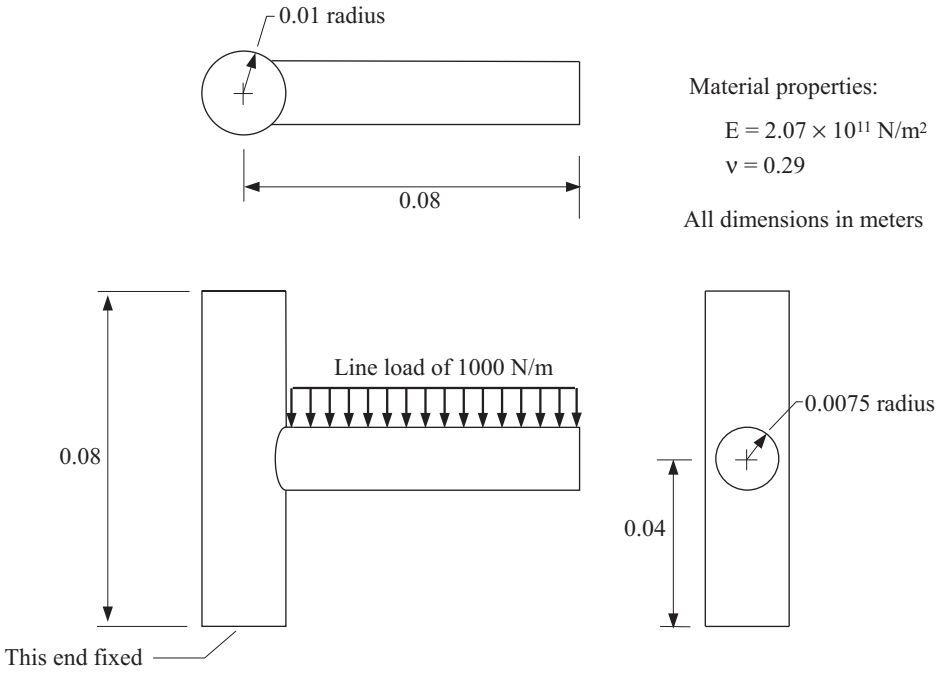
Problem description

It is desired to analyze the shell-shell intersection shown:



Shell-shell intersection dimensions and loading

The purpose of this analysis is to demonstrate the usage of ADINA-M/PS (the ADINA Modeler using the Parasolid geometry kernel) and ADINA-M/OC (the ADINA Modeler using the Open Cascade geometry kernel) in the analysis of shell structures.

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

- Meshing ADINA-M faces with shell elements
- Plotting bending moments and membrane forces in shells
- Using ADINA-M/OC

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.

Note that you must have an ADINA-M/PS license to do the Parasolid part of this problem, and an ADINA-M/OC license to do the Open Cascade part of this problem.

This problem cannot be solved with the 900 nodes version of the ADINA System because the 900 nodes version of the ADINA System does not include ADINA-M/PS or ADINA-M/OC.

Analysis using ADINA-M/PS

Invoking the AUI and choosing the finite element program

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

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 29: Analysis of a shell-shell intersection with ADINA-M/PS" and click OK.

Defining model geometry

Vertical pipe: Click the Define Bodies icon  , add body 1, set the Type to Cylinder, the Radius to 0.01, the Length to 0.08, the Center Position to (0.0, 0.0, 0.04), the Axis to Z and click Save.

Horizontal pipe: Add body 2, set the Type to Cylinder, the Radius to 0.0075, the Length to 0.08, the Center Position to (0.0, 0.04, 0.04), the Axis to Y and click OK.

