

Problem description

A shell corner is analyzed first for its static response due to a concentrated load, then for its natural frequencies and mode shapes.

In the static analysis, we will demonstrate the following topics that have not been presented in previous problems:

- Defining shell thicknesses
- Plotting 5 and 6 DOF nodes
- Plotting shell thicknesses
- Plotting results on the top, midsurface and bottom of the shell

In the frequency analysis, we will demonstrate the following topics that have not been presented in previous problems:

- Setting up a frequency analysis
- Plotting mode shapes

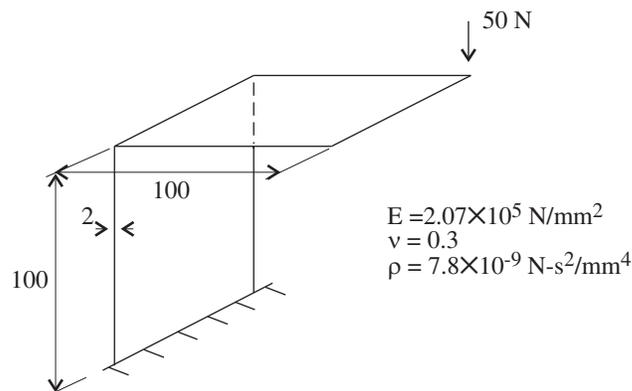
We assume that you have worked through problems 1 to 7, or have equivalent experience with the ADINA System.

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the 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.

Static analysis



Invoking the AUI and choosing the finite element program

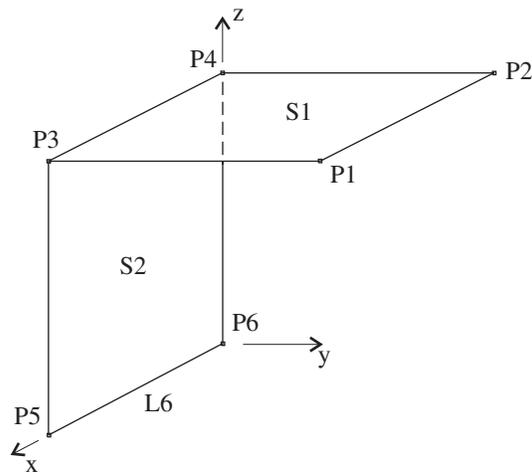
Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading “Problem 8: Analysis of a shell corner, static analysis” and click OK.

Defining model geometry

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



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

Point #	X1	X2	X3
1	100	100	100
2	0	100	100
3	100	0	100
4	0	0	100
5	100	0	0
6	0	0	0

Geometry surfaces: Click the Define Surfaces icon , define the following surfaces and click OK.

Surface Number	Type	Point 1	Point 2	Point 3	Point 4
1	Vertex	1	2	4	3
2	Vertex	3	4	6	5

Defining and applying loads

Click the Apply Load icon , make sure that the Load Type is set to 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 50, the Z direction to -1 and click OK. In the first row of the table in the Apply Load dialog box, set the Point # to 2, then click OK.

Applying the boundary conditions

We will fix the bottom of the structure, which corresponds to line 6 in the model (use the Query icon  and the mouse to confirm the line number). Click the Apply Fixity icon , set the “Apply to” field to Edge/Line, set the Edge/Line to 6 in the first row of the table and click OK.

Click the Boundary Plot icon  and the Load Plot icon . Then use the mouse to rotate the mesh plot until the graphics window looks something like the figure on the next page.

