

Introduction

This tutorial examines the flow of ink as it is ejected from the nozzle of a printhead in an inkjet printer. Using ANSYS 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 ANSYS 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 18.1](#). The axial symmetry of the problem allows a 2D geometry to be used. The computation mesh consists of 24,600 cells. The domain consists of two regions: an ink chamber and an air chamber. The dimensions are summarized in [Table 18.1](#).

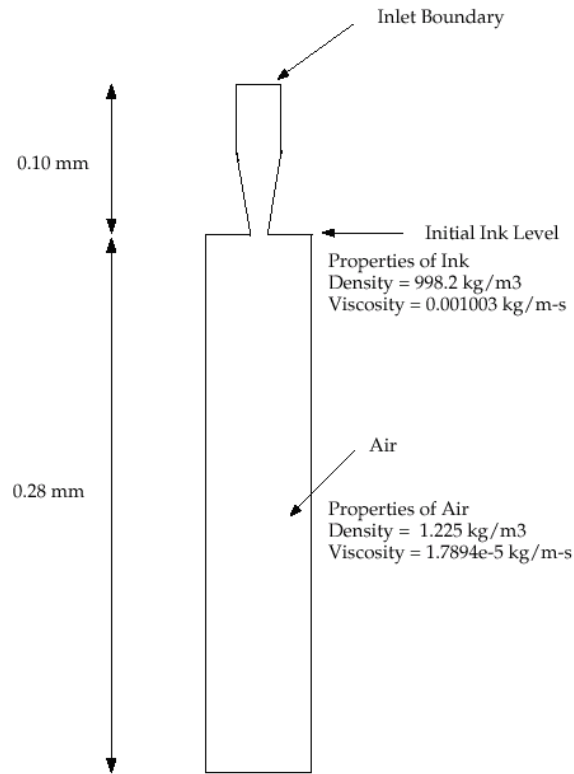


Figure 18.1: Schematic of the Problem

Table 18.1: 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 ANSYS 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 [User Services Center](#) to your working folder (as described in Tutorial 1).
2. Unzip `vof.zip`.

The files `inkjet.msh` and `inlet1.c` can be found in the `vof` folder created on unzipping the file.

3. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.
4. Enable Double-Precision.

For more information about FLUENT Launcher, see Section 1.1.2 in the separate [User's Guide](#).

Note: *The Display Options are enabled by default. Therefore, once you read in the mesh, it will be displayed in the embedded graphics windows.*

Step 1: Mesh

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

File → Read → Mesh..

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

2. Examine the mesh (Figure 18.2).

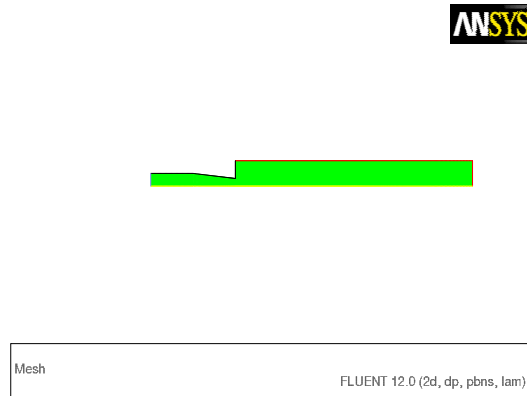


Figure 18.2: Default Display of the Nozzle Mesh

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 18.3).*

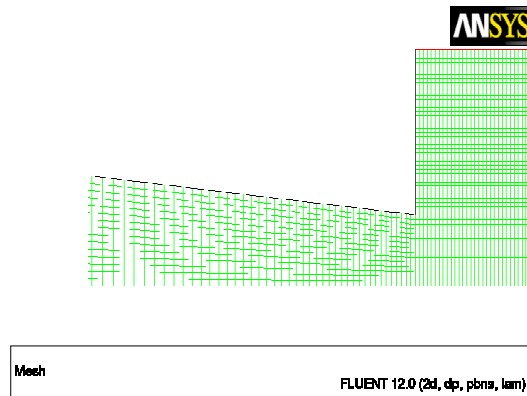
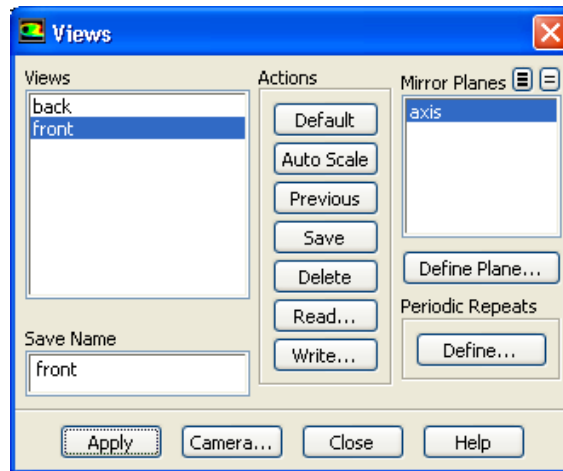


Figure 18.3: The Quadrilateral Mesh

3. Manipulate the mesh display to show the full chamber upright.

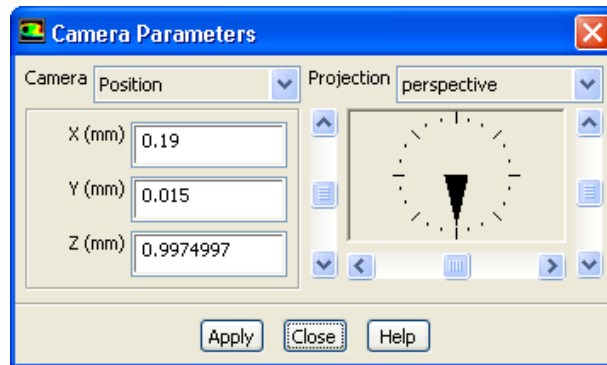
◆ Graphics and Animations → Views...



