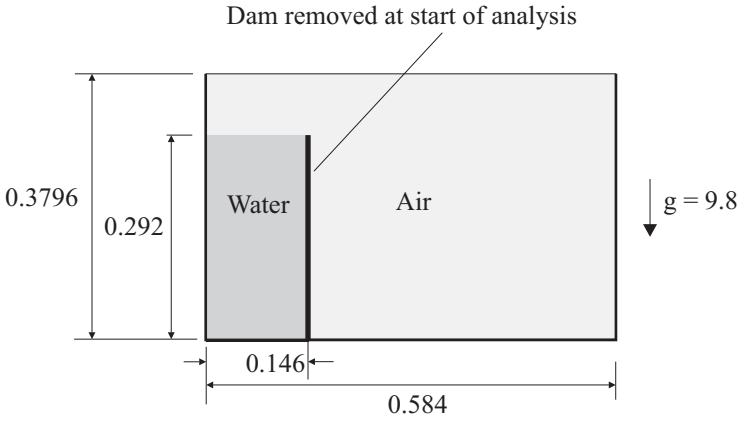
Problem description

It is desired to analyze the motion of water within a basin. Initially, the basin contains a dam, and the water is confined by the dam as shown. At the start of the analysis, the dam is removed and the water flows into the rest of the basin.



SI units used.

Water: $\mu = 10^{-3}$, $\rho = 1000$ Air: $\mu = 10^{-5}$, $\rho = 1$

Slip walls are used to model the basin.

We use the VOF (volume of fluid) method to solve this problem. During the analysis, the water is represented by fluid for which the value of the VOF species is greater than ½.

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

- Defining fluids with VOF species.
- Defining and applying initial conditions to geometry.

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.

Much of the input for this problem is stored in files prob32_1.in and prob32_2.in. You need to copy files prob32_1.in and prob32_2.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.

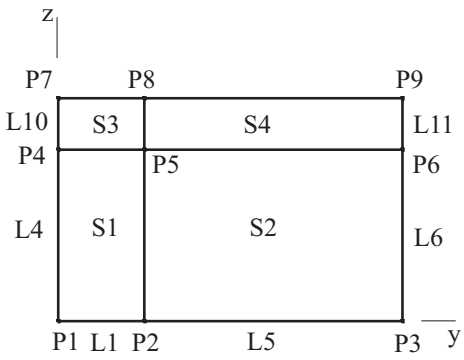
Defining model control data

Analysis type: Set the Analysis Type drop-down list to Transient.

VOF control parameters: Choose Model→Flow Assumptions, check the "Use VOF Method" field and click the VOF Control… button. Set the "Max. Number of Iterations Allowed" to 50 and click OK twice to close both dialog boxes.

Defining time steps, model geometry, boundary conditions and material properties

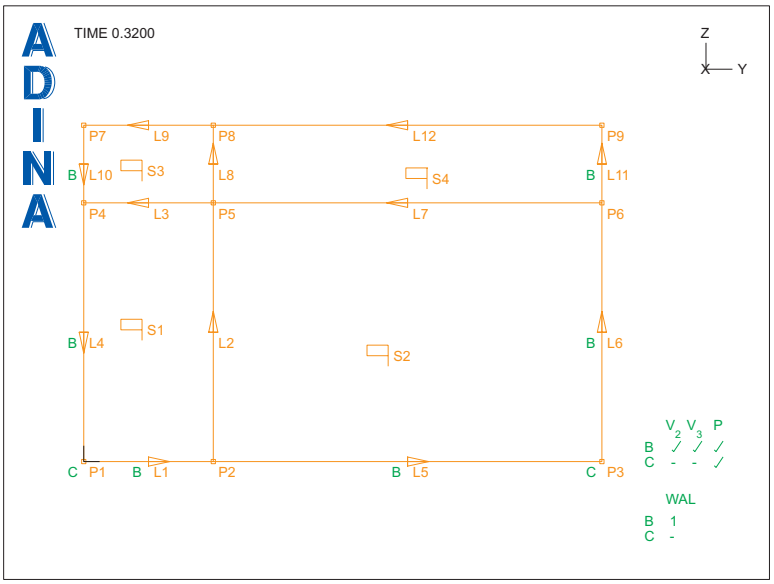
The following figure shows the key geometry used in defining this model.



