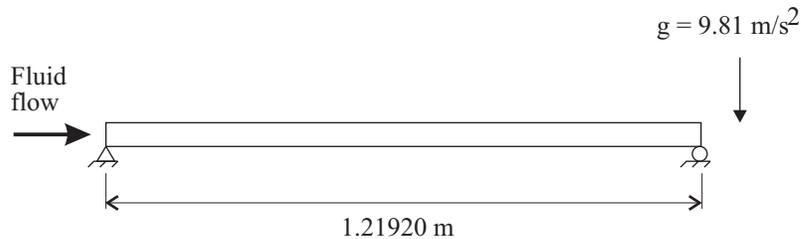


### Problem description

The pipe shown below is filled with water flowing at constant velocity. It is desired to determine the pipe deformation due to gravity and to the fluid flow.



$$E_{\text{pipe}} = 2.07 \times 10^{11} \text{ N/m}^2 \quad \mu = 100 \times \mu_{\text{water}} = 100 \times 0.001 \text{ kg/m-s} = 0.1 \text{ kg/m-s}$$

$$v_{\text{pipe}} = 0.29 \quad \rho_{\text{water}} = 890 \text{ kg/m}^3$$

$$\rho_{\text{pipe}} = 7800 \text{ kg/m}^3$$

$$\text{Pipe outer diameter} = 5.0800 \times 10^{-2} \text{ m}$$

$$\text{Pipe inner diameter} = 4.9022 \times 10^{-2} \text{ m}$$

$$\text{Pipe midsurface radius} = 2.49555 \times 10^{-2} \text{ m}$$

$$\text{Pipe wall thickness} = 8.89 \times 10^{-4} \text{ m}$$

**Pipe model:** The walls of the pipe are modeled with 9-node shell elements. Small displacements are assumed in the pipe model.

**Fluid model:** The fluid is modeled with 8-node FCBI elements. A uniform velocity is applied to the fluid at the left end of the pipe.

On the right end of the pipe, the transverse velocities are set to zero. (If this condition is omitted, the gravity applied to the fluid will cause fluid flow at the pipe outlet.)

Large structural displacements are assumed in the fluid model.

Slip fluid-structure interaction boundary conditions are used on the pipe wall. This assumption is consistent with the comparison solution and allows us to use a coarse fluid model (only two elements across the pipe diameter). The result is that the fluid velocity is constant in the pipe. As a consequence, the viscosity does not affect the solution, and therefore the viscosity of the water is scaled by a factor of 100, so that the model converges.

We must set up two finite element models, an ADINA Structures model for the pipe and an ADINA CFD model for the fluid. In this case we set up the models separately in two separate AUI databases, but note that you could also have set up both models in the same AUI database.

### *Problem 24: Deformation of a pipe due to internal fluid flow*

---

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

- Defining a 3-D FSI model.
- Defining mass-proportional loads.
- Defining fluid-structure boundary conditions with slip.
- Adding comparison solutions to a graph

We assume that you have worked through problems 1 to 23, 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.

#### **Invoking the AUI and choosing the finite element program**

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

#### **Defining the ADINA Structures model**

##### **Defining model control data**

*Problem heading:* Choose Control→Heading, enter “Problem 24: Deformation of a pipe due to internal fluid flow, pipe model” into the heading field and click OK.

*Interaction with fluid flow:* Set the Multiphysics Coupling drop-down list to 'with CFD'.

##### **Defining the model geometry**

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



*Geometry point:* Click the Define Points icon , enter the following point into the table and click OK:

Point #	X1	X2	X3
1	0	0.0249555	0

*Geometry lines:* Click the Define Lines icon , add line 1, set the Type to Revolved, set the Initial Point to 1, the Angle of Rotation to 180, make sure that the Axis is set to X and click Save. Add line 2, make sure that the Type is set to Revolved, set the Initial Point to 2, the Angle of Rotation to 180, make sure that the Axis is set to X and click OK.

