

ANSYS Beginner Space Frame Demo (draft 3, 11/27/05)

Problem description

To verify that you understand the somewhat tricky input features of a space frame (general beam bending, torsion, and axial load) it is wise to start with a published solution. An example from [1] is a three member spatial “Z” shaped space frame that is fixed at both ends, and loaded with a vertical point load at one of the two free nodes. That simple system is illustrated in Figure 1. The top portion of the figure presents the desired orientations of the (assumed) rectangular cross-section of each of the three members. The bottom portion displays the usual line drawing representation used by most finite element systems for line elements. The data to be given below were simply taken from reference [1].

The beam is assumed to have a rectangular cross-section. Thus, the only tricky part of the study is to assure that the principle inertia axes of each rectangular beam section are properly orientated in 3D space. The desired member placement is seen in the top of \$ Figure while the usual line element format is in the bottom of that figure. The single line format can let you overlook a beam orientation error and possibly lead to seriously incorrect results.

Each beam has a local orthogonal coordinate system associated with it. The local x-axis runs from the first node to the second. It is used to compute the local torsional inertia, I_{XX} . The local cross-section bending inertias, I_{YY} and I_{ZZ} , are input relative to the local y- and z-axes, respectively. This is not a problem when the beam has a circular cross-section since $I_{YY} = I_{ZZ}$ for any choice of the local z-axis. That is not true for non-circular sections, so the local z-axis must be carefully established for each member (then the y-axis is computed by the “right hand rule”). The local z-axis can be defined in two ways in ANSYS. Here the method of using a third node will be employed. The three nodes define the local x-z plane, and therefore the cross-sectional axis used by you to compute I_{ZZ} . Knowing that, the I_{YY} value is calculated with respect to the remaining orthogonal cross-section axis.

When selecting candidates for use as the third node it is usually preferred to pick an existing structure node. On some occasions you have to introduce a (full restrained) non-structural node simply to use as a geometric reference point. Ideally you could find structural nodes that make it easy to keep I_{YY} and I_{ZZ} constant for a group of elements. Consider the first member in this system. It happens to go from node 1 to node 2. Try to pick a third node (from nodes 3, 4, or 5) that will allow the same I_{YY} and I_{ZZ} values to be used in other elements. In Figure you can see that the plane formed by nodes 1-2-3 allow the first two members to share the same maximum and minimum local inertia axes, but the use of structural node 4 or geometry node 5 do not. Therefore, pick node 3 to define the local z-axis for member 1, as seen in Figure . Similarly, in Figure , the second member, along line 2-3 is best, or more consistently, oriented by picking node 1 as its third node. For the last member, you have to make a choice between keeping the same I_{YY} and I_{ZZ} values (i.e., same set of real constants) by creating node 5 as a non-structural node, or picking structure node 2 and reversing the I_{YY} and I_{ZZ} values. In other words, picking node 2, which is done here, will

require a second set of real constants, and remembering when to activate that set. You will be reminded of these choices when the actual data are input.

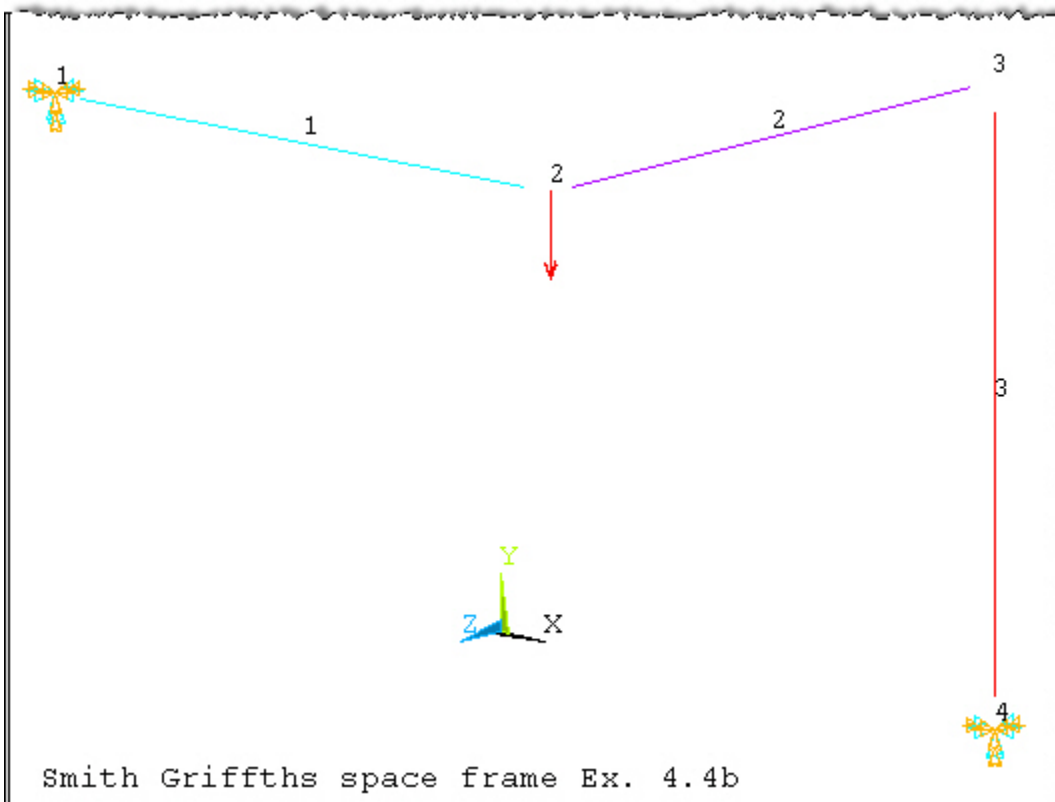
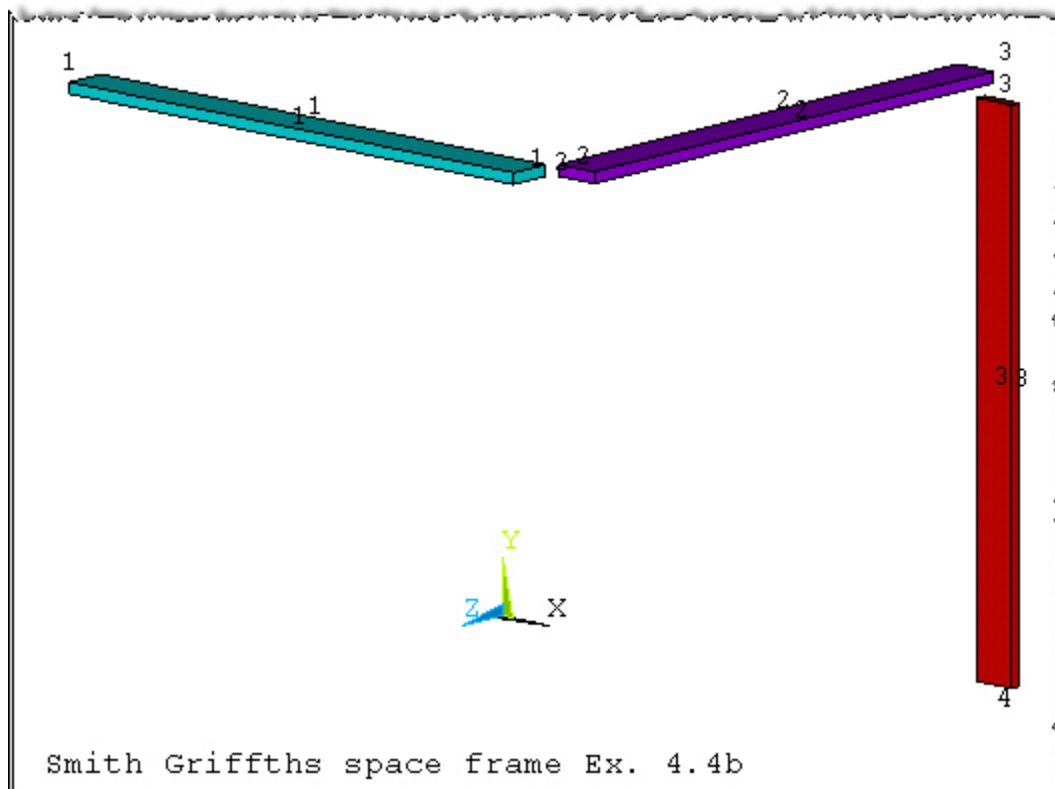


Figure 1 A three member space frame with a vertical load

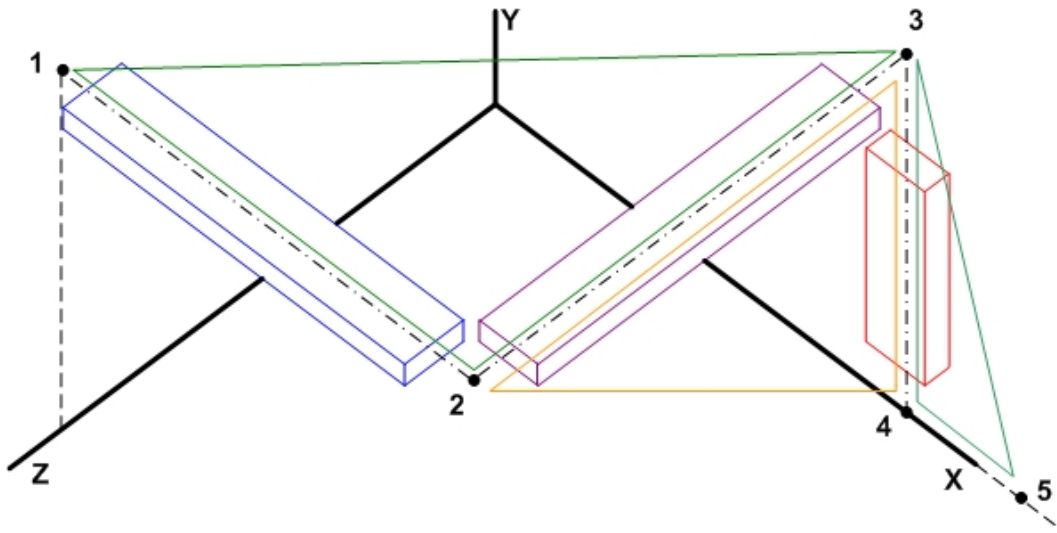


Figure 2 Options for members local x-z planes (triangles)

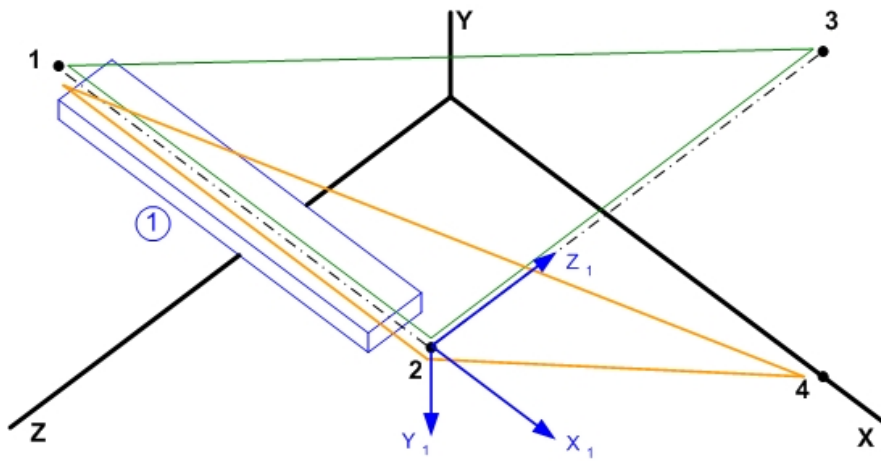


Figure 3 Insert local axes for the first member (plane 1-2-3)

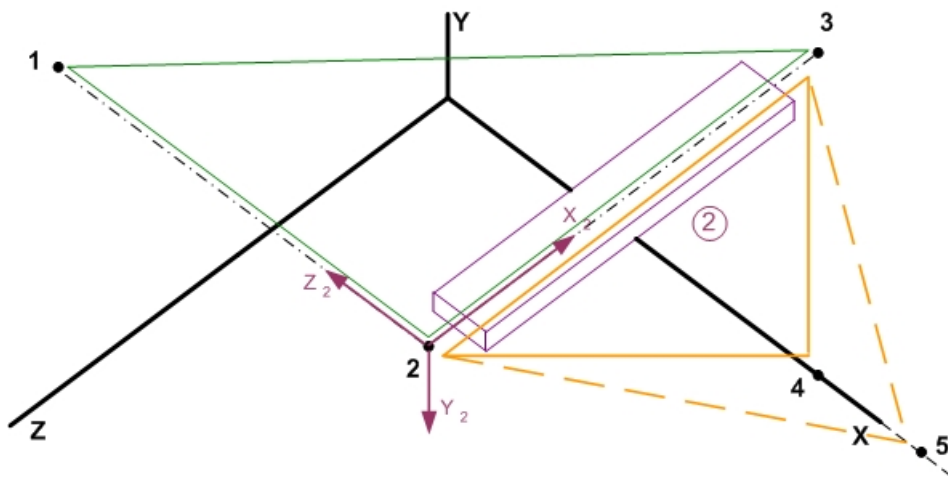


Figure 4 Choice for second member local axis (plane 2-3-1)

Start ANSYS

ANSYS is short for **AN**alysis **SY**stem. That pretty much sums up its capabilities. Most areas of engineering calculation are included: stress, thermal, fluids, dynamics, vibrations, frequency analysis, acoustics, electromagnetism, optimization, etc. are present and usually have non-linear abilities as well. ANSYS was first released in 1971 and has generally been one of the most popular finite element systems since then, worldwide. Since it has a huge list of abilities its menu system can get relatively long. That is also true because it (and all FEA systems at that time) was run in a “batch mode” on the largest available computers. In other words, you could, and still can, execute ANSYS through an input text file without using a Graphical User Interface (GUI). Once you are fully experienced with the code that can be a real time saver. Every GUI session is saved in that format so that you can simply edit it with a text editor and make small changes for a similar problem.

