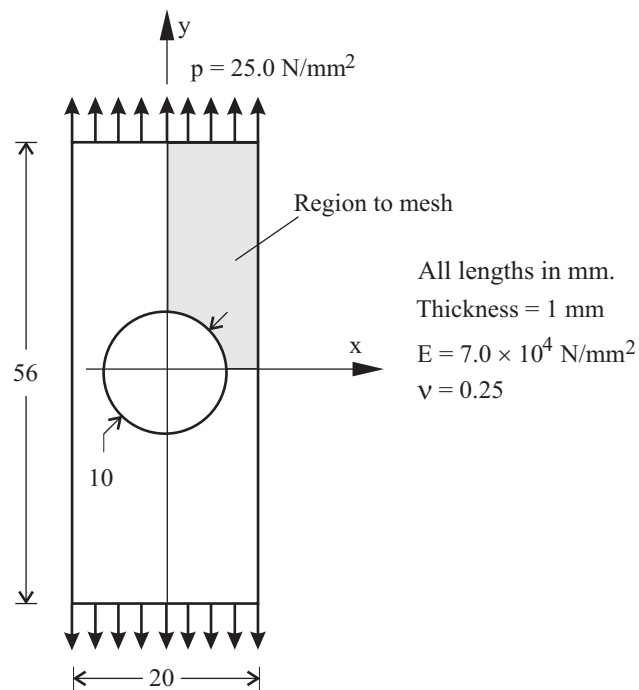


Problem description

A plate with a hole is subjected to tension as shown:



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

- Defining a problem heading
- Selecting master degrees of freedom
- Entering a geometry point in a cylindrical coordinate system
- Defining several geometry surfaces
- Defining boundary conditions
- Querying the graphics with the mouse
- Generating elements onto geometry surfaces, ensuring compatibility
- Displaying geometry point, line and surface numbers
- Zooming into the graphics window
- Plotting original and deformed meshes
- Moving and resizing mesh plots with the mouse
- Deleting unwanted text with the mouse
- Plotting element vectors
- Plotting stress bands

Problem 2: Plate with a hole in tension

- Graphing stresses along a line
- Modifying a graph plot

We assume that you have worked through problem 1, 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.

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

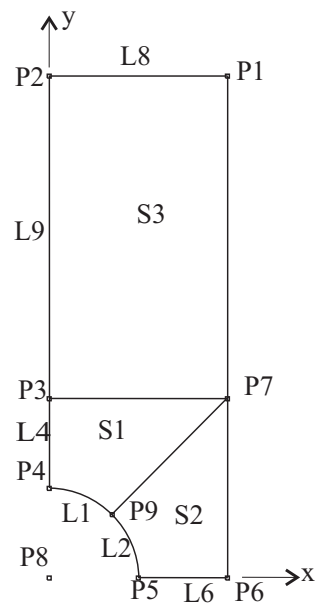
Problem heading: Choose Control→Heading, enter the heading “Problem 2: Plate with a hole in tension” and click OK.


Plane for 2D elements: Choose Control→Miscellaneous Options, set the "2D Solid Elements in" field to "XY-Plane, Y-Axisymmetric" and click OK.

Master degrees of freedom: Choose Control→Degrees of Freedom, uncheck the Z-Translation, X-Rotation, Y-Rotation and Z-Rotation buttons and click OK. (We perform this step because the two-dimensional solid elements that we will use only provide stiffness for the x-translation and y-translation degrees of freedom. If you omit this step, the AUI will delete the z-translation, x-rotation, y-rotation and z-rotation degrees of freedom for all the nodes when you generate the ADINA Structures data file. So this step is not necessary; ADINA Structures will give the same solution whether or not you perform this step. However, ADINA Structures runs more efficiently if you perform this step.)

Defining model geometry


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



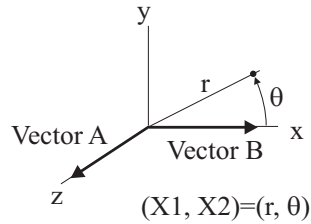
Points: Click the Define Points icon , enter the following information into the X1, X2 columns of the table (you can leave the X3 column blank), then click OK.


Point #	X1	X2
1	10	28
2	0	28
3	0	10
4	0	5
5	5	0
6	10	0
7	10	10
8	0	0

Problem 2: Plate with a hole in tension

We also need a point mid-way along the hole. The coordinates of this point are most conveniently entered using a cylindrical coordinate system. Click the Coordinate Systems icon , add coordinate system 1 and set the Type to Cylindrical. Set the components of Vector A to (0, 0, 1), the components of Vector B to (1, 0, 0) and click OK.

