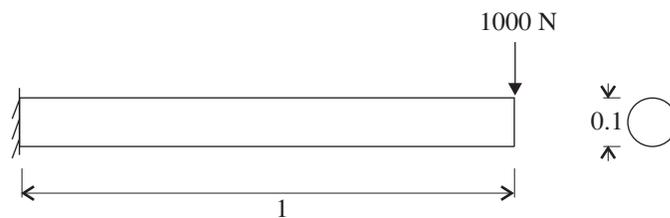


Problem description

A solid cylinder is subjected to a tip load as shown:



All lengths in meters
 $E = 2.07 \times 10^{11} \text{ N/m}^2$
 $\nu = 0.29$

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

- Defining a geometry line by extruding a point
- Defining a geometry surface by revolving a line
- Defining a geometry volume by extruding a surface
- Generating hexahedral and prismatic elements
- Rotating a mesh plot with the mouse
- Smoothing the stresses
- Creating a node-set
- Calculating section forces and moments
- Using the Split Zone feature

We assume that you have worked through problems 1 to 3, or have equivalent experience with the ADINA System.

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the GUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This problem can be solved with the 900 nodes version of the ADINA System.

Invoking the GUI and choosing the finite element program

Invoke the GUI and set the Program Module drop-down list to ADINA Structures.

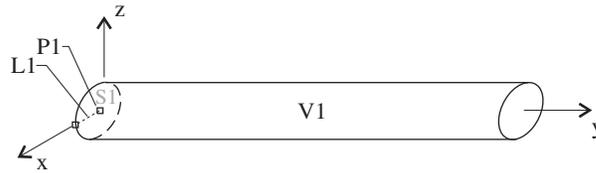
Problem 4: Cylinder subjected to tip load

Defining model control data

Problem heading: Choose Control→Heading, enter the heading “Problem 4: Cylinder subjected to tip load” and click OK.

Defining model geometry

Here is a diagram showing the key geometry used in defining this model:



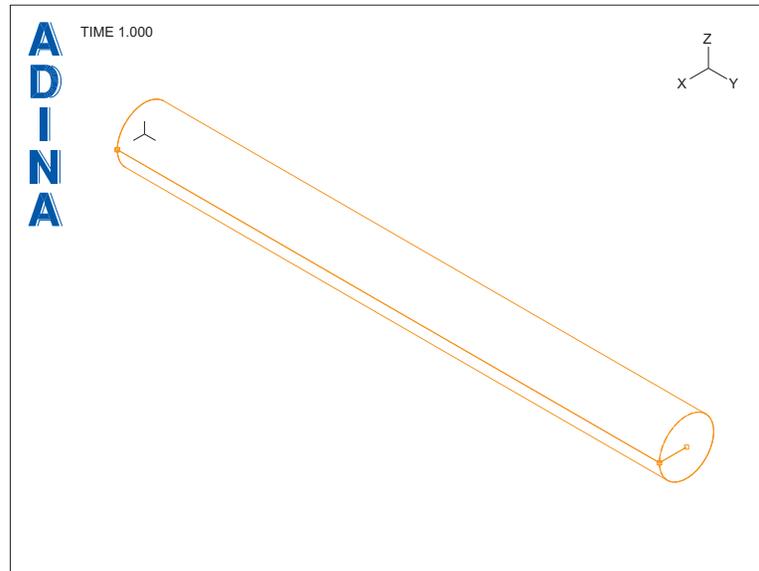
Geometry point: Click the Define Points icon , enter the following information into the table and click OK.

Point #	X1	X2	X3
1	0	0	0

Geometry line: Click the Define Lines icon , add line 1, set the Type to Extruded, set the Initial Point to 1, the components of the Vector to 0.05, 0.0, 0.0 and click OK.

Geometry surface: Click the Define Surfaces icon , add surface 1, set the Type to Revolved, set the Initial Line to 1, the Angle of Rotation to 360, the Axis to Y, uncheck the Check Coincidence button and click OK.

Geometry volume: Click the Define Volumes icon , add volume 1, set the Type to Extruded, set the Initial Surface to 1, the components of the Vector to 0.0, 1.0, 0.0, uncheck the Check Coincidence button and click OK. The graphics window should look something like the figure on the next page.



