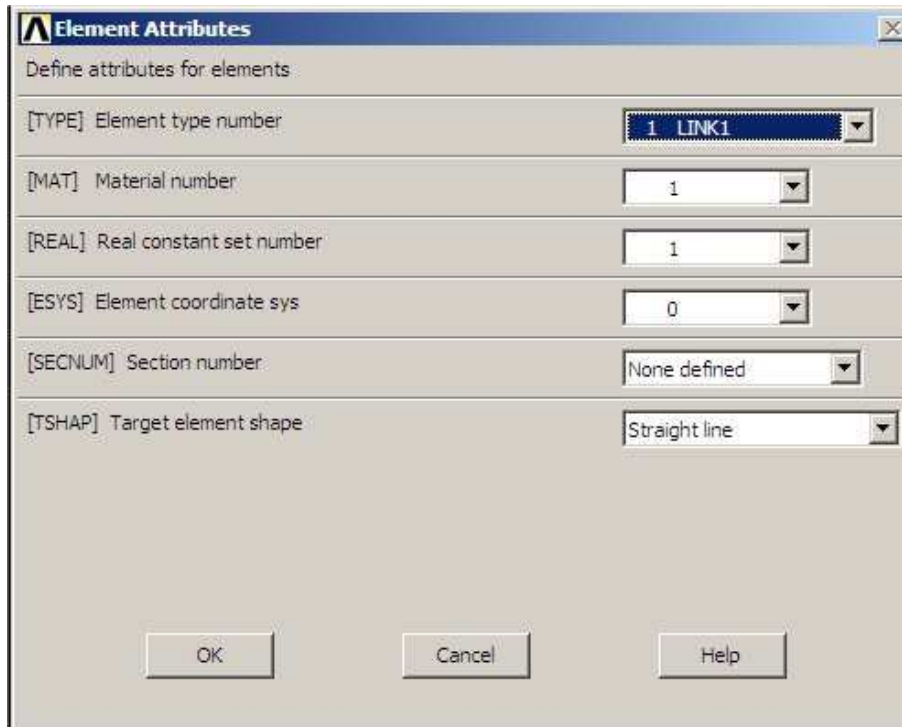
- Repeat the above step for nodes #2 and #3. Note that you must click on **OK** instead of **Apply** after entering data for the final node.

6. Define Elements

(a) Define Bar Element

Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes

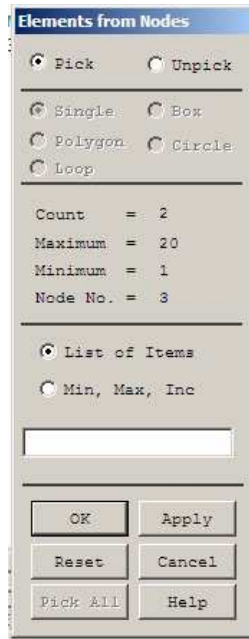
- This brings up the 'Element Attributes' window. Click on **OK**.