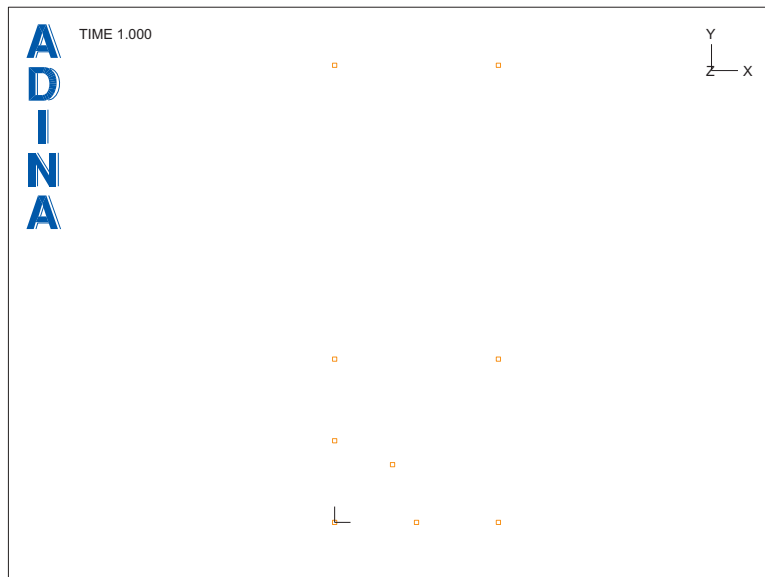
(Vector A gives the direction of the cylindrical axis, and vector B is used to determine the line corresponding to zero angle θ , as shown in the following figure.)




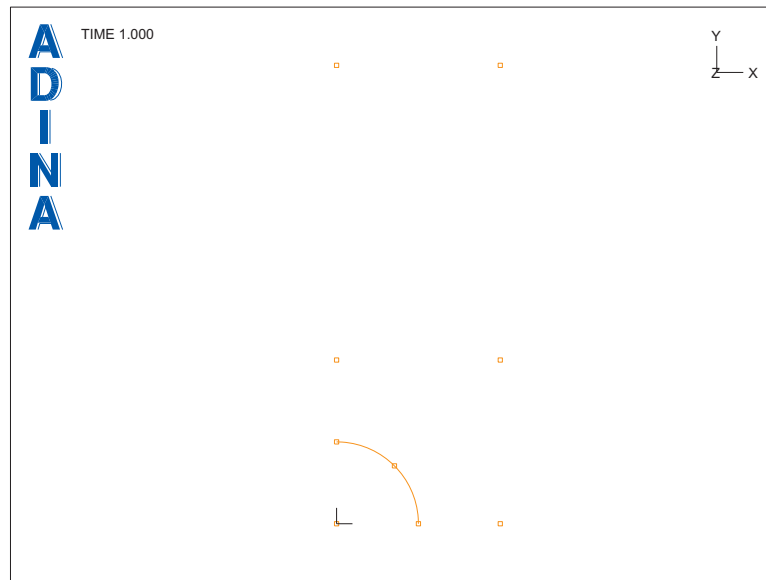
Then click the Define Points icon , add an additional row to the table if necessary, enter


Point #	X1	X2	X3
9	5	45	0

and click OK. The graphics window should look something like this:






Arc lines: Click the Define Lines icon , add line 1, set the Type to Arc, set P1 to 4, P2 to 9, Center to 8 and click Save. Then add line 2, set P1 to 9, P2 to 5, Center to 8 and click OK. The graphics window should look something like this:

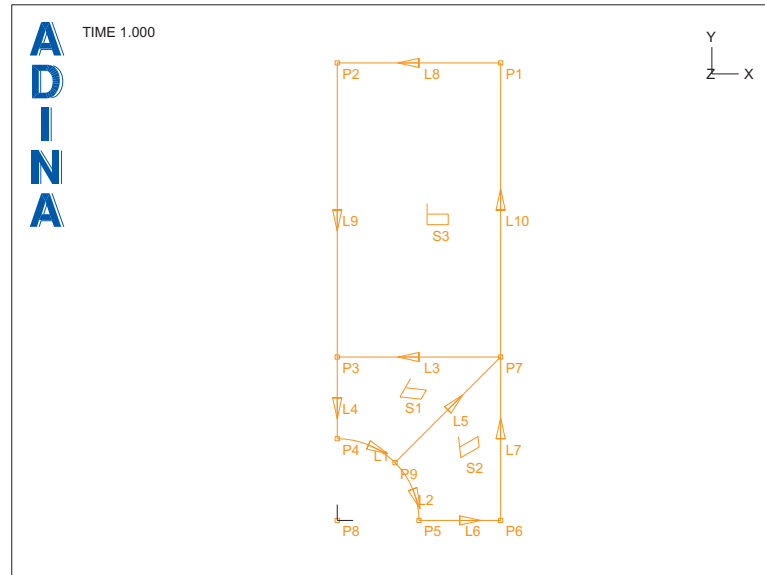


Surfaces: Click the Define Surfaces icon , add surface 1, make sure that the Type is set to Vertex, define the following surfaces and click OK. (Click Add after defining surfaces 1 and 2.)


Surface Number	Point 1	Point 2	Point 3	Point 4
1	7	3	4	9
2	7	9	5	6
3	1	2	3	7

To display the geometry point, line and surface numbers, click the Point Labels icon , Line/Edge Labels icon  and Surface/Face Labels icon . The graphics window should look something like the figure on the next page.


