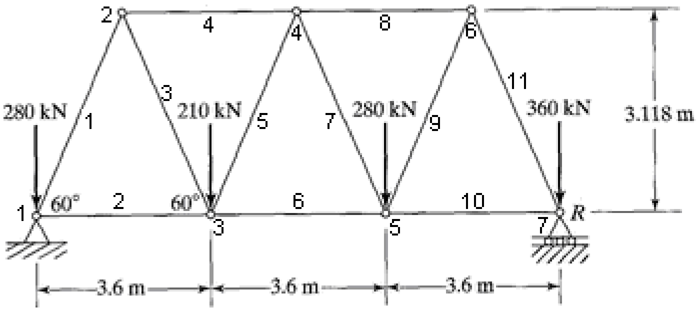
Two Dimensional Truss

Introduction

This tutorial was created using ANSYS 7.0 to solve a simple 2D Truss problem. This is the first of four introductory ANSYS tutorials.

Problem Description

Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200GPa, A = 3250mm^2).



(Modified from Chandrupatla & Belegunda, Introduction to Finite Elements in Engineering, p.123)

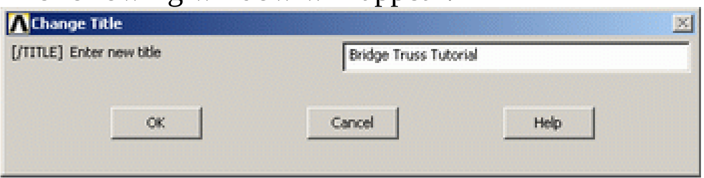
Preprocessing: Defining the Problem

1. **Give the Simplified Version 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**

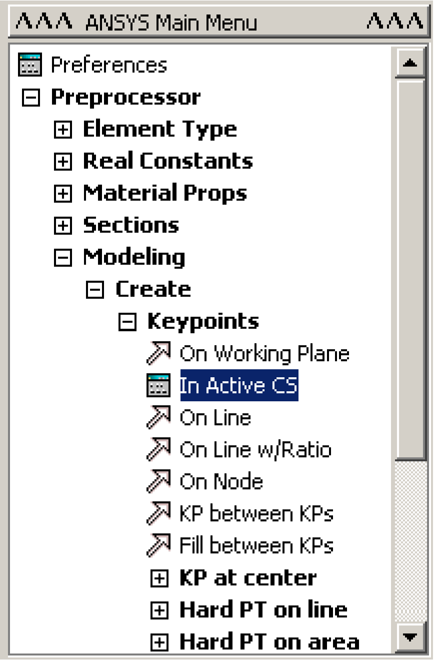
2. **Enter Keypoints**

   The overall geometry is defined in ANSYS using keypoints which specify various principal coordinates to define the body. For this example, these keypoints are the ends of each truss.

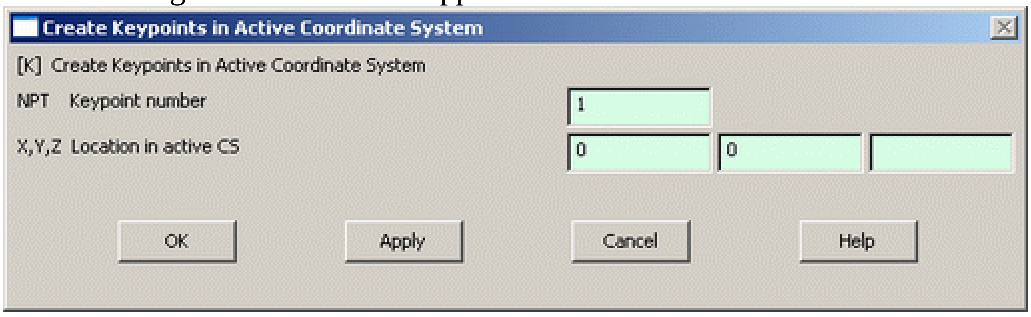
   - We are going to define 7 keypoints for the simplified structure as given in the following table

keypoint	coordinate	
	x	**y**
1	0	0
2	1800	3118
3	3600	0
4	5400	3118
5	7200	0
6	9000	3118
7	10800	0

   - (these keypoints are depicted by numbers in the above figure)
   - 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 (ie 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

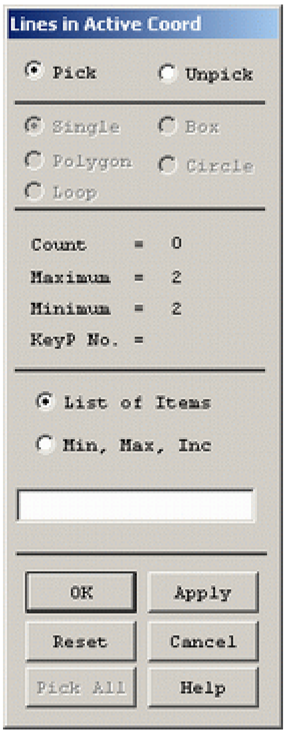
When defining keypoints, lines, areas, volumes, elements, constraints and loads you are bound to make mistakes. Fortunately these are easily corrected so that you don't need to begin from scratch every time an error is made! Every 'Create' menu for generating these various entities also has a corresponding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

3. **Form Lines**

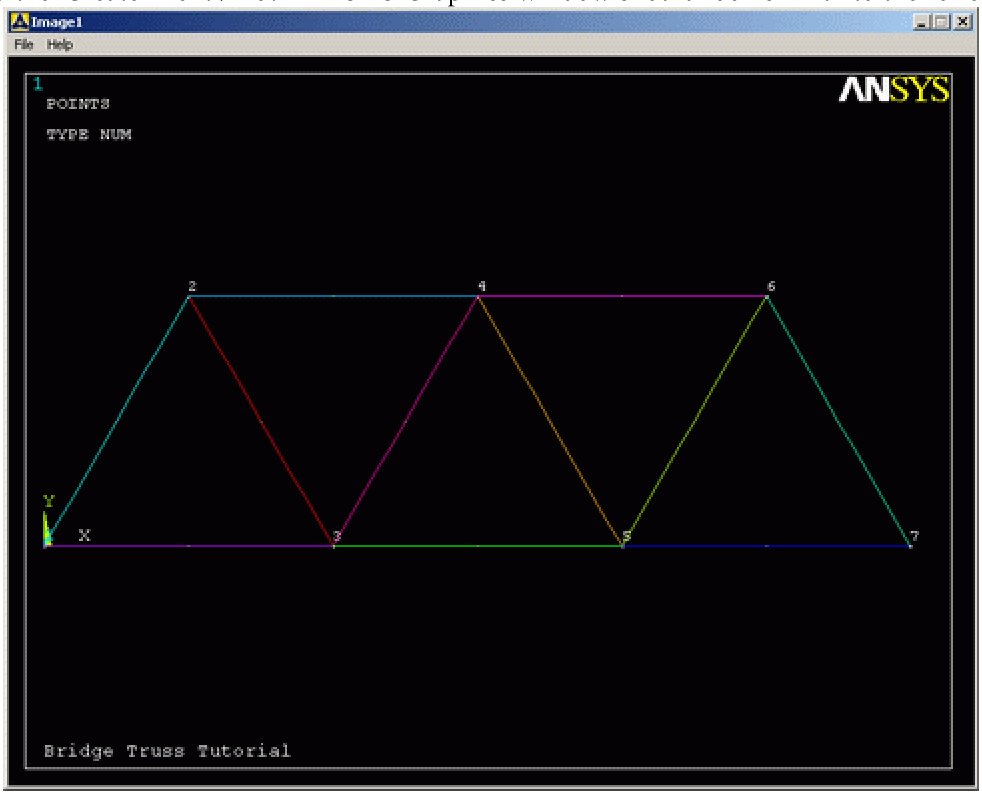
   The keypoints must now be connected

   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 Coord**.
     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 Coord' window, minimize the 'Lines' menu and the 'Create' menu. Your ANSYS Graphics window should look similar to the following figure.



