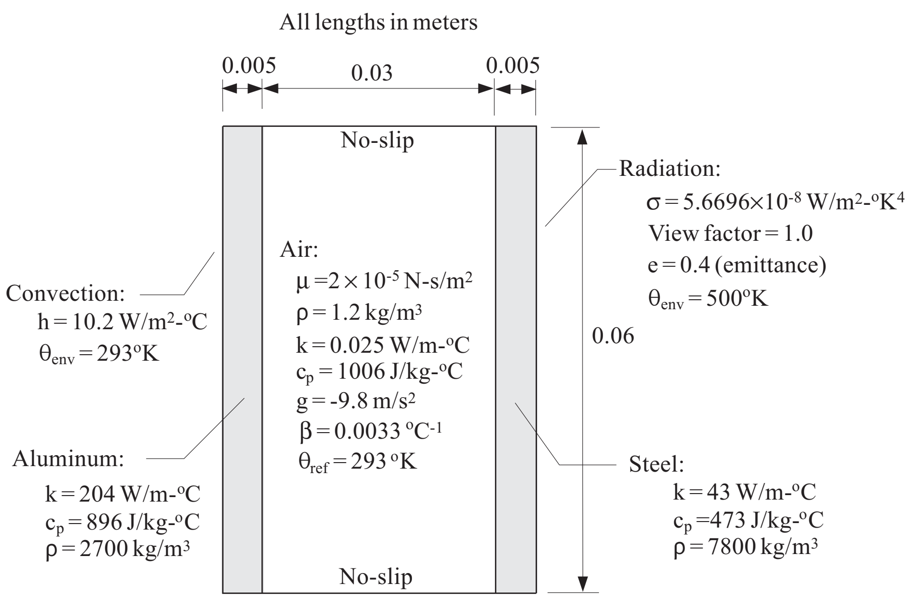
Problem description

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



Both the solid walls and fluid within the enclosure are modeled using ADINA CFD. The solid walls are subjected to radiation and convection boundary conditions.

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

- The use of "solid" element groups in ADINA CFD
- Assignment of convection and radiation boundary conditions in ADINA CFD
- Particle trace plots showing the motions of single particles
- Animation of particle trace plots

We assume that you have worked through problems 1 to 20, 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 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 CFD.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading “Problem 21: Conjugate heat transfer and natural convection within an enclosure” and click OK.

Flow assumptions: Choose Model→Flow Assumptions, set the Flow Dimension field 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 100 and click OK twice to close both dialog boxes.

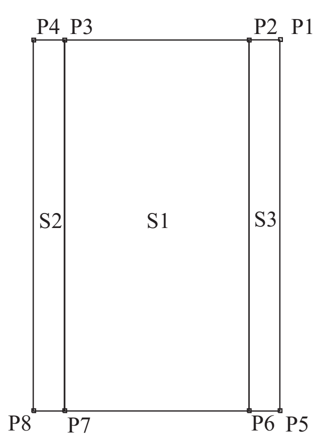
Initial temperature: Choose Control→Default Temperature, set the Default Initial Temperature to 400 and click OK.

Relative pressure: Choose Control→Miscellaneous Options, uncheck the Include Hydrostatic Pressure button and click OK.

Non-dimensionalization: Choose Control→Solution Process, check the Non-Dimensional Analysis button, click the ... button to the right of that field, set the Length Scale to 0.01, the Velocity Scale to 0.01, the Density Scale to 1.2, the Specific Heat Scale to 1006.0, the Temperature Scale to 500.0, the Temperature Datum to 500.0, then click OK twice to close both dialog boxes.

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	0.02	0.03
2	0.015	0.03
3	-0.015	0.03
4	-0.02	0.03
5	0.02	-0.03
6	0.015	-0.03
7	-0.015	-0.03
8	-0.02	-0.03

Now click the Point Labels icon to display the point numbers.

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

Surface number	Surface type	Point 1	Point 2	Point 3	Point 4
1	Vertex	2	3	7	6
2	Vertex	1	2	6	5
3	Vertex	3	4	8	7

