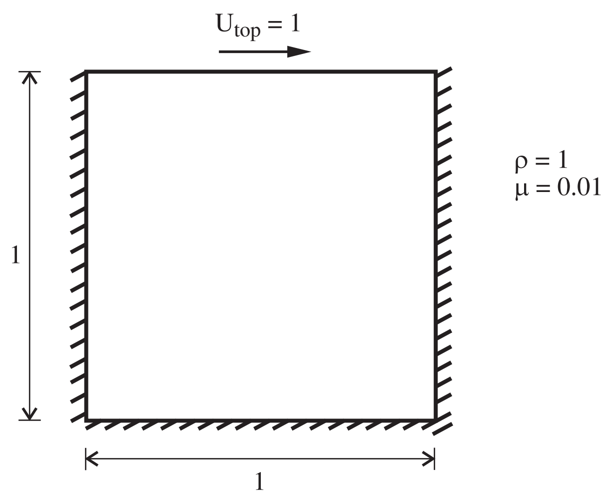
## Problem description

In this problem we determine the fluid flow within a square wall-driven cavity as shown:




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

- CFD analysis with ADINA-CFD
- Setting the flow assumptions
- Defining and applying special boundary conditions
- Plotting the outline of the mesh
- Plotting velocities as vectors
- Plotting particle traces
- Calculating the total force applied to the model

We assume that you have worked through problems 1 to 5, or have equivalent experience with the ADINA System.

**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.

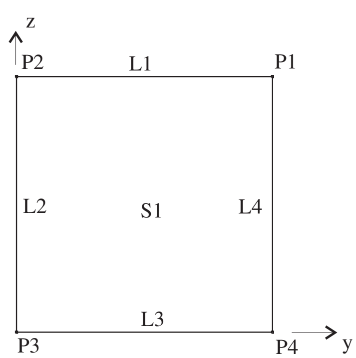
**Defining model control data**

*Problem heading:* Choose Control→Heading, enter the heading "Problem 6: Square wall-driven cavity" and click OK.

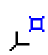
*Flow assumptions:* Choose Model→Flow Assumptions, set the “Flow Dimension” to “2D (in YZ Plane)”, uncheck the “Includes Heat Transfer” button and click OK.

**Defining model geometry**

Here is a diagram showing the key geometry used in defining this model:

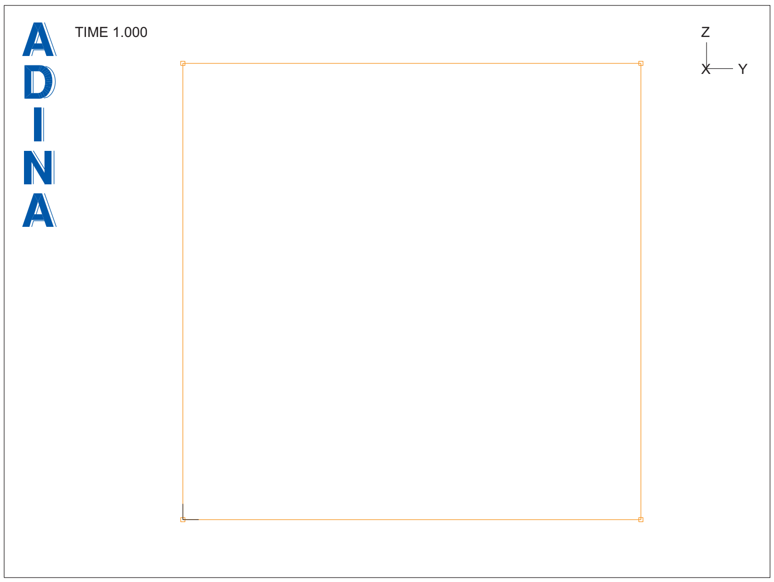



Notice that 2D models in ADINA CFD are defined in the y-z plane.

*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 | 1 | 1 |
| 2 | 0 | 1 |
| 3 | 0 | 0 |
| 4 | 1 | 0 |

*Geometry surfaces:* Click the Define Surfaces icon , add surface number 1, set the Type to Vertex if necessary, set Point 1 to 1, Point 2 to 2, Point 3 to 3, Point 4 to 4 and click OK. You can use the P button and the mouse to easily select these points. The graphics window should look something like the figure on the next page.