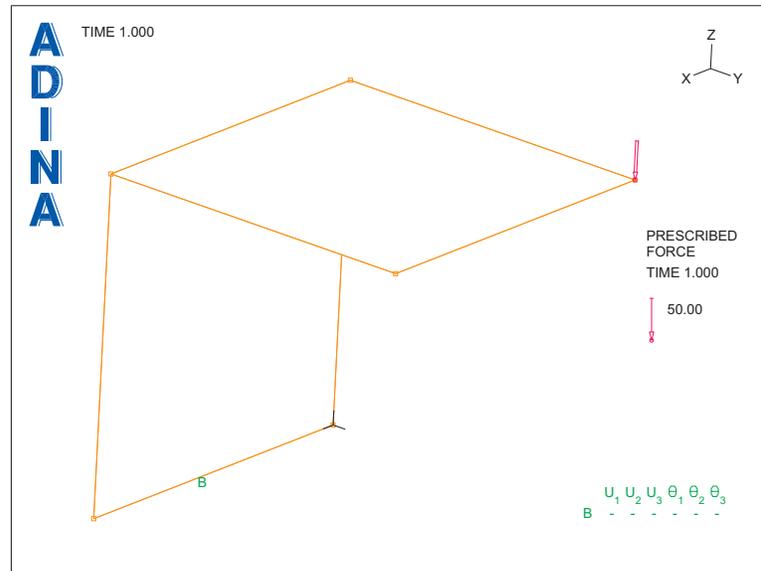
Defining the material

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 2.07E5, the Poisson's ratio to 0.29 and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the shell thicknesses

Choose Geometry→Surfaces→Thickness, enter 2.0 in the Thickness column for both surfaces and click OK.

Problem 8: Analysis of a shell corner

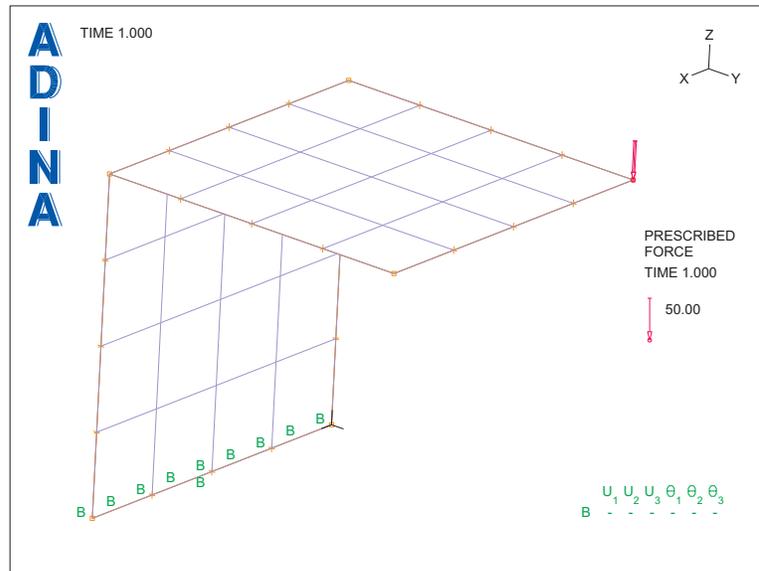


Defining the elements

Element group: Click the Element Groups icon , add element group number 1, set the Type to Shell, click the Advanced tab, set the Stress Reference System to Local and click OK.

Subdivision data: We will use a uniform mesh for the solution. Choose Meshing→ Mesh Density→Complete Model, set the “Subdivision Mode” to “Use Length”, set the “Element Edge Length” to 25 and click OK.

Element generation: Click the Mesh Surfaces icon , set the “Nodes per Element” to 9, enter 1 and 2 in the first two rows of the table and click OK. The graphics window should look something like the figure on the next page.



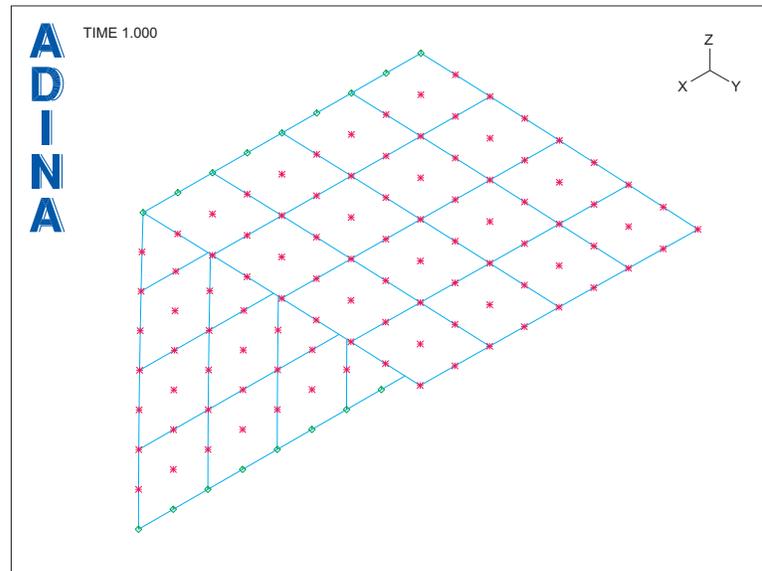
Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

First click the Save icon  and save the database to file prob08. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon , set the file name to prob08, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes. Then set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob08.

