Problem description
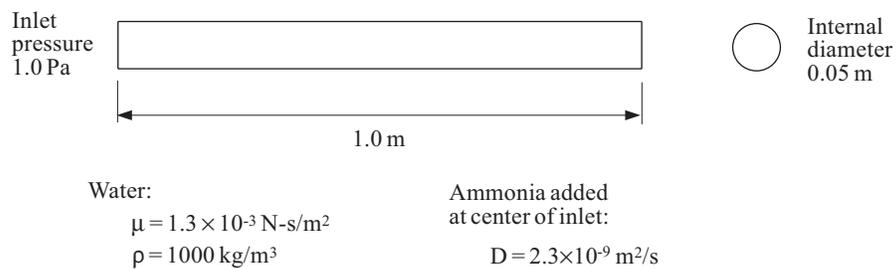
We determine the fluid flow and mass transfer within a pipe subjected to an inlet pressure as shown:

Inlet
pressure
1.0 Pa

Internal
diameter
0.05 m

1.0 m

Water:
$$\mu = 1.3 \times 10^{-3}\ \text{N-s/m}^2$$
$$\rho = 1000\ \text{kg/m}^3$$

Ammonia added
at center of inlet:
$$D = 2.3 \times 10^{-9}\ \text{m}^2/\text{s}$$

This problem could be solved using a 2D analysis, but we choose to solve it using a 3D analysis as a demonstration.

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

• 3D fluid flow analysis
• Assignment of mass transfer properties
• Defining and using a mesh style
• Determining the volume flux through a pipe
• Viewing the solution using multiple cutting planes

We assume that you have worked through problems 1 to 18, or have equivalent experience with the ADINA System. Therefore we will not describe every user selection or button press.

We will work on this problem in two steps: 1) set up the fluid model, then 2) add mass transfer.

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.

Fluid model

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 19: Fluid flow in a pipe" and click OK.

Flow assumptions: Choose Model→Flow Assumptions, verify that the Flow Dimension field is set to 3D, uncheck the "Includes Heat Transfer" button and click OK.

Defining the model geometry

Here is a diagram showing the key geometry used in defining the ADINA-CFD model.
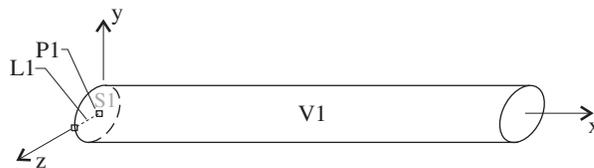
It is, of course, possible to use ADINA-M to define the geometry, but we use the native AUI geometry in this problem.

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

Point #	X1	X2	X3
1	0	0	0

Now 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, 0.0, 0.025 and click OK.

Now 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.

Now click the Define Volumes icon , add volume 1, set the Type to Extruded, set the Initial Surface to 1, make sure that the components of the Vector are set to 1.0, 0.0, 0.0 and click OK.

The graphics window should look something like this (note: in this view you are viewing the outlet of the pipe):
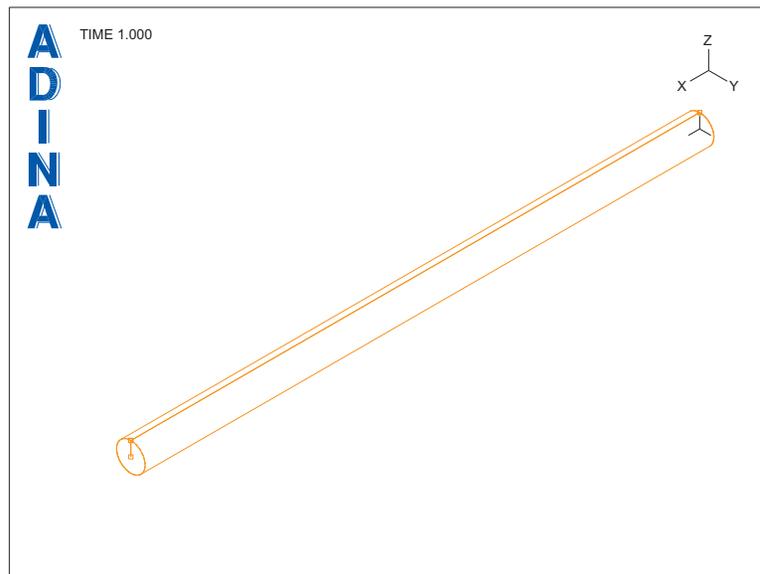
Defining the boundary conditions and loading

Wall boundary conditions: We need to apply no-slip wall boundary conditions on the pipe walls. Use the Query icon ? and the mouse to determine the surface number of the pipe wall (it should be 4). Now click the Special Boundary Conditions icon SBC, add special boundary condition 1 and verify that the Type is Wall. Add surface number 4 to the first row and column of the table, then click OK to close the Define Special Boundary Condition dialog box.