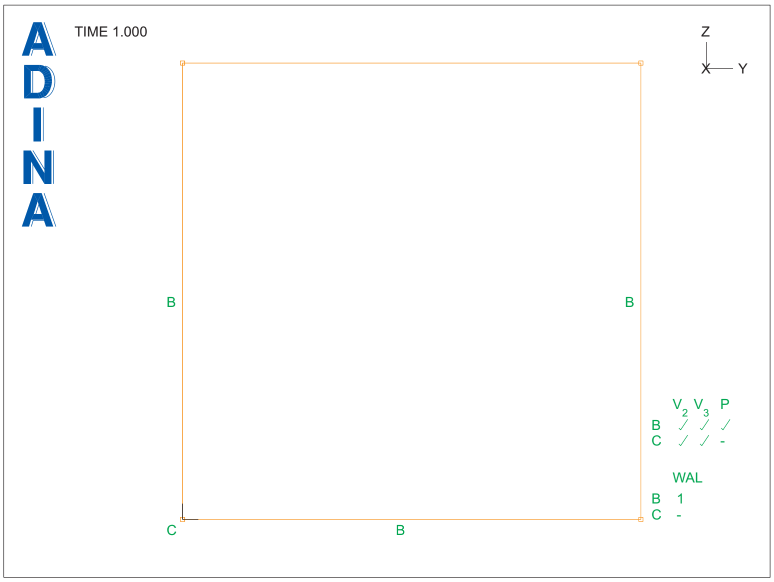



**Defining and applying boundary conditions**

*No-slip boundary conditions:* We need to apply the no-slip boundary conditions to three sides of the square. Click the Special Boundary Conditions icon, add boundary condition 1 and verify that the Type is Wall. Double-click in the first row and column of the table, use the mouse to pick the left, bottom and right lines and press the Esc key to return to the Define Special Boundary Condition dialog box. Make sure that the line numbers in the table are lines 2, 3, 4 (the order of the lines is not important). Click OK to close the Special Boundary Conditions 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 field 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 the figure on the next page.




*Velocity:* We need to apply the normal and tangential velocities to the line at the top of the square. Click the Apply Load icon, verify that the Load Type is Velocity and click the Define... button to the right of the Load Number field. In the Define Velocity dialog box, add velocity 1, set the Y and Z Prescribed Velocities to 1 and 0 respectively, and click OK. In the Apply Usual Boundary Conditions/Loads dialog box, set the "Apply to" field to Line, and, in the first row of the table, set the Line # to 1. Click OK to close the Apply Usual Boundary Conditions/Loads dialog box.

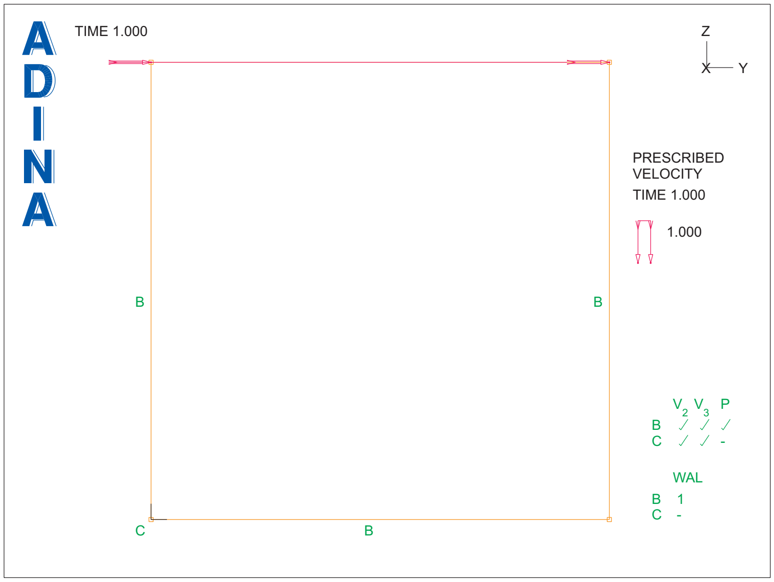
When you click the Load Plot icon, the graphics window should look something like the figure on the next page.