Begin ANSYS with **Start→All Programs→ANSYS 10.0→ANSYS**. That will bring you to the main ANSYS **Utility Menu** as seen here in \$.

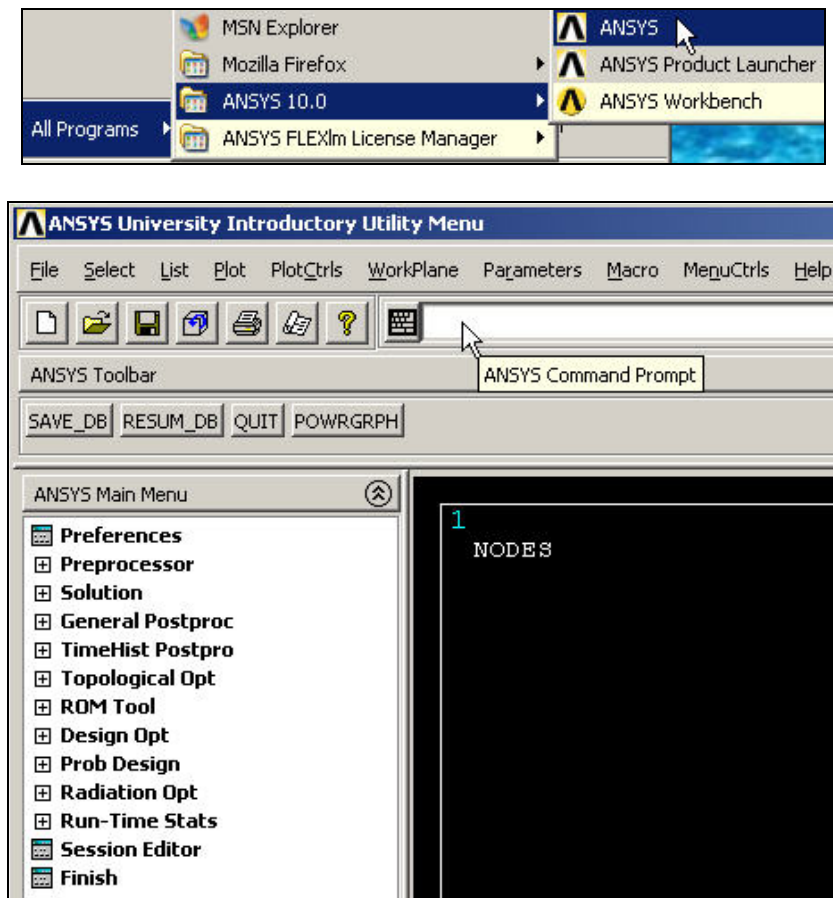



Figure 5 Opening ANSYS to the Utility Menu and graphics window

Select job name and analysis type

The various menus below will sometimes get moved to a back (hidden) window. If you think that has occurred hit the **Raise Hidden** button, . You will always need a job name:

1. **Utility Menu**→**File**→**Change Jobname**.
2. **Change_Jobname**, type in the new name, **OK** (as seen \$).

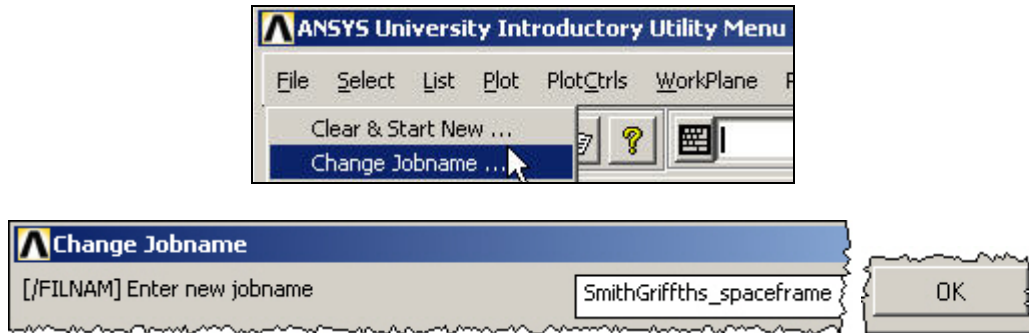


Figure 1 Setting the new job name

The ANSYS file sizes for real engineering problems get to be quite large, so have a directory dedicated to ANSYS:

1. **Utility Menu**→**File**→**Change Directory**.
2. **Browse for Folder**→**Change Working Directory**, pick your directory (ANSYS_dir in Figure 2), **OK**.

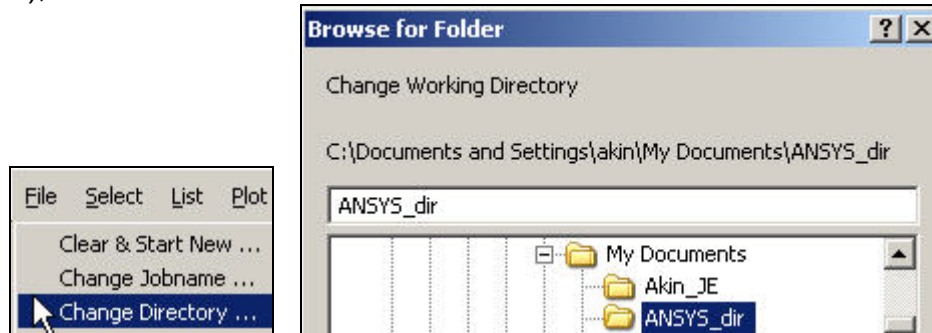


Figure 2 Establish a directory for the analysis files

To keep up with your analysis studies over time create descriptive titles:

1. **Utility Menu**→**File**→**Change Title**.
2. **Change Title**, enter a descriptive title, **OK** (see Figure 3).

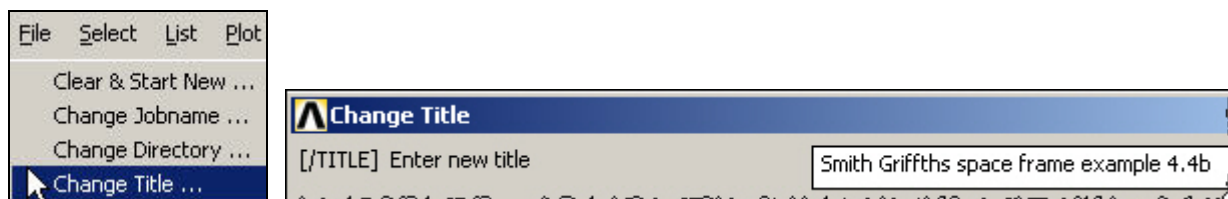


Figure 3 Provide a descriptive title

Element type data

You will need a 3D structural line element, or frame member, that transmits axial forces, axial torsion, and transverse bending moments. Open and add to element types you need:

1. **Main Menu**→**Preferences**→**Preferences for GUI Filtering**.
2. Check **Structural**, accept default **h-Method**, **OK**.
3. **Main Menu**→**Preprocessor**→**Element Type**→**Add/Edit/Delete**, as in Figure 4.
4. In **Element Types**, seen in Figure 5, pick **Add**→**Library of Element Types**.
5. Select (Structural) **Beam** and **3D elastic 4** (that is, beam4), **OK**.
6. In **Element Types** pick **Close**.

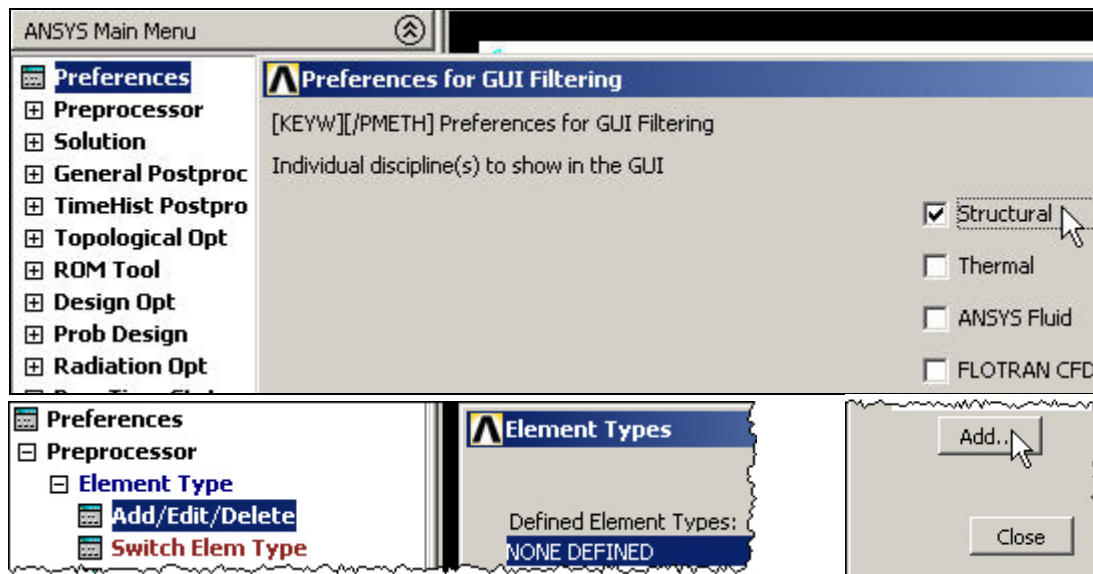


Figure 4 Declare the selection of a structural analysis

The Beam4 element is 3D and by default is used in the X,Y,Z space and usually requires planning to orientate the local second area moment of inertia axes, IYY and IZZ.

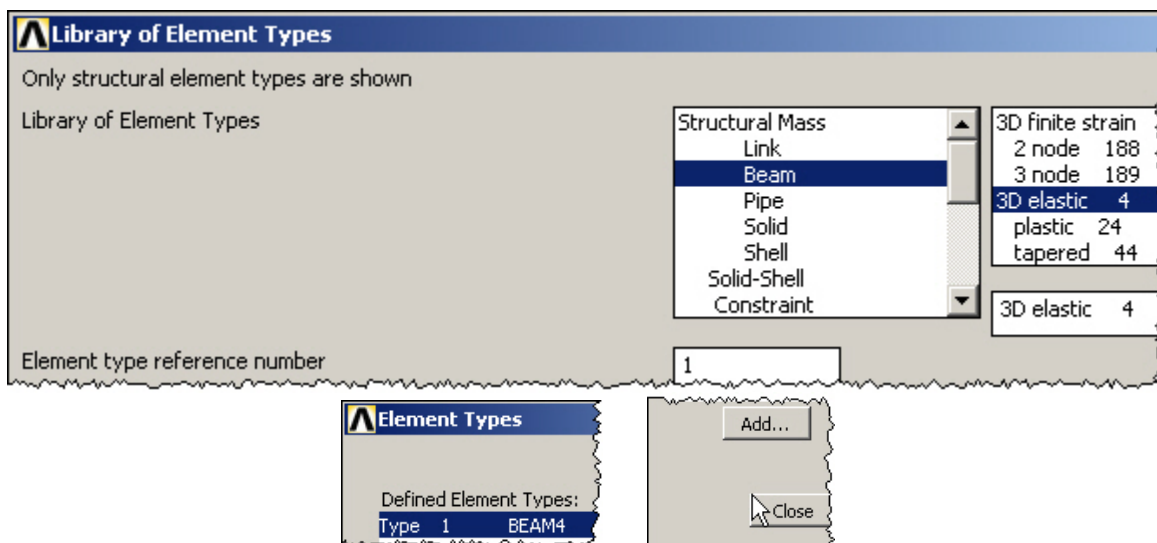


Figure 5 Select space frame element type

Element geometric properties data

Every element type requires one or more real constants, like area or moment of inertia, to describe it. Here, you need several geometric items for each type of beam. You must enter at least one: IYY (although giving just the area makes it a truss member, which can be a useful trick sometimes). Paraphrasing from the ANSYS manual on the BEAM4 element:

“The geometry, node locations, and coordinate systems for this element are shown in Figure 6]. The element is defined by two or three nodes, the cross-sectional area, two area moments of inertia (IZZ and IYY), two thicknesses (TKY and TKZ), an angle of orientation (Θ) about the element x-axis, the torsional moment of inertia (IXX), and the material properties. ... If IXX is not specified or is equal to 0.0, it is assumed equal to the polar moment of inertia (IYY+IZZ). IXX should be positive and is usually less than the polar moment of inertia. The element torsional stiffness decreases with decreasing values of IXX. ... “

The element x-axis is oriented from node I toward node J. For the two-node option, the default ($\Theta=0^\circ$) orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in [Figure 6]. For the case where the element is parallel to the global Z axis (or within a 0.01 percent slope of it), the element y axis is oriented parallel to the global Y axis (as shown). For user control of the element orientation about the element x-axis, use the Θ angle (THETA) or the third node option. If both are defined, the third node option takes precedence. The third node (K), if used, defines a plane (with I and J) containing the element x and z axes (as shown).” [End paraphrasing.]

