

Introduction

This tutorial examines the flow of ink as it is ejected from the nozzle of a printhead in an inkjet printer. Using FLUENT's volume of fluid (VOF) multiphase modeling capability, you will be able to predict the shape and motion of the resulting droplets in an air chamber.

This tutorial demonstrates how to do the following:

- Set up and solve a transient problem using the pressure-based solver and VOF model.
- Copy material from the property database.
- Define time-dependent boundary conditions with a user-defined function (UDF).
- Patch initial conditions in a subset of the domain.
- Automatically save data files at defined points during the solution.
- Examine the flow and interface of the two fluids using volume fraction contours.

Prerequisites

This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have completed [Tutorial 1](#). Some steps in the setup and solution procedure will not be shown explicitly.

Problem Description

The problem considers the transient tracking of a liquid-gas interface in the geometry shown in [Figure 17.1](#). The axial symmetry of the problem allows a 2D geometry to be used. The computation grid consists of 24,600 cells. The domain consists of two regions: an ink chamber and an air chamber. The dimensions are summarized in [Table 17.1](#).

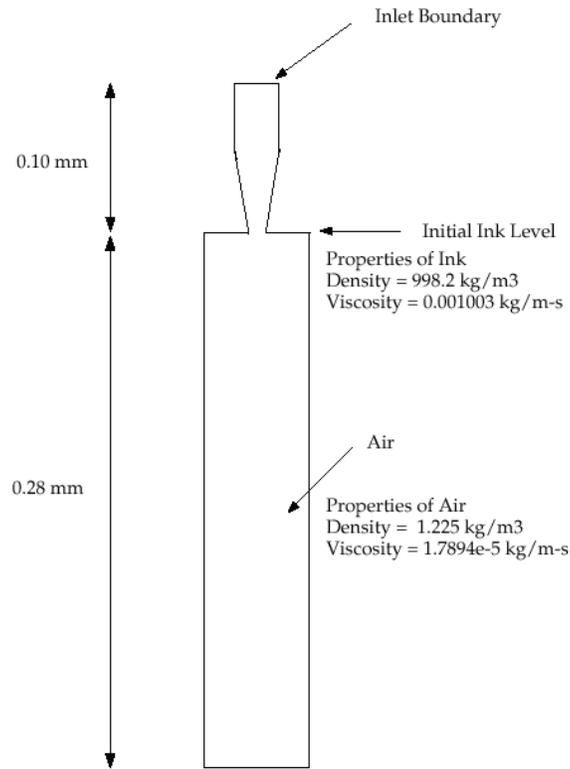


Figure 17.1: Schematic of the Problem

Table 17.1: Ink Chamber Dimensions

Ink Chamber, Cylindrical Region: Radius (mm)	0.015
Ink Chamber, Cylindrical Region: Length (mm)	0.050
Ink Chamber, Tapered Region: Final Radius (mm)	0.009
Ink Chamber, Tapered Region: Length (mm)	0.050
Air Chamber: Radius (mm)	0.030
Air Chamber: Length (mm)	0.280

The following is the chronology of events modeled in this simulation:

- At time zero, the nozzle is filled with ink, while the rest of the domain is filled with air. Both fluids are assumed to be at rest. To initiate the ejection, the ink velocity at the inlet boundary (which is modeled in this simulation by a user-defined function) suddenly increases from 0 to 3.58 m/s and then decreases according to a cosine law.
- After 10 microseconds, the velocity returns to zero.

The calculation is run for 30 microseconds overall, i.e., three times longer than the duration of the initial impulse.

Because the dimensions are small, the double-precision version of FLUENT will be used. Air will be designated as the primary phase, and ink (which will be modeled with the properties of liquid water) will be designated as the secondary phase. Patching will be required to fill the ink chamber with the secondary phase. Gravity will not be included in the simulation. To capture the capillary effect of the ejected ink, the surface tension and prescription of the wetting angle will be specified. The surface inside the nozzle will be modeled as neutrally wettable, while the surface surrounding the nozzle orifice will be non-wettable.

Setup and Solution

Preparation

1. Download `vof.zip` from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD to your working folder (as described in Tutorial 1).
2. Unzip `vof.zip`.
inkjet.msh and inlet.c can be found in the vof folder created on unzipping the file.
3. Start the 2DDP (`2ddp`) version of FLUENT.

Step 1: Grid

1. Read the mesh file `inkjet.msh`.

File → **Read** → Case...

A warning message will be displayed twice in the console. You need not take any action at this point, as the issue will be rectified when you define the solver settings in Step 2.

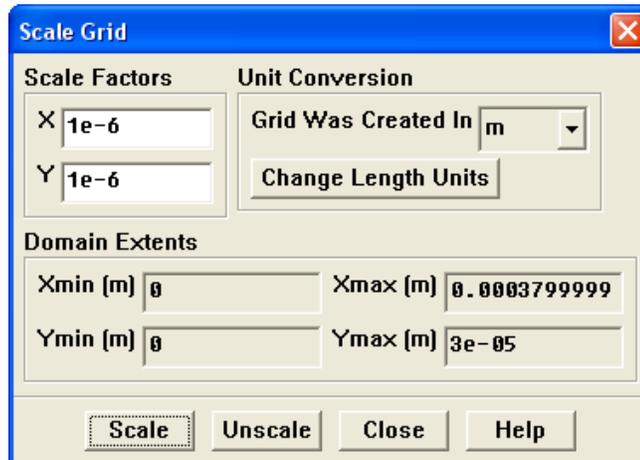
2. Check the grid.

Grid → Check

FLUENT will perform various checks on the mesh and report the progress in the console. Make sure that the minimum volume reported is a positive number.

3. Scale the grid.

Grid → Scale...



- (a) Enter $1e-6$ for X and Y in the Scale Factors group box.
- (b) Click Scale and close the Scale Grid panel.

4. Define the units for the grid.

Define → Units...



- Select length from the Quantities list.
- Select mm from the Units list.
- Select surface-tension from the Quantities list.
- Select dyn/cm from the Units list.
- Close the Set Units panel.

5. Display the grid with the default settings (Figure 17.2).

Display → Grid...



Figure 17.2: Default Display of the Nozzle Grid

Extra: *By zooming in with the middle mouse button, you can see that the interior of the model is composed of a fine mesh of quadrilateral cells (see Figure 17.3).*

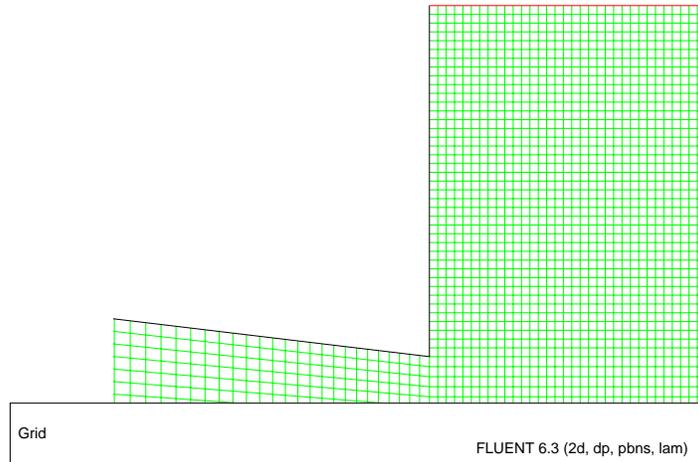
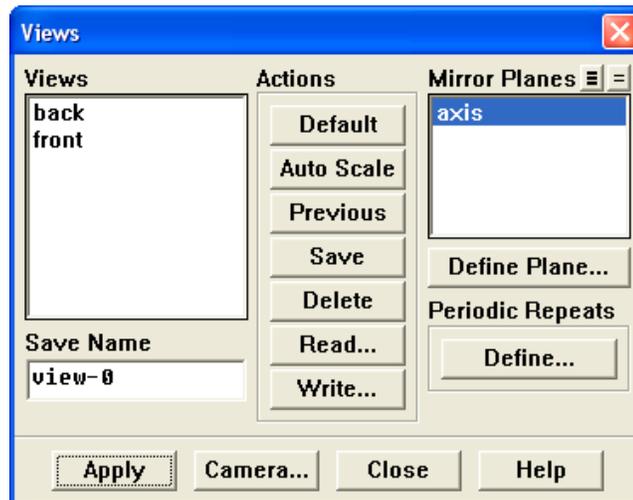


