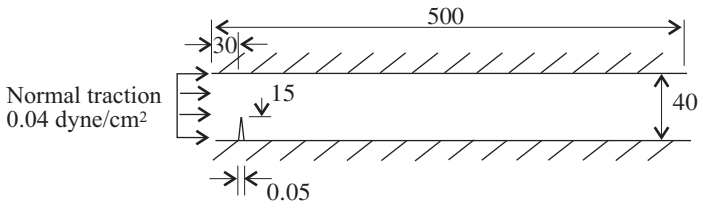
Problem description

We determine the fluid flow and the deformation of a very thin flexible structure within a two-dimensional channel, as shown:



Fluid:
$\mu = 1.7 \times 10^{-4}$ g/cm-s
$\rho = 0.001$ g/cm^3
All lengths in cm

Structure:
Elastic material:
$E = 1 \times 10^6$ dyne/cm^2
$\nu = 0.3$

In this problem solution, we use a fluid model for the fluid in the channel and a solid model for the structure. The fluid model is entered as an ADINA-CFD model and the solid model is entered as an ADINA model; however the analysis itself is performed as a fully coupled analysis using ADINA-FSI.

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

- Performing a fluid-structure-interaction (FSI) analysis in which separate AUI databases are used for the solid and fluid models.
- Plotting a model using stretch factors.
- Defining leader-follower points.
- Resizing element vectors with the mouse.

We assume that you have worked through problems 1 to 17, 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 the 900 nodes version of the ADINA System does not include ADINA-FSI.

ADINA model

Invoking the AUI and choosing the finite element program

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

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 18: Fluid flow over a flexible structure in a channel, ADINA input" and click OK.

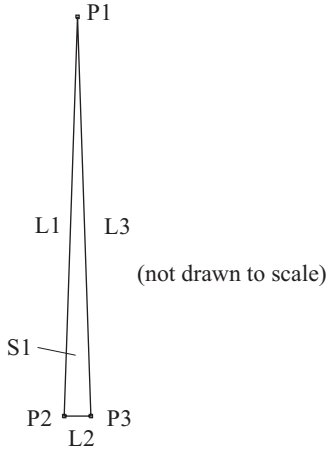
FSI analysis: Set the FSI/No FSI drop-down list to FSI.

Master degrees of freedom: Choose Control→Degrees of Freedom, uncheck the X-Translation, X-Rotation, Y-Rotation and Z-Rotation buttons and click OK.

Analysis assumptions: We anticipate that the structural displacements will be large, but that the strains will be small. Choose Control→Analysis Assumptions→Kinematics, set the "Displacements/Rotations" field to Large and click OK. (Note: the strains will be small because the structure is so thin.)

Defining model geometry

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



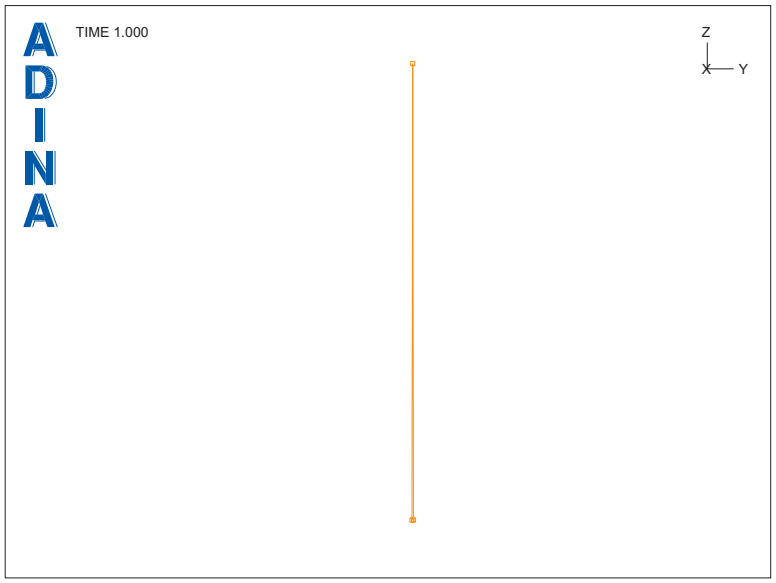
Geometry points: Click the Define Points icon , enter the following information into the table (remember to leave the X1 column blank) and click OK.