For circular sections $IYY = IZZ$ and you just need to locate the two end nodes to define the local x-axis in space. But in the common case where IYY and IZZ are different you must exercise care in using the third node to define the local z-axis. That is, the local x-z plane coincides with the plane that contains the line (beam) element and the third (orientation) node, K. The local IZZ is relative to that local z-axis and the IYY is defined relative to the local y-axis which is defined by the “right hand rule” from the local x- and z-axes. *Experience shows that this is the most common user error in employing 3D beam elements.*

The selection of the third node occurs later, when the element connectivity is picked in the graphics area, or read fro a file. Sometimes it is useful to have a few nodes just for orientation use, if the needed structure nodes do not do that. If that approach is used they should be completely restrained to avoid any potential problems in the equation solver.

To enter the various real constants to be referenced by the element types:

1. **Main Menu**→**Preferences**→**Real Constants**→ **Add/Edit/Delete**.
2. **Real Constants**→ **Add**→**Element Type for Real Constants**.
3. Choose element type: **Type 1 BEAM4, OK**, as in Figure 7.
4. In **Real Constant Set Number 1**, for **BEAM4** verify **Set No. 1**.
5. Enter 4.0 (m²) for **Cross-sectional area (AREA)**; 1.0 (m⁴) for **IZZ**, 0.3 (m⁴) for **IYY**, and 0.3 (m⁴) for **IXX** (see Figure 8 left). The thicknesses (width and depth) of the beam cross-section were not given in the problem statement. Thus, assume

values of 0.4 (m) and 0.1 (m) in the z- and y-directions for TKZ and TKY, respectively, to help illustrate a non-circular cross-section (like that in Figure 1), OK.

6. **Set 1** appears in **Real Constants**. Select **Add**,
7. In **Real Constant Set Number 2**, for **BEAM4** verify **Set No. 2**.
8. Reverse the y- and z- values used in step 5. Enter 4.0 (m²) for **Cross-sectional area (AREA)**; 1.0 (m⁴) for **IYY**, 0.3 (m⁴) for **IZZ**, and 0.3 (m⁴) for **IXX**, 0.1 (m) for **TKZ** and 0.4 (m) for **TKY** (see Figure 8 right), **OK**.
9. **Set 2** appears in **Real Constants**. Select **Close**.

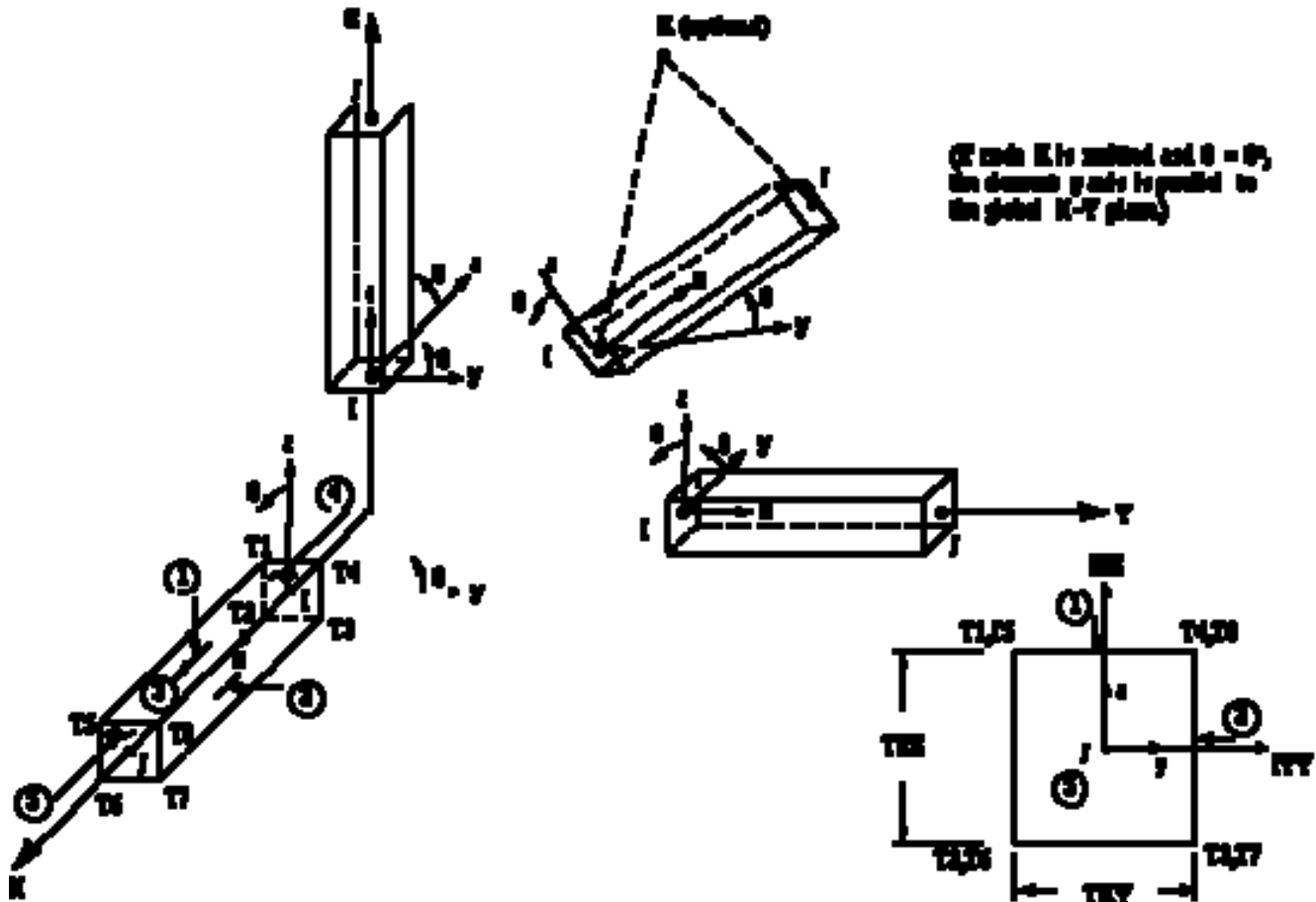


Figure 6 Beam4 orientation data from ANSYS manual

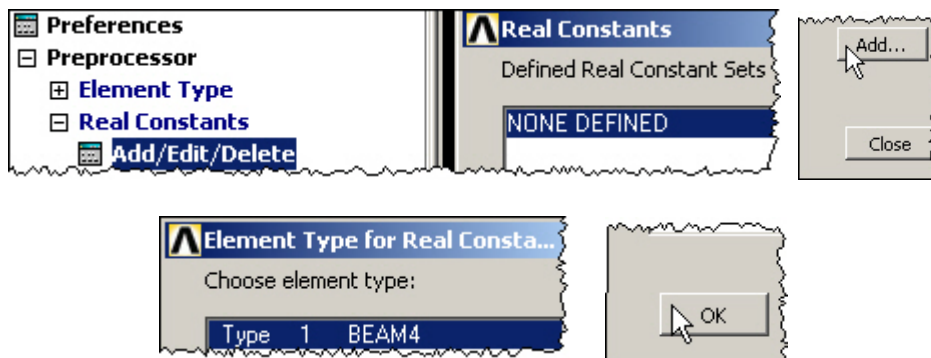


Figure 7 Starting the first set of real constants

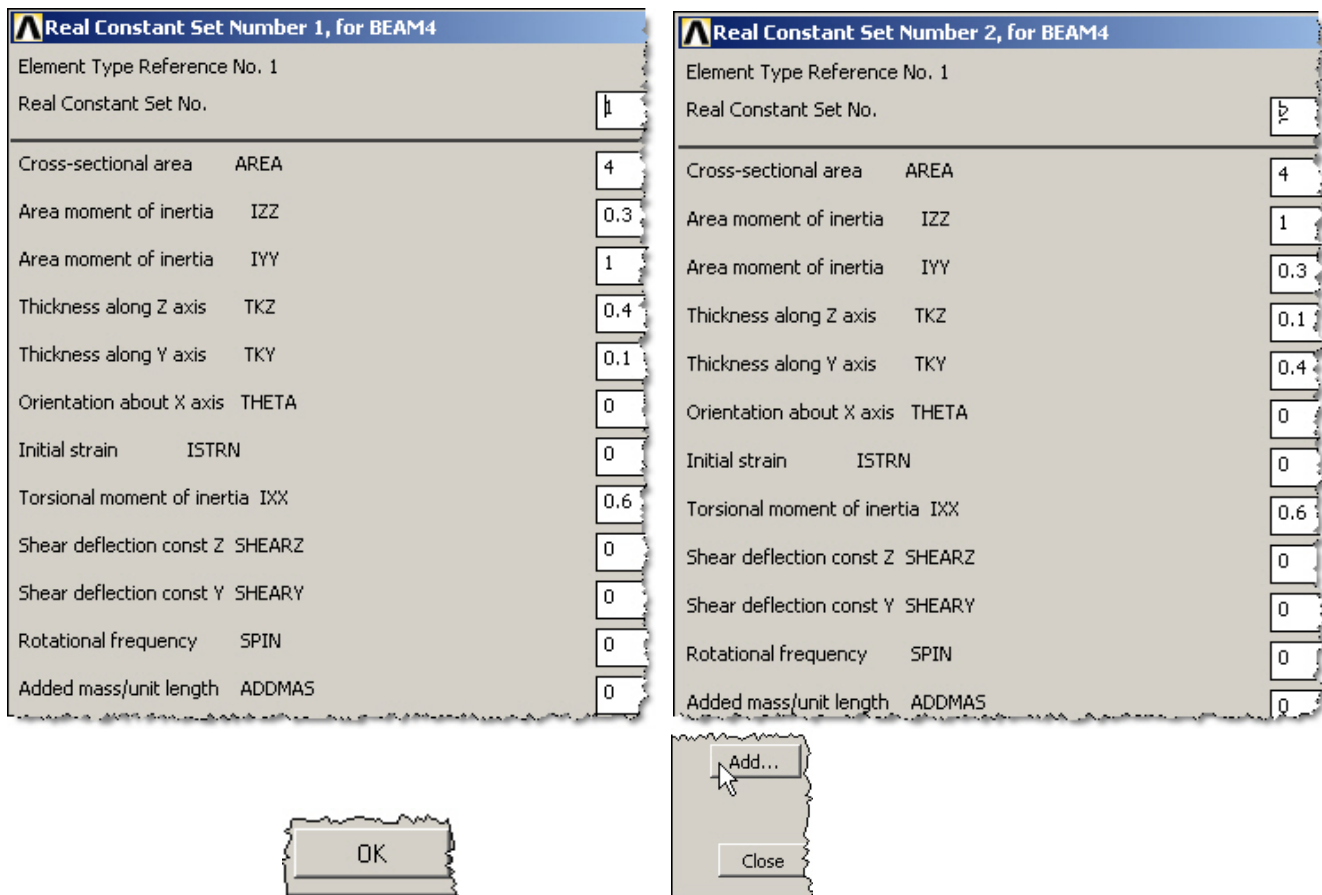


Figure 8 Real constants for beams 1 and 2, and for beam 3, respectively

Note that there are 12 items for each real constants set. The same data can be entered through a text file, as seen at the end of this example, but only 6 items are allowed on each of two lines of input.

One use of the attributes for the Thickness in the Z (TKZ) and Y (TKY) directions is to plot the approximate shape of each member in its local coordinate system, as seen in Figure . To create such a plot:

1. **Utility Menu**→**PlotCtrls**→**Style**→**Size and Shape ...**
2. **Size and Shape**→ **Element SHAPE** (ESHAPE) on. **Shrink** if desired (Figure 9).

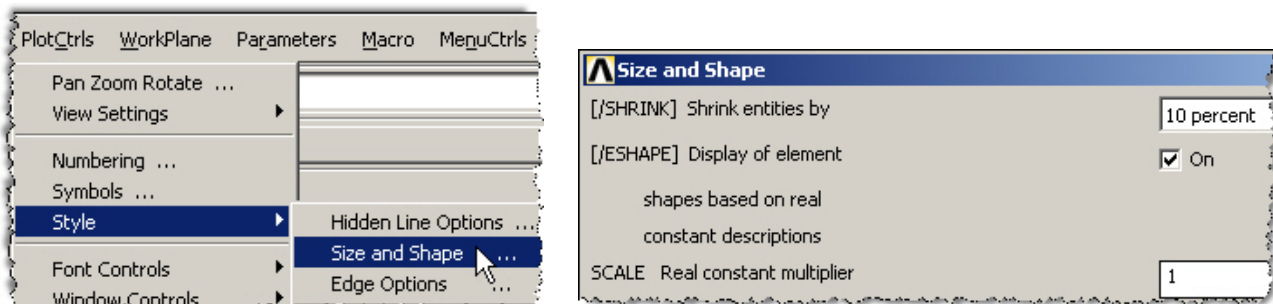


Figure 9 Displaying element shapes

Define member material properties:

Here you will use the simplest linear, isotropic, 1D material description. ANSYS has full anisotropic (completely directionally dependent), as well as non-linear material “constitutive laws”. Activate the material properties with:

1. **Main Menu**→**Preprocessor**→**Material Props**→**Material Models**.
2. Material Model Number 1 appears in **Define Material Model Behavior**.
3. Double click on **Structural**, then **Linear**, then **Elastic**, then **Isotropic**.
4. In **Linear Isotropic Properties for Material Number 1** enter 1.e6 (kN/m²) for elastic modulus, **EX**, and 0.0 for Poisson’s ratio (**PRXY**), as in Figure 10, **OK**.
5. **Close** (X) the Define Material Model Behavior window.

