

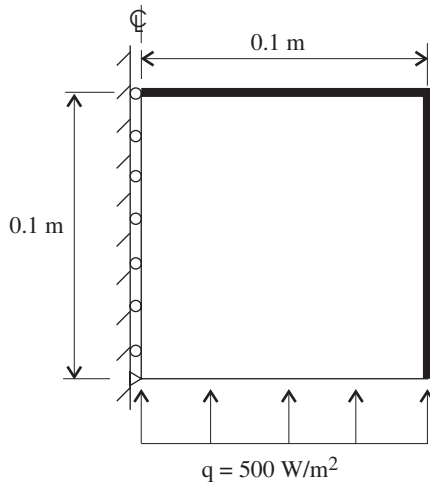
Problem 9

Thermal stress analysis of a cylinder

This page intentionally left blank.

Problem description

A cylinder is subjected to heat flux loadings as shown:



Thermal properties:

$$k = 0.5 \text{ W/m}\cdot\text{°C}$$

$$h = 5 \text{ W/m}^2\cdot\text{°C}$$

$$\epsilon = 0.2$$

$$\sigma = 5.669 \times 10^{-8} \text{ W/m}^2\cdot\text{°K}^4$$

Environmental temperature = 20 °C

Convection and radiation takes place on boundaries marked with a thick line

Structural properties:

$$E = 6.9 \times 10^{10} \text{ N/m}^2$$

$$\nu = 0.30$$

$$\alpha = 4.5 \times 10^{-6} \text{ m/m}$$

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

- Thermal analysis using ADINA-Thermal
- Definition of boundary convection and radiation elements
- Thermal stress analysis using the same mesh layout in ADINA-Thermal and ADINA
- Thermal stress analysis using a different mesh layout in ADINA-Thermal and ADINA

(Note, this problem can also be solved with the ADINA TMC feature of ADINA Structures, see primer problem 46.)

We assume that you have worked through problems 1 to 8, or have equivalent experience with the AUI.

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.

Thermal analysis

Invoking the AUI and choosing the finite element program

Invoke the AUI and choose ADINA Thermal from the Program Module drop-down list.

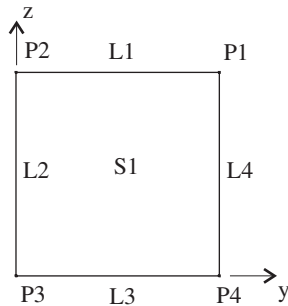
Defining model control data


Problem heading: Choose Control→Heading, enter the heading “Problem 9: Thermal stress analysis of a cylinder - thermal analysis” and click OK.

Mapping file option: Choose Control→Mapping, check the “Create Mapping File” button and click OK. The mapping file is required in thermal stress analysis when the structural mesh has a different layout than the thermal mesh.


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 (you can leave the X1 column blank) and click OK.

| Point # | X2 | X3 |
|---------|-----|-----|
| 1 | 0.1 | 0.1 |
| 2 | 0.0 | 0.1 |
| 3 | 0.0 | 0.0 |
| 4 | 0.1 | 0.0 |

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


| Surface Number | Type | Point 1 | Point 2 | Point 3 | Point 4 |
|----------------|--------|---------|---------|---------|---------|
| 1 | Vertex | 1 | 2 | 3 | 4 |

Defining and applying loads


Click the Apply Load icon , set the Load Type to Distributed Heat Flux and click the Define... button to the right of the Load Number field. In the Define Distributed Heat Flux dialog box, add heat flux number 1, set the Magnitude to 500 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 Site # to 3. Click OK to close the Apply Load dialog box.

Defining the environmental temperatures for convection and radiation


We will impose convection and radiation environmental temperatures onto lines 1 and 4 of the model.