Figure 17.3: The Quadrilateral Mesh

6. Manipulate the grid display to show the full chamber upright.

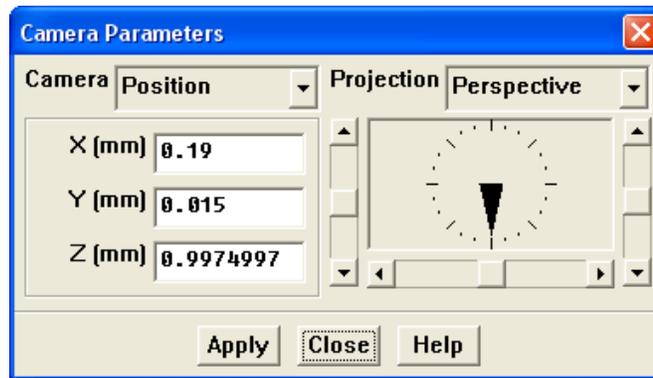
Display → Views...



- Select axis from the Mirror Planes selection list.
- Click Apply.

The grid display will be updated to show both sides of the chamber.

(c) Click the Camera... button to open the Camera Parameters panel.



i. Drag the indicator of the dial with the left mouse button in the clockwise direction until the upright view is displayed (Figure 17.4).

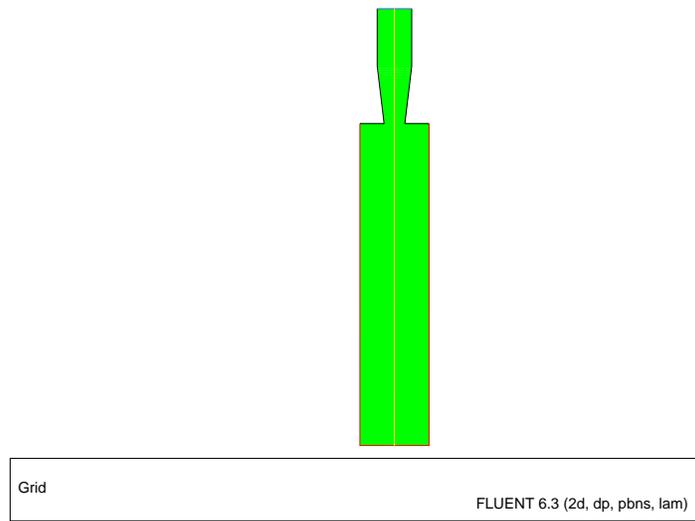


Figure 17.4: Grid Display of the Nozzle Mirrored and Upright

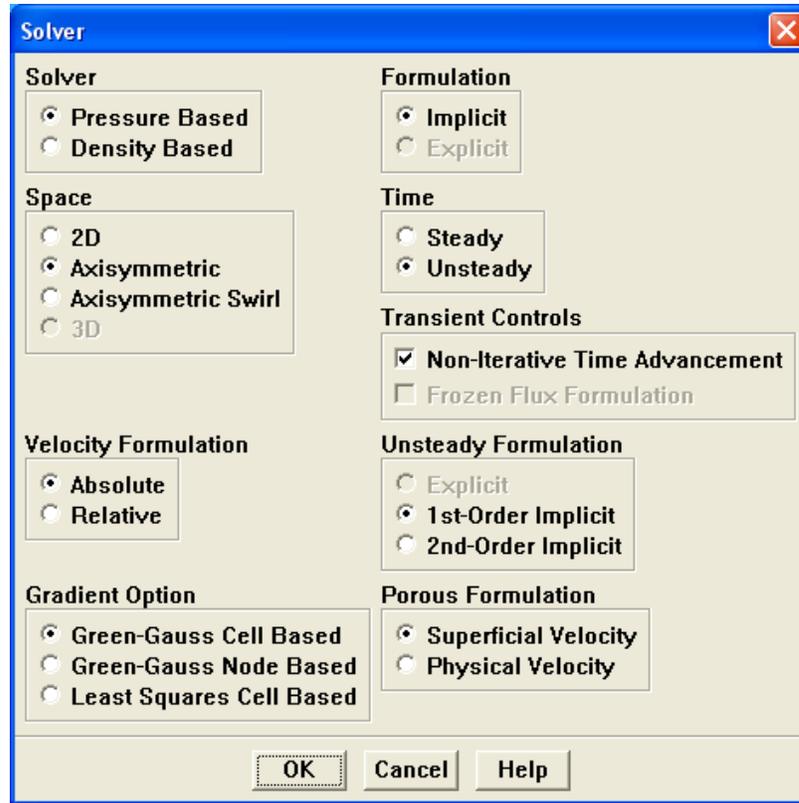
ii. Close the Camera Parameters panel.

(d) Close the Views panel.

Step 2: Models

1. Define the solver settings.

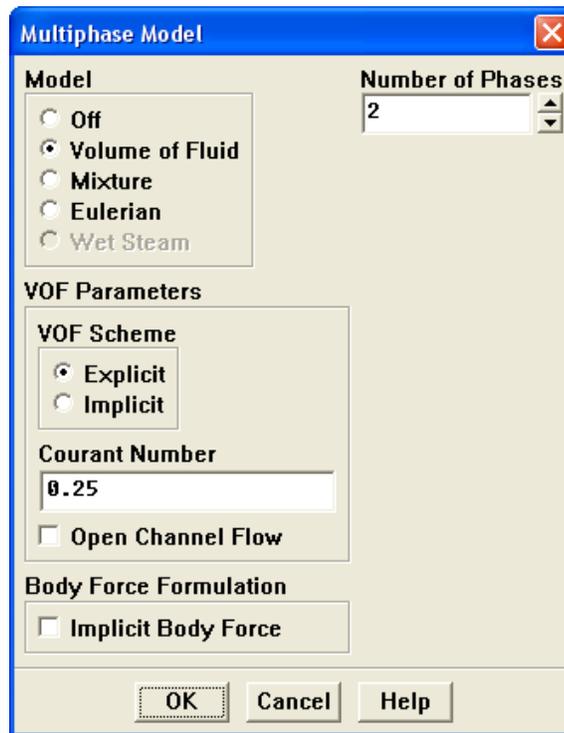
Define → Models → Solver...



- (a) Retain the default setting of Pressure Based in the Solver list.
- (b) Select Axisymmetric from the Space list.
- (c) Select Unsteady from the Time list.
The Solver panel will expand.
- (d) Enable Non-Iterative Time Advancement in the Transient Controls group box.
- (e) Click OK to close the Solver panel.

2. Enable the Volume of Fluid multiphase model.

Define → Models → Multiphase...