- (a) Select front from the Views selection list.
- (b) Select axis from the Mirror Planes selection list.
- (c) Click Apply.

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

- (d) Click the Camera... button to open the Camera Parameters dialog box.



- i. Drag the indicator of the dial with the left mouse button in the clockwise direction until the upright view is displayed (Figure 18.4).

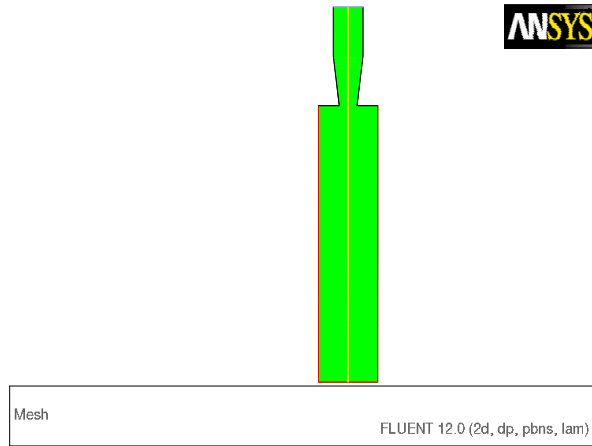


Figure 18.4: Mesh Display of the Nozzle Mirrored and Upright

- ii. Close the Camera Parameters dialog box.
- (e) Close the Views dialog box.

Step 2: General Settings

◆ General

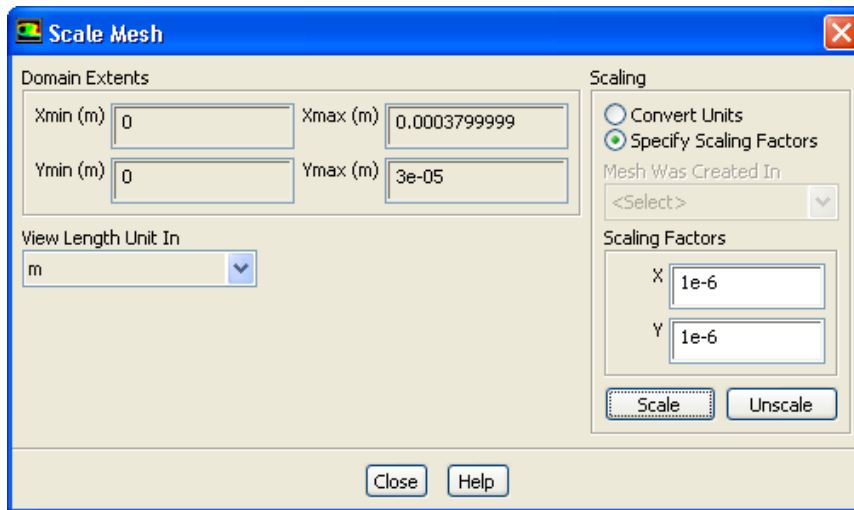
1. Check the mesh.

◆ General → Check

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

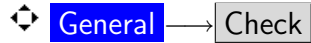
2. Scale the mesh.

◆ General → Scale...



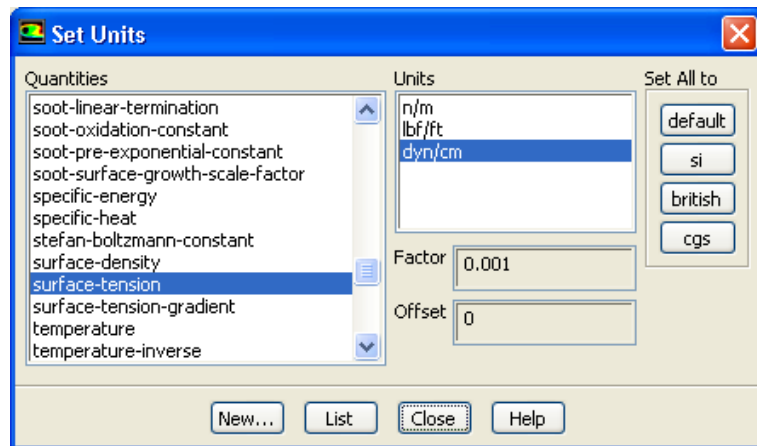
- (a) Select Specify Scaling Factors from the Scaling group box.
- (b) Enter $1e-6$ for X and Y in the Scaling Factors group box.
- (c) Click Scale and close the Scale Mesh dialog box.

3. Check the mesh.



Note: *It is a good idea to check the mesh after you manipulate it (i.e., scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.*

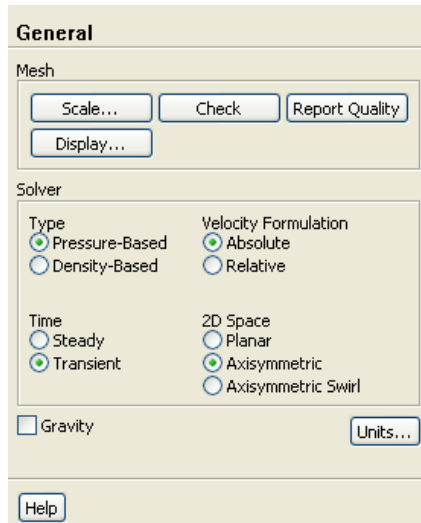
4. Define the units for the mesh.



- (a) Select length from the Quantities list.
- (b) Select mm from the Units list.
- (c) Select surface-tension from the Quantities list.
- (d) Select dyn/cm from the Units list.
- (e) Close the Set Units dialog box.

