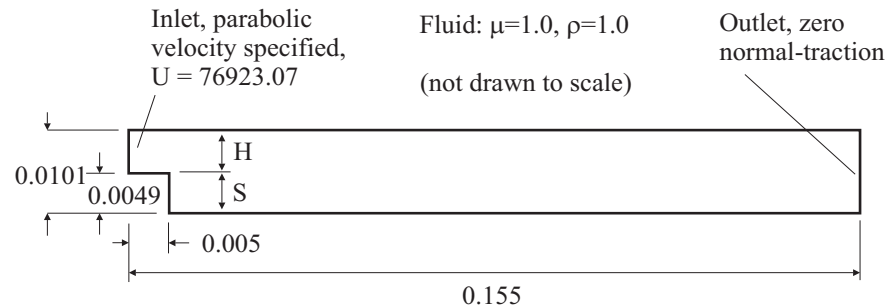


### Problem description

The step-wall diffuser has become a basic configuration for validation of CFD codes due to its simplicity and large amount of available numerical and experimental data. For a detailed compilation of results, see the following reference:

P. T. Williams and A. J. Baker, "Incompressible Computational Fluid Dynamics and the Continuity Constraint Method for the Three-Dimensional Navier-Stokes Equations", *Numerical Heat Transfer, Part B*, 29:137-273, 1996.

Here we determine the fluid flow in a step-wall diffuser using 4-node FCBI elements:



The flow is isothermal and the properties are chosen such that the Reynolds number is 800. The Reynolds number  $Re = U(2H)\rho / \mu$  is based on twice the upstream channel height,  $H$ , and the average velocity,  $U$ . The inlet fully-developed velocity profile is given by

$$u(z) = \frac{3}{2}U \left[ 1 - \left( \frac{z}{H/2} \right)^2 \right], \quad -H/2 < z < H/2$$

No-slip walls are used to model the channel.

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

- Specifying a parabolic velocity profile as a boundary condition

### 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.

*Problem 31: Analysis of a laminar step-wall diffuser*

---

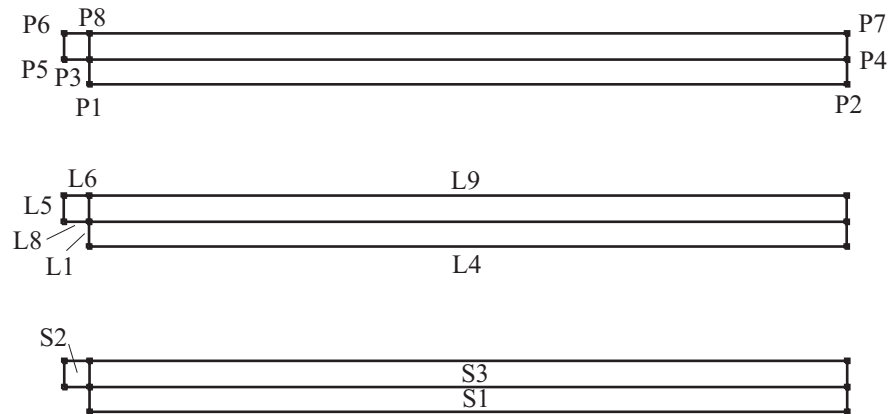
Much of the input for this problem is stored in file prob31\_1.in. You need to copy file prob31\_1.in from the folder samples\primer into a working directory or folder before beginning this analysis.

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

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

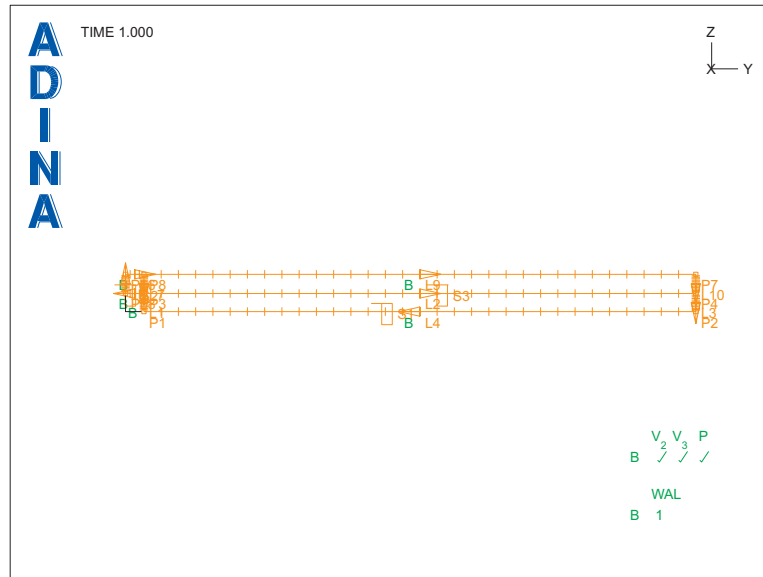
**Defining model control data, model geometry, boundary conditions, material properties, element group definitions and subdivision data**

The key geometry used to define this geometry is shown here:




We have placed all of the remaining model control data, time step definitions, geometry definitions, material definitions, boundary conditions, element group definitions and subdivision data in batch file prob31\_1.in. Choose File→Open Batch, navigate to the working directory or folder, select the file prob31\_1.in and click Open. The AUI processes the commands in the batch file.