Point #	X2	X3
1	30.025	15.0
2	30.0	0.0
3	30.05	0.0

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

Surface #	Type	Point 1	Point 2	Point 3	Point 4
1	Vertex	1	2	3	1

The graphics window should look something like this:

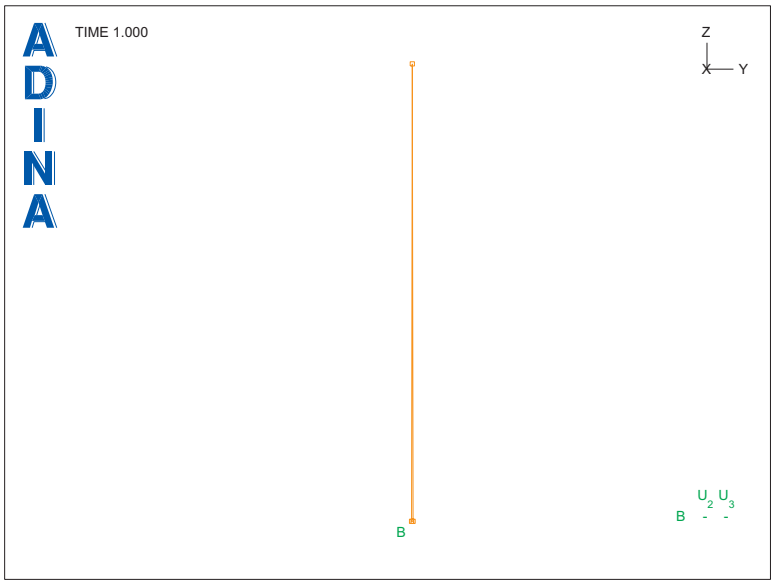


Defining the boundary conditions

Fixities: We will fix the bottom line of the structure. Click the Apply Fixity icon , change the "Apply to" option to Lines, enter line number 2 in the first row of the table, leave the fixity column blank and click OK.

Fluid-structure boundaries: Lines 1 and 3 are fluid-structure boundaries. Choose Model→ Boundary Conditions→FSI Boundary, add FSI boundary number 1, enter 1, 3 in the first two rows of the table and click OK.

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



Defining the material

Click the Manage Materials icon and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 1, set the Young's Modulus to 1E6, the Poisson's ratio to 0.3 and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the elements

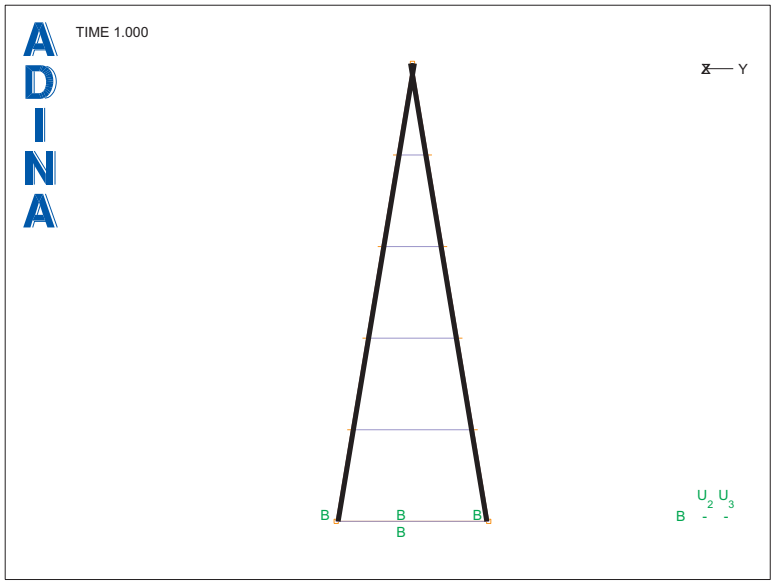
Element group: Click the Define Element Groups icon , add element group 1, set the Type to 2-D Solid, the Element Sub-Type to Plane Strain and click OK.

