ANSYS - Truss 8 Postprocess the results content

Step 8: Postprocess the results

Postprocessing is the step where we look at and analyze the results obtained from the ANSYS solution.
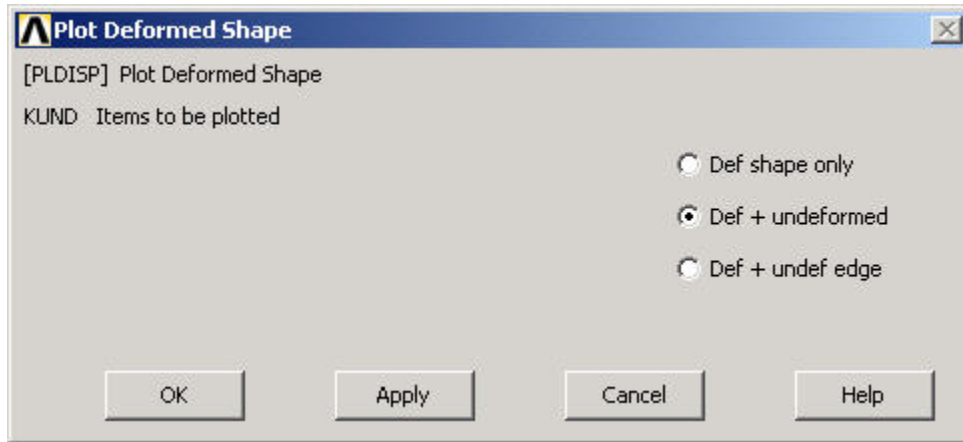
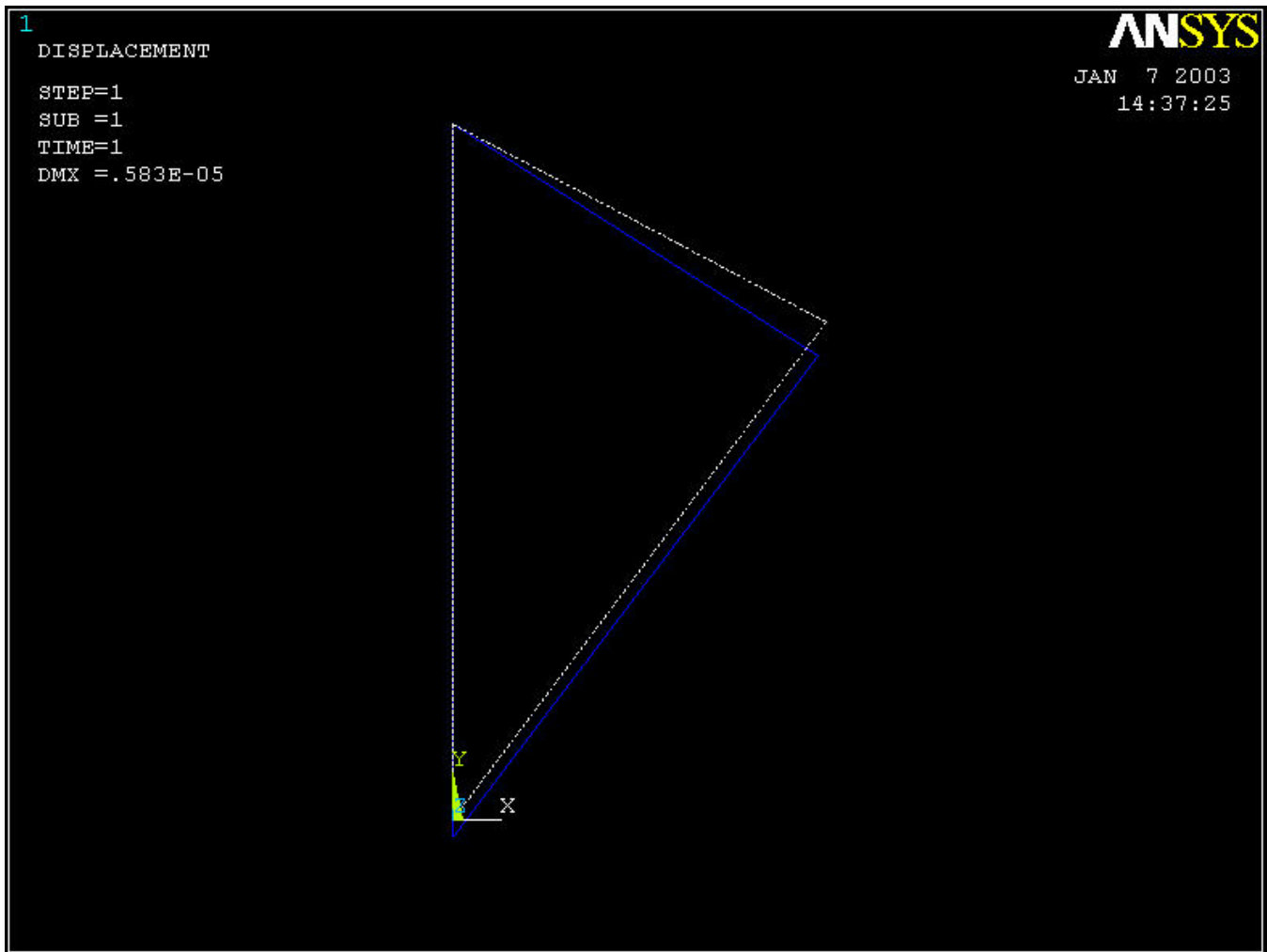Enter the *General Postprocessing* module_:_