*Geometry surfaces:* Click the Define Surfaces icon , add surface 1, set the Type to Extruded, set the Initial Line to 1, the Vector to (1.2192, 0, 0), enter 2 in the first row of the table and click OK.

Choose Geometry→Surfaces→Thickness, set the Thickness for both surfaces to 0.000889 and click OK.

When you click the Wire Frame icon , Point Labels icon  and Surface/Face Labels icon , the graphics window should look something like the top figure on the next page.

### Defining boundary conditions

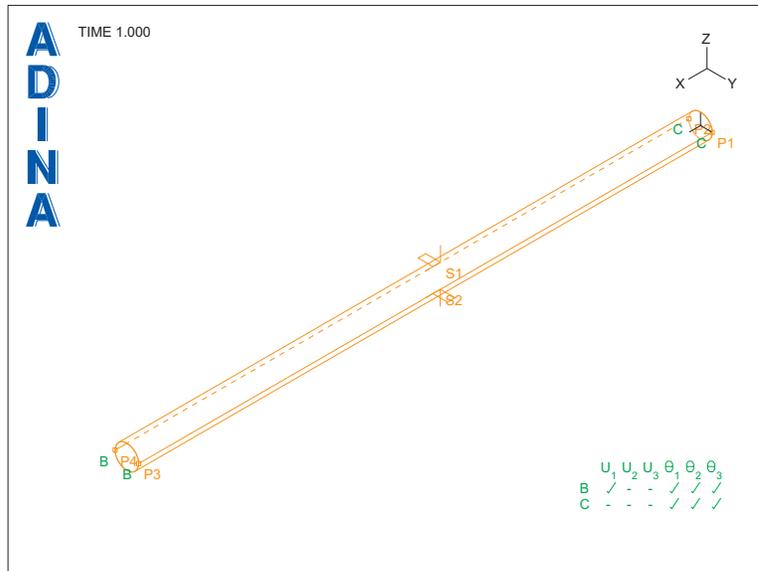
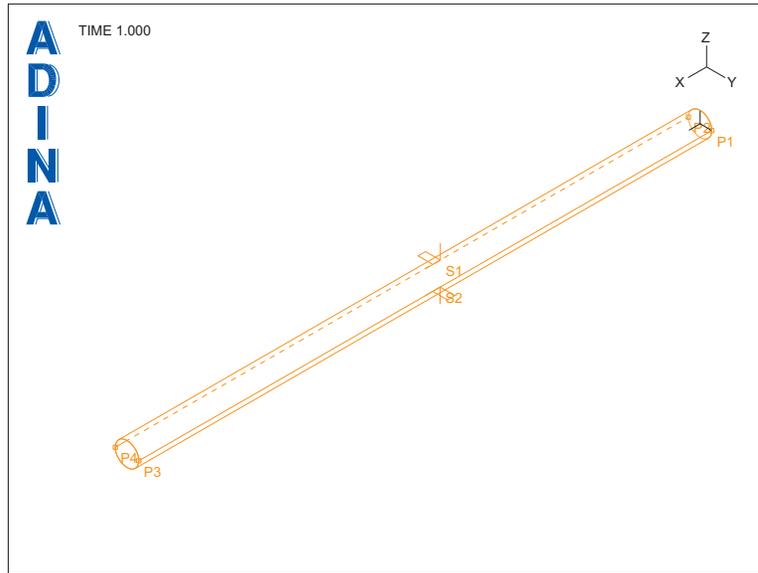
*Supports:* Click the Apply Fixity icon , and click the Define... button. Add boundary condition PIN, check the X-Translation, Y-Translation and Z-Translation fields and click Save. Then add boundary condition ROLLER, check the Y-Translation and Z-Translation fields and click OK.