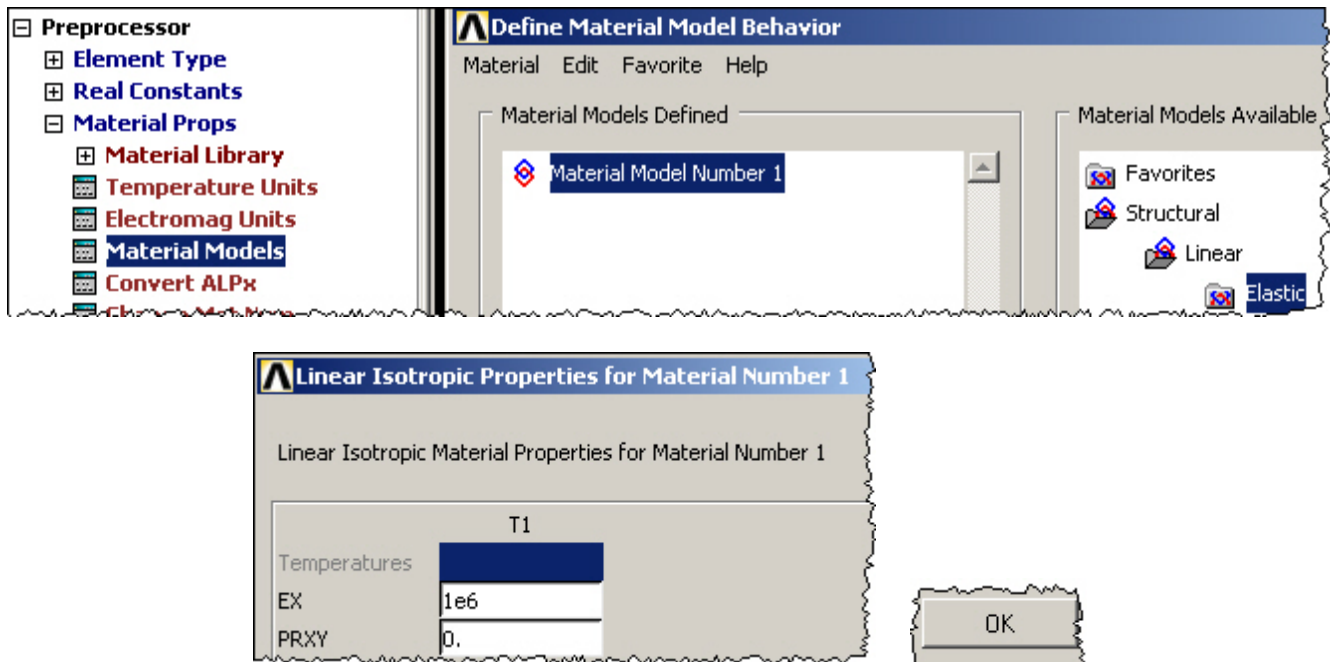


Figure 10 Define the isotropic elastic properties

Define nodal data:

Of course, ANSYS has powerful mesh generation capabilities. However, for beginners or small problems with only a few nodes you can type in the coordinates, or use cursor input via the graphics window, or read them from a file. Use the first approach:

1. **Main Menu**→**Preprocessor**→**Modeling**→**Create**→**Nodes**→**In Active CS**.
2. In **Create Nodes in Active Coordinate System** of Figure 11 enter 1 for **Node number**, **X = 0.**, **Y = 5.**, **Z = 5 (m)**, **Apply**, enter 2 for **Node number**, **X = 5.**, **Y = 5.**, **Z = 5.**, **Apply**, enter 3 for **Node number**, **X = 5.**, **Y = 5.**, **Z = 0.**, **Apply**, enter 4 for **Node number**, **X = 5.**, **Y = 0.**, **Z = 0.**, **Apply**.
3. You can also add a non-structural node for beam orientation purposes. If so, enter 5 for **Node number**, **X = 5.**, **Y = 0.**, **Z = 5.**, **Apply**, **OK**.
4. If you make a mistake you can return and correct it in the above window, or delete them all with **Preprocessor**→**Modeling**→**Delete**.

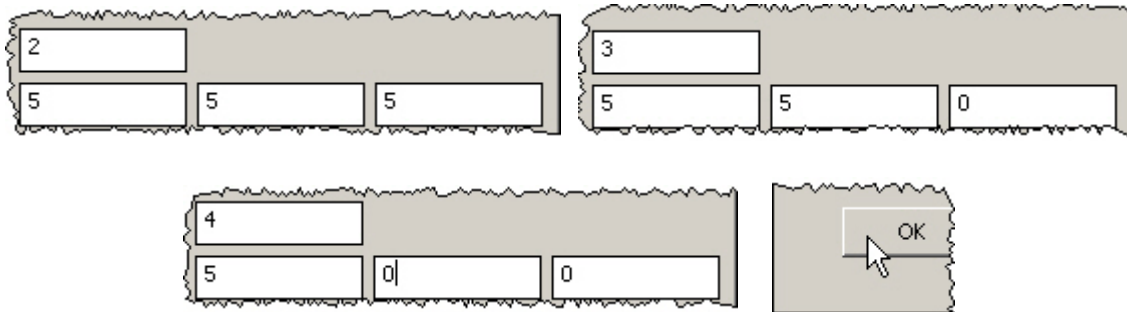
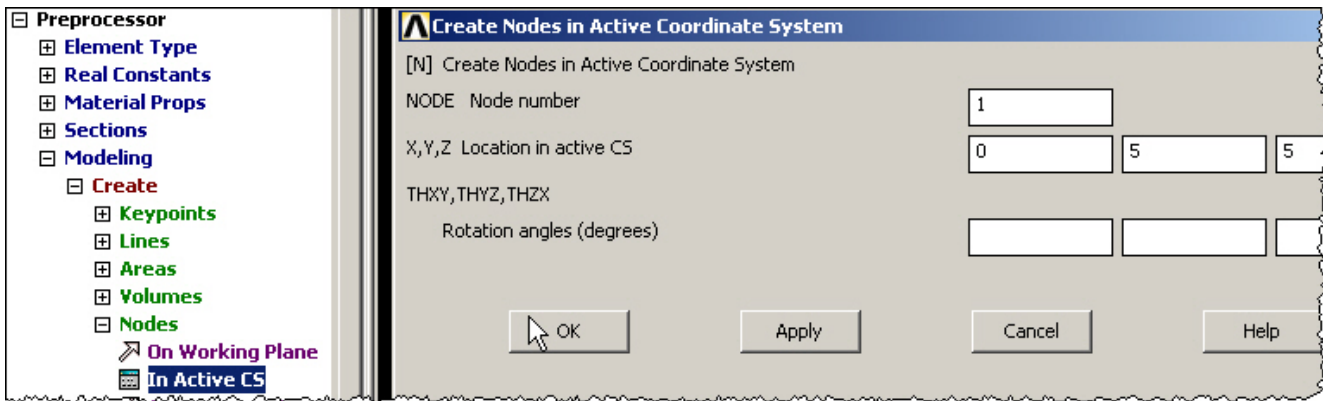


Figure 11 Build the list of nodal coordinates

Now, plot the nodal values input (node 1 may be hidden behind the axis symbol):

1. **Utility Menu**→**PlotCtrls**→**Numbering**.
2. In **Plot Numbering Controls** check **node numbers** and select **element numbers** in the pull down menu, **OK**.
3. **Utility Menu**→**PlotCtrls**→**Numbers** and review the plot that is similar to Figure .
4. To get the reverse video white background of that figure use **PlotCtrls**→**Style**→**Color**→**Reverse Video**.

Define element attributes

Next you have to associate each of the elements with your previous material numbers and real constant sets. Plan ahead and input those elements of the same type in sequence:

1. **Main Menu**→**Preprocessor**→**Modeling**→**Create**→**Elements**→**Elem Attributes**.
2. In **Element Attributes** select defaults (type number = 1, material number = 1, real constant set = 1), **OK**.
3. **Main Menu**→**Preprocessor**→**Modeling**→**Create**→**Elements**→**Auto numbered**→**Thru Nodes**, as in Figure 13.
4. In **Elements from nodes** verify that **Pick** is checked on.
5. In the graphics window, seen in Figure 14, define the first element by picking **node 1** (a square symbol appears) then **node 2**, then orientation **node 3**, **OK**, and the next element number (1) appears in the graphics area.
6. Repeat this process for the last member sharing the current real constants set: **Thru Nodes**, pick **nodes 2, 3**, and **1** for element 2 (bottom of Figure 14).

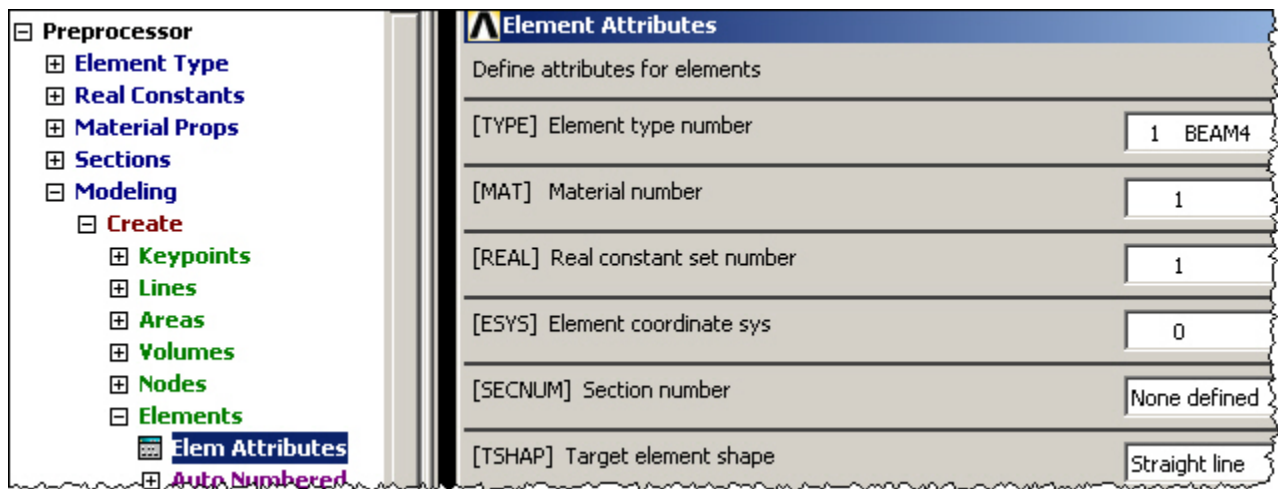


Figure 12 Confirm the beam attributes for the first two members

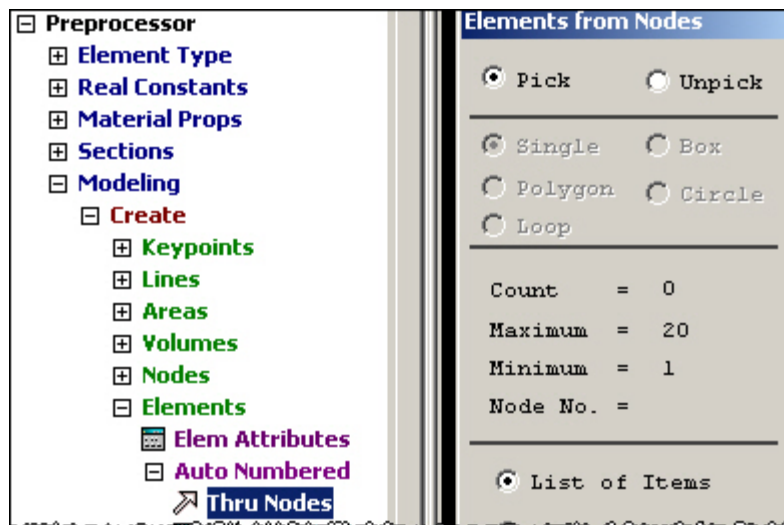


Figure 13 Prepare to graphically pick the element connectivity

For the next (and last) element you need to change the number of its real constants data set from 1 to 2. If those data were still correct but the next element was made of a different material then a similar change would be required at this point. To accomplish the change, seen in Figure 15, use:

1. **Preprocessor**→**Modeling**→**Create**→**Elements**→**Elem Attributes**.
2. In **Element Attributes** change the **Real constants set number** from 1 to 2.

If you are using text file input rather than the GUI then a similar change (REAL,2) is needed.

Now you can continue with picking the nodes for the final member in the mesh.