Subdivision data: We will use a 5 × 1 mesh, so we assign 5 subdivisions to lines 1 and 3.

Click the Subdivide Lines icon , select line 1, set the Number of Subdivisions to 5, enter 3 in the first row of the table and click OK.

Element generation: Click the Mesh Surfaces icon, check the "Triangular Surfaces Treated as Degenerate" button, enter 1 in the first row of the table and click OK.

It's difficult to see the elements because the model is so thin. We will plot the model using a stretch factor. To do this, click the Modify Mesh Plot icon, click the View… button, set the X Stretch factor to 100.0 and click OK twice to close both dialog boxes. The graphics window should look something like this:



Generating the ADINA data file, saving the database

Click the Data File/Solution icon, set the file name to prob18_a, uncheck the Run Solution button and click Save. Now click the Save icon and save the database to file prob18_a.

ADINA-CFD model

Click the New icon to create a new database. Choose ADINA CFD from the Program Module drop-down list.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 18: Fluid flow over a structure in a channel, ADINA-CFD model" and click OK.

FSI analysis: Set the FSI/No FSI drop-down list to FSI.

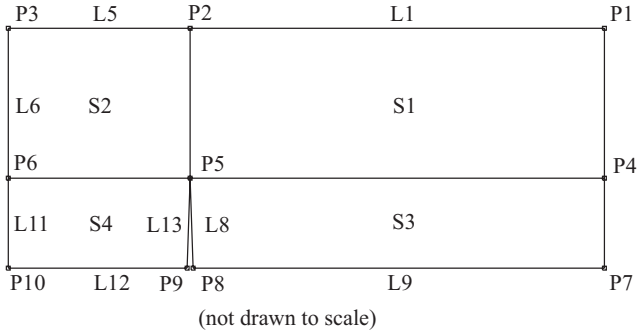
Flow assumptions: Choose Model→Flow Assumptions, set the Flow Dimension to 2D (in YZ Plane), uncheck the Includes Heat Transfer button and click OK.

Time steps and time function: In this model, we will apply the normal traction in 70 steps. Choose Control→Time Step, set the number of steps to 70 in the first row of the table and click OK. Now choose Control→Time Function, edit the table as follows and click OK.

Time	Value
0	0.0
1	0.0001
2	0.0003
3	0.0008
20	0.0024
30	0.0044
40	0.01
70	0.04

Defining the model geometry

Here is a diagram showing the key geometry used in defining the ADINA-CFD model.



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

Point #	X2	X3
1	500	40
2	30.025	40
3	0	40
4	500	15
5	30.025	15
6	0	15
7	500	0
8	30.05	0
9	30	0
10	0	0

Now 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	1	2	5	4
2	Vertex	2	3	6	5
3	Vertex	4	5	8	7
4	Vertex	5	6	10	9