In the Apply Fixity dialog box, set the “Fixity” to PIN, make sure that the “Apply to” field is set to Point, enter 1, 2 in the first two rows of the table, then click Apply. Set the “Fixity” to ROLLER, enter 3, 4 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 the bottom figure on the next page.

*Fluid-structure boundaries:* Surfaces 1 and 2 are fluid-structure boundaries. Choose Model→Boundary Conditions→FSI Boundary, add FSI boundary number 1, set the “Apply to” field to Surfaces, enter 1, 2 in the first two rows of the table and click OK.

Problem 24: Deformation of a pipe due to internal fluid flow



### Defining the load

The pipe is subjected to gravity loading due to its own weight. Click the Apply Load icon , set the Load Type to Mass Proportional and click the Define... button to the right of the Load Number field. In the Define Mass-Proportional Loading dialog box, add load number 1, set the Magnitude to 9.81, make sure that the Direction is set to (0, 0, -1) and click OK. In the Apply Load dialog box, in the first row of the table, set the Time Function to 1 and click OK.

### 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 2.07E11, the Poisson's ratio to 0.29, the Density to 7800 and click OK. Click Close to close the Manage Material Definitions dialog box.

### Defining the finite elements and nodes

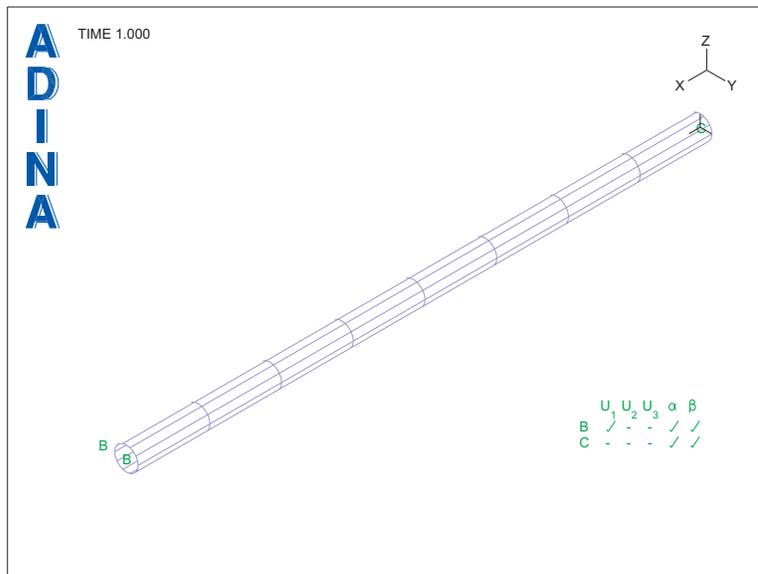
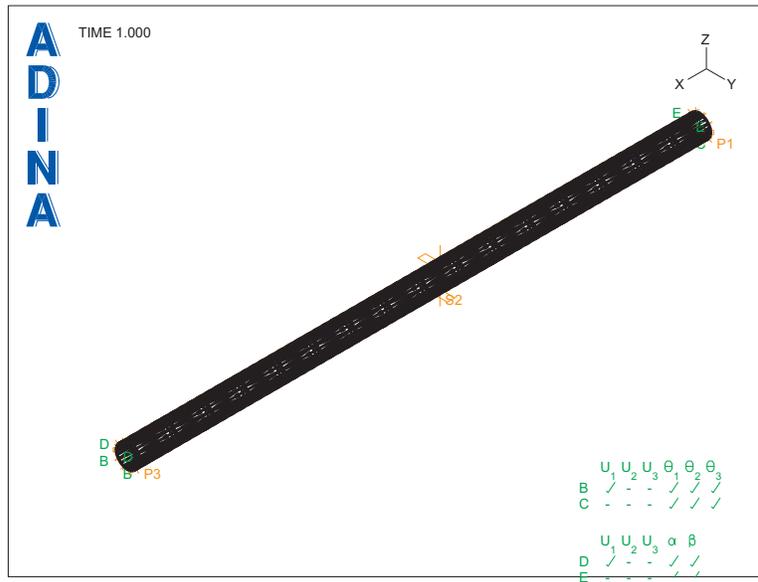
*Element groups:* Click the Element Groups icon , add element group 1, set the Type to Shell and click OK.