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





### Defining and applying a parabolic velocity profile

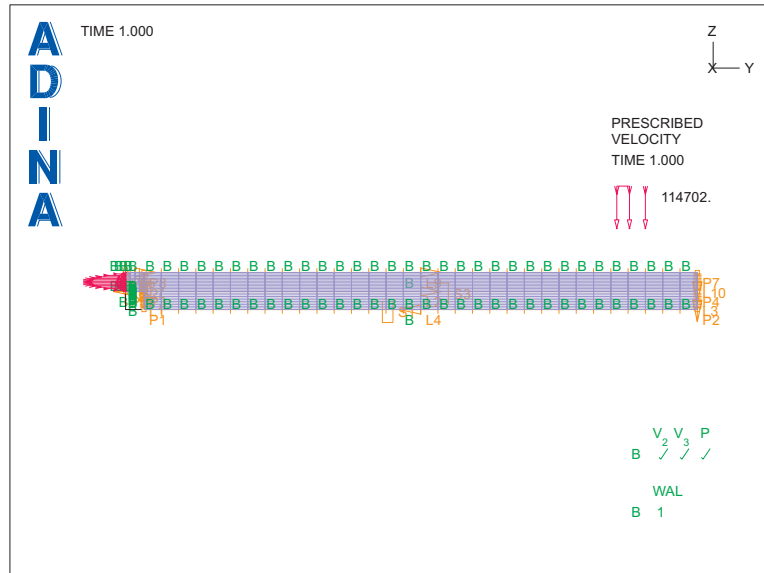
*Spatial-function:* We need to define a spatial-function corresponding to the parabolic velocity. Choose Geometry→Spatial Functions→Line and add Function Number 1. Set the Type to Quadratic and set “u = 0” to 0.0, “u=0.5” to 1.5 and “u=1” to 0.0. Click OK to close the dialog box.

*Velocity profile:* 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 1, set the Y velocity to 76923.07, set the Z velocity to 0.0 and click OK. In the Apply Usual Boundary Conditions/Loads dialog box, set the “Apply to” field to Line, then, in the first row of the table, set the Line # to 5 and the Spatial Function to 1. Click OK to close the dialog box.



### Meshing


Click the Mesh Surfaces icon , enter 1, 2, 3 in the first three rows of the table and click OK. When you click the Load Plot icon , the graphics window should look something like the figure on the next page.

**Problem 31: Analysis of a laminar step-wall diffuser**







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

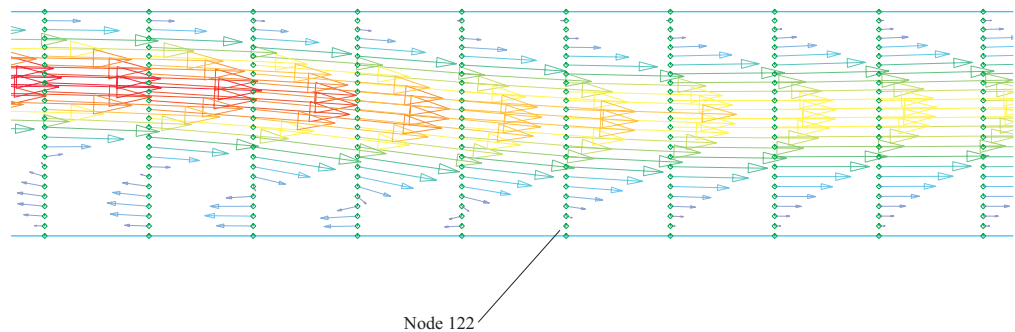
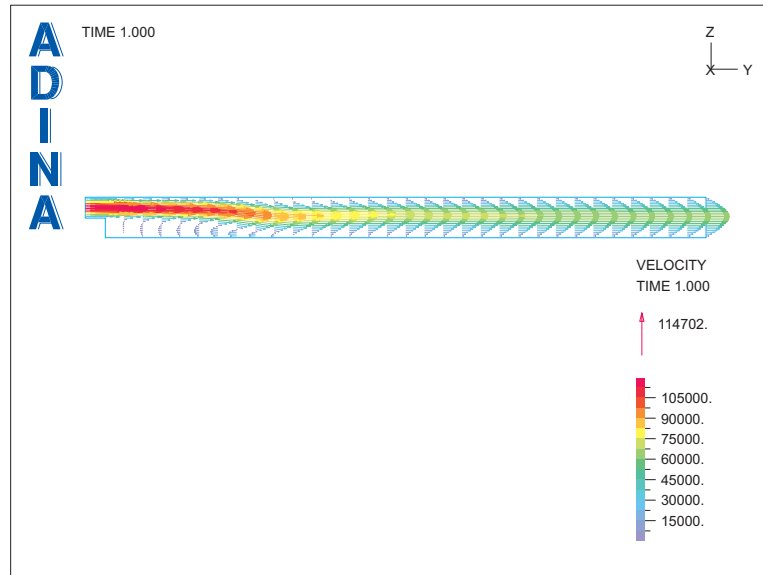
Click the Save icon  and save the database to file prob31. Click the Data File/Solution icon , set the file name to prob31, make sure that the Run Solution button is checked and click Save.


When ADINA-CFD is finished, 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 prob31.

**Plotting the solution**

Click the Model Outline icon , then click the Quick Vector Plot icon . The graphics window should look something like the top figure on the next page.

We are interested in the length of the recirculation region to the right of the step. Click the Node Symbols icon  and use the Zoom icon  to enlarge the region near the end of the recirculation region. The graphics window should look something like the bottom figure on the next page.



The recirculation region appears to end at the indicated node. Click the Query icon  and click on the node. The AUI writes the following message to the message window:

Node 122, curr=( 0.00000E+00, 5.18750E-02, 4.45455E-04)

So the recirculation region has length  $(0.0519 - 0.005) = 0.0469$ , which is comparable with the experimental result of  $14S = 0.0686$ , where  $S$  is the step height (see the reference given at the beginning of this problem description for the experimental result).

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

*Problem 31: Analysis of a laminar step-wall diffuser*

---

This page intentionally left blank.