- (a) Select Volume of Fluid from the Model list.
The Multiphase Model panel will expand.
- (b) Retain the default settings and click OK to close the Multiphase Model panel.

Step 3: Materials

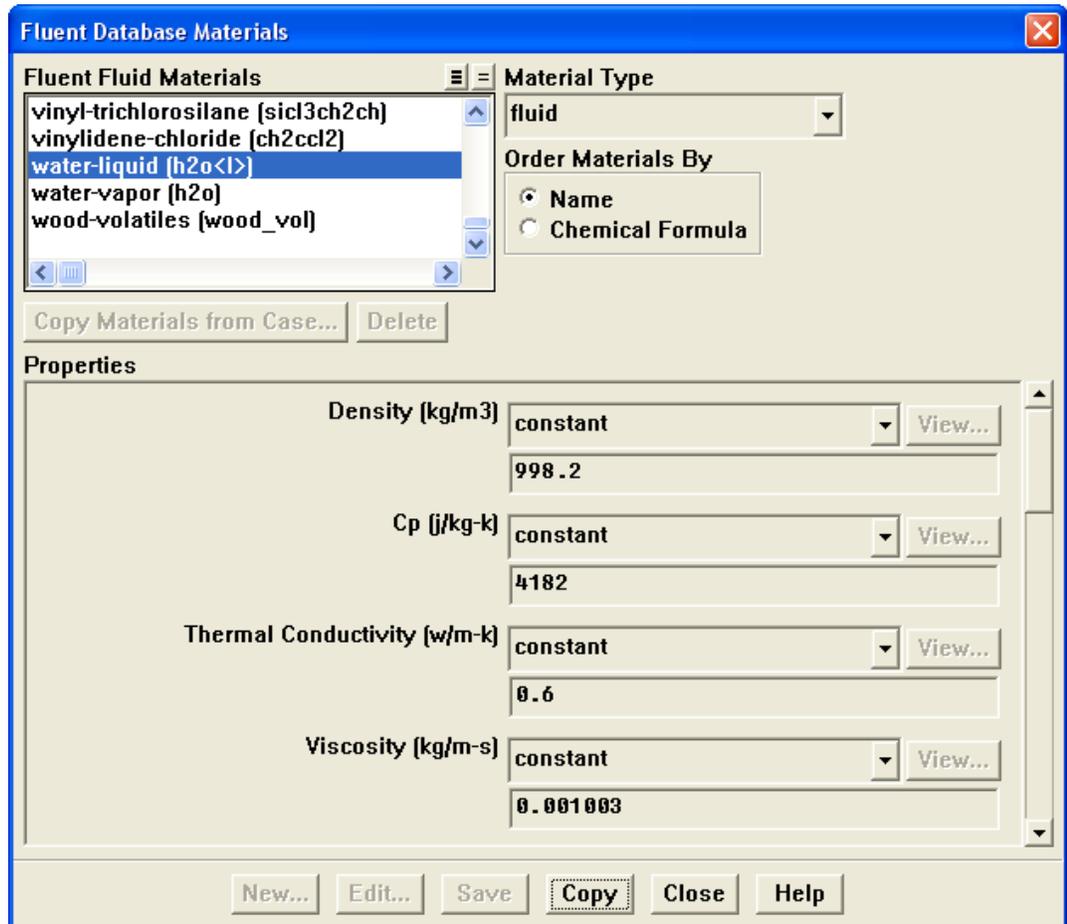
The default properties of air and water defined in FLUENT are suitable for this problem. In this step, you will make sure that both materials are available for selection in later steps.

1. Add water to the list of fluid materials by copying it from the FLUENT materials database.

Define → Materials...

The screenshot shows the 'Materials' dialog box in FLUENT. The 'Name' field is set to 'air'. The 'Material Type' is 'fluid'. The 'Order Materials By' is 'Name'. The 'Chemical Formula' field is empty. The 'Fluent Fluid Materials' dropdown is set to 'air'. The 'Mixture' dropdown is set to 'none'. The 'Properties' section shows 'Density (kg/m3)' set to 'constant' with a value of '1.225' and 'Viscosity (kg/m-s)' set to 'constant' with a value of '1.7894e-05'. At the bottom, there are buttons for 'Change/Create', 'Delete', 'Close', and 'Help'.

- (a) Click the Fluent Database... button to open the Fluent Database Materials panel.

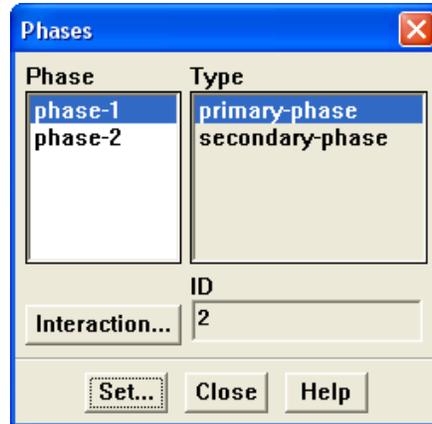


- i. Select water-liquid (h2o<l>) from the Fluent Fluid Materials list.
Scroll down the Fluent Fluid Materials list to locate water-liquid (h2o<l>).
 - ii. Click Copy to copy the information for water to your list of fluid materials.
 - iii. Close the Fluent Database Materials panel.
- (b) Close the Materials panel.

Step 4: Phases

In the following steps, you will define water as the secondary phase. When you define the initial solution, you will patch water in the nozzle region. In general, you can specify the primary and secondary phases whichever way you prefer. It is a good idea to consider how your choice will affect the ease of problem setup, especially with more complicated problems.

Define → Phases...

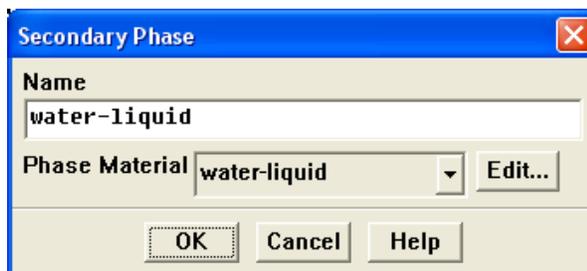


1. Specify air (air) as the primary phase.
 - (a) Select phase-1 in the Phase list.
 - (b) Make sure that primary-phase is selected in the Type list.
 - (c) Click the Set... button to open the Primary Phase panel.

