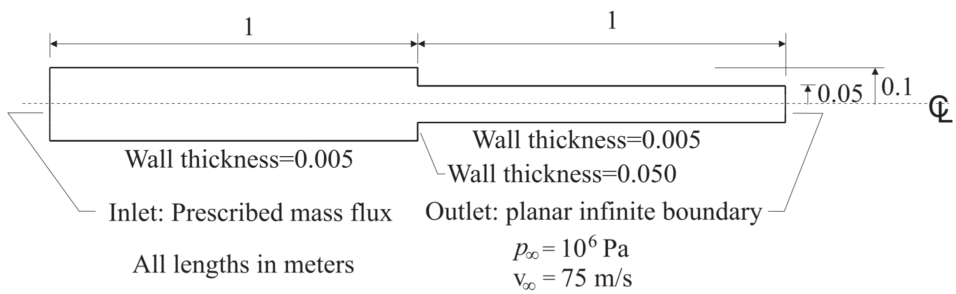
Problem description

It is desired to analyze the flow and structural response within the following pipe constriction:



The fluid material properties are $\rho = 1000 \text{ kg/m}^3$, $\kappa = 2.1\times10^9 \text{ Pa}$, and the solid material properties are $E = 2.07\times10^{11} \text{ Pa}$, $\nu = 0.3$, $\rho = 7800 \text{ kg/m}^3$.

This problem could of course be solved with ADINA-FSI, but it is more efficient to solve it using the subsonic potential-based fluid formulation of ADINA Structures.

First we determine the steady-state flow, then we determine the unsteady flow due to a sinusoidal variation of the prescribed mass flux. For the steady-state flow, it is necessary to prescribe a mass flux at the inlet that is equal to the mass flux at the outlet. The mass flux at the outlet is $\rho v A$ where, using the prescribed pressure at infinity, $\rho = 1000.476$ kg/m^3 at the outlet. Hence the prescribed mass flux per unit area at the inlet for steady state is 18758.929 kg/(s-m^2).

Once we determine the steady-state flow, we then determine the unsteady flow and structural motions due to a sinusoidal variation of the prescribed mass flux by $\pm10\%$ with a frequency of 5000 Hz. The time step size used for the unsteady flow is 10^{-5} sec , corresponding to 20 time steps per cycle.

Note that the wave propagation in the fluid is not greatly affected by the compliance of the pipe walls, because the pipe walls are quite stiff. Therefore the a potential-interface of type infinite boundary elements can be used to model the outlet boundary condition.

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

- Defining axisymmetric shell elements.
- Setting the solution start time to be less than zero.
- Using the subsonic potential-based formulation.

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.

The data for the time function is stored in a separate file `prob30_tf.txt`. You need to copy the file `prob30_tf.txt` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA Structures. Choose Edit→Memory Usage and make sure that the ADINA/AUI memory is at least 80 MB.

The memory allocation is required to perform the animations at the end of this example.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 30: Analysis of fluid-structure interaction within a pipe constriction" and click OK.

Analysis type: Set the Analysis Type drop-down list to Dynamics-Implicit. Notice that both the steady-state solution and the transient solution are obtained in the dynamic analysis.

Master degrees of freedom: Choose Control→Degrees of Freedom, uncheck the X-Translation, Y-Rotation and Z-Rotation buttons and click OK. You need to leave the X-Rotation button checked because the axisymmetric shell elements use the X-Rotation degree of freedom.

Equilibrium iteration tolerances: We will change the convergence tolerances used during equilibrium iterations. Choose Control→Solution Process, click the Tolerances... button, set the Energy Tolerance to 1E-7 and click OK twice to close both dialog boxes.

Solution start time: This run consists of two parts. The first part determines the steady-state response. In the first part, we use one long time step of 99.9999 and then 10 short time steps of 1E-5 to verify that steady-state is reached. In the second part, we use 500 short time steps of 1E-5 in the transient solution.

We set up the problem so that time 0 corresponds to the beginning of the second part of the run. That means that the first part of the run has solution times less than zero. The solution start time is –100.0. Choose Control→Solution Process, set the Solution Start Time to –100.0 and click OK.