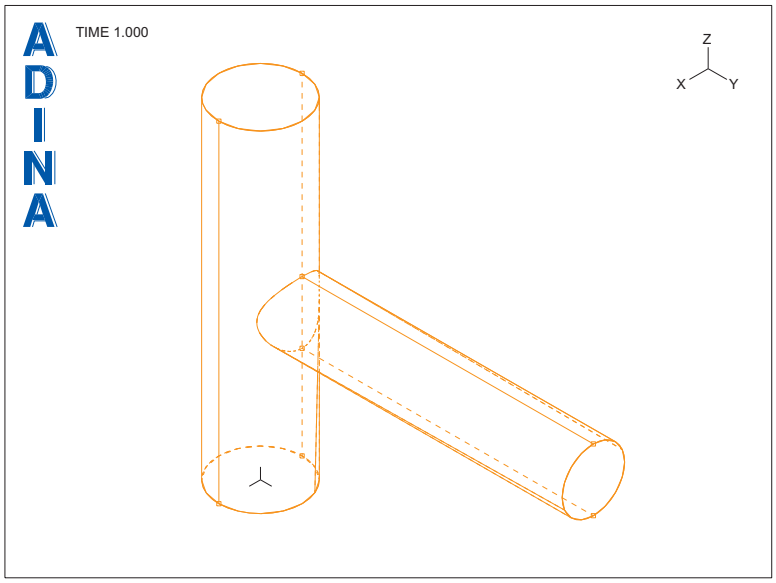
Merging the pipes: Click the Boolean Operator icon  , make sure that the Operator Type is Merge and that the Target Body is 1, enter 2 in the first row of the table and click OK.

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

Specifying boundary conditions, loads and the material

Fixities: Click the Apply Fixity icon  , set the "Apply to" field to Edge/Line, enter the following information in the first two rows of the table and click OK.

Edge/Line #	Body #
1	1
4	1

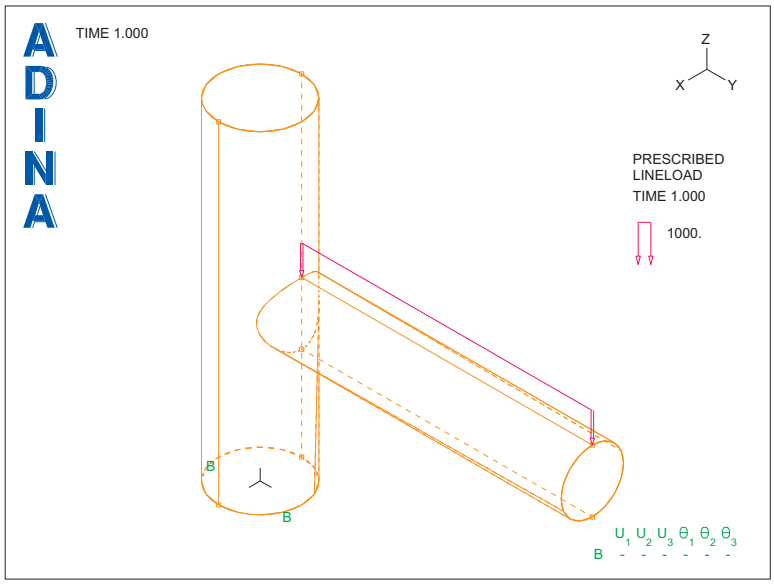


Loads: Click the Apply Load icon , set the Load Type to Distributed Line Load and click the Define... button to the right of the Load Number field. In the Define Distributed Line Load dialog box, add line load 1, set the Magnitude to -1000 and click OK. In the Apply Load dialog box, set the "Apply To" field to Edge, then, in the first row of the table, set the Edge # to 10, the Body # to 1 and the Aux Point to 8. Click OK to close the Apply Load dialog box.

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

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.

Shell thicknesses: Choose Geometry→Faces→Thickness, set the thickness for faces 1, 4, 5, 7 to 0.0005 and click OK.