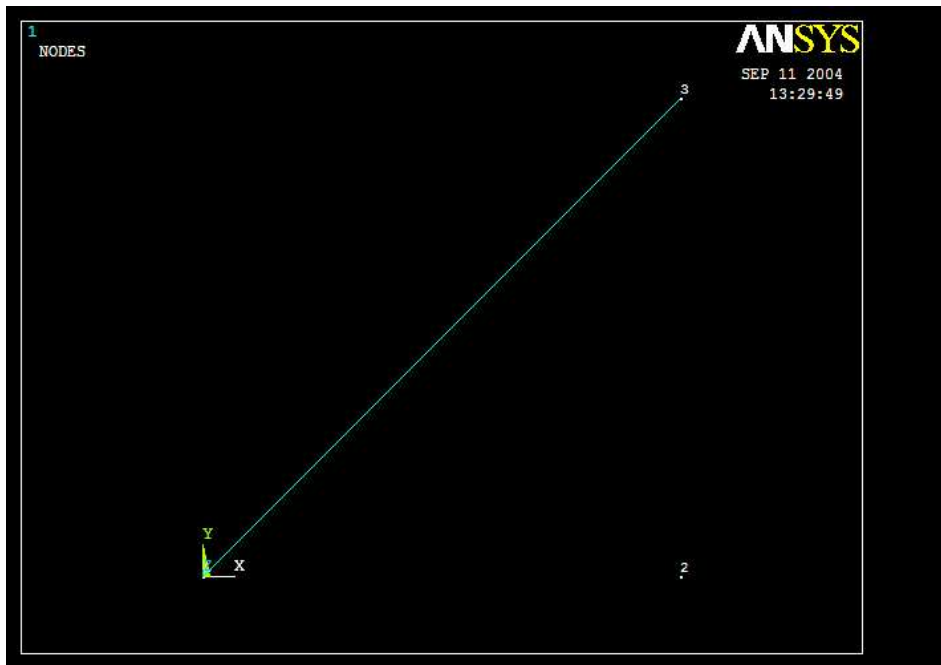
(b) Create Bar Element

Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru Nodes

- The 'Element from Nodes' window opens as shown,



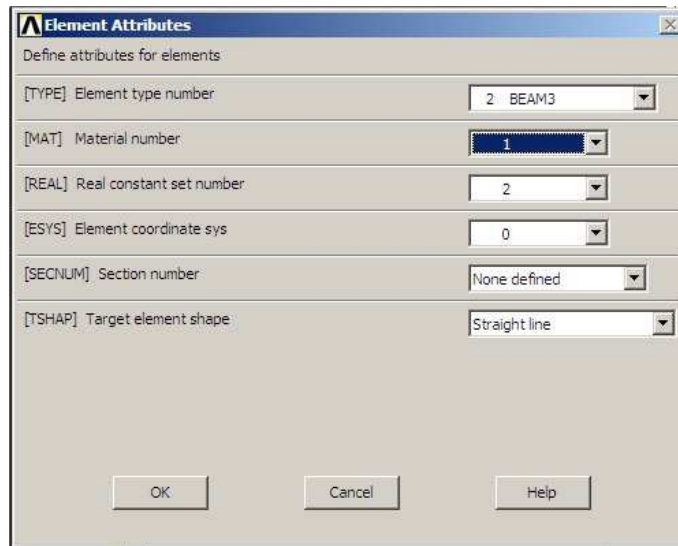
- Select node 1, then node 3.
- Click on **OK** in the 'Element from Nodes' window. Your graphics window should look like this,



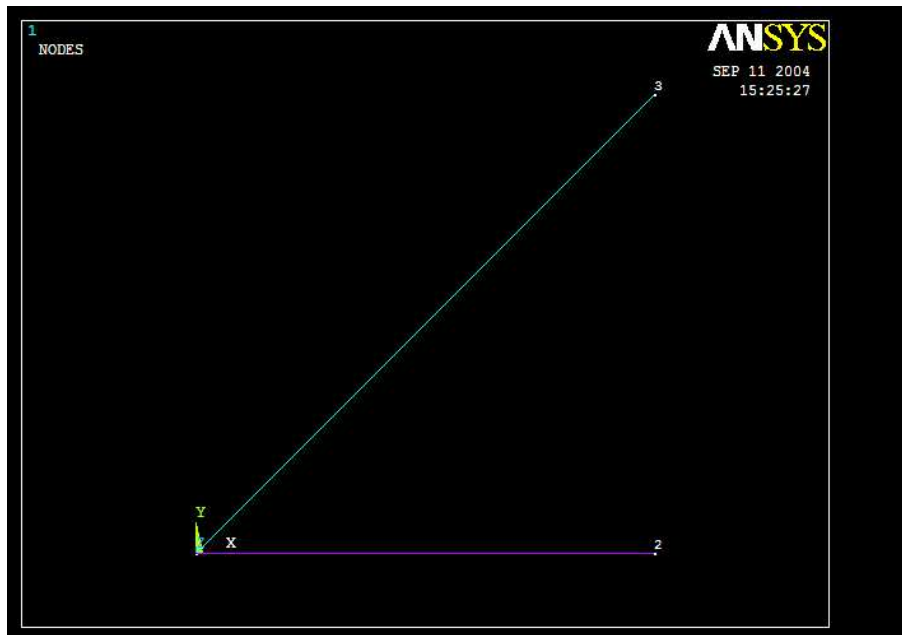
(c) Define Beam Element

Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes

- In the 'Element Attributes' window that opens, select '2 BEAM3' for 'Element type number', and '2' for 'Real constant set number' as shown,