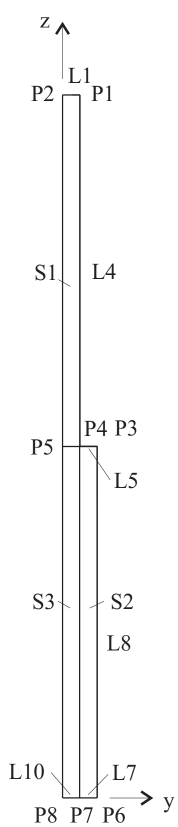
Time steps: Choose Control→Time Step, edit the table to contain the following data and click OK.

Number of Steps	Magnitude
1	99.9999
510	1E-5

Time function: Choose Control→Time Function, clear the table, import file `prob30_tf.txt` and click OK. `Prob30_tf.txt` contains a ramp function to time –0.0001, then a constant function to time 0.0, then a superposed sinusoidal time function with frequency of 5 kHz.

Defining model geometry

The following figure shows the key geometry used in defining this model.



The model is defined in the y-z plane because the 2D potential-based elements and the axisymmetric shell elements must be defined in the y-z plane.

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

Point #	X2	X3
1	0.05	2
2	0	2
3	0.1	1
4	0.05	1
5	0	1
6	0.1	0
7	0.05	0
8	0	0

Click the Point Labels icon to display the point numbers.

Surfaces: Click the Define Surfaces icon and enter the following surfaces, then click OK.

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

Axisymmetric shell thicknesses: Choose Geometry→Lines→Thickness, set the thickness for lines 4 and 8 to 0.005, the thickness for line 5 to 0.050 and click OK.

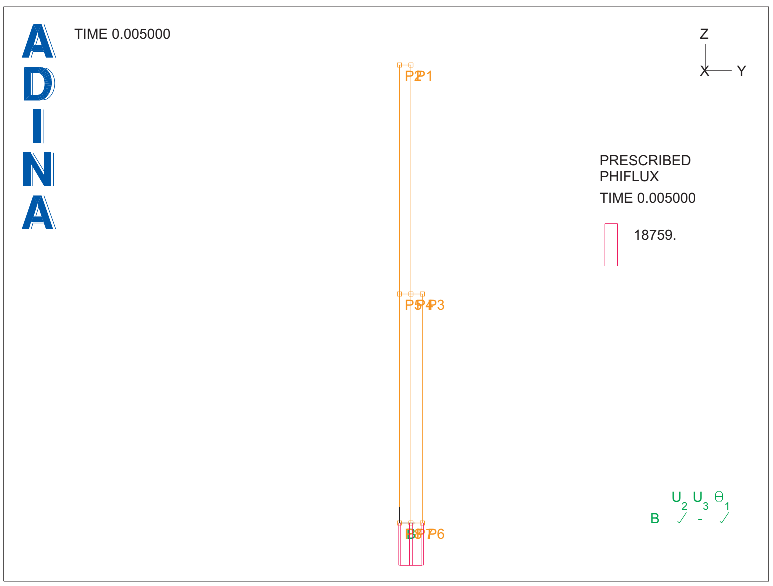
Specifying boundary conditions, loads and the material

Fixity for pipe: Click the Apply Fixity icon and click the Define... button. In the Define Fixity dialog box, add fixity name ZT, check the Z-Translation button and click OK. In the Apply Fixity dialog box, set the "Fixity" field to ZT, make sure that the "Apply to" field is set to Point, enter 6 in the first row of the table and click OK.

Infinite boundary condition: Choose Model→Boundary Conditions→Potential Interface, add potential interface number 1, set the Type to Fluid – Infinite Region, verify that the Boundary Type is Planar, set the Pressure at Infinity to 1E6 and set the Velocity at Infinity to 75. Enter 1 in the first row of the table, then click OK.

Mass-flux loads: Click the Apply Load icon, set the Load Type to Distributed Fluid Potential Flux, and click the Define... button to the right of the Load Number field. In the Define Distributed Fluid Potential Flux dialog box, add number 1, set the Magnitude to 18758.929 and click OK. In the Apply Load dialog box, make sure that the "Apply To" field is set to Line, then, in the first two rows of the table, set the Line # to 7 and 10. 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:



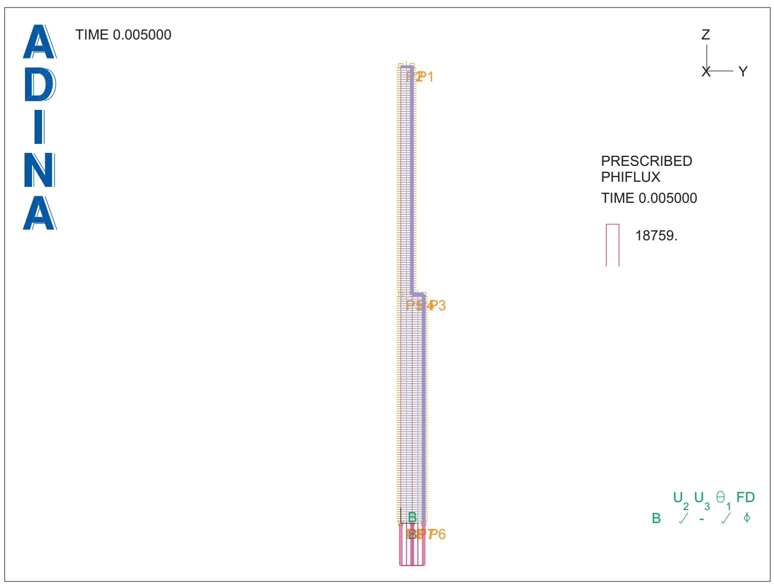
Materials: Click the Manage Materials icon and click the Potential-based Fluid button. In the Define Potential-based Fluid Material dialog box, add material 1, set the Bulk Modulus to 2.1E9, the Density to 1000 and click OK. Now click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 2, set the Young's Modulus to 2.07E11, the Poisson's ratio to 0.3, the Density to 7800 and click OK. Click Close to close the Manage Material Definitions dialog box.

Meshing

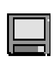
Element groups: Click the Element Groups icon, add element group number 1, set the Type to 2-D Fluid, set the Formulation to Subsonic Potential-Based Element and click Save. Now add element group number 2, set the Type to Isobeam, set the Element Sub-Type to Axisymmetric Shell, set the Default Material to 2 and click OK.

Subdivision data: Click the Subdivide Surfaces icon, select surface number 1, set the Number of Subdivisions in the u- and v-directions to 2 and 100 respectively, enter 2 and 3 in the first two rows of the table and click OK.

Meshing: Click the Mesh Surfaces icon , set the Type to 2-D Fluid, set the Nodes per Element to 4, enter 1, 2, 3 in the first three rows of the table and click OK. Now click the Mesh Lines icon , click the Nodal Options tab, in the Nodal Coincidence Checking box, set the Check field to "All Generated Nodes", click the Basic tab, enter 4, 5, 8 in the first three rows of the table and click OK. The graphics window should look something like this:



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

Click the Save icon  and save the database to file `prob30`. Click the Data File/Solution icon , set the file name to `prob30`, make sure that the Run Solution button is checked and click Save.

Notice that the AUI writes "Model completion information for potential-based elements" into the Log Window. This is because the AUI completes the potential-based model when it generates the ADINA Structures data file. In this case the AUI generates 202 fluid-structure-interface elements. The AUI also notes that there are 204 uncovered element sides in element group 1. These sides correspond to the inlet line and symmetry line.

When ADINA Structures 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 `prob30`.