5. Retain the default setting of Pressure-Based in the Solver list.

◆ **General**



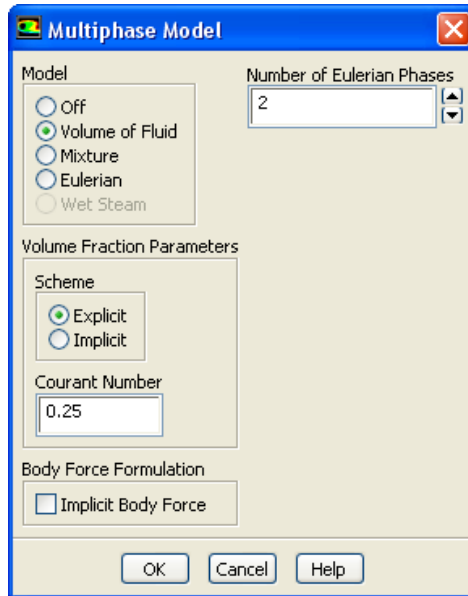
6. Select Transient from the Time list.
7. Select Axisymmetric from the 2D Space list.

Step 3: Models

◆ **Models**

1. Enable the Volume of Fluid multiphase model.

◆ **Models** → **Multiphase** → **Edit...**



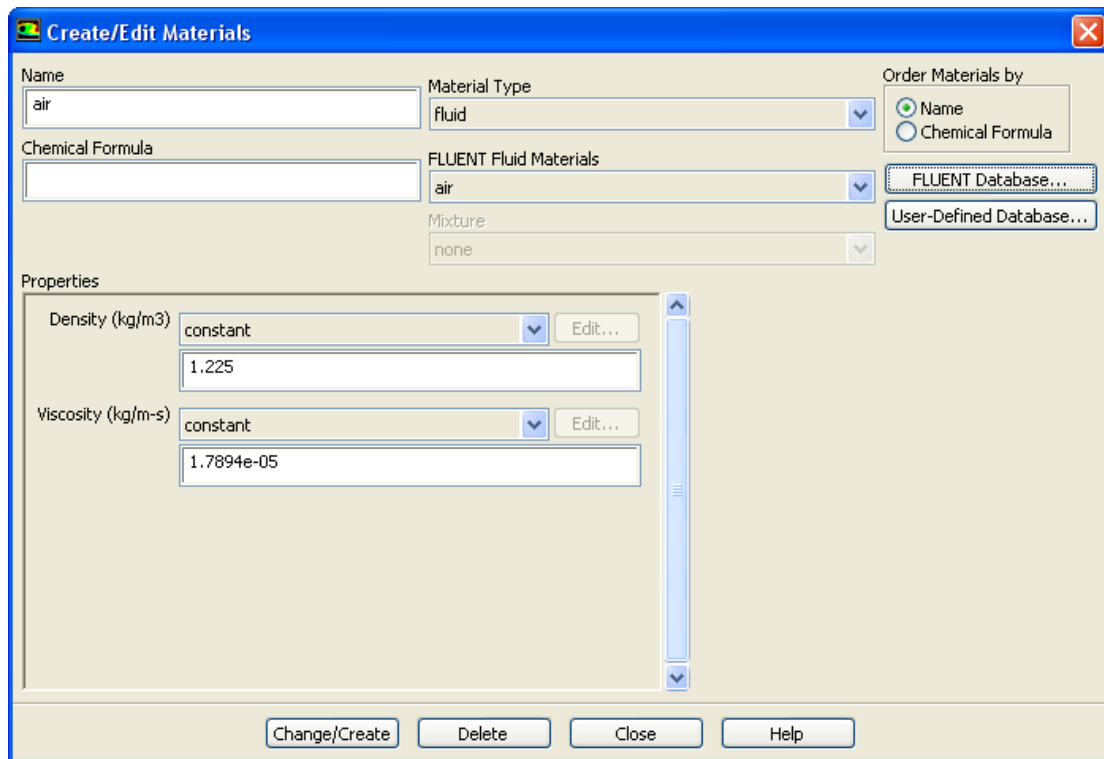
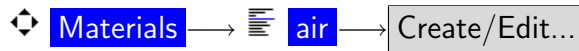
- (a) Select Volume of Fluid from the Model list.
The Multiphase Model dialog box will expand.
- (b) Retain the default settings and click OK to close the Multiphase Model dialog box.