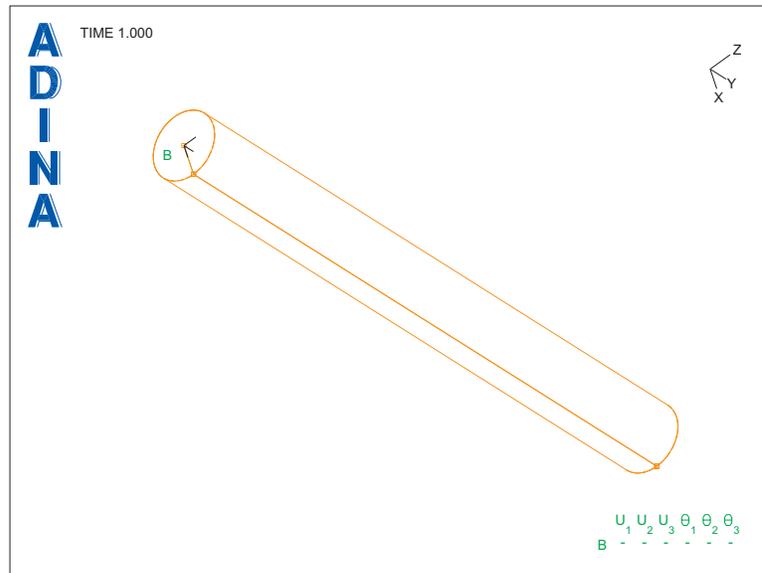
Applying boundary conditions

We will fix surface 1. Click the Apply Fixity icon , set the “Apply to” field to Face/Surface, enter 1 in the first row and column of the table and click OK. Click the Boundary Plot icon  to display the boundary conditions.

Since surface 1 is hidden, we can't see if the boundary condition is applied or not. Rotate the mesh out-of-plane with the mouse until the graphics window looks something like the figure on the next page.

Here are detailed instructions for rotating the mesh out-of-plane. Click the Pick icon  and the Dynamic Rotate (XY) icon  and highlight the mesh plot. Press and hold the left mouse button, then move the mouse. The mesh plot rotates along with the mouse motion. When the mesh plot is in the correct position, release the left mouse button. (You can also rotate the mesh out-of-plane when the Dynamic Pan icon  is pressed by holding down the Shift key while dragging the mouse.)

Problem 4: Cylinder subjected to tip load



Defining and applying loads

Click the Apply Load icon , make sure that the Load Type is Force and click the Define... button to the right of the Load Number field. In the Define Concentrated Force dialog box, add force 1, set the Magnitude to 1000, the Direction to (-1.0, 0.0, 0.0) and click OK. In the Apply Load dialog box, in the first row of the table, set the Point # to 6. Click OK to close the Apply Load dialog box.

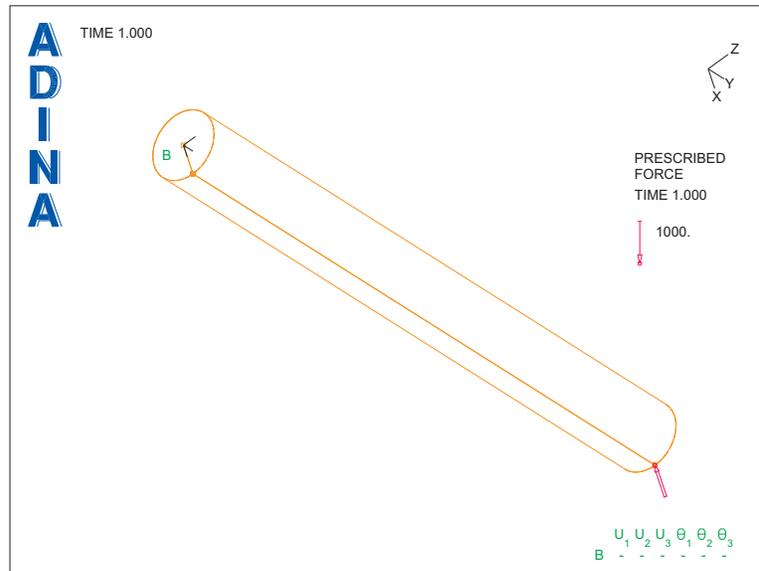
When you click the Load Plot icon , the graphics window should look something like the figure on the next page.

Defining the material

Click the Manage Materials icon  and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 1, set the Young's Modulus to 2.07E11, the Poisson's ratio to 0.29 and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the elements

Element group: Click the Element Groups icon , add group number 1, set the Type to 3-D Solid and click OK.