Problem 2: Plate with a hole in tension



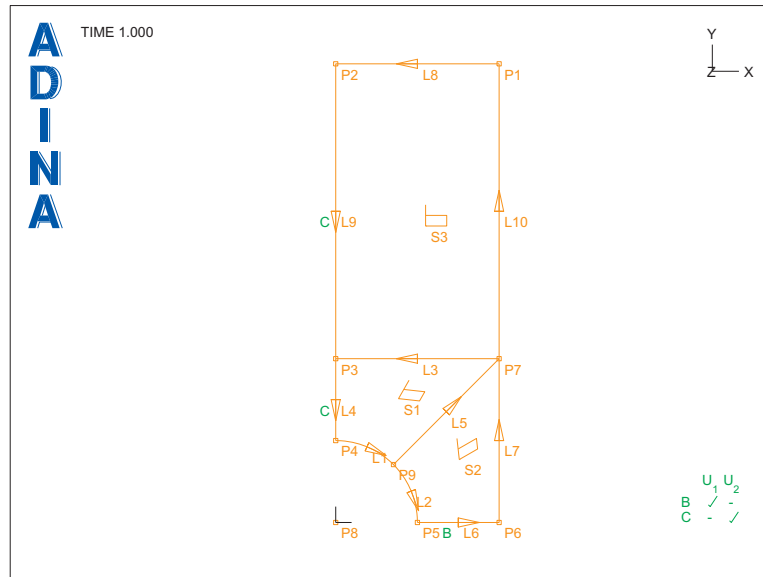
Defining and applying boundary conditions

We need two boundary conditions for modeling symmetry. Click the Apply Fixity icon  and click the Define... button. In the Define Fixity dialog box, add fixity name YT, check the Y-Translation button and click Save. Then add fixity name XT, check the X-Translation button and click OK.


In the Apply Fixity dialog box, set the “Apply to” field to Edge/Line. To set the fixity for lines 4 and 9 to XT, set the Fixity Name to XT and enter 4 and 9 in the first two rows of the table. To set the fixity for line 6 to YT, set the Fixity Name to YT and enter 6 in the first row of the table. Then click OK.


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

Notice that line 6 is marked with a B. The table at the lower right of the graphics window indicates that the U_1 (x) degree of freedom is free and that the U_2 (y) degree of freedom is fixed. Similarly, lines 4 and 9 are marked with a C, and the table indicates that the U_1 (x) degree of freedom is fixed and that the U_2 (y) degree of freedom is free.




Defining and applying loads

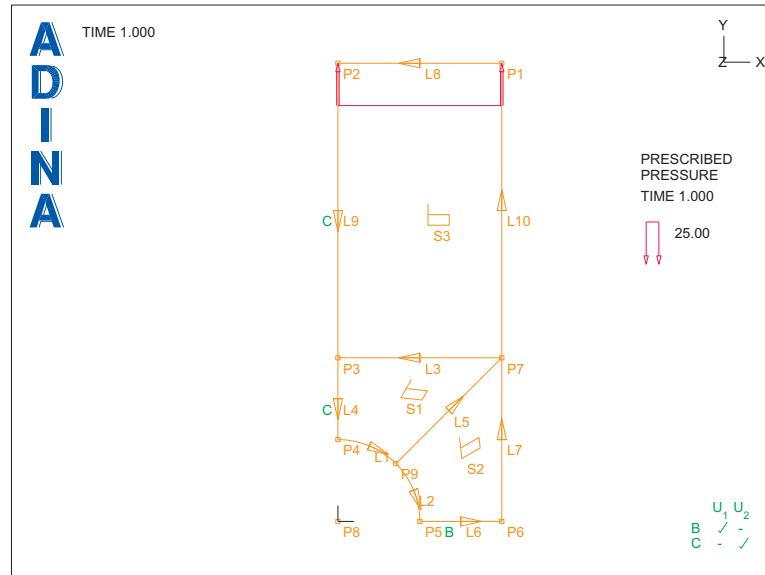
Click the Apply Load icon , set the Load Type to Pressure and click the Define... button to the right of the Load Number field. In the Define Pressure dialog box, add pressure 1, set the Magnitude to -25 and click OK. In the Apply Load dialog box, make sure that the “Apply to” field is set to Line and, in the first row of the table, set the Line # to 8. 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 7E4, the Poisson's ratio to 0.25 and click OK. Click Close to close the Manage Material Definitions dialog box.

Problem 2: Plate with a hole in tension




Defining the elements

Element group: Click the Element Groups icon , add group number 1, set the Type to 2-D Solid, set the Element Sub-Type to Plane Stress and click OK.

