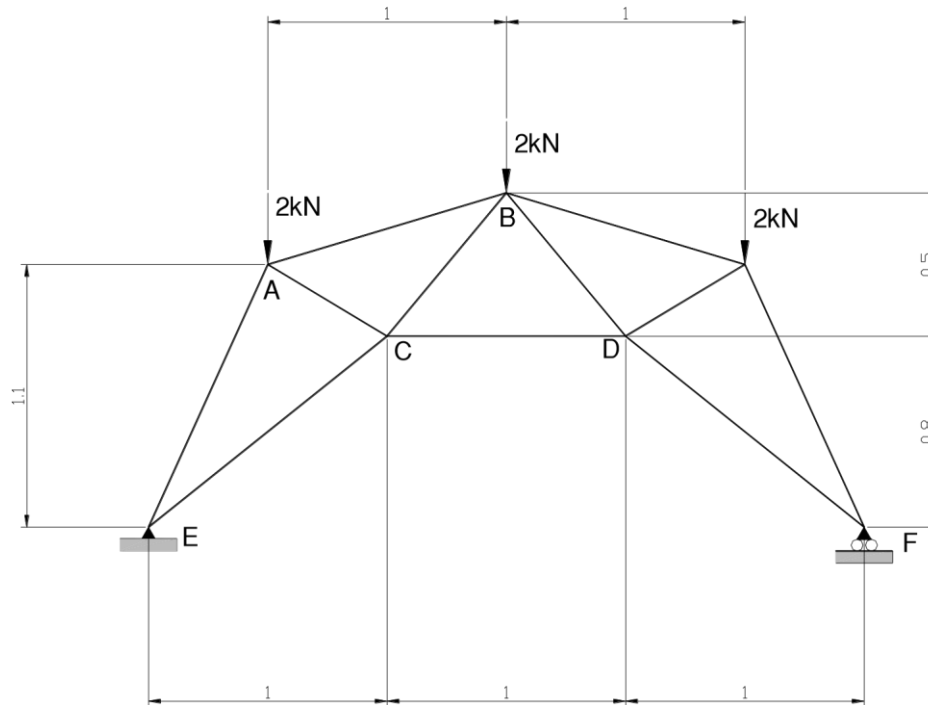


# Truss Elements

## Plane Pin-jointed Frame Example



**Determine the maximum displacement and the forces in members AC, CD and AB give that the frame is constructed from 30mm diameter bars with a Young's modulus of elasticity of  $210\text{GN/m}^2$ .**

**What are the forces in the members? Comment on the results.**

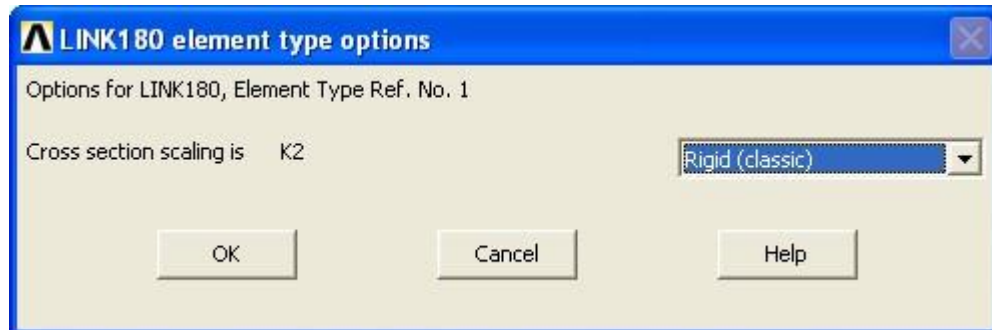
There are three stages to a finite element analysis namely Pre-processing, Processing (Solution) and Post-processing.

Pre-processing involves defining the problem geometry, materials and the element types to be used. The loads and boundary conditions may also be applied at this stage.