Main Menu > General Postproc

Plot Deformed Shape

Main Menu > General Postproc > Plot Results > Deformed Shape

Select Def + undeformed and click OK.

This plots the deformed and undeformed shapes in the *Graphics* window.

The deformed shape is shown as a solid line and the undeformed shape as a dotted line. The maximum displacement *DMX* is 0.583E-05m as reported in the *Graphics* window. This is small but plausible. Note that the deformation is magnified in the plot so as to be easily visible.

To save the deformation plot in a file, use Utility Menu > PlotCtrls > Hard Copy > To File. Select the file format you want and type in a filename of your choice under **Save to:** and click **OK**. The file will be created in your working directory. You can print out this file as necessary.

Animate the deformation:

Utility Menu > PlotCtrls > Animate > Deformed Shape

Select Def + undeformed and click OK. Select Forward Only in the *Animation Controller*.

Node 1 (Pin A) doesn't move and node 2 (Pin C) moves only in the vertical direction. Node 3 (Pin B) moves more or less in the direction of the applied force. The deformation of the structure agrees with the applied boundary conditions and matches with what one would expect from intuition.
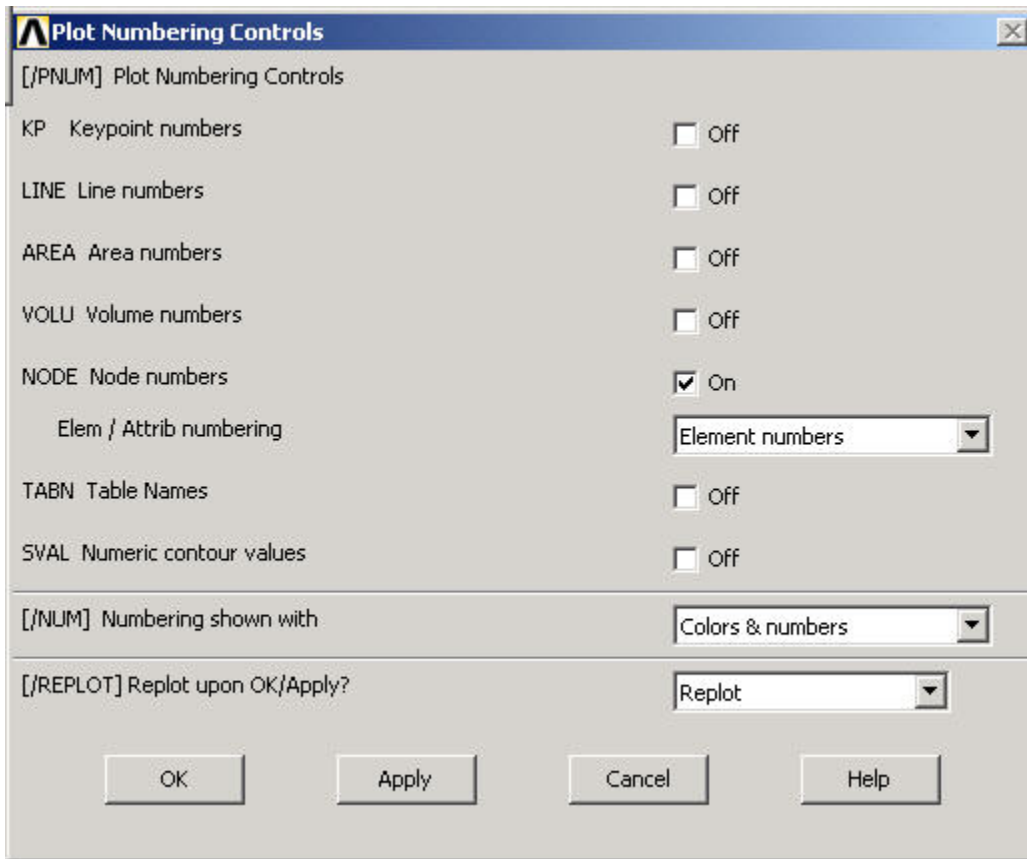
Turn On Node and Element Numbers

In order to interpret the results that ANSYS reports, it's useful to turn on the node and element numbers in the *Graphics* window.

Utility Menu > PlotCtrls > Numbering

The *Plot Numbering Controls* menu is used to control the numbering of the various entities in a finite-element model.

Turn on Node numbers. Under Elem/Attrib numbering, select Element numbers. Click OK.
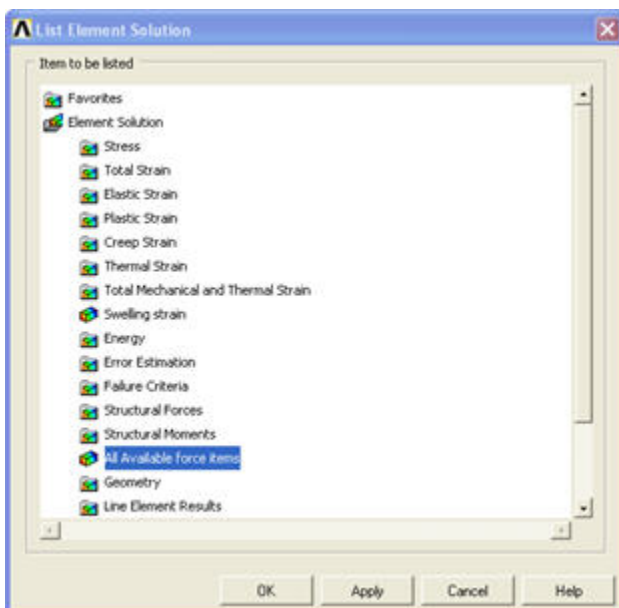
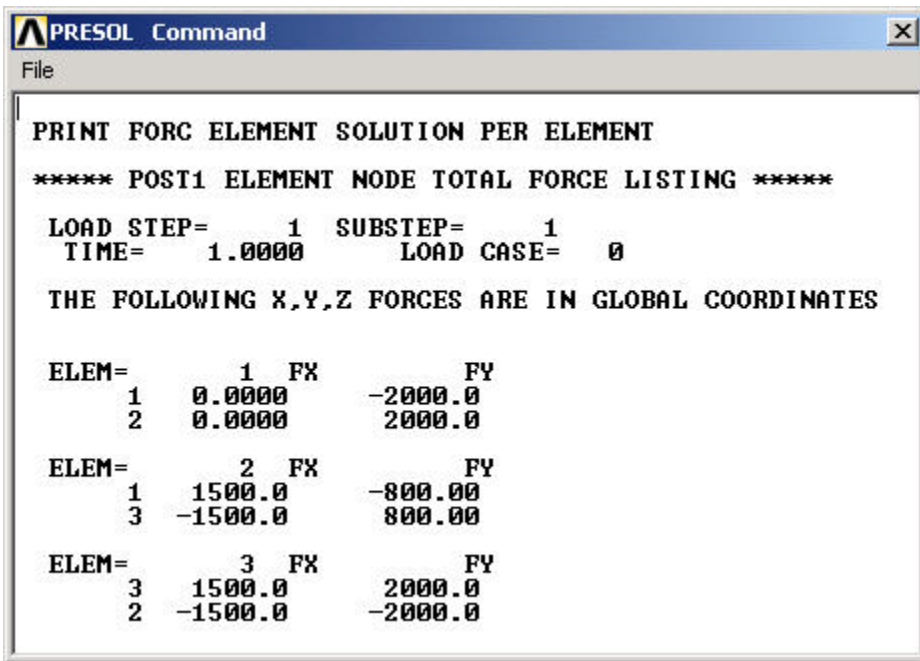The node and element numbers will now appear in the *Graphics* window.

