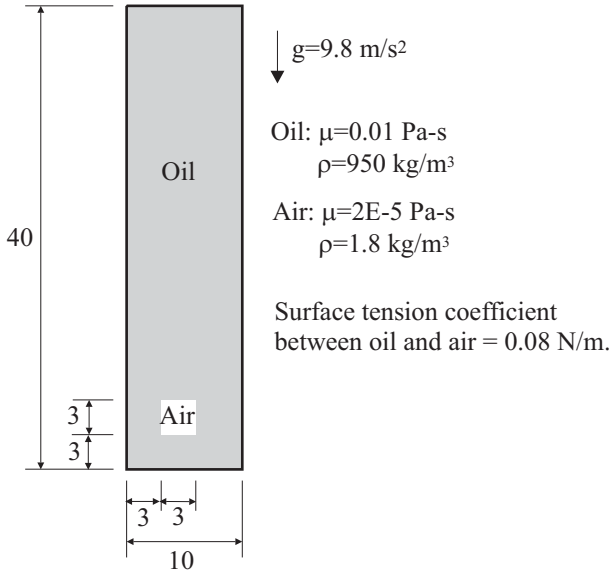
Problem description

An air bubble rises in a column of oil as shown:



All lengths in mm

Both the air and the oil are modeled as incompressible Newtonian fluids. The oil is considered to be the primary fluid and the air is considered to be VOF species 1.

The model is planar and two-dimensional.

Eventually the bubble hits the top of the column. We control the interface angle between the bubble and the wall at the top of the column by setting the VOF wall angle to 0.0.

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

- Defining a boundary condition of type VOF wall angle

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 `prob36_1.in`, `prob36_2.in`, `prob36_1.plo` and `prob36_2.plo`. You need to copy files `prob36_1.in`, `prob36_2.in`, `prob36_1.plo`, `prob36_2.plo` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

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

Defining model control data, geometry, subdivision data, boundary conditions and materials

We have prepared a batch file (`prob36_1.in`) that performs the following operations:

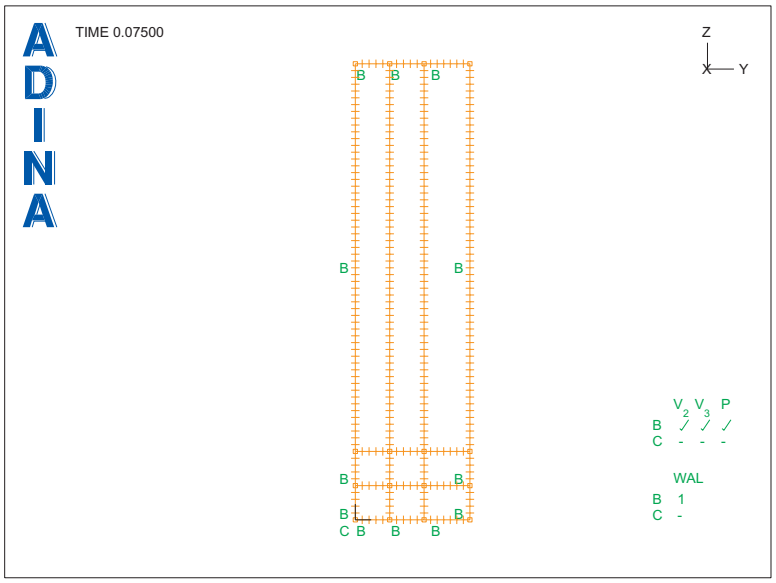
- ► Specifies a transient analysis.
- ► Sets the Courant number and other parameters used in automatic time-stepping.
- ► Defines points, lines and surfaces.
- ► Subdivides the surfaces.
- ► Defines a wall boundary condition and fixes the pressure at one point.
- ► Defines the oil and air materials (materials 1 and 2 respectively)
- ► Plots the model

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob36_1.in` and click Open. The graphics window should look something like the figure on the next page.

Defining the VOF analysis

Choose Model→Flow Assumptions and set VOF to "Yes". Click the VOF Control… button, set the "Max. Number of Iterations Allowed" to 150 and click OK twice to close both dialog boxes.

Click the Element Groups icon  and add group 1, set the Element Sub-Type to Planar, click the Advanced tab and click the … button to the right of the Associated VOF Material field. In the VOF Material dialog box, add VOF material 1, set the First Species Material Number to 2, set the "Surface Tension Coefficient between Primary and First Species" to 0.08 and click OK. Click OK to close the Define Element Group dialog box.