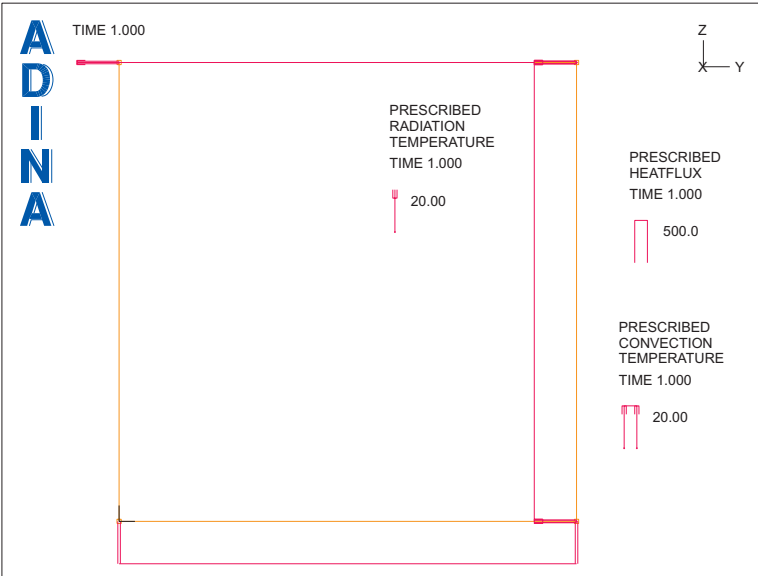
Convection: Click the Apply Load icon , set the Load Type to Convection and click the Define... button to the right of the Load Number field. In the Define Convection dialog box, add convection 1, set the Environment Temperature to 20 and click OK. In the Apply Load dialog box, set the “Apply to” field to Line and, in the first two rows of the table, set the Site # to 1 and 4 respectively. Click Apply in the Apply Load dialog box (do not close the dialog box yet).

Radiation: Set the Load Type to Radiation and click the Define... button to the right of the Load Number field. In the Define Radiation dialog box, add radiation 1, set the Environment Temperature to 20 and click OK. In the Apply Load dialog box, set the “Apply to” field to Line and, in the first two rows of the table, set the Site # to 1 and 4 respectively. 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 materials

Click the Manage Materials icon  and click the “k isotropic, c constant” button. In the Define Constant Isotropic Material dialog box, add material 1, set the Thermal Conductivity to 0.5 and click OK. (Do not close the Manage Material Definitions dialog box yet.)




Click the Convection Constant button. In the Define Constant Convection Material dialog box, add material 2, set the Convection Coefficient to 5 and click OK.


Click the Radiation Constant button. In the Define Constant Radiation Material dialog box, add material 3, set the Emissivity Coefficient to 0.2, set the Temperature Unit to Celsius, the Stefan-Boltzmann constant to $5.669\text{E-}8$ and click OK.


Click Close to close the Manage Material Definitions dialog box.

Defining the elements

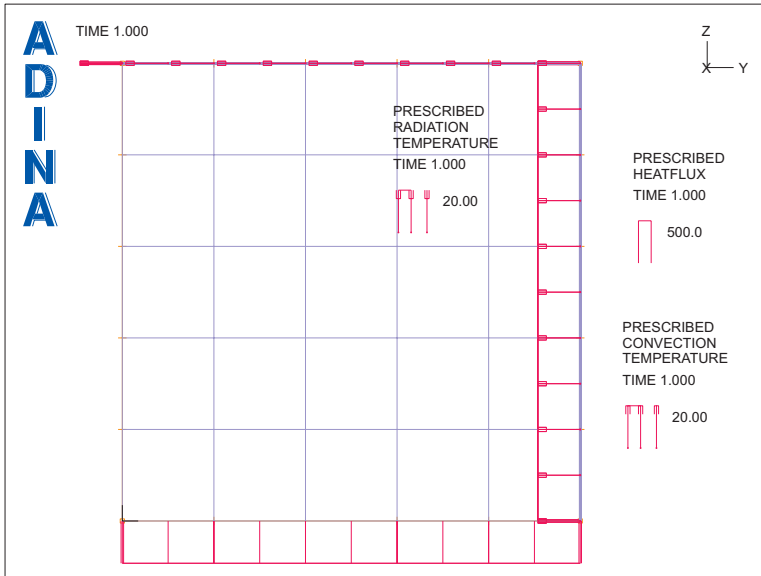
Element groups: Click the Define Element Groups icon , add group number 1, set the Type to 2-D Conduction, make sure that the Element Sub-Type is Axisymmetric and click Save. Then add group number 2, set the Type to Boundary Convection, set the Element Sub-Type to Axisymmetric, set the Default Material to 2 and click Save. Finally add group number 3, set the Type to Boundary Radiation, set the Element Sub-Type to Axisymmetric, set the Default Material to 3 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 0.02 and click OK.