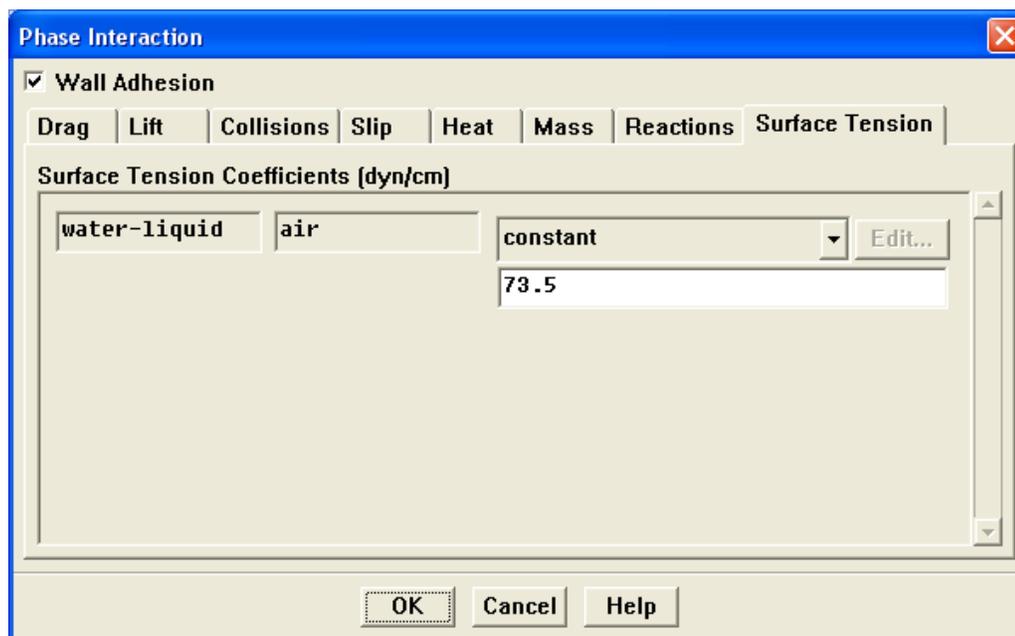


- i. Enter air for Name.
 - ii. Retain the default selection of air in the Phase Material drop-down list.
 - iii. Click OK to close the Primary Phase panel.
2. Specify water (water-liquid) as the secondary phase.
 - (a) Select phase-2 in the Phase list.
 - (b) Make sure that secondary-phase is selected in the Type list.

- (c) Click the Set... button to open the Secondary Phase panel.



- i. Enter water-liquid for Name.
 - ii. Select water-liquid from the Phase Material drop-down list.
 - iii. Click OK to close the Secondary Phase panel.
3. Specify the interphase interaction by clicking the Interaction... button to open the Phase Interaction panel.



- (a) Enable the Wall Adhesion option so that contact angles can be prescribed.
 - (b) Click the Surface Tension tab.

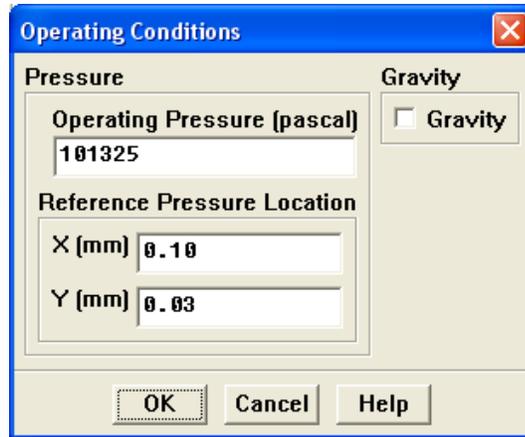
The surface tension coefficient inputs will be displayed.

 - i. Select constant from the Surface Tension Coefficient drop-down list.
 - ii. Enter 73.5 dyn/cm for the Surface Tension Coefficient.
 - (c) Click OK to close the Phase Interaction panel.
4. Close the Phases panel.

Step 5: Operating Conditions

1. Set the operating reference pressure location.

Define → Operating Conditions...



You will set the Reference Pressure Location to be a point where the fluid will always be 100% air.

- (a) Enter 0.10 mm for X.
- (b) Enter 0.03 mm for Y.
- (c) Click OK to close the Operating Conditions panel.

Step 6: User-Defined Function (UDF)

1. Interpret the UDF source file for the ink velocity distribution (`inlet1.c`).

Define → User-Defined → Functions → Interpreted...



- (a) Enter `inlet1.c` for Source File Name.

*If the UDF source file is not in your working folder, then you must enter the entire folder path for Source File Name instead of just entering the file name. Alternatively, click the **Browse...** button and select `inlet1.c` in the `voif` folder that was created after you unzipped the original file.*

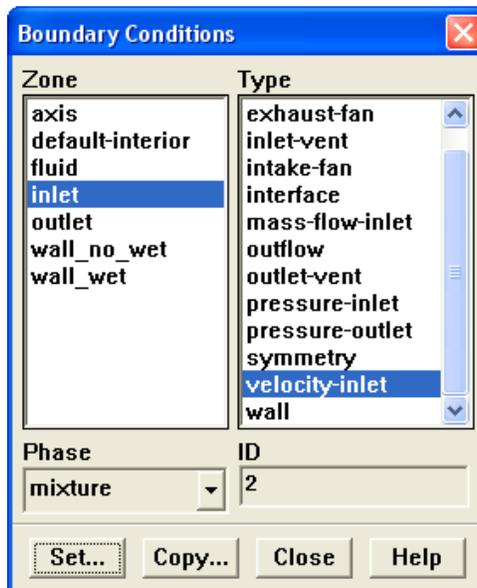
- (b) Click Interpret.

The UDF defined in `inlet1.c` will now be visible and available for selection as `udf membrane_speed` in the drop-down lists of relevant graphical user interface panels.

- (c) Close the Interpreted UDFs panel.

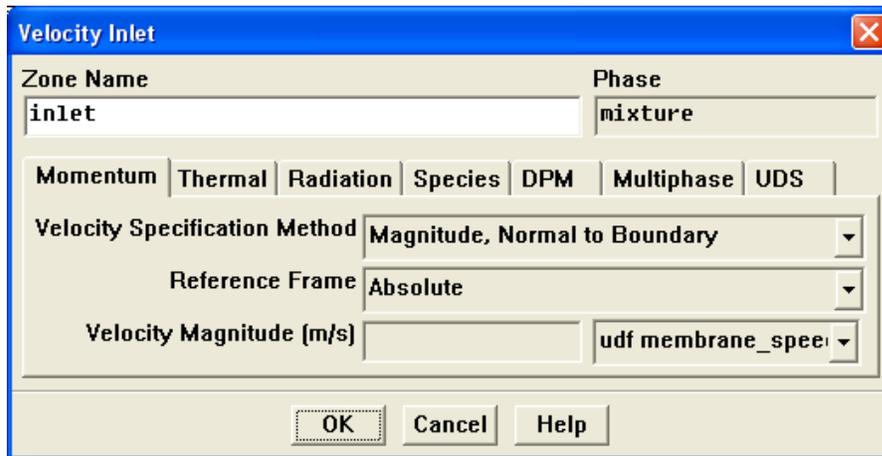
Step 7: Boundary Conditions

Define → Boundary Conditions...

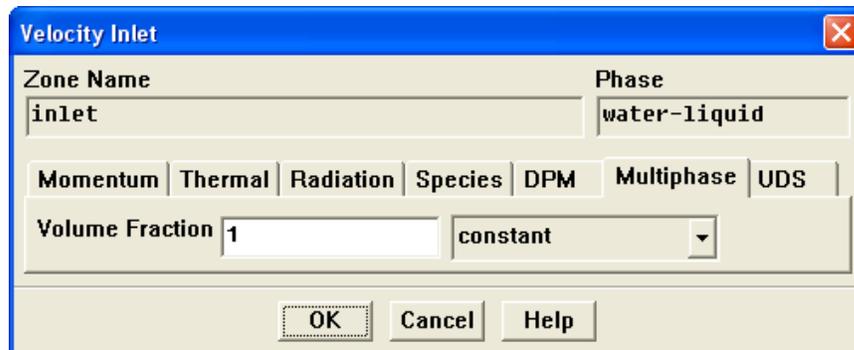


1. Set the boundary conditions at the inlet (`inlet`) for the mixture.
 - (a) Select `inlet` in the Zone list.
 - (b) Retain the default selection of `mixture` in the Phase drop-down list.

- (c) Click the Set... button to open the Velocity Inlet panel.