We have placed all of the time step definitions, geometry definitions, material definitions and boundary conditions in batch file prob32_1.in. Choose File→Open Batch, navigate to the working directory or folder, select the file prob32_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.

Notice that a wall boundary condition is used to model the basin. This is a slip-wall boundary condition. Also notice that the velocities are fixed at points 1 and 3. If we had not fixed the velocities at these points, there would have been slip at these points, corresponding to the averaged normals from the adjacent lines.



Also notice that the dam itself is not modeled. The presence of the dam is included in the model only through the choice of initial conditions.

Defining and applying initial conditions

Initially, water occupies geometry surface 1 and air occupies the other surfaces. Choose Model→Initial Conditions→Define, add name INIT, and, in the first row of the table, set the Variable to VOF-SPECIES1 and the Value to 1.0. Click OK to close the dialog box.

Now choose Model→Initial Conditions→Apply, set the "Apply to" field to "Surfaces", and, in the first row of the table, set the Surface # to 1 and the Initial Condition to INIT. Click OK to close the dialog box.

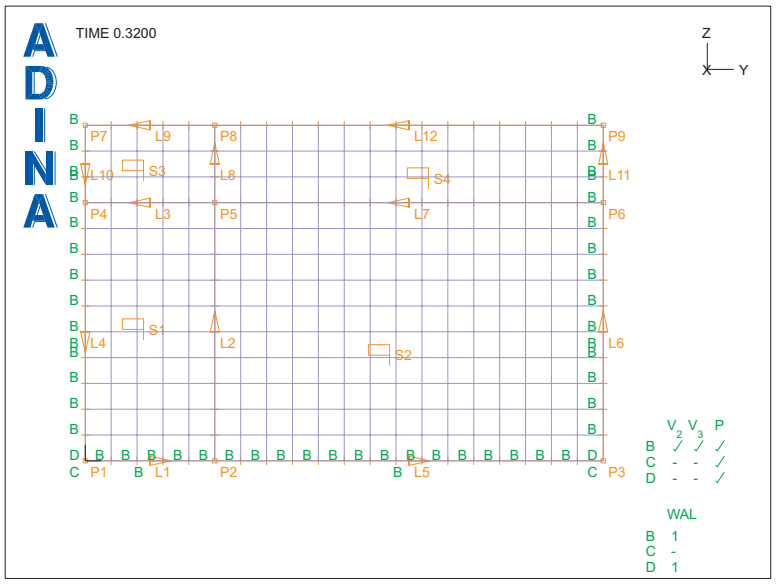
Defining the element group and VOF material

Click the Define Element Groups icon and add group 1. Set the Element Sub-Type to Planar and the Default Material to 2. Click the Advanced tab, make sure that the Associated VOF Material is 1 and click the … button to the right of that field. In the VOF Material dialog box, add VOF Material Number 1, make sure that, in the First Species dialog box, the Material Number is 1 and click OK twice to close both dialog boxes.

Meshing

We have placed the subdivision and meshing commands in batch file prob32_2.in. Choose File→Open Batch, select the file prob32_2.in and click Open. The AUI processes the commands in the batch file.