List Forces in Truss Members: Method 1

Main Menu > General Postproc > List Results > Element Solution

From the list, under *Element Solution*, select All Available force items . Click OK.

This brings up a window listing the forces that the elements apply on each of their nodes:

```
 ∧ 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          FY
        1     0.0000      -2000.0
        2     0.0000       2000.0

   ELEM=        2  FX          FY
        1     1500.0      -800.00
        3    -1500.0       800.00

   ELEM=        3  FX          FY
        3     1500.0       2000.0
        2    -1500.0      -2000.0
```

These element forces reported by ANSYS are forces ON its environment BY the element, not the converse. For example, Element 2 (or member AB) applies a force of 1500 N in the x-direction and 800 N in the negative y-direction on node 1 (or pin A). This means that the total force in AB is . The resultant acts from A to B i.e. the member is pulling on pin A. So it must be in tension. Similarly, the force in Element 1 (AC) is 2000 N (tension) and in Element 3 (BC) is 2500 N (compression). Note that your node and element numbers might be different from the above since they depend on the order in which the lines were created.

Close the *PRESOL Command* window.

List Forces in Truss Members: Method 2

Bring up the help page for LINK1 element:

Utility Menu > Help > Help Topics

Under the ***Contents*** tab, select

Release 10.0 Documentation for ANSYS > Element Reference > Element Library > LINK1

In the *LINK1* help page, scroll down to *LINK1 Element Output Definitions.* You'll see the item

MFORX: Member force in the element coordinate system X direction

The figure at the top of the *LINK1* help page shows that the x-direction in the element coordinate system is along the line. So *MFORX* is basically the axial force in the element.

So how do we get the *MFORX* values for our three elements from ANSYS? ANSYS has a quirky way of doing this as we shall see. If you scroll down the help page further, you'll see the *LINK1 Item and Sequence Numbers:*

MFORX SMISC 1

The output data is broken down into item groups with *SMISC* being one of the groups. Each item within an item group has an identifying "sequence" number. So *MFORX* is the item with sequence number 1 in the *SMISC* group.