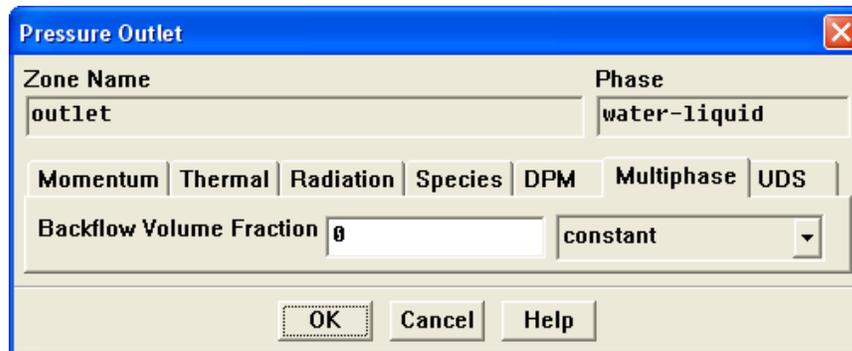


- i. Select udf membrane_speed from the Velocity Magnitude drop-down list.
 - ii. Click OK to close the Velocity Inlet panel.
2. Set the boundary conditions at the inlet (inlet) for the secondary phase.
- (a) Make sure that inlet is selected in the Zone list.
 - (b) Select water-liquid from the Phase drop-down list.
 - (c) Click the Set... button to open the Velocity Inlet panel.



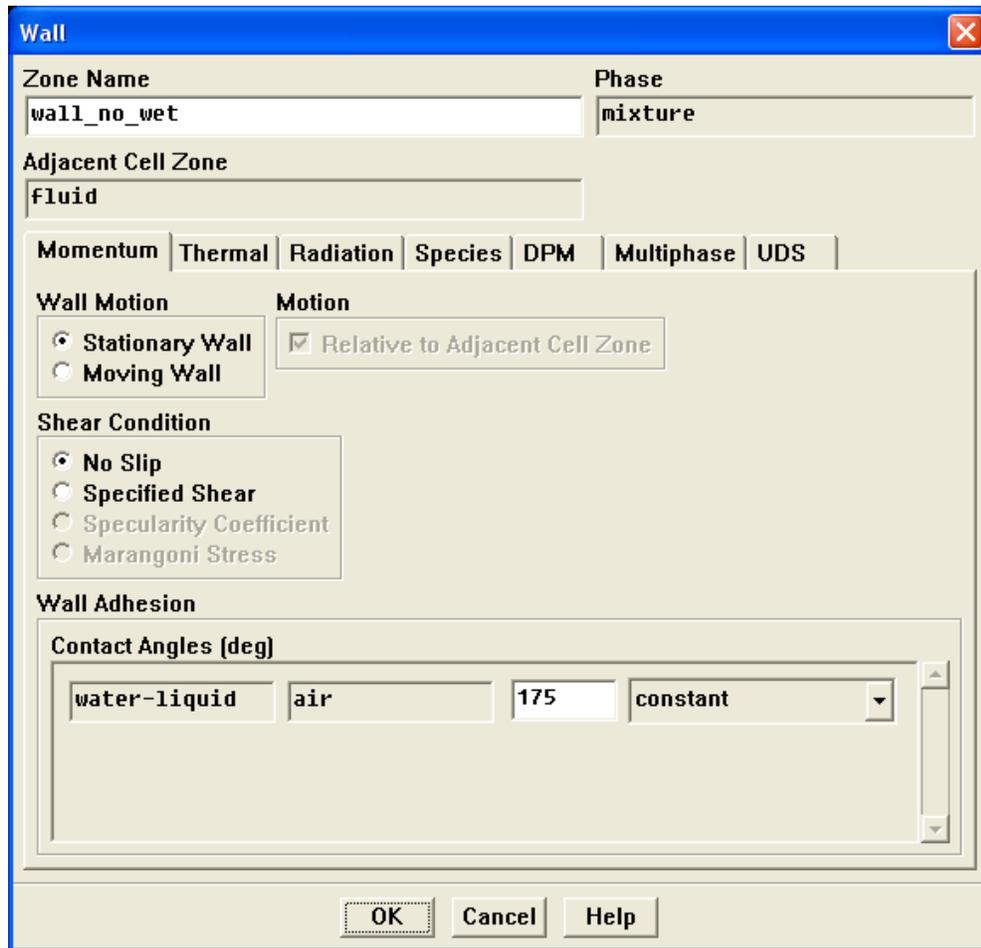
- i. Click the Multiphase tab and enter 1 for the Volume Fraction.
- ii. Click OK to close the Velocity Inlet panel.

3. Set the boundary conditions at the outlet (`outlet`) for the secondary phase.
 - (a) Select `outlet` in the Zone list.
 - (b) Retain the selection of `water-liquid` from the Phase drop-down list.
 - (c) Click the `Set...` button to open the Pressure Outlet panel.



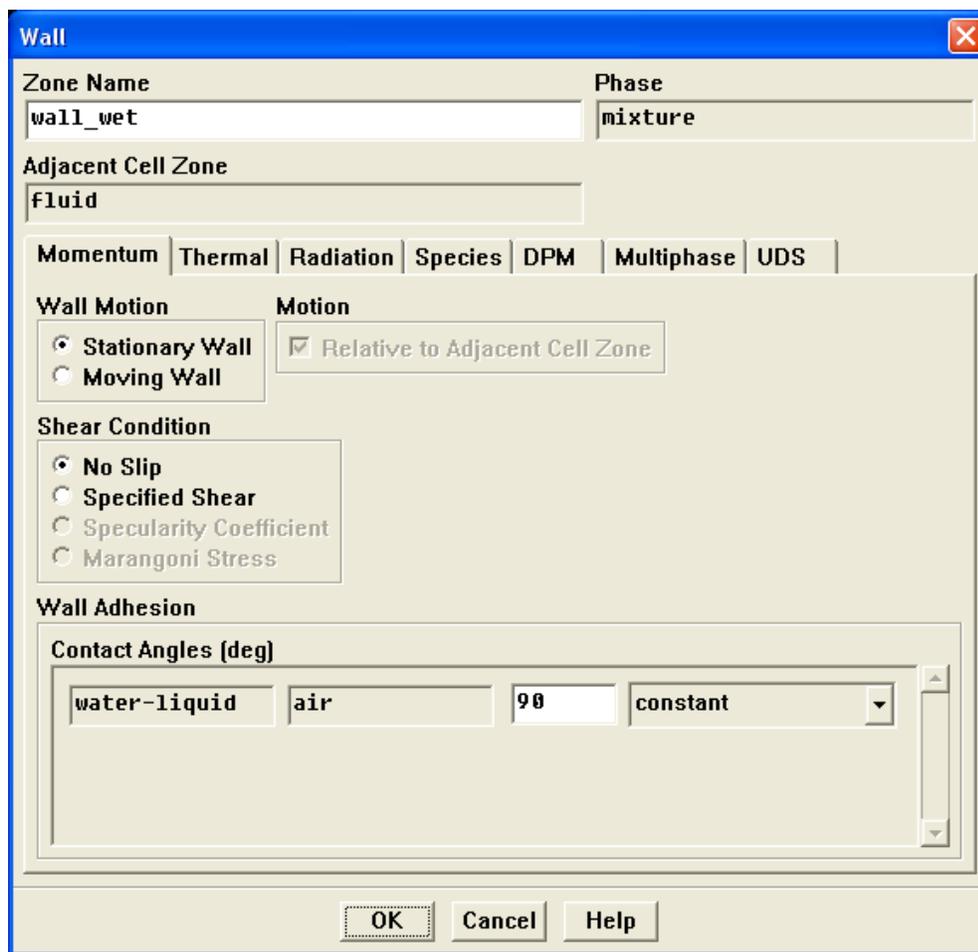
- i. Click the `Multiphase` tab and retain the default setting of 0 for the `Backflow Volume Fraction`.
 - ii. Click `OK` to close the `Pressure Outlet` panel.
4. Set the conditions at the top wall of the air chamber (`wall_no_wet`) for the mixture.
 - (a) Select `wall_no_wet` in the Zone list.
 - (b) Select `mixture` from the Phase drop-down list.