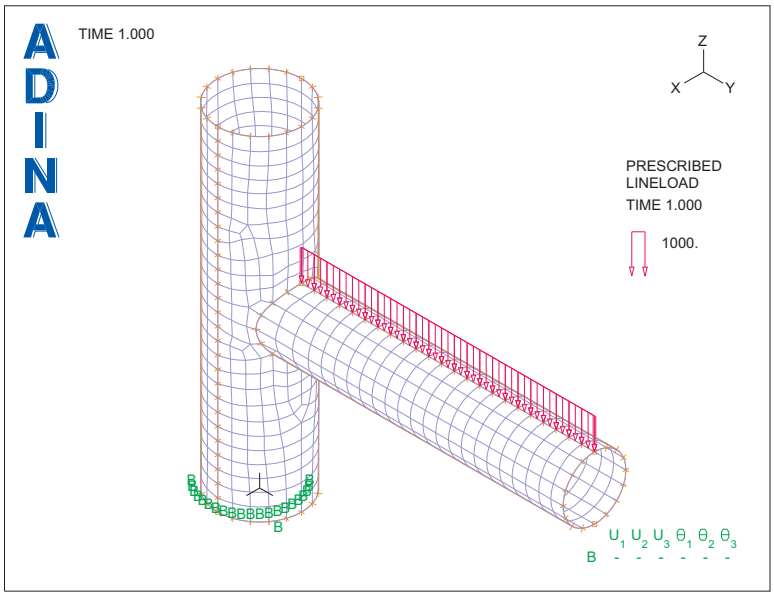


Meshing

Element group: Click the Element Groups icon  , add element group number 1, set the Type to Shell, click the Advanced tab, check the "Calculate Midsurface Forces and Moments" button and click OK. It is necessary to calculate the midsurface forces and moments so that bending moments can be displayed.

Subdivision data: We will specify a uniform element size throughout the ADINA-M geometry. Choose Meshing→Mesh Density→Complete Model, set the "Subdivision Mode" to "Use Length", set the Element Edge Length to 0.003 and click OK.

Meshing: First, click the Hidden Surfaces Removed icon  (we do not want to see dashed hidden lines in the elements that we generate). Now click the Mesh Faces icon  , set the Nodes per Element to 9, enter 1, 4, 5, 7 in the first four rows of the table, and click OK. The graphics window should look something like the figure on the next page.