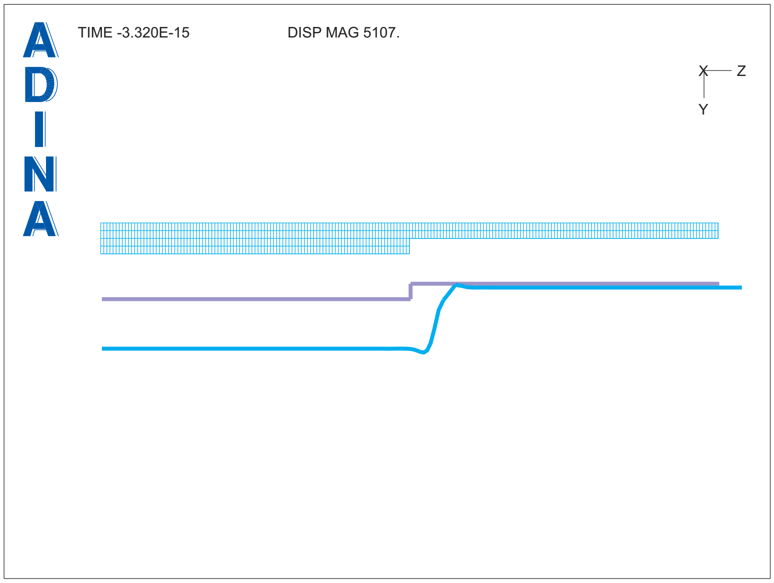
Plotting the steady-state solution

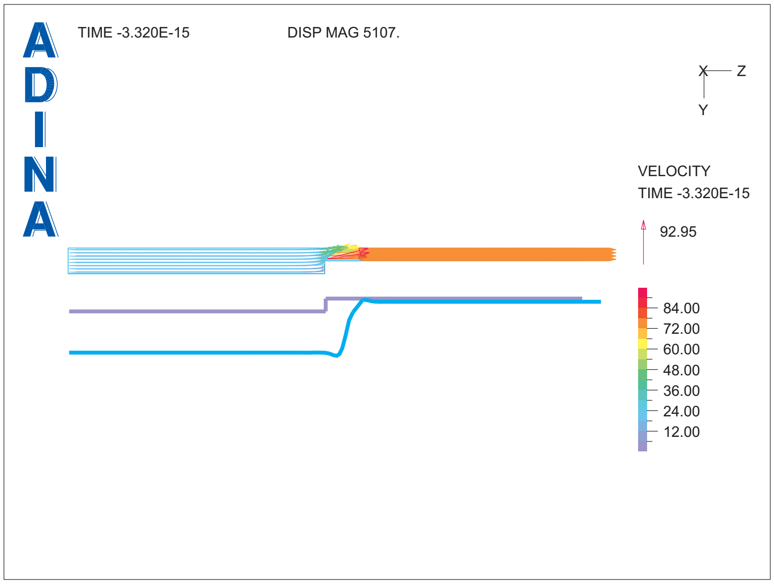
Choose Definitions→Response, make sure that the Response Name is set to DEFAULT, set the Solution Time to 0 and click OK. Then click the Clear icon and the Mesh Plot icon . To set the view, click the Modify Mesh Plot icon , click the View… button, set the Angle of Rotation to –90 and click OK. To suppress the thick lines used to draw the fluid interface elements, click the Element Depiction… button, click the "Contact, etc." tab, set the Contact Surface Line Width to 0.0 and click OK twice to close both dialog boxes. Move the mesh plot to the center of the graphics window with the mouse. Click the Save Mesh Plot Style icon so that successive mesh plots are displayed with the rotated view.

Now click the Scale Displacements icon . The displacements of the axisymmetric shell elements are plotted. Since there are no displacements of the nodes within the fluid elements, these nodes remain at their original positions.

To obtain a nicer picture, we need to plot the two groups separately. Click the Clear icon , click the Display Zone icon , set the Zone Name to EG1 and click Apply. Then set the Zone Name to EG2 and click OK. Use the mouse to separate the two mesh plots. There are also two "TIME …" texts and two sets of axes that have the same locations. Use the mouse to separate them. Then highlight the mesh plot of the axisymmetric shell elements and click the Scale Displacements icon and the Show Original Mesh icon . Use the mouse to rearrange the plots and delete extra text and axes until the graphics window looks something like the top figure on the next page.

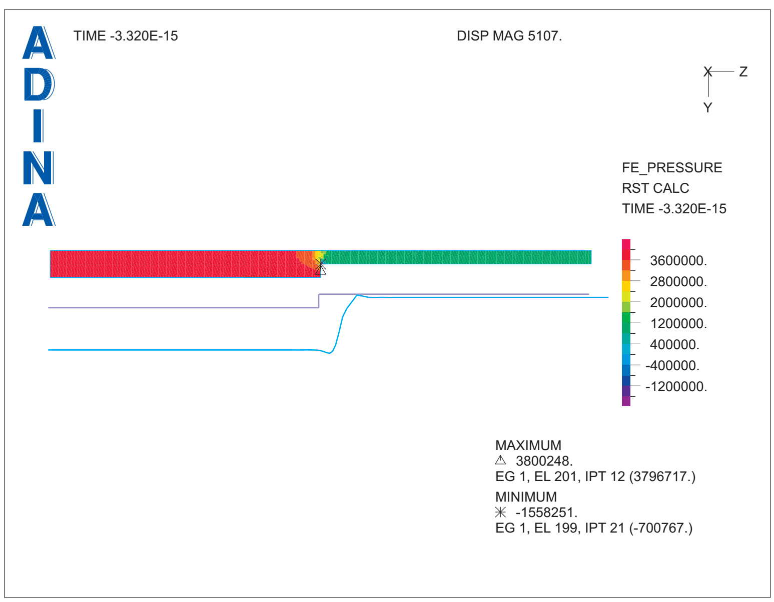
To display velocity vectors in the fluid, highlight the fluid mesh and click the Model Outline icon . Then click the Create Vector Plot icon , set the Mesh Plot Name to MESHPLOT00001 and click OK. (Note: MESHPLOT00001 is the fluid mesh plot and MESHPLOT00002 is the axisymmetric shell mesh plot, because we created the fluid mesh plot before the axisymmetric shell mesh plot.) Use the mouse to rearrange the plots until the graphics window looks something like the bottom figure on the next page.