Examining the solution

Visualizing the number of rotational degrees of freedom for the nodes: We would like to see which nodes are assigned 5 degrees of freedom and which nodes are assigned 6 degrees of freedom. Choose Display→Geometry/Mesh Plot→Define Style, set the Node Depiction field to ROTATIONAL_DOF using the drop-down list and click OK. Then click the Clear icon  and the Mesh Plot icon . The graphics window should look something like the figure on the next page.

Problem 8: Analysis of a shell corner



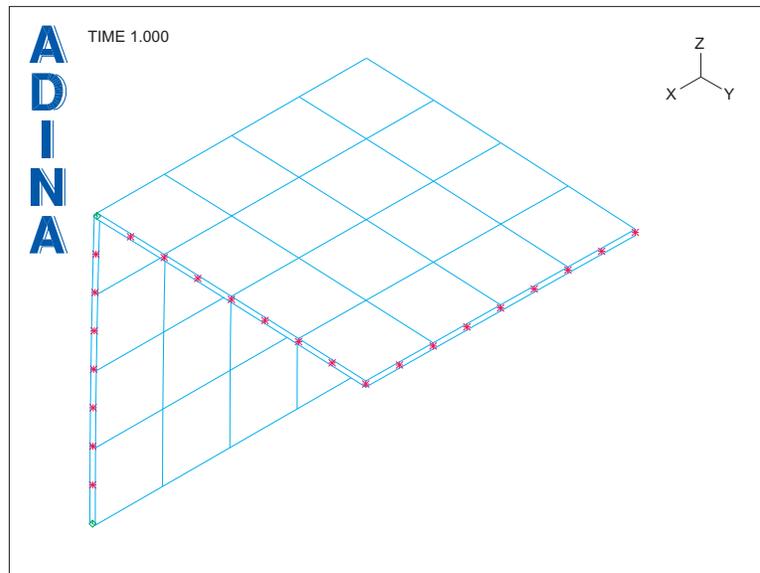
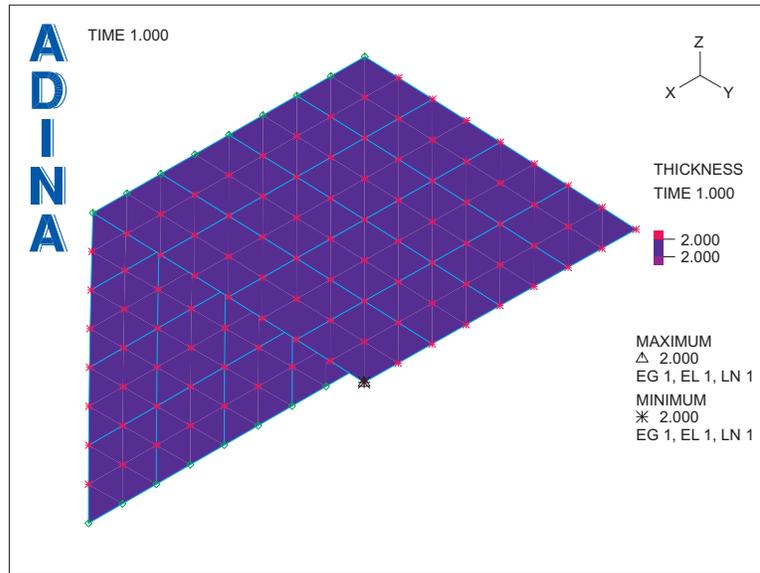
The nodes that are assigned 5 degrees of freedom are drawn in red and the nodes that are assigned 6 degrees of freedom are drawn in green. Notice that the 6 degree of freedom nodes are on the shell-shell intersection and on the built-in edge.

Visualizing the shell thickness: Click the Create Band Plot icon , set the Band Plot Variable to (Thickness: THICKNESS) and click OK. The graphics window should look something like the top figure on the next page.

Click the Clear Band Plot icon  to delete the band plot.