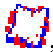
**Defining the material**

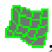
Click the Manage Materials icon and click the Laminar button. In the Define Laminar Material dialog box, add material 1, set the Viscosity to 0.01, the Density to 1 and click OK. Click Close to close the Manage Material Definitions dialog box.

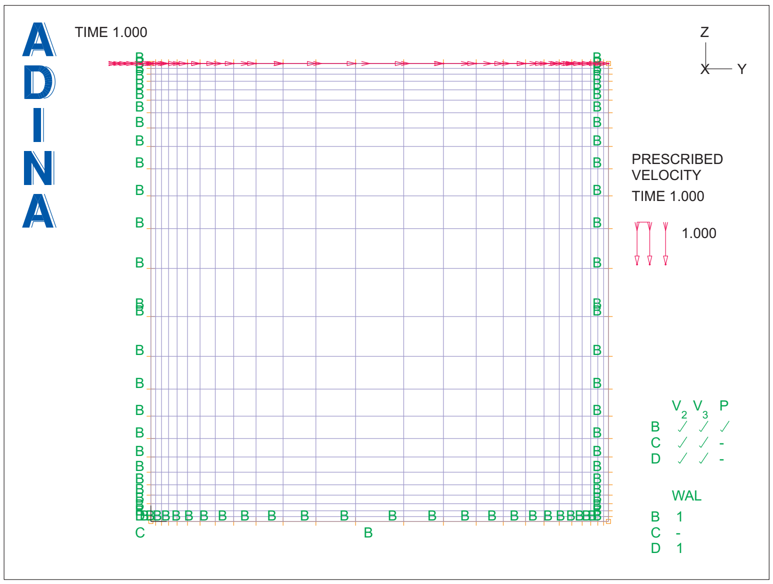
**Defining the elements**

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




*Subdivision data:* We use a 25×25 mesh for the solution, with a finer mesh near the corners. (This mesh is not fine enough to give an accurate solution; for an accurate solution, a more refined mesh would be required.) Click the Subdivide Surfaces icon , set the "Number of Subdivisions" for both the u and v directions to 25, set the "Length Ratio of Element Edges" for both the u and v directions to 10, check both of the "Use Central Biasing" buttons 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.




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

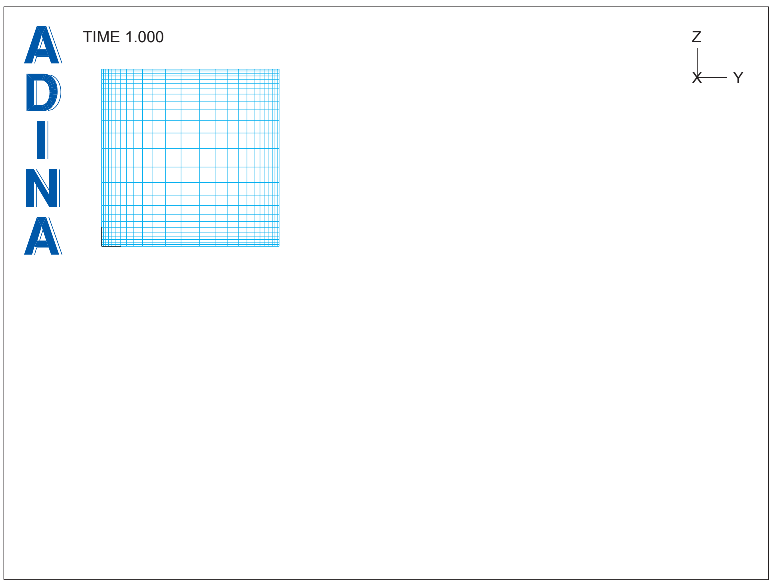
First click the Save icon and save the database to file `prob06`. To generate the ADINA-CFD data file and run ADINA-CFD, click the Data File/Solution icon, set the file name to `prob06`, make sure that the Run Solution button is checked and click Save. When ADINA-CFD is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon, set the "File type" field to "ADINA-IN Database Files (*.idb)", open database file `prob06`, click the Open icon and open porthole file `prob06`.

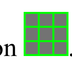
Please notice that we first opened the ADINA-IN database, then loaded the porthole file. We did this so that we can calculate the total force applied to the model later on.