Step 4: Materials

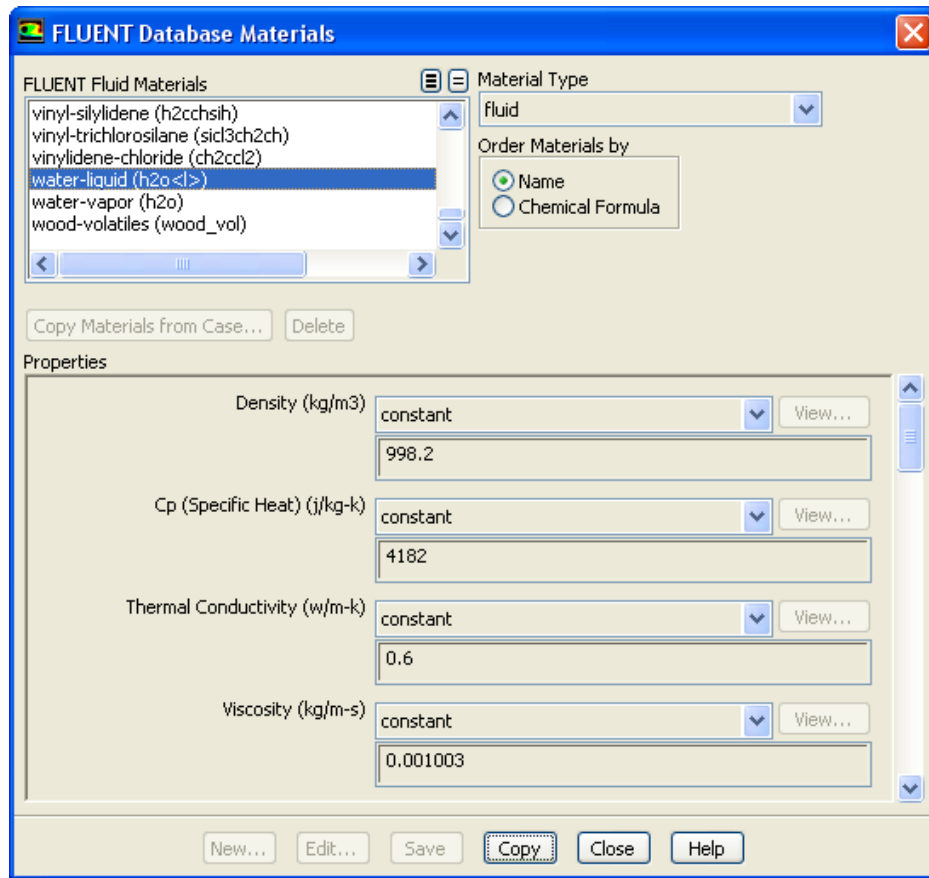
◆ Materials

The default properties of air and water defined in ANSYS 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 ANSYS FLUENT materials database.



- (a) Click FLUENT Database... in the Create/Edit Materials dialog box to open the FLUENT Database Materials dialog box.

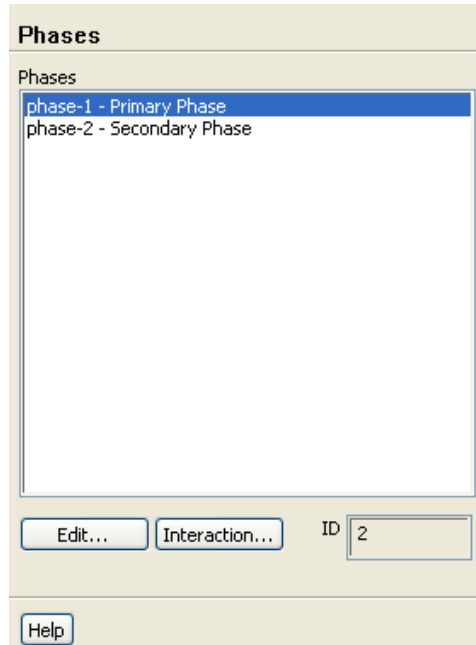


- i. Select water-liquid (h2o<l>) from the FLUENT Fluid Materials selection list.
Scroll down the FLUENT Fluid Materials list to locate water-liquid (h2o<l>).
 - ii. Click Copy to copy the information for water to your list of fluid materials.
 - iii. Close the FLUENT Database Materials dialog box.
- (b) Click Change/Create and close the Create/Edit Materials dialog box.

Step 5: Phases

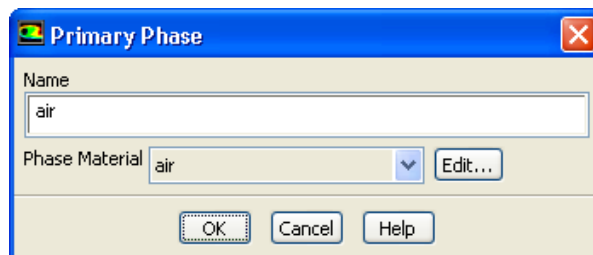
◆ Phases

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.



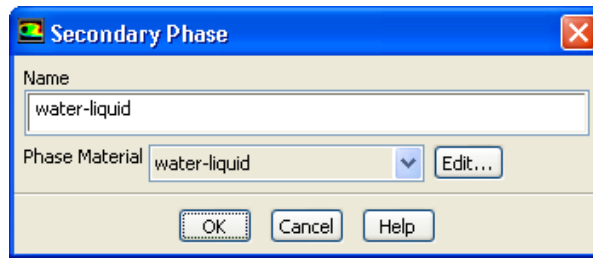
1. Specify air (air) as the primary phase.

◆ Phases → ☰ phase-1 - Primary Phase → Edit...



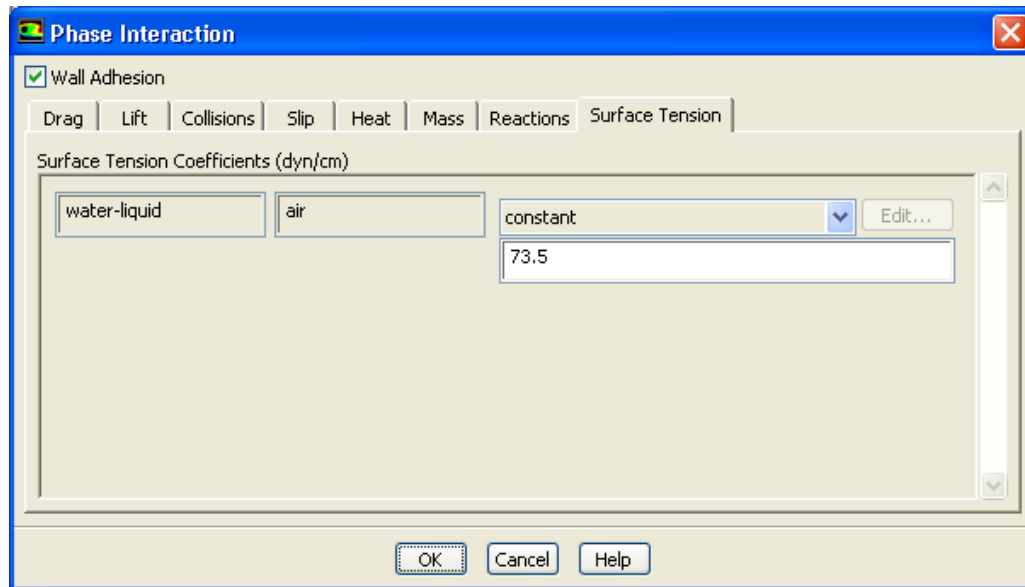
- (a) Enter **air** for Name.
- (b) Retain the default selection of **air** in the Phase Material drop-down list.
- (c) Click **OK** to close the Primary Phase dialog box.