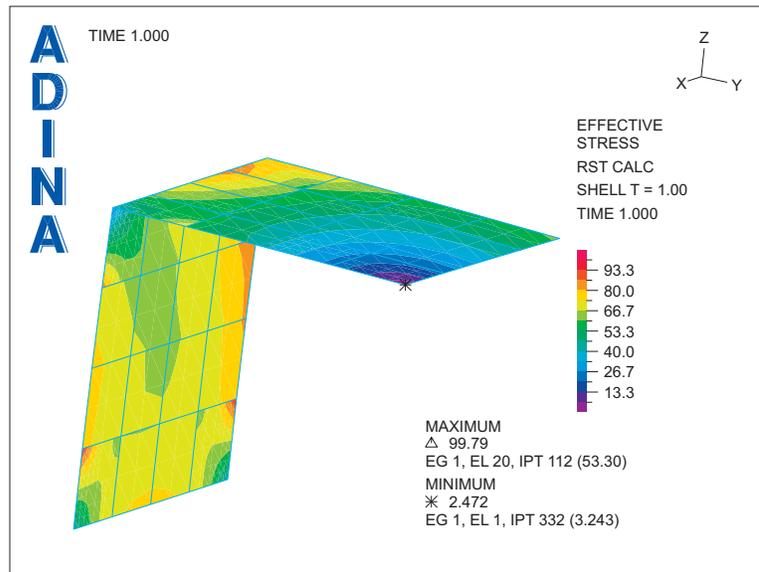
You can also plot the shell elements as if they were solid elements (that is, with the actual shell thicknesses). Click the Modify Mesh Plot icon , click the Element Depiction... button, set the “Appearance of Shell Element” to Top/Bottom and click OK twice to close both dialog boxes. The graphics window should look something like the bottom figure on the next page.

For the remaining plots, we do not want to plot the nodes. Click the Reset Mesh Plot Style  icon. Then click the Clear icon  and the Mesh Plot icon .



Problem 8: Analysis of a shell corner

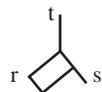
Effective stresses: Click the Quick Band Plot icon . Use the Pick icon  and the mouse until the graphics window looks something like this:



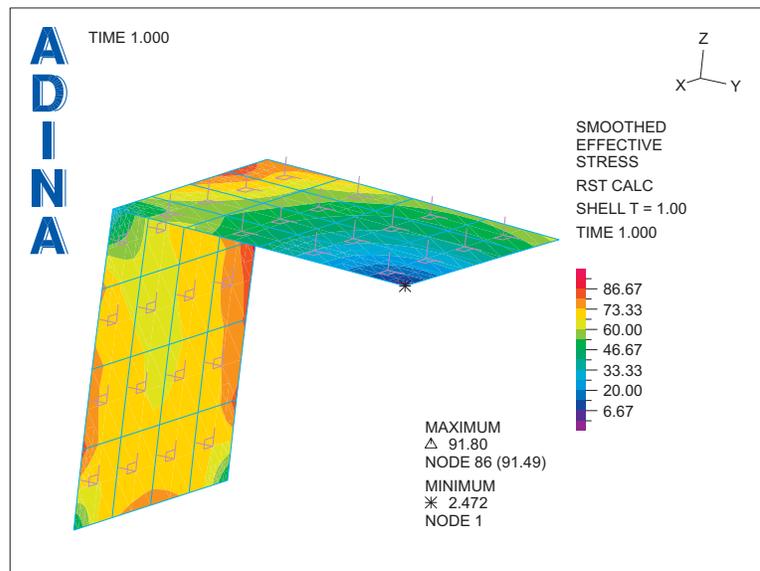
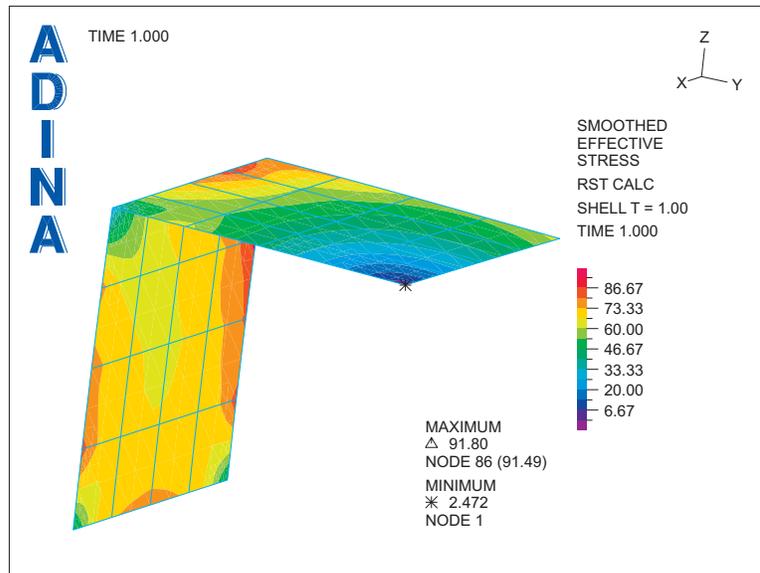
The bands appear to have breaks between elements. (This is an indication that the mesh may not be fine enough.) To smooth the band plot, click the Smooth Plots icon . The graphics window should look something like the top figure on the next page.