Disappearing Lines

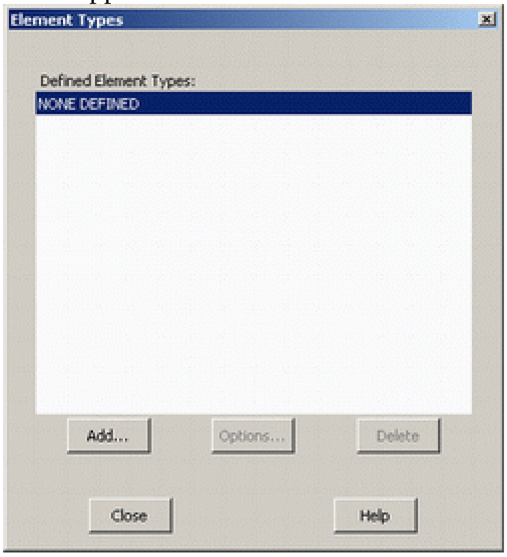
Please note that any lines you have created may 'disappear' throughout your analysis. However, they have most likely **NOT** been deleted. If this occurs at any time from the **Utility Menu** select:

Plot > Lines

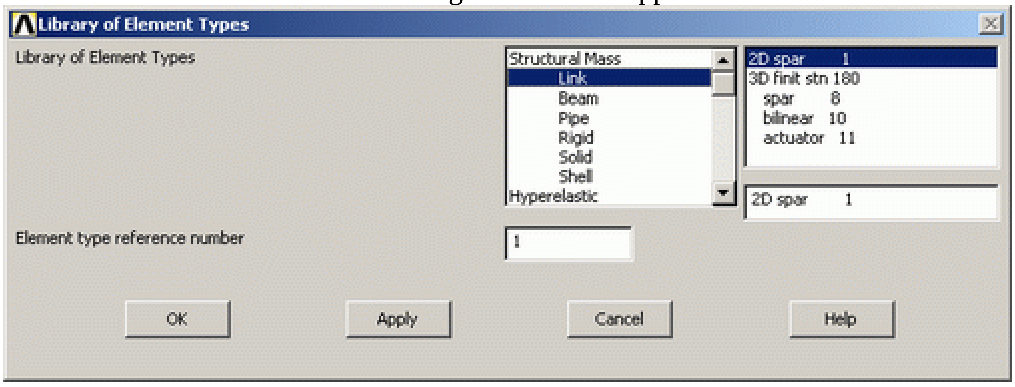
4. **Define the Type of Element**

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:



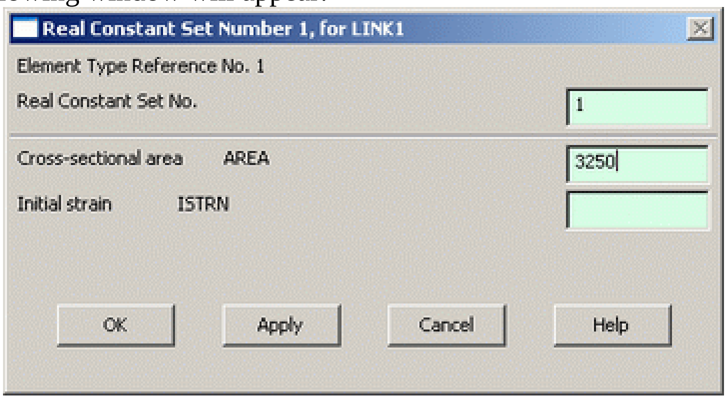
- For this example, we will use the 2D spar element as selected in the above figure. Select the element shown and click 'OK'. You should see 'Type 1 LINK1' in the 'Element Types' window.
- Click on 'Close' in the 'Element Types' dialog box.

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:

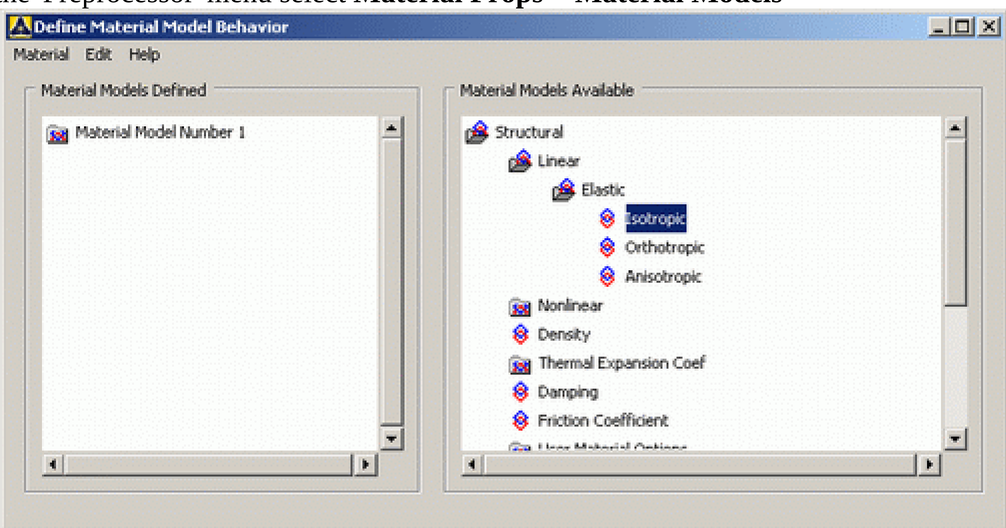


- As shown in the window above, enter the cross-sectional area (3250mm):
- Click on 'OK'.
- 'Set 1' now appears in the dialog box. Click on 'Close' in the 'Real Constants' window.

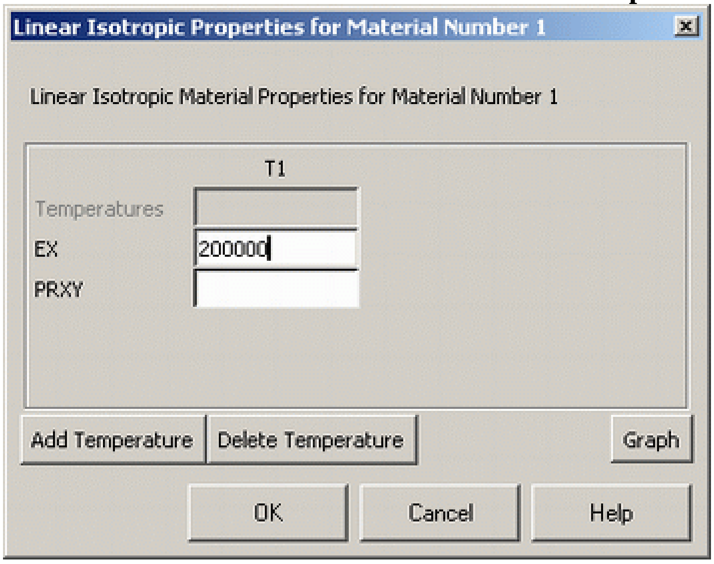
6. **Element Material Properties**

   You then need to specify material properties:

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



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



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

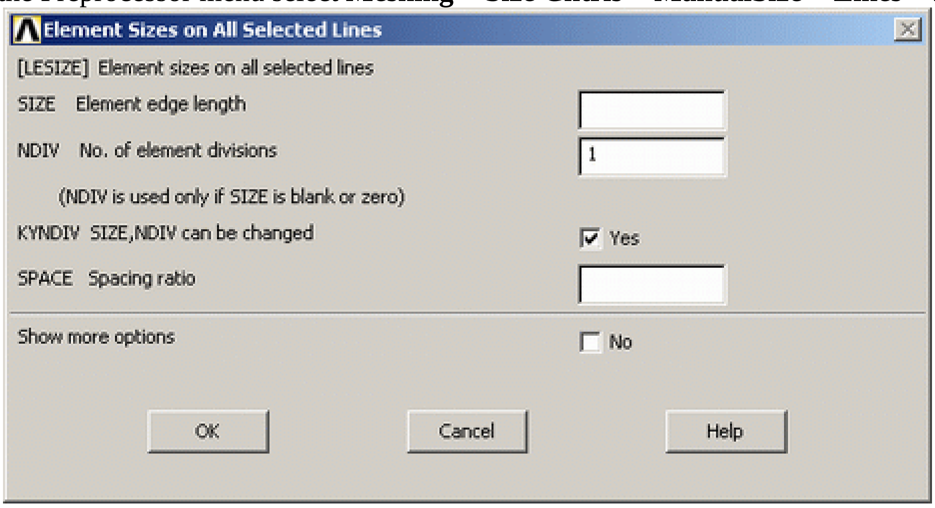
EX
200000

- o Set these properties and click on 'OK'. Note: You may obtain the note 'PRXY will be set to 0.0'. This is poisson's ratio and is not required for this element type. Click 'OK' on the window to continue. Close the "Define Material Model Behavior" by clicking on the 'X' box in the upper right hand corner.

7. **Mesh Size**

The last step before meshing is to tell ANSYS what size the elements should be. There are a variety of ways to do this but we will just deal with one method for now.

- o In the Preprocessor menu select **Meshing** > **Size Cntrls** > **ManualSize** > **Lines** > **All Lines**



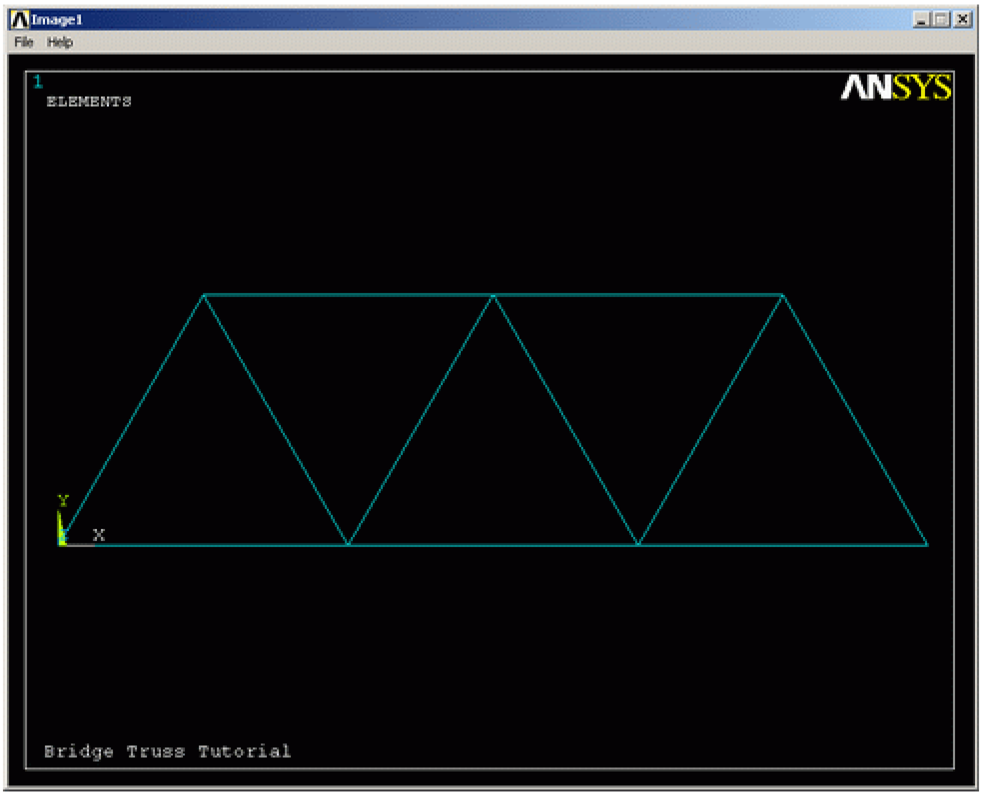
- o In the size 'NDIV' field, enter the desired number of divisions per line. For this example we want only 1 division per line, therefore, enter '1' and then click 'OK'. Note that we have not yet meshed the geometry, we have simply defined the element sizes.

8. **Mesh**

Now the frame can be meshed.

- o In the 'Preprocessor' menu select **Meshing** > **Mesh** > **Lines** and click 'Pick All' in the 'Mesh Lines' Window

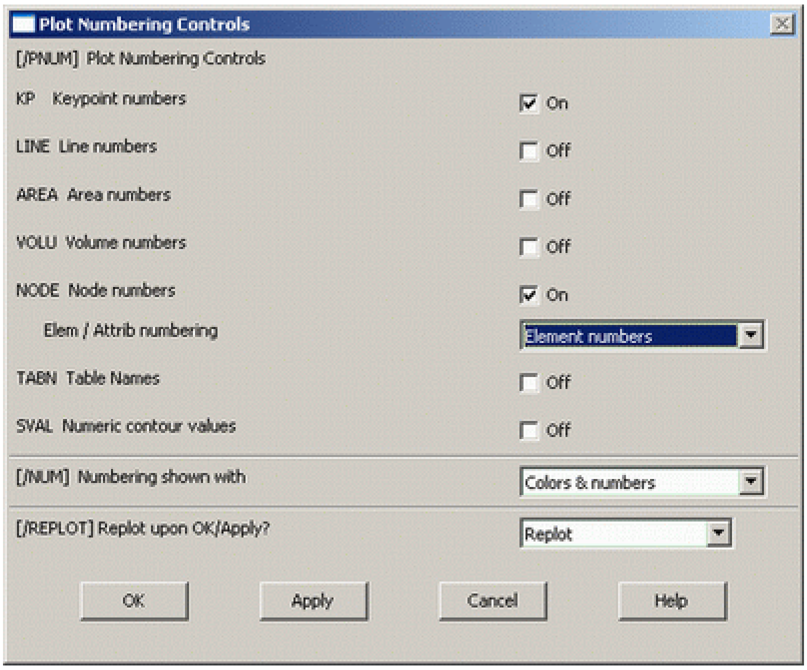
Your model should now appear as shown in the following window



Plot Numbering

To show the line numbers, keypoint numbers, node numbers...

- From the **Utility Menu** (top of screen) select **PlotCtrls > Numbering...**
- Fill in the Window as shown below and click 'OK'



Now you can turn numbering on or off at your discretion

Saving Your Work

Save the model at this time, so if you make some mistakes later on, you will at least be able to come back to this point. To do this, on the **Utility Menu** select **File > Save as...**. Select the name and location where you want to save your file.

It is a good idea to save your job at different times throughout the building and analysis of the model to backup your work in case of a system crash or what have you.