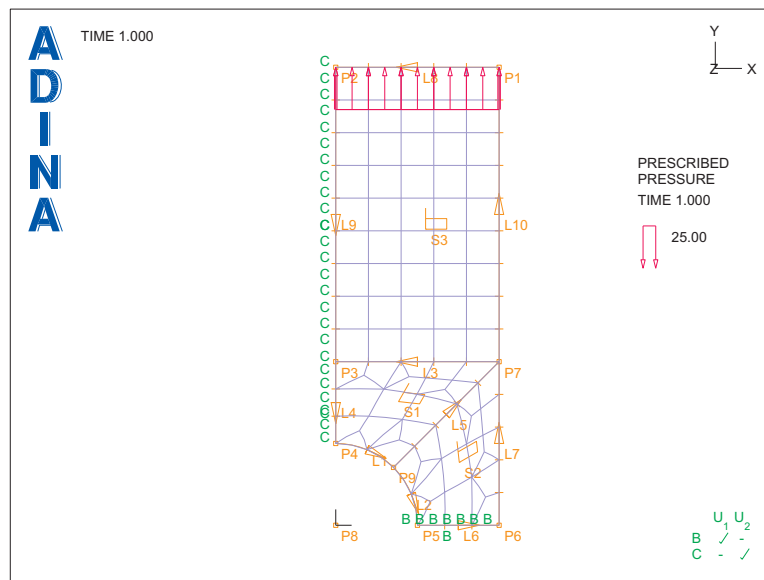
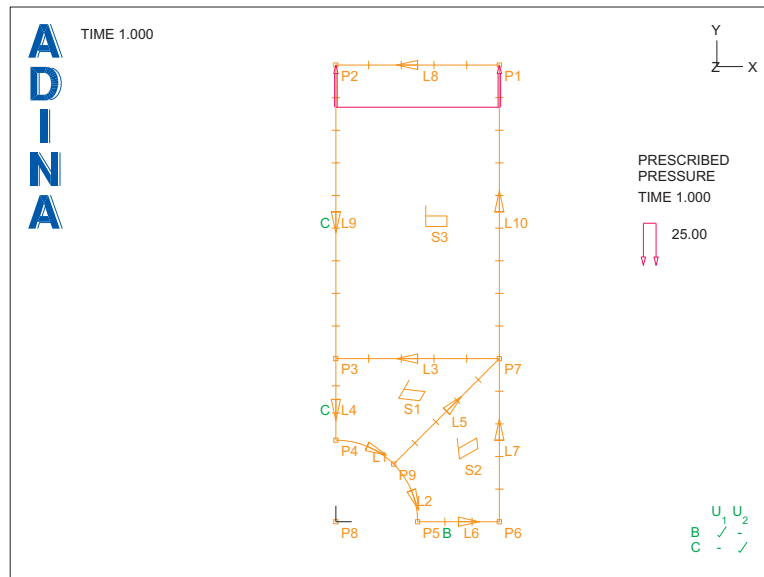
Subdivision data: In this mesh, we will assign a uniform point size to all points and have the AUI automatically compute the subdivisions. Choose Meshing→Mesh Density→Complete Model, make sure that the “Subdivision Mode” is set to “Use End-Point Sizes” and click OK. Now choose Meshing→Mesh Density→Point Size, set the “Points Defined from” field to “All Geometry Points”, set the Maximum to 2 and click OK.


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

Element generation: Click the Mesh Surfaces icon , enter 1, 2, 3 in the first three rows of the Surface # table and click OK. The graphics window should look something like the bottom figure on the next page.

Note, if the pressure magnitude is 0, return to the Control→Miscellaneous Options dialog box and make sure that the “2D Solid Elements in” field is set to “XY-Plane, Y-Axisymmetric”.

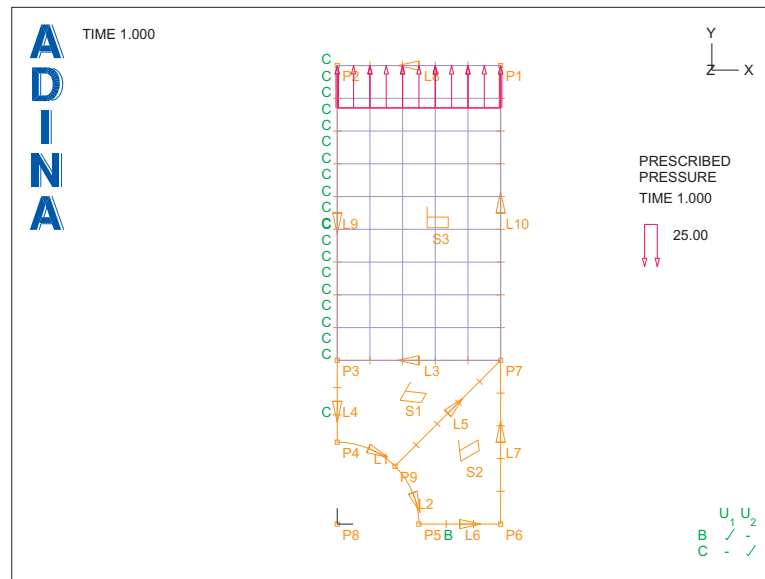
Problem 2: Plate with a hole in tension



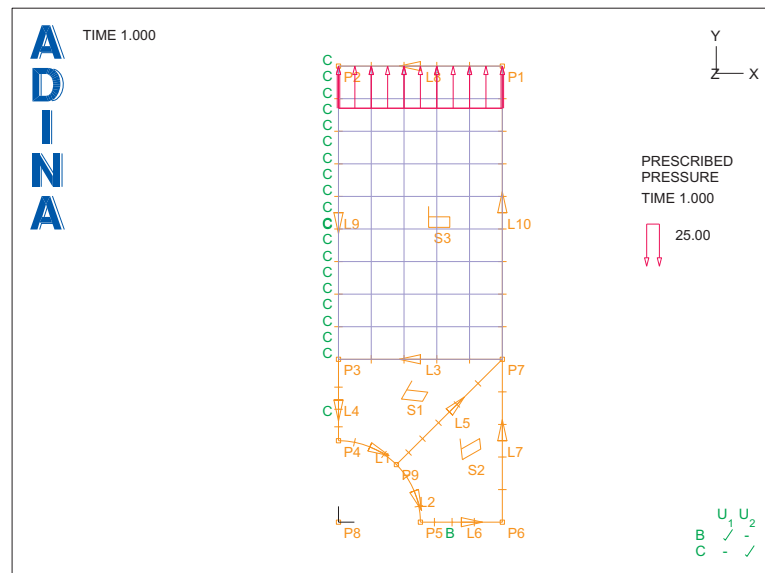
The mesh does not look very pleasing near the hole. We will remesh near the hole, adding more elements. Click the Delete Mesh/Elements icon , set the "Delete Mesh from" field to On Surfaces, enter 1, 2 in the Surface # table and click OK.