**Examining the solution**

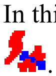
*Element mesh plot:* In all of the mesh plots that we will display, we do not want to show the geometry, loads or boundary conditions. Click the Show Geometry icon to hide the geometry, the Load Plot icon to hide the loads and the Boundary Plot icon to hide the boundary conditions. Then click the Save Mesh Plot Style icon to update the defaults.

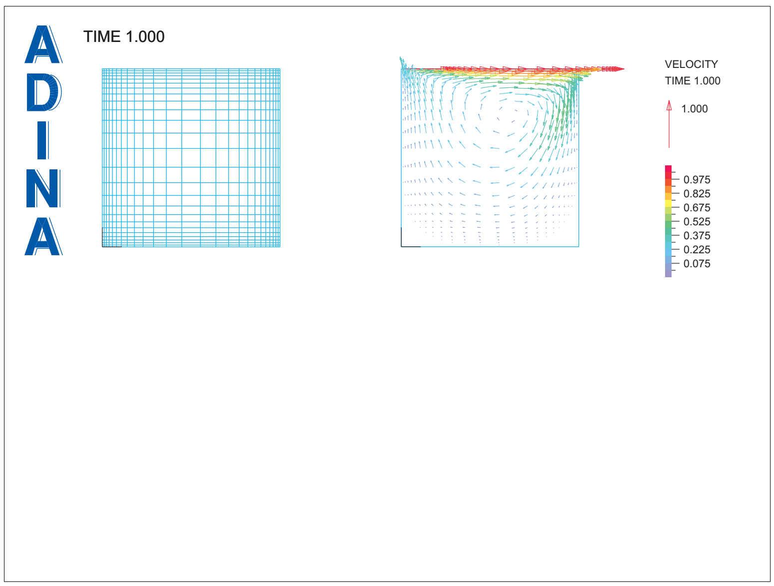
We will plot this mesh along with other mesh plots showing the solution. To accomplish this, shrink the mesh plot using the mouse and move it to the upper left-hand corner of the graphics window, so that the graphics window looks something like this:

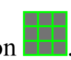



*Velocity vectors:* Click the Mesh Plot icon . Use the mouse to resize and move the new mesh plot to the upper right-hand corner of the graphics window. Also remove any extra plotted axes and "TIME 1.000" texts using the mouse.

Now click the Model Outline icon  to show only the mesh outline. Click the Save Mesh Plot Style icon  to update the mesh plot defaults.

In this plot we would like to show the velocity as vectors. Click the Quick Vector Plot icon . Use the mouse to move and resize the mesh and the "Velocity" annotation until you obtain the figure on the next page.

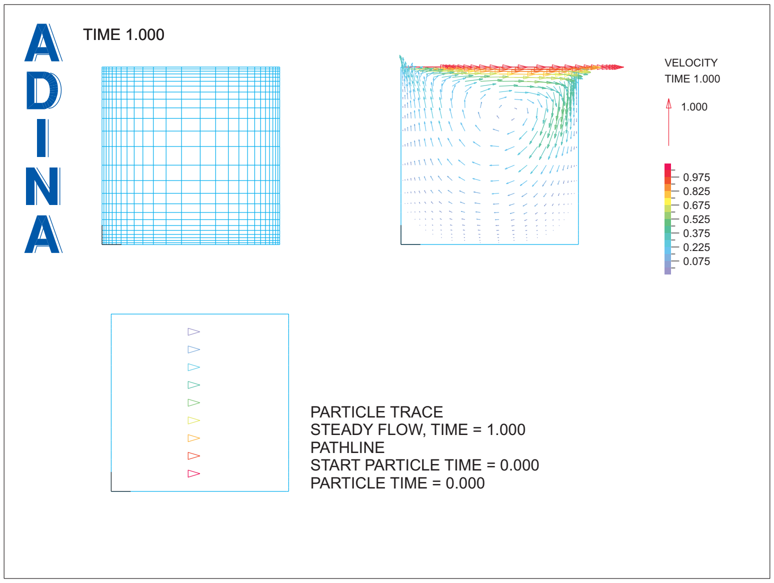



*Pathlines:* Click the Mesh Plot icon . The mesh plot might be plotted on top of the previous mesh plot, so you might have to move the previous mesh plot out of the way to see the new mesh plot. Use the mouse to resize and move the new mesh plot to the lower left-hand corner of the graphics window. Also, if necessary, remove the plotted axes and "TIME 1.000" text using the mouse.