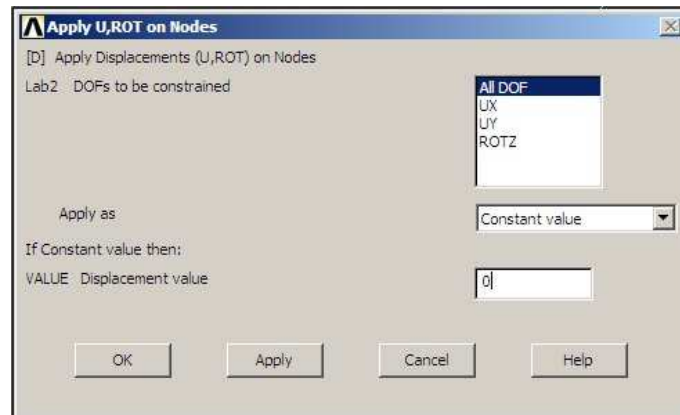


- Click on **OK**.
- (d) Create Beam Element
- Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru Nodes**
- Select node 1, then node 2.
 - Click on **OK**. Your graphics window should look like this,



7. Mesh the Model
No need because we have defined the model using nodes and elements.
8. Apply Boundary Conditions
Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes

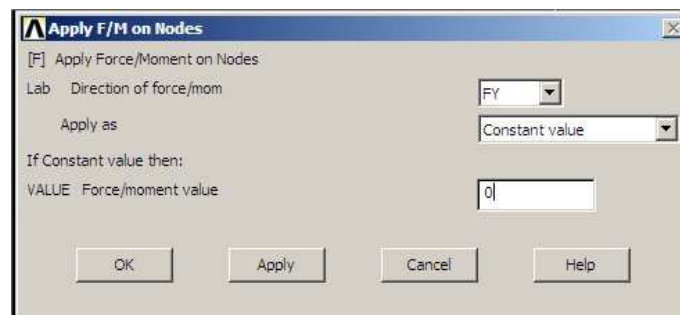
- Pick nodes 2 and 3, then click **OK** in the picking window.
- In the following window that comes up, select 'All DOF' for 'DOFs to be constrained', 'Constant value' for 'Apply as' and enter '0' for 'Displacement value', then click **OK**.



9. Apply Loads

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Nodes

- Pick node 1, then click **OK** in the picking window.
- In the following window that opens, select 'FY' for 'Direction of force/mom', 'Constant value' for 'Apply as' and enter '-5E5' for 'Force/moment value', (a force of 500,000 N in the negative direction of the Y axis), then click on **OK**.



5 Processing (Solving)

Main Menu > Solution > Analysis Type > New Analysis

- Make sure that 'Static' is selected. Click **OK**.

Main Menu > Solution > Solve > Current LS

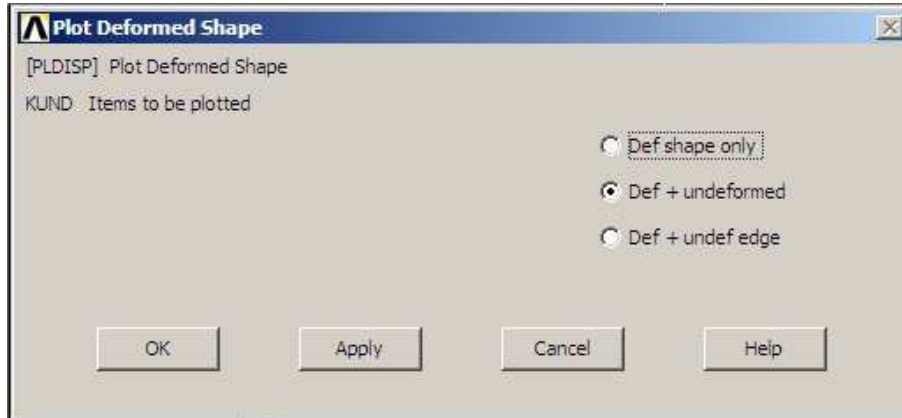
- Check your solution options listed in the '/STATUS Command' window.
- Click the **OK** button in the 'Solve Current Load Step' window.
- You should see the message 'Solution is done!' in the 'Note' window that comes up. Close the 'Note' and '/STATUS Command' windows.