Loading: We will apply a normal-traction at the channel inlet (surface 1). 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, set the "Apply to" field to Surface, then set the Surface # to 1 in the first row of the table. Click OK to close the dialog box.

Note: by default, the normal traction is zero at the channel outlet, so no action is required at the channel outlet.

Time steps and time function: In this model, we will apply the normal traction in 2 steps. Choose Control→Time Step, set the number of steps to 2 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
2	1.0

When you click the Wire Frame icon [icon], then the Boundary Plot icon [icon] and the Load Plot icon [icon], you should be able to use the mouse to rotate and resize the mesh plot until the graphics window looks something like this:
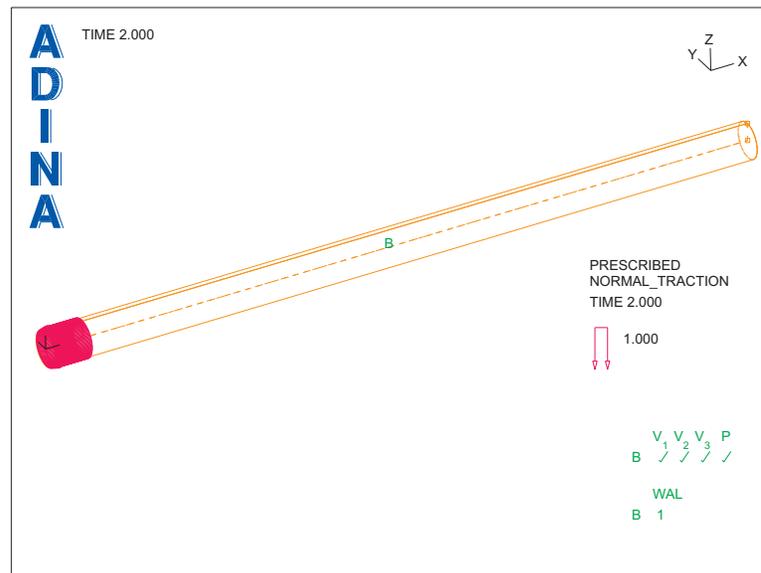
Defining material properties

Click the Manage Materials icon **M** and click the Laminar button. In the Define Laminar Material dialog box, add material 1, set the Viscosity to 0.0013, the Density to 1000.0 and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the elements

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

Subdivision data: In this mesh, we will assign the number of subdivisions in the u, v and w directions of the volume. In this case, the u direction is the tangential direction, the v direction is the axial direction (extrusion direction) and the w direction is the radial direction.

Click the Subdivide Volumes icon and set the Number of Subdivisions in the u, v and w directions to 16, 8 and 6 respectively. Also set the "Length Ratio of Element Edges in w-direction" to 0.5. (We do this to make the element size smaller in the radial direction near the pipe wall.) Then click OK.