On the menu on the left hand side of the ANSYS window select the following:

Preprocessor

Element Type – Add/Edit/Delete – [ADD] – Structural - Link - 3D finit str 180 –  
[OK]  
[Options...] *Change Function of Stretch to Rigid (classic)*



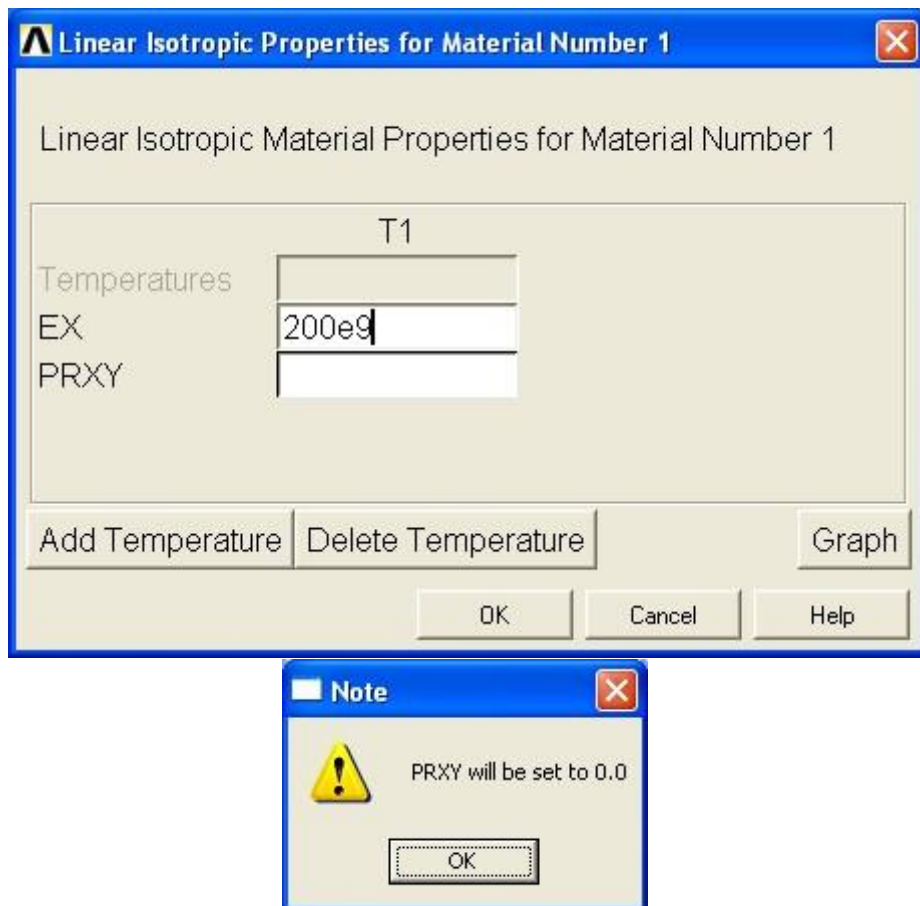
[OK]  
[Close]

The cross-sectional area of the element is added. This is done using the Section command. Please note that you **MUST** enter a value on 1 when asked for the section ID. The default Section Number when the analysis come to solve is 1.

Note that when entering the value for the area, the section can be given a name. The equation below is  $\frac{22}{7}$  approximately  $\pi$ ,  $0.03^{**2}$  is Ansys's format for  $0.03^2$  (the diameter squared).

Sections – Link - Add– Add Link Section with ID (*Enter the ID as 1*) – [OK] –  
Link Area (*Enter Area =  $\frac{22}{7} * 0.03^{**2} / 4$* ) – [OK] – [CLOSE]

Material Props – Material Models – Structural – Linear - Elastic – Isotropic –  
EX (*Enter Young's modulus of Elasticity, PRXY is not needed*) – [OK]



[OK]

From the menu bar on the *Define Material Model Behavior* dialogue box select

Material - Exit

Modelling – Create – Nodes – On working plane (*enter coordinates of each joint followed by [Enter] / [RETURN] key*).