Minimize the help window. To list *MFORX* values:

Main Menu > General Postproc > List Results > Element Solution

Under Element Solution, select Miscellaneous Items > Summable data (SMISC,1). Since *MFORX* is sequence number 1 in the *SMISC* group, enter 1 next to Sequent number SMIS in the editable field. Click OK. Click OK in the List Element Solution window.
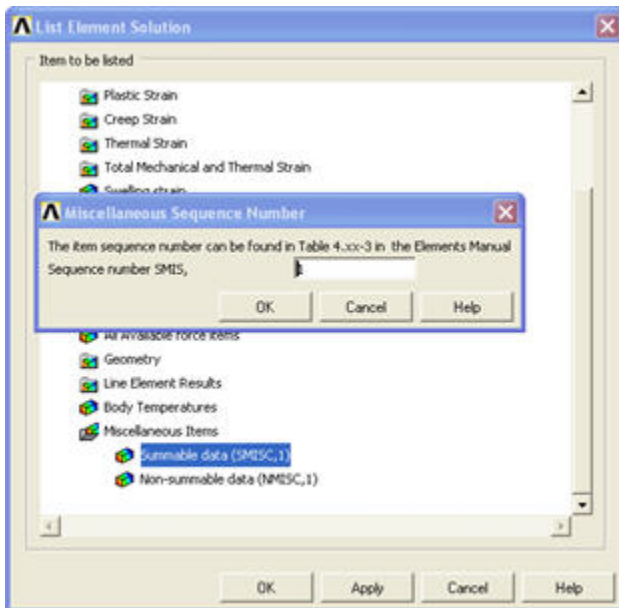
This brings up a window with the axial forces in the elements. Positive values indicate tension and negative values compression. Do these values match what we got in method 1?
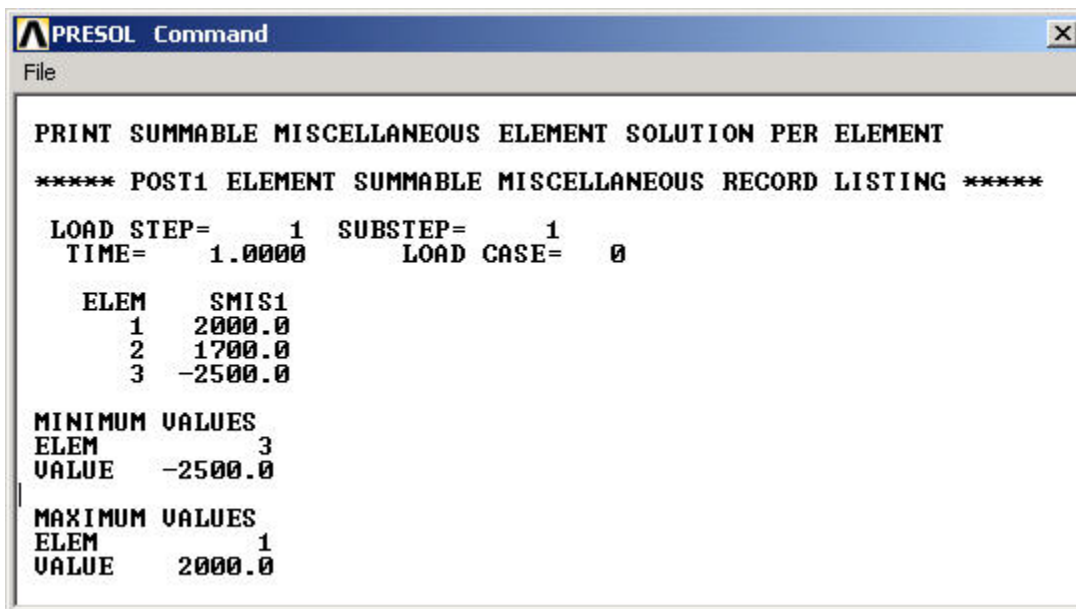

You can also **plot** the items listed under *Element Output Definitions* using the sequence number.