*Subdivision data:* Click the Subdivide Surfaces icon , select surface 1, set the "Number of Subdivisions in u-direction" to 5, the "Number of Subdivisions in v-direction" to 8, enter 2 in the first row of the table and click OK.

*Element generation:* Click the Mesh Surfaces icon , set the "Nodes per Element" to 9, enter 1 and 2 in the first two rows of the table and click OK. The graphics window should look something like the top figure on the next page.

The thick lines indicate the fluid-structure boundaries. To hide the thick lines, click the Show Fluid Structure Boundary icon . Now click the Show Geometry icon  (to hide the geometry) and the Hidden Surfaces Removed icon . The graphics window should look something like the bottom figure on the next page.

Problem 24: Deformation of a pipe due to internal fluid flow



### Creating the ADINA Structures data file, saving the database file

Click the Data File/Solution icon , set the file name to prob24\_a, uncheck the Run Solution button and click Save. Now click the Save icon  and save the database to file prob24\_a.

### Defining the ADINA CFD model

Click the New icon  to create a new AUI database. Then set the Program Module drop-down list to ADINA CFD.

### Defining the model control data

*Heading:* Choose Control→Heading, enter “Problem 24: Deformation of a pipe due to internal fluid flow, fluid model” and click OK.

*Interaction with structure:* Set the Multiphysics Coupling drop-down list to 'with Structures'. Then click the Coupling Options icon , set the FSI Solution Coupling to Direct, set the Maximum Number of Fluid-Structure Iterations to 50 and click OK.

*Flow assumptions:* Choose Model→Flow Assumptions, make sure that the Flow Dimension is set to 3D, uncheck the “Includes Heat Transfer” button, and click OK.

*Number of iterations and iteration tolerances:* Choose Control→Solution Process, click the Iteration Method... button, set the Maximum Number of Iterations to 30 and click OK to close the Iteration Method dialog box. Now 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.

*Time steps and time function:* We will apply only the gravity load in the first step and then increase the flow velocity to 200 m/s in 20 equal steps. Choose Control→Time Step, and, in the first row of the table, set the Number of Steps to 21, then click OK. Now choose Control→Time Function, edit the table for time function 1 as follows and click OK.

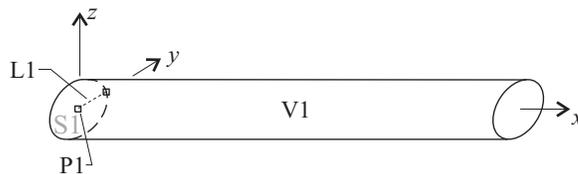
Time	Value
0	0
1	0
21	200

*Problem 24: Deformation of a pipe due to internal fluid flow*

---

**Defining the model geometry**

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



Inlet:  $v_x$  prescribed

Outlet:  $v_y = v_z = 0$

*Geometry point:* Click the Define Points icon , enter the following point into the table and click OK.

Point #	X1	X2	X3
1	0	0	0

*Geometry line:* Click the Define Lines icon , add line 1, set the Type to Extruded, set the Initial Point to 1, the components of the Vector to 0, 0.0249555, 0 and click OK.

*Geometry surface:* Click the Define Surfaces icon , add surface 1, set the Type to Revolved, set the Initial Line to 1, the Angle of Rotation to 360, make sure that the Axis is set to X, uncheck the Check Coincidence button and click OK.

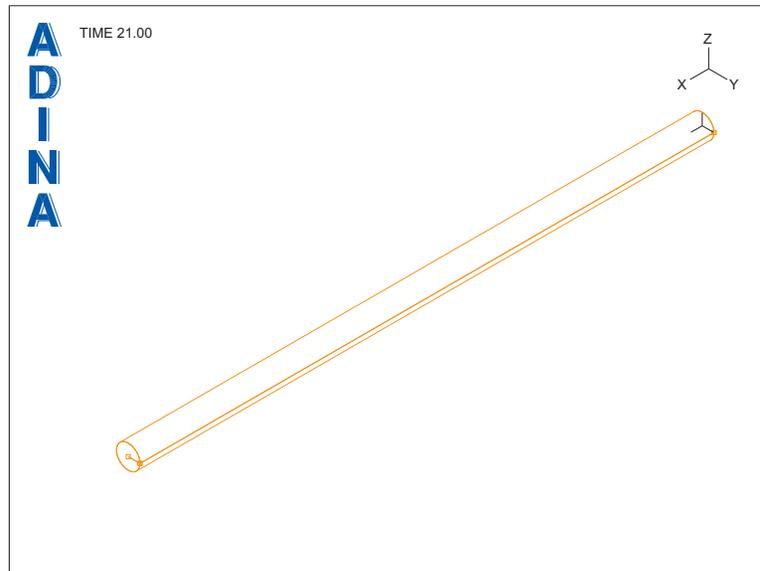
*Geometry volume:* Click the Define Volumes icon , add volume 1, set the Type to Extruded, set the Initial Surface to 1, the components of the Vector to 1.2192, 0, 0 and click OK.

The graphics window should look something like the figure on the next page (note: in this view you are viewing the outlet of the pipe).

**Defining boundary conditions**

*FSI boundary conditions:* We need to define the fluid-structure boundary conditions

corresponding to the pipe wall (surface 4). Click the Special Boundary Conditions icon , add Condition Number 1, set the Type to Fluid Structure Interface, make sure that the Fluid-Structure Boundary # is set to 1 and set the Slip Condition to Yes. Then enter 4 in the first row and column of the table and click OK.



*Inlet velocity:* We need to define and apply the prescribed velocity at the inlet (surface 1).

Click the Apply Load icon , make sure 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 number 1, set the X Prescribed Value of Velocity to 1 and click OK. In the Apply Usual Boundary Conditions/Loads dialog box, set the “Apply to” field to Surface, and, in the first row of the table, set the Surface # to 1. Click OK to close the dialog box.

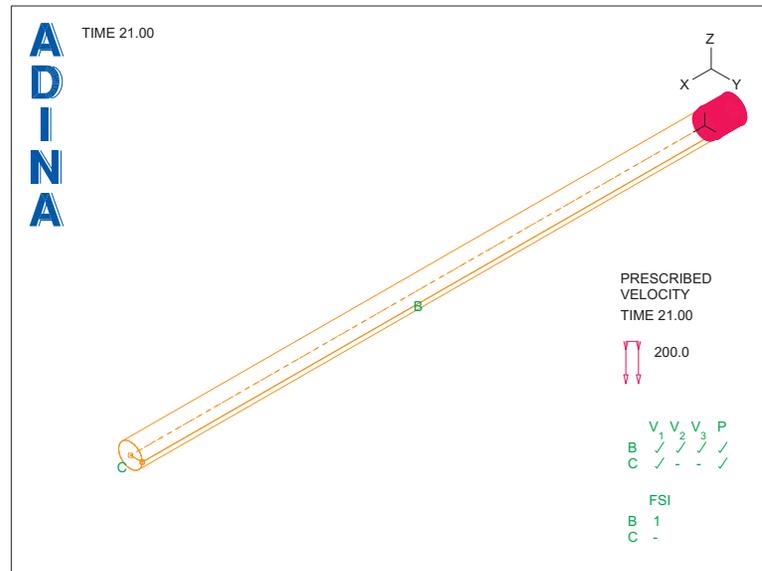
*Outlet velocity:* We need to set the y and z velocities to be zero at the outlet (surface 5). Click the Apply Fixity icon  and click the Define... button. In the Define Zero Values dialog box, add zero values name YZ, check the Y-Velocity and Z-Velocity fields, then click OK. In the Apply Zero Values dialog box, set “Zero Values” to YZ, set the “Apply to” field to Face/Surface, enter 5 in the first row and column of the table and click OK.

Click the Boundary Plot icon , the Load Plot icon  and the Wire Frame icon  to show the FSI boundary conditions and the prescribed velocities. The graphics window should look something like the figure on the next page.

### Defining the 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 0.1, the Density to 890, the Z Acceleration due to Gravity to -9.81 and click OK. Click Close to close the Manage Material Definitions dialog box.

*Problem 24: Deformation of a pipe due to internal fluid flow*

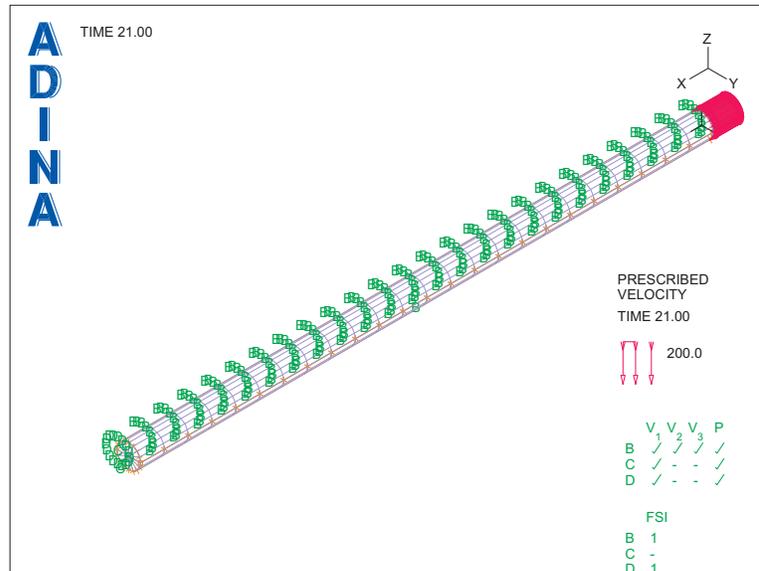


**Defining the finite elements and nodes**

*Element group:* Click the Element Groups icon , add element group 1, set the Type to 3-D Fluid, and click OK.

*Subdivision data:* We deliberately use a different number of elements along the pipe in the fluid model than in the structural model, to show that a different number of elements can be used along the fluid-structure boundaries in each model. Click the Subdivide Volumes icon , set the number of subdivisions in the u, v and w directions to 16, 24 and 1 respectively, then click OK.

*Element generation:* Click the Hidden Surfaces Removed icon , then click the Mesh Volumes 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.



### Creating the ADINA CFD data file, saving the database file

Click the Data File/Solution icon , set the file name to `prob24_f`, uncheck the Run Solution button and click Save. Now click the Save icon  and save the database to file `prob24_f`.

### Running ADINA-FSI

Choose Solution→Run ADINA-FSI, click the Start... button, select file `prob24_f`, then hold down the Ctrl key and select file `prob24_a`. The File name field should display both file names in quotes. Then click Start.

The run stops in time step 20 with the message

```
*** ERROR *** CODE ADF3069
  Unsuccessful during FSI iteration
```

This is OK; as we will see, the run stops because the critical velocity was exceeded. Close all open dialog boxes.

## Post-processing the ADINA CFD model

In the following, we demonstrate post-processing of the ADINA CFD model only. You can also post-process the ADINA Structures model to check the model and to determine the deformations due to the fluid velocity.