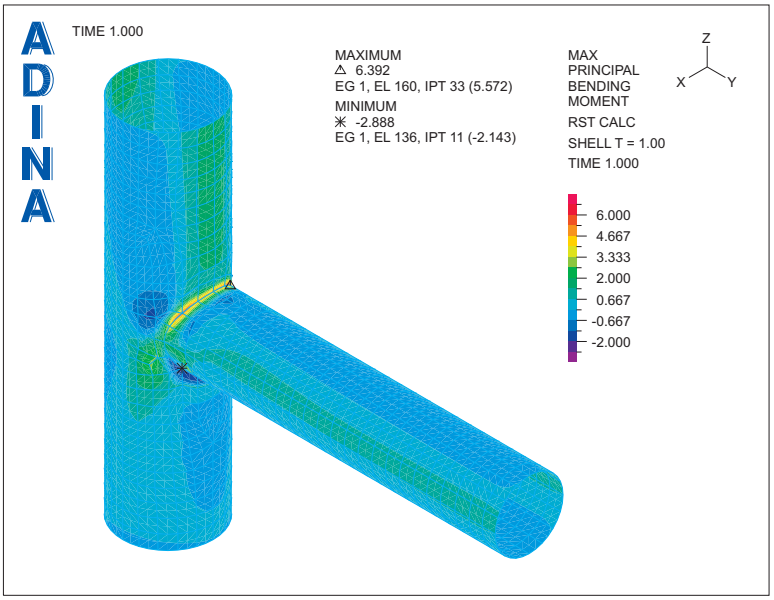


Generating the data file, running ADINA Structures, loading the porthole file

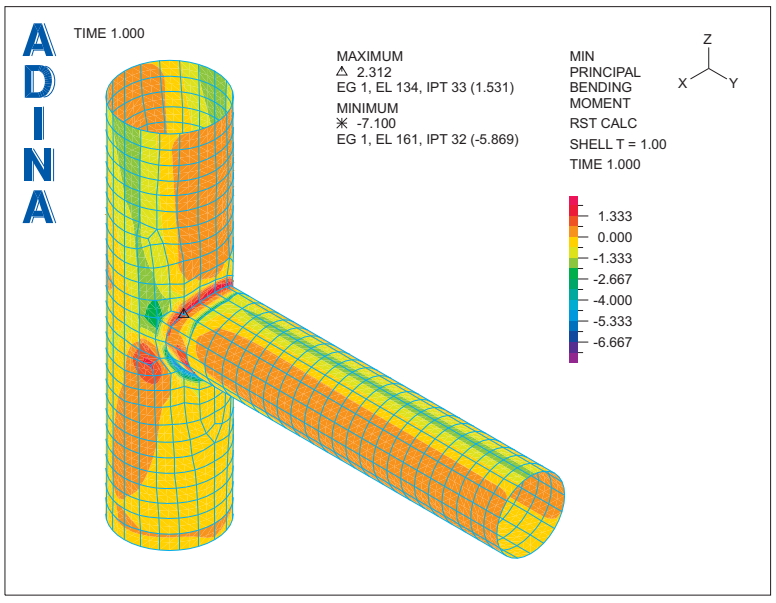
Click the Save icon and save the database to file `prob29`. Click the Data File/Solution icon, set the file name to `prob29`, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob29`.

Plotting the bending moments and membrane forces

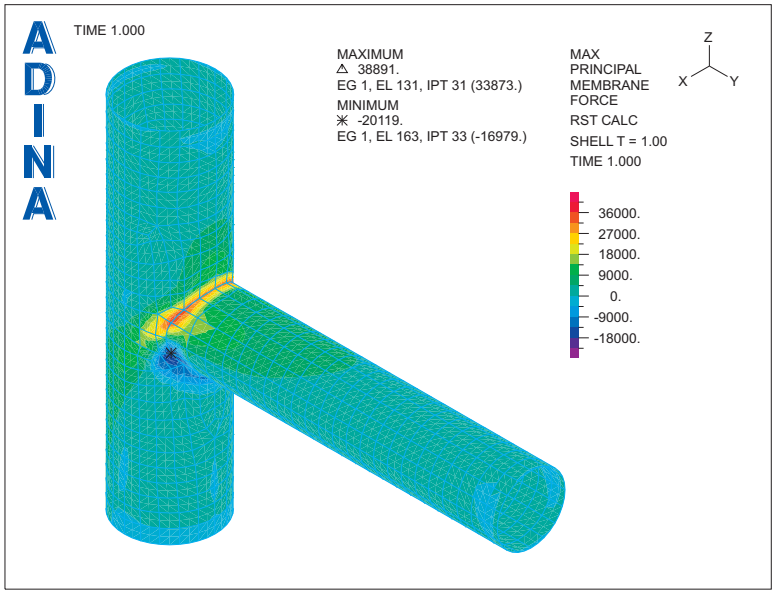
Bending moments: Click the Create Band Plot icon, set the Band Plot Variable to (Force: MAX_PRINCIPAL_BENDING_MOMENT) and click OK. The graphics window should look something like the top figure on the next page. The maximum principal bending moment is about 6.392 (N-m/m). (Your results may be slightly different because free meshing produces different meshes on different platforms.)



Now click the Modify Band Plot icon  , set the Band Plot Variable to (Force: MIN_PRINCIPAL_BENDING_MOMENT) and click OK. The graphics window should look something like this The minimum principal bending moment is about -7.100 (N-m/m).