1. Repeat the previous connectivity definition process.
2. **Thru Nodes**, pick **nodes 3, 4, and 2** for element 3, as illustrated in Figure 16.

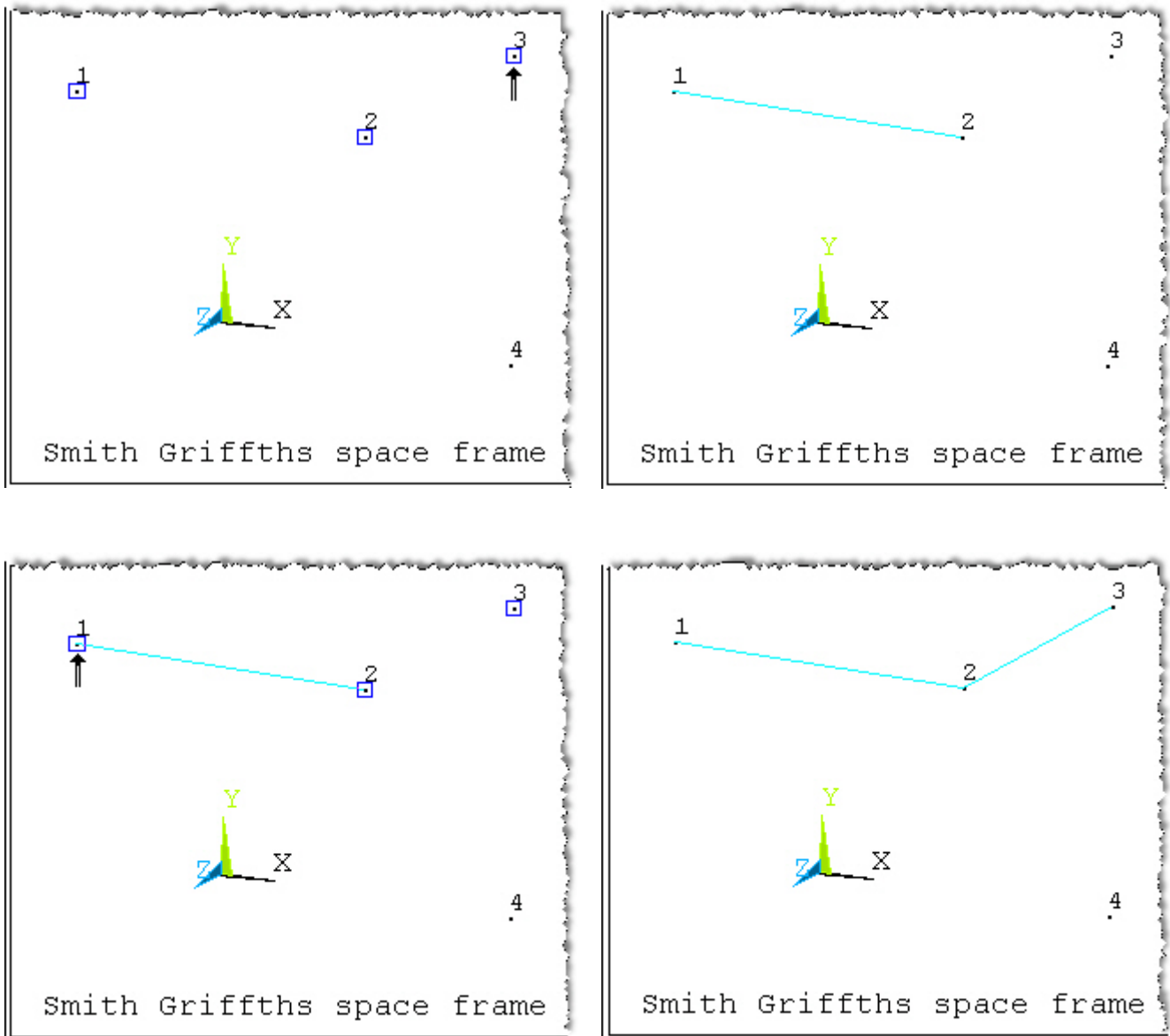


Figure 14 Define two elements for first real constants set

| Element Attributes | |
|---------------------------------|---------|
| Define attributes for elements | |
| [TYPE] Element type number | 1 BEAM4 |
| [MAT] Material number | 1 |
| [REAL] Real constant set number | 2 |

Figure 15 Set new real constants set for next element(s)

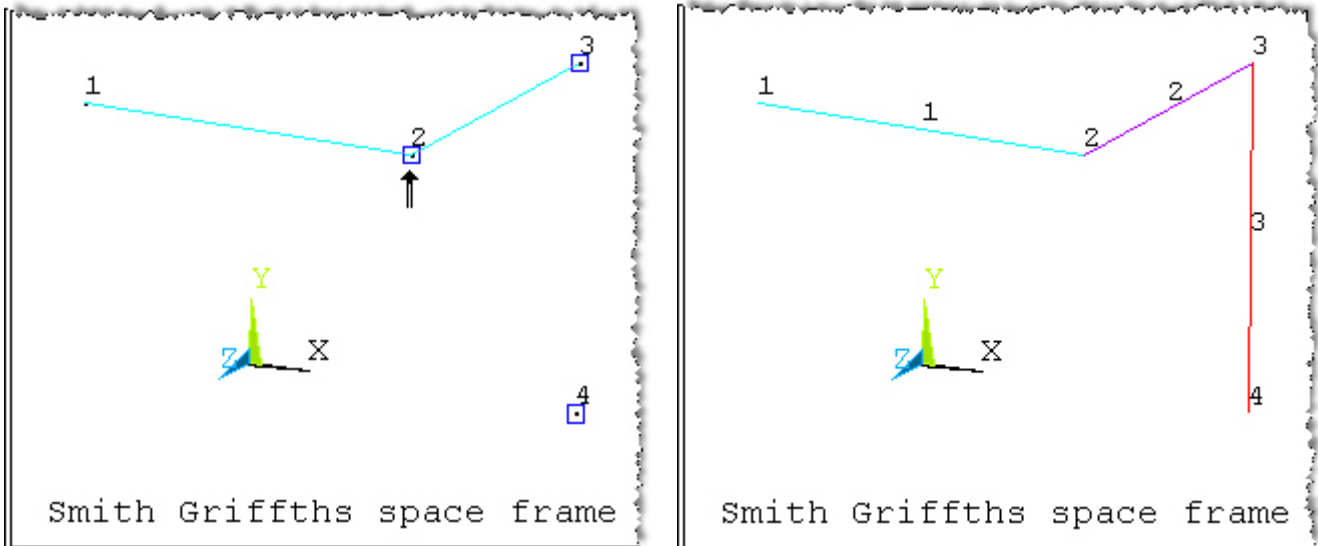


Figure 16 Final element pick for the space frame mesh

Check mesh data

It is wise to check such element input by plotting the nodes and elements, and/or the element's real constant set numbers via:

1. **Utility Menu**→**PlotCtrls**→**Numbers ...**
2. **Plot Numbering Controls**→**Elem /Attrib numbering** pick real const num in Figure 17 (left) to create the associated display (right).

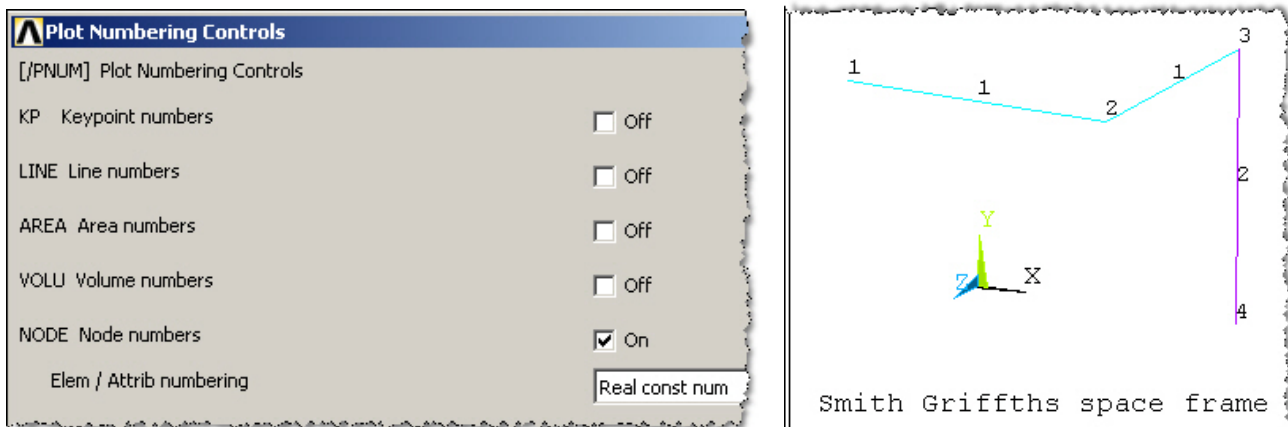
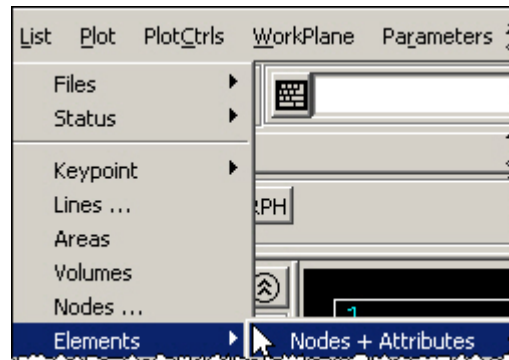


Figure 17 Displaying the real constant set numbers

Likewise, you can view a list of the data with:

1. **Utility Menu**→**List**→**Elements**→**Nodes+Attr+RealConst**.
2. When the **ELIST (ELe ment LIST) window** appears check those data and close it.



| ELIST Command | | | | | | | | | | |
|--|-----|-----|----------|-----|-----|----------|----------|---|----------|---------|
| File | | | | | | | | | | |
| LIST ALL SELECTED ELEMENTS. <LIST NODES> | | | | | | | | | | |
| ELEM | MAT | TYP | REL | ESY | SEC | NODES | | | | |
| 1 | 1 | 1 | 1 | 0 | 1 | 1 | 2 | 3 | | |
| AREA | | | IZZ | | | IYY | TKZ | | TKY | THET |
| ISTR | | | IXX | | | SHRZ | SHRY | | SPIN | ADMS |
| 4.00000 | | | 0.300000 | | | 1.00000 | 0.400000 | | 0.100000 | 0.00000 |
| 0.00000 | | | 0.600000 | | | 0.00000 | 0.00000 | | 0.00000 | 0.00000 |
| 2 | 1 | 1 | 1 | 0 | 1 | 2 | 3 | 1 | | |
| 4.00000 | | | 0.300000 | | | 1.00000 | 0.400000 | | 0.100000 | 0.00000 |
| 0.00000 | | | 0.600000 | | | 0.00000 | 0.00000 | | 0.00000 | 0.00000 |
| 3 | 1 | 1 | 2 | 0 | 1 | 3 | 4 | 2 | | |
| AREA | | | IZZ | | | IYY | TKZ | | TKY | THET |
| ISTR | | | IXX | | | SHRZ | SHRY | | SPIN | ADMS |
| 4.00000 | | | 1.00000 | | | 0.300000 | 0.100000 | | 0.400000 | 0.00000 |
| 0.00000 | | | 0.600000 | | | 0.00000 | 0.00000 | | 0.00000 | 0.00000 |

Figure 18 Listing the element attributes for checking

Apply displacement restraints

The displacement restraints must be applied to reflect the physical support (often the most unclear part of an analysis) as well as eliminating all the “rigid body motions” (RBM). Here there are three translational RBM plus three rotational ones. Apply the system nodal displacement restraints via:

1. **Main Menu**→**Preprocessor**→**Loads**→**Define Loads**→**Apply**→**Structural**→**Displacement**→**On Nodes** (see Figure 19).
2. In **Apply U, Rot on Nodes** verify that **Pick** and **Single** are checked on.
3. In the graphics window, Figure 20, select top **node 1** and bottom **node 4** (for complete restraints), **OK**. After the panel changes form, highlight the fixed option for all of the degrees of freedom components (**All DOF**) as the **DOF to be constrained**
4. Under **Constant value** enter 0, **OK**.

Note that these restraint operations are shown in the graphics window as triangles pointing in the direction of restraint, at each restrained node. To list the current restraints:

1. **Utility Menu**→**List**→**Loads**→**DOF Constraints**→**On All Nodes**.
 2. When the (Displacement **LIST**) **DLIST** window appears check those data and close it
- Review the graphical restraint symbols in Figure 20 to verify the choice you picked from the list of available displacement restraints.