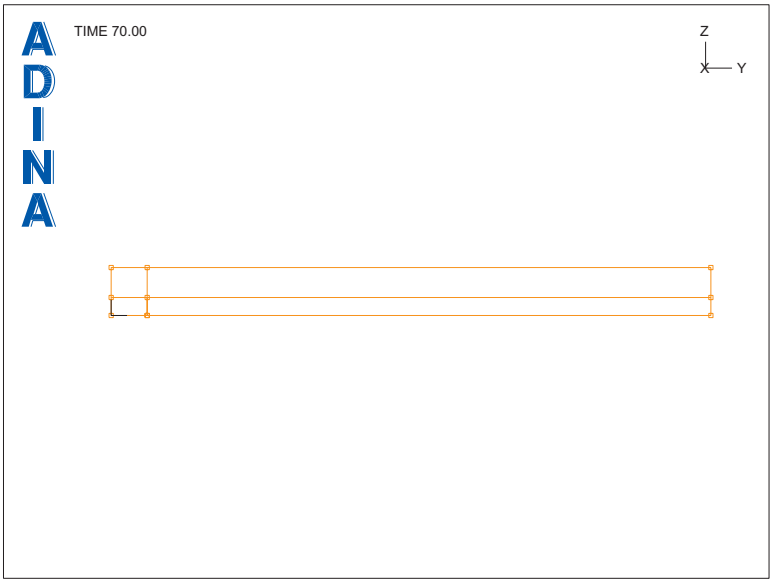
The graphics window should look something like the figure on the next page.

Defining material properties

Click the Manage Materials icon and click the Constant button. In the Define Material with Constant Properties dialog box, add material 1, set the Viscosity to 1.7E-4, the Density to 0.001and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the boundary conditions and loading

Wall boundary conditions: We will apply no-slip wall boundary conditions on the channel upper and lower walls. Click the Special Boundary Conditions icon , add special boundary condition 1 and verify that the Type is Wall. Enter 1, 5, 9 and 12 in the first four rows of the Line # table. Click OK to close the Special Boundary Condition dialog box.



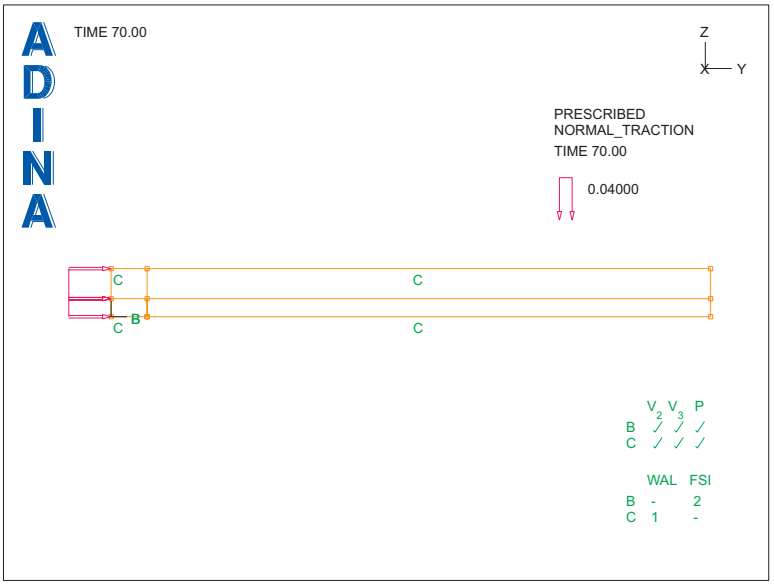
FSI boundary conditions: Click the Special Boundary Conditions icon  , add special boundary condition 2, set the Type to Fluid-Structure Interface and make sure that the Fluid-Structure Boundary # is 1. Enter 8 and 13 in the first two rows of the Line # table. Click OK to close the Special Boundary Condition dialog box.

Loading: We will apply a normal-traction at the channel inlet. Click the Apply Load icon  , set the Load Type to Normal Traction and click the Define... button to the right of the Load Number field. In the Define Normal Traction dialog box, add Normal Traction 1, set the Magnitude to 1.0 and click OK. In the Apply Usual Boundary Conditions/Loads dialog box, in the first two rows of the table, set the Line # to 6 and 11 respectively, then click OK.

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

Defining leader-follower relationships

To keep good mesh quality, we want point 2 to move along the channel wall following point 5. Choose Meshing→ALE Mesh Constraints→Leader-Follower. In the first row of the table, set the Label # to 1, the Leader Point # to 5, the Follower Point # to 2 and click OK.