Subdivision data: In this mesh, we will assign the number of subdivisions in the u, v and w directions of the volume. In this case, the u direction is the tangential direction, the v direction is the axial direction and the w direction is the radial direction.

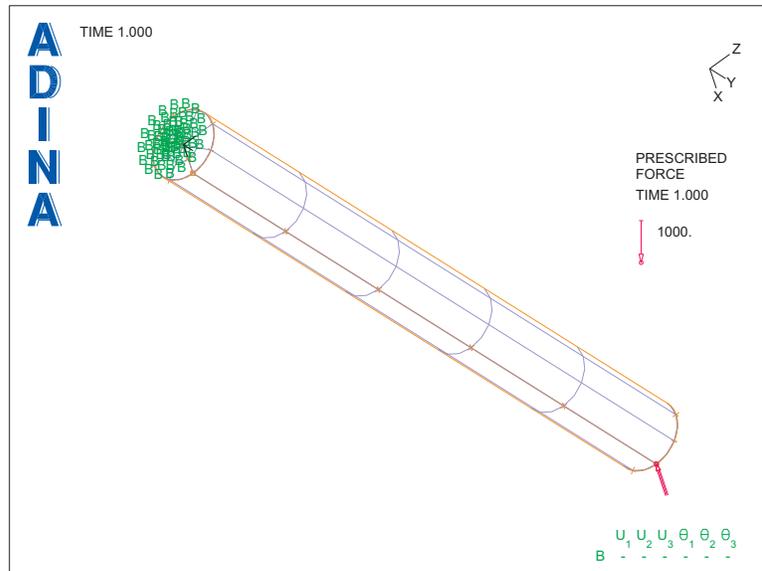
Click the Subdivide Volumes icon  and set the Number of Subdivisions in the u, v and w directions to 8, 5 and 2 respectively. Then click OK.

Element generation: Click the Mesh Volumes icon , enter 1 in the first row of the table and click OK. The graphics window should look something like the figure on the next page.

Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

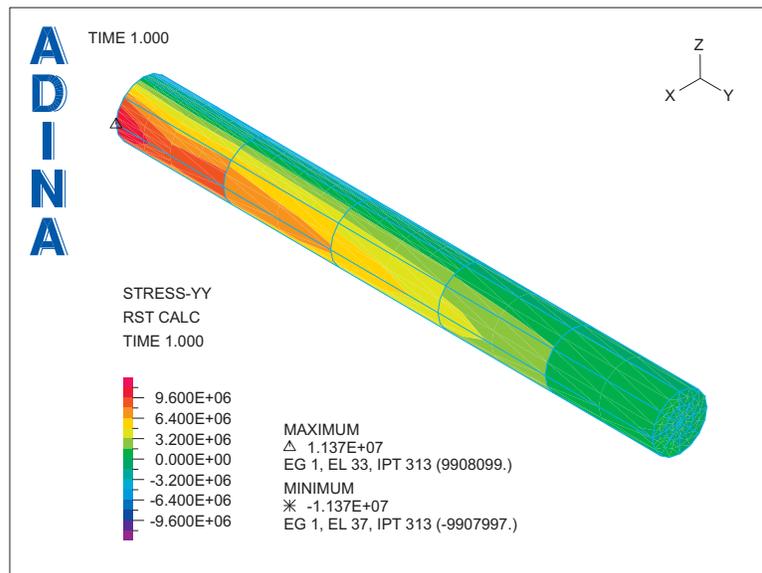
First click the Save icon  and save the database to file prob04. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon , set the file name to prob04, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob04.