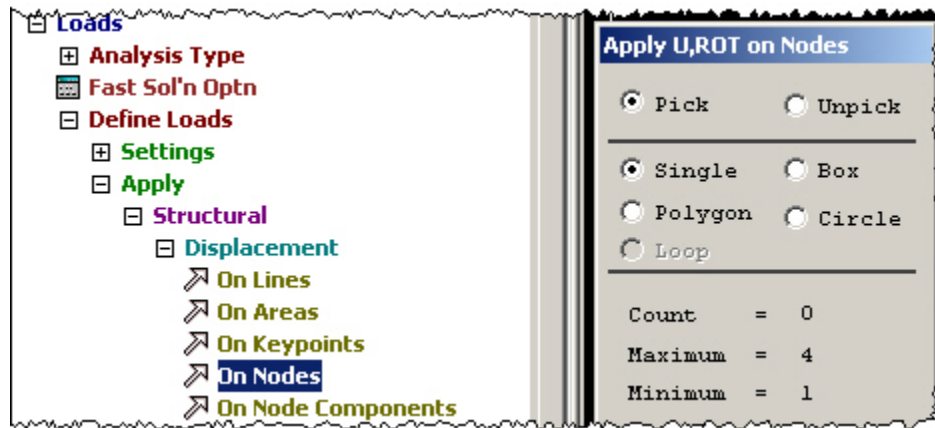


Figure 19 Preparing to graphically assign displacement restraints

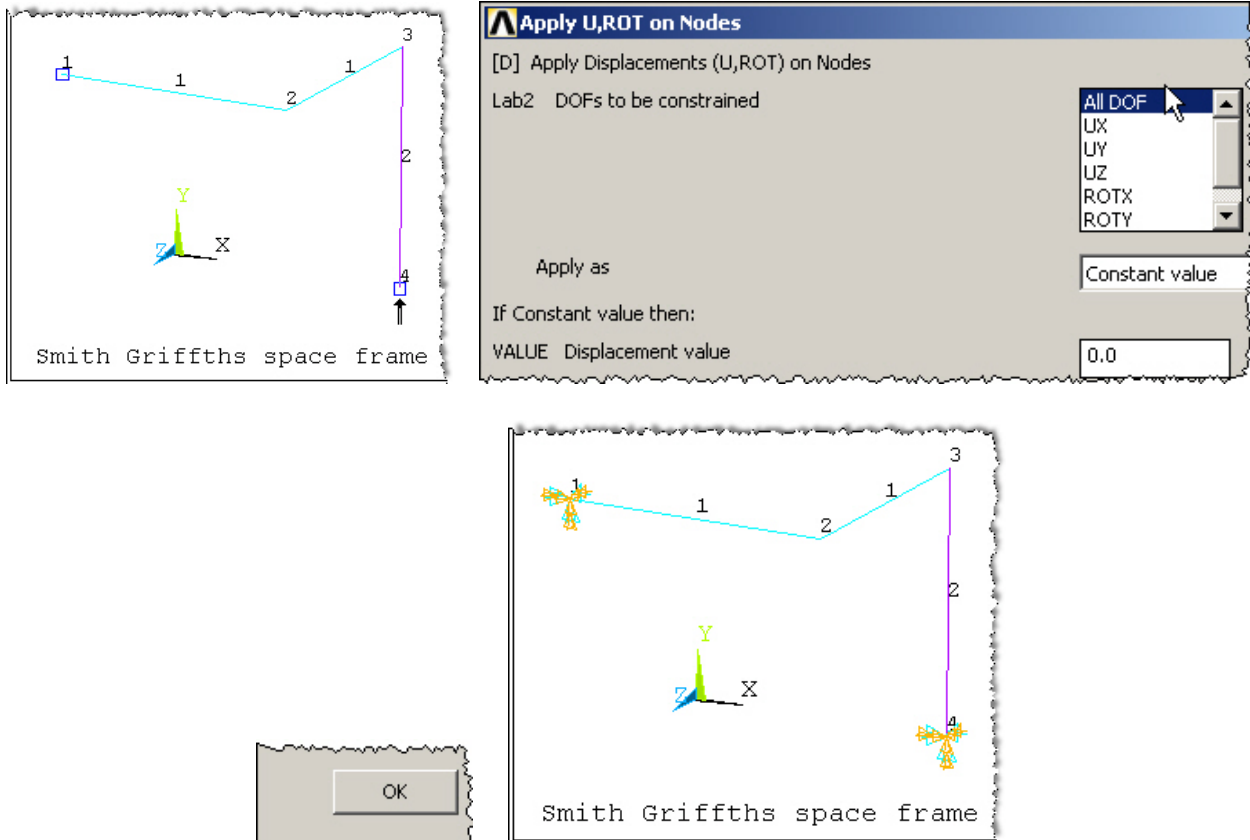


Figure 20 Picking nodes to restrain all displacements and rotations

Apply nodal loads

This space frame is intended to carry a vertical load of -100 (kN) at node 2. Specify that y-component force with:

1. **Main Menu**→**Preprocessor**→**Loads**→**Define Loads**→**Apply**→**Structural**→**Force/Moment**→**On Nodes**.
2. In **Apply F/M on Nodes** verify that **Pick** and **Single** are on, and then pick node 2, **OK**.
3. When the window changes form (lower part of Figure 21, pick a vertical force (**FY**) for the **Direction** of force and -100 (kN) for the **Constant value**, **OK**.
4. Then the arrows representing the load should appear as in Figure 22.

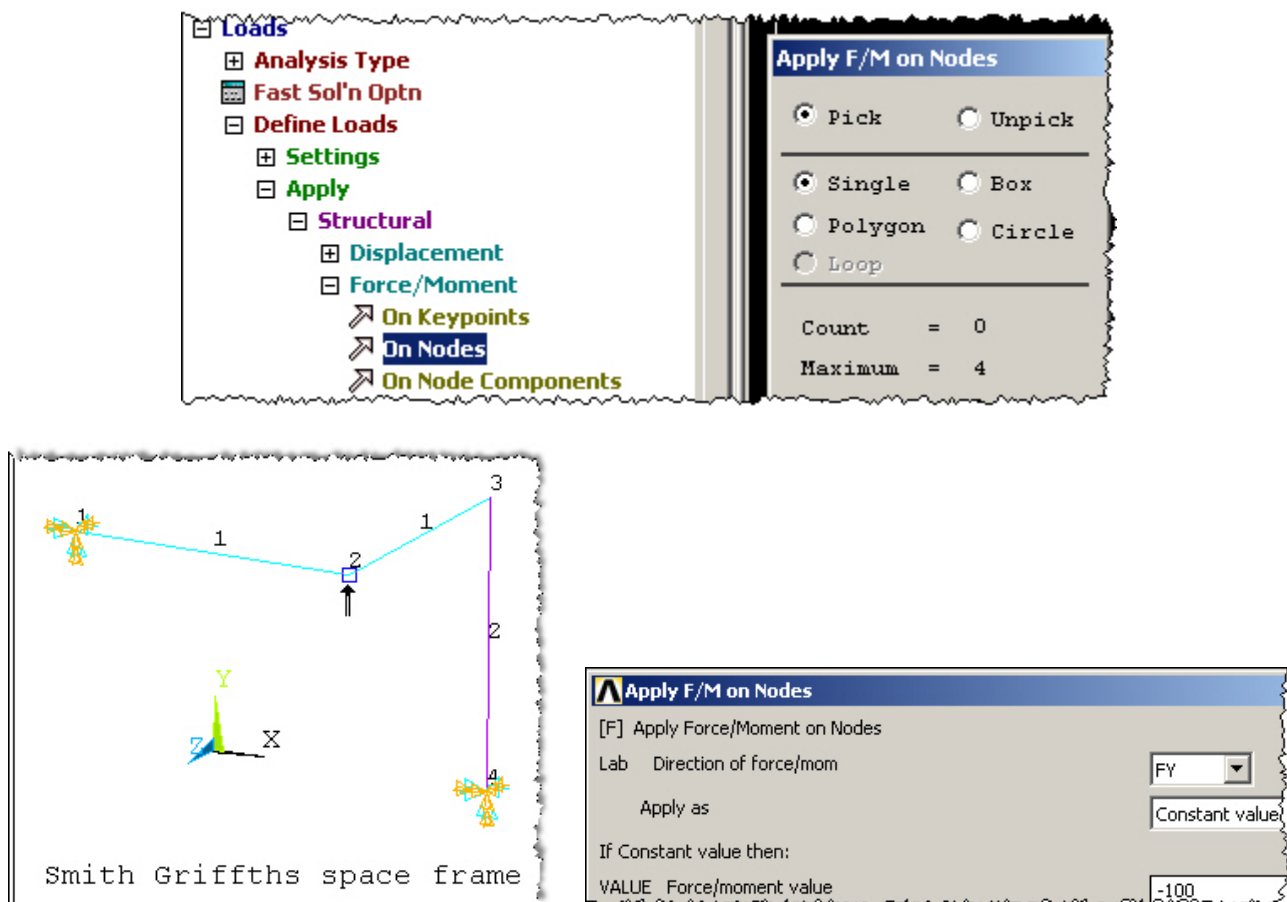


Figure 21 Graphically picking loaded nodes (lower left) and assigning values

Note that these load operations are shown in the graphics window as arrows pointing in the direction of load, at each restrained node. If your plot does not show everything you have defined also try **PlotCtrls**→**Symbols...**→**Symbols** and check that the items you want are turned on.

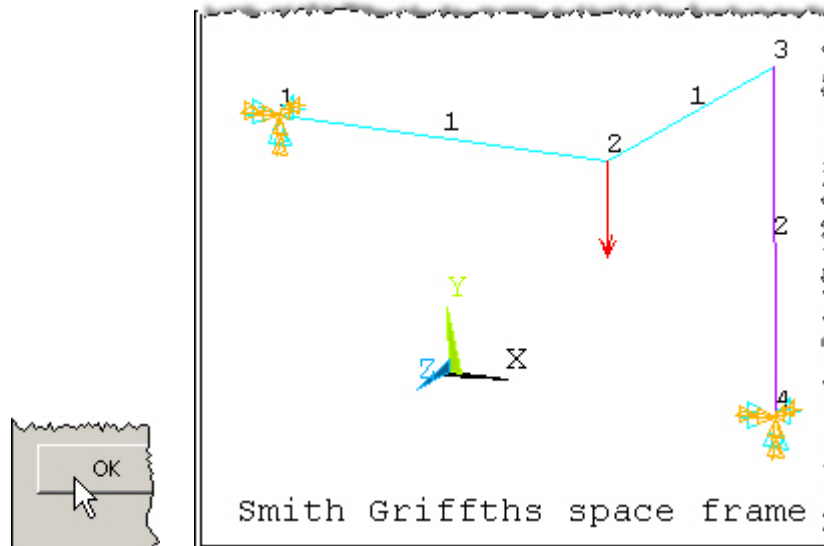


Figure 22 Loaded node's force arrow(s) appear

You may also wish to list the current loads:

1. **Utility Menu**→**List**→**Loads**→**Forces**→**On All Nodes**.
2. When the **Force LIST (FLIST)** window appears check those data and close it.

Saves and restarts

At this point you may want to save your data and restart the actual analysis later. If so:

1. **Utility Menu**→**File**→**Save as Jobname.db**.
2. Upon return open ANSYS and enter **Utility Menu**→**File**→**Resume from...** to get the list of your ANSYS database (db) files so you can select the one you want.

Solve for displacements and secondary variables

To use the current (and only) load system (LS) enter:

1. **Main Menu**→**Solution**→**Solve**→**Current LS**, review the listed summary, **OK**.
2. When the solution of the simultaneous equations is complete you will be alerted that the solution is done.

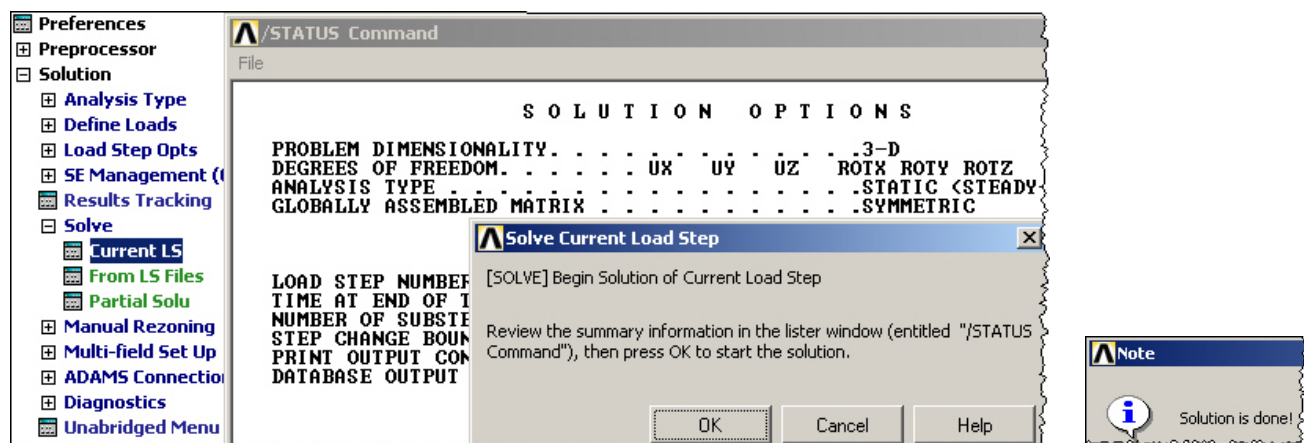


Figure 23 Solving displacements for the current load set

Post-processing

Displacements

It is always wise to visually check the computed displacements:

1. **Main Menu**→**General Postproc**→**Plot Results**→**Deformed Shape**.
2. In **Plot Deformed Shape** pick the combined **deflected and undeformed** option for the Items to be plotted (KUND), **OK**. Check the plot in the graphics window, such as the one seen in Figure 24.
3. Animate the computed deflections with:**Utility Menu**→**PlotCtrls**→**Animate**→**Deformed Shape**.
4. Pick Def+undeformed in **Animate Deformed Shape**, **OK**.
5. If desired, employ the Animation Controller that appears, or simply pick Stop, **Close**.