**Create Nodes on WP**

☒ Pick    ☐ Unpick

---

Count        =        0  
Maximum      =        1000  
Minimum      =        1  
WP X          =  
Y              =  
Global X      =  
Y              =  
Z              =

---

☐ WP Coordinates  
☒ Global Cartesian

0,0

---

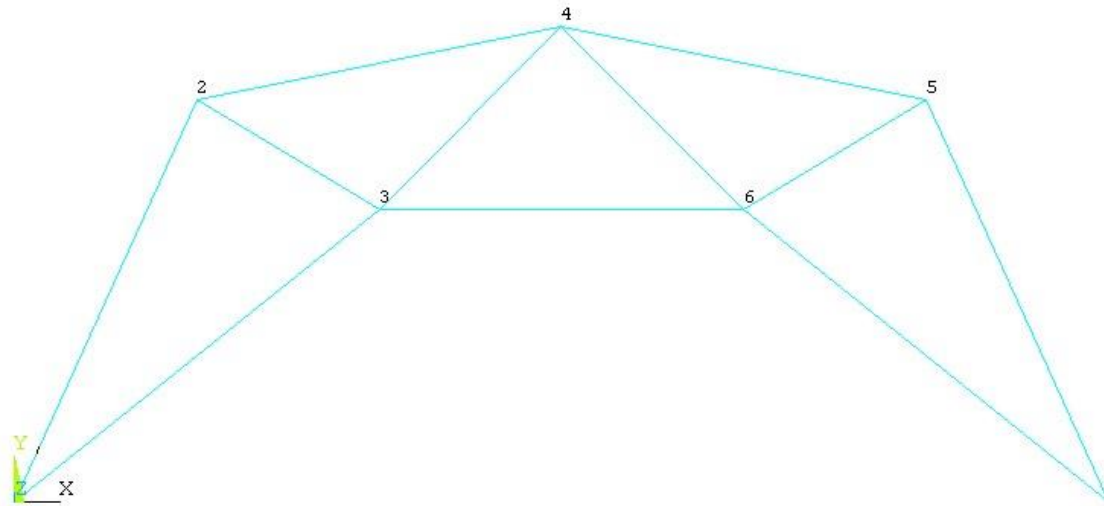
OK      Apply  
Reset    Cancel  
Help

Once all the nodes have been entered select  
[OK]

**Note: Any nodes which are specified in the wrong position can be deleted using the **Delete – Nodes** option.**

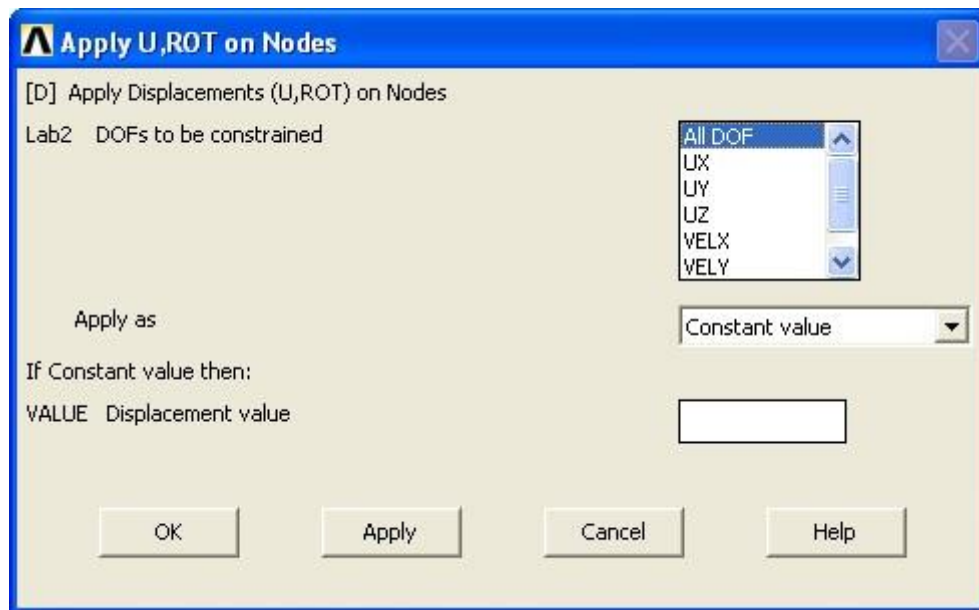
Modelling – Create – Elements – Auto Numbered – Thru' Nodes (*select node to node in the graphics window then [Enter] to form the required elements*).