These are the results as presented onto the top of the shells. To find out which direction corresponds to the top of the shells, click the Modify Mesh Plot icon  and click the Element Depiction... button. Check the Display Local System Triad button and click OK twice to close both dialog boxes.

The graphics window should look something like the bottom figure on the next page. A triad is plotted within each element showing the direction of the element local coordinate system. The triad is interpreted as follows:

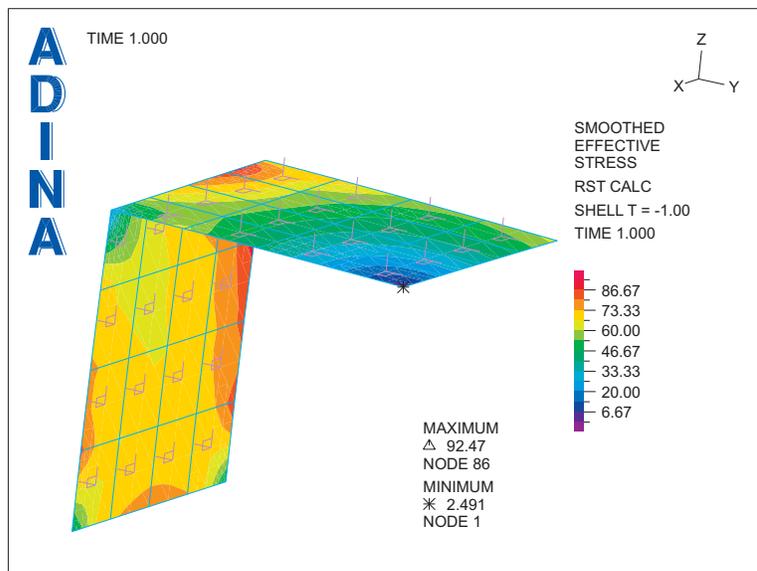


in which r , s and t are the element local coordinates.