Element generation: Click the Mesh Volumes icon , enter 1 in the first row of the table and click OK. The graphics window should look something like this:
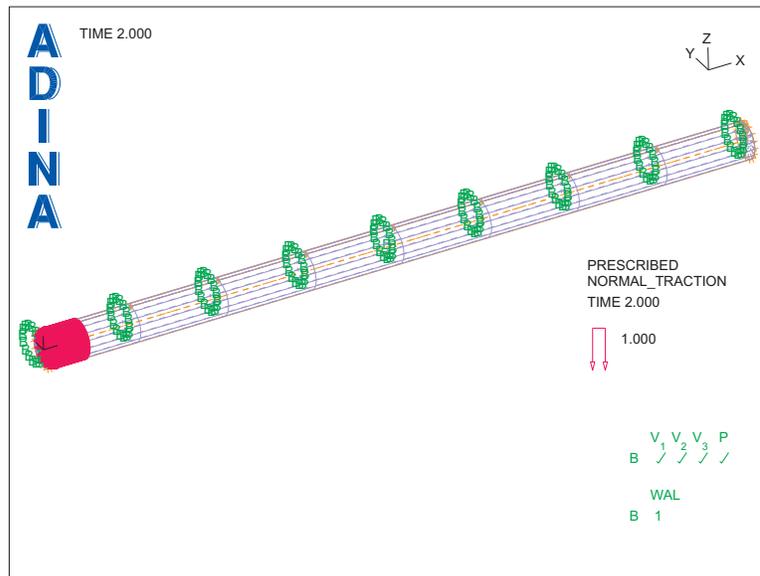
Notice that the AUI does not display element lines in the interior of the volume.

Use the Pick icon and the mouse to display the model as seen end-on from the channel inlet. The graphics window should look something like the figure on the next page. Here you can see that, in the radial direction, the element size gets smaller near the pipe wall.
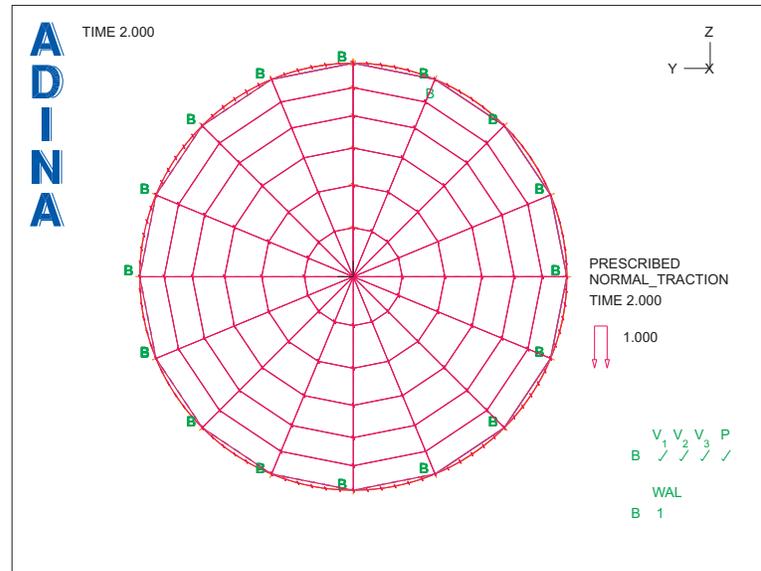
Generating the data file, running ADINA-CFD, loading the porthole file

Click the Save icon ▦ and save the database to file `prob19`. Click the Data File/Solution

icon ▤❗, set the file name to `prob19_1`, 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 `prob19_1`.

Examining the solution

We want to create plots of the results within the pipe. So we will define a cutting plane, then change the view and remove all of the cutplane interior lines.

Cutting plane: Click the Cut Surface icon 🔳. Set the Type to Cutting Plane, set "Defined by" to Y-Plane, uncheck the Display the Plane(s) button and click OK.

View and cutting plane interior lines: Click the XZ View icon 🔳 and the Model Outline

icon 🔳.

Save mesh plot defaults: We will use this mesh plot appearance for several plots. So we save the mesh plot defaults so that we don't have to repeat the above steps for each plot. Click the

Save Mesh Plot Style icon [S].