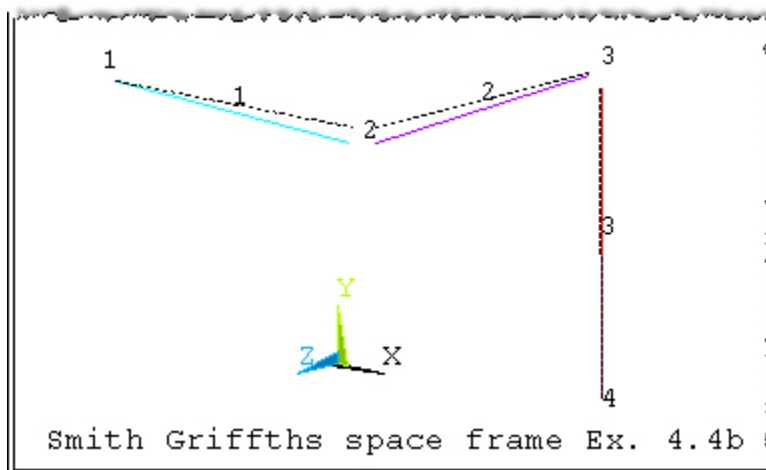
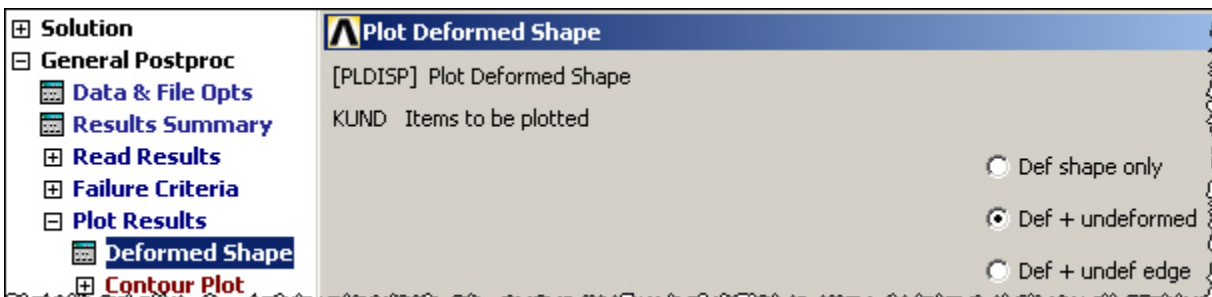


Figure 24 The deformed and undeformed space frame

To create a hardcopy (you may need to try various background colors):

1. **Utility Menu**→**PlotCtrls**→**Hard Copy**→Printer (or →**File**), select your printer name, **Print**.
2. To get the reverse video white background of Figure 24 use **PlotCtrls**→**Style**→**Color**→**Reverse Video**.

Since displacements and (infinitesimal) rotations are vector quantities it is wise to plot them in that mode as a visual check of the response of the system. To do that:

1. **General Postproc**→**Plot Results**→**Vector Plot**→**Predefined**.
2. In **Vector Plot of Predefined Vectors** select **DOF solution**, **Translation U**, **Vector mode**, and element **nodes**.

The resulting color plot will display the vectors with scaled lengths and with a color matching the color bar scale, as seen in Figure 25.

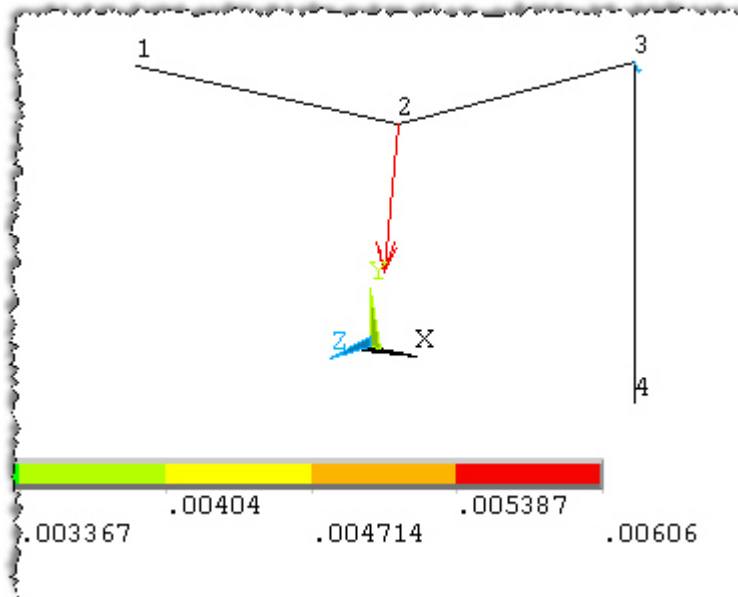
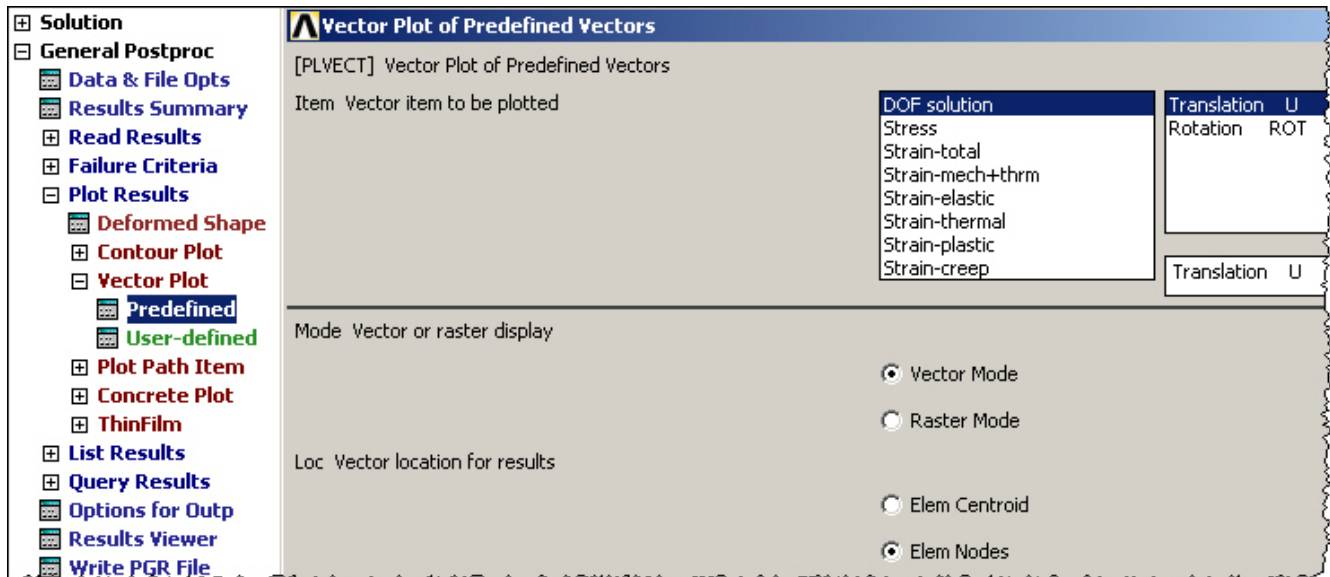
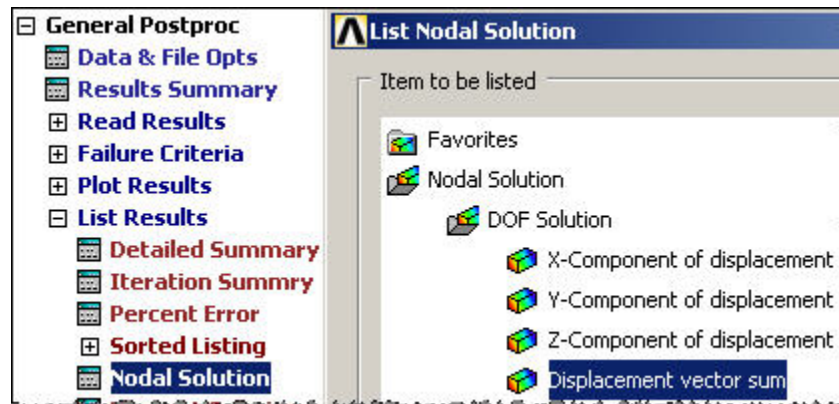


Figure 25 Nodal displacement vector plot

To see a (potentially long) list of displacement results:

1. **Preferences**→**General Postproc**→**List Results**→**Nodal Solution**.
2. In **List Nodal Solution**→**Nodal Solutions**→**DOF Solution**→**Displacement vector sum**, **OK**.
3. Examine the results in the **PRNSOL (PRint Nodal SOLution) Command window** of Figure 26 and close it.

4. Likewise, to see the nodal rotations, also in Figure 26, use **List Nodal Solution** → **Nodal Solutions** → **DOF Solution** → **Rotation vector sum**, OK.



```

PRNSOL Command
File
PRINT U  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 THE GLOBAL COORDINATE SYSTEM

```

| NODE | UX | UY | UZ | USUM |
|------|--------------|--------------|-------------|-------------|
| 1 | 0.0000 | 0.0000 | 0.0000 | 0.0000 |
| 2 | -0.30390E-05 | -0.59965E-02 | 0.87687E-03 | 0.60603E-02 |
| 3 | 0.95706E-03 | -0.45359E-04 | 0.91129E-03 | 0.13223E-02 |
| 4 | 0.0000 | 0.0000 | 0.0000 | 0.0000 |

```

MAXIMUM ABSOLUTE VALUES
NODE 3 2 3 2
VALUE 0.95706E-03 -0.59965E-02 0.91129E-03 0.60603E-02

```

```

PRNSOL Command
File
PRINT ROT  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 THE GLOBAL COORDINATE SYSTEM

```

| NODE | ROTX | ROTY | ROTZ | RSUM |
|------|-------------|--------------|--------------|-------------|
| 1 | 0.0000 | 0.0000 | 0.0000 | 0.0000 |
| 2 | 0.11295E-02 | -0.23600E-03 | -0.15137E-02 | 0.19033E-02 |
| 3 | 0.74700E-03 | -0.15816E-03 | -0.37269E-03 | 0.84966E-03 |
| 4 | 0.0000 | 0.0000 | 0.0000 | 0.0000 |

```

MAXIMUM ABSOLUTE VALUES
NODE 2 2 2 2
VALUE 0.11295E-02 -0.23600E-03 -0.15137E-02 0.19033E-02

```

Figure 26 Listing the nodal displacements and rotations

Displacement solution validation

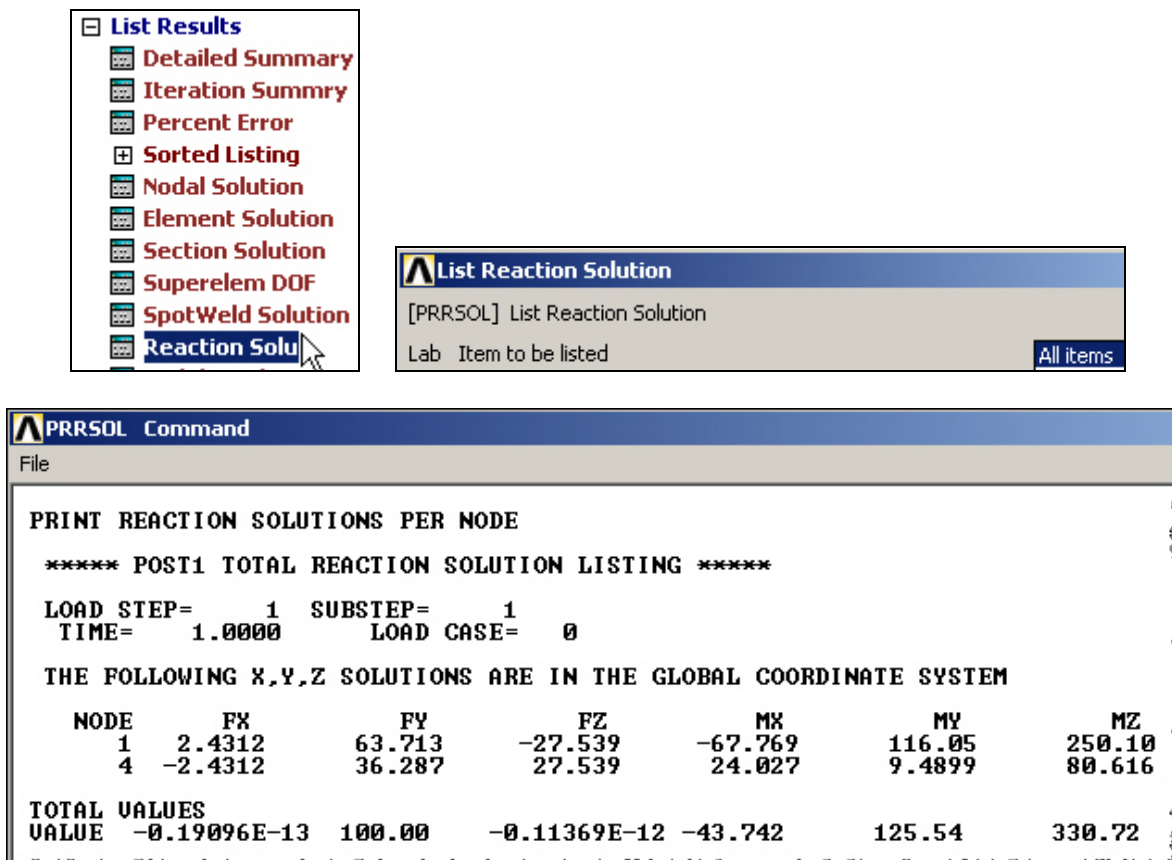
The generalized nodal displacements shown in Figure 26 agree with the (four significant figure) results in [1]. Therefore, you can conclude that you have followed the correct steps for the analysis of a space frame. Next you can review the system reactions, the member reactions and member stresses.