6 Postprocessing

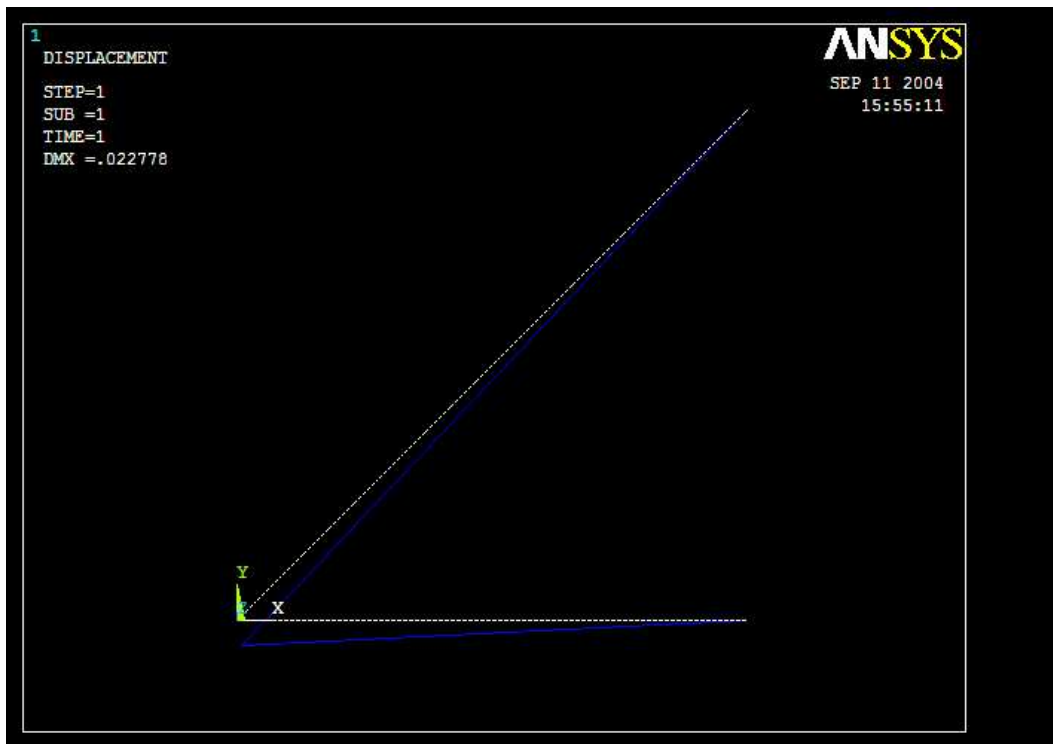
1. Plot the deformed shape

Main Menu > General Postproc > Plot Results > Deformed shape

- This brings up the 'Plot Deformed Shape' window. Select 'Def+undeformed' as shown,



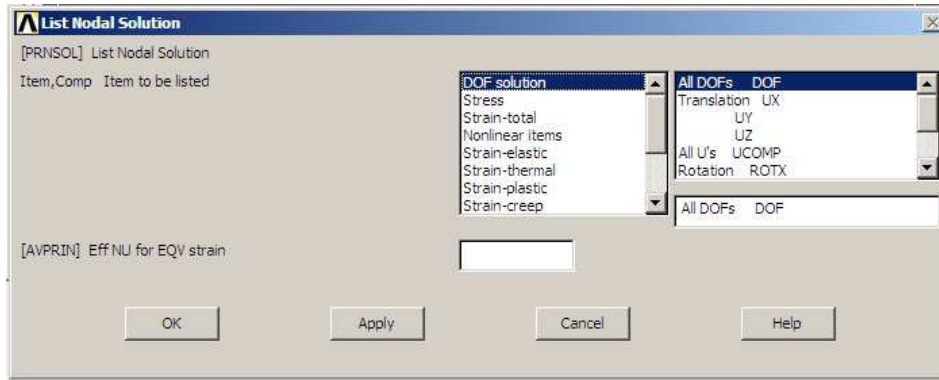
- Click on **OK**. Your ANSYS Graphics windows should look like this,