TIME -3.320E-15
DISP MAG 5107.
X
Z
Y


TIME -3.320E-15
DISP MAG 5107.
X
Z
Y
VELOCITY
TIME -3.320E-15
92.95
84.00
72.00
60.00
48.00
36.00
24.00
12.00

Make sure that the fluid mesh is highlighted. Then click the Previous Solution icon a few times to verify that the vector plot is unchanged for times less than 0.0. Then click the Next Solution icon a few times until the solution time is 0.0 again. Click the Clear Vector Plot icon to remove the velocity vectors.

To display pressures in the fluid, highlight the fluid mesh and click the Create Band Plot icon , set the Band Plot Variable to (Stress: FE_PRESSURE) and click OK. Use the mouse to rearrange the plots until the graphics window looks something like this:

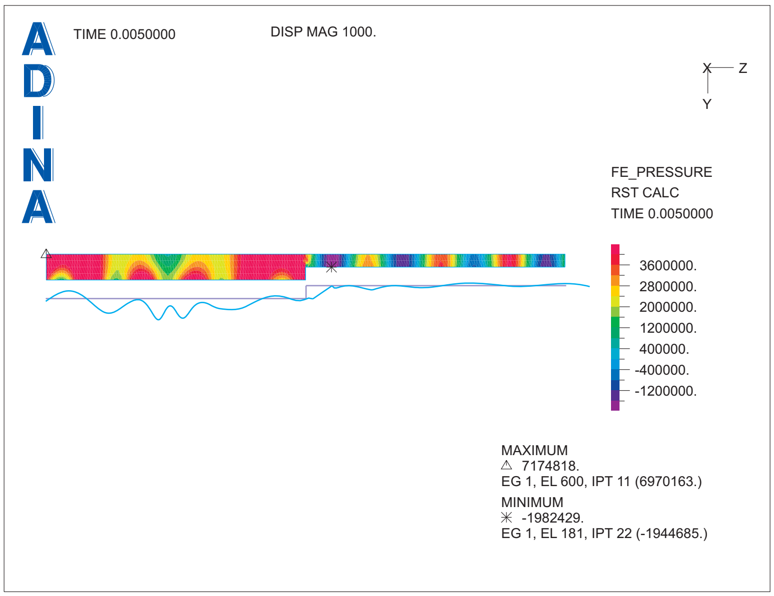


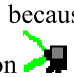
The pressure is higher at the inlet because the velocity is lower. In fact, the inlet and outlet pressures and velocities satisfy the Bernoulli equation $\left(\frac{p}{\rho}+\frac{v^2}{2}\right)_{inlet}=\left(\frac{p}{\rho}+\frac{v^2}{2}\right)_{outlet}$.

Animating the transient solution

Let's look at the last computed solution. Highlight the fluid mesh and click the Last Solution icon , then highlight the axisymmetric shell mesh and click the Last Solution icon . To reduce the magnification factor for the axisymmetric shell displacements, click the Modify Mesh Plot icon , click the Model Depiction… button, set the Magnification Factor to 1000 and click OK twice to close both dialog boxes.

The graphics window should look something like this:



To animate the solution, choose Display→Movie Shoot→Load Step, set the Start Time to 0.0 and click OK. Pressure waves move from the inlet to the outlet. Eventually a standing wave pattern forms in the wide area of the pipe, because of reflections off of the constriction. However the waves always travel towards the outlet in the narrow area of the pipe, because of the infinite boundary condition. When the movie is finished, click the Animate icon  to display the animation. When you are finished viewing the animation, click the Refresh icon  to restore the graphics window.

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