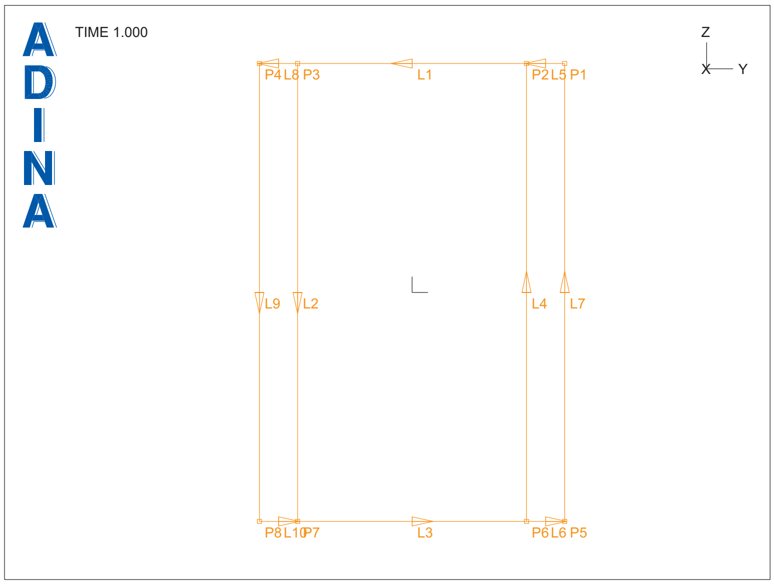
After you click the Line/Edge Labels icon, the graphics window should look something like the figure on the next page.

Defining material properties

Click the Manage Materials icon and click the Laminar button.

Air: In the Define Laminar Material dialog box, add material 1, set the Viscosity to 2.0E-5, the Density to 1.2, the Coefficient of Volume Expansion to 0.0033, the Reference Temperature to 293.0, the Thermal Conductivity to 0.025, the Specific Heat at Constant Pressure to 1006.0, the Acceleration due to Gravity, Z to -9.8 and click Save.

Steel: In the Define Laminar Material dialog box, add material 2, set the Density to 7800.0, the Thermal Conductivity to 43.0, the Specific Heat at Constant Pressure to 473.0, and click Save.



Aluminum: In the Define Laminar Material dialog box, add material 3, set the Density to 2700.0, the Thermal Conductivity to 204.0, the Specific Heat at Constant Pressure to 896.0, and click OK.

Click Close to close the Manage Material Definitions dialog box.

Defining the boundary conditions

Wall boundary conditions: The lines on which we need to assign no-slip boundary conditions are lines 1 to 4. Click the Special Boundary Conditions icon , add special boundary condition 1 and verify that the Type is Wall. Enter 1, 2, 3, 4 in the first four rows of the table and click Save (do not close the dialog box yet). Note: it is recommended to assign wall boundary conditions to the lines between the solid regions and the fluid region.

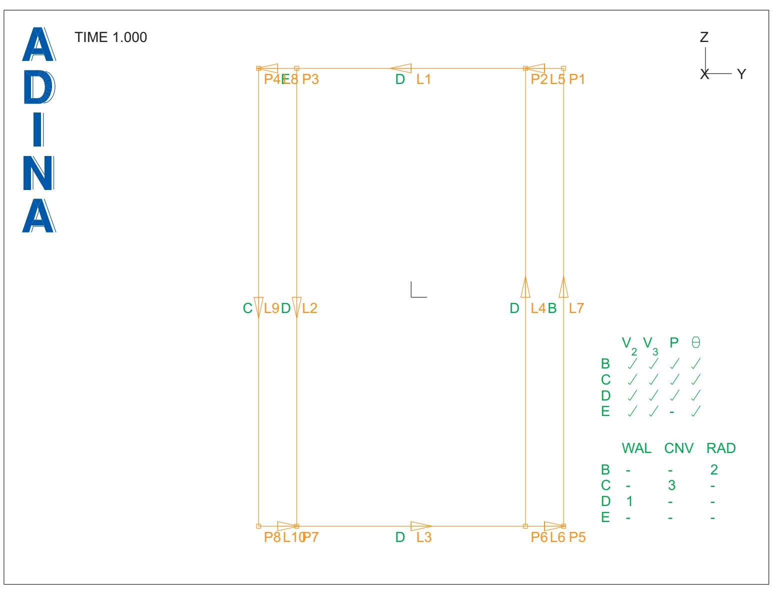
Radiation boundary condition: We need to prescribe a radiation boundary condition to the right-hand line of the model (line 7). In the Special Boundary Condition dialog box, add special boundary condition 2, set the Type to Heat Transfer Radiation, set the View Factor to 1.0, the Stefan-Boltzmann constant to 5.6696E-8, the Radiation Coefficient Function Multiplier to 0.4 and the Environment Temperature Function Multiplier to 500.0. Enter 7 in the Line # column in the first row of the table and click Save.

