

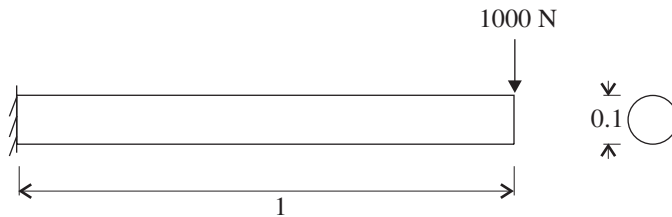
## **Problem 4**

**Cylinder subjected to tip load**

This page intentionally left blank.

## 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

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

### Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI 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 AUI and choosing the finite element program

Invoke the AUI and choose ADINA Structures from the Program Module drop-down list.

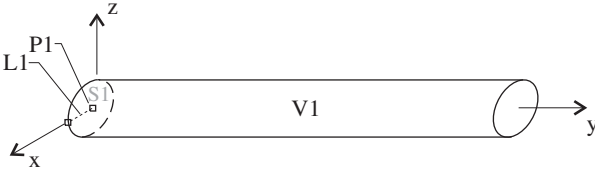
### Defining model control data


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

*Master degrees of freedom:* Choose Control→Degrees of Freedom, uncheck the X-Rotation, Y-Rotation and Z-Rotation buttons and click OK.


### Defining model geometry


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




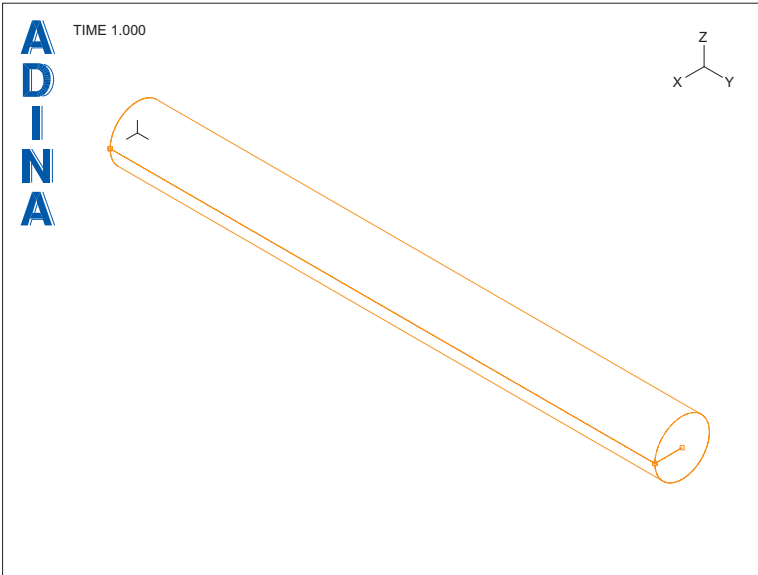
*Geometry point:* Click the Define Points icon , enter the following information into the table and click OK.

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.

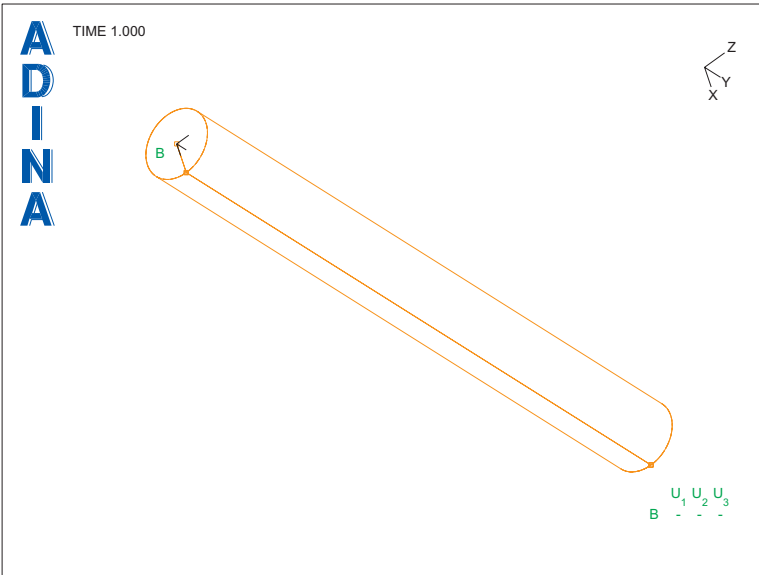


### Applying boundary conditions


We will fix surface 1. Click the Apply Fixity icon , set the “Apply to” field to Surfaces, 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.)




### 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 Site # to 6. Click OK to close the Apply Load dialog box.

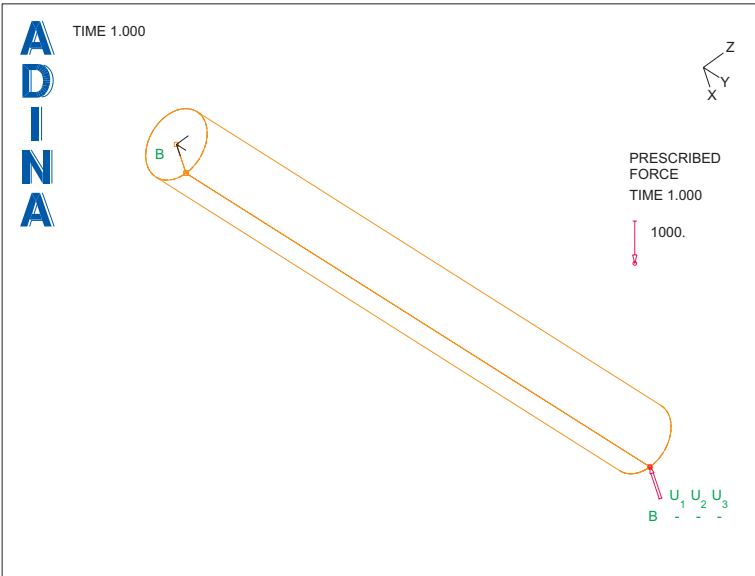
When you click the Load Plot icon , the graphics window should look something like the figure on the next page.