- Specify water (water-liquid) as the secondary phase.



- Enter water-liquid for Name.
- Select water-liquid from the Phase Material drop-down list.
- Click OK to close the Secondary Phase dialog box.

- Specify the interphase interaction.



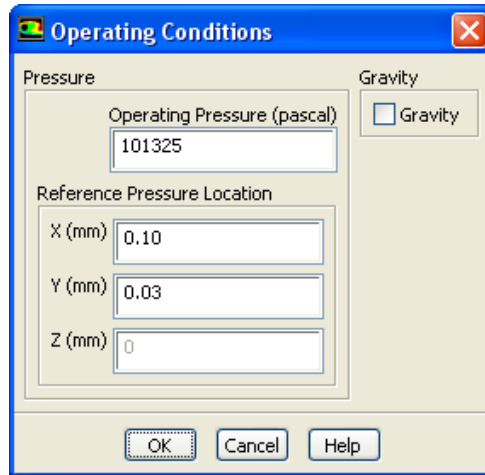
- Enable Wall Adhesion so that contact angles can be prescribed.
- Click the Surface Tension tab.
The surface tension coefficient inputs will be displayed.
 - Select constant from the Surface Tension Coefficient drop-down list.
 - Enter 73.5 dyn/cm for Surface Tension Coefficient.
- Click OK to close the Phase Interaction dialog box.

Step 6: Operating Conditions

◆ Boundary Conditions

1. Set the operating reference pressure location.

◆ **Boundary Conditions** → **Operating Conditions...**

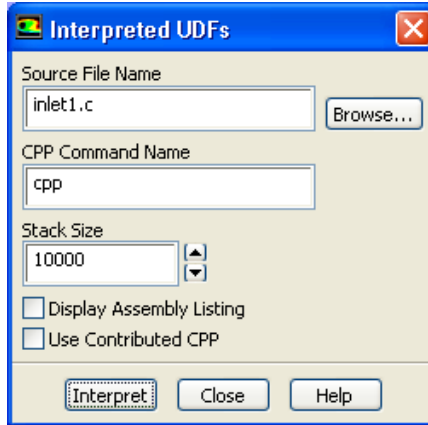
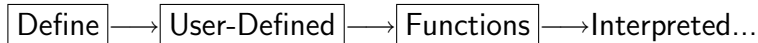


You will set the Reference Pressure Location to be a point where the fluid will always be 100% air.

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

Step 7: User-Defined Function (UDF)

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



- (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 vof 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 dialog boxes.

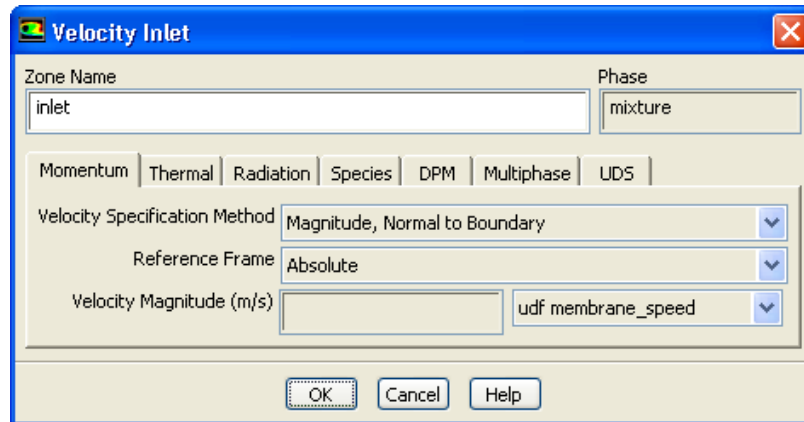
- (c) Close the Interpreted UDFs dialog box.

Step 8: Boundary Conditions

◆ Boundary Conditions


1. Set the boundary conditions at the inlet (inlet) for the mixture by selecting mixture from the Phase drop-down list in the Boundary Conditions task page.

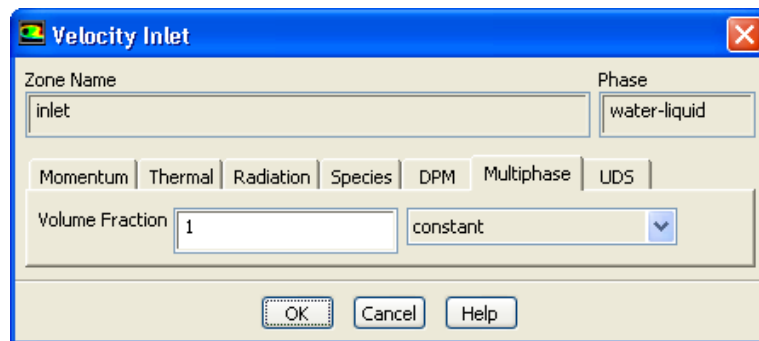
◆ Boundary Conditions →  inlet → Edit...



- (a) Select `udf membrane_speed` from the Velocity Magnitude drop-down list.
- (b) Click OK to close the Velocity Inlet dialog box.