Problem 2: Plate with a hole in tension

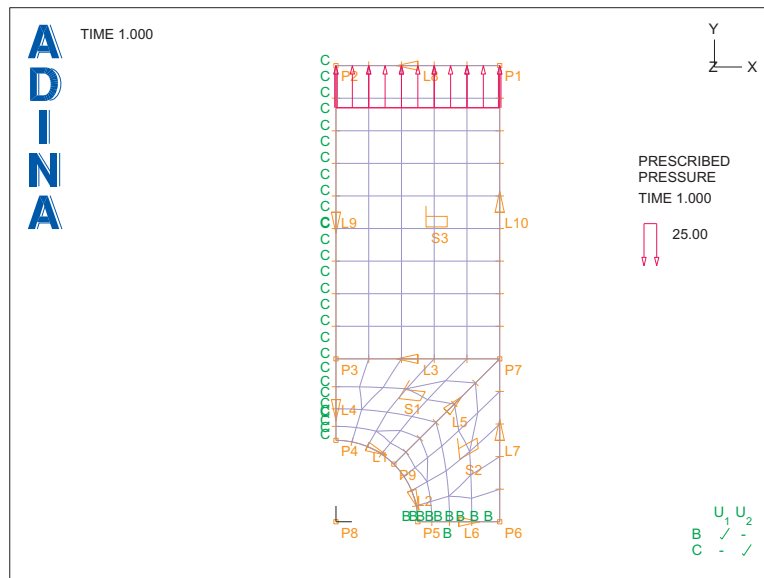
The graphics window should look something like this:







Now we decrease the point size of the points on the hole. Choose Meshing→Mesh Density→Point Size, and, in the table, enter 1.0 for the Mesh Size for points 4, 5, 9 and click OK. The graphics window should look something like this:



Now we generate the elements onto surfaces 1 and 2. Click the Mesh Surfaces icon , enter 1, 2 in the first two rows of the Surface # table and click OK. The graphics window should look something like this:



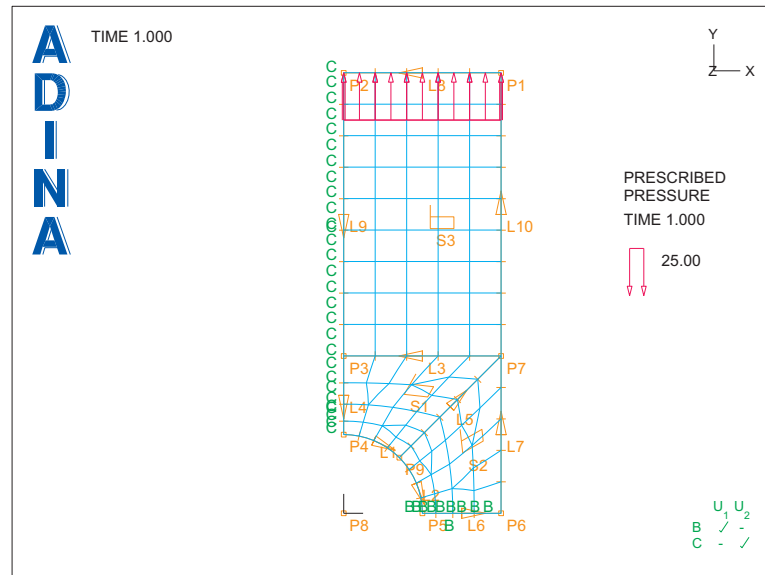
Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

First click the Save icon  and save the database to file prob02. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon , set the file name to prob02, 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 and discard all changes. Then click the Open icon , set the “File type” field to “ADINA-IN Database Files (*.idb)”, choose file prob02 and click Open. Then click the Open icon  and open porthole file prob02.



Please notice that we first opened the ADINA-IN database, then loaded the porthole file. We did this so that we can create a graph of the stresses along a geometry line later on.

The graphics window should look something like the figure on the next page. Notice that the geometry and the deformed mesh are displayed.




