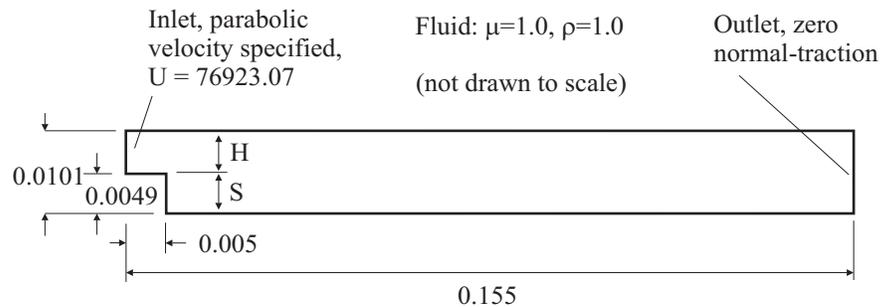


Problem description

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 and FCBI-C 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 topics that have not been presented in previous problems:

- Specifying a parabolic velocity profile as a boundary condition
- Using FCBI-C elements

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.

Problem 31: Analysis of a laminar step-wall diffuser

This problem can be solved with the 900 nodes version of the ADINA System.

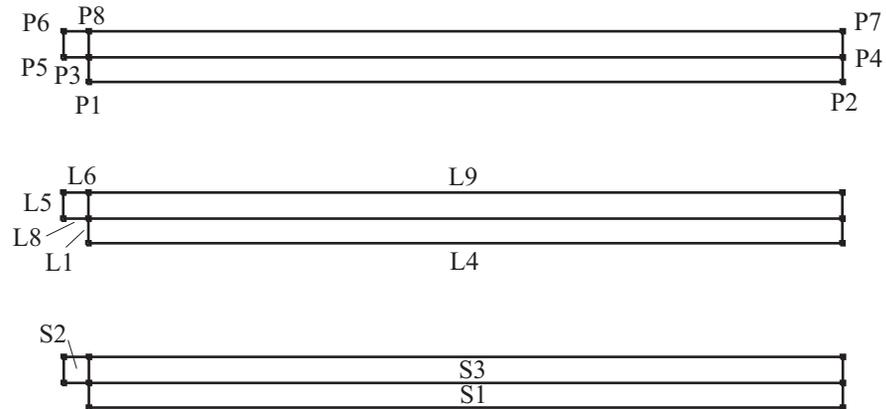
Much of the input for this problem is stored in file prob31_1.in. You need to copy file prob31_1.in from the folder samples\primer into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

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

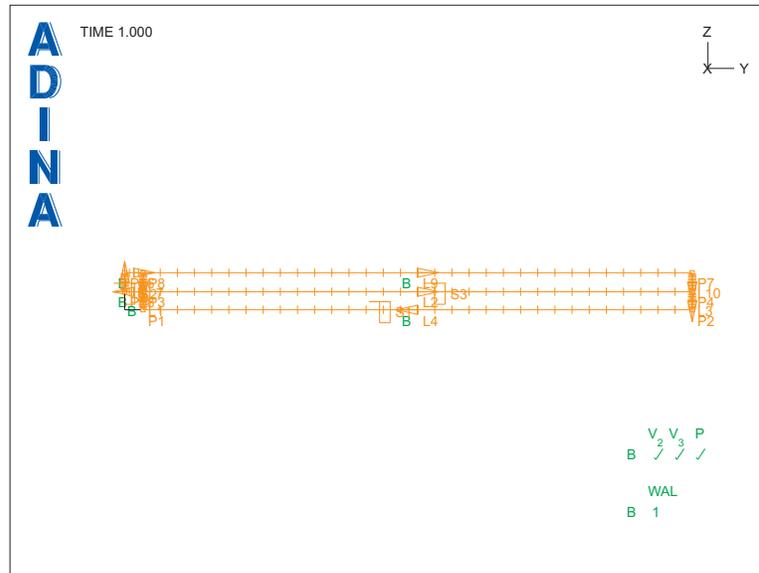
Defining model control data, model geometry, boundary conditions, material properties, element group definitions and subdivision data