Solution Phase: Assigning Loads and Solving

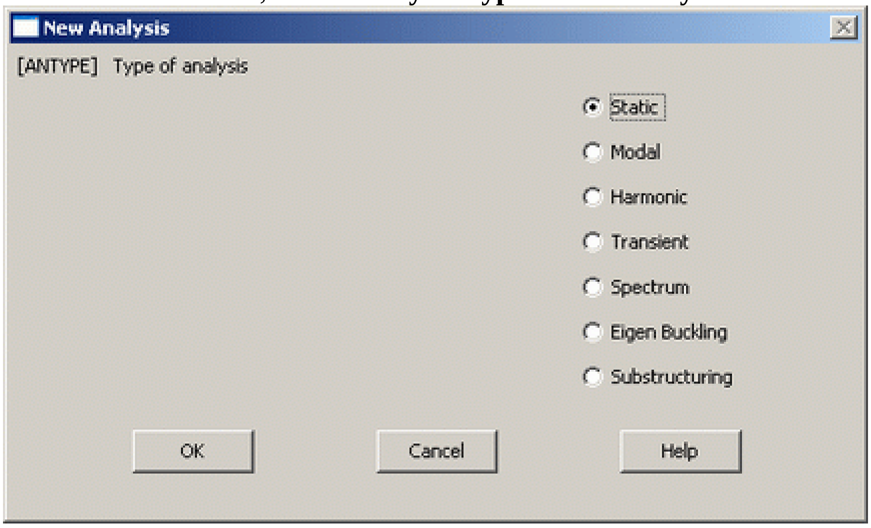
You have now defined your model. It is now time to apply the load(s) and constraint(s) and solve the the resulting system of equations.

Open up the 'Solution' menu (from the same 'ANSYS Main Menu').

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**.

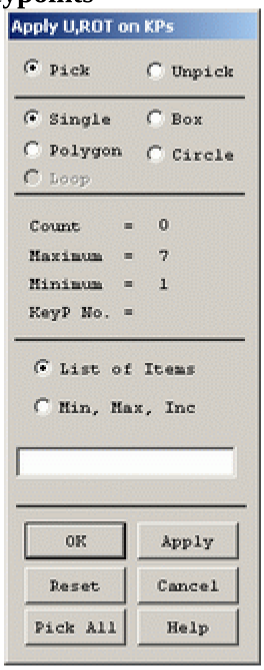


   - Ensure that 'Static' is selected; i.e. you are going to do a static analysis on the truss as opposed to a dynamic analysis, for example.
   - Click 'OK'.

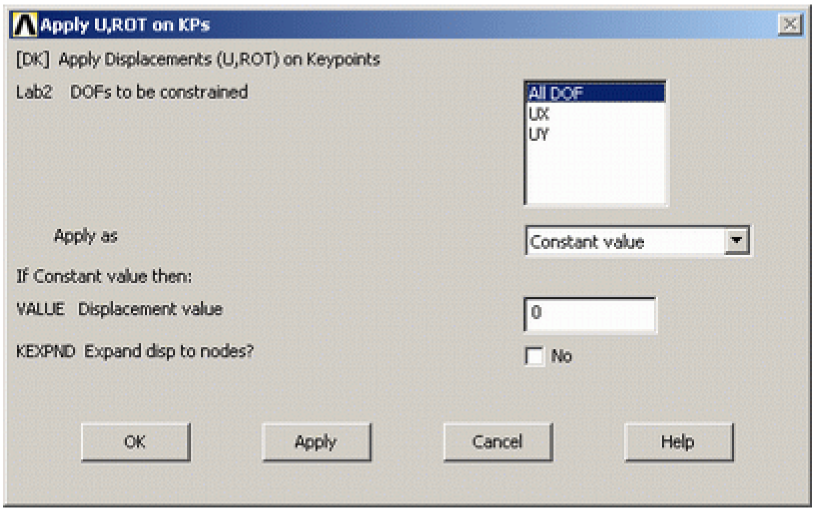
2. **Apply Constraints**

   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' in the 'Apply U,ROT on KPs' window.

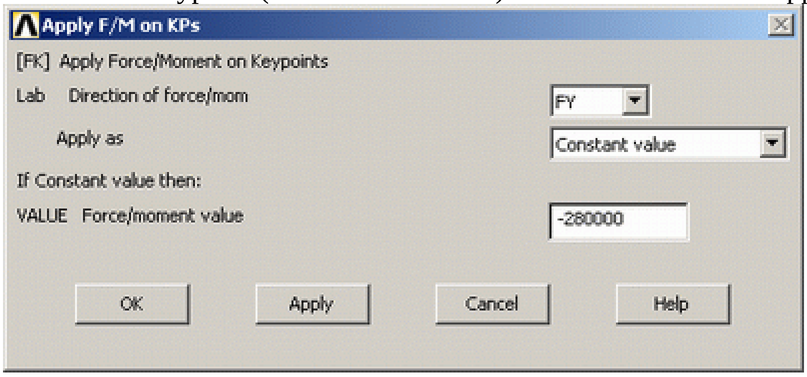


- This location is fixed which means that all translational and rotational degrees of freedom (DOFs) are constrained. Therefore, select 'All DOF' by clicking on it and enter '0' in the Value field and click 'OK'.

  You will see some blue triangles in the graphics window indicating the displacement contraints.
- Using the same method, apply the roller connection to the right end (UY constrained). Note that more than one DOF constraint can be selected at a time in the "Apply U,ROT on KPs" window. Therefore, you may need to 'deselect' the 'All DOF' option to select just the 'UY' option.

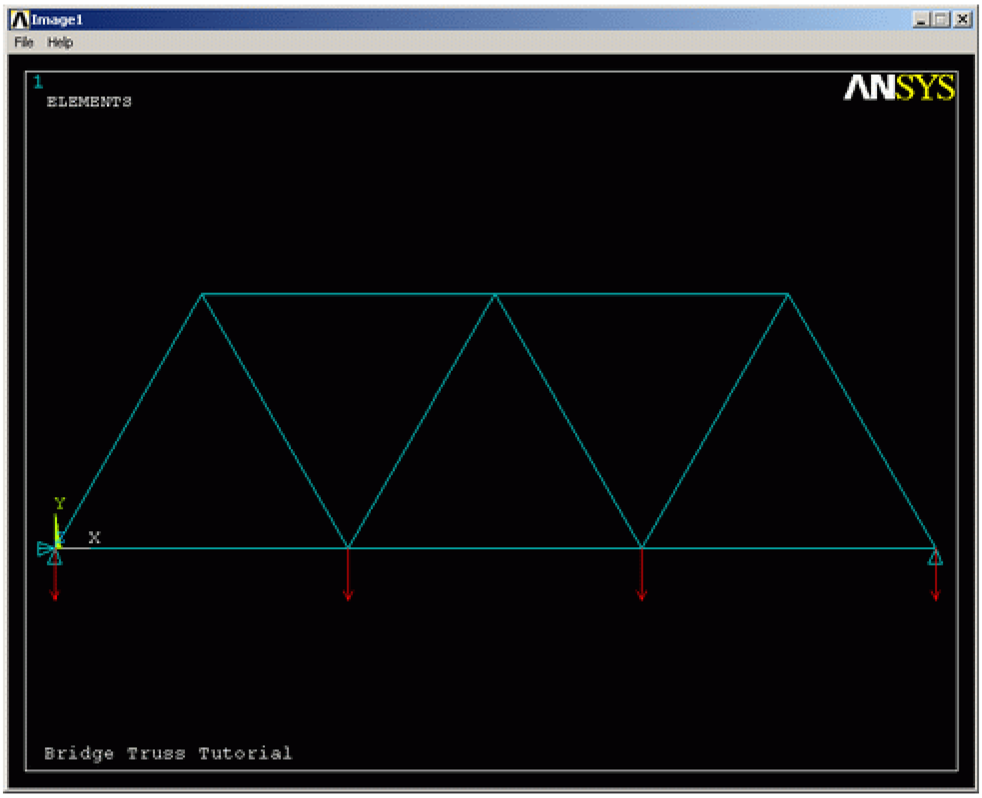
3. **Apply Loads**

   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 the first Keypoint (left end of the truss) and click 'OK' in the 'Apply F/M on KPs' window.