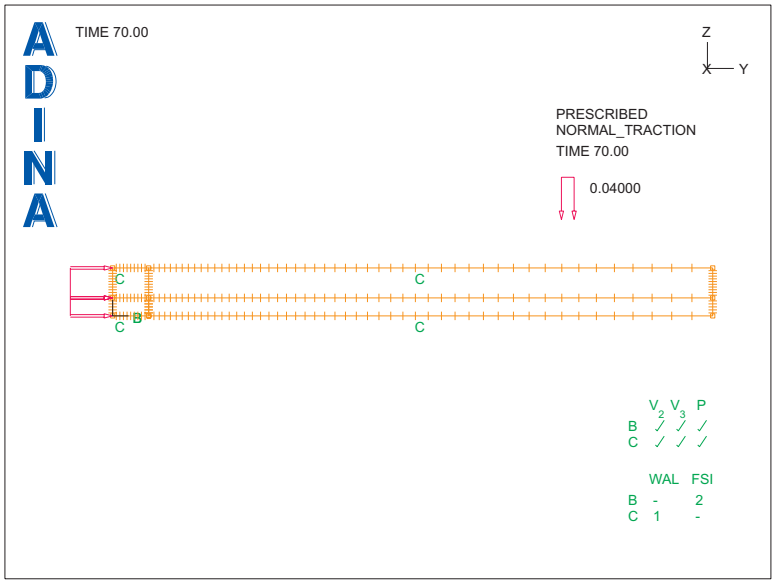
Defining the elements

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

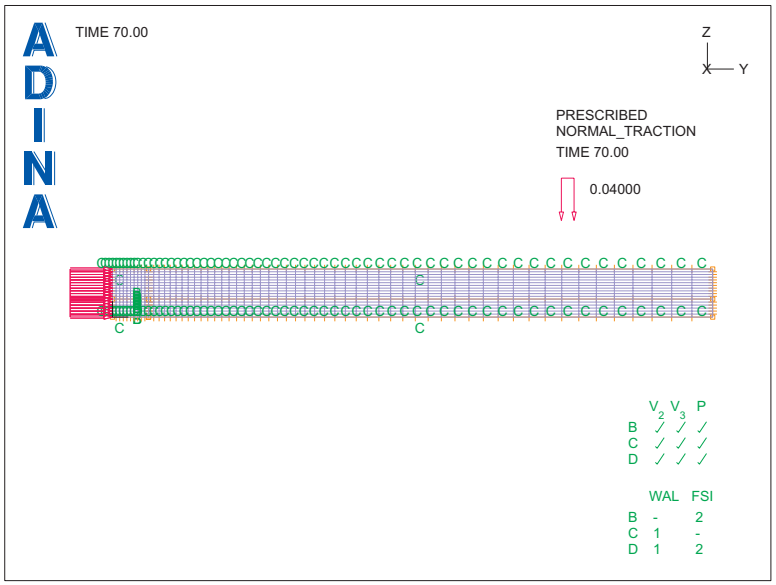
Subdivision data: We will set the subdivisions of the surfaces explicitly. Click the Subdivide Surfaces icon, set the subdivision data for each surface as shown in the following table and click OK.

Surface Number	Number of Subdivisions, u	Number of Subdivisions, v	Length Ratio of Element Edges, u	Length Ratio of Element Edges, v
1	50	11	4	1
2	10	11	1	1
3	50	6	4	1
4	10	6	1	1

The graphics window should look something like the top figure on the next page.



Element generation: Click the Mesh Surfaces icon , enter 1, 2, 3, 4 in the first four rows of the table and click OK. The graphics window should look something like this:



Notice that there are six fluid elements along each structural boundary, but only five solid elements in the structure. Also the fluid elements are 4-node elements and the solid elements are 9-node elements.