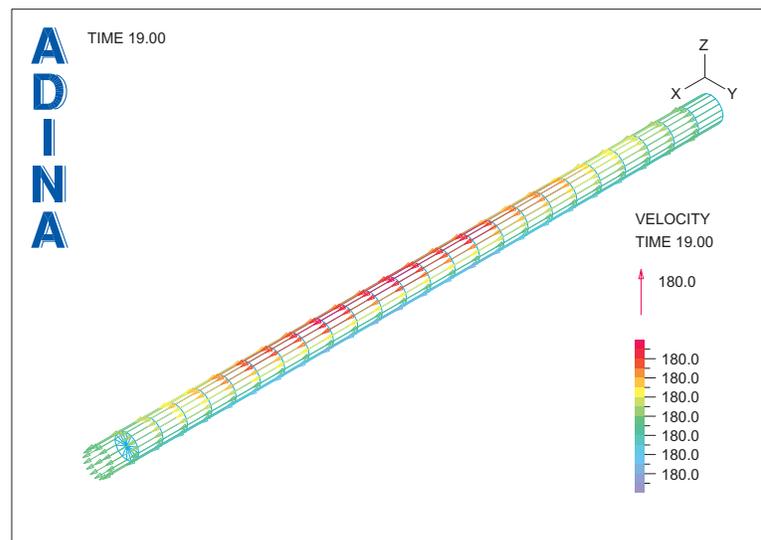
Set the Program Module drop-down list to Post-Processing (you can discard all changes).

Click the Open icon  and open porthole file prob24\_f.

Choose List→Info→Response to see which solutions were computed. There are 20 load steps from times 0.0 to 19.0. Notice that there is no solution for times 20.0 and 21.0, because the model did not converge for time step 20.0. Click Close to close the dialog box.

## Checking the model solution

Click the Quick Vector Plot icon  to plot the velocity vectors. The graphics window should look something like this:



The velocities are nearly constant, showing that the fluid-structure boundary condition with slip is working properly.

**Plotting the deformation as a function of velocity**

Click the Clear icon  to clear the graphics window.

*Velocity:* We can extract the input velocity as the prescribed velocity at one of the nodes on the left end of the pipe. Node 425 is one of the nodes on the left end of the pipe. Choose Definitions→Model Point→Node, add point LEFTEND, set the node number to 425 and click Save.

*Deformation at the mid-span of the pipe:* We need a node point definition at the mid-span of the pipe. Node 193 is at the mid-span. Add point MIDSPAN, set the node number to 193 and click OK. We also need a resultant for the displacement to change its sign. Choose Definitions→Variable→Resultant, add resultant DISPLACEMENT, define it as -<Z-DISPLACEMENT> and click OK.

*Creating the graph plot:* Choose Graph→Response Curve (Model Point). For X, set the Variable to (Prescribed Load: X-PRESCRIBED\_VELOCITY) and set the Model Point to LEFTEND. For Y, set the Variable to (User Defined: DISPLACEMENT) and set the Model Point to MIDSPAN. Then click OK.

*Adding a comparison solution:* A good approximation to the theoretical solution is

$$w = \frac{w_{static}}{1 - \left(\frac{v}{v_c}\right)^2}$$

where

$$w_{static} = \frac{5L^4}{384EI} \frac{(mg)_{total}}{L}, \quad v_c = \frac{\pi}{L} \sqrt{\frac{EI}{\rho A}}$$

Note that  $A = \frac{\pi d^2}{4}$  where  $d$  is the internal diameter of the pipe cross-section, and  $\rho$  is the density of the water. This approximate solution is found in Harris, *Shock and Vibration Handbook, 3<sup>rd</sup> edition*, McGraw-Hill Book Company, 1988: Chapter 29, Section “Internal Flow in Pipes”. (The approximation given in the reference is obtained using only the first mode of the “exact” solution.)

For our case, the static deformation is 8.6889E-5 m and the critical velocity  $v_c$  is 188.48 m/s.