Convection boundary condition: We need to prescribe a convection boundary condition to the left-hand line of the model (line 9). In the Special Boundary Condition dialog box, add special boundary condition 3, set the Type to Heat Transfer Convection, set the Convection Coefficient Function Multiplier to 10.2 and the Environment Temperature Function Multiplier to 293.0. Enter 9 in the Line # column in the first row of the table and 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 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 this:

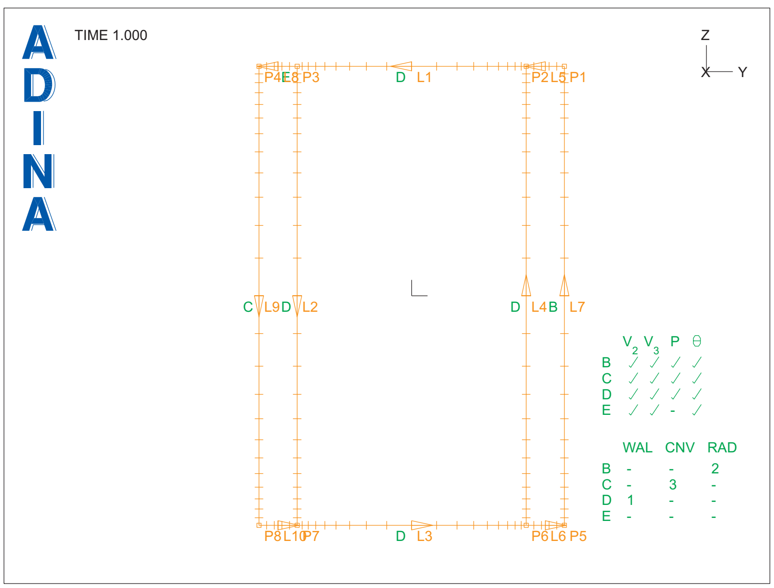


Defining subdivision data

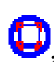
We will grade the mesh so that the fluid mesh is refined near the walls, therefore we will use a nonuniform mesh size with central biasing. Click the Subdivide Surfaces icon, enter the following data and click OK.

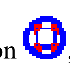
Surface #	Number of Subd. in u-dir	Number of Subd. in v-dir	Length Ratio in u-dir	Length Ratio in v-dir	Use Central Biasing for u-dir	Use Central Biasing for v-dir
1	18	24	5	5	Yes	Yes
2	5	24	1	5	No	Yes
3	5	24	1	5	No	Yes

The graphics window should look something like this:



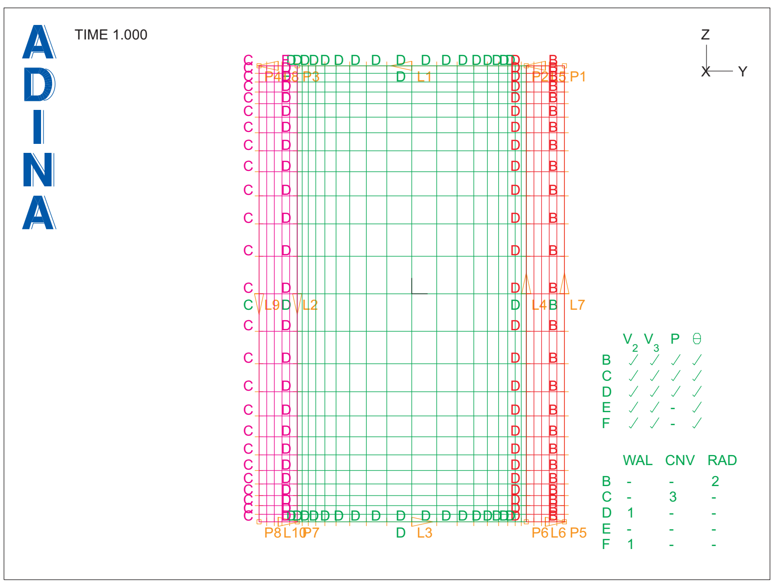
Defining the elements

Air: 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. Now click the Mesh Surfaces icon , enter 1 in the first row of the table and click OK.

Steel: Click the Element Groups icon , add element group 2, verify that the Type is 2-D Fluid, set the Element Sub-Type to Planar and set the Default Material to 2. Set the Element Option to Solid and click OK. Now click the Mesh Surfaces icon , enter 2 in the first row of the table and click OK.

Aluminum: Click the Element Groups icon, add element group 3, verify that the Type is 2-D Fluid, set the Element Sub-Type to Planar and set the Default Material to 3. Set the Element Option to Solid and click OK. Now click the Mesh Surfaces icon, enter 3 in the first row of the table and click OK.

