ANSYS - Truss Step 8

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Step 8: Postprocess the results

Postprocessing is the step where we look at and analyze the results obtained from the ANSYS solution.

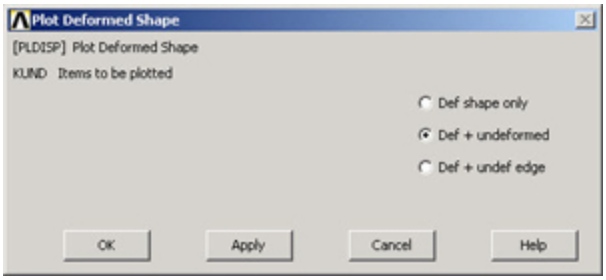
Enter the *General Postprocessing* module:

Main Menu > General Postproc

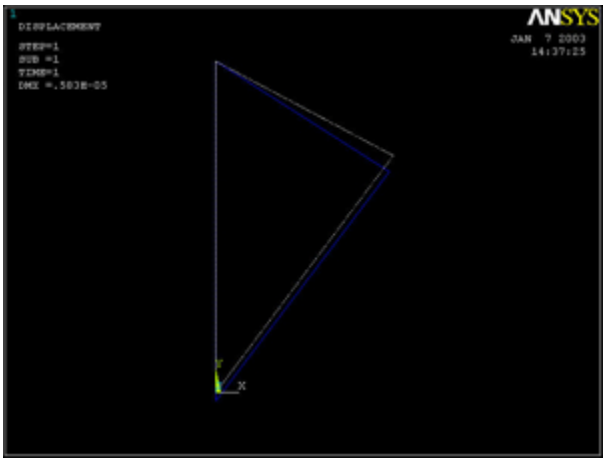
Plot Deformed Shape

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. Stop the animation and close the animation control window.

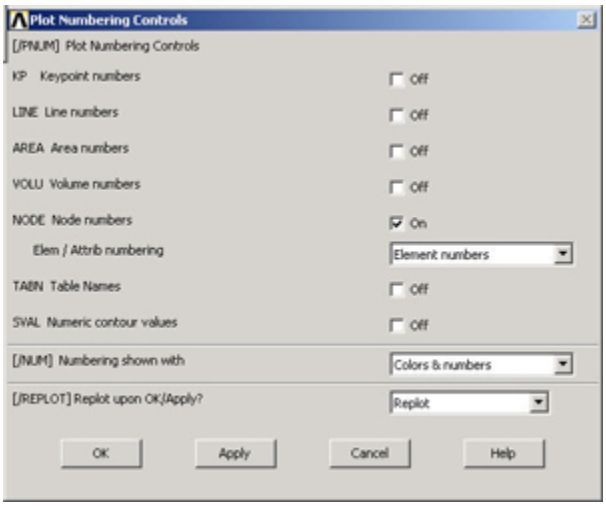
Turn On Node and Element Numbers

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

Bring up the help page for LINK1 element:

Utility Menu > Help > Help Topics

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

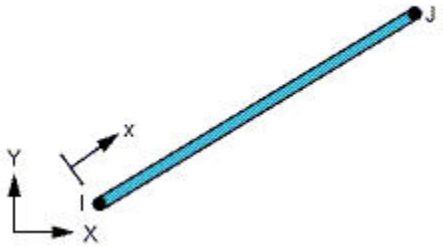
Release 11.0 Documentation for ANSYS > Elements Reference > Element Library > LINK1

In the *LINK1* help page, scroll down to *Table 1.1: LINK1 Element Output Definitions*. This table lists all the output items you can get for the LINK1 element. Let's scour this table for the element output that corresponds to the force in a truss member. We see the following item in the table that looks like a promising candidate:

MFORX: Member force in the element coordinate system X direction

The figure at the top of the *LINK1* help page, reproduced below, shows that the x-direction in the *element coordinate system* is along the element.