Membrane forces: Click the Modify Band Plot icon  , set the Band Plot Variable to (Force: MAX_PRINCIPAL_MEMBRANE_FORCE) and click OK. The graphics window should look something like this. The maximum principal membrane force is about 38891 (N/m).



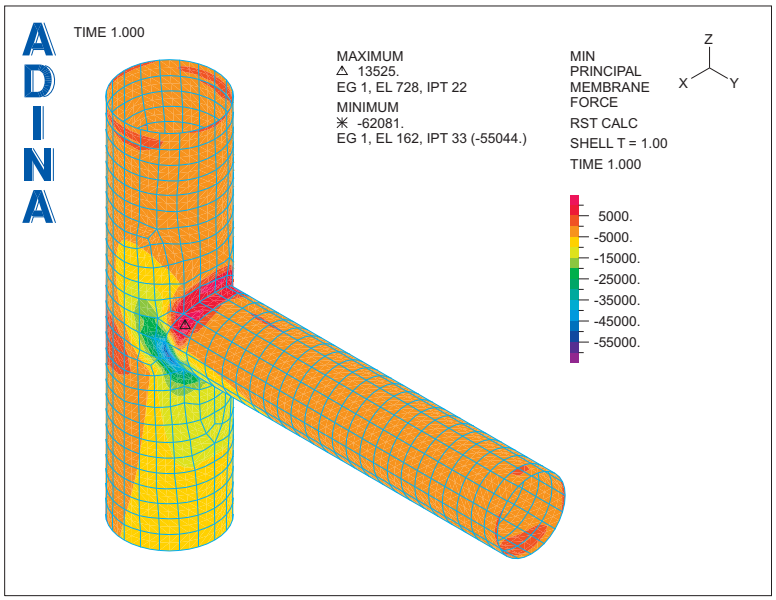
Now click the Modify Band Plot icon  , set the Band Plot Variable to (Force: MIN_PRINCIPAL_MEMBRANE_FORCE) and click OK. The graphics window should look something like the top figure on the next page The minimum principal membrane force is about -62081 (N/m).

Exiting the AUI: Choose File→Exit (you can discard all changes).

Analysis using ADINA-M/OC

Invoking the AUI and choosing the finite element program

Invoke the AUI with the ADINA-M/OC modeler (for example, using the command `aui9.3 -occ` for Linux versions) and set the Program Module drop-down list to ADINA Structures.



Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 29: Analysis of a shell-shell intersection with ADINA-M/OC" and click OK.

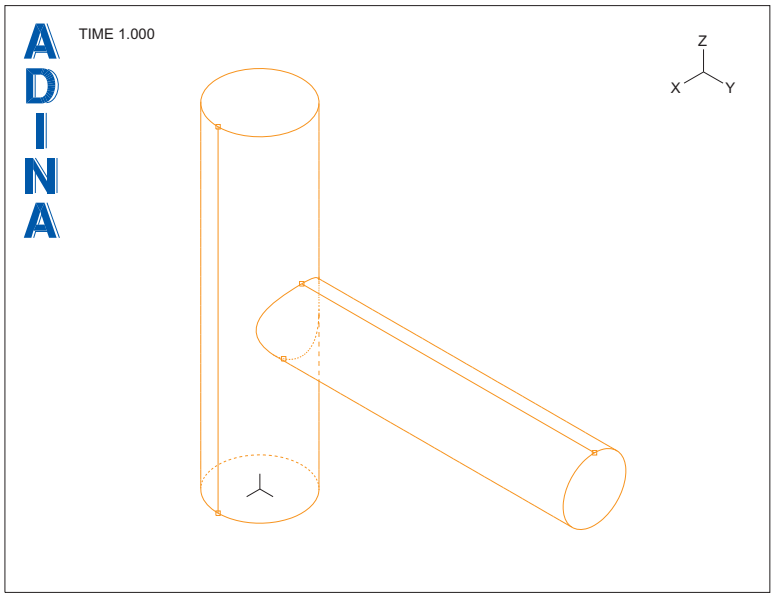
Defining model geometry

Vertical pipe: Click the Define Bodies icon  , add body 1, set the Type to Cylinder, the Radius to 0.01, the Length to 0.08, the Center Position to (0.0, 0.0, 0.04), the Axis to Z and click Save.

