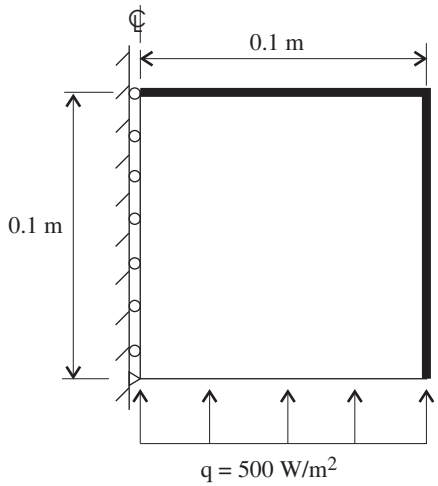
Problem description

A cylinder is subjected to heat flux loadings as shown:



Thermal properties:
- $k = 0.5$ W/m-°C
- $h = 5$ W/m^2-°C
- $\varepsilon = 0.2$
- $\sigma = 5.669 \times 10^{-8}$ W/m^2-°K^4
- Environmental temperature = 20 °C
- Convection and radiation takes place on boundaries marked with a thick line

Structural properties:
- $E = 6.9 \times 10^{10}$ N/m^2
- $\nu = 0.30$
- $\alpha = 4.5 \times 10^{-6}$ m/m

This is the same problem as was considered in primer problem 9. However, in this analysis, we will analyze the cylinder entirely within ADINA Structures, using the TMC model feature. Also the model will be solved in the x-y plane.

(Note, do not confuse the TMC model feature within ADINA Structures with the ADINA-TMC solution capability.)

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

- Using the TMC model feature within ADINA Structures

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 46: Thermal stress analysis of a cylinder – ADINA TMC model" and click OK.

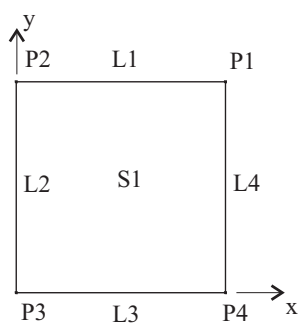
Plane for 2D elements: Choose Control→Miscellaneous Options, set the "2D Solid Elements in" field to "XY-Plane, Y-Axisymmetric" and click OK.

Heat transfer solution: Click the Coupling Options icon , set the "Type of Solution" to "TMC One-Way Coupling" and click OK.

Saving of heat fluxes: We would like to save the element heat fluxes. Choose Control→ Porthole→Select Element Results, add Result Selection Number 1, set Thermal to All 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 (you can leave the X3 column blank) and click OK.

Point #	X1	X2
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 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 XT, check the X-Translation button and click OK.

In the Apply Fixity dialog box, set the "Fixity" to XT, set the "Apply to" field to Edge/Line, enter line 2 in the first row and column of the table and click Apply.

We also need to fix one point in the model. In the Apply Fixity dialog box, set the "Fixity" to ALL, set the "Apply to" field to Point, enter 3 in the first row of the table and click OK.

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 Line # to 3. Click OK to close the Apply Load dialog box.

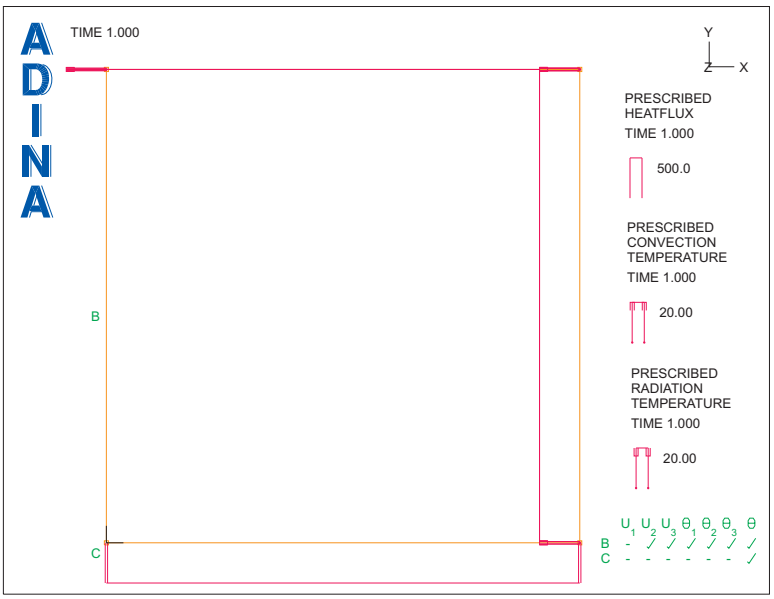
Defining the convection and radiation boundary conditions

We will impose convection and radiation boundary conditions 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, and click the … button to the right of the Convection Property field. In the Define Convection Property dialog box, add Property 1, make sure that the Type is set to CONSTANT, set the Convection Coefficient to 5 and click OK. In the Define Convection Load dialog box, set the Environment Temperature to 20, set the Convection Property to 1 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 Line # 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, and click the … button to the right of the Radiation Property field. In the Define Radiation Property dialog box, add Property 1, make sure that the Type is set to CONSTANT, set the Temperature Unit to Centigrade, set the Emissivity Coefficient to 0.2, set the Stefan-Boltzmann Constant to 5.669E-08 and click OK. In the Define Radiation Load dialog box, set the Environment Temperature to 20, set the Radiation Property to 1 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 Line # to 1 and 4 respectively. Click OK to close the Apply Load dialog box.

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



Defining the element group and materials

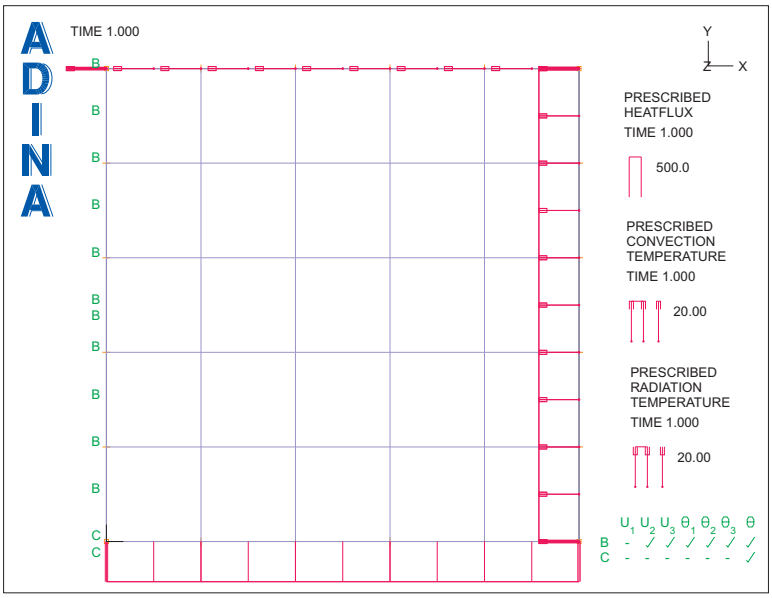
Element group: Click the Element Groups icon, add group number 1, set the Type to 2-D Solid and make sure that the Element Sub-Type is Axisymmetric. Click the … button to the right of the Default Material field and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog 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. In the Define Element Group dialog box, click the … button to the right of the Thermal Material field 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. Click Close to close the

Manage Material Definitions dialog box, and click OK to close the Define Element Groups dialog box.

Defining the elements

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: Click the Mesh Surfaces icon , set the Type to 2-D Solid if necessary, enter 1 in the first row of the Surface # table and click OK. The graphics window should look something like this:

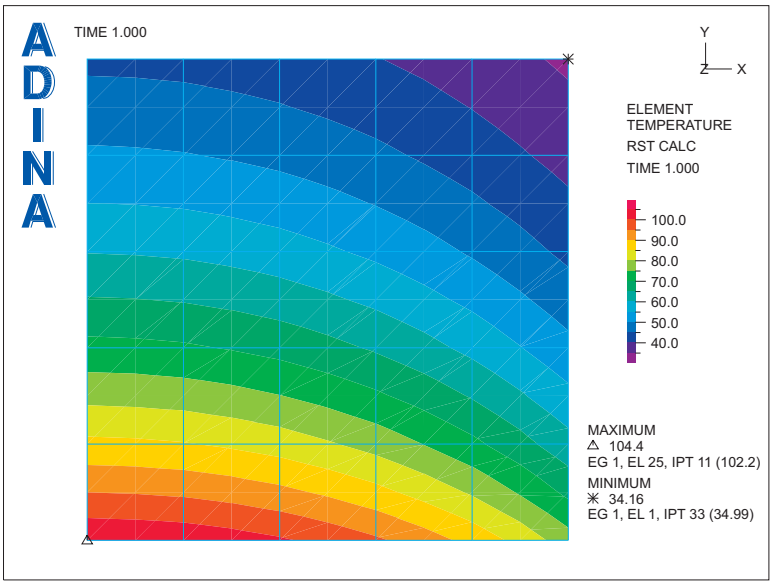


Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

First click the Save icon and save the database to file `prob46`. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon , set the file name to `prob46`, 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 `prob46`.

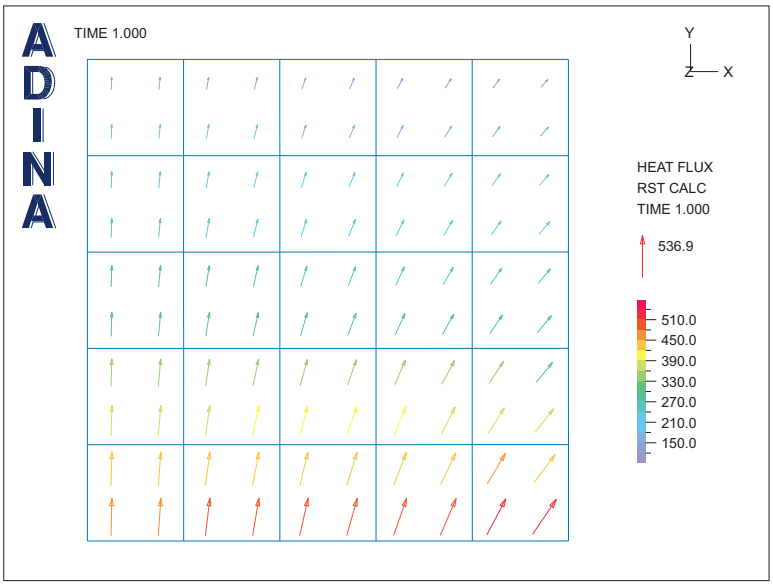
Examining the solution

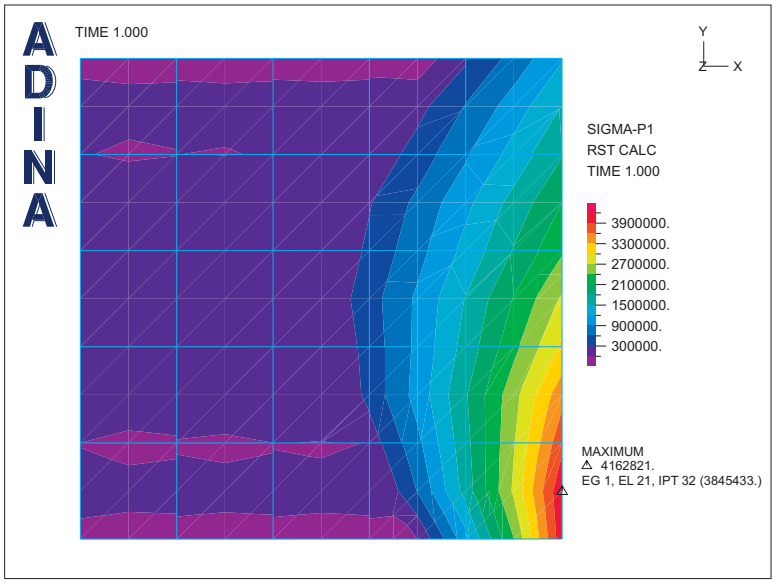
Temperatures: Click the Create Band Plot icon , set the Band Plot Variable to (Temperature: ELEMENT_TEMPERATURE). The graphics window should look something like this:



Heat fluxes: Click the Clear Band Plot icon , click the Create Vector Plot icon , set the Vector Quantity to HEAT_FLUX and click OK. The graphics window should look something like the top figure on the next page.

Maximum principal stresses: Click the Clear Vector Plot icon , click the Create Band Plot icon , choose variable (Stress:SIGMA-P1) and click OK to display the maximum principal stress. Click the Modify Band Plot icon , click the Band Rendering... button, set the "Extreme Values" field to "Plot the Maximum" and click OK twice to close both dialog boxes. The graphics window should look something like the bottom figure on the next page.





The temperature and stress solutions are almost exactly the same as were obtained in primer problem 9.

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

This page intentionally left blank.