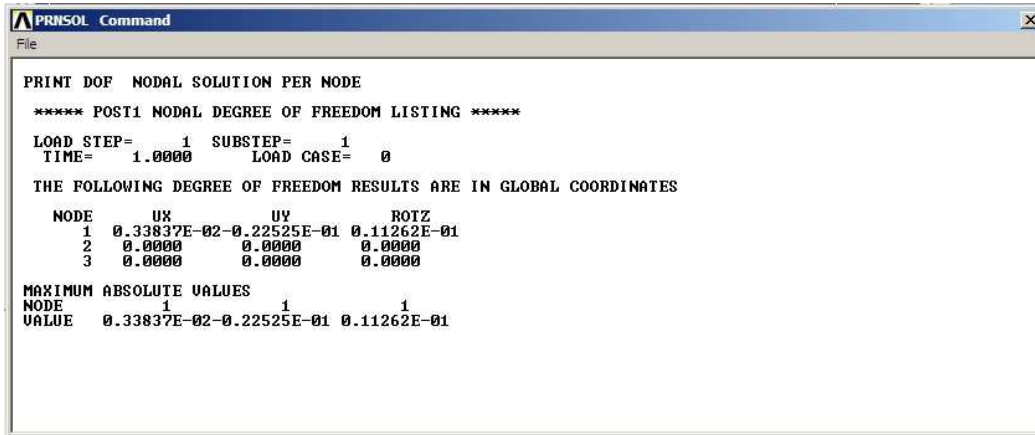
2. List Nodal Displacement Results

Main Menu > General Postproc > List Results > Nodal Solution

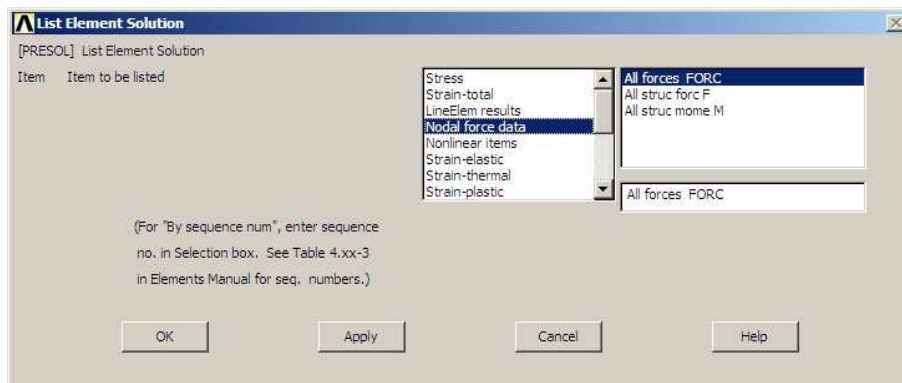
- In the 'List Nodal Solution' window that appears, select 'DOF Solution' and 'All DOFs DOF' as shown,



- Clicking on **OK** will bring up the following 'PRESOL Command' window. This window shows that the displacements in the *X* and *Y* directions, and rotation about the *Z* axis at node 1 are 0.33837×10^{-2} m, -0.22525×10^{-1} m and 0.11262×10^{-1} rad, respectively.



- Close the 'PRESOL Command' window.
3. List Nodal Force Results
Main Menu > General Postproc > List Results > Element Solution
- In the 'List Element Solution' window that opens, select 'Nodal force data' and 'All forces FORC' as shown,