The finished truss should appear as follows.



The boundary conditions or restraints are now applied. This involves defining known values of displacement at given nodes to simulate how the structure is fixed. In this case there are three sets of restraints required. The first is on the pinned joint on the right hand side which is fixed from moving in both the x and y-directions or all degrees of freedom. The second condition is at the roller joint, which restricts movement in the y-direction only. Lastly, all the nodes need to be restrained from moving in the z-direction as these elements are defined in three-dimensional space. Note that the displacement is zero in these cases; this is the default value and does not need to be specified. Non-zero displacements can be specified by entering the value of the displacement and defining the direction with the sign (plus or minus).

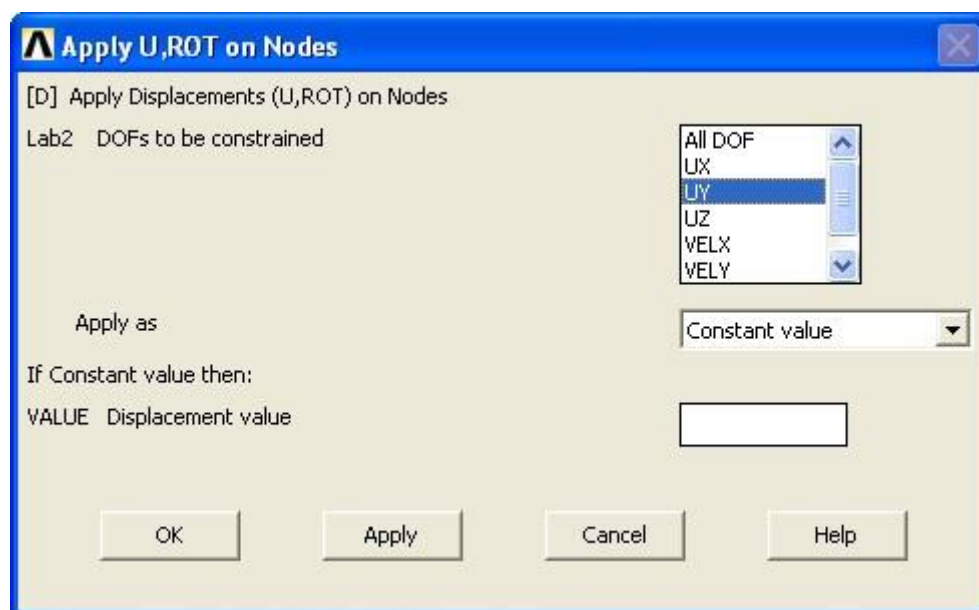
**Loads – Define loads - Apply- Structural -Displacement – On Nodes** (*Select the node in the graphics window that is fully restrained i.e node 1 at origin*)



Select ALL DOF

[OK]

*Repeat the command – choose the right most node which is shown as being on a roller. Deselect ALL DOF and then select UY.*



[OK]

*Repeat the command – Choose all nodes (bottom left box in Apply U,ROT on Nodes dialogue box)*