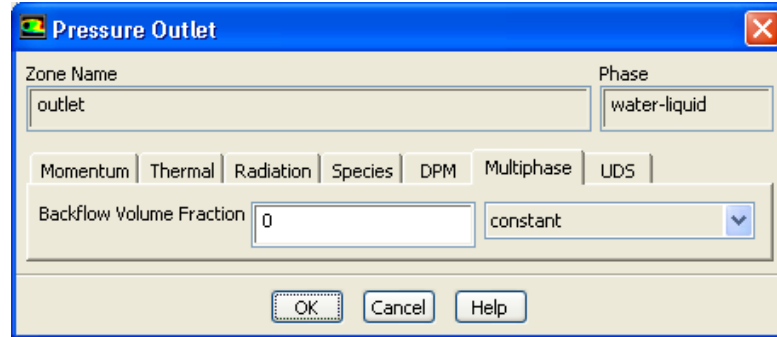
2. Set the boundary conditions at the inlet (inlet) for the secondary phase by selecting water-liquid from the Phase drop-down list in the Boundary Conditions task page.

◆ Boundary Conditions →  inlet → Edit...



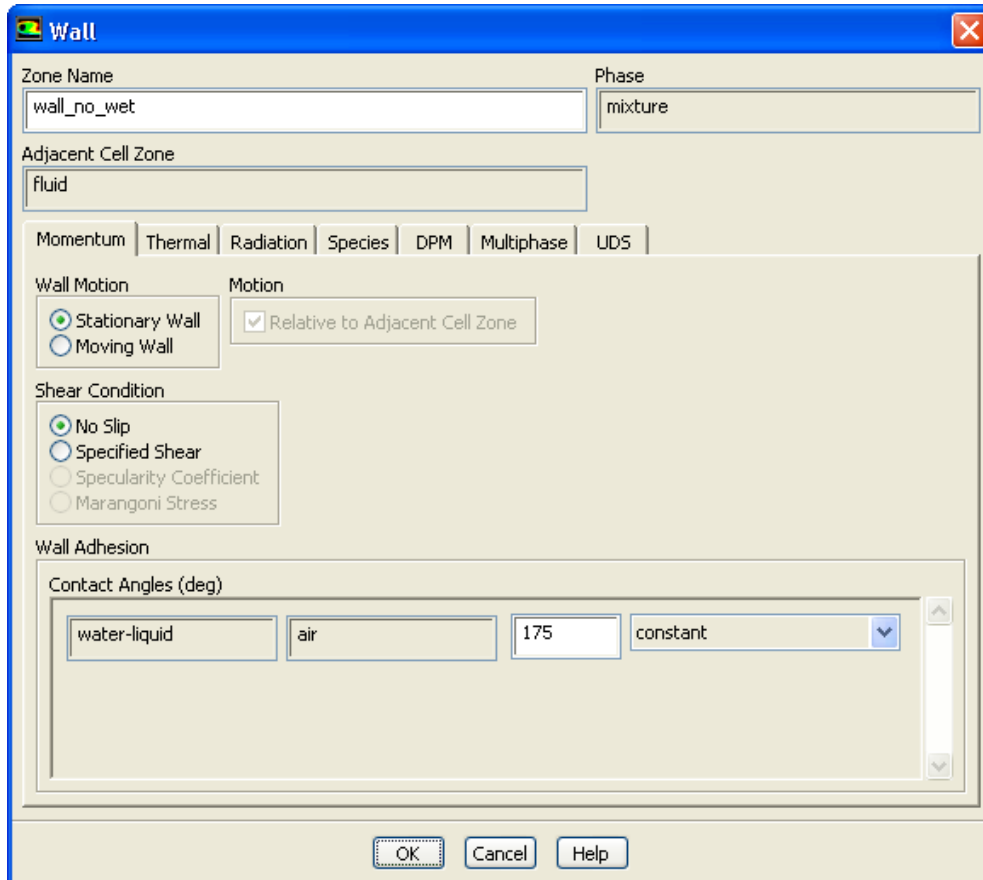
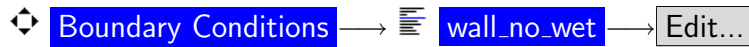
- (a) Click the Multiphase tab and enter 1 for the Volume Fraction.
- (b) Click OK to close the Velocity Inlet dialog box.

- Set the boundary conditions at the outlet (**outlet**) for the secondary phase by selecting **water-liquid** from the Phase drop-down list in the Boundary Conditions task page.



- Click the **Multiphase** tab and retain the default setting of 0 for the **Backflow Volume Fraction**.
- Click **OK** to close the **Pressure Outlet** dialog box.

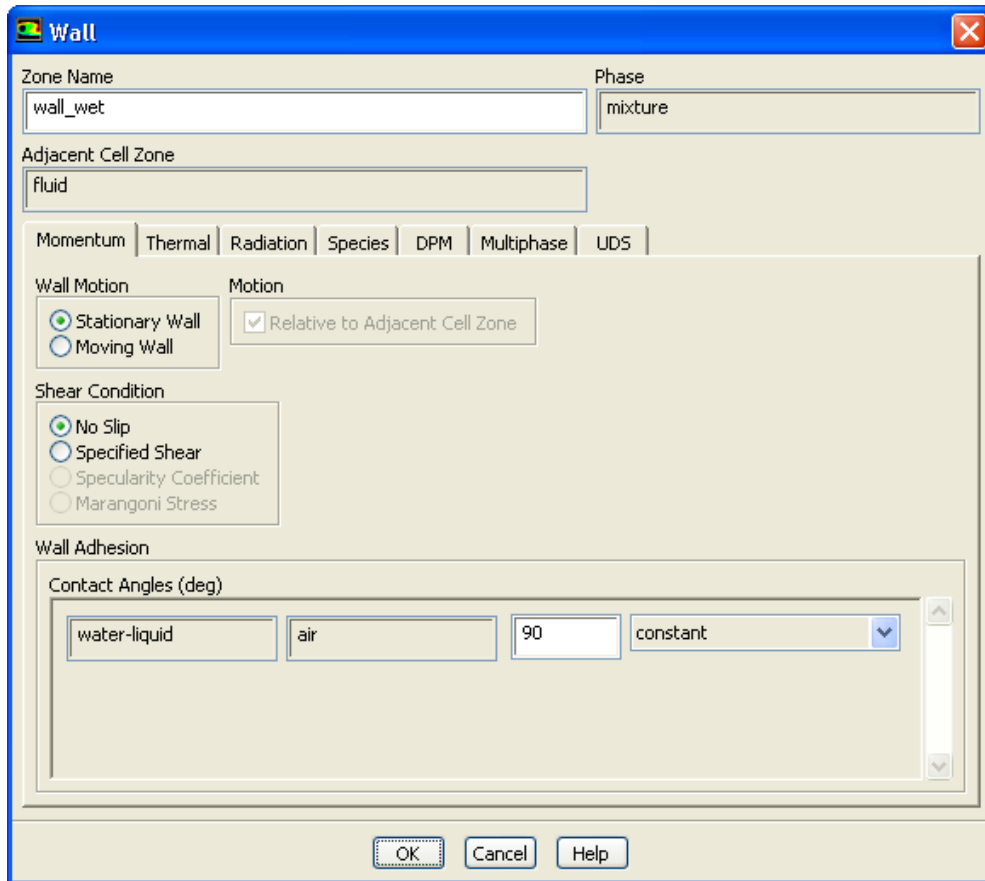
4. Set the conditions at the top wall of the air chamber (wall_no_wet) for the mixture by selecting mixture from the Phase drop-down list in the Boundary Conditions task page.