(c) Click the Set... button to open the Wall panel.



- i. Enter 175 degrees in the text-entry field for Contact Angles.
 - ii. Click OK to close the Wall panel.
5. Set the conditions at the side wall of the ink chamber (wall_wet) for the mixture.
- (a) Select wall_wet in the Zone list.
 - (b) Retain the selection of mixture from the Phase drop-down list.

(c) Click the Set... button to open the Wall panel.



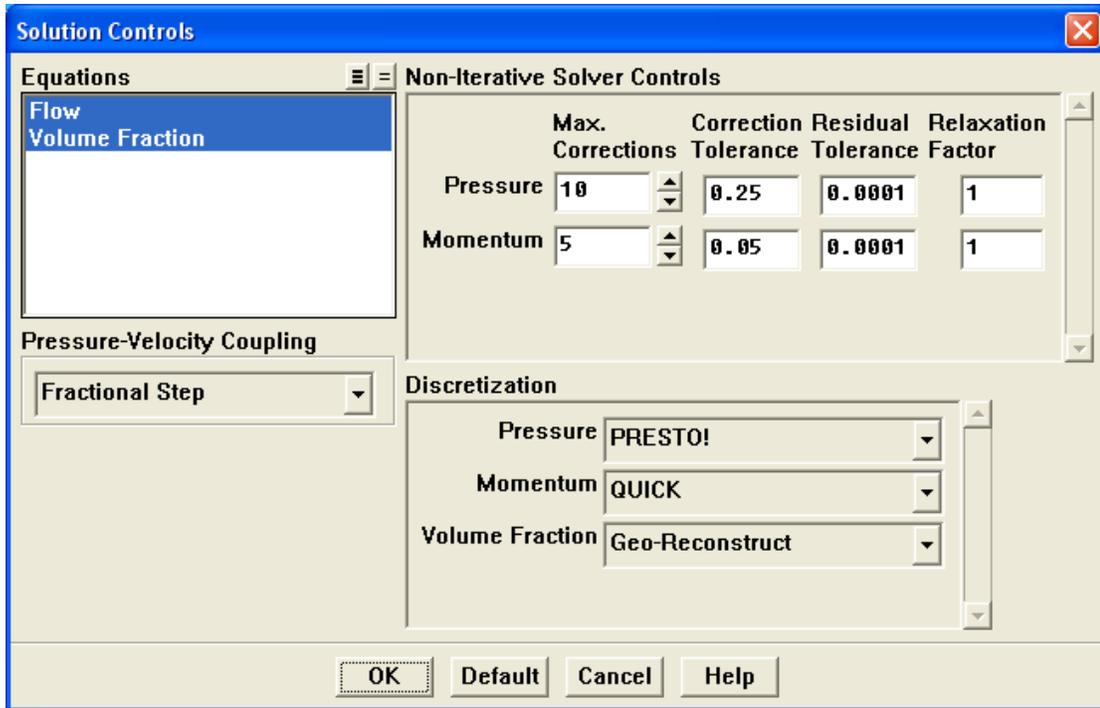
- i. Retain the default setting of 90 degrees in the text-entry field for Contact Angles.
- ii. Click OK to close the Wall panel.

6. Close the Boundary Conditions panel.

Step 8: Solution

1. Set the solution parameters.

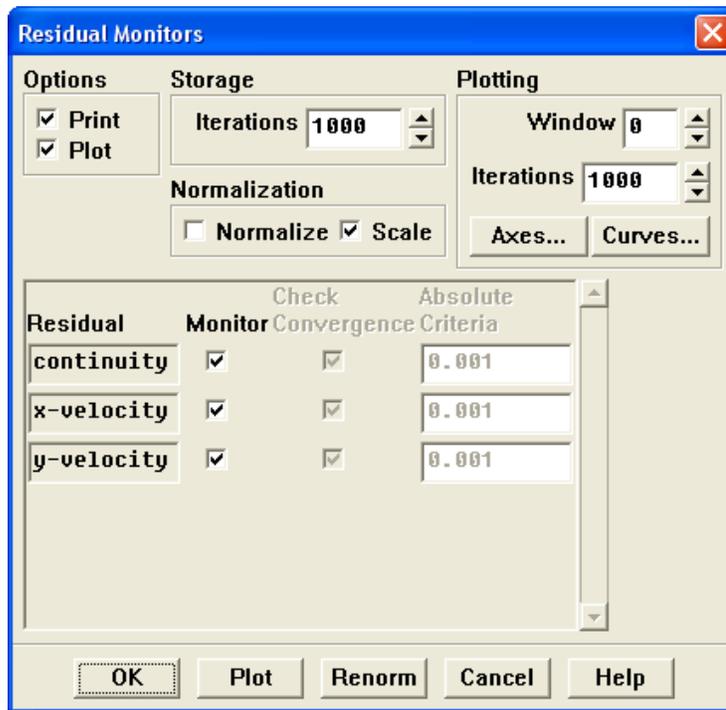
Solve → Controls → Solution...



- (a) Select Fractional Step from the Pressure-Velocity Coupling drop-down list.
- (b) Retain the default selection of PRESTO! in the Pressure drop-down list in the Discretization group box.
- (c) Select QUICK from the Momentum drop-down list.
- (d) Click OK to close the Solution Controls panel.

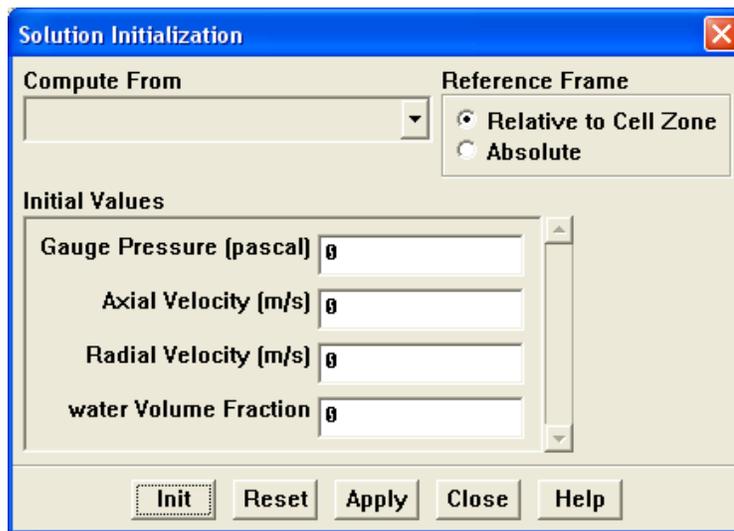
2. Enable the plotting of residuals during the calculation.

Solve → Monitors → Residual...



- (a) Enable Plot in the Options group box.
 - (b) Click OK to close the Residual Monitors panel.
3. Initialize the solution using the default initial values.

Solve → Initialize → Initialize...



- (a) Retain the default settings for all the parameters.
 - (b) Click Init and close the Solution Initialization panel.
4. Define a register for the ink chamber region.

Adapt → Region...



