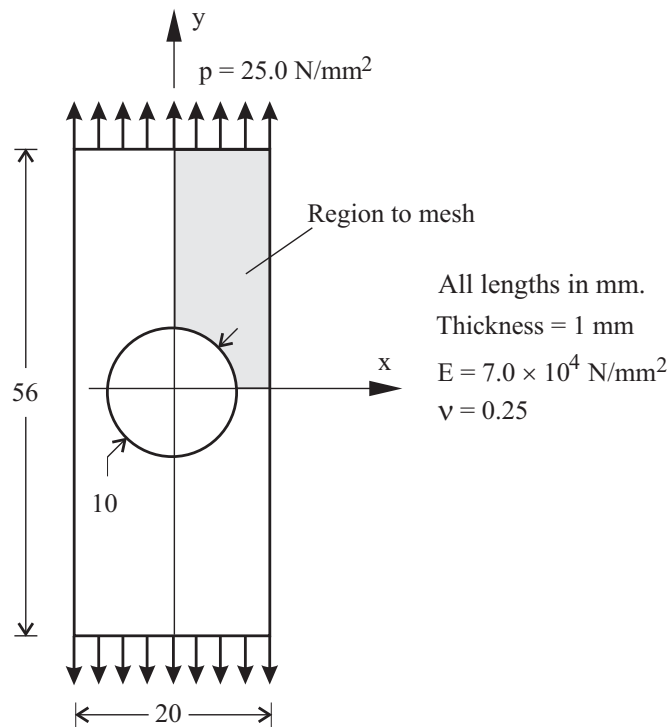


Problem description

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



This is the same model and loading as problem 2. We deliberately solve the problem using the relatively ineffective 3 and 4-node elements (without incompatible modes), so that the results are inaccurate when a coarse mesh is used. In this way we can demonstrate the mesh quality visualization features of the AUI.

In this problem solution, we will demonstrate the following topics:

- Plotting and listing error indicators
- Plotting repeating bands

We assume that you have worked through problems 1 to 12, or have equivalent experience with the ADINA System. Therefore we will not describe every user selection or button press.

Problem 13: Visualizing the mesh quality

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 the model



As the model is almost the same as problem 2, we only briefly give the steps needed to define the model.

Problem heading: Choose Control→Heading, enter the heading “Problem 13: Visualizing the mesh quality” 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.

Geometry: Click the Define Points icon , define the following points (remember to keep the X3 column blank) and 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


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. Then click the Define Points icon , add the following information to the table, and click OK.


Point #	X1	X2	X3
9	5	45	0


To define the 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.


To define the surfaces, click the Define Surfaces icon , make sure that the Type is set to Vertex, define the following surfaces and click OK.

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

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 “Fixity” to XT, the “Apply to” field to Edge/Line, enter 4 and 9 in the first two rows of the table and click Apply. Then set the “Fixity” to YT, enter 6 in the first row of the table and click OK.

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.




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.

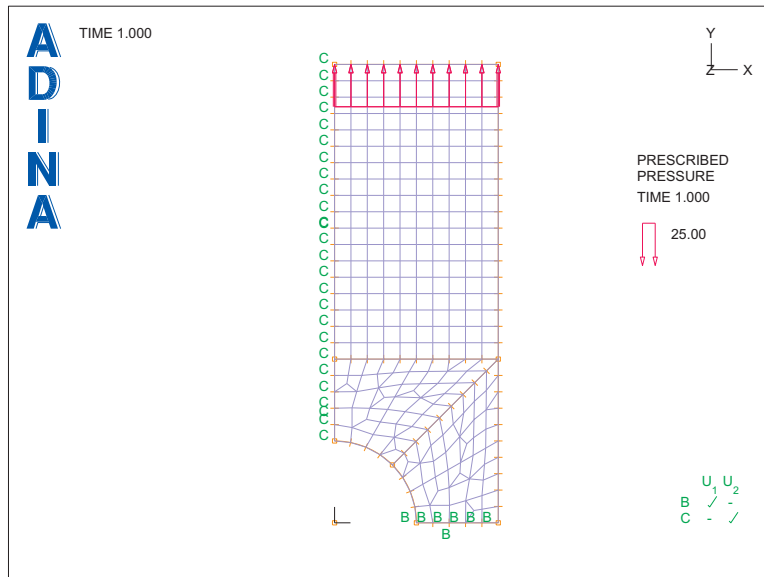
Element group: Click the Element Groups icon , add element group number 1, set the Type to 2-D Solid, set the Element Sub-Type to Plane Stress and click OK.

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, verify 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 1.0 and click OK.

Problem 13: Visualizing the mesh quality


Element generation: Click the Mesh Surfaces icon , set the “Nodes per Element” to 4, enter 1, 2, 3 in the first three rows of the table and click OK. When you click the Boundary Plot icon  and the Load Plot icon , the graphics window should look something like this:



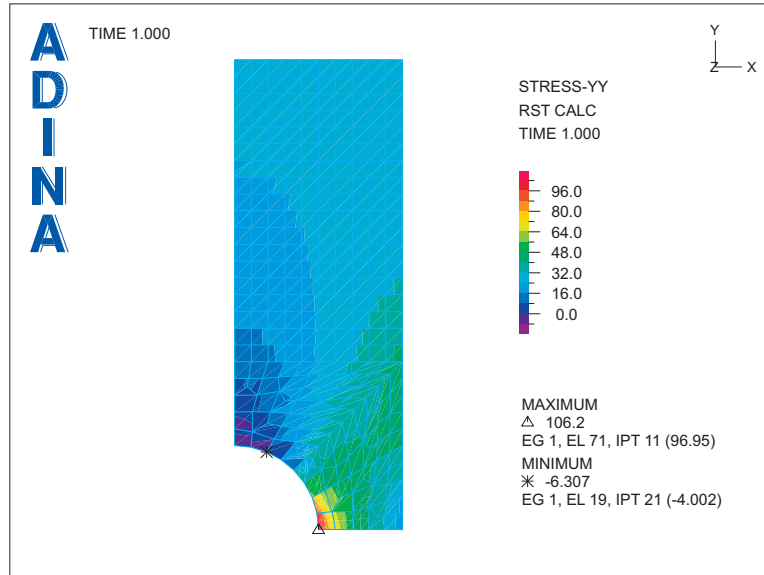
Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file


Click the Save icon  and save the database to file prob13. Click the Data File/Solution icon , set the file name to prob13, 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 prob13.

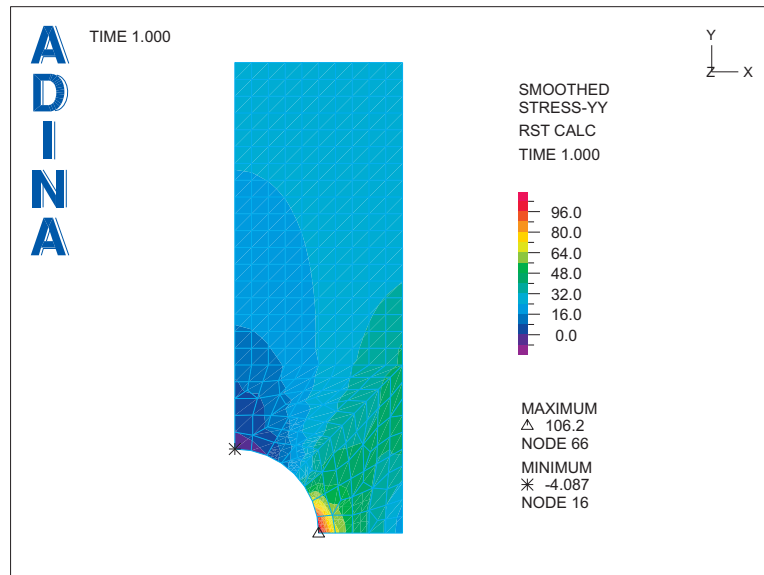
Examining the solution

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 the top figure on the next page.


Problem 13: Visualizing the mesh quality

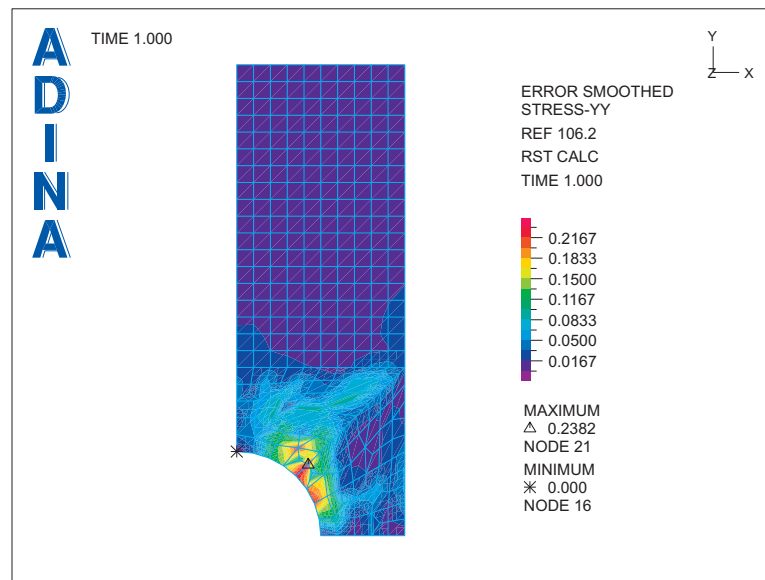


Note the jagged nature of the bands. To smooth the bands, click the Smooth Plots icon . The graphics window should look something like this:




Problem 13: Visualizing the mesh quality


Error indicators: The AUI allows you to plot error indicators as a guide for determining where the mesh should be refined. To plot error indicators, click the Error Plots icon . The graphics window should look something like this:

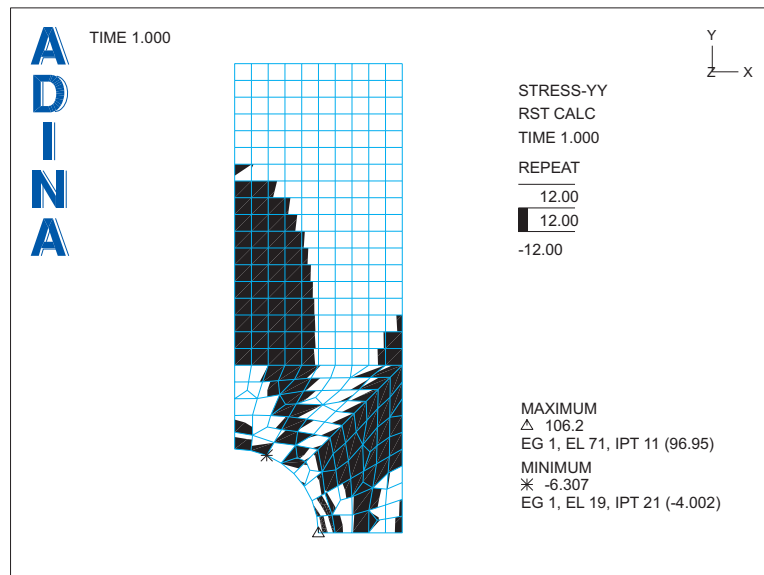
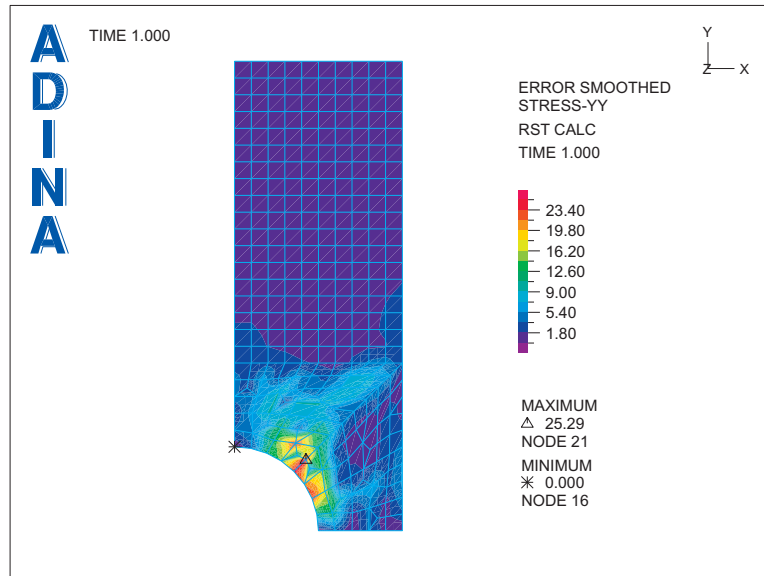


This plot shows that the maximum stress jump (difference between stresses evaluated at the same node) is about 24% of the maximum stress value.

You can, if desired, scale the error indicator so that the stress jump is not divided by a reference value. Click the Modify Band Plot icon , click the ... button next to the Smoothing Technique field, set the Error Reference Value to 1 and click OK twice to close both dialog boxes. The graphics window should look something like the top figure on the next page.

It is also possible to list the nodes for which the error indicator is highest. Choose List→Extreme Values→Zone, set the Smoothing Technique to BANDPLOT00001, Variable 1 to (Stress:STRESS-YY) and click Apply. The AUI lists the value of 2.52870E+01 for node 21. Click Close to close the dialog box.

Repeating bands: Another way to present the error is to plot repeating bands of unsmoothed stresses. Click the Modify Band Plot icon , set the Smoothing Technique to NONE, click the Band Table... button, set the Type to Repeating and click OK twice to close both dialog boxes. The graphics window should look something like the bottom figure on the next page.




The fact that the bands become indistinct near the hole shows that further mesh refinement is needed.


Problem 13: Visualizing the mesh quality

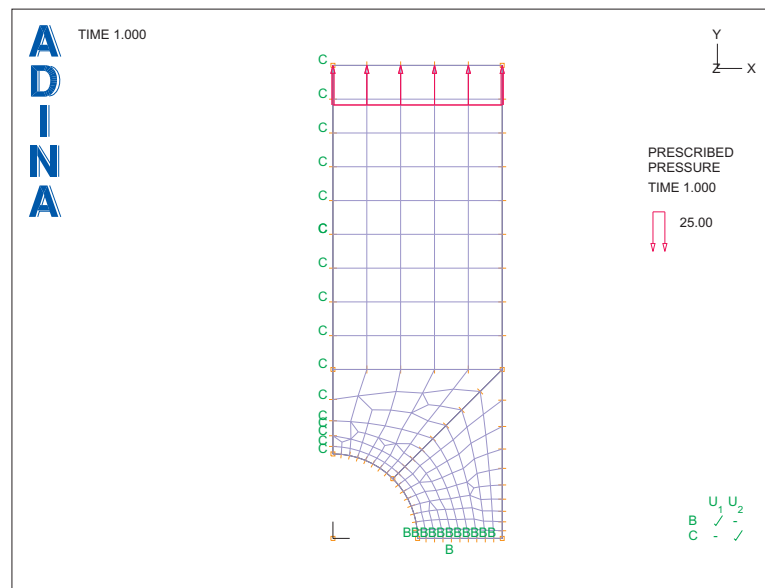
Refining the mesh

Preparing to modify the model: Set the Program Module drop-down list to ADINA Structures (you can discard all changes). Choose database file prob13 from the recent file list near the bottom of the File menu.

Deleting the elements: Click the Delete Mesh/Elements icon , set the “Delete Mesh from” field to Surface if necessary, enter 1, 2, 3 in the first three rows of the table and click OK.

Creating a refined mesh: In this mesh refinement, we would like to use fewer elements away from the hole and more elements closer to the hole. Choose Meshing→Mesh Density→Point Size, set the “Points Defined From” field to “Vertices of Specified Surfaces”, enter 1, 0.5; 2, 0.5; 3, 2.0 in the first three rows of the table, then click OK.

Now click the Mesh Surfaces icon , set the “Nodes per Element” to 4, enter 1, 2, 3 in the first three rows of the 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

Save the database, generate the ADINA Structures data file, run ADINA Structures, choose Post-Processing and load the porthole file in the same way as before, this time using name prob13a.

Examining the solution

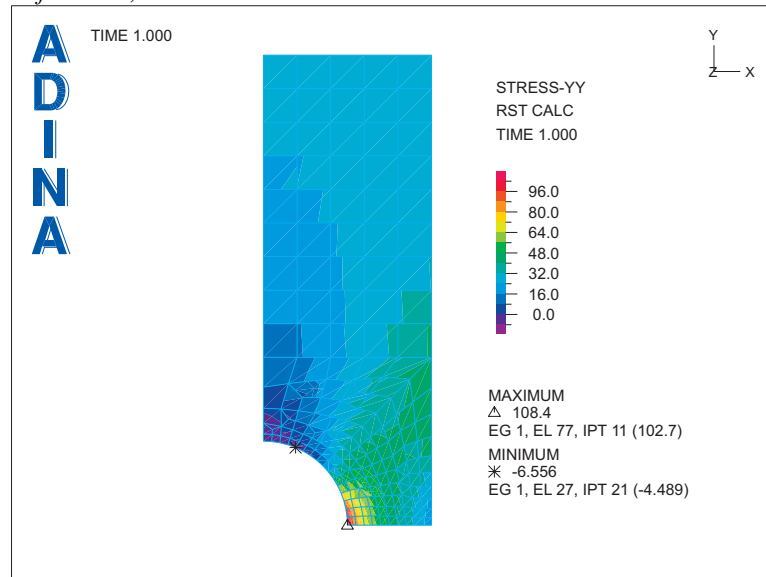
Follow the instructions given above to plot the stresses. We obtain the plots shown on pages 13-10 and 13-11.

The numerical value of the error indicator has dropped, showing that the solution has in fact improved. Also the repeating bands are more distinct near the hole.