Generating the ADINA-CFD data file, saving the ADINA-IN database

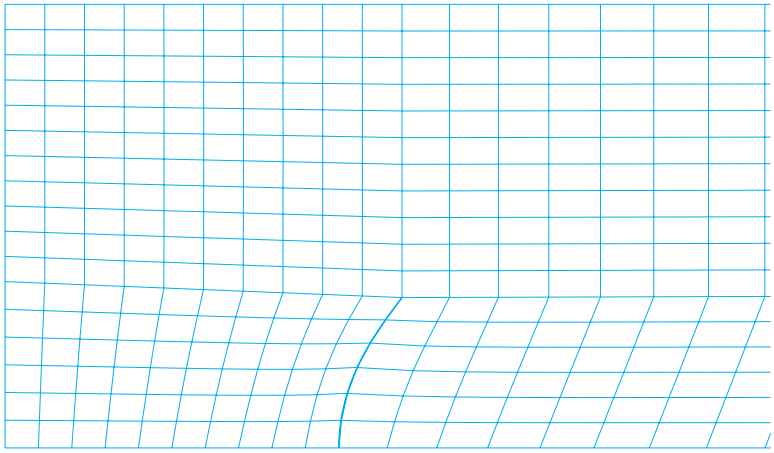
Click the Data File/Solution icon  , set the file name to prob18_f, make sure that the Run Solution button is unchecked and click Save. Save the database file to file prob18_f.

Running ADINA-FSI

Choose Solution→Run ADINA-FSI, click the Start button, select file prob18_f, then hold down the Ctrl key and select file prob18_a. The File name field should display both file names in quotes. Then click Start. When ADINA-FSI finishes, 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 prob18_f. Then click the Open icon  and open porthole file prob18_a.

Examining the solution

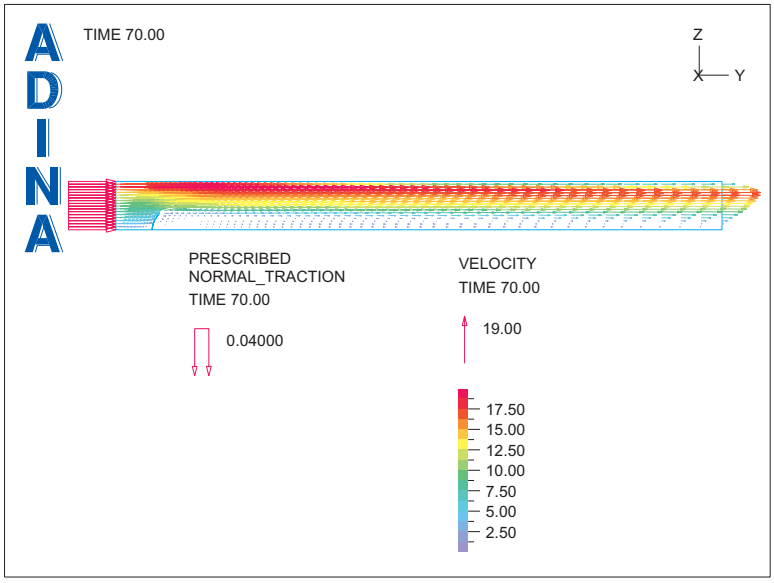
Mesh motion: Use the Zoom icon  and the mouse to enlarge the left end of the model. The graphics window should look something like this:



Use the icons that change the solution time (Previous Solution icon  , Next Solution icon  , Fast Rewind icon  , Fast Forward icon  , First Solution icon  and Last

Solution icon) to study in detail how the mesh moves. Notice that the vertical element boundaries in the fluid above the structure remain vertical. They remain vertical because we defined a leader-follower relationship between geometry points 5 and 2. When you are finished using these icons, click the Last Solution icon to display the last solution, then click the Unzoom All icon.