- Enter 175 degrees for Contact Angles.
- Click OK to close the Wall dialog box.

Note: This angle affects the dynamics of droplet formation. You can repeat this simulation to find out how the result changes when the wall is hydrophilic (i.e., using a small contact angle, say 10 degrees).

5. Set the conditions at the side wall of the ink chamber (wall_wet) for the mixture.



- (a) Retain the default setting of 90 degrees for Contact Angles.
- (b) Click OK to close the Wall dialog box.

Step 9: Solution

1. Set the solution methods.

↻ **Solution Methods**

Solution Methods

Pressure-Velocity Coupling

Scheme
Fractional Step

Spatial Discretization

Gradient
Least Squares Cell Based

Pressure
PRESTO!

Momentum
QUICK

Volume Fraction
Geo-Reconstruct

Transient Formulation

First Order Implicit

Non-Iterative Time Advancement

Frozen Flux Formulation

Default

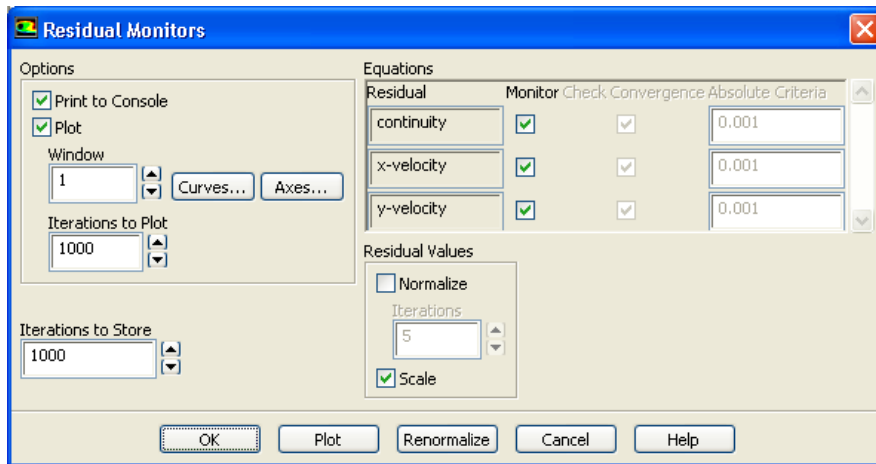
Help

- (a) Enable Non-Iterative Time Advancement.

The non-iterative time advancement (NITA) scheme is often advantageous compared to the iterative schemes as it is less CPU intensive. Although smaller time steps must be used with NITA compared to the iterative schemes, the total CPU expense is often smaller. If the NITA scheme leads to convergence difficulties, then the iterative schemes (e.g. PISO, SIMPLE) should be used instead.

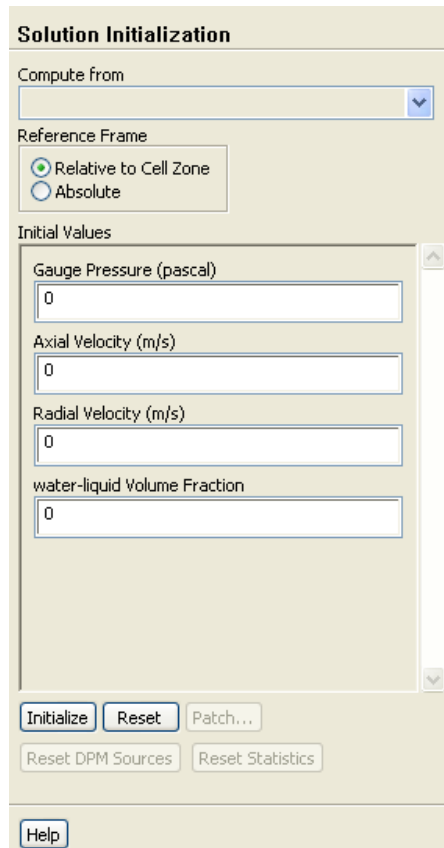
- (b) Select Fractional Step from the Scheme drop-down list in the Pressure-Velocity Coupling group box.
- (c) Retain the default selection of Least Squares Cell Based from the Gradient drop-down list in the Spatial Discretization group box.
- (d) Retain the default selection of PRESTO! from the Pressure drop-down list.
- (e) Select QUICK from the Momentum drop-down list.