In order to plot this solution, we need to define constants and a resultant. Choose Definitions→Variable→Constant, add W\_STATIC, set the Value to 8.6889E-5 and click Save. Then add VC, set the Value to 188.48 and click OK. Now choose Definitions→Variable→Resultant, add W\_VELOCITY, enter the expression

*Problem 24: Deformation of a pipe due to internal fluid flow*

---

$W\_STATIC / (1.0 - (<X-PRESCRIBED\_VELOCITY> / VC) ** 2)$

and click OK.

Now choose Graph→Response Curve (Model Point). For X, set the Variable to (Prescribed Load:X-PRESCRIBED\_VELOCITY) and set the Model Point to LEFTEND. For Y, set the Variable to (User Defined:W\_VELOCITY) and set the Model Point to LEFTEND. Then set the Plot Name to PREVIOUS and click OK.

Let's show the theoretical critical velocity as a separate curve. Choose Graph→Define User Data, add user data VC, enter 188.48, 0 in the first row of the table, enter 188.48, 0.001 in the second row of the table and click OK. Now choose Graph→Plot User Data, make sure that the Data Name is set to VC, set the Plot Name to PREVIOUS and click OK.

We will now change the curve legends and symbols. Choose Graph→Modify, set the "Action" to "Modify the Curve Depiction", click the P button, highlight the green curve (the one marked with circles) and click the ... button to the right of the Curve Depiction field. Click the Legend tab, and, in the Legend Attributes box, set the Type to Custom, enter "ADINA-FSI" in the Legend table and click OK. Click Apply to see the new curve legend.

For the second curve, set the "Action" to "Modify the Curve Depiction", click the P button, highlight the red curve (the one marked with triangles) and click the ... button to the right of the Curve Depiction field. Uncheck the "Display Curve Symbol" button, and, in the Legend Attributes box, set the Type to Custom, enter "Comparison solution" in the Legend table and click OK. Click Apply to see the new curve legend.

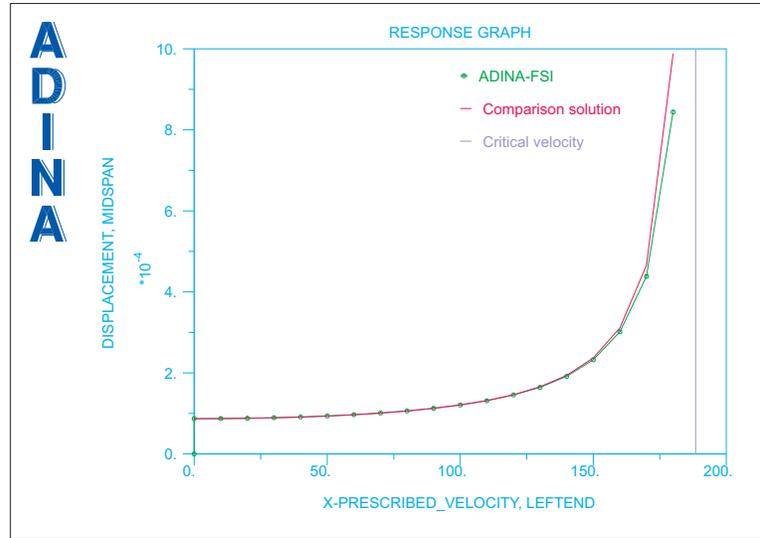
In a similar way, remove the curve symbols and change the curve legend to "Critical velocity" for the third curve. Click OK twice to close both dialog boxes.

The graphics window should look something like the figure on the next page. It is also possible to change the graph title and axes, as was shown in problem 2.

To see the numerical values displayed in the curves, choose Graph→List. In the first listing, at time 1.90000E+01, the midspan displacement should be 8.41999E-04 (m). Click Close to close the dialog box.

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

Problem 24: Deformation of a pipe due to internal fluid flow



*Problem 24: Deformation of a pipe due to internal fluid flow*

---

This page intentionally left blank.