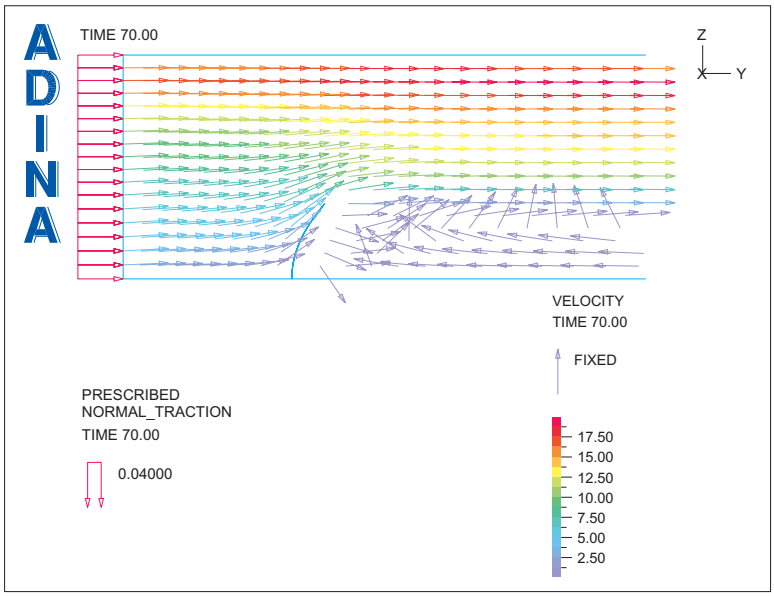
Velocity vectors: Click the Model Outline icon, then click the Load Plot icon to display the loads. Now click the Quick Vector Plot icon. The AUI shows the velocities in the fluid and the stresses in the structure. To remove the stresses in the structure, click the Modify Vector Plot icon, verify that the Vector Quantity is Stress, click the Delete button and click Yes to answer the prompt, then click OK to close the dialog box. Use the Pick icon and the mouse to rearrange the graphics until the graphics window looks something like this:



Use the icons that change the solution time to study in detail how the velocities change as the load is increased. For small loads (at time 1.0, for example), the recirculation region is short and is entirely contained in the fluid model. As the load is increased, the recirculation region becomes longer. When you are finished using these icons, click the Last Solution icon to display the last solution.

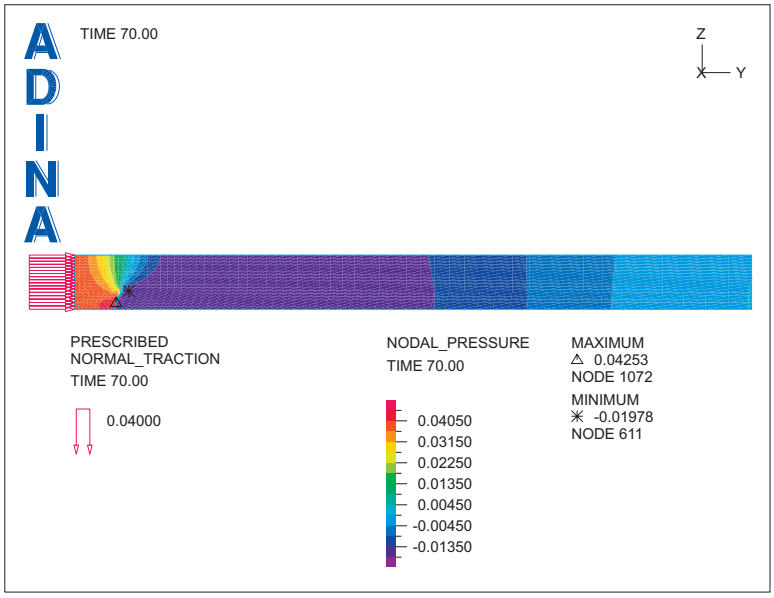
It is easier to visualize the recirculation region if the vectors are longer. Click the Pick icon , then pick one of the vectors. The AUI draws a diamond-shaped box around the vector. Enlarge the vectors by holding down the Ctrl key and dragging the mouse diagonally upwards and to the right. You can also shorten the vectors by dragging the mouse diagonally downwards and to the left.