- Click on **OK** to obtain the following results:

```

PRESOL Command
File
PRINT FORC ELEMENT SOLUTION PER ELEMENT
***** POST1 ELEMENT NODE TOTAL FORCE LISTING *****
LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
THE FOLLOWING X,Y,Z FORCES ARE IN GLOBAL COORDINATES

ELEM= 1 FX FV
1 0.47372E+06 0.47372E+06
3 -0.47372E+06 -0.47372E+06

ELEM= 2 FX FV MZ
1 -0.47372E+06 26279. 0.29104E-10
2 0.47372E+06 -26279. 78837.

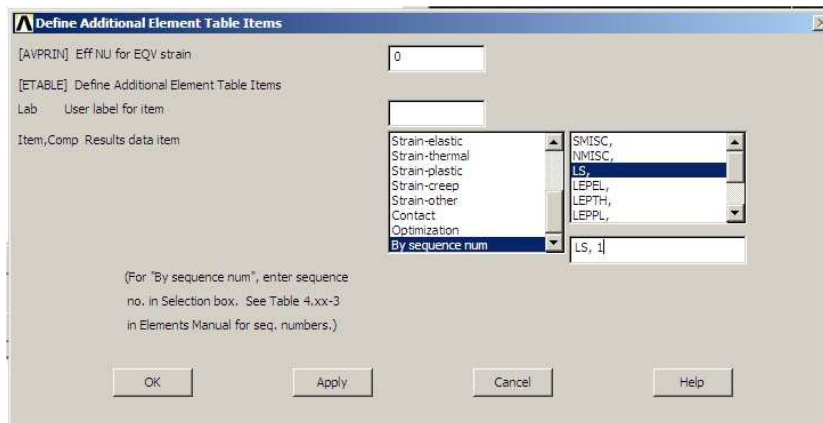
```

- Close the above window.

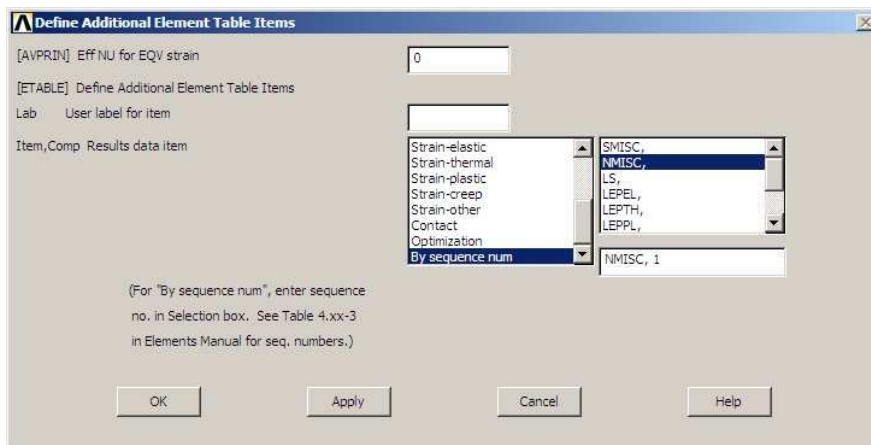
4. List Stress Results

Main Menu > General Postproc > Element Table > Define Table

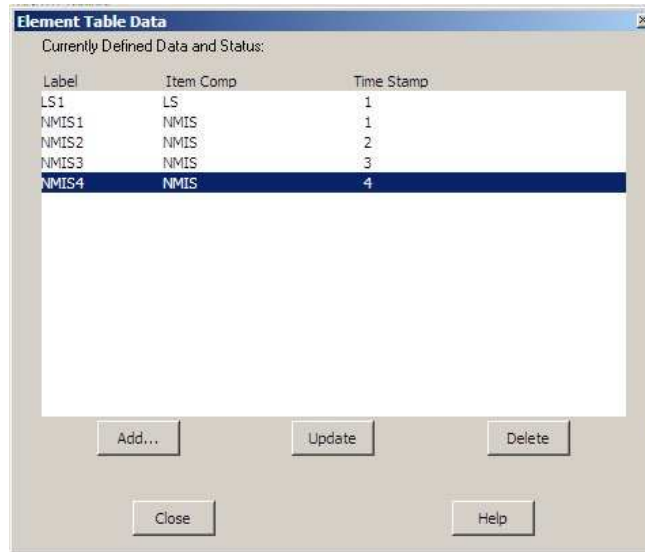
- Click on **Add ...** in the 'Element Table Data' window. In the 'Define Additional Element Table Items' window that comes up, enter sequence number 'LS, 1' as follows:



- Click on **Apply**, then enter sequence number 'NMISC, 1' as shown,



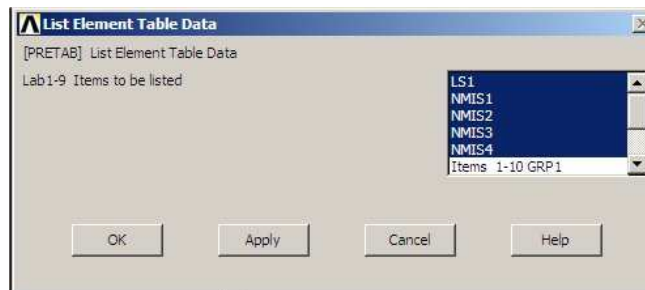
- Repeat the previous step for the sequence numbers 'NMISC, 2', 'NMISC, 3' and 'NMISC, 4'.
- Click on **OK** in the 'Define Additional Element Table Items' window. Your 'Element Table Data' window should look like this,



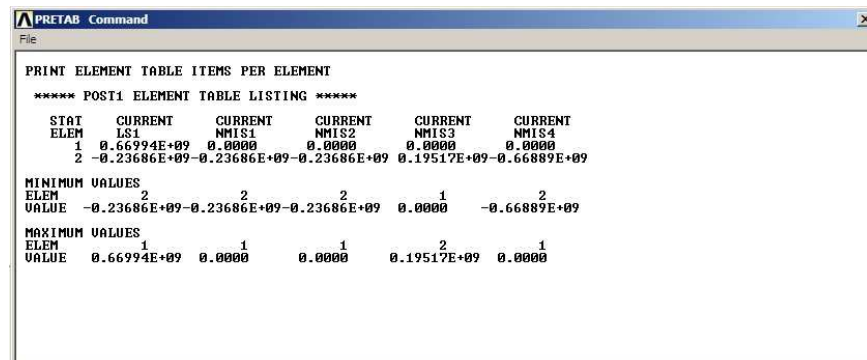
- Click on **Close** in the 'Element Table Data'.

Main Menu > General Postproc > Element Table > List Elem Table

- In the 'List Element Table Data' window that appears, select the first 5 items as shown,



- Click on **OK**. This opens the following 'PRETAB Command' window:



According to this ANSYS solution, the stress distribution in the bar and beam elements can be plotted as shown in Fig. 3.

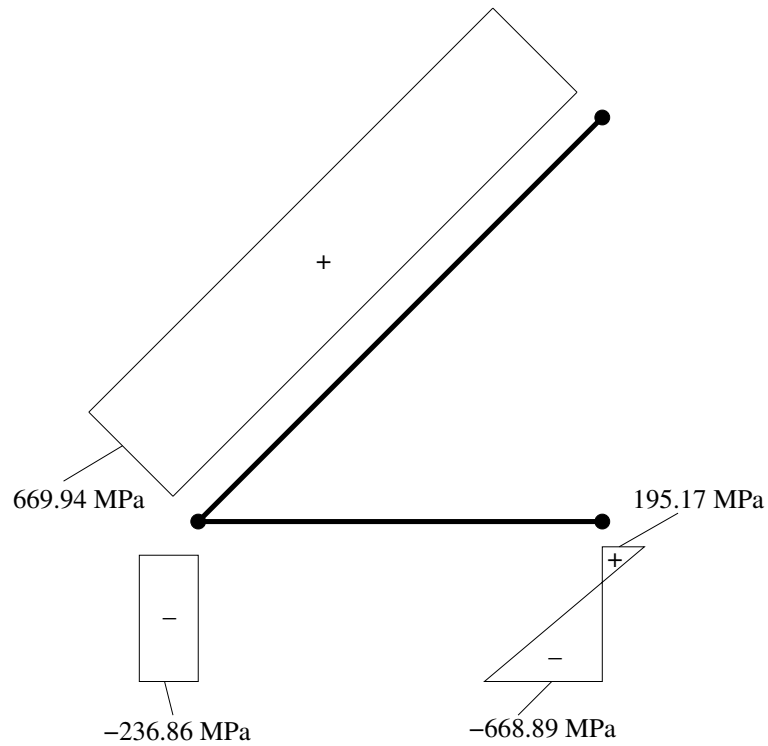


Figure 3: Stress distribution in the system

- Close the 'PRETAB Command' window.
5. Exit ANSYS, Saving All Data
Utility Menu > File > Exit ...
- In the window that opens, select 'Save Everything' and click on **OK**.