System reaction forces

If the solver does not fail then your reactions will be equal and opposite of your resultant forces and moments. Therefore they let you check the level of loads actually applied, versus what you intended to apply. That is helpful especially for pressure loads. Check the system reactions with:

1. **Main Menu**→**General Postproc**→**List Results**→**Reaction Solution**.
2. In **List Reaction Solution** pick **All items** for Item to be listed, **OK**.
3. Review the **Print Reaction SOLution (PRRSOL)** Command window of Figure 27.

Those system reactions also agree with [1] to four significant figures.



The screenshot shows the ANSYS software interface. On the left, the 'List Results' menu is open, with 'Reaction Solution' selected. In the center, the 'List Reaction Solution' dialog box is open, showing '[PRRSOL] List Reaction Solution' and 'Lab Item to be listed' with 'All items' selected. On the right, the 'PRRSOL Command' window is open, displaying the following text:

```
PRRSOL Command
File
PRINT REACTION SOLUTIONS PER NODE
***** POST1 TOTAL REACTION SOLUTION LISTING *****
LOAD STEP=      1  SUBSTEP=      1
TIME=      1.0000  LOAD CASE=      0
THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM
      NODE      FX      FY      FZ      MX      MY      MZ
      1      2.4312      63.713      -27.539      -67.769      116.05      250.10
      4     -2.4312      36.287      27.539      24.027      9.4899      80.616
TOTAL VALUES
VALUE  -0.19096E-13  100.00  -0.11369E-12  -43.742  125.54  330.72
```

Figure 27 Examine the system reaction forces and moments

Member end forces and moments

The element (member) end forces and moments, also known as the member reactions, in the space frame resulting from the computed displacements can also be recovered and listed:

1. **Main Menu**→**General Postproc**→**List Results**→**Element Solution**.
 2. Get the member axial forces, in global components, from **List Element Solution**→**Element Solution**→**All Available Force Items**, **OK**.
 3. Review the **PRESOL (P**rint **E**lement **S**olution) **Command** display of Figure 28.
- Those element level reactions also agree with [1] to four significant figures.

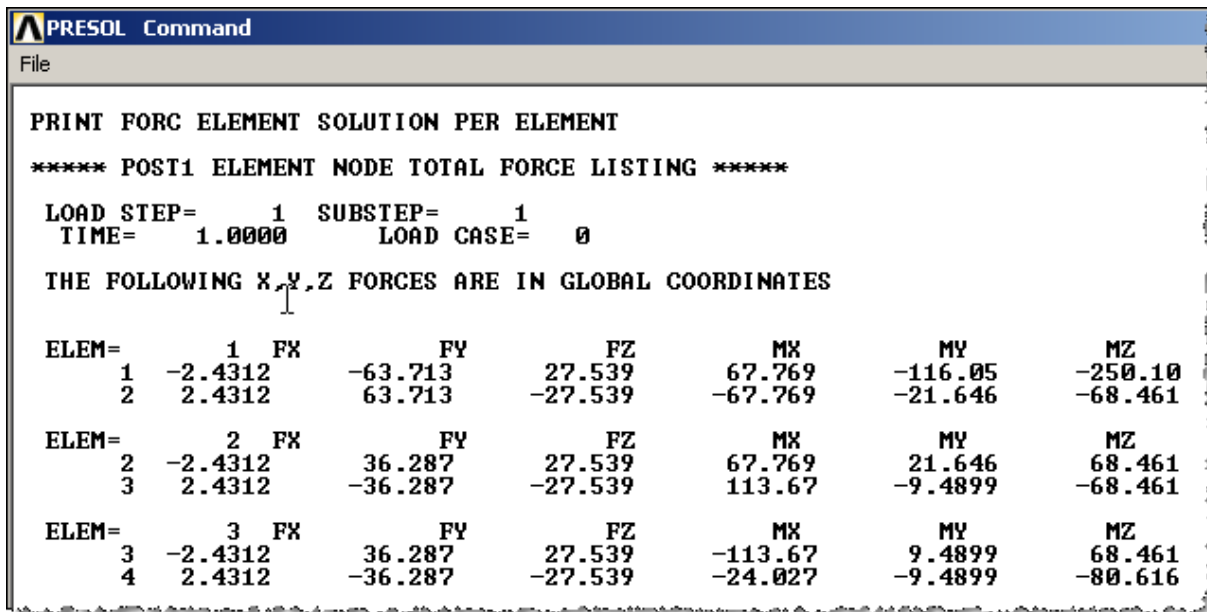


Figure 28 Individual member end reactions

Member stress state

Reference [1] does not give the member stresses. ANSYS does give them and the strains also. Here, the stress data, other than the neutral axis stress SDIR, are not meaningful because the Y- and Z-thickness were not known but were chosen to allow visual checks, as seen in Figure . When thickness values consistent with the area, IYY, and IZZ are given then the reported stresses are the bending stresses superimposed on the axial stresses.

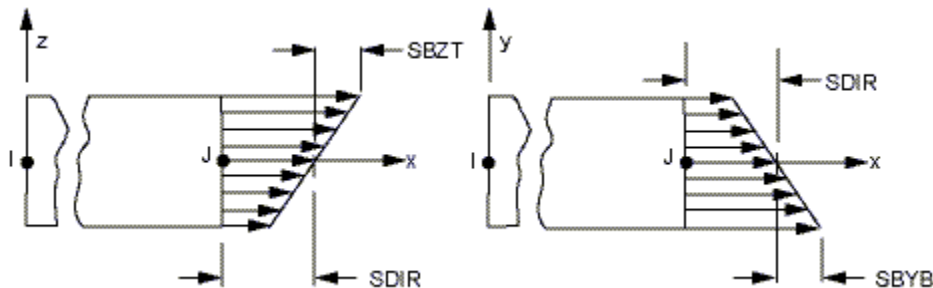


Figure 29 Definition of the BEAM4 stresses

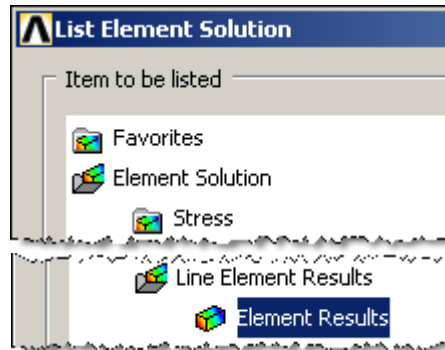
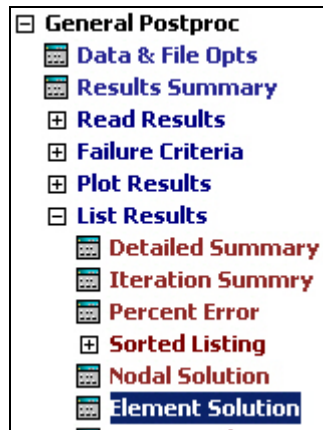


Figure 30 Selecting line element stress results

PRE SOL Command

File

```

PRINT ELEM ELEMENT SOLUTION PER ELEMENT
***** POST1 ELEMENT SOLUTION LISTING *****
LOAD STEP      1 SUBSTEP=      1
TIME=          1.0000 LOAD CASE= 0

EL=            1  NODES=        1    2    3  MAT= 1
BEAM4
TEMP =         0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00
LOCATION  SDIR      SBYT      SBYB      SBZT      SBZB
  1 (I) -0.60780 -41.684  41.684  23.209 -23.209
  2 (J) -0.60780  11.410 -11.410  -4.3292  4.3292
LOCATION  SMAX      SMIN
  1 (I)  64.286 -65.501
  2 (J)  15.131 -16.347

EL=            2  NODES=        2    3    1  MAT= 1
BEAM4
TEMP =         0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00
LOCATION  SDIR      SBYT      SBYB      SBZT      SBZB
  1 (I) -6.8846  11.295 -11.295  -4.3292  4.3292
  2 (J) -6.8846 -18.944  18.944  -1.8980  1.8980
LOCATION  SMAX      SMIN
  1 (I)  8.7393 -22.509
  2 (J)  13.958 -27.727

EL=            3  NODES=        3    4    2  MAT= 1
BEAM4
TEMP =         0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00
LOCATION  SDIR      SBYT      SBYB      SBZT      SBZB
  1 (I) -9.0717 -13.692  13.692 -18.944  18.944
  2 (J) -9.0717 -16.123  16.123  4.0044 -4.0044
LOCATION  SMAX      SMIN
  1 (I)  23.565 -41.708
  2 (J)  11.056 -29.199

```

* S = Stress, DIR = DIRect (axial), BYT, BYB = Bending Y-axis Top, Bottom, BZT, BZB = Bending Z-axis Top, Bottom

Figure 31 List of member local stresses*

Exiting ANSYS

You can save the results to your data base, as described above, and close with **File→Exit**. To begin a different analysis, instead of picking Exit, use **File→Clear & Start New ...**, **OK**.

Closure

For this class of loading this three element model will be close to giving exact results for the generalized nodal displacements. The beam deflections are cubic in space but most finite element codes execute a linear plot between the two end nodes, as in Figure 24. If you wanted to see the more realistic deformed shape the simplest thing to do is to divide each of the elements into more elements, say 10 each. ANSYS can easily do that for you.

The job data for almost any analysis can be entered by a text file. The data for this example are in Figure 32. One thing you should note about this alternate input approach is that the data for the real constants sets will only accept six entries per line. Therefore, keyword *r* (for real) must be followed with the keyword *rmore* in order to supply the IXX term here. Lines beginning with a slash (/) are ANSYS commands. Comments are preceded with an exclamation symbol (!) and extend to the end of the line.

To read such a data text file into ANSYS begin in the **Utility Menu**:

1. **File→Read Input From ...**
2. Pick the **Directory, Filename, OK**.

Then you can use the GUI to pick up your study at the stage where you chose to terminate the batch text file. The entire analysis process can be controlled from a batch file. Experienced users often find this feature to be a time saver.

Reference:

I.M. Smith, D.V. Griffiths, Programming the Finite Element Method, 3rd Ed, Chichester, John Wiley, 1998

```

/FILNAM, S_G_SF_44
/title, Smith Griffths space frame Ex. 4.4b
/prep7
!
! General Summary
! Smith Griffths space frame Ex. 4.4b page137-141
! Units: m, kN, m^2, m^4, kN/m^2
! AREA = 4, IX = 0.3, IY = 1, IZ = 0.3, P = 100
! ELASTIC = 1.e6, SHEAR = 1.e6
!
! #1 Fixed          *3      Node 2 displacements:
! \                P      / |      -3.039E-6 -5.997E-3  8.769E-4
! \                :      / |      Node 2 rotations:
! (1)\            : /((2) |(3)  1.130E-3 -2.360E-4 -1.514E-3
! \                \ v/      |      Node 1 force reactions:
!          *2          |      2.431E+0, 6.371E+1, -2.754E+1
!
! #4 Fixed

n, 1, 0, 5, 5 ! begin node definition
n, 2, 5, 5, 5
n, 3, 5, 5, 0
n, 4, 5, 0, 0

et,1,beam4 ! first element type

mp,ex,1,1e6 ! first material type
mp,prxy,1,0.0

! first real constant set (elements 1 & 2)
! AREA, IZZ, IYY, TkZ, TkY, THETA (6 MAX !)
r, 1, 4., 0.3, 1.0, 0.4, 0.1, 0
! ISTRN, IXX, SHEARZ, SHEARY, SPIN, DDMAS
rmore, 0, 0.6, 0, 0, 0, 0

! second real constant set (elements 3)
! AREA, IZZ, IYY, TkZ, TkY, THETA (6 MAX !)
r, 2, 4., 1.0, 0.3, 0.1, 0.4, 0
! ISTRN, IXX, SHEARZ, SHEARY, SPIN, ADDMAS
rmore, 0, 0.6, 0, 0, 0, 0

! Define elements of et=1, mt=1. First 2 nodes set
! local beam x-axis, third node sets local xz-plane
REAL,1 ! turn on real property set #1
en, 1, 1, 2, 3 ! elem i, j, k
en, 2, 2, 3, 1
REAL,2 ! activate real property set #2
en, 3, 3, 4, 2

d, 1, all, 0. ! displacement restraints
d, 4, all, 0.

f, 2, fy, -100. ! force at joint

finish ! batch input, go to GUI

```

Figure 32 Text file to describe this example mesh, restraints, and loads