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

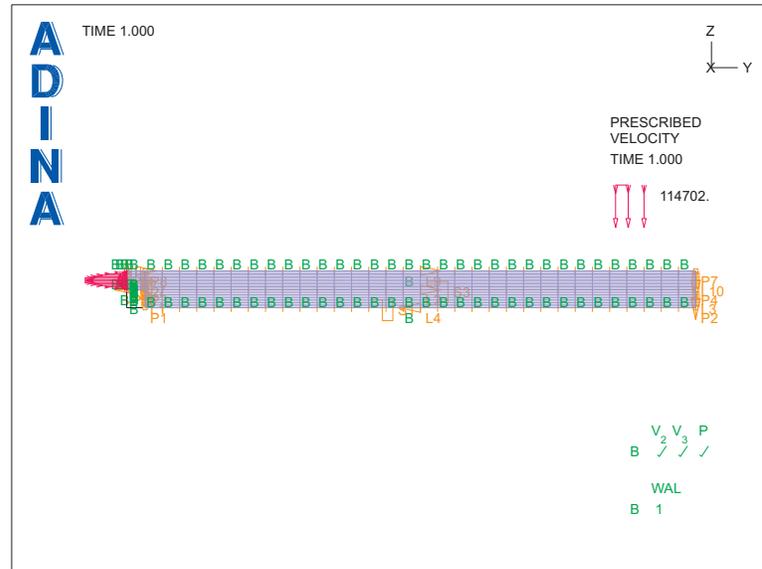
Spatial-function: We need to define a spatial-function corresponding to the parabolic velocity. Choose Geometry→Spatial Functions→Line and add Function Number 1. Set the Type to Quadratic and set “u = 0” to 0.0, “u=0.5” to 1.5 and “u=1” to 0.0. Click OK to close the dialog box.

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.

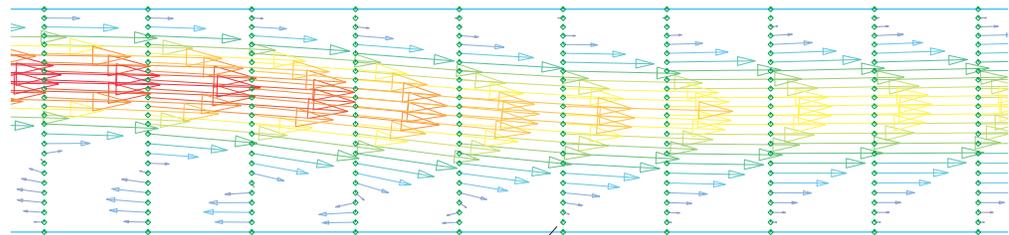
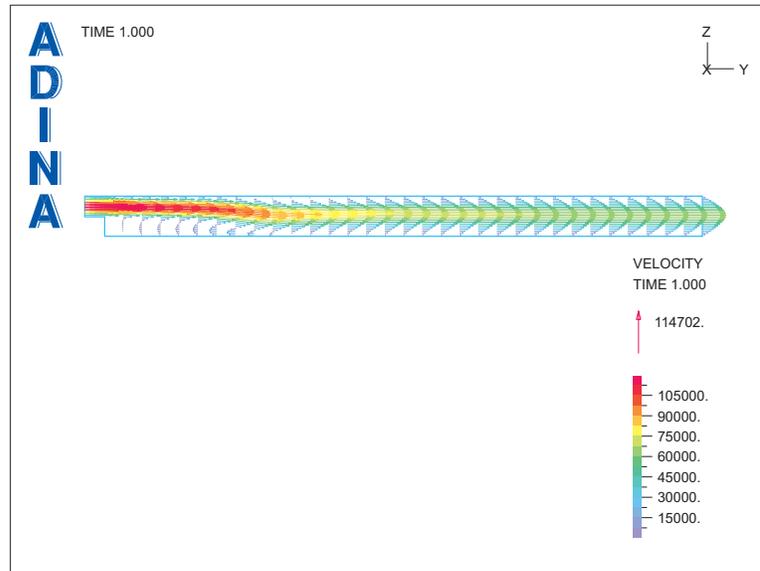
When ADINA CFD is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob31.

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.

Problem 31: Analysis of a laminar step-wall diffuser



Node 122

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).

FCBI-C element

In the following, we demonstrate how to use the FCBI-C element for this laminar step-wall diffuser problem. See the notes at the end of this problem description for more information about the FCBI-C element.

Set the Program Module drop-down list to ADINA CFD (you can discard all changes) and choose file prob31.i db from the recent file list near the bottom of the File menu.

Choosing the FCBI-C element and setting the solution process parameters

Choose Control→Solution Process... and set the "Flow-Condition-Based-Interpolation-Elements" to FCBI-C. Read and close the warning message.

We will change the residual tolerances used in the FCBI-C solution. In the Solution Process dialog box, click the Outer Iteration... button, and, in the Outer Iteration Settings dialog box, click the Advanced Settings... button. In the Equation Residual box, set "Use" to "All" and set the associated tolerance to 1E-5. In the Variable Residual box, make sure that "Use" is set to "All" and set the associated tolerance to 1E-10. Then click OK three times to close all three dialog boxes.