Click the Color Element Groups icon, then use the mouse to rearrange the graphics window until it looks something like this:



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

Click the Save icon and save the database to file `prob21`. Click the Data File/ Solution icon, set the file name to `prob21`, 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 `prob21`.

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.

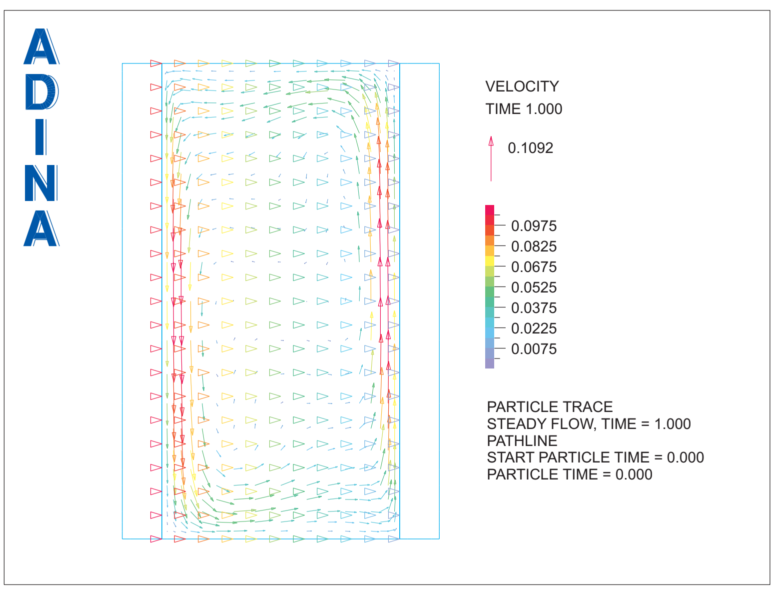
Click the Group Outline icon to plot just the outlines of the element groups. Use the mouse to erase the "TIME 1.000" text and the coordinate axes. Then click the Save Mesh Plot Style icon to save the mesh plot defaults.

Velocity vectors: Click the Quick Vector Plot icon.

Particle traces: We will show the particle traces in the same mesh plot. 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.

X	Y	Z	Plane	Shape	Side 1 Length	NSIDE1	Side 2 Length	NSIDE2
0.0	0.0	0.0	X-Plane	Rectangular	0.03	11	0.06	21

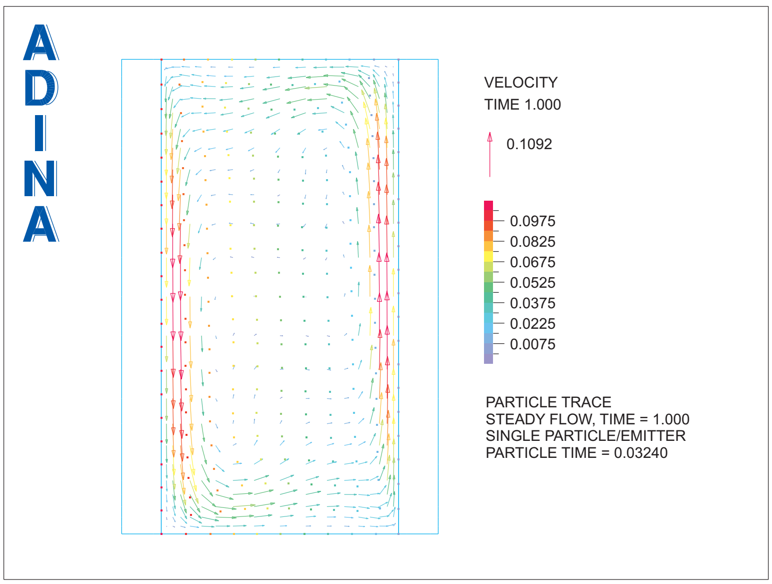
Click OK to close the Create Particle Trace Plot dialog box. Then use the mouse to rearrange the graphics window until it looks something like this:



Now click the Trace Downstream icon to start the pathlines.

We would rather display the actual particles instead of the particle paths. Choose Display→ Particle Trace Plot→Modify and click the … button to the right of the Trace Calculation field. Set the Trace Option to Single Particle and click OK twice to close both dialog boxes.