**Apply U,ROT on Nodes**

☒ Pick      ☐ Unpick

---

☒ Single      ☐ Box  
☐ Polygon      ☐ Circle  
☐ Loop

---

Count      =      0  
 Maximum    =      7  
 Minimum    =      1  
 Node No.   =

---

☒ List of Items  
☐ Min, Max, Inc

---

*Deselect UY and select UZ.*

[OK]

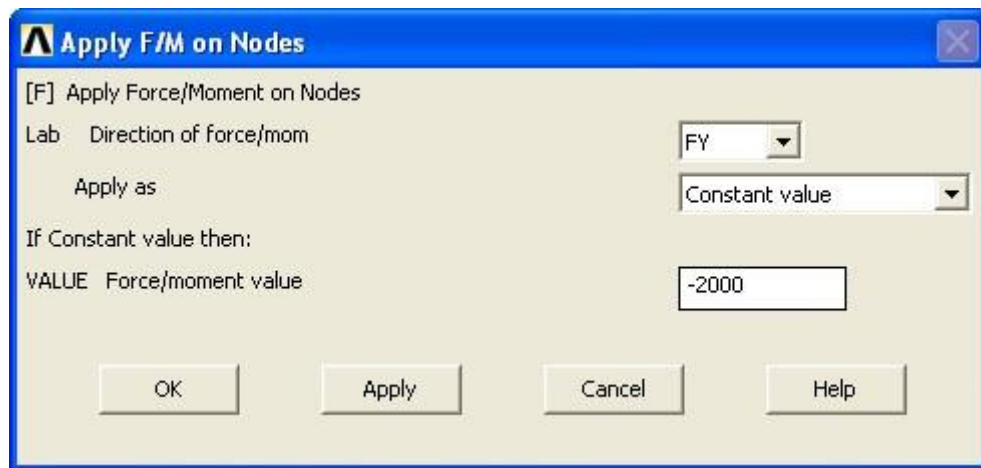
The loads are applied to the model. The applied loads should reflect the actual physical loading. In this case there is a load applied at three joints, which is identical and can therefore be applied in a single command.

Loads – Define loads - Apply – Structural - Forces/Moments – On Nodes

*(Select the nodes to which load is applied, enter value of load i.e. the three top most nodes.)*

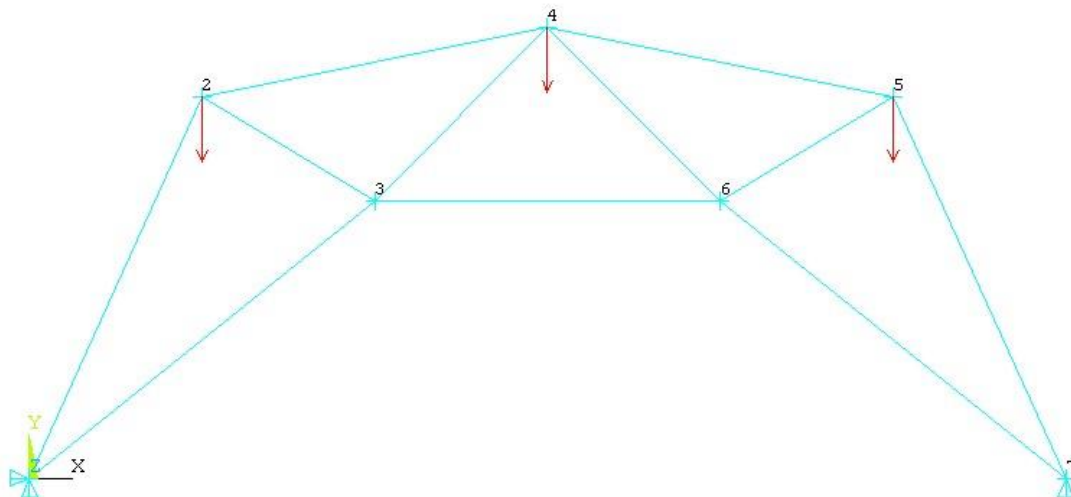
*Change Lab from FX to FY*

*Value -2000 (minus indicates in the negative y-direction i.e. downwards)*



[OK]

The truss should appear as flows on the screen.