Element generation: To generate the 2-D conduction elements, click the Mesh Surfaces icon , set the Type to 2-D Conduction, enter 1 in the Surface # table and click OK.

To generate the boundary convection elements, click the Mesh Lines icon , set the Type to Boundary Convection, enter 1 and 4 in the Line # table and click Apply.


To generate the boundary radiation elements, set the Type to Boundary Radiation, enter 1 and 4 in the Line # table and click OK. The graphics window should look something like this:



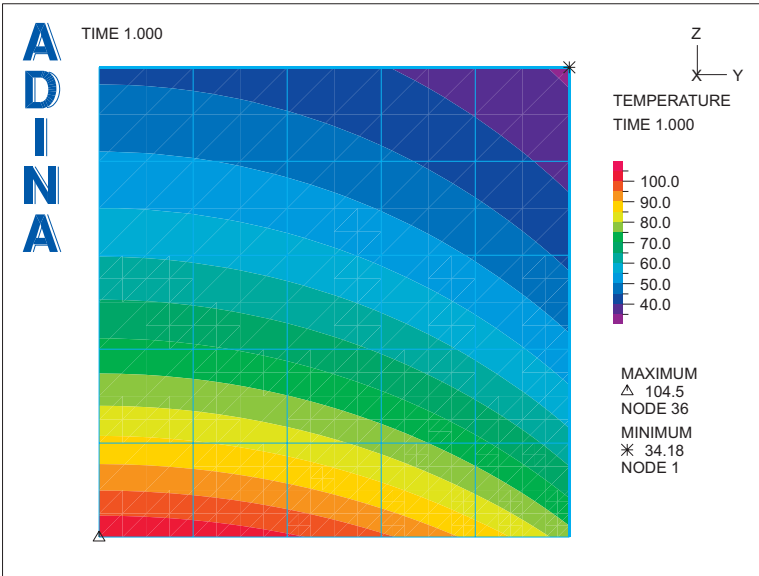
Generating the ADINA-Thermal data file, running ADINA-Thermal, loading the porthole file



First click the Save icon  and save the database to file prob09. To generate the ADINA-Thermal data file and run ADINA-Thermal, click the Data File/Solution icon , set the file name to prob09, make sure that the Run Solution button is checked and click Save. When ADINA-Thermal is finished, close all open dialog boxes. Then choose Post-Processing from the Program Module drop-down list (you can discard all changes), click the Open icon  and open porthole file prob09.

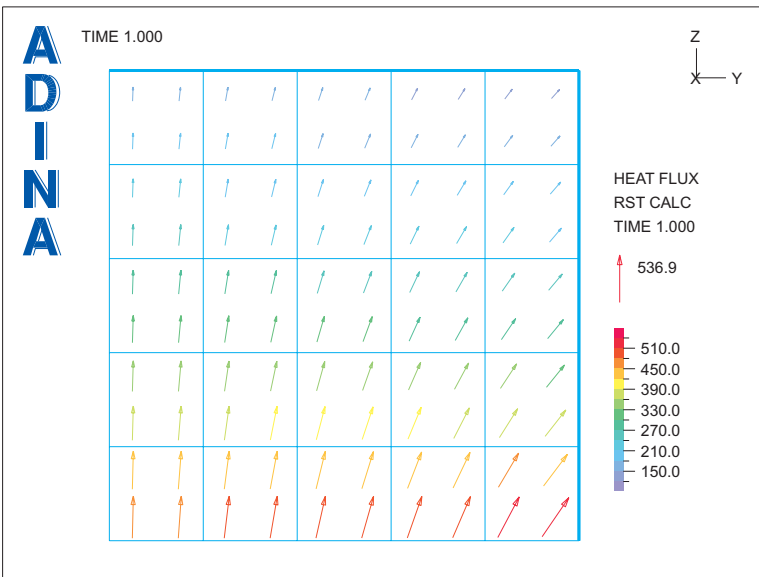
Examining the solution

Click the Quick Band Plot  icon to display the temperatures. The graphics window should look something like the top figure on the next page.

Problem 9: Thermal stress analysis of a cylinder



Now click the Clear icon  and click the Quick Vector Plot icon  to display the heat fluxes. The graphics window should look something like this:



Stress analysis using the thermal mesh layout


Now we will perform a thermal stress analysis of the cylinder. First we will use the same mesh layout as was used for the thermal analysis, then we will use a different mesh layout.

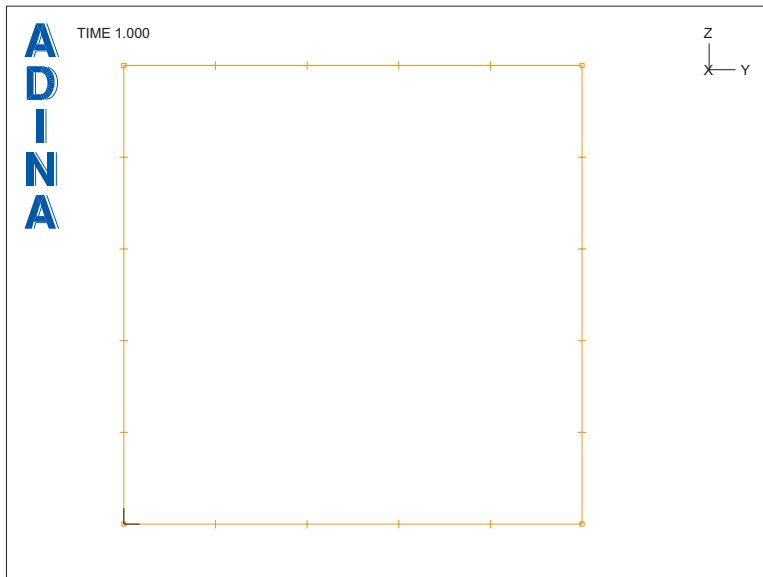
Choose ADINA Thermal from the Program Module drop-down list (you can discard all changes). Choose prob09.idb from the recent file list near the bottom of the File menu.

Deleting the ADINA-Thermal finite element model

Choose Meshing→Delete F.E. Model and click Yes to answer the prompt.

We also want to delete all of the load applications on the geometry. In the Model Tree, click on the + next to the Loading text, then, for each of the load applications, right-click on the text, choose Delete... and click Yes to answer the prompt.

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



The geometry is still available for model definition.

Defining model control data


Choosing ADINA as the finite element program: Choose ADINA Structures from the Program Module drop-down list.

Problem heading: Choose Control→Heading, enter the heading "Problem 9: Thermal stress analysis of a cylinder - stress analysis" 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.


Temperature input: Choose Control→Miscellaneous File I/O, set the "Temperatures" field to "Data Read from File" and click OK.

Defining and applying boundary conditions


We need a boundary condition corresponding to rollers on the left line of the square. Click the Apply Fixity icon  and click the Define... button. In the Fixity dialog box, add fixity name YT, check the Y-Translation button and click OK.

In the Apply Fixity dialog box, set the "Apply to" field to Lines, enter line 2, fixity YT in the first row of the table and click Save.


We also need to fix one point in the model. In the Apply Fixity dialog box, set the "Apply to" field to Points, 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 top figure on the next page.

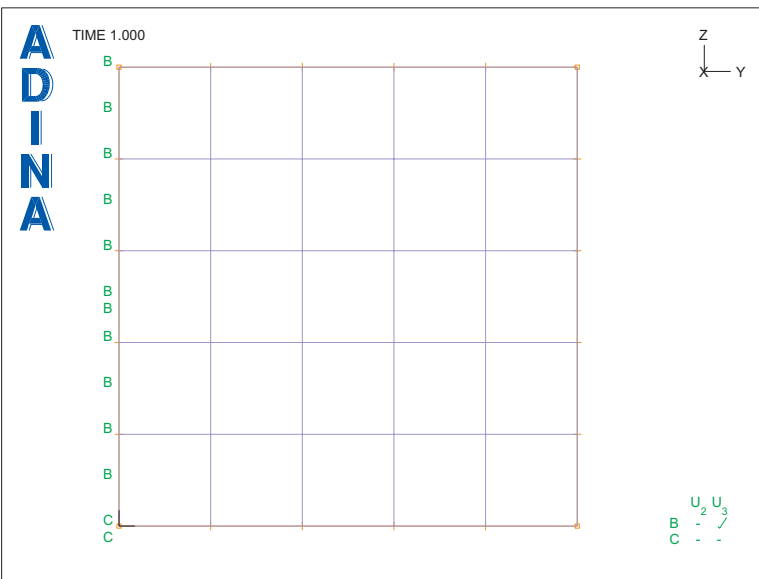
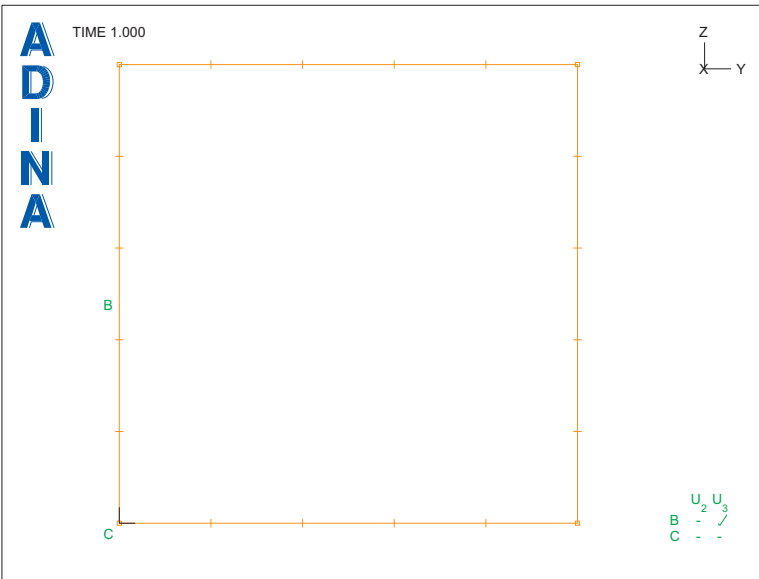
Defining the material

Click the Manage Materials icon  and click the "Elastic Isotropic" button. In the Define Isotropic Thermo-Elastic Material dialog box, add material 1, set the Young's Modulus to 6.9E10, the Poisson's ratio to 0.3, the Coef of Thermal Expansion to 4.5E-6, then click OK. Click Close to close the Manage Material Definitions dialog box.


Defining the elements

Element group: Click the Define Element Groups icon , add group number 1, set the Type to 2-D Solid, make sure that the Element Sub-Type is Axisymmetric and click OK.


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




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



First choose File→Save As and save the database to file prob09a. To generate the ADINA data file and run ADINA, click the Data File/Solution icon , set the file name to prob09a, make sure that the Run Solution button is checked and click Save.


The AUI will prompt you to specify the temperature file. Change the current directory or folder to the directory or folder that you used to run the ADINA-Thermal model, choose temperature file prob09 and click Copy.

When the ADINA job is completed, close all open dialog boxes, choose Post-Processing from the Program Module drop-down list (you can discard all changes), click the Open icon  and open porthole file prob09a.