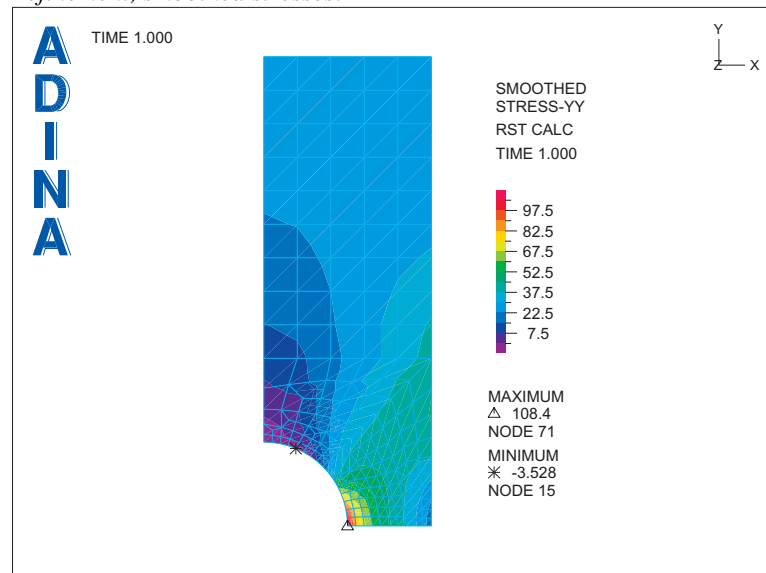
Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.

Problem 13: Visualizing the mesh quality

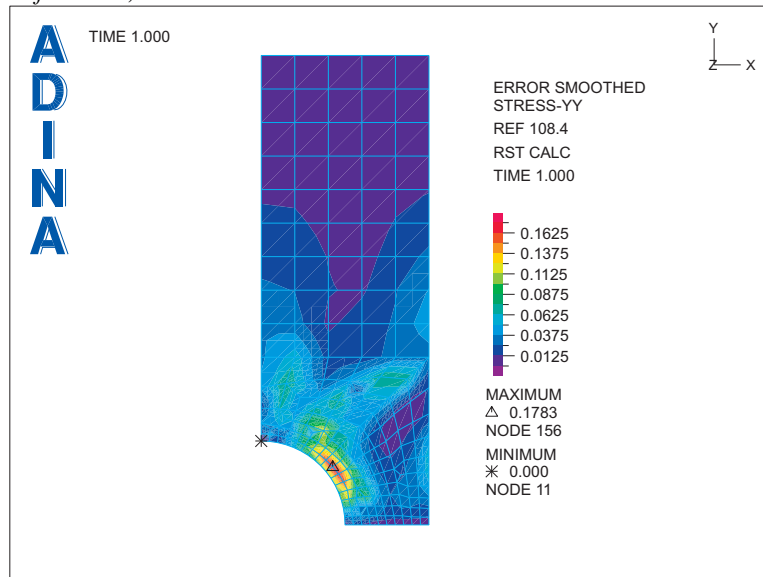
Refinement, unsmoothed stresses:



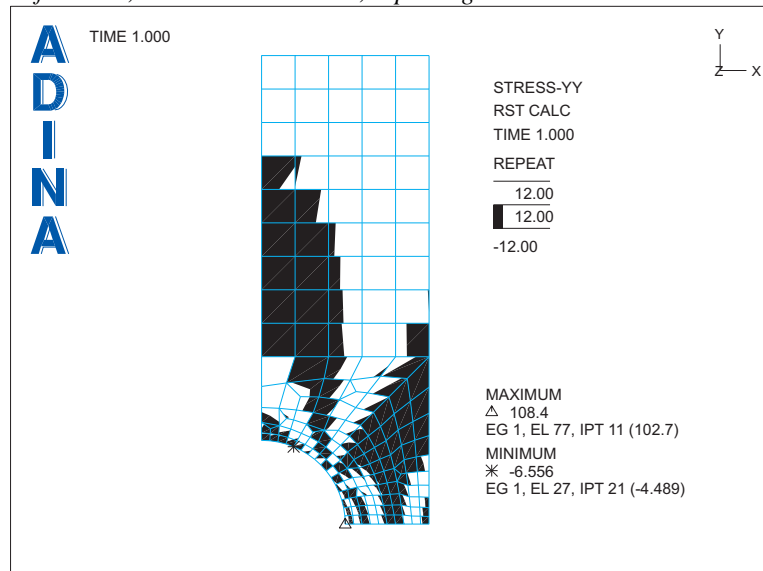
Refinement, smoothed stresses:



Refinement, error indicator:



Refinement, unsmoothed stresses, repeating bands:



Problem 13: Visualizing the mesh quality

This page intentionally left blank.