Horizontal pipe: Add body 2, set the Type to Cylinder, the Radius to 0.0075, the Length to 0.08, the Center Position to (0.0, 0.04, 0.04), the Axis to Y and click OK.

Merging the pipes: Click the Boolean Operator icon  , make sure that the Operator Type is Merge and that the Target Body is 1, enter 2 in the first row of the table and click OK.

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



Notice that the boundary representation of the body by faces is different between Parasolid and Open Cascade. In addition, the boundary representation is slightly different between the Open Cascade version on Linux and the Open Cascade version on Windows. (The above picture is obtained using the Linux version.)

Linux: Notice that geometry point 6 is not on a line of symmetry of the model.

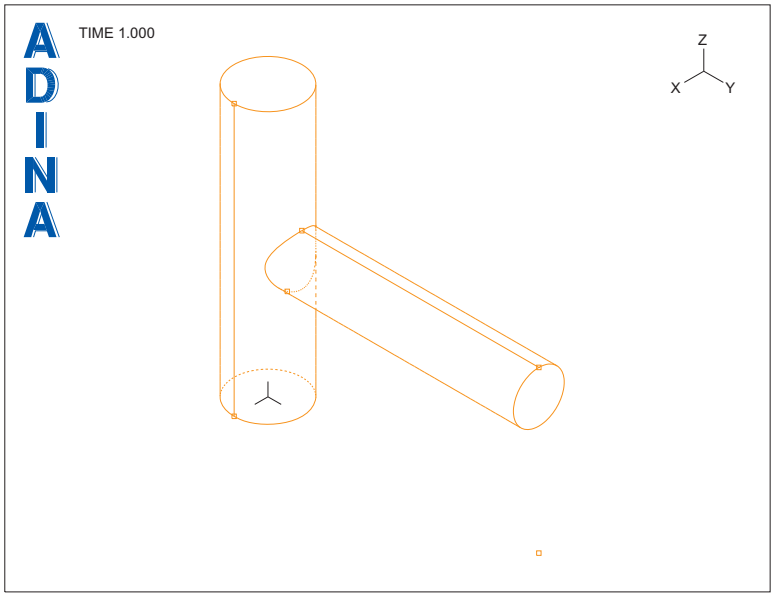
Windows: Notice that geometry point 5 is not on a line of symmetry of the model.

Specifying boundary conditions, loads and the material

Fixities: Click the Apply Fixity icon, set the "Apply to" field to Edge/Line, enter the following information in the first row of the table and click OK.

Edge/Line #	Body #
5	1

Loads: Recall that in the Parasolid model, we defined the distributed line load using an auxiliary point on the symmetry plane of the model. For the Open Cascade model, there is no suitable point on the symmetry plane of the model. Click the Define Points icon, add point 7 at coordinate (0, 0.08, 0) and click OK. To see this point, you might need to use the Pick icon and the mouse to shrink the mesh plot slightly. The graphics window should look something like the figure on the next page.