Problem 4: Cylinder subjected to tip load



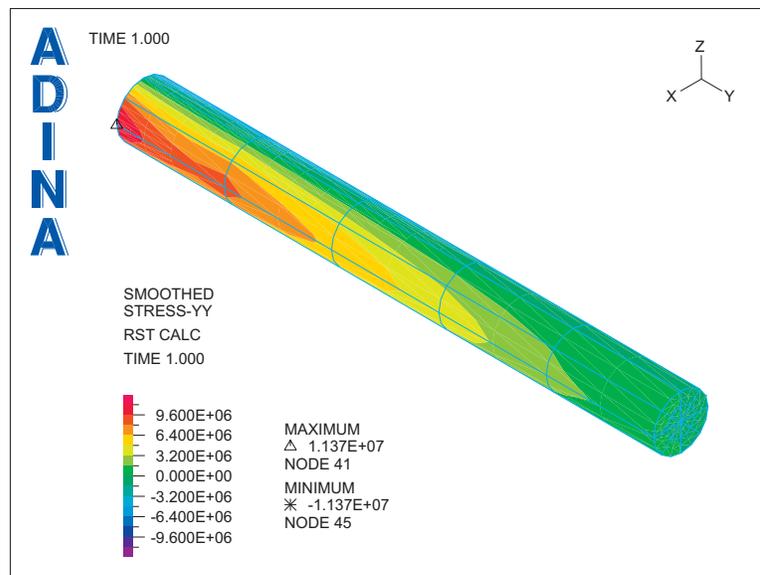
Examining the solution

Stress band plot: Click the Create Band Plot icon , set the Band Plot Variable to (Stress: STRESS-YY) and click OK. The graphics window should look something like this:



You will notice discontinuities or “jumps” in the bands near the built-in end of the cylinder.

To smooth the stress band plot, click the Smooth Plots icon . The graphics window should look something like this:



(Note: smoothing the stresses does not make them more accurate.)

Calculating section forces and moments - model definition

In order to calculate section forces and moments, it is necessary for ADINA Structures to compute the element nodal point forces.

Set the Program Module to ADINA Structures (you can discard all changes) and choose database file prob04.i db from the recent file list near the bottom of the File menu.

Click the Element Groups icon , set the “Element Result Output” to “Nodal Forces” and click OK.

Calculating section forces and moments - running ADINA Structures

Click the Save icon  to save the database to file prob04. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon , set the file name to prob04b, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes. Set the Program Module

Problem 4: Cylinder subjected to tip load

drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob04b.

Checking global equilibrium

First we will check that the sum of the nodal point forces and moments within the entire finite element assemblage is zero.

Choose Definitions→Model Point (Special) →Element Force, add point name `WHOLE_MODEL_POINT`, make sure that the “Zone Containing Elements” is set to `WHOLE_MODEL`, and, in the 'Defined by Nodes In' box, set the Zone to `WHOLE_MODEL`. The element force point contains the nodes in both of the two zones; since these zones are both the `WHOLE_MODEL` zone, the element force point contains all of the nodes in the model. Click OK to close the dialog box.

Now choose List→Value List→Model Point, set Variable 1 to (Force: `FORCE_SUM-X`), Variable 2 to (Force: `FORCE_SUM-Y`), Variable 3 to (Force: `FORCE_SUM-Z`), Variable 4 to (Force: `MOMENT_SUM-X`), Variable 5 to (Force: `MOMENT_SUM-Y`), Variable 6 to (Force: `MOMENT_SUM-Z`) and click Apply. The value of these variables at time 1 are all very small (less than $1E-7$ on our computer). Click Close to close the dialog box.

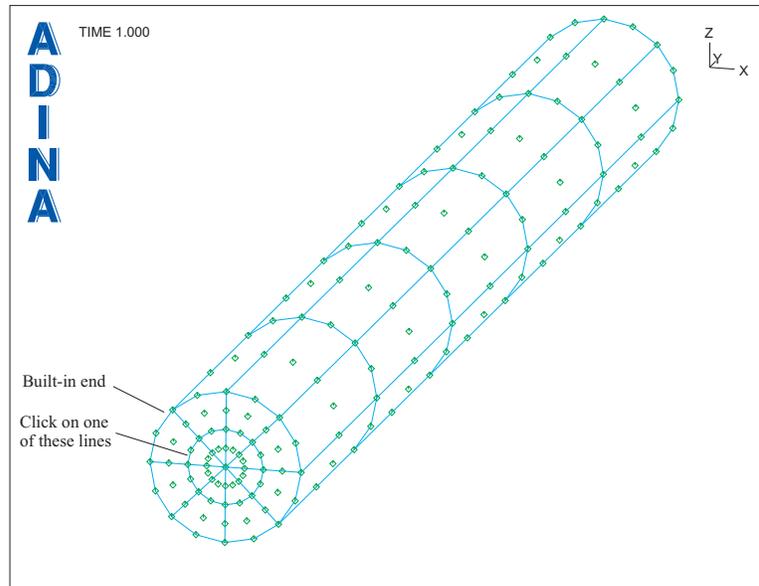
Calculating the forces and moments at the built-in end

In order to calculate the forces and moments at the built-in end, we need to select the nodes on the built-in end. One way to do this is to define a node-set with these nodes.