The graphics window should look something like this:



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

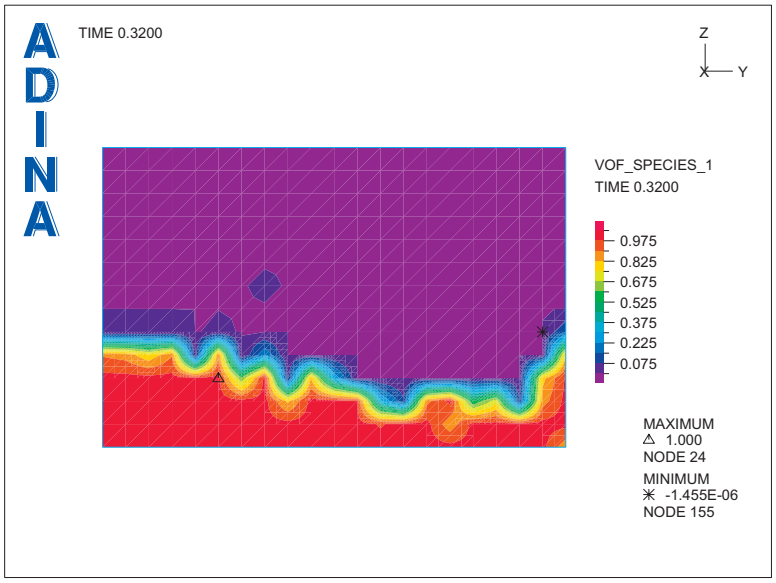
Click the Save icon and save the database to file prob32. Click the Data File/Solution icon, set the file name to prob32, make sure that the Run Solution button is checked and click Save.

The ADINA-CFD run finishes in 180 time steps. 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 prob32.

Plotting the solution

Click the Model Outline icon, then click the Create Band Plot icon, set the Band Plot Variable to (Fluid Variable: VOF_SPECIES_1) and click OK. Use the Pick icon and the

mouse to rearrange the graphics window until it looks something like this:



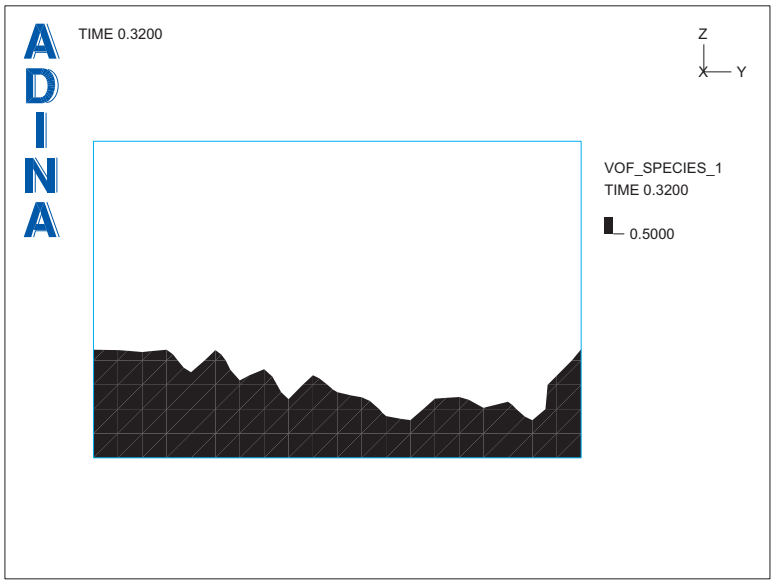
In this plot, the water appears as red and the air appears as dark blue. Other colors correspond to a mixture of water and air.

For presentation purposes, we assume that any region for which the VOF species is greater than ½ corresponds to water. Click the Modify Band Plot icon and click the Band Table… button. In the Define Band Table Depiction dialog box, set the Number of Colors to 2, set the "Color for Minimum" to BLACK, the "Color for Maximum" to WHITE and click OK. Now click the Band Rendering… button, set the "Extreme Values" to "Do not Plot" and click OK twice to close both dialog boxes.

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

Now click the Movie Load Step icon to create an animation. The water flows out of its confined region into the rest of the basin. 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).


ADINA
TIME 0.3200
Z
X
Y
VOF_SPECIES_1
TIME 0.3200
0.5000