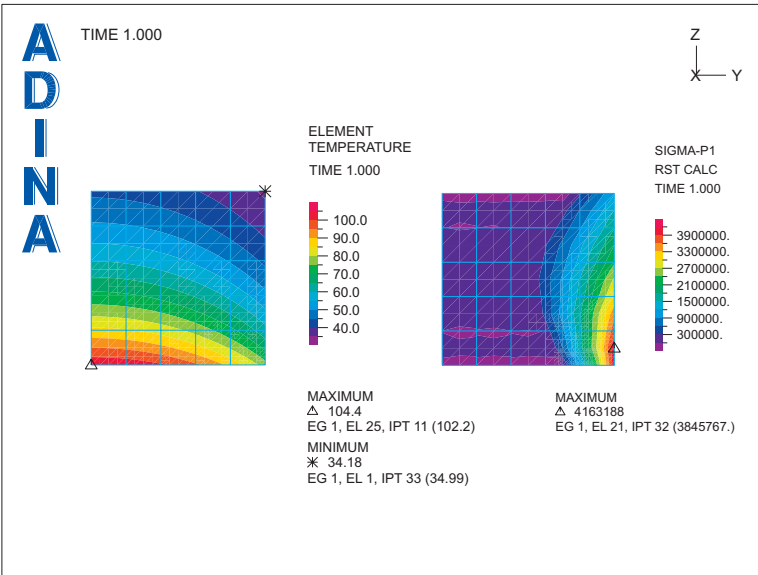
Examining the solution

Click the Create Band Plot icon , choose variable (Temperature: ELEMENT_TEMPERATURE) and click OK to display the temperature field. This is, of course, the same temperature field calculated by ADINA-Thermal. Use the mouse to resize this mesh plot into the left-hand part of the graphics window.

Now click the Mesh Plot icon  to display another mesh plot, then click the Create Band Plot icon , choose variable (Stress:SIGMA-P1) and click OK to display the maximum principal stress.

To suppress the display of the minimum value of the maximum principal stress (which, by the way, is not the minimum value of the minimum principal stress), click the Modify Band Plot icon , select band plot BANDPLOT00002, click the Band Rendering... button, set the “Extreme Values” field to “Plot the Maximum” and click OK twice to close both dialog boxes.

Remove extra text and axes and resize the mesh plots until the graphics window looks something like the figure on the next page.



Stress analysis using a finer mesh layout


Let's use a finer structural finite element mesh to solve the same problem. Choose ADINA Structures from the Program Module drop-down list (you can discard all changes). Then choose prob09a.idb from the recent file list near the bottom of the File menu.

Deleting the ADINA finite element model

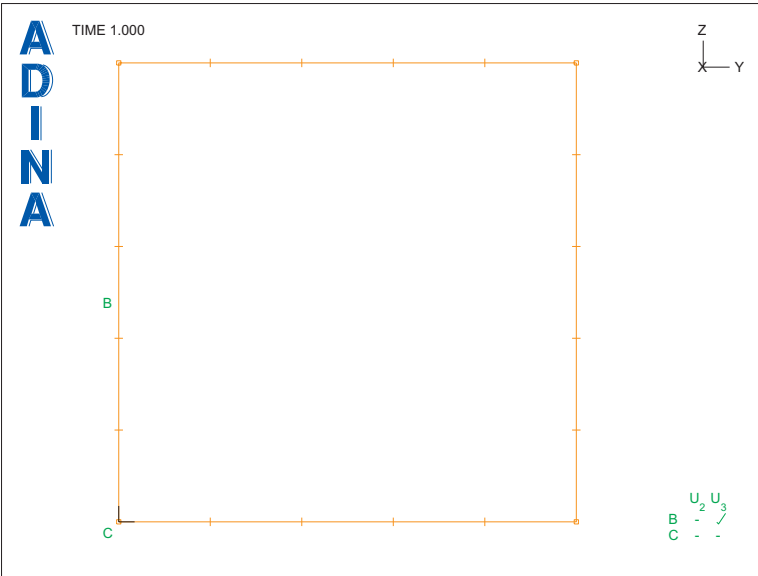
To remove the finite element model, choose Meshing→Delete F.E. Model and click Yes to answer the prompt. The graphics window should look something like the figure on the next page.

Notice that the boundary conditions defined on the geometry are still present. The material definition is also still present.


Defining the elements

Element group: Click the Define Element Groups icon , add group number 1, set the Type to 2-D Solid, make sure that the Element Sub-Type is Axisymmetric and click OK.