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 Coordinates and click the Auto… button. In the Auto Generation dialog box, enter the following information into the table and click OK.

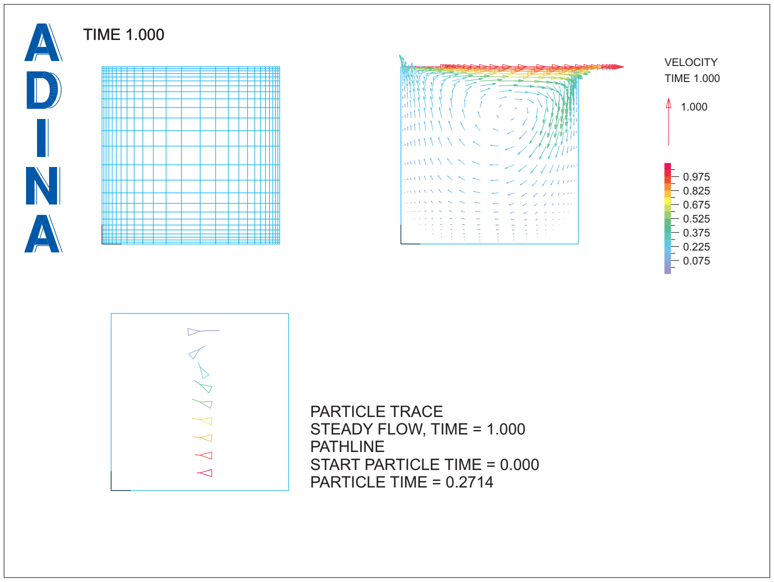
| X | Y | Z |
|---|---|---|
| | 0.5 | 0.1 |
| | | 0.1 |
| | 0.5 | 0.9 |

At this point, the table in the Define Trace Rake dialog box should contain 9 rows, in which Z=0.1, 0.2, …, 0.9. Click OK twice to close the Define Trace Rake dialog box and the Create Particle Trace Plot dialog box. The graphics window should look something like the figure on the next page.




The trace rake contains 9 injectors evenly spaced along a vertical line in the center of the model.

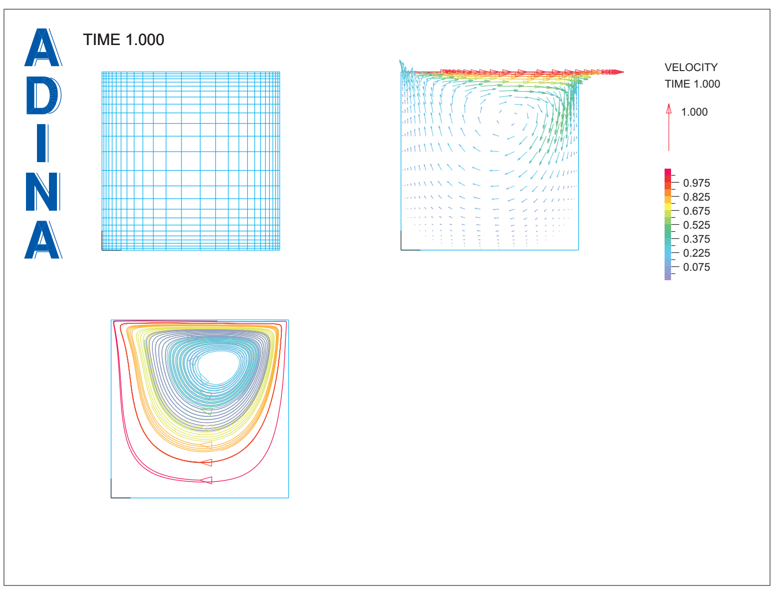
Now click the Trace Downstream icon  once. The graphics window should look something like this:



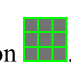
Notice that the injector triangles have rotated to correspond to the flow velocity and each injector has emitted a short pathline. In this plot, the start particle time is 0.0 and the particle time is 0.2714. This means that particles that are located at the injectors at particle time 0.0 have moved to the positions shown by particle time 0.2714. (Note that we use the term "particle time" to distinguish the time used in the particle tracing from the solution time.)

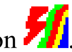
Click the Trace Downstream icon  a few more times to watch the pathlines grow. Each time you click the Trace Downstream icon, the particle time increases and the pathlines grow longer.