Click the Node Symbols icon , then use the Pick icon  and the mouse to rotate the mesh plot until the built-in end is visible. The graphics window should look something like the figure on the next page.

Now click the Node Set icon , add Node Set 1 and set the Method to “Auto-Chain Element Faces”. Double-click in the Face column of the table, click on one of the element lines on the built-in end, press the Esc key and click Save. The nodes on the built-in end should be highlighted (you might need to move the dialog box away from the mesh plot). Click OK to close the dialog box.

Choose Definitions→Model Point (Special)→Element Force, add point name `BUILT_IN`, make sure that the “Zone Containing Elements” is set to `WHOLE_MODEL`, and, in the “Defined by Nodes In” box, set the Node Set to 1. Click OK to close the dialog box.



Now choose List→Value List→Model Point, set the Point Name to BUILT_IN, set Variable 1 to (Force: FORCE_SUM-X), Variable 2 to (Force: FORCE_SUM-Y), Variable 3 to (Force: FORCE_SUM-Z), Variable 4 to (Force: MOMENT_SUM-X), Variable 5 to (Force: MOMENT_SUM-Y), Variable 6 to (Force: MOMENT_SUM-Z) and click Apply. FORCE_SUM-X is equal to 1.00000E+03 (N) and MOMENT_SUM-Z is equal to -1.00000E+03 (N-m), and the other variables are very small. These sums are the expected values. Click Close to close the dialog box.

Calculating the forces and moments on a section half-way down the cylinder

In order to calculate the forces and moments within the model, we need to select the correct elements and nodes. This can be done in two steps:

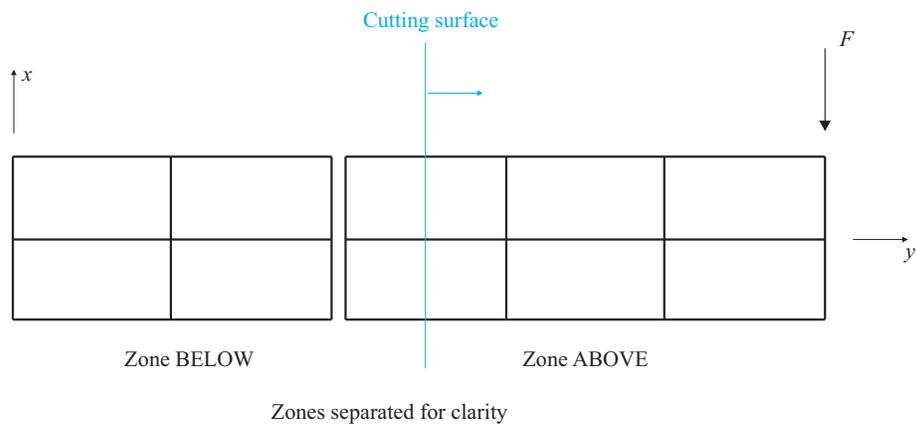
- 1) Split the model into two zones.
- 2) Select the elements and nodes on the intersection of the two zones.

Splitting the model into two zones

Click the Split Zone icon  and click the ... button to the right of the "With Cutting Plane" field. In the Define Cutsurface Depiction dialog box, set 'Defined by' to Y-Plane, the Coordinate Value to 0.5 and click OK. In the Split Zone dialog box, set 'Place Elements Above Cutting Plane into Zone' to 'ABOVE' (by typing the word ABOVE into this field), and set 'Place Elements Below Cutting Plane into Zone' to 'BELOW'. Click OK to close the dialog box.

Problem 4: Cylinder subjected to tip load

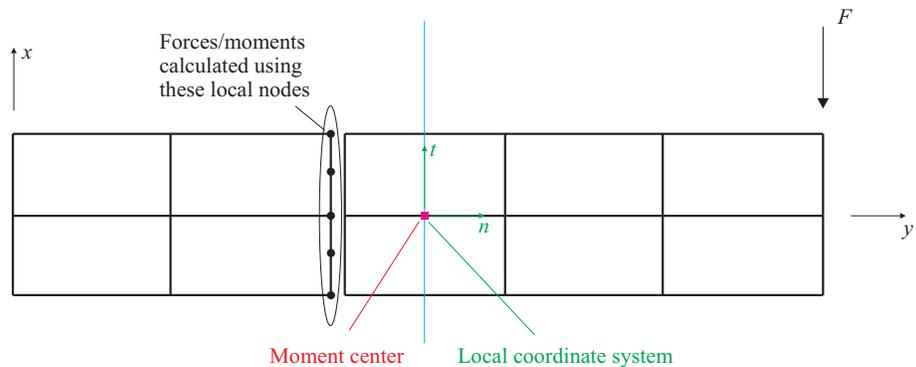
Within the Model Tree, expand the Zone entry, then click on zone ABOVE to highlight zone ABOVE, then click on zone BELOW to highlight zone BELOW. You should be able to confirm that these zones are defined as shown in the following figure:



Selecting the elements and nodes on the intersection of the two parts

Choose Definitions→Model Point (Special) →Element Force, add point name HALFWAY, set “Zone Containing Elements” to BELOW, and, in the “Defined by Nodes In” box, set the Zone to ABOVE. Also set the Local Coordinate System Cut Surface to CUTPLANE_SPLITZONE. Click OK to close the dialog box.

The element local nodes in the element force point are chosen to be those that are in the zone specified by “Zone Containing Elements”, as shown in the following figure:



The element local nodes do not lie on the cutting plane, see notes at the end of this problem description for a discussion.

Calculating forces and moments

Choose List→Value List→Model Point, set the Point Name to HALFWAY, set Variable 1 to (Force: FORCE_SUM-X), Variable 2 to (Force: FORCE_SUM-Y), Variable 3 to (Force: FORCE_SUM-Z), Variable 4 to (Force: MOMENT_SUM-X), Variable 5 to (Force: MOMENT_SUM-Y), Variable 6 to (Force: MOMENT_SUM-Z) and click Apply. FORCE_SUM-X is equal to $-1.00000E+03$ (N) and MOMENT_SUM-Z is equal to $5.00000E+02$ (N-m), and the other variables are very small. These sums are the expected values. (Do not close the dialog box yet.)

In moment calculations, the position of the moment center is very important. Let's verify the position of the moment center used in point HALFWAY. Set Variable 1 to (Coordinate: MOMENT_CENTER-X), Variable 2 to (Coordinate: MOMENT_CENTER-Y), Variable 3 to (Coordinate: MOMENT_CENTER-Z) and click Apply. The moment center is seen to be at coordinate (0, 0.5, 0). (This is because, by default, the moment center is calculated using the Local Coordinate System Cut Surface, and also the projection of the selected element faces onto that cutting surface.) (Do not close the dialog box yet.)

Calculating forces and moments in a local coordinate system

In this model, the results in the global coordinate system are physically meaningful, however in a more complicated model, it is necessary to choose a local coordinate system aligned with the section in order to obtain meaningful results.

By default a local coordinate system with components t, b, n is defined in the element force point. In order to determine the directions of this coordinate system, set Variable 1 to (Miscellaneous: LOCAL_T_DIRECTION-X), Variable 2 to (Miscellaneous: LOCAL_T_DIRECTION-Y), Variable 3 to (Miscellaneous: LOCAL_T_DIRECTION-Z), Variable 4 to (Miscellaneous: LOCAL_N_DIRECTION-X), Variable 5 to (Miscellaneous: LOCAL_N_DIRECTION-Y), Variable 6 to (Miscellaneous: LOCAL_N_DIRECTION-Z) and click Apply. The t direction is (1,0,0) and the n direction is (0,1,0). (The b direction can be obtained in a similar way).

Now set Variable 1 to (Force: FORCE_SUM-T), Variable 2 to (Force: FORCE_SUM-B), Variable 3 to (Force: FORCE_SUM-N), Variable 4 to (Force: MOMENT_SUM-T), Variable 5 to (Force: MOMENT_SUM-B), Variable 6 to (Force: MOMENT_SUM-N) and click Apply. FORCE_SUM-T is equal to $-1.00000E+03$ (N) and MOMENT_SUM-B is equal to $-5.00000E+02$ (N-m), and the other variables are very small. These sums are the expected values. Click OK to close the dialog box.

Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.

Note

1) In the calculation of force and moments using point HALFWAY, it is clear from the figure that the local nodes do not lie on the cutting plane. We deliberately choose to do this in order to emphasize that the local nodes need not lie on the cutting plane. Of course, in many cases, you might choose to place the element boundaries at the section at which you want to extract forces and moments, then the local nodes will lie on the cutting plane. However, when a free form mesh is used, the local nodes need not lie on the section. The techniques demonstrated here work for both mapped and free-form meshes.