- Select FY in the 'Direction of force/mom'. This indicate 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.

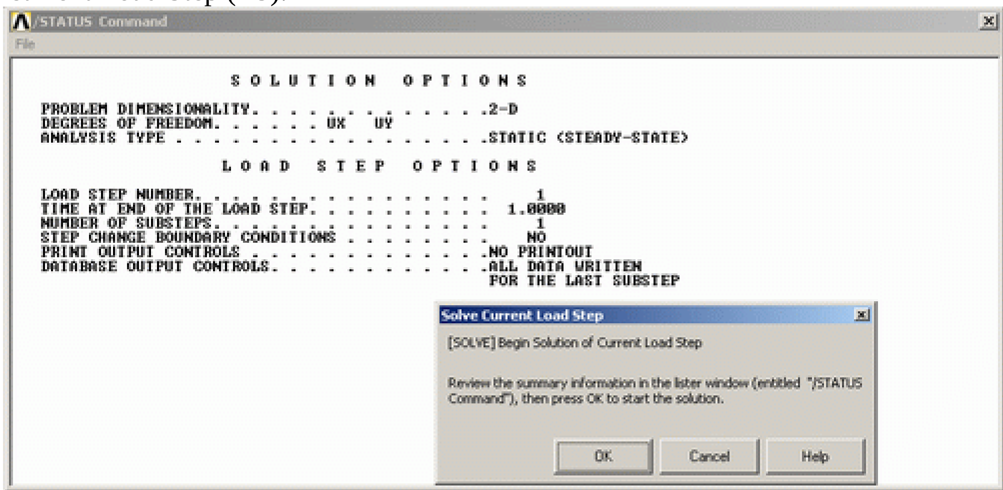
The applied loads and constraints should now appear as shown below.



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..

Postprocessing: Viewing the Results

1. **Hand Calculations**

We will first calculate the forces and stress in element 1 (as labeled in the problem description).

$$\circlearrowleft \sum M_1 = 0 = -210\,\text{kN}(3.6\,\text{m}) - 280\,\text{kN}(7.2\,\text{m}) - 360\,\text{kN}(10.8\,\text{m}) + F_7(10.8\,\text{m})$$

$$F_7 = \frac{210\,\text{kN}(3.6\,\text{m}) + 280\,\text{kN}(7.2\,\text{m}) + 360\,\text{kN}(10.8\,\text{m})}{10.8\,\text{m}} = 617\,\text{kN}$$

$$\uparrow \sum F_y = 0 = -280\,\text{kN} - 210\,\text{kN} - 280\,\text{kN} - 360\,\text{kN} + 617\,\text{kN} + F_1$$

$$F_1 = 280\,\text{kN} + 210\,\text{kN} + 280\,\text{kN} + 360\,\text{kN} - 617\,\text{kN} = 513\,\text{kN}$$

Element 1 Forces/Stress

$$F_{E1} = \frac{513\,\text{kN} - 280\,\text{kN}}{\cos(30)} = 269\,\text{kN}$$

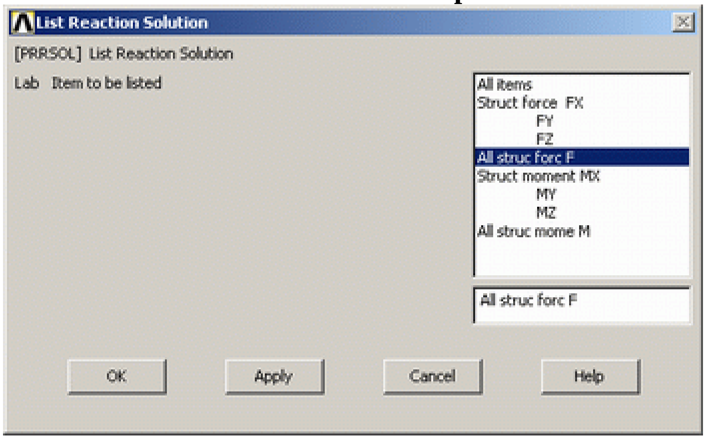
$$\sigma_{E1} = \frac{F_{E1}}{A} = \frac{269\,\text{kN}}{3250\,mm^2} = 82.8\,\text{MPa}$$

2. **Results Using ANSYS**

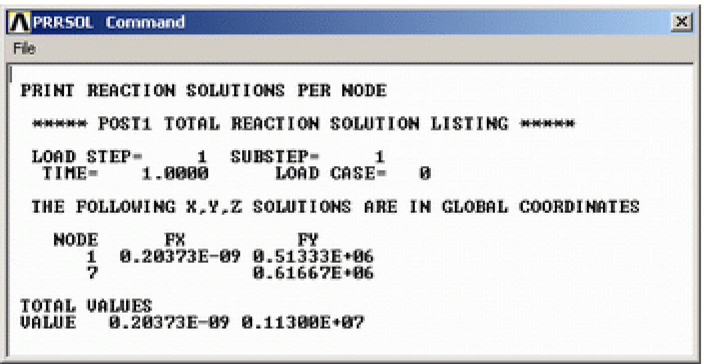
Reaction Forces

A list of the resulting reaction forces can be obtained for this element

- from the Main Menu select **General Postproc > List Results > Reaction Solu**.



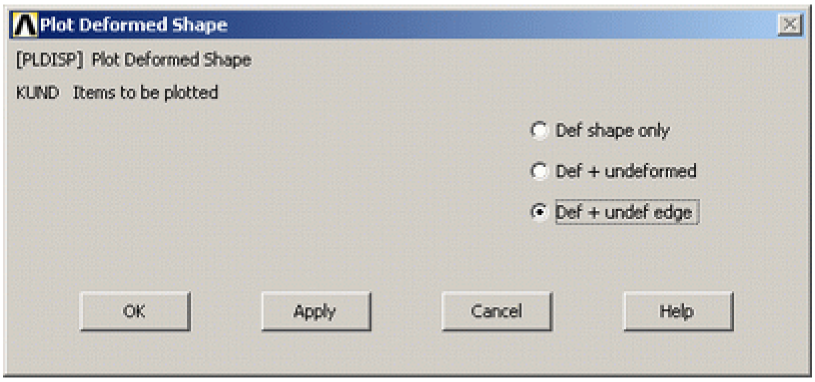
- Select 'All struc forc F' as shown above and click 'OK'



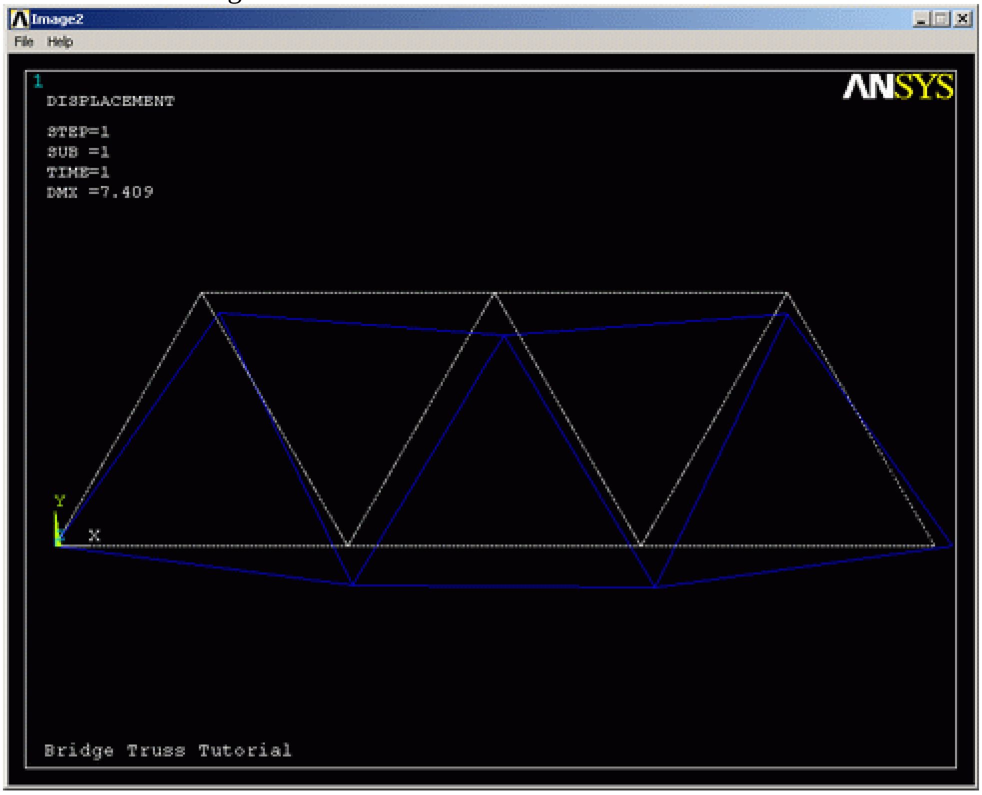
These values agree with the reaction forces claculated by hand above.