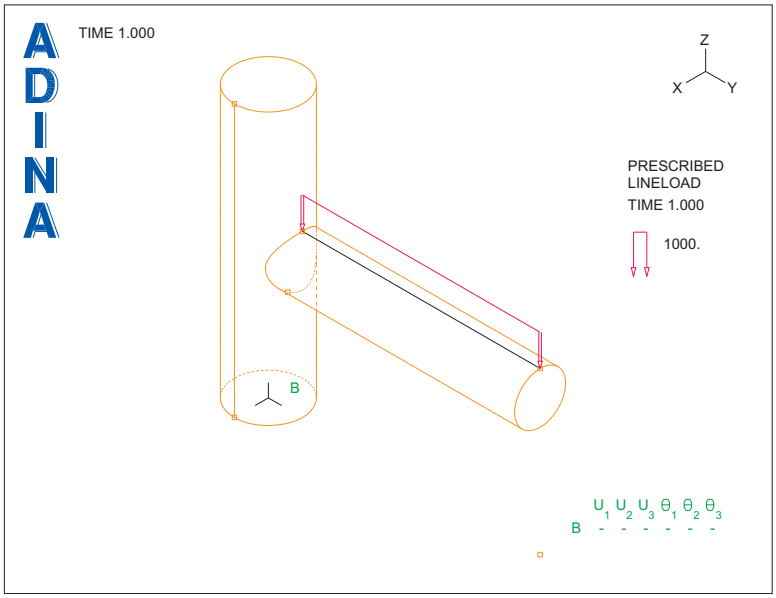


Now click the Apply Load icon, set the Load Type to Distributed Line Load and click the Define... button to the right of the Load Number field. In the Define Distributed Line Load dialog box, add line load 1, set the Magnitude to -1000 and click OK. In the Apply Load dialog box, set the "Apply To" field to Edge, then, in the first row of the table, set the Edge # to 6, the Body # to 1 and the Aux Point to 7. Click OK to close the Apply Load dialog box.

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

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.

Shell thicknesses: Choose Geometry→Faces→Thickness, set the thickness for faces 2, 3 to 0.0005 and click OK.



Meshing

Element group: Click the Element Groups icon, add element group number 1, set the Type to Shell, click the Advanced tab, check the "Calculate Midsurface Forces and Moments" button and click OK.

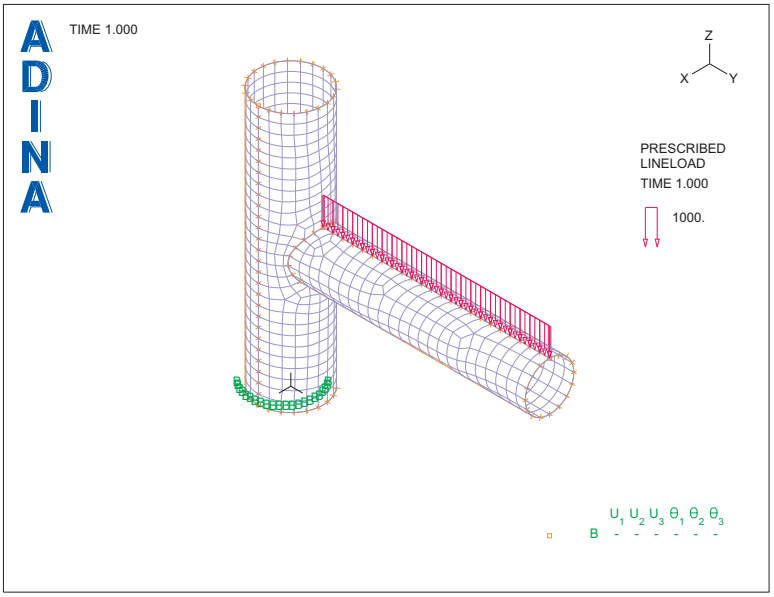
Subdivision data: We will specify a uniform element size throughout the ADINA-M geometry. Choose Meshing→Mesh Density→Complete Model, set the "Subdivision Mode" to "Use Length", set the Element Edge Length to 0.003 and click OK.