Velocities and pressures: Click the Quick Vector Plot icon [icon] to display the velocities. We would also like to show the pressure within the pipe. Use the mouse to move the mesh plot to the top of the graphics window. Then click the Mesh Plot icon [icon] and the Quick Band Plot icon [icon]. Use the mouse to delete any extra text and axes, and to resize the mesh plots and annotations, until the graphics window looks something like this:
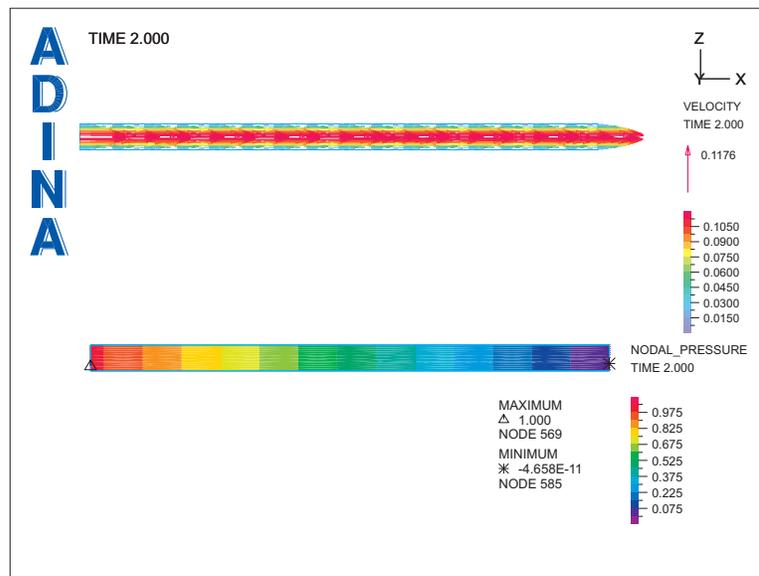

Total volume flux, average velocity, maximum velocity: Now we will determine the total volume flux, average velocity and maximum velocity of the flow. To do this, we cut the model with a plane perpendicular to the direction of flow, then integrate and search for the maximum velocity on the plane.

First we reset the mesh plot defaults. Click the Reset Mesh Plot Style icon [R].

Now click the Clear icon [CLEAR], then click the Mesh Plot icon [icon]. Click the Cut Surface icon [icon], set the Type to Cutting Plane, make sure that "Defined by" is set to X-Plane, set the Coordinate Value to 0.7 and click OK.

The graphics window should look something like this:
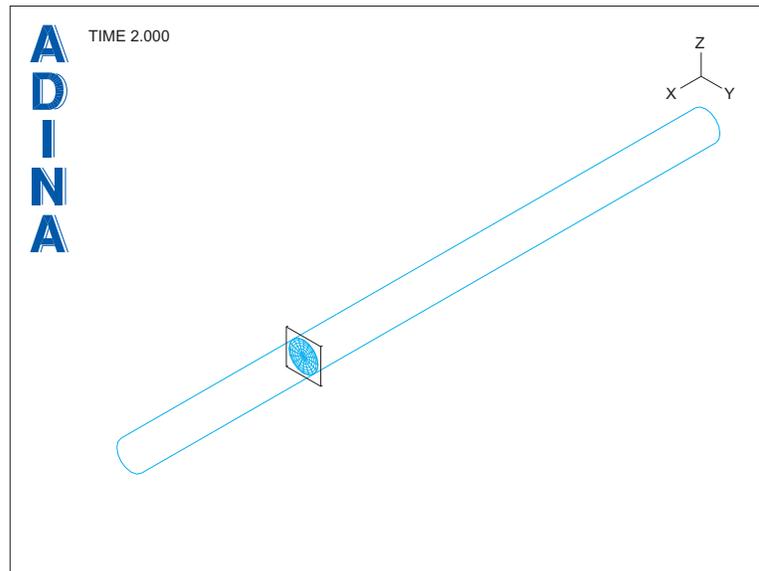
Total volume flux: To create a model point for determining the total volume flux over the cutting plane, choose Definitions→Model Point (Special) →Mesh Integration, add point name X_FLUX, verify that the Integration Type is Integral and click OK. Now choose List→ Value List→Model Point, verify that the point name is X_FLUX, set Variable 1 to (Flux: VOLUME_FLUX_SURFACE) and click Apply. The volume flux should be 1.09308E-4 (the analytical solution for laminar flow is 1.18E-4). Click Close to close the dialog box.

Average velocity: To create a model point for determining the average velocity over the cutting plane, choose Definitions→Model Point (Special) →Mesh Integration, add point name X_AVERAGE, set the Integration Type to Averaged and click OK. Now choose List→ Value List→Model Point, verify that the point name is X_AVERAGE, set Variable 1 to (Velocity:X-VELOCITY) and click Apply. The average velocity should be 5.72472E-2 (the analytical solution for laminar flow is 6.01E-2). Click Close to close the dialog box.