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.

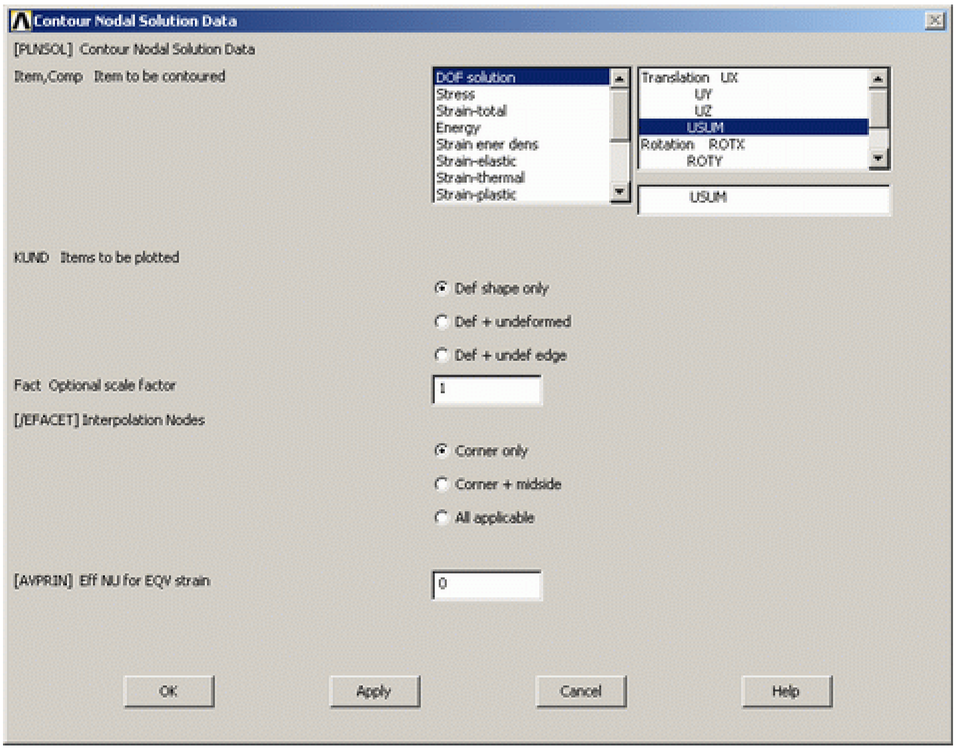


- Observe the value of the maximum deflection in the upper left hand corner (DMX=7.409). One should also observe that the constrained degrees of freedom appear to have a deflection of 0 (as expected!)

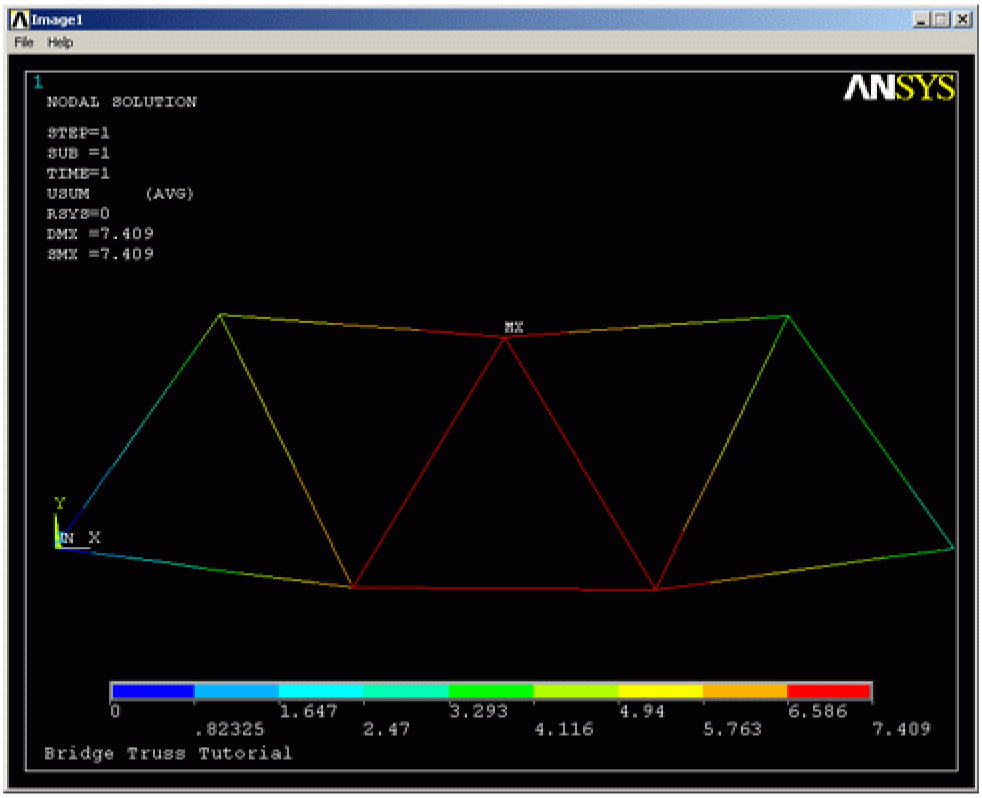
Deflection

For a more detailed version of the deflection of the beam,

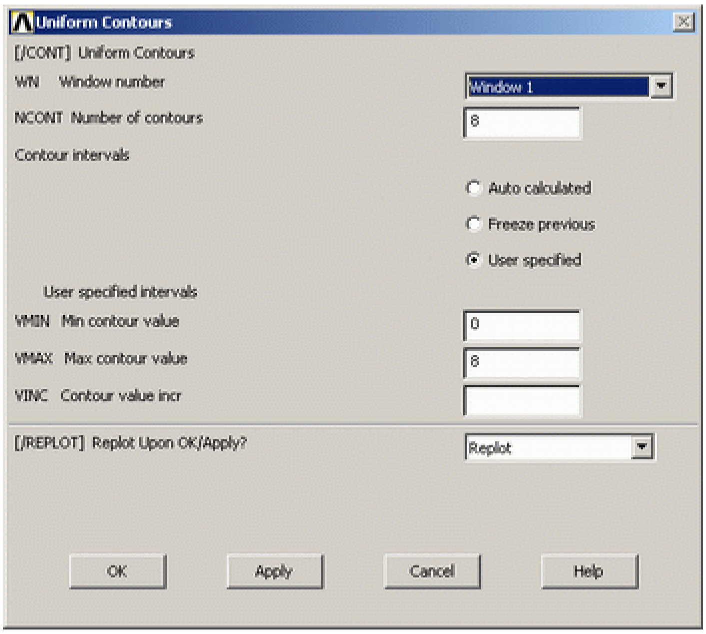
- From the 'General Postproc' menu select **Plot results > Contour Plot > Nodal Solution**. The following window will appear.