In most cases, you plot stresses using Main menu > General Postproc > Plot Results > Contour Plot >Nodal Solu. But for line elements like *LINK1*, this doesn't work and you'll get zero values for the stresses. So you'll have to use the sequence numbers to make stress plots for line elements.
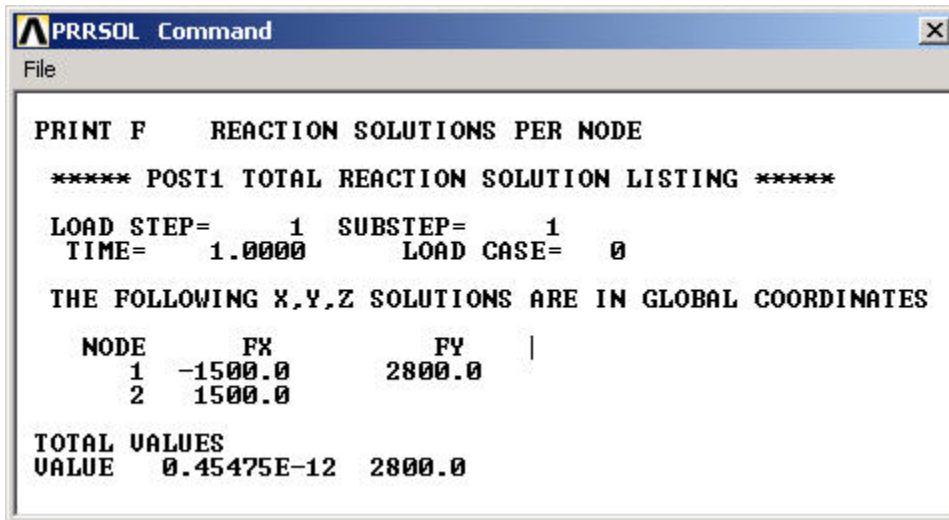
List Reaction Forces at Nodes

Main Menu > General Postproc > List Results > Reaction Solu

Select All struc forc F for Item to be listed and click OK.
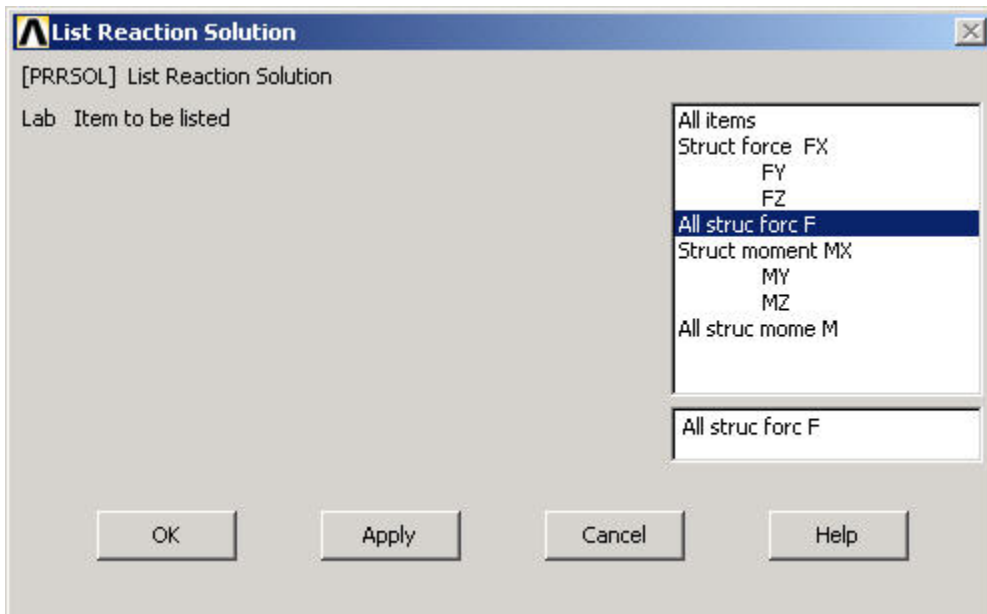
This brings up a window with the reaction forces at the nodes.

The sum of the reaction forces balances the applied load as should be the case for static equilibrium.

Close the *PRRSOL Command* window.