Problem 8: Analysis of a shell corner

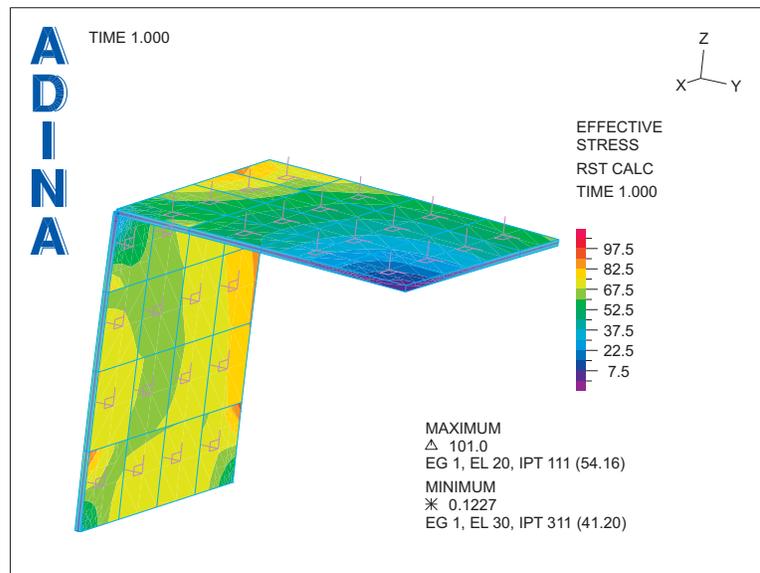
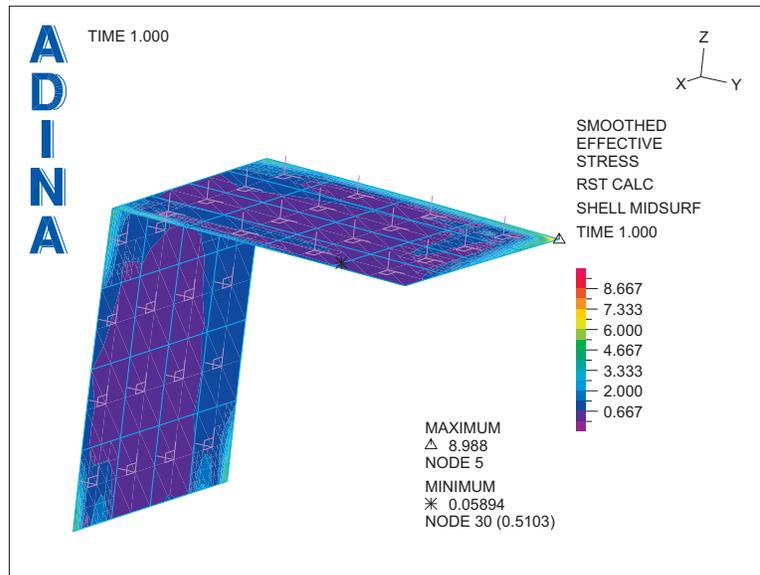
To plot the results corresponding to the bottom of the shells, click the Modify Band Plot icon , click the Result Control... button, set the t coordinate field within the “Calculation of Shell Element Results on Midsurface” box to -1 and click OK twice to close both dialog boxes. The graphics window should look something like this:



To plot the results corresponding to the shell midsurface, click the Modify Band Plot icon  and click the Result Control... button. Choose “From Shell Midsurface” within the “Calculation of Shell Element Results on Midsurface” box and click OK twice to close both dialog boxes.

The graphics window should look something like the top figure on the next page.

You can also plot the stresses on the top-bottom depiction of the shell elements that we presented earlier. Click the Modify Mesh Plot icon , click the Element Depiction... button, set the “Appearance of Shell Element” to Top/Bottom and click OK twice to close both dialog boxes. The AUI doesn’t display anything, because the AUI cannot plot the smoothed stresses on a top-bottom depiction. Click the Clear Band Plot icon , then click the Quick Band Plot icon . The graphics window should look something like the bottom figure on the next page.



Frequency analysis

Now we will determine the first few natural frequencies and mode shapes for the shell structure.

Invoking the pre-processor: Set the Program Module drop-down list to ADINA Structures (you can discard all changes). Choose prob08.ldb from the recent file list near the bottom of the File menu.

Change the heading to “Problem 8: Analysis of a shell corner, frequency analysis”.

Removing the concentrated load: In the Model Tree, click on the + next to the Loading text, then right-click on the ‘1. Force 1 on Point 2’ text, choose Delete... and click Yes to answer the prompt. Click the Load Plot icon  to plot the mesh without the load.

Defining a frequency analysis: Set the Analysis Type drop-down list to Frequencies/Modes. Then click the Analysis Options icon , set the Number of Frequencies/Mode Shapes to 6 and click OK.

Modifying the material: In the Model Tree, click on the + next to the Material text, then right-click on the ‘1. Elastic’ text and choose Modify.... In the Define Isotropic Linear Elastic Material dialog box, set the Density to 7.8E-9 and click OK.

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

First choose File→Save As and save the database to file prob08a. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon , set the file name to prob08a, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes. Then set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob08a.

Examining the solution

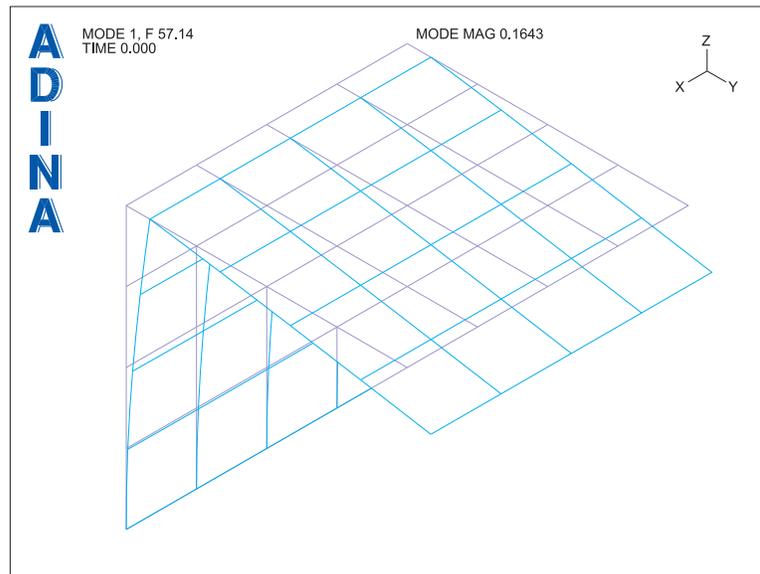
Determining which solutions were calculated by ADINA Structures: Choose List→Info→Response. ADINA Structures calculated the following modal data for reference time 0.0: natural frequencies, physical error norms and mode shapes, all for modes 1 to 6. Click Close to close the dialog box.

Listing the natural frequencies: Choose List→Value List→Zone, set Variable 1 to (Frequency/Mode: FREQUENCY) and click Apply. The frequencies are: 5.71434E+01 (Hz),

1.09713E+02, 1.56364E+02, 3.75121E+02, 7.60300E+02, 1.11636E+03. Click Close to close the dialog box.

Plotting the first mode shape: The AUI displays the first mode shape. We would also like to display the original (undeformed) mesh in the same plot. Click the Show Original Mesh icon

. The graphics window should look something like this:



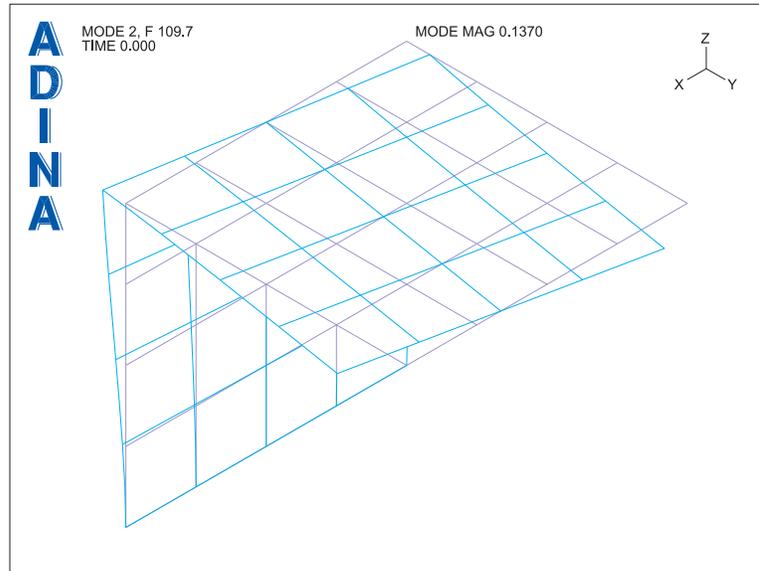
To create a movie showing the model moving in its first mode shape, click the Movie Mode Shape icon . To animate the movie, click the Animate icon . To loop the movie 10 times, choose Display→Animate, set the Number of Cycles to 10 and click OK.

Click the Refresh icon  to clear the animation. Now, to show mode shape 2, click the Next Solution icon . The graphics window should look something like the figure on the next page.

You can use the Previous Solution icon , Next Solution icon , First Solution icon  and Last Solution icon  to view all of the other modes.

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

Problem 8: Analysis of a shell corner



Additional note

Because the model in this primer problem is linear, the natural frequencies do not depend on the applied loads (in this case, the concentrated load).

However, if the model is nonlinear, the natural frequencies depend on the applied loads. To obtain the natural frequencies of a nonlinear model in the presence of applied loads, it is necessary to first obtain the static solution corresponding to the applied loads, then perform a frequency analysis using the static solution as initial conditions. This frequency analysis is performed as a restart run. Primer problem 26 demonstrates the procedure.