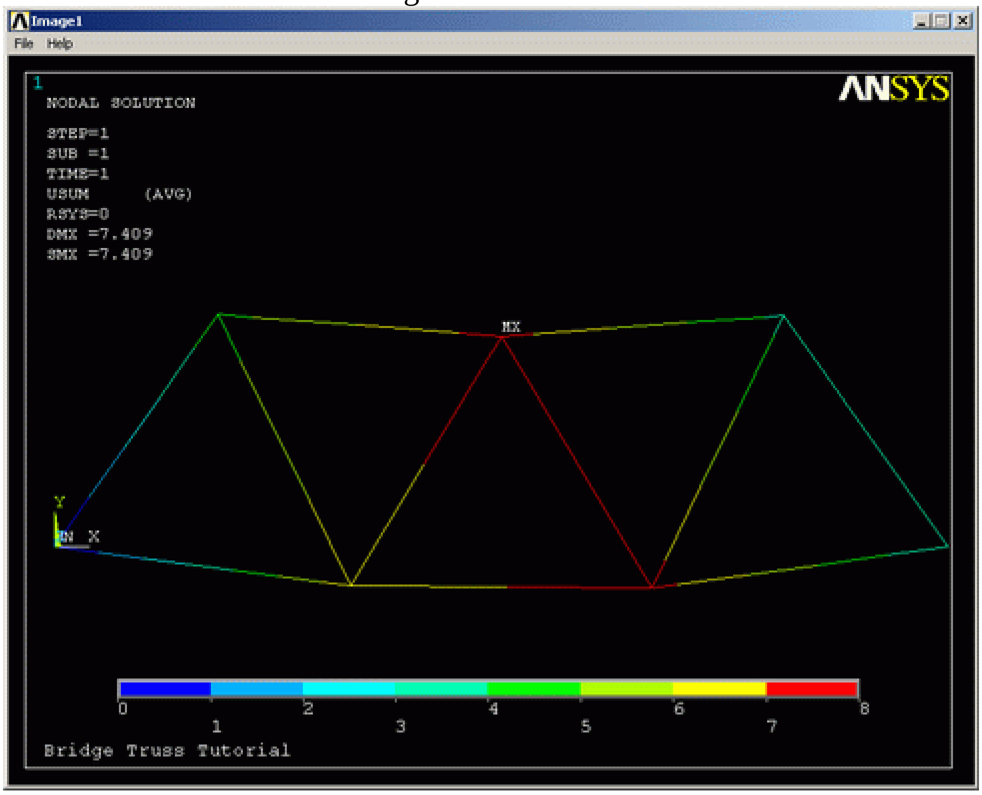
- Select 'DOF solution' and 'USUM' as shown in the above window. Leave the other selections as the default values. Click 'OK'.



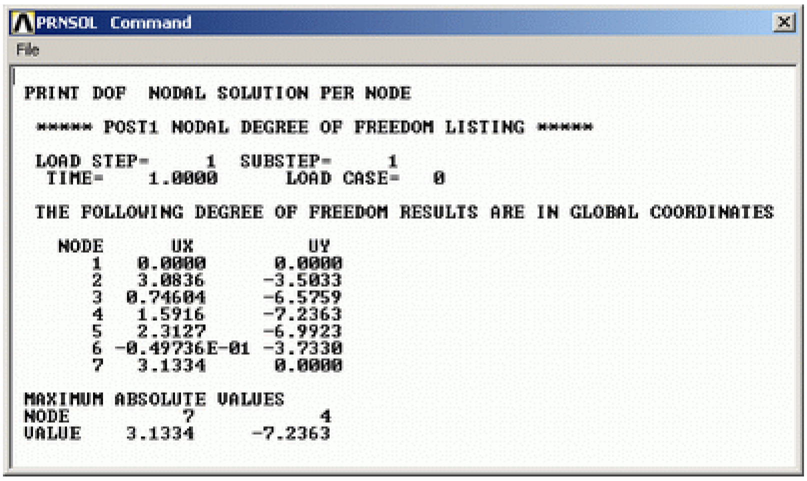
- Looking at the scale, you may want to use more useful intervals. From the **Utility Menu** select **Plot Controls > Style > Contours > Uniform Contours...**
- Fill in the following window as shown and click 'OK'.



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.



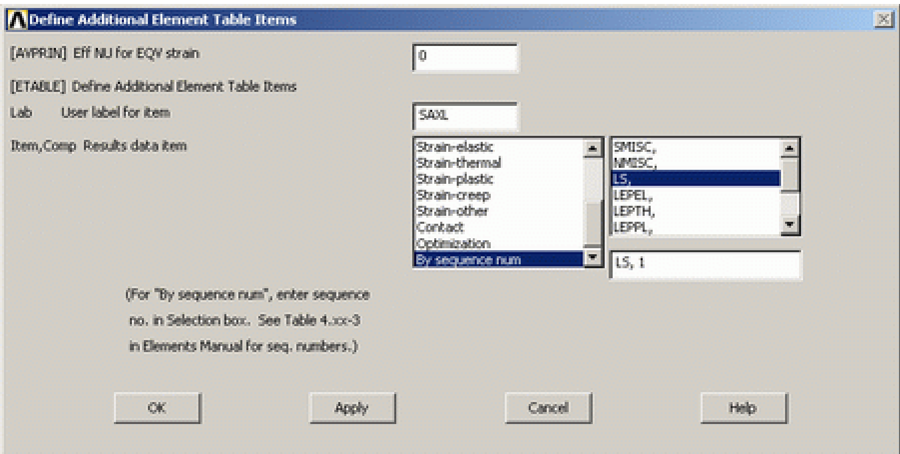
```
PRNSOL Command
File

PRINT DOF  NODAL SOLUTION PER NODE

 ***** 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 GLOBAL COORDINATES

   NODE       UX           UY
      1    0.0000       0.0000
      2    3.0836      -3.5033
      3   0.74604      -6.5759
      4    1.5916      -7.2363
      5    2.3127      -6.9923
      6 -0.49736E-01   -3.7330
      7    3.1334       0.0000

MAXIMUM ABSOLUTE VALUES
NODE           7            4
VALUE     3.1334      -7.2363
```

- Are these results what you expected? Note that all the degrees of freedom were constrained to zero at node 1, while UY was constrained to zero at node 7.
- If you wanted to save these results to a file, select 'File' within the results window (at the upper left-hand corner of this list window) and select 'Save as'.

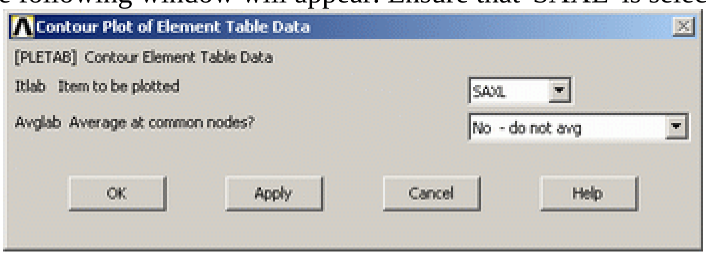
Axial Stress

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 to compare with the hand calculations. The Element Table is different for each element, therefore, we need to look at the help file for LINK1 (Type `help link1` into the Input Line). From Table 1.2 in the Help file, we can see that SAXL can be obtained through the ETABLE, using the item 'LS,1'

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

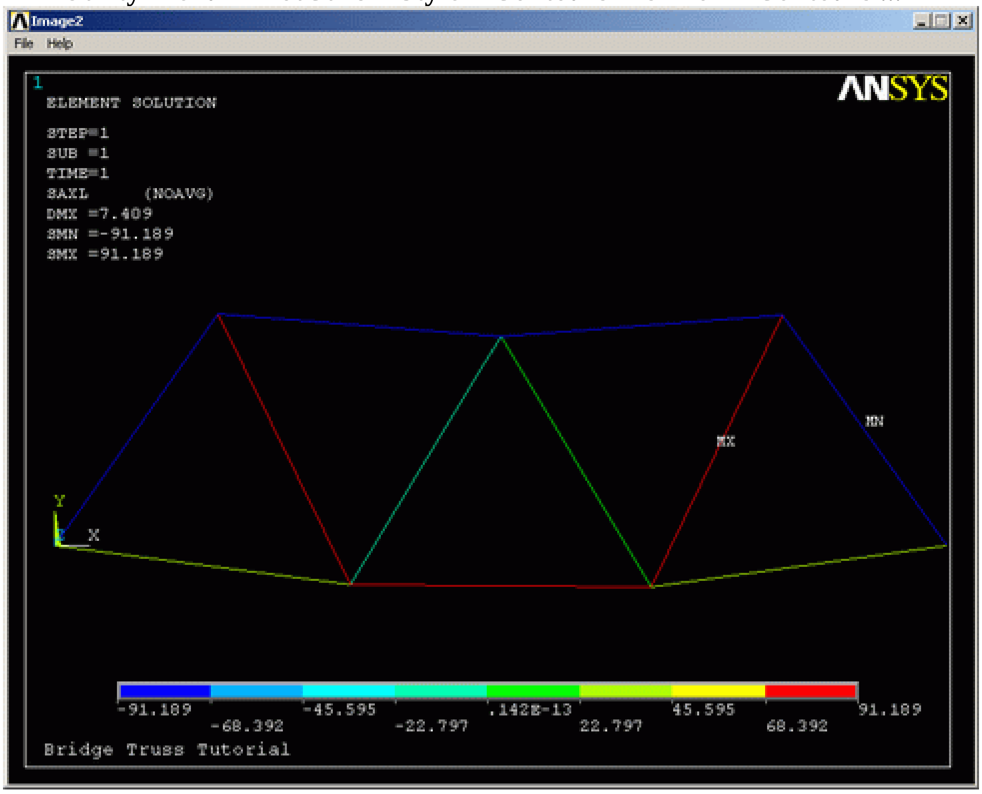


- As shown above, enter 'SAXL' in the 'Lab' box. This specifies the name of the item you are defining. Next, in the 'Item,Comp' boxes, select 'By sequence number' and 'LS,'. Then enter 1 after LS, in the selection box
- Click on 'OK' and close the 'Element Table Data' window.
- Plot the Stresses by selecting **Element Table > Plot Elem Table**
- The following window will appear. Ensure that 'SAXL' is selected and click 'OK'



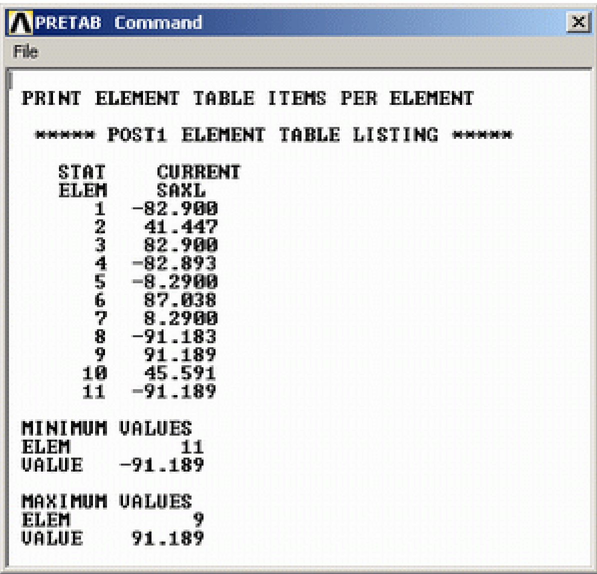
- Because you changed the contour intervals for the Displacement plot to "User Specified" - you need to switch this back to "Auto calculated" to obtain new values for VMIN/VMAX.

Utility Menu > PlotCtrls > Style > Contours > Uniform Contours ...



Again, you may wish to select more appropriate intervals for the contour plot

- List the Stresses
  - From the 'Element Table' menu, select **'List Elem Table'**
  - From the 'List Element Table Data' window which appears ensure 'SAXL' is highlighted
  - Click 'OK'



PRETAB Command
File

```
PRINT ELEMENT TABLE ITEMS PER ELEMENT

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

    STAT     CURRENT
    ELEM     SAXL
       1   -82.900
       2    41.447
       3    82.900
       4   -82.893
       5   -8.2900
       6    87.038
       7    8.2900
       8   -91.183
       9    91.189
      10    45.591
      11   -91.189

MINIMUM VALUES
ELEM         11
VALUE   -91.189

MAXIMUM VALUES
ELEM          9
VALUE    91.189
```

Note that the axial stress in Element 1 is 82.9MPa as predicted analytically.

Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Copy and paste following code into Notepad or a similar text editor and save it to your computer. Now go to **'File > Read input from...'** and select the file.

```
! ANSYS command file to perform 2D Truss Tutorial (Chandrupatla p.123)
!
```

```
/title, Bridge Truss Tutorial
/PREP7                  ! preprocessor phase
!
! define parameters (mm)
height = 3118
width = 3600
!
! define keypoints
!
K,1,     0,    0             ! keypoint, #, x, y
K,2,   width/2,height
K,3,     width,  0
K,4, 3*width/2,  height
K,5,   2*width,    0
K,6, 5*width/2,  height
K,7,   3*width,  0
!
! define lines
!
L,1,2                   ! line connecting kpoint 1 and 2
L,1,3
L,2,3
L,2,4
L,3,4
L,3,5
L,4,5
L,4,6
L,5,6
L,5,7
L,6,7

!
! element definition
!
ET,1,LINK1              ! element type      #1; spring element
R,1,3250                ! real constant     #1; Xsect area: 3200 mm^2
MP,EX,1,200e3          ! material property #1; Young's modulus: 200 GPa

LESIZE,ALL, , ,1,1,1    ! specify divisions on unmeshed lines
LMESH,all               ! mesh all lines
!
FINISH                  ! finish pre-processor
!
/SOLU                   ! enter solution phase
!
! apply some constraints
DK,1,ALL,0        ! define a DOF constraint at a keypoint
DK,7,UY,0
!
! apply loads
!
FK,1,FY,-280e3          ! define a force load to a keypoint
FK,3,FY,-210e3
FK,5,FY,-280e3
FK,7,FY,-360e3
!
SOLVE                   ! solve the resulting system of equations
FINISH                  ! finish solution

/POST1
PRRSOL,F                ! List Reaction Forces
PLDISP,2                ! Plot Deformed shape
PLNSOL,U,SUM,0,1        ! Contour Plot of deflection
```

```
ETABLE,SAXL,LS, 1          ! Axial Stress
PRETAB,SAXL                ! List Element Table
PLETAB,SAXL,NOAV           ! Plot Axial Stress
```

Quitting ANSYS

To quit ANSYS, select 'QUIT' from the ANSYS Toolbar or select **Utility Menu/File/Exit...**. In the dialog box that appears, click on 'Save Everything' (assuming that you want to) and then click on 'OK'.

Reference

http://www.mece.ualberta.ca/tutorials/ansys