Problem 2: Plate with a hole in tension



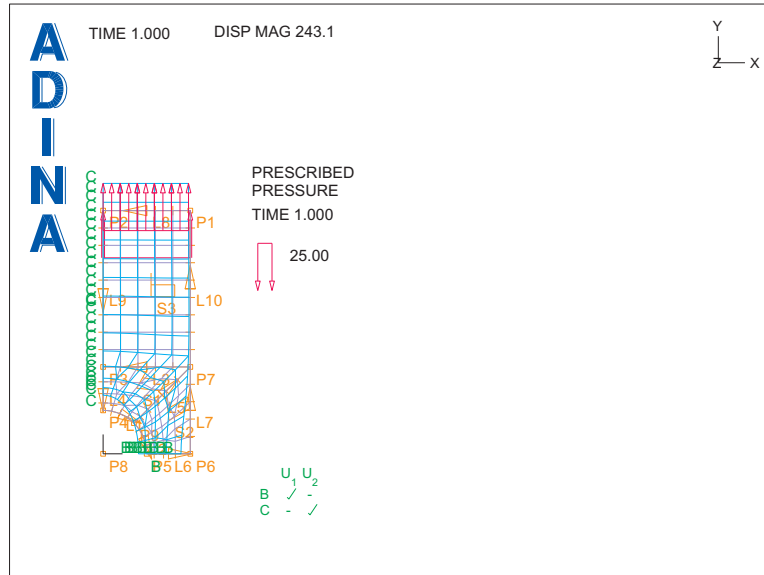
Examining the solution



Original and deformed meshes: Click the Show Original Mesh icon  and the Scale Displacements icon  10%.


We will plot this mesh along with other mesh plots showing the solution. To make room for the other mesh plots, we need to shrink this mesh using the mouse and move it to the left, so that we obtain the picture shown at the top of the next page.


Here are detailed instructions for resizing and moving the mesh plot. To shift the mesh plot, click the Pick icon  and the Dynamic Pan icon , then click on one of the lines of the mesh plot. A bounding box appears around the mesh plot; this bounding box indicates the mesh plot is highlighted. Now hold down the left mouse button and move the cursor. The mesh plot moves along with the mouse cursor. When the mesh plot is in the proper position, release the left mouse button. To resize the mesh plot, click the Dynamic Resize icon , hold down the left mouse button and move the mouse diagonally up and to the right to enlarge the mesh plot and move the mouse diagonally down and to the left to shrink the mesh plot. The mesh plot dynamically resizes as you move the mouse. When the mesh plot is the proper size, release the left mouse button. To unhighlight the mesh plot, move the cursor onto empty space within the graphics window and click the left mouse button.


Move and resize the boundary condition table and the load legend in the same way.




Stress band plot: To display another mesh plot, click the Mesh Plot icon . Use the mouse to move and resize the new mesh plot to the right of the first mesh plot. This time, to resize the mesh, make sure that the Dynamic Pan icon  is pressed, pick the mesh plot, then drag the mouse while holding down the Ctrl key.

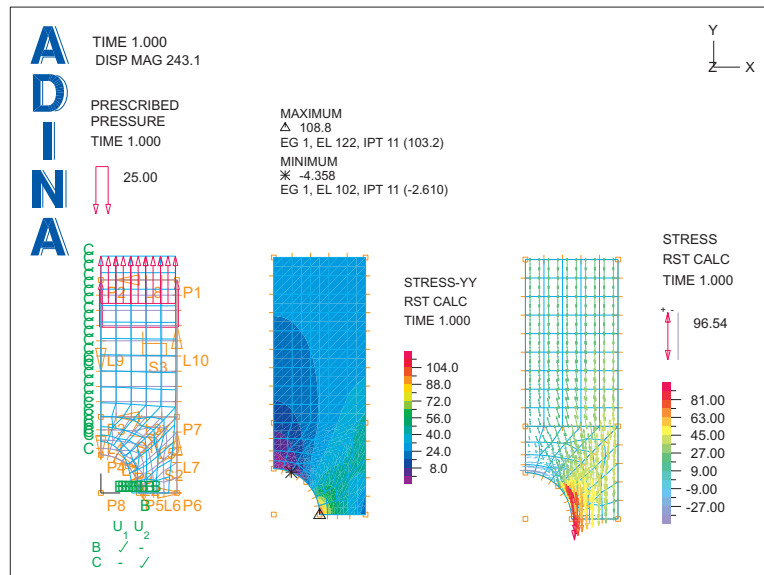
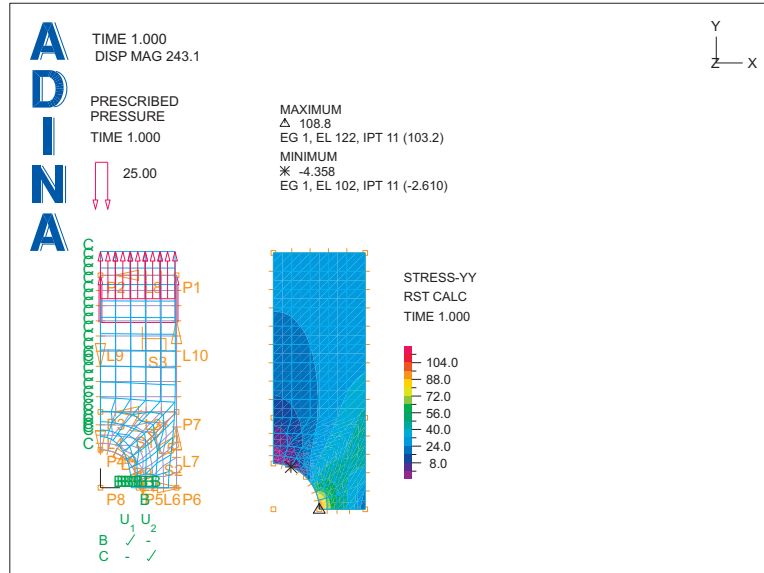
At this point, there are two sets of axes and two sets of “TIME 1.000” texts displayed. To remove unwanted text, make sure that the Pick icon  is pressed, then move the cursor onto the text and click the left mouse button. The text becomes highlighted. Then press the Del or Delete button on the keyboard to erase the text. Remove both sets of axes, both sets of “TIME 1.000” texts and the “DISP MAG” text in the same way.