Choose Display→Particle Trace Plot→Modify and click the … button to the right of the Trace Rendering field. Uncheck the "Display Symbols at Injector Locations" button and click OK twice to close both dialog boxes. The graphics window should look something like this:

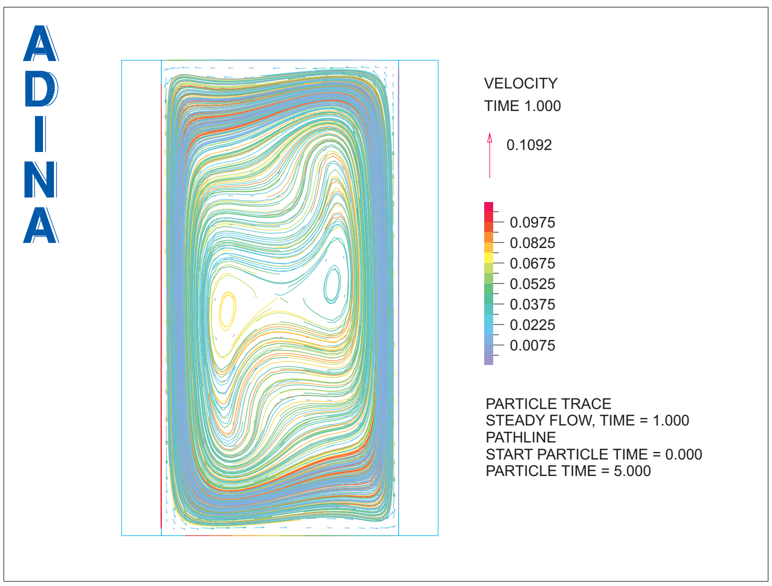


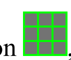
Now click the Trace Downstream icon several times. The particles move as the particle time increases. Notice that the particles near the boundaries of the fluid move faster than the particles near the center of the fluid. Also the particles move in the directions given by the velocity vectors.

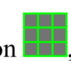
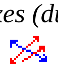
Now we will create an animation of the particles moving. Choose Display→Movie Shoot→Trace Step, set the End Time to 5.0 and click OK. The AUI computes the particle traces corresponding to particle times 0 to 5. Click the Animate icon to display the animation.

It is difficult to visualize the particle motions because the particles move too far between successive frames. Click the Refresh icon to clear the animation, then choose Display→Movie Shoot→Trace Step, set the End Time to 5.0, the Number of Frames to 201 and click OK. Click the Animate icon to display the animation. To slow down the animation further, choose Display→Animate, increase the Minimum Delay and click Apply. Click Cancel to close the Animate dialog box and click the Refresh icon to clear the animation.

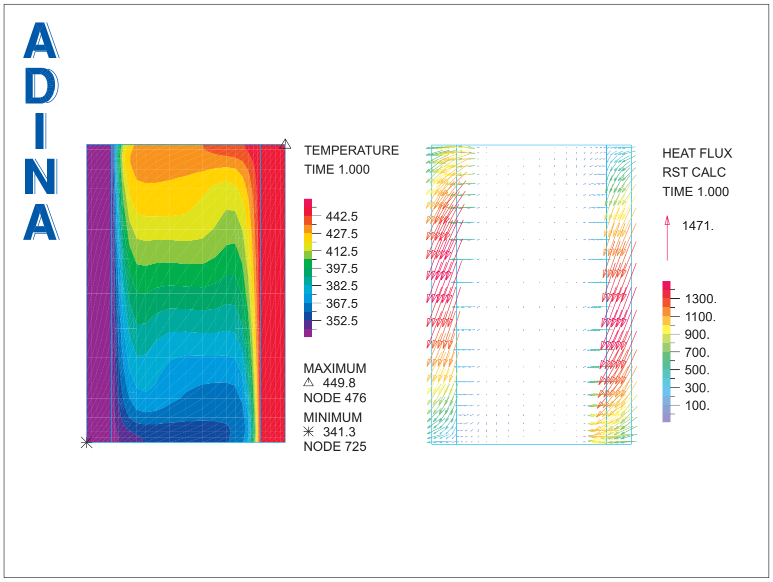
Let's create a pathline plot for the same range of particle times. Choose Display→ Particle Trace Plot→Modify and click the … button to the right of the Trace Calculation field. Set the Trace Option to Pathline, the Current Particle Time to 5.0 and click OK twice to close both dialog boxes. The graphics 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 mouse to move the mesh to the left half of the graphics window.

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

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


ADINA
TEMPERATURE
TIME 1.000
442.5
427.5
412.5
397.5
382.5
367.5
352.5
MAXIMUM
△ 449.8
NODE 476
MINIMUM
✳ 341.3
NODE 725
HEAT FLUX
RST CALC
TIME 1.000
1471.
1300.
1100.
900.
700.
500.
300.
100.

This page intentionally left blank.