Figure 1.1 LINK1 Geometry



Thus, *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 *Table 1.2 LINK1 Item and Sequence Numbers* with the following entry for *MFORX*:

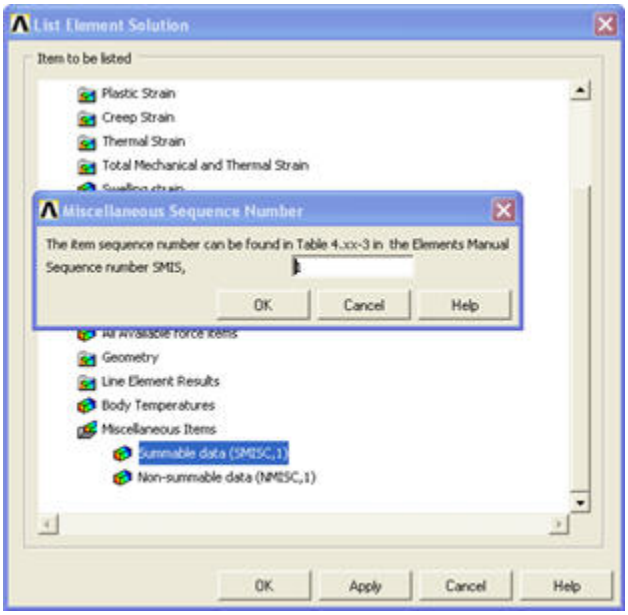
MFORX SMISC 1

The output data for any element type is broken down into item groups with *SMISC* being one of the groups (*SMISC* stands for "Summable Miscellaneous" items). Each item within a group such as *SMISC* has an identifying "sequence" number. So, as per the above entry in Table 1.2, *MFORX* is an item in the *S MISC* group with a sequence number 1. In effect, it's entry #1 in the SMISC group for the LINK1 element.

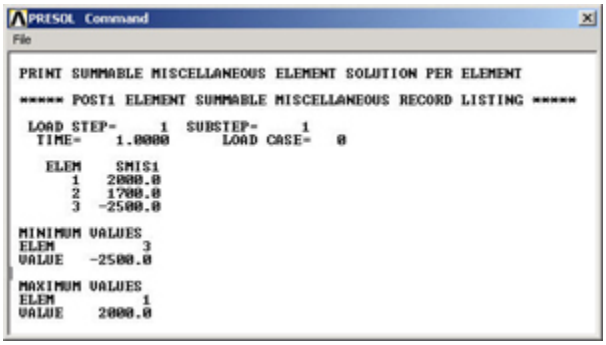
Minimize the help window. To list *MFORX* values, select:

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 ***Sequence 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.



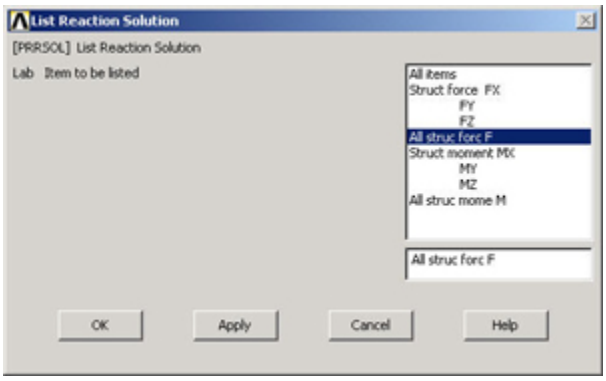
You can also *plot* the items listed under *Element Output Definitions* using the sequence number.

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.

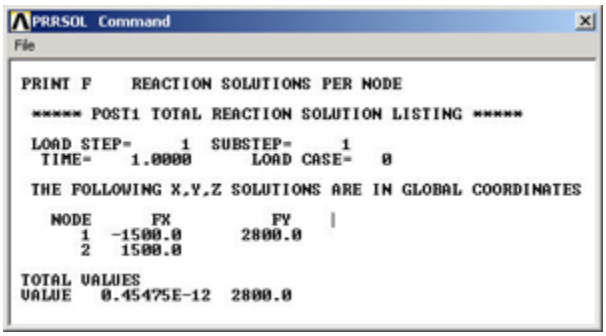
List Reaction Forces at Nodes

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.

Go to Step 9: Validate the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules