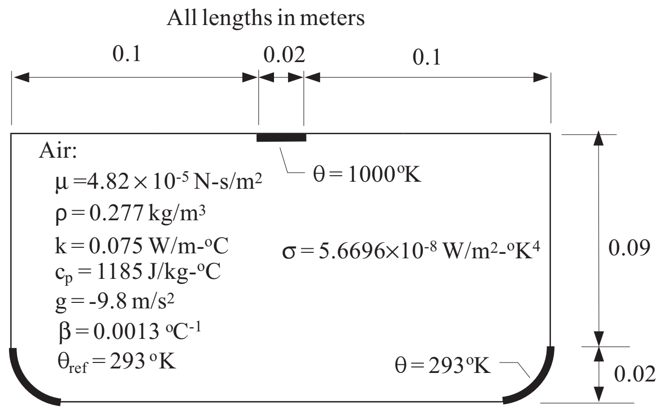
Problem description

We determine the fluid flow and temperature distribution within the enclosure shown in the figure.



Boundaries marked with ▬ are reflecting boundaries with $s = 0.5$ (specular reflectivity), $d=0.5$ (diffusivity)

Other boundaries are absorbing boundaries with $s=0.0$, $d=0.1$.

The enclosure includes three reflectors and an absorbing boundary. Radiation heat transfer occurs between these items. Heat transfer also occurs due to natural convection of the fluid and thermal conduction within the fluid.

Notice that the temperatures are prescribed at two of the reflectors. The temperature at the third reflector is not prescribed and is solved for as part of the solution process.

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

- Assignment of material data for natural convection
- Assignment of initial temperatures
- The use of relative pressure within the analysis
- Automatic nondimensionalization
- Assignment of specular boundary conditions
- Free-form meshing within geometry surfaces
- Defining a trace rake of type grid
- Setting the particle time step size

We assume that you have worked through problems 1 to 19, or have equivalent experience with the ADINA System. Therefore we will not describe every user selection or button press.

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 cannot be solved with the 900 nodes version of the ADINA System because there are 3191 nodes in the model.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA CFD.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 20: Natural convection and specular radiation within an enclosure" and click OK.

Flow assumptions: Choose Model→Flow Assumptions, set the Flow Dimension to 2D (in YZ Plane) and click OK.

Number of iterations: Choose Control→Solution Process, click the Iteration Method… button, set the Maximum Number of Iterations to 50 and click OK twice to close both dialog boxes.

Tolerances: Choose Control→Solution Process, click the Iteration Tolerances… button, set the "Relative Tolerance for Degrees of Freedom" to 0.01 and click OK twice to close both dialog boxes.

Initial temperature: We want to set the initial temperature to 293^o K. Choose Control→ Default Temperature, set the Default Initial Temperature to 293 and click OK.

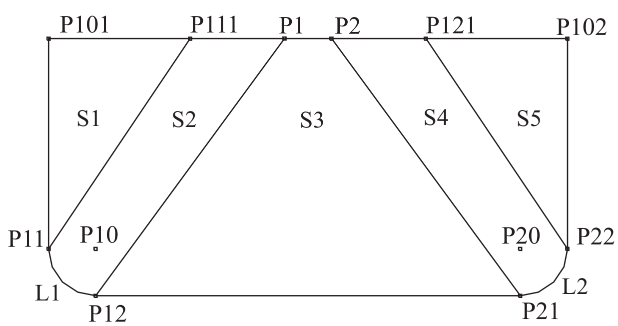
Relative pressure: Choose Control→Miscellaneous Options, uncheck the "Include Hydrostatic Pressure" button and click OK. This means that the pressure variable in the solution procedure will not include hydrostatic effects; this technique is usually used when buoyancy is present in the model.

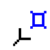
Non-dimensionalization: Choose Control→Solution Process, check the Non-Dimensional Analysis button, click the ... button to the right of that button, set the Length Scale to 0.01, the Velocity Scale to 0.1, the Density Scale to 0.277, the Specific Heat Scale to 1185.0, the Temperature Scale to 1000.0, the Temperature Datum to 293.0, then click OK twice to close both dialog boxes (be careful to close the Non-Dimensional Analysis dialog box first).
The velocity scale is determined so that the convective term in the energy equation has the

same order of magnitude as the radiation term. The temperature scale is used to reduce the temperatures used in the solution process; if the temperatures are not reduced, the numerical procedure will diverge due to large θ^4 values from the radiation terms.

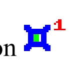
Defining the model geometry

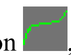
The following diagram shows the key geometry used in defining the ADINA CFD model.



Geometry points: Click the Define Points icon , enter the following points (you can leave the X1 column blank) and click OK.

Point #	X2	X3
1	-0.01	0.1
2	0.01	0.1
10	-0.09	0.01
11	-0.11	0.01
12	-0.09	-0.01
20	0.09	0.01
21	0.09	-0.01
22	0.11	0.01
101	-0.11	0.1
102	0.11	0.1
111	-0.05	0.1
121	0.05	0.1

Click the Point Labels icon  to show the geometry point labels.

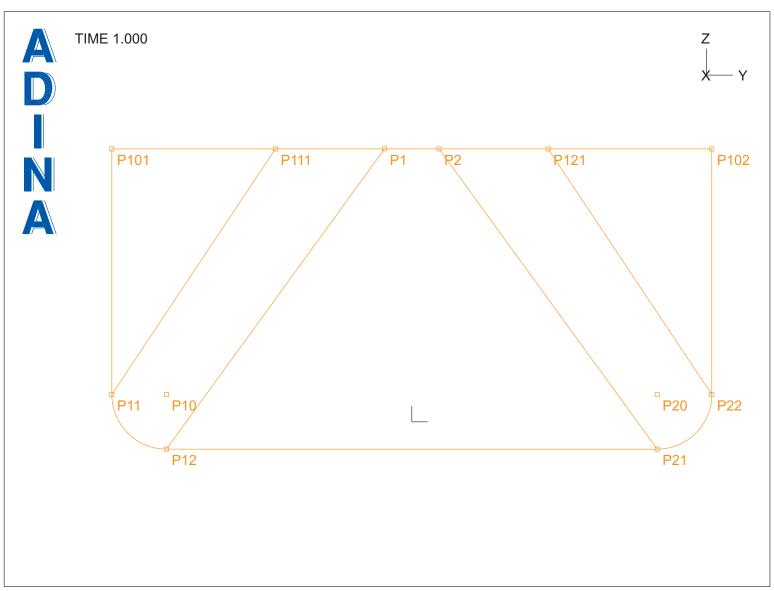
Geometry lines: Click the Define Lines icon , define the following lines and click OK:

Line number	Type	Defined by	P1	P2	Center
1	Arc	P1, P2, Center	11	12	10
2	Arc	P1, P2, Center	21	22	20

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

Surface number	Type	Point 1	Point 2	Point 3	Point 4
1	Vertex	111	101	11	111
2	Vertex	1	111	11	12
3	Vertex	2	1	12	21
4	Vertex	121	2	21	22
5	Vertex	121	22	102	121

Notice that surfaces 1 and 5 are triangular surfaces. The graphics window should look something like this:



Defining material properties

Click the Manage Materials icon and click the Laminar button. In the Define Laminar Material dialog box, add material 1, set the Viscosity to 4.82E-5, the Density to 0.277, the Coefficient of Volume Expansion to 0.0013, the Reference Temperature to 293, the Thermal Conductivity to 0.075, the Specific Heat at Constant Pressure to 1185.0, the Acceleration due to Gravity, Z to -9.8 and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the boundary conditions

Wall boundary conditions: Click the Special Boundary Conditions icon, add special boundary condition 1 and verify that the Type is Wall. Enter line numbers 1, 2, 3, 4, 6, 8, 9, 11, 13, 14 in the first 10 rows of the table. (Note: the order of the lines in the table doesn't matter.) Click OK to close the dialog box.

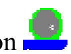
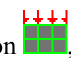
Pressure zero value: Because the flow is incompressible and we are specifying the velocity along the entire boundary, the pressure solution is not completely determined. In order to completely determine the pressure solution, we set the pressure to zero at one point in the model. Click the Apply Fixity icon and click the Define… button. In the Define Zero Values dialog box, add zero values name PRESSURE, check the Pressure degree of freedom and click OK.

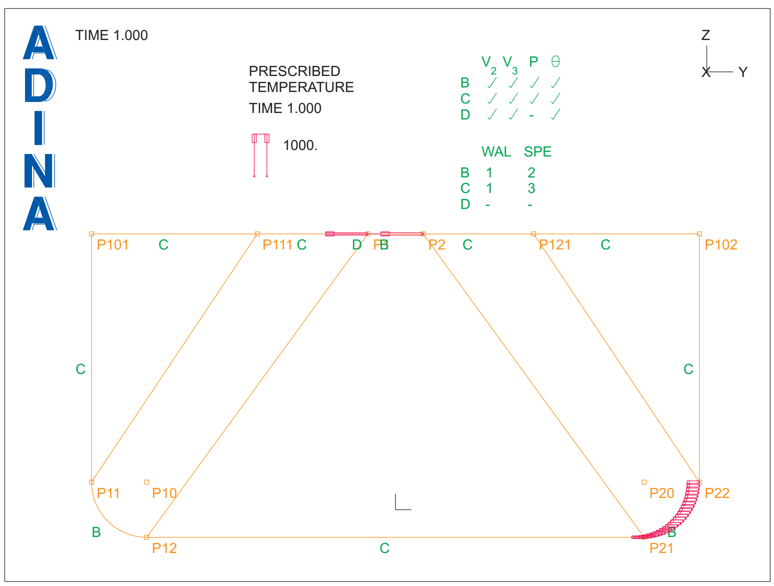
In the Apply Zero Values dialog box, set the Zero Values name to PRESSURE, verify that the "Apply to" field is Point, enter 1 in the first row of the table and click OK.

Prescribed temperatures: We need to prescribe the temperatures on the top center line (line 8) and the right-hand arc line (line 2). Click the Apply Load icon, set the Load Type to Temperature and click the Define... button to the right of the Load Number field. In the Define Temperature dialog box, add temperature 1, set the Magnitude to 1000, click Save, add temperature 2, set the Magnitude to 293 and click OK. In the Apply Usual Boundary Conditions/Loads dialog box, verify that the Load Number is set to 1, set the "Apply to" field to Line, set the Line # to 8 in the first row of the table and click Apply. Now set the Load Number to 2, verify that the "Apply to" field is set to Line, set the Line # to 2 in the first row of the table and click OK.

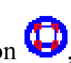
Specular boundary conditions: First we define the boundary condition for the reflectors (lines 8, 1, 2). Click the Special Boundary Conditions icon, add special boundary condition 2, set the Type to Specular-Diffusive-Radiation, the Stefan-Boltzmann Constant to 5.6696E-8, the "Number of Rays Emitted between Normal and Tangent Direction" to 20, the "Specular Reflectivity Function Multiplier" to 0.5 and the "Diffuse Reflectivity Function Multiplier" to 0.5. Enter line numbers 8, 1, 2 in the first three rows of the table and click Save.

Now we define the boundary condition for the remainder of the enclosure (lines 3, 4, 6, 9, 11, 13, 14). Copy special boundary condition 2 to 3, set the Specular Reflectivity Function Multiplier to 0.0 and the Diffuse Reflectivity Function Multiplier to 0.1. Clear the table and enter line numbers 3, 4, 6, 9, 11, 13, 14 in the first seven rows of the table. Click OK to close the dialog box.

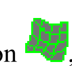
When you click the Boundary Plot icon  and the Load Plot icon , the graphics window should look something like this:

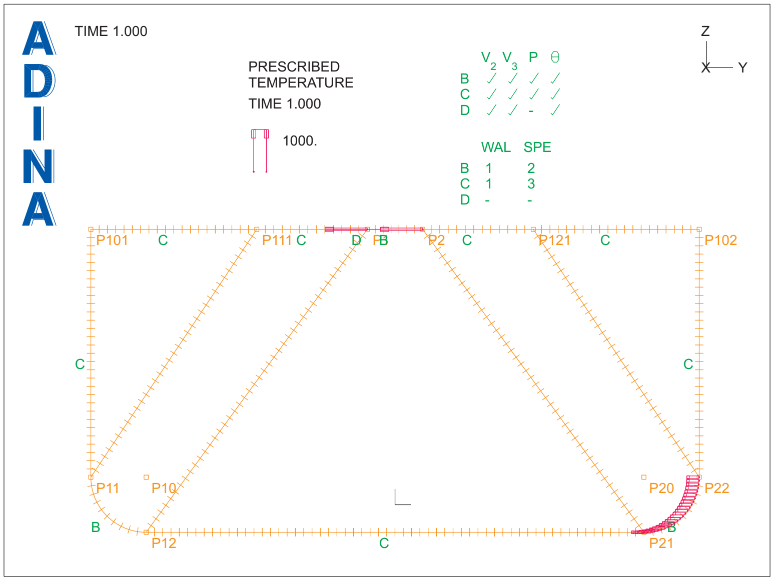


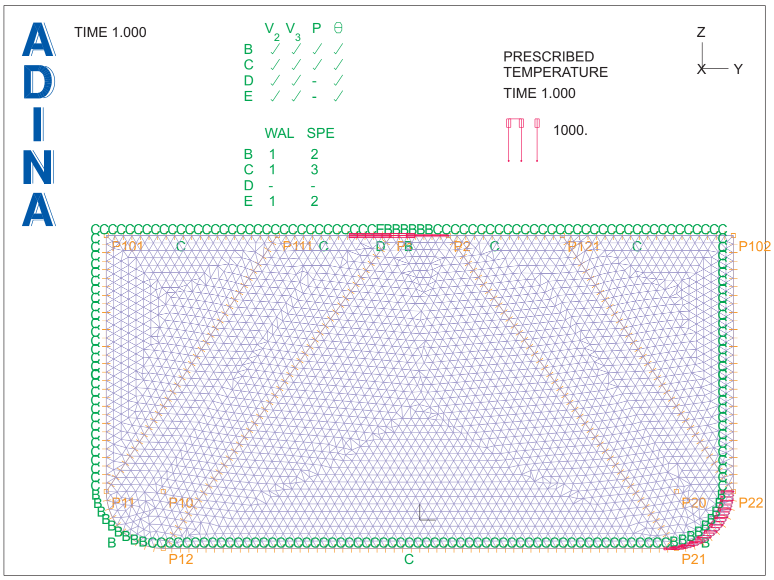
Defining the elements

Element group: Click the Element Groups icon , add element group 1, verify that the Type is 2-D Fluid, set the Element Sub-Type to Planar and click OK.

Subdivision data: We will use a uniform mesh size. Choose Meshing→Mesh Density→ Complete Model, set the “Subdivision Mode” to Use Length, set the “Element Edge Length” to 0.003 and click OK. The graphics window should look something like the top figure on the next page.

Now click the Mesh Surfaces icon , set the Meshing Type to Free-Form, set the Nodes per Element to 3, enter 1, 2, 3, 4, 5 in the first five rows of the table and click OK. Use the mouse to rearrange the graphics window until it looks something like the bottom figure on the next page.


TIME 1.000
PRESCRIBED
TEMPERATURE
TIME 1.000
1000.
WAL SPE
B 1 2
C 1 3
D - -
P101
P111
P2
P121
P102
P11
P10
P20
P22
P12
P21


TIME 1.000
WAL SPE
B 1 2
C 1 3
D - -
E 1 2
PRESCRIBED
TEMPERATURE
TIME 1.000
1000.
P101
P111
P2
P121
P102
P11
P10
P20
P22
P12
P21

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

Click the Save icon and save the database to file `prob20`. Click the Data File/Solution icon , set the file name to `prob20`, make sure that the Run Solution button is checked and click Save. When ADINA CFD 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 `prob20`.

Examining the solution

We will create plots of the results within the enclosure. As the underlying mesh plots will all have the same appearance, we set the appearance of the first mesh plot, then set the defaults to that appearance.

Click the Model Outline icon to plot just the mesh outline. Use the Pick icon and the mouse to erase the "TIME 1.000" text and the coordinate axes. Then click the Save Mesh Plot Style icon to save the mesh plot defaults.

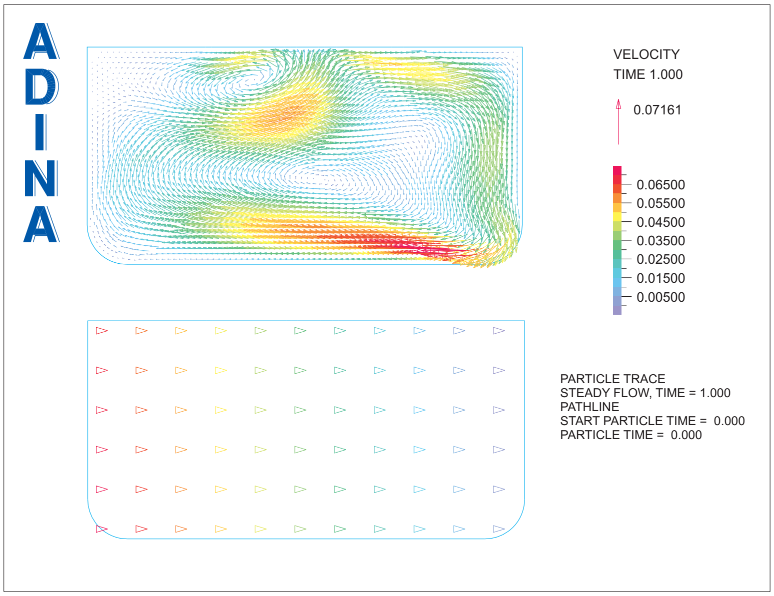
Velocity vectors: Click the Quick Vector Plot icon . Use the Pick icon and the mouse to move the mesh to the upper half of the graphics window.

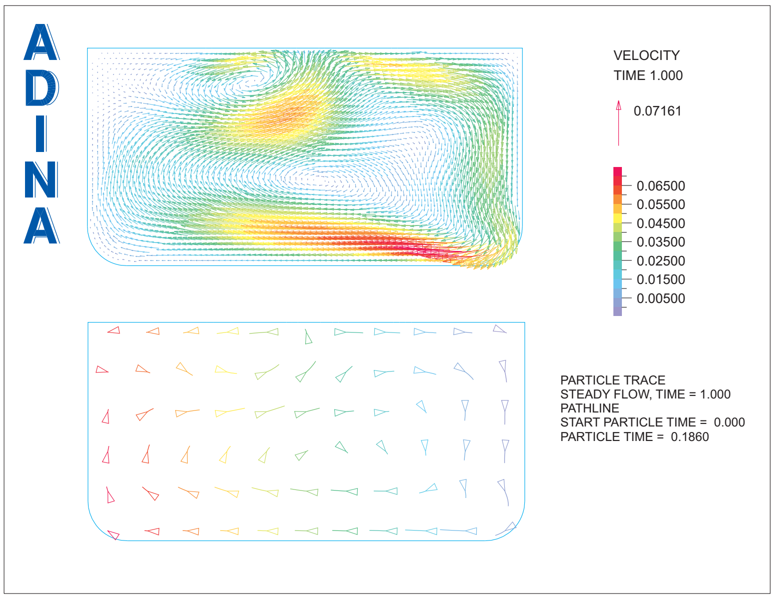
Pathlines: Click the Mesh Plot icon and move the mesh to the lower half of the graphics window. Now choose Display→Particle Trace Plot→Create and click the … button to the right of the Trace Rake field. In the Define Trace Rake dialog box, set the Type to Grids, enter the following data in the first row of the table and click OK twice to close both dialog boxes.

X	Y	Z	Plane	Shape	Side 1 Length	NSIDE1	Side 2 Length	NSIDE2
0.0	0.0	0.045	X-Plane	Rectangular	0.2	11	0.1	6

Move the particle trace legend until the graphics window looks something like the top figure on the next page. The rake is a rectangular grid of injectors with center (0,0,0.045) with side lengths 0.2 and 0.1.

Now click the Trace Downstream icon 5 times. The graphics window should look something like the bottom figure on the next page.


ADINA
VELOCITY
TIME 1.000
0.07161
0.06500
0.05500
0.04500
0.03500
0.02500
0.01500
0.00500
PARTICLE TRACE
STEADY FLOW, TIME = 1.000
PATHLINE
START PARTICLE TIME = 0.000
PARTICLE TIME = 0.000


ADINA
VELOCITY
TIME 1.000
0.07161
0.06500
0.05500
0.04500
0.03500
0.02500
0.01500
0.00500
PARTICLE TRACE
STEADY FLOW, TIME = 1.000
PATHLINE
START PARTICLE TIME = 0.000
PARTICLE TIME = 0.1860

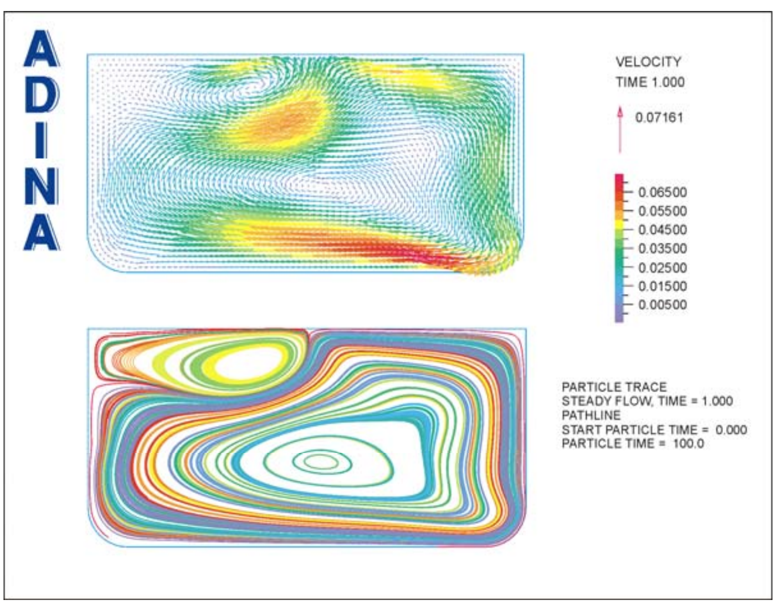
It seems like the default particle time step is too small, since we have to click the Trace Step Downstream icon several times before we notice any difference in the plot. To reset the default particle time step, choose Display→Particle Trace Plot→Modify and click the … button to the right of the Trace Calculation field. The Particle Time Step Size is currently 0.037194. Set the Particle Time Step Size to 1.0 and click OK twice to close both dialog boxes. The plot doesn't change since we have not changed the particle time.

(Note: do not confuse the particle time step with the time step that the AUI uses for the numerical integration of the particle traces. The numerical integration of the particle traces is completely separate from the particle time step. The particle time step is used only to provide a time step for the Trace Downstream and Trace Upstream icons.)

Now click the Trace Downstream icon  twice. Each time you click the icon, the particle time increases by 1.0.

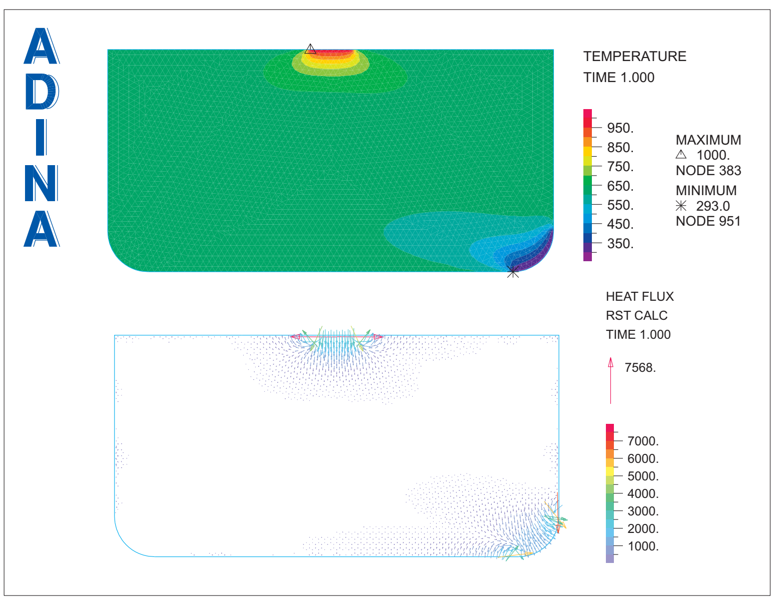
To remove the injector triangles, choose Display→Particle Trace Plot→Modify and click the … button to the right of the Trace Rendering field. In the Define Trace Rendering Depiction dialog box, uncheck the "Display Symbols at Injector Locations" button and click OK twice to close both dialog boxes.

To create longer pathlines, choose Display→Particle Trace Plot→Modify and click the … button to the right of the Trace Calculation field. Set the Current Particle Time to 100 and click OK twice to close both dialog boxes. The graphics window should look something like this:



Temperature: Click the Clear icon, then the Mesh Plot icon, then click the Create Band Plot icon, set the Band Plot Variable to (Temperature:TEMPERATURE) and click OK. Use the Pick icon and the mouse to move the mesh to the upper half of the graphics window.

Heat fluxes (due to conduction within the fluid): Click the Mesh Plot icon, then click the Create Vector Plot icon, set the Vector Quantity to HEAT_FLUX and click OK. Use the Pick icon and the mouse to rearrange the graphics, until the graphics window looks something like this:



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

This page intentionally left blank.