Now we directly specify the particle time. Choose Display→Particle Trace Plot→Modify and click the … button to the right of the Trace Calculation field. Set the Current Particle Time to 50 and click OK twice to close both dialog boxes. After you delete the particle trace legend with the mouse, the graphics window should look something like this:

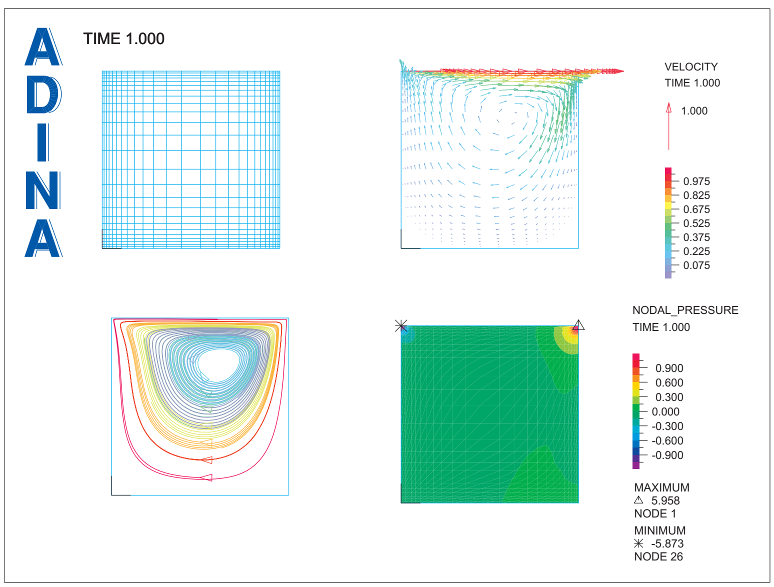



Notice that the outermost particle trace doesn't close completely. This is due to the coarseness of the mesh.

*Pressure band plot:* Click the Mesh Plot icon . Use the mouse to resize and move the new mesh plot to the lower right-hand corner of the graphics window. Also, if necessary, remove the plotted axes and "TIME 1.000" text using the mouse.

In this plot we would like to show the pressure. Click the Quick Band Plot icon . The pressures are most extreme at the top corners of the square, where the fluid flow turns through a right angle. We don't see much detail for the remaining plot because the scaling is set by the extreme values. To rescale the band plot, click the Modify Band Plot icon  and click the Band Table button. In the Value Range box, set the Maximum to 1 and the Minimum to -1, then click OK twice to close both dialog boxes.

After you move and resize the band table and the “Maximum” legend, the graphics window should look something like this:




*Total applied force:* To determine the total force applied to the cavity, we need to sum all of the reaction forces applied to the nodes on the walls of the cavity. Choose Definitions→ Model Point (Combination)→General, add name CAVITY, enter the text strings “LINE 2”, “LINE 3” and “LINE 4” (you don't need to enter the quotes) in the first three rows of the table and click OK. The AUI displays the message “76 nodes in gncombination” in the message window and at the bottom of the AUI control window.

Choose List→Value List→Model Point, set variable 1 to (Reaction: Y-REACTION) and click Apply. The AUI should output the value -2.47360E-01. The AUI computes this value by summing the y reactions over the nodes attached to geometry lines 2, 3 and 4. Hence the total force applied to the top wall of the cavity is +2.47360E-01. Click Close to close the dialog box.

(Please note: there are two ways to compute the total force. The first way is to add up all of the reaction forces on all of the nodes on which the velocities are prescribed. The second way is to add up all of the reaction forces on all of the nodes on which the velocities are fixed, then negate the result. We choose the second way because the input is easier.

It might seem logical to request the total reaction forces applied to the nodes on line 1. But this gives the wrong results for the following reason. Line 1 contains nodes 1 to 26. Nodes 1 and 26 are nodes on which the y velocities are fixed, and nodes 2 to 25 are nodes on which the y velocities are prescribed. Hence line 1 contains both prescribed velocity nodes and fixed velocity nodes.

If you sum the reaction force applied to nodes 2, …, 25, this gives the correct result, but the input is more difficult, since you then need to enter nodes 2, …, 25 directly into the Definitions→Model Point Combination→Node dialog box.)

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