Subdivision data: Now we redefine the subdivision data, placing more elements at the lower right-hand corner of the square. We will now control the subdivision data using end-point sizes. Choose Meshing→Mesh Density→Complete Model, make sure that the “Subdivision Mode” is “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 0.02 and click Apply. Then enter 0.01 for the mesh size at point 4 and click OK.

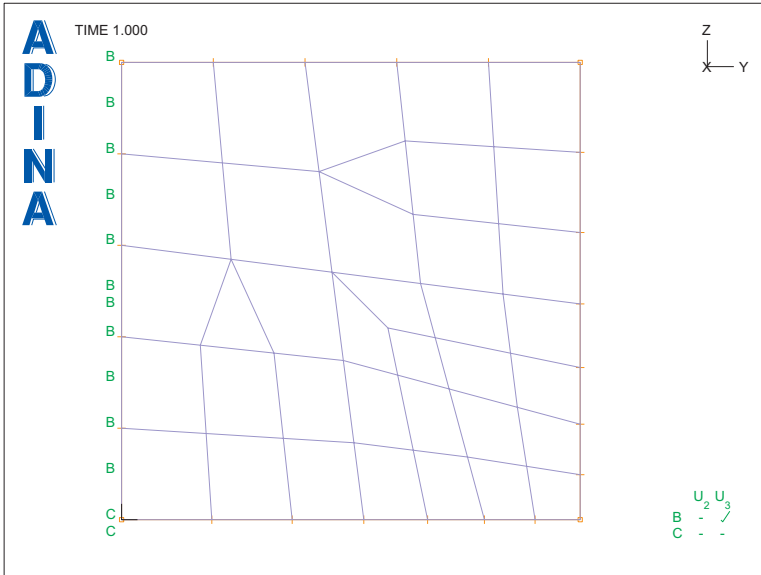
Element generation: Click the Mesh Surfaces icon , enter 1 in the first row of the table and click OK. The graphics window should look something like the figure on the next page.

Specifying the mapping file

Because the structural mesh has a different layout than the thermal mesh, we have to use the mapping file previously generated by ADINA-Thermal. Choose File→Thermal Mapping→Define, select file prob09.map and click Open.

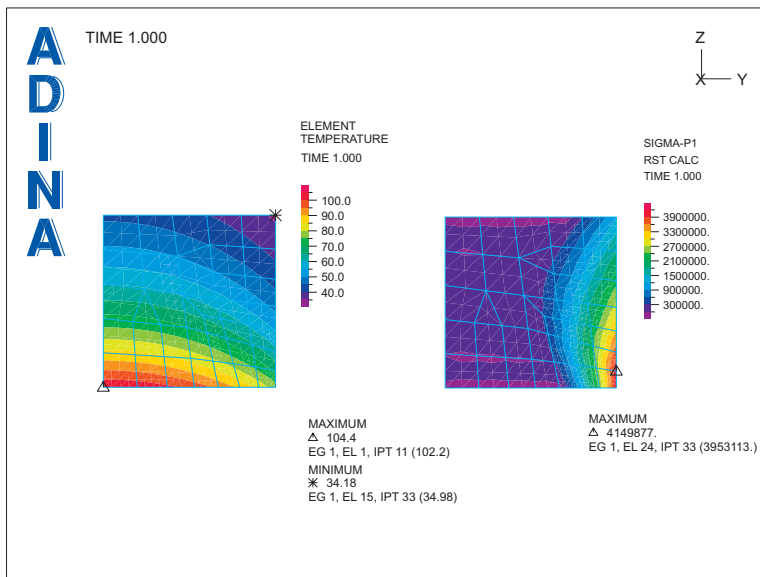
Generating the ADINA data file, running ADINA, loading the porthole file

First choose File→Save As and save the database to file prob09b. To generate the ADINA data file and run ADINA, click the Data File/Solution icon , set the file name to prob09b, make sure that the Run Solution button is checked and click Save. When the ADINA job is completed, close all open dialog boxes, choose Post-Processing from the Program Module drop-down list (you can discard all changes), click the Open icon  and open porthole file prob09b.



Examining the solution

Follow exactly the same steps as for the previous structural mesh to examine the solution. The graphics window should look something like this:



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

This page intentionally left blank.