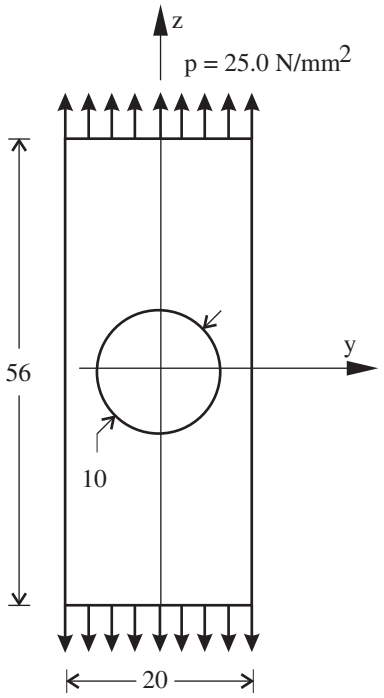
Problem description

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



All lengths in mm.
Thickness = 1 mm
$E = 7.0 \times 10^4$ N/mm^2
$\nu = 0.25$

This is the same problem as problem 2. However, in this problem we will demonstrate the body sheet feature of ADINA-M/PS (the ADINA Modeler, based on the Parasolid geometry kernel). Also, we will solve this problem in the y-z plane.

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

- Defining a combined line
- Defining a body of type sheet using ADINA-M/PS
- Meshing a body of type sheet
- Plotting stresses in a cylindrical coordinate system

We assume that you have worked through problems 1 and 2, 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.

Note that you must have an ADINA-M/PS license to do 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.

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 3: Plate with a hole in tension using ADINA-M/PS" and click OK.

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

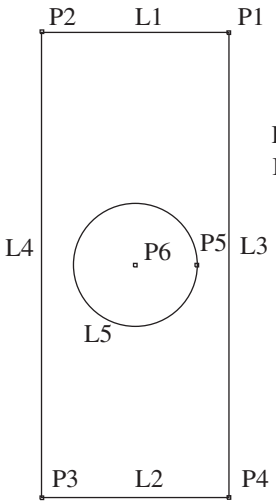
Defining model geometry

The diagram on the next page shows the key geometry used in defining this model. Notice that the lines are organized into loops. The external loop completely surrounds the model. The internal loop represents the hole in the model.

(Note: it is also possible to define the external loop by a single line of type polyline (with straight line segments). However we choose to use four separate lines, then combine them, in order to demonstrate the combined line feature.)

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

Point #	X2	X3
1	10	28
2	-10	28
3	-10	-28
4	10	-28
5	5	0
6	0	0



External loop: L6=L1 + L4 + L2 + L3
Internal loop: L5

Lines: Click the Define Lines icon and add the following lines:

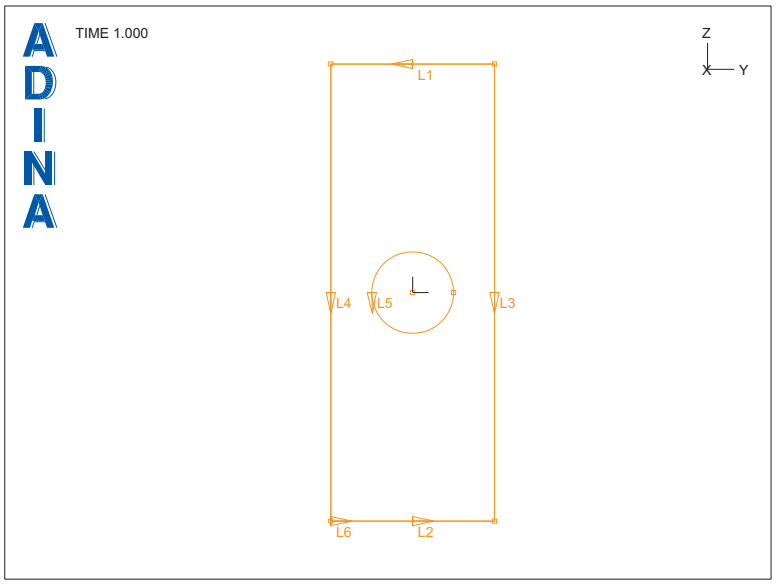
Line number	Type	Point 1	Point 2
1	Straight	1	2
2	Straight	3	4
3	Straight	1	4
4	Straight	2	3

Now add line 5, set the Type to Circle, make sure that "Defined by" is set to "Center, P1, P3", set Center to 6, P1 to 5, P3 to 2 and click OK.

Combined line: As indicated in the sketch above, we need to define an external loop as a single line. This line is constructed by combining lines 1 to 4.

Click the Define Lines icon, add line 6, set the Type to Combined, enter 1, 4, 2, 3 in the first four rows of the table (the order of the lines is important) and click OK.

When you click the Line/Edge Labels icon, the graphics window should look something like the figure on the next page.



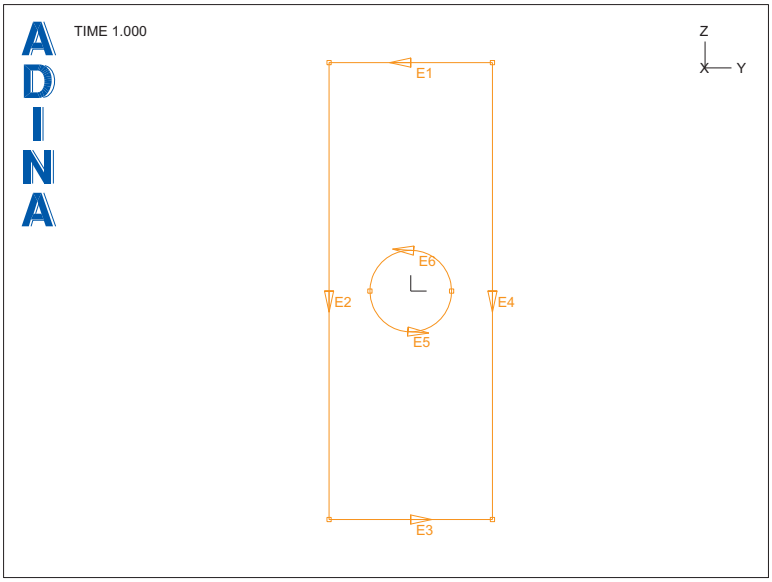
Sheet body: Now we construct a sheet body using line 6 as the external loop and line 5 as an internal loop. Click the Define Bodies icon , add body 1, set the Type to Sheet, set the External Loop Line # to 6, enter 5 in the first row of the table and click OK. The graphics window should look something like the top figure on the next page.

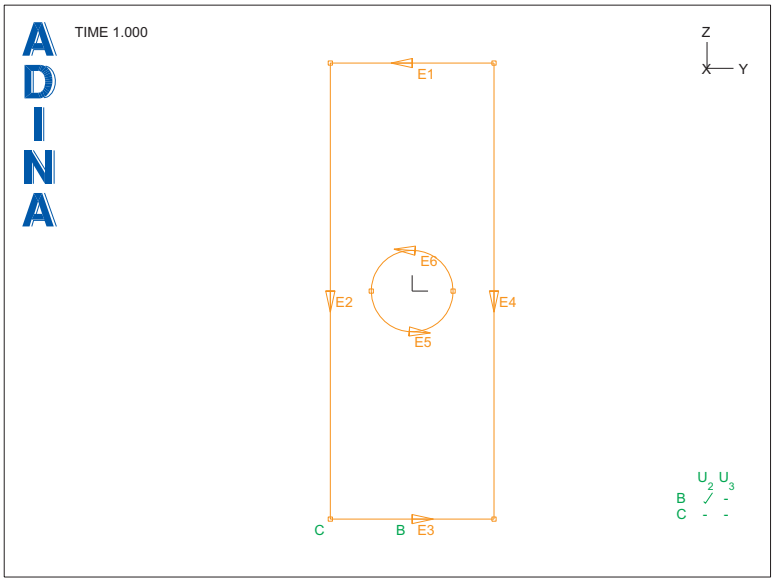
Notice that the lines have been replaced by edges.

Defining and applying boundary conditions

We will put rollers on the bottom edge of the model. Click the Apply Fixity icon and click the Define... button. In the Define Fixity dialog box, add fixity name ZT, check the Z-Translation button and click OK. In the Apply Fixity dialog box, set the "Fixity" to ZT and the "Apply to" field to Edge/Line. Enter 3, 1 in the first row of the table and click Apply.

We need to eliminate the rigid-body motion in the y direction. We do this by fixing point 3 (which is the lower left-hand point). In the Apply Fixity dialog box, set the "Fixity" to ALL and the "Apply to" field to Point, enter 3 in the first row of the table and click OK. When you click the Boundary Plot icon , the graphics window should look something like the bottom figure on the next page.

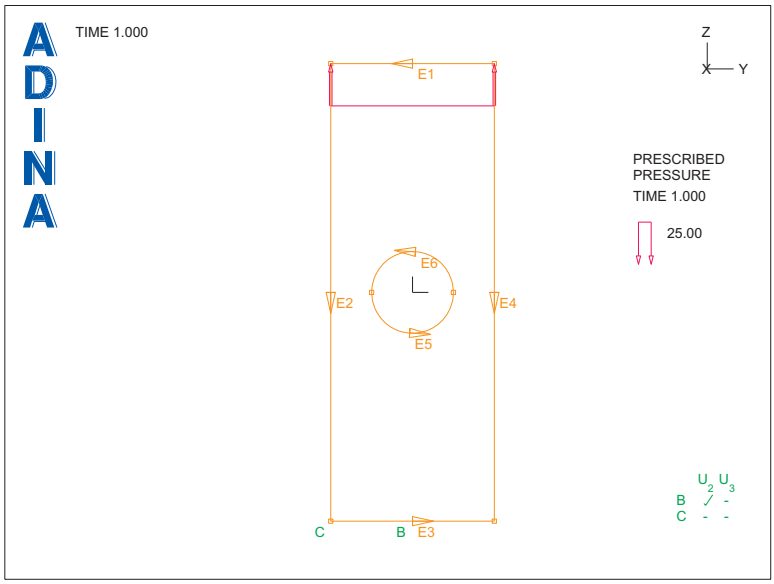

ADINA
TIME 1.000
Z
X
Y
E1
E6
E2
E4
E5
E3


ADINA
TIME 1.000
Z
X
Y
E1
E6
E2
E4
E5
C
B
E3
U2 U3
B
C

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, set the "Apply to" field to Edge and, in the first row of the table, set the Edge # to 1 and the Body # to 1. Click OK to close the Apply Load dialog box.

When you click the Load Plot icon , the graphics window should look something like this:



Defining the material

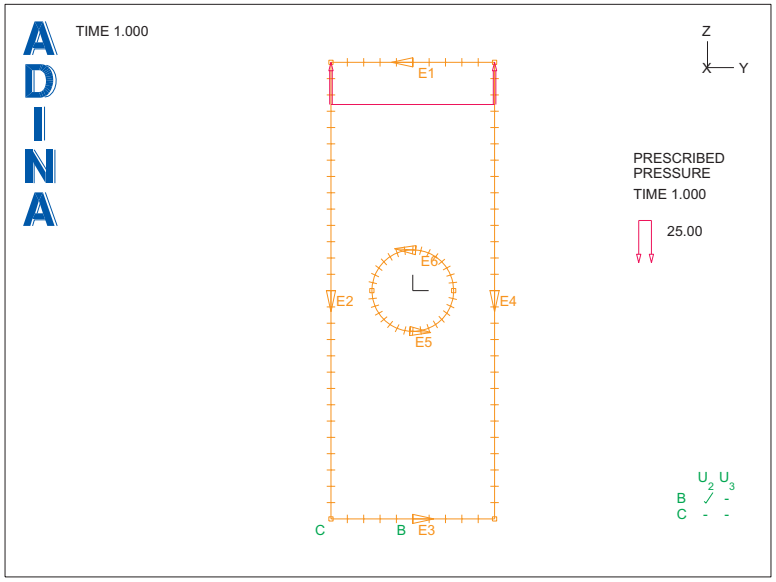
Click the Manage Materials icon and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 1, set the Young's Modulus to 7E4, the Poisson's ratio to 0.25 and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the elements

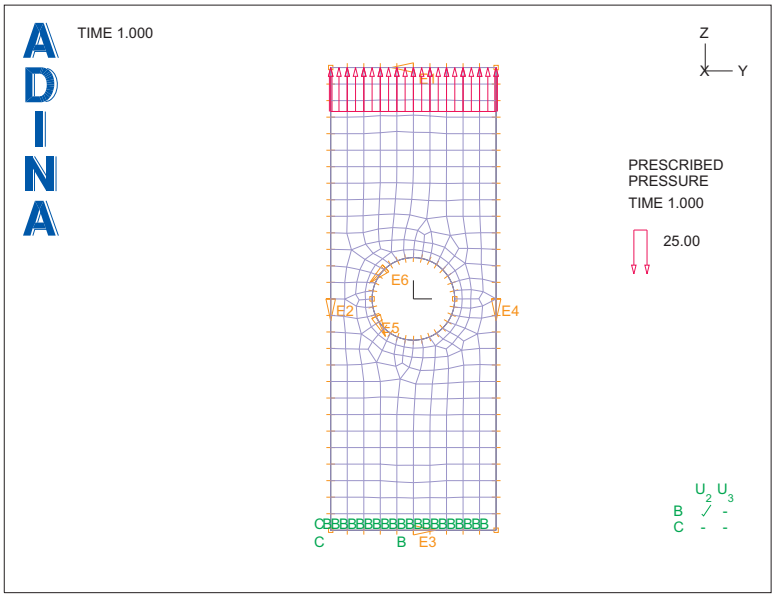
Element group: Click the Element Groups icon , add group number 1, set the Type to 2-D Solid, set the Element Sub-Type to Plane Stress and click OK.

Subdivision data: In this mesh, we will set the lengths of all element edges to be the same, then set the lengths of element edges on the hole (edges 5 and 6) to be smaller. Choose Meshing→Mesh Density→Complete Model, set the "Subdivision Mode" to Use Length, set the "Element Edge Length" to 2 and click OK. Now click the Subdivide Edges icon  , choose edge 5, set the "Element Edge Length" to 1, enter 6 in the first row of the table and click OK.

The graphics window should look something like this:



Element generation: Click the Mesh Faces 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. Your mesh may look slightly different than the mesh shown in the picture.

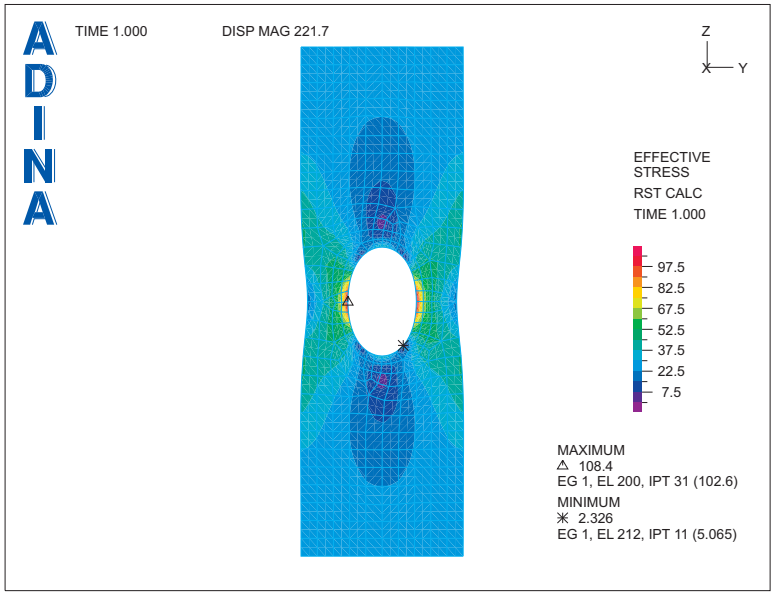


Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

First click the Save icon and save the database to file `prob03`. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon, set the file name to `prob03`, 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 and open porthole file `prob03`.

Examining the solution

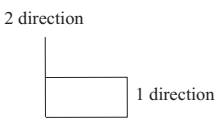
Click the Scale Displacements icon, then click the Quick Band Plot icon. The graphics window should look something like the figure on the next page. Your results will be slightly different if the mesh is different than the mesh shown in the picture.



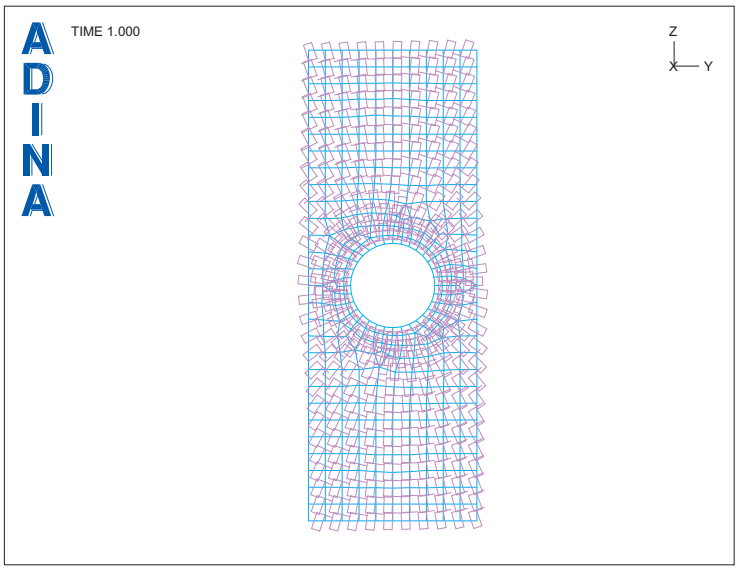
Plotting the stresses in a cylindrical coordinate system: Let's plot the tangential stress component in a cylindrical coordinate system. First click the Clear icon  to clear the mesh plot and band plot. Choose Definitions→Result Control, make sure that the Result Control Name is DEFAULT and click the … button to the right of the "Coordinate System for Transformed Results" field. In the Define Coordinate System dialog box, add system 1, set the Type to Cylindrical and click OK. In the Define Result Control Depiction dialog box, make sure that "Coord. Sys. for Transformed Results" is set to 1 and click OK.

To check the coordinate system directions, click the Mesh Plot icon  . Click the Modify Mesh Plot icon  . and click the Element Depiction… button. Check the Display Local System Triad button, set the Type to Result Transformation System, then click OK twice to close both dialog boxes. The graphics window should look something like the top figure on the next page.

Each of the element symbols shows the coordinate system directions, as follows:

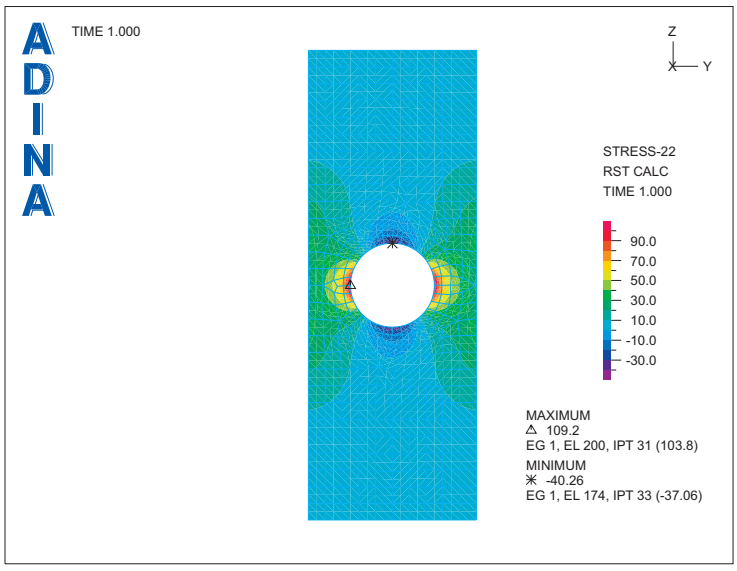


Evidently the 1 direction is the radial direction, the 2 direction is the tangential direction and the 3 direction is the axial direction.



Now click the Clear icon 

, then the Mesh Plot icon  , then the Create Band Plot icon  . Set the Band Plot Variable to (Stress:STRESS-22) and click OK. The graphics window should look something like this:



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