- (a) Retain the default setting of 0 mm for X Min and Y Min in the Input Coordinates group box.
- (b) Enter 0.10 mm for X Max.
- (c) Enter 0.03 mm for Y Max.
- (d) Click Mark.

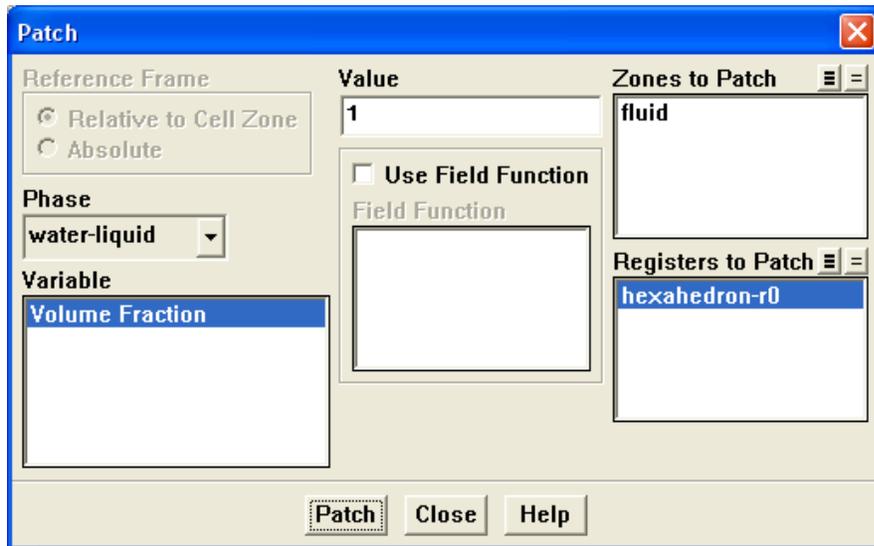
FLUENT will report in the console that 1500 cells were marked for refinement while zero cells were marked for coarsening.

Extra: You can display and manipulate adaption registers, which are generated using the Mark command, using the Manage Adaption Registers panel. Click the Manage... button in the Region Adaption panel to open the Manage Adaption Registers panel.

- (e) Close the Region Adaption panel.

5. Patch the initial distribution of the secondary phase (water-liquid).

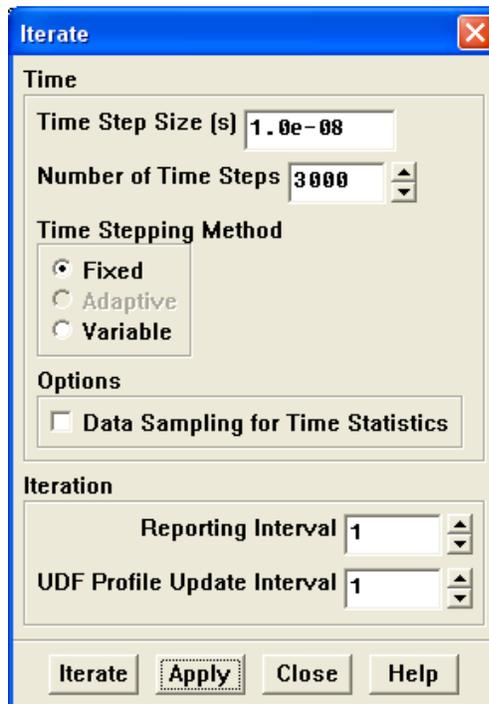
Solve → Initialize → Patch...



- (a) Select water-liquid from the Phase drop-down list.
- (b) Select Volume Fraction from the Variable list.
- (c) Enter 1 for Value.
- (d) Select hexahedron-r0 from the Registers to Patch selection list.
- (e) Click Patch and close the Patch panel.

6. Set the time-stepping parameters.

Solve → Iterate...



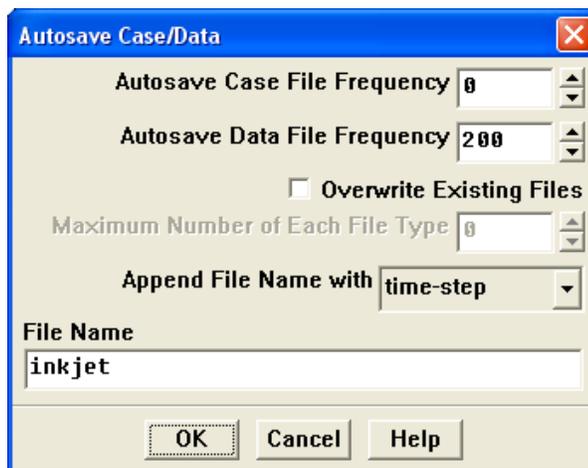
- (a) Enter 1.0e-8 seconds for the Time Step Size.

Note: *Small time steps are required to capture the oscillation of the droplet interface and the associated high velocities. Failure to use sufficiently small time steps may cause differences in the results between platforms.*

- (b) Enter 3000 for the Number of Time Steps.
(c) Retain the default selection of Fixed in the Time Stepping Method list.
(d) Click Apply.

7. Request the saving of data files every 200 steps.

File → Write → Autosave...



- (a) Retain the default setting of 0 for the Autosave Case File Frequency.
- (b) Enter 200 for the Autosave Data File Frequency.
- (c) Make sure that time-step is selected from the Append File Name with drop-down list.
- (d) Enter inkjet for the File Name.

FLUENT will append the time step value to the file name prefix (inkjet). The standard .dat extension will also be appended. This will yield file names of the form inkjet0200.dat, where 200 is the time step number.

Optionally, you can add the extension .gz to the end of the file name (e.g., inkjet.gz), which will instruct FLUENT to save the data files in a compressed format, yielding file names of the form inkjet0200.dat.gz.

- (e) Click OK to close the Autosave Case/Data panel.

8. Save the initial case file (inkjet.cas).

File → Write → Case...

9. Run the calculation.

Solve → Iterate...

- (a) Click Iterate.

The solution will run for 3000 iterations.

- (b) Close the Iterate panel.

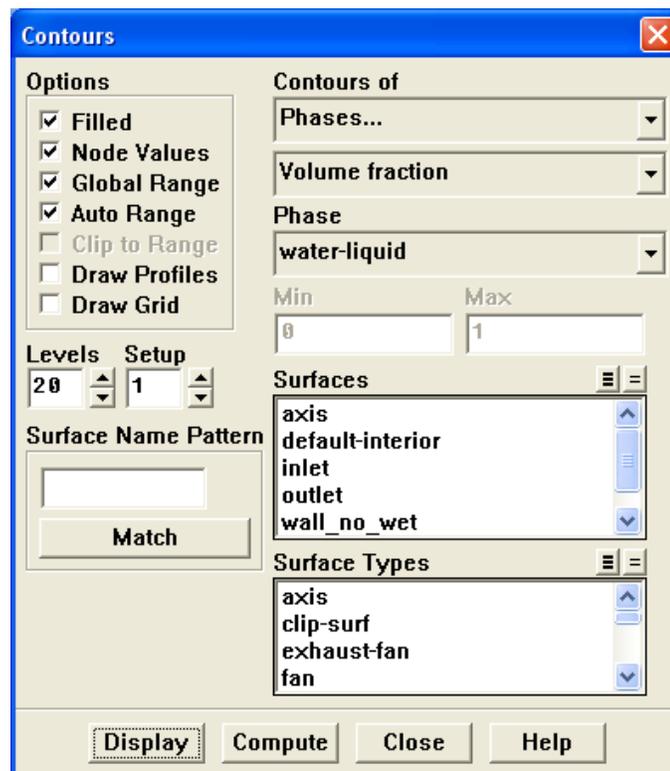
Step 9: Postprocessing

1. Read the data file for the solution after 6 microseconds (inkjet0600.dat).

File → **Read** → Data...