## Solution

Processing or Solution is the stage where the analysis type is defined along with the options for the solution. In this case all the default options are acceptable. The problem can be solved.

Solution

Solve – Current LS ( $LS = Load\ step$ )

[OK]

[CLOSE] - to “Solution is done” dialogue box.

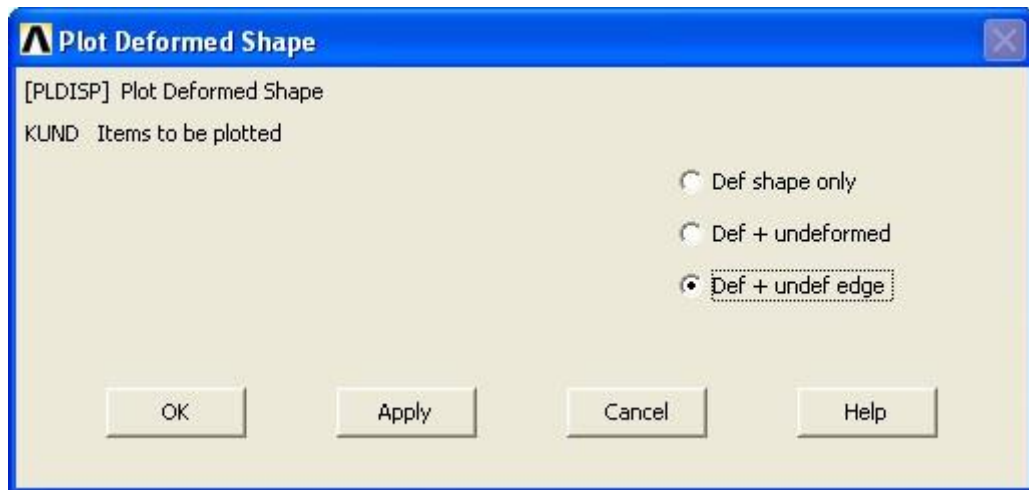


## Post-processing

Post-processing is the stage where the results are reviewed. The first result to check is the displacement.