Maximum velocity: To create a model point for determining the maximum velocity over the cutting plane, choose Definitions→Model Point (Special) →Mesh Extreme, add point name X_MAX, verify that the Extreme Value Type is Absolute Maximum and click OK. Now choose List→Value List→Model Point, set the point name to X_MAX, set Variable 1 to (Velocity:X-VELOCITY) and click Apply. The maximum velocity should be 1.17643E-1 (the analytical solution for laminar flow is 1.202E-1). Click Close to close the dialog box.

Adding mass transfer

Set the Program Module drop-down list to ADINA CFD (you can discard all changes). Choose database file `prob19` from the recent file list near the bottom of the File menu.

Modifying the model

Heading: Choose Control→Heading, change the heading to "Problem 19: Mass transfer in a pipe", and click OK.

Mass transfer: Choose Model→Flow Assumptions, check the "Includes Mass Transfer" button, set the Total Number of Species to 1 and click OK.

Mass ratio loading: First we apply zero mass-ratio of the species at the inlet surface. Click the Apply Load icon , set the Load Type to Mass Ratio and click the Define... button to the right of the Load Number field. In the Define Mass Ratio dialog box, add mass ratio 1, make sure that the Mass Ratio is 0 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 and click Apply. (Do not close the dialog box yet.)

Now we apply unity mass-ratio of the species at the center of the inlet. In the Apply Usual Boundary Conditions/Loads dialog box, make sure that the Load Type is set to Mass Ratio and click the Define... button to the right of the Load Number field. In the Define Mass Ratio dialog box, add mass ratio 2, set the Mass Ratio to 1 and click OK. In the Apply Usual Boundary Conditions/Loads dialog box, set the Load Number to 2, set the "Apply to" field to Point and, in the first row of the table, set the Point # to 1. Click OK to close the dialog box.

Mass material: Choose Model→Mass Transfer→Mass Transfer Materials, add material 2 and verify that the Type is Constant. Set the Constant Diffusive Coeff. to 2.3E-9 and click OK.

We also must update the element group data to reference the mass material. Click the Element Groups icon , set the Mass Transfer Material to 2 and click OK.

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

First click the Save icon to save the database. Click the Data File/Solution icon , set the file name to `prob19_2`, 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 `prob19_2`.


Examining the solution

In this post-processing session, we examine how the mass ratio changes along the length of the pipe. Click the Cut Surface icon ![icon] and set the Type to Cutting Plane, verify that the "Defined by" field is set to X-Plane, set the Coordinate Value to 0.1, set the "Distance Between Planes" to 0.2 and click OK.

We now have five cutting planes evenly spaced along the pipe. Click the Model Outline icon ![icon] to remove the interior lines of the cut elements.

Now click the Create Band Plot icon ![icon], set the Band Plot Variable to (Fluid Variable: MASS_RATIO_1) and click OK. The graphics window should look something like this:
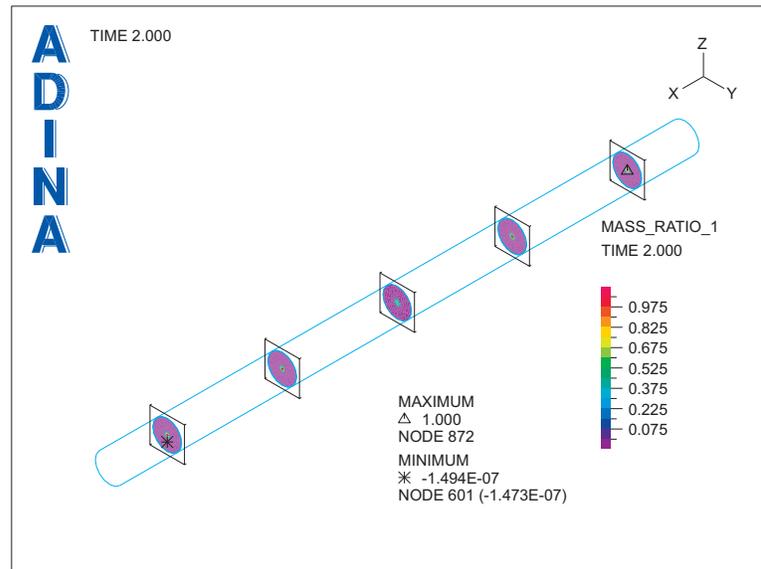
The plot shows that the high mass ratio remains at the center of the pipe (that is, the ammonia remains concentrated at the center of the pipe).

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