Now click the Create Band Plot icon , set the Band Plot Variable to (Stress: STRESS-YY) and click OK. Move the band plot legends until the graphics window looks something like the top figure on the next page.



Stress vector plot: To display another mesh plot, click the Mesh Plot icon . Use the mouse to resize and move the new mesh plot to the right of the previous mesh plot. Remove the new axes and the new TIME 1.000 text.


Now click the Quick Vector Plot icon . Move the vector plot legend until the graphics window looks something like the bottom figure on the next page.

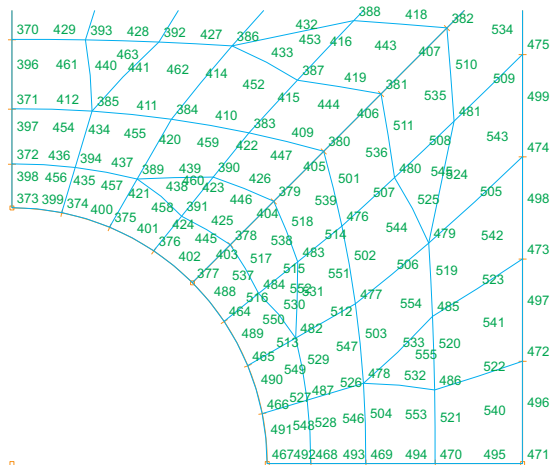
Problem 2: Plate with a hole in tension




Nodes and elements near the hole:

We will view the node numbers in the mesh near the hole. Click the Clear icon , then click the Node Labels icon .

Since there are so many nodes, we have to zoom in to see them. Click the Zoom icon , move the cursor to a point near the top of the hole, press and hold the left mouse button, drag the cursor down and to the right so that the rubber-band box encloses the mesh region near the hole and release the left mouse button. The graphics window should look something like this:



To learn the current coordinates of node 467, click the Query icon , move the cursor onto the label of node 467 and click the left mouse button. The AUI writes a message such as "Node 467, curr=(4.99614E+00, 0.00000E+00, 0.00000E+00)" into the message window and into the status bar at the bottom of the control window. For more information about node 467, right-click and choose More. The AUI writes more information into the message window, including the element that is attached to node 467. (To display the message window, choose View→Message Window.)

To learn the numbers of the elements near the hole, move the cursor onto one of the elements and click the left mouse button. The AUI writes a message such as "Element group 1, element 122, side -1" into the message window and into the status bar at the bottom of the control window. Repeatedly click the left mouse button to cycle through all of the graphics

Problem 2: Plate with a hole in tension


objects that can be queried at the cursor location. For more information about a graphics object, right-click and choose More.

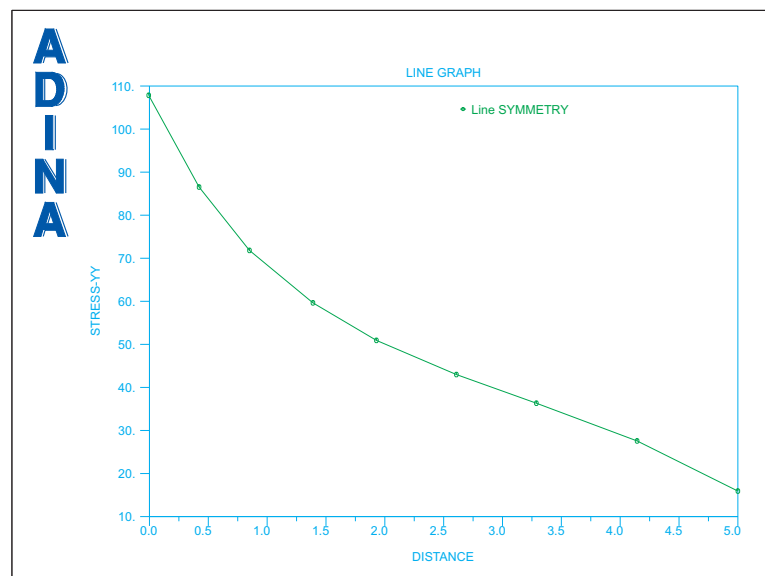
Now surround some of the graphics using a marquee box. The AUI highlights the selected graphics and writes messages for each graphics object (you may need to display the message window and use the vertical scrollbar in the message window to see all of the messages).

Graph of stresses on the horizontal symmetry line: Now we will graph the stresses on the horizontal symmetry line. To do this, we create a “node line”, which lists the nodes on the horizontal symmetry line.