### 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 Define 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.

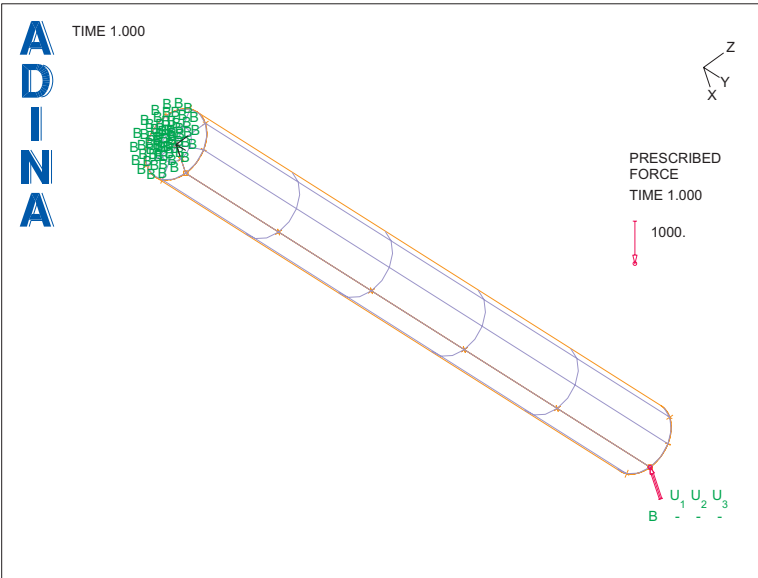
Click the Subdivide Volumes icon  and set the Number of Subdivisions in the u, v and w directions to 8, 5 and 2 respectively. Then click OK.

*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 data file, running ADINA, loading the porthole file

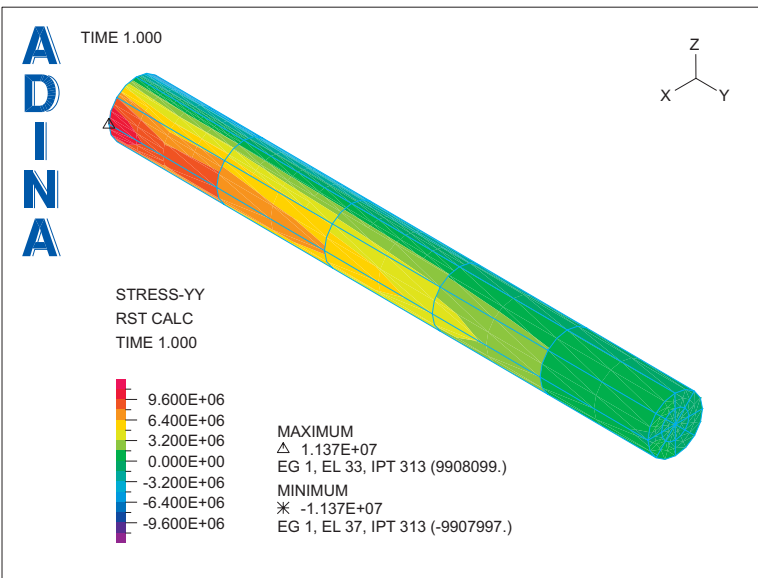
First click the Save icon  and save the database to file prob04. To generate the ADINA data file and run ADINA, click the Data File/Solution icon , set the file name to prob04, make sure that the Run Solution button is checked and click Save. When ADINA is finished, close all open dialog boxes. Choose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

Problem 4: Cylinder subjected to tip load




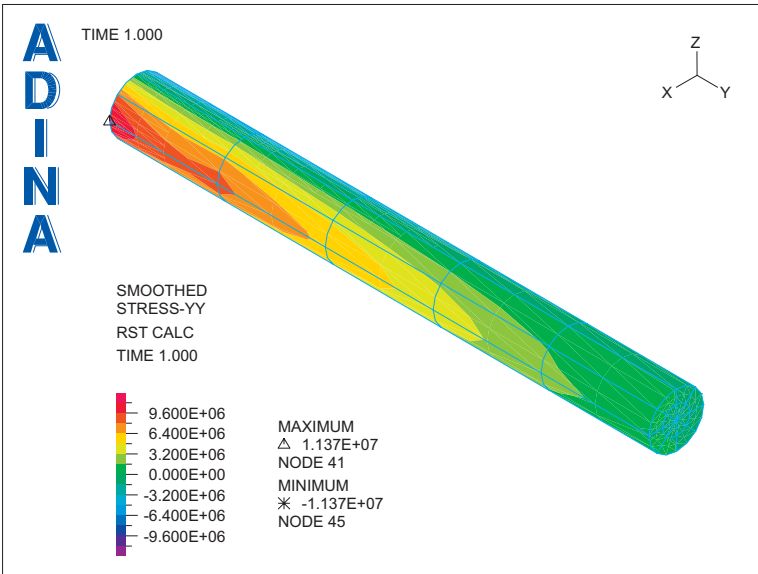
Examining the solution

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.)

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

This page intentionally left blank.