2. Display filled contours of water volume fraction after 6 microseconds (Figure 17.5).

Display → Contours...



- (a) Enable Filled in the Options group box.
 (b) Select Phases... and Volume fraction from the Contours of drop-down lists.
 (c) Select water-liquid in the Phase drop-down list.
 (d) Click Display.

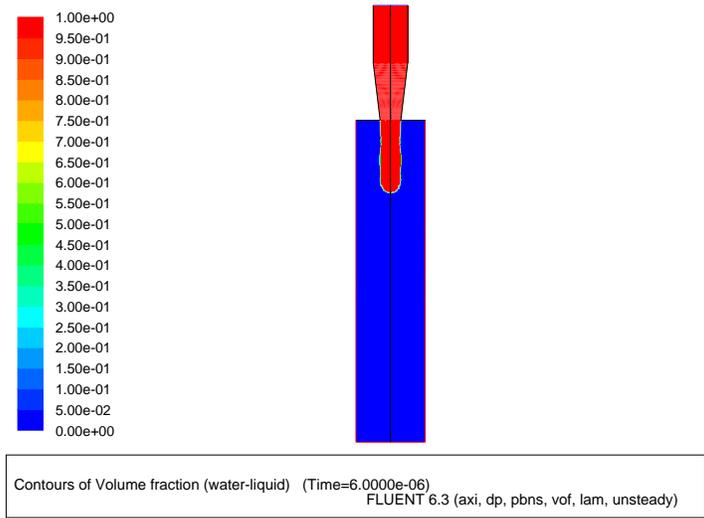


Figure 17.5: Contours of Water Volume Fraction After 6 μ s

3. Similarly, display contours of water volume fraction after 12, 18, 24, and 30 microseconds (Figures 17.6—17.9).

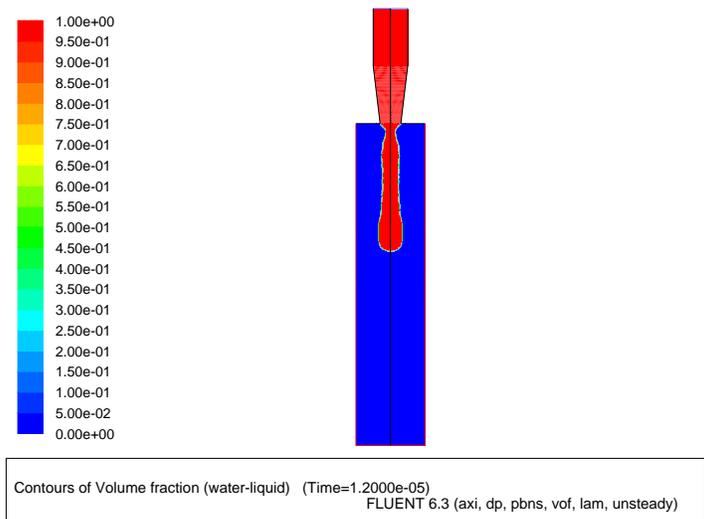


Figure 17.6: Contours of Water Volume Fraction After 12 μ s

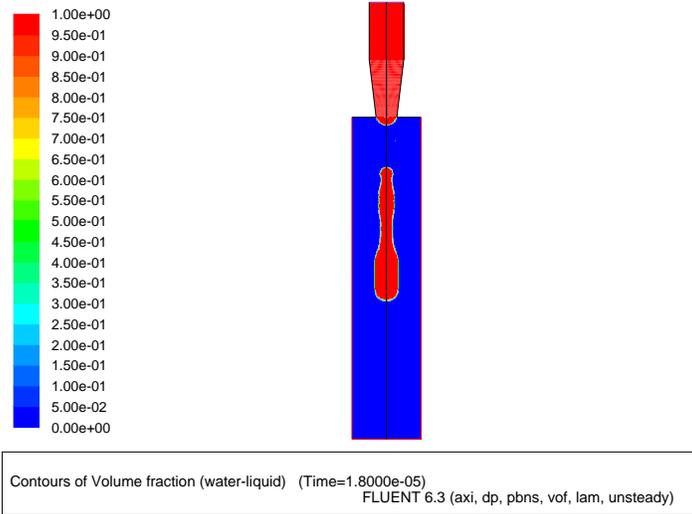


Figure 17.7: Contours of Water Volume Fraction After 18 μs

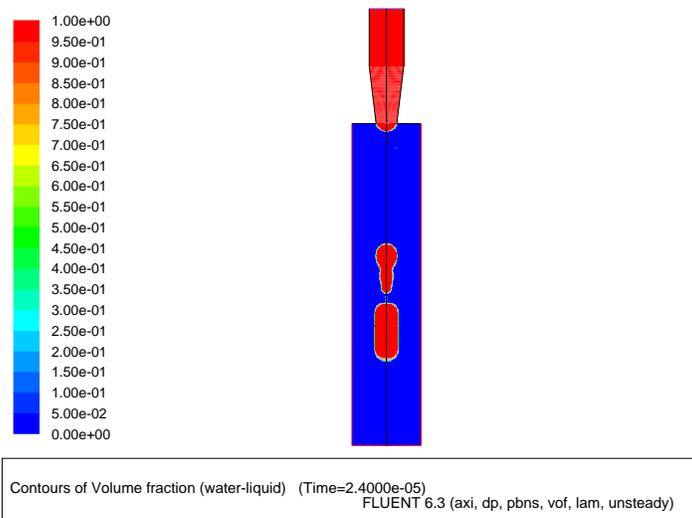


Figure 17.8: Contours of Water Volume Fraction After 24 μs

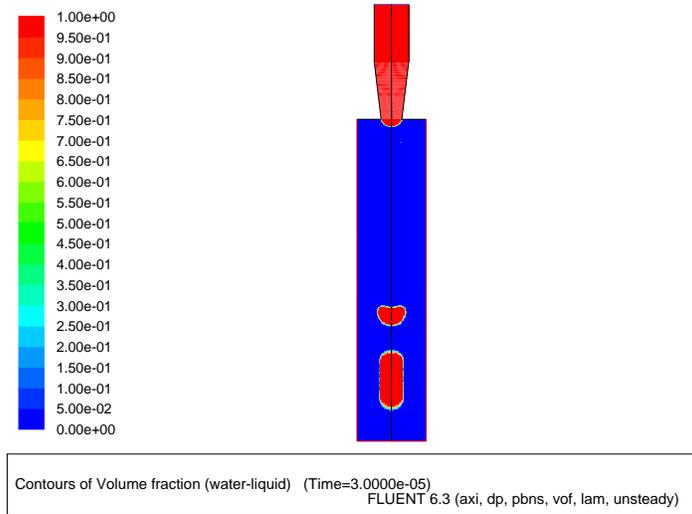


Figure 17.9: Contours of Water Volume Fraction After 30 μs

Summary

This tutorial demonstrated the application of the volume of fluid method with surface tension effects. The problem involved the 2D axisymmetric modeling of a transient liquid-gas interface, and postprocessing showed how the position and shape of the surface between the two immiscible fluids changed over time.

See Section 23.3 of the [User's Guide](#) for additional details about VOF multiphase flow modeling.

Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the grid. Grid adaption can also ensure that the solution is independent of the grid. These steps are demonstrated in [Tutorial 1](#).