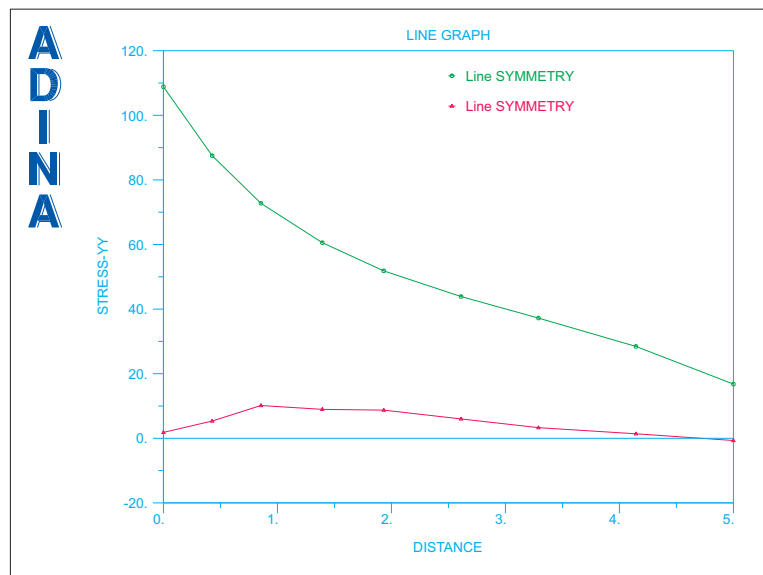
Choose Definitions→Model Line→General, add line SYMMETRY, enter the text “LINE 6” (you don't need to enter the quotes) into the first row and column of the table and click OK. The AUI writes the message “9 nodes in gline” to the message window and to the bottom of the AUI control window.

Please note, the above step is possible only if you open the ADINA-IN database before loading the porthole file. This is because the geometry information comes from the ADINA-IN database.

Now click the Clear icon  and choose Graph→Response Curve (Model Line). Verify that the Model Line Name is SYMMETRY, make sure that the X Variable is set to (Coordinate: DISTANCE), set the Y Variable to (Stress:STRESS-YY), set the Y Smoothing Technique to AVERAGED and click Apply. The graphics window should look something like this:




Let's add another stress component to the same graph. In the Display Response Curve (Model Line) dialog box, verify that the Line Name is SYMMETRY, make sure that the X Variable is set to (Coordinate:DISTANCE), set the Y Variable to (Stress:STRESS-XX) and make sure that the Y Smoothing Technique is set to AVERAGED. Also make sure that the Plot Name in the Graph Attributes box is set to PREVIOUS. Then click OK. The graphics window should look something like this:



Now we will customize the graph title, axes and curves. Choose Graph→Modify.

Title: Click the P button, move the cursor to the graph box and click to highlight it. Then click the ... button to the right of the Graph Depiction field. In the Title Attributes box, set the Type to Custom, enter “Stresses on horizontal symmetry line” in the Graph Title table (you don't need to enter the quotes) and click OK. Click Apply to see the updated title.

Use the Pick icon  and the mouse to center the title.

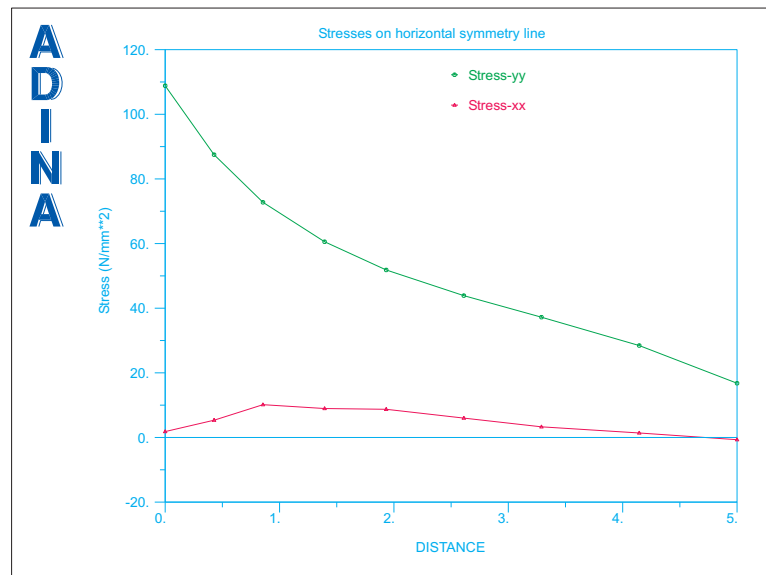
Axes: Set the Action to “Modify the Axis Depiction”. Click the P button, move the cursor to one of the numbers on the Y axis and click to highlight the Y axis. Then click the ... button to the right of the Axis Depiction field. In the Label Attributes box, set the Type to Custom, enter “Stress (N/mm**2)” in the Label table and click OK. Click Apply to see the updated axis.

Curves: Set the Action to “Modify the Curve Depiction”. Click the P button, move the cursor to the upper curve and click to highlight it. Then click the ... button to the right of the Curve

Problem 2: Plate with a hole in tension

Depiction field. Click the Legend tab, then, in the Legend Attributes box, set the Type to Custom, enter “Stress-yy” in the Legend table and click OK. Click Apply to see the updated curve and legend.

In a similar way, change the legend for the lower curve to “Stress-xx”. The graphics window should look something like this:



Numerical values: Choose Graph→List. The value of STRESS-YY at distance 0.0 should be 1.08832E+02 (N/mm²). Click Close to close the dialog box.

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