We can also make all of the velocity vectors the same length. Click the Modify Vector Plot icon , click the Rendering… button, set the Vector Length to All Same Length and click OK twice to close both dialog boxes. Use the Mesh Zoom icon and the mouse to enlarge the left end of the model. The graphics window should look something like this:



Click the Refit icon to show the whole model.

Pressure field: Now we will show the pressure field. Click the Clear Vector Plot icon , then click the Quick Band Plot icon . The AUI shows the pressures in the fluid and the effective stresses in the structure. To remove the stresses in the structure, click the Modify Band Plot icon , verify that the Band Plot Variable is (Stress: EFFECTIVE_STRESS), click the Delete button and click Yes to answer the prompt, then click OK to close the dialog box. Use the Pick icon and the mouse to rearrange the graphics until the graphics window looks something like this figure on the next page.



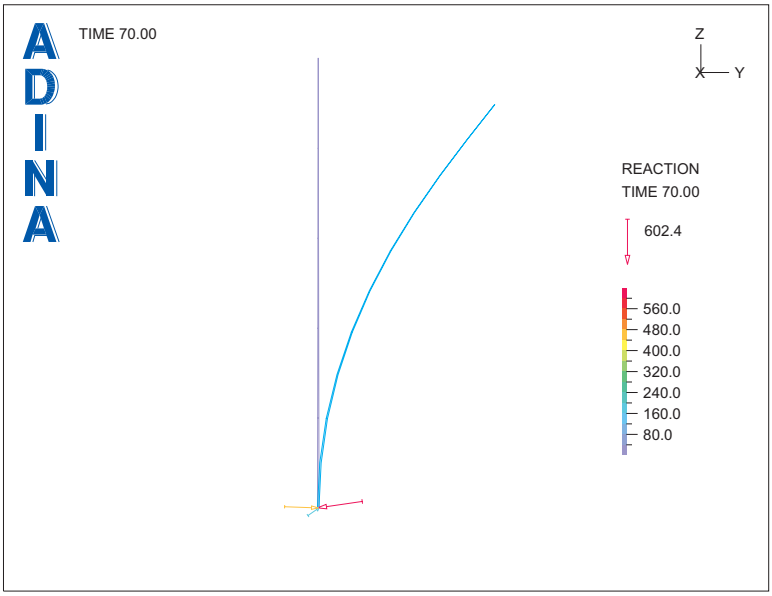
Use the icons that change the solution time to study in detail how the pressure field changes as the load is increased. When you are finished using these icons, click the Last Solution icon to display the last solution.

Plotting the structural motion

To display only the ADINA (solid) model, click the Clear icon, then, in the Model Tree, expand the Zone entry, right click on '1. ADINA' and choose Display.

We would like to compare the deformed mesh to the original mesh. Click the Show Original Mesh icon. Use the Pick icon and the mouse to shrink the plot so that the original mesh is entirely displayed.

Now we will show the reactions. Choose Display→Reaction Plot→Create, verify that the Reaction Quantity is REACTION and click OK. The graphics window should look something like the figure on the next page.



Let's list the maximum displacements. For the listing commands, we must first select the finite element model. Choose ADINA Structures from the FE Model (not the Program Module!) drop-down list. Then choose List→Extreme Values→Zone, set Variable 1 to (Displacement: Y-DISPLACEMENT), Variable 2 to (Displacement:Z-DISPLACEMENT) and click Apply. The maximum y displacement should be 5.87995 at node 1 and the maximum z displacement should be –1.54817 at node 1. Click Close to close the dialog box.

Exiting the AUI: Choose File→Exit to exit the AUI (you can discard all changes).

This page intentionally left blank.