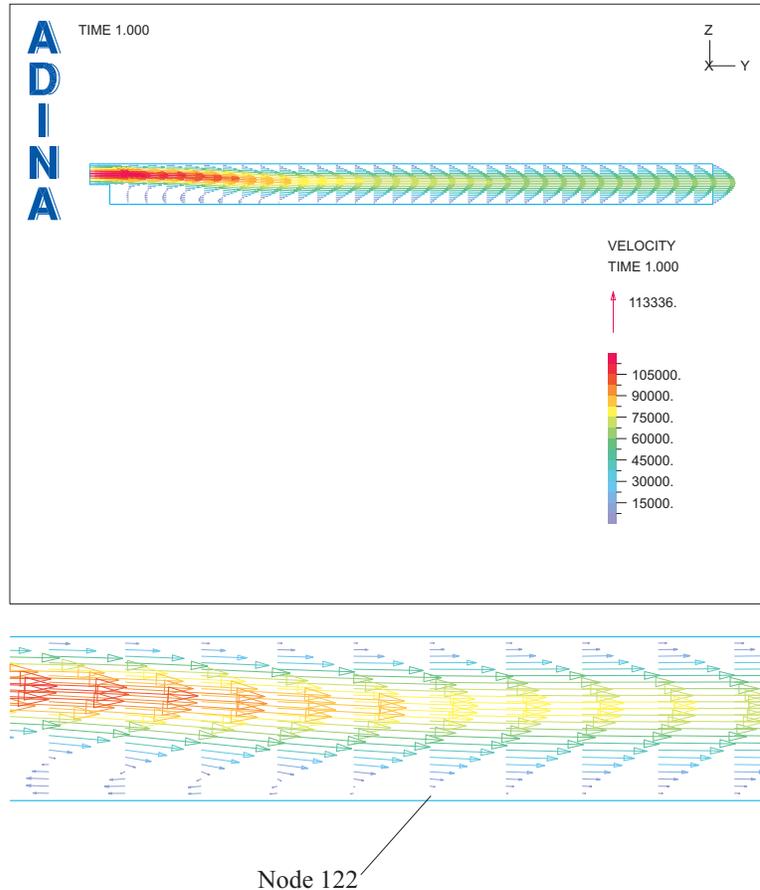
Generating the data file, running ADINA CFD, loading the porthole file

Choose File→ Save As to save the database to file prob31_c. Click the Data File/Solution icon , set the file name to prob31_c, 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 prob31_c.

Plotting the solution

Follows the same plotting steps given above, we can plot the velocity profile shown in the figures on the next page.



It can be seen the velocity profile and length of the recirculation region computed with the FCBI-C element are very similar to those computed with the FCBI element.

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

Notes for the FCBI-C element

1) The default element used in ADINA CFD is the FCBI element. In the solution process associated with the FCBI element, all variables for all equations are solved simultaneously. Thus the solution process for the FCBI element requires a large amount of memory to store all variables. This may not be an issue for small or medium sized models, but it may become an issue for large models due to limited memory.

In ADINA CFD, we have another option for large models, the FCBI-C element. The solution process associated with the FCBI-C element uses iterative methods to solve the fluid model,

Problem 31: Analysis of a laminar step-wall diffuser

in which the fluid governing equations are solved one by one in a certain order. Compared to the solution process for the FCBI element, the iterative solution process associated with the FCBI-C element requires less memory. Therefore the FCBI-C element is typically recommended for large fluid models.

2) The default equation solver for the FCBI element is the sparse solver, but the default solver for the FCBI-C element is the AMG (type 1) solver. The details on the iterative equation solvers for the FCBI-C element can be found in the ADINA CFD TMG manual section 11.3.3. Generally the AMG (type 1) solver can be used for many practical applications.

3) Because of the iterative nature of the solution process for the FCBI-C element, solving each fluid governing equation may require different residual tolerances. These tolerances are set in the Outer Iteration Advanced Settings dialog box. In the "Convergence Criteria" box, there are two tolerance criteria:

- (1) Equation Residual (ER)
- (2) Variable Residual (VR)

The computation stops when one of these two criteria is satisfied. The default ER is for mass conservation (Mass) equation. Its default value is 0.0001. The default VR is for all variables (All). Its default value is 0.001. These default criteria are good enough for many practical applications. But they may need to be changed for different problems. For example, if a pure heat conduction problem is solved by using the ADINA CFD FCBI-C element, "Energy" may be a better ER criterion than "Mass". The definitions of ER and VR and how to choose different ER and VR for different physical models can be found in the ADINA CFD TMG manual section 11.2.2.1.

4) In this model, only the continuity and momentum equations are solved, and the default residual settings can be used. However, we demonstrate the changes of the residual settings by using the "All" option for both ER and VR, and we tighten the ER "All" tolerance to 1.0E-05, and the VR "All" tolerance to 1.0E-10. The VR tolerance value is much smaller than the ER tolerance value. This will force the computation to continue until the ER criterion is satisfied.