Linux version

Meshing: First, click the Hidden Surfaces Removed icon, then click the Mesh Faces icon, set the Nodes per Element to 9, enter 2, 3 in the first two rows of the table, and click OK. The AUI gives the following warning message:

```
Odd number of subdivisions for face 2 of body 1.
Program will automatically refine edge 1 of body 1.
Odd number of subdivisions for face 3 of body 1.
Program will automatically refine edge 2 of body 1.
Odd number of subdivisions for face 2 of body 1.
Program will automatically refine edge 5 of body 1.
```

Click OK to close the warning message. The graphics window should look something like this:

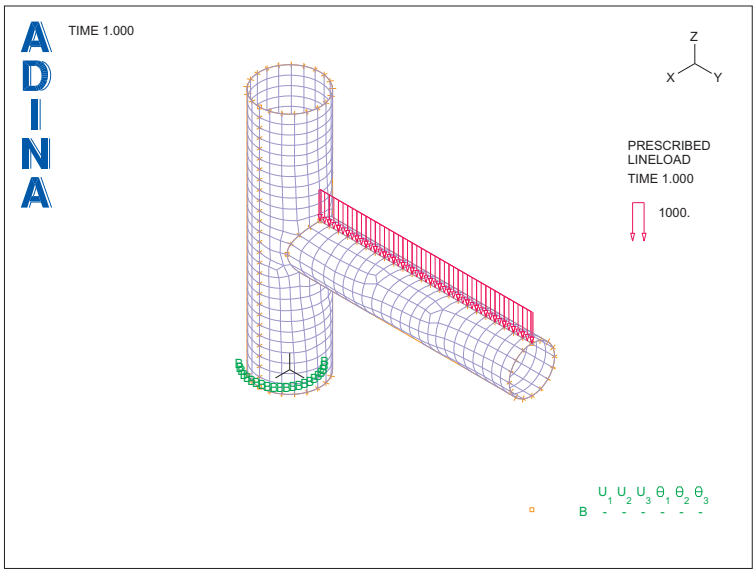


Windows version

Meshing: First, click the Hidden Surfaces Removed icon, then click the Mesh Faces icon, set the Nodes per Element to 9, enter 2, 3 in the first two rows of the table, and click OK. The graphics window should look something like the top figure on the next page.

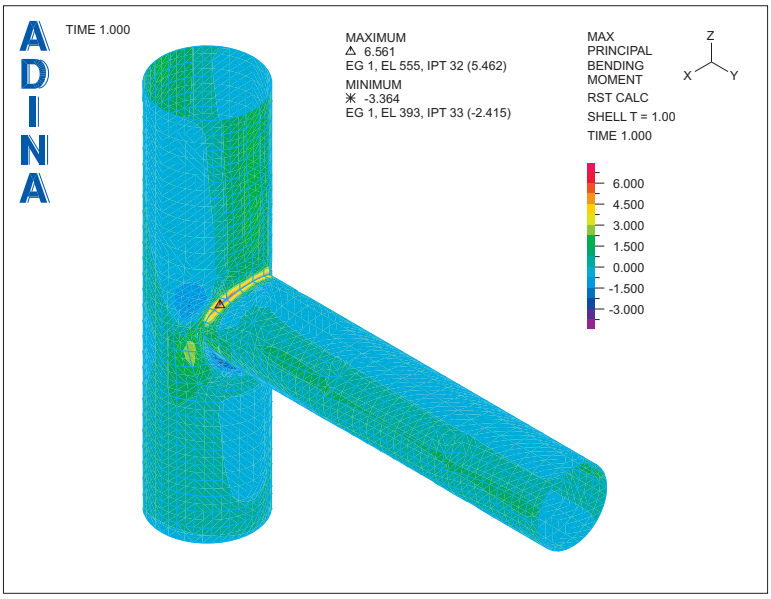
Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon and save the database to file `prob29_oc`. Click the Data File/Solution icon, set the file name to `prob29_oc`, 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 `prob29_oc`.



Plotting the bending moments and membrane forces

Follow the instructions given above to plot the principal bending moments and membrane forces. For example, the maximum principal bending moment should look something like this (this figure is obtained using the Linux version):



The results from the Parasolid and Open Cascade analyses are very similar.

Exiting the AUI: Choose File→Exit (you can discard all changes).