### General Postproc

#### Plot results – Deformed Shape

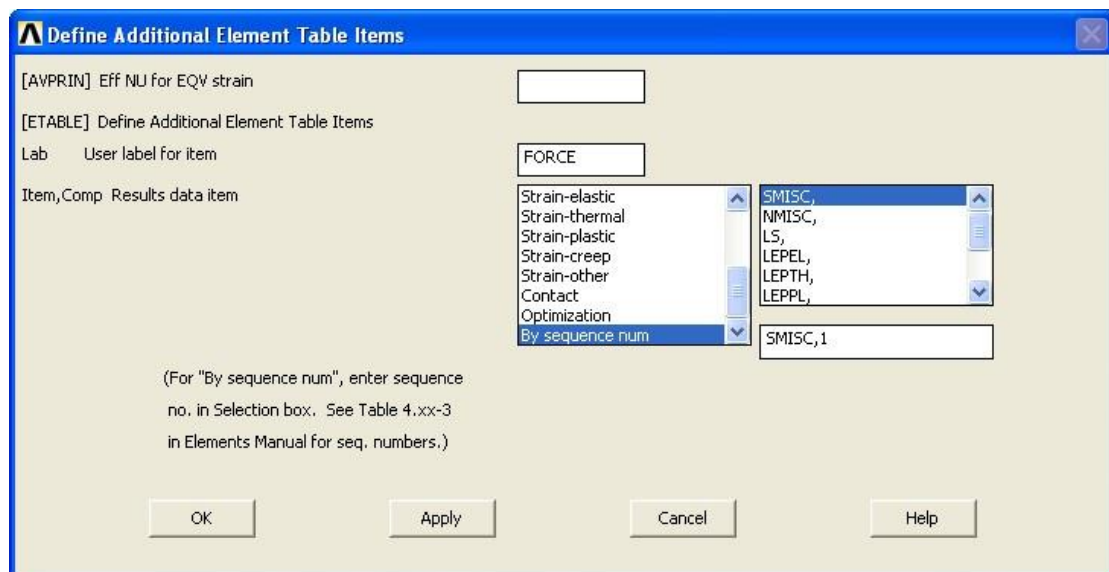


To plot the element forces:

#### Element Table – Define Table – [Add...]

##### Lab – FORCE

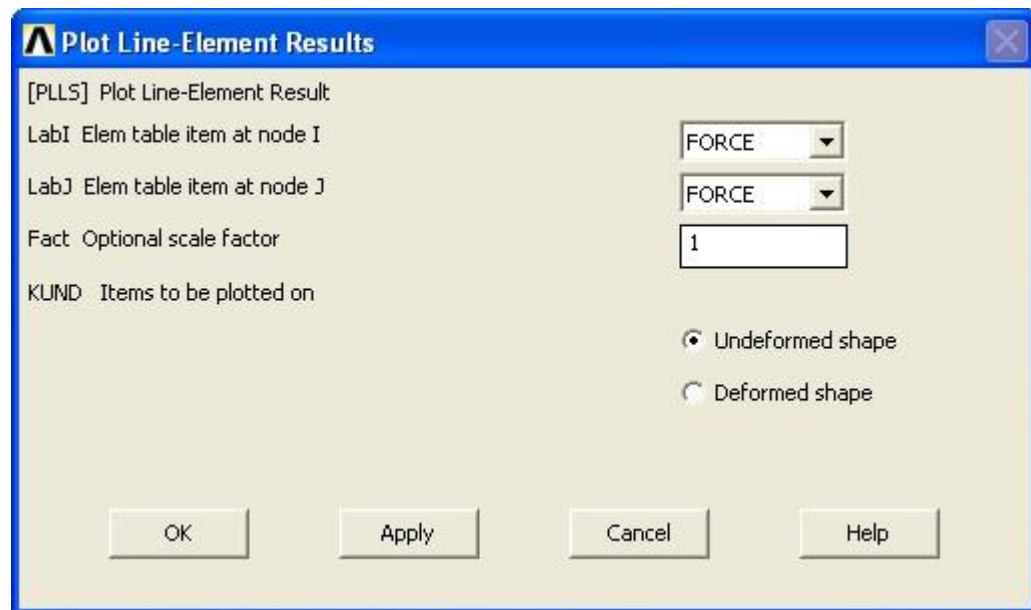
##### Item, Comp ... - By sequence num – SMISC,1



[OK]

[CLOSE]

### Plot Results – Contour Plot – Line Elem Res



The image shows a software dialog box titled "Plot Line-Element Results". It contains several input fields and options for configuring a contour plot. The fields are: "LabI Elem table item at node I" with a dropdown menu showing "FORCE"; "LabJ Elem table item at node J" with a dropdown menu showing "FORCE"; "Fact Optional scale factor" with a text box containing "1"; and "KUND Items to be plotted on" with two radio button options: "Undeformed shape" (which is selected) and "Deformed shape". At the bottom of the dialog are four buttons: "OK", "Apply", "Cancel", and "Help".

[PLLS] Plot Line-Element Result	
LabI Elem table item at node I	FORCE
LabJ Elem table item at node J	FORCE
Fact Optional scale factor	1
KUND Items to be plotted on	<input checked="" type="radio"/> Undeformed shape <input type="radio"/> Deformed shape
OK Apply Cancel Help	

[OK]

The results should be checked to ensure that the analysis has been carried out correctly. Once this has been done then recommendations can be made regarding the design.