Click the Special Boundary Conditions icon SBC, add condition 2 and set the Type to VOF Wall Angle. Set the "Wall Angle between Primary Fluid and First Species" to 0.0, and enter the following line numbers in the first 12 rows of the table: 3, 4, 7, 9, 10, 13, 17, 18, 20, 21, 23, 24 (these are the same lines as are used in the wall boundary condition). Click OK to close the dialog box.

When you click the Redraw icon , the graphics window should look something like the top figure on the next page.

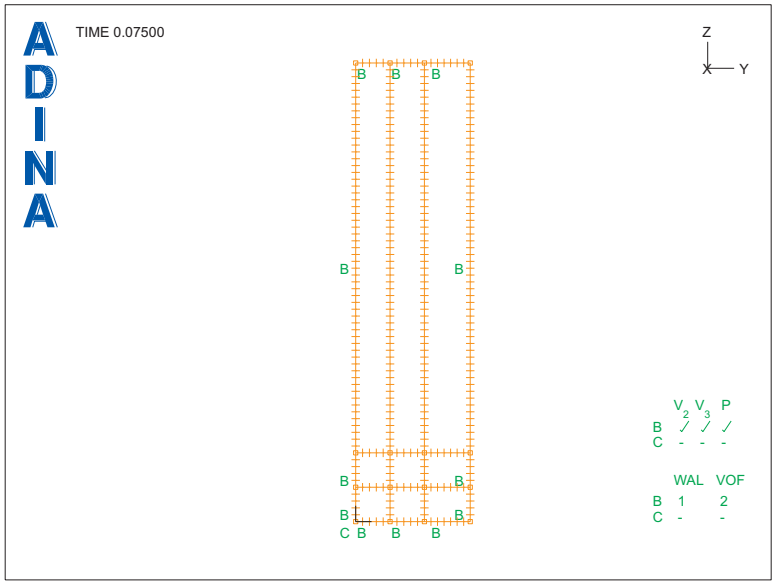
Initial conditions: Choose Model→Initial Conditions→Define, add initial condition BUBBLE, and, in the first row of the table, set the Variable to VOF-SPECIES1 and the Value to 1.0. Now click the Apply... button and, in the Apply Initial Conditions dialog box, set the "Apply to" field to "Face/Surface" and enter 5 in the first row and column of the table. Click OK twice to close both dialog boxes.

Meshing

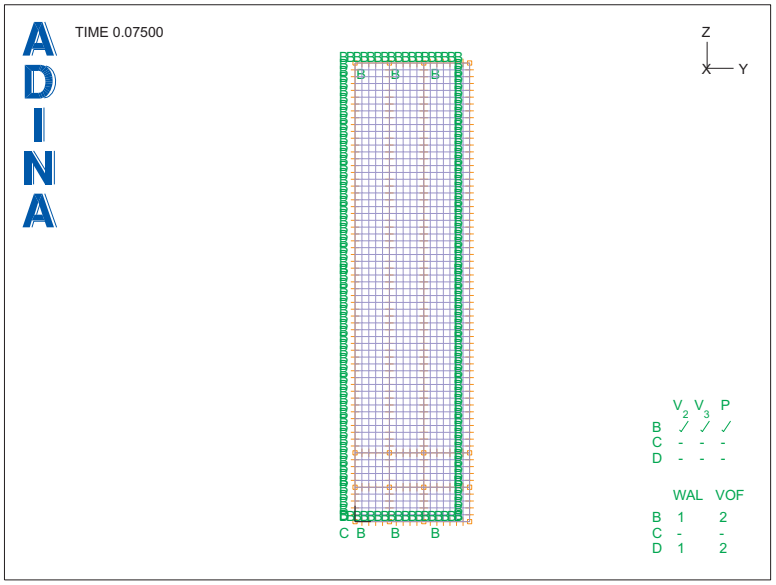
We have prepared a batch file (`prob36_2.in`) that performs the following operations:

- ► Meshes the geometry (using 4-node FCBI elements)
- ► Regenerates the graphics

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob36_2.in` and click Open. The AUI processes the commands in the batch file.



The graphics window should look something like this:



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

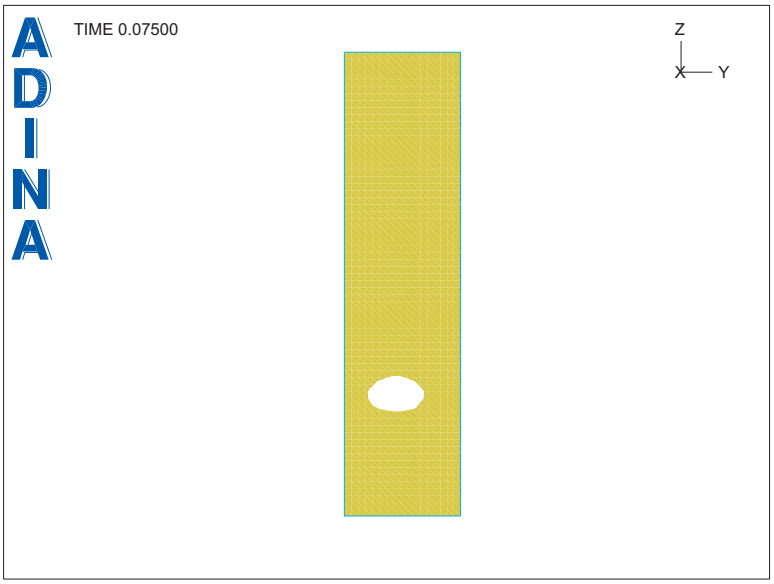
Click the Save icon and save the database to file `prob36`. Click the Data File/ Solution icon, set the file name to `prob36`, make sure that the Run Solution button is checked and click Save.

ADINA CFD runs for 150 solution steps.

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 `prob36`.

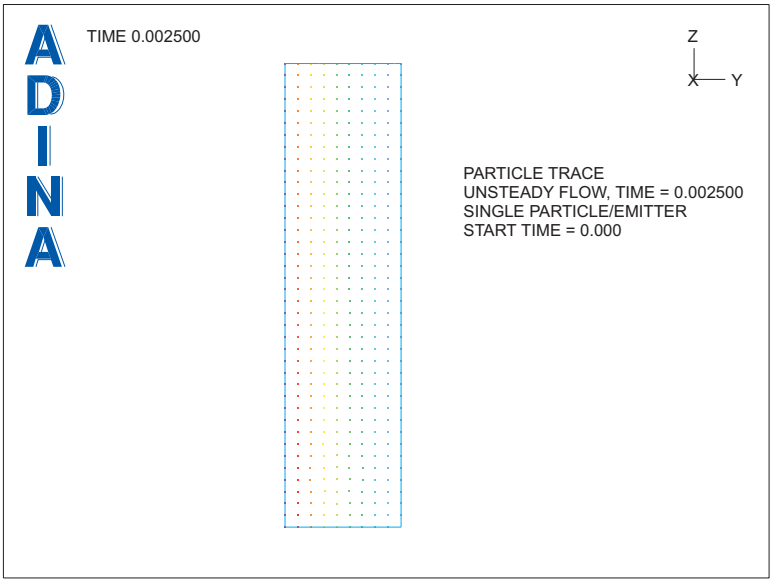
Post-processing

Visualizing the bubble motion: For presentation purposes, we assume that any region for which the VOF species is greater than ½ corresponds to air. We have put the necessary commands in a batch file (`prob36_1.plo`). Choose File→Open Batch, navigate to the working directory or folder, select the file prob36_1.plo and click Open. The graphics window should look something like this:



Click the Movie Load Step icon, then the Animate icon. The bubble immediately becomes rounded and starts to rise, oscillating slightly as it rises. Click the Refresh icon to clear the animation.

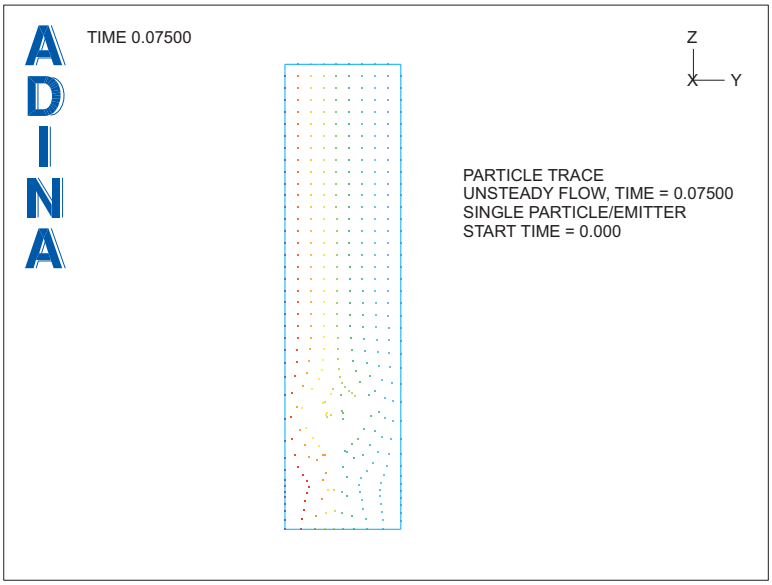
Visualizing the oil motion: We can use the particle tracing feature to visualize the motion of the oil. We have put the necessary commands in a batch file (`prob36_2.plo`). Choose File→Open Batch, navigate to the working directory or folder, select the file `prob36_2.plo` and click Open. Use the Pick icon and the mouse to rearrange the graphics until the graphics window looks something like this:



At this point, the particle traces near the beginning of the solution are displayed.

Now click the Last Solution icon to compute the particle traces. The graphics window should look something like the figure on the next page.

Click the Movie Load Step icon, then the Animate icon. The particles are seen to move out of the way of the rising bubble. Click the Refresh icon to clear the animation.



Solution with more steps

In the preceding, we only ran the solution for 150 steps so that the analysis would not take too long. However, it is of interest to run the solution for more steps, if your computer is fast enough.

Set the Program Module drop-down list to ADINA CFD (you can discard all changes), and choose database file `prob36.idb` from the recent file list near the bottom of the File menu.

Choose Control→Time Step, set the Number of Steps to 720 in the first row of the table and click OK.

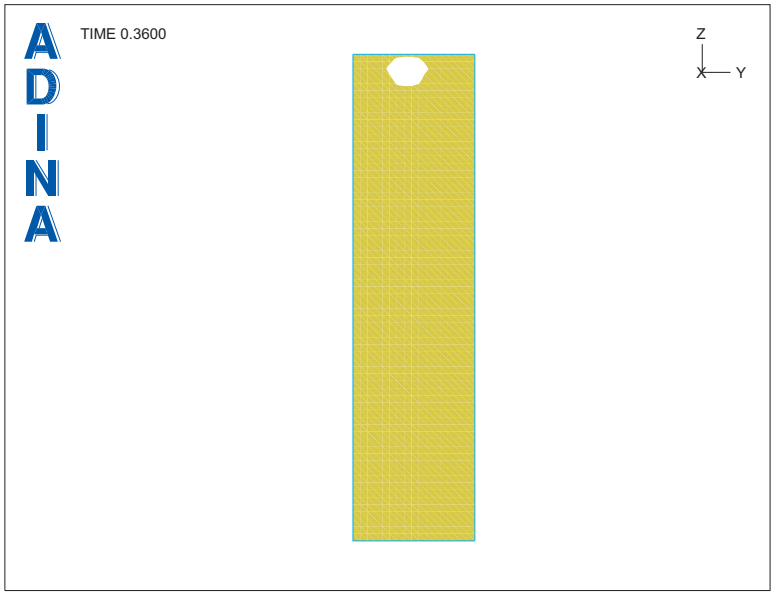
Click the Data File/Solution icon  , set the file name to `prob36b`, make sure that the Run Solution button is checked and click Save.

ADINA CFD runs for 720 solution steps.

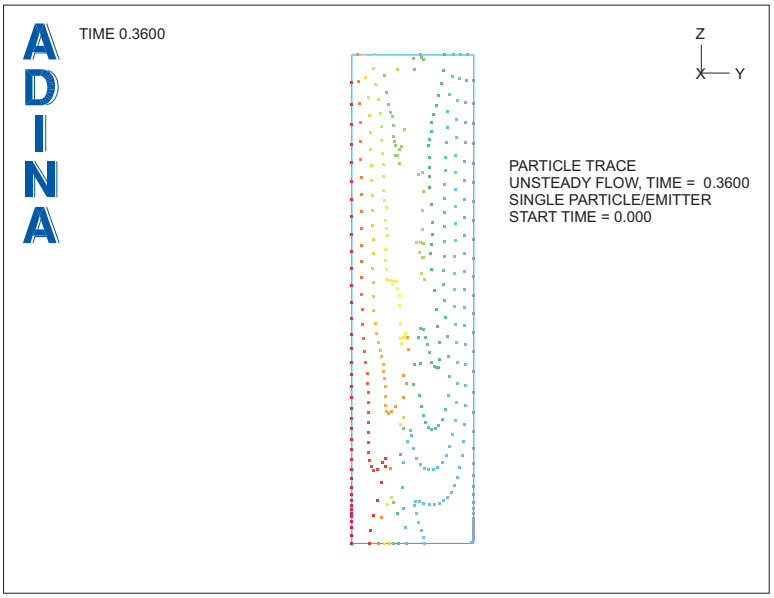
When ADINA CFD is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file `prob36b`.

Post-process the model exactly as for the previous analysis. You should see the following plots.

Visualization of bubble motion:



Visualization of oil motion:



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