2. Enable the plotting of residuals during the calculation.



- (a) Make sure Plot is enabled in the Options group box.
- (b) Click OK to close the Residual Monitors dialog box.

3. Initialize the solution using the default initial values.



- (a) Retain the default settings for all the parameters and click Initialize.
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.

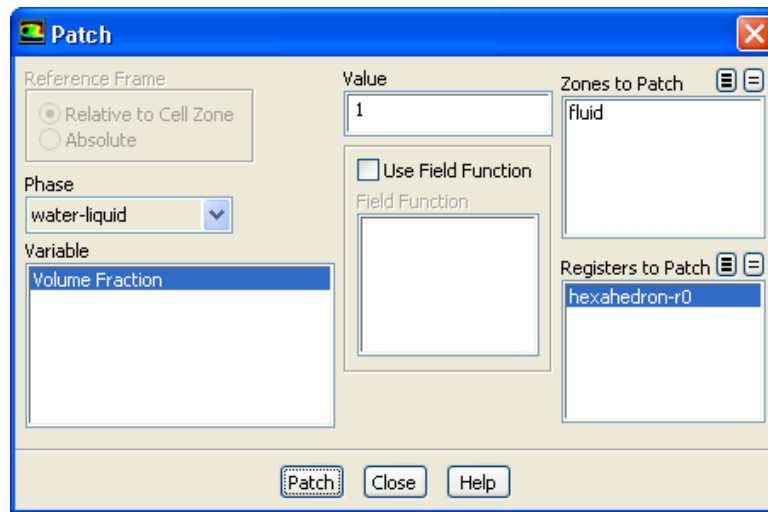
ANSYS FLUENT will report in the console that 1500 cells were marked for refinement while zero cells were marked for coarsening.

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

- (e) Close the Region Adaption dialog box.

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

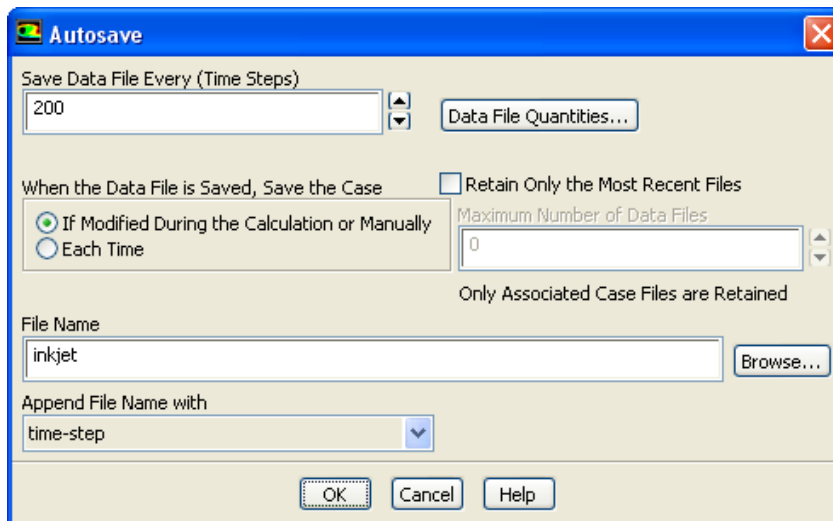
◆ **Solution Initialization** → Patch...



- (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 dialog box.

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

◆ **Calculation Activities** (Autosave Every (Time Steps)) → Edit....



- (a) Enter 200 for Save Data File Every (Time Steps).

- (b) Make sure that **time-step** is selected from the Append File Name with drop-down list.
- (c) Enter **inkjet** for the File Name.

ANSYS 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 inkjet-1-00200.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 ANSYS FLUENT to save the data files in a compressed format, yielding file names of the form inkjet-1-00200.dat.gz.

- (d) Click **OK** to close the Autosave dialog box.

7. Save the initial case file (**inkjet.cas.gz**).

File → **Write** → Case...

8. Run the calculation.

◆ **Run Calculation**

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

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 drop-down list.
- (d) Click Calculate.

The solution will run for 3000 iterations.

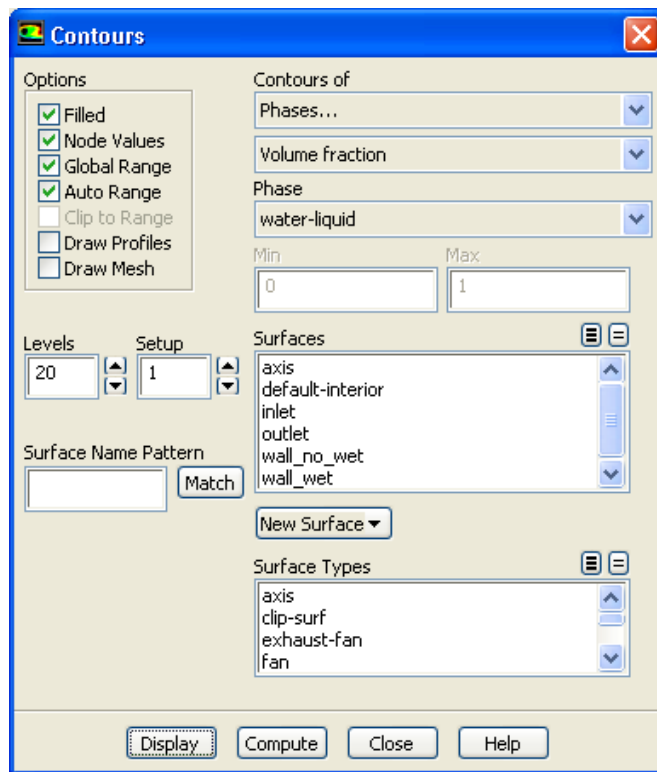
Step 10: Postprocessing

1. Read the data file for the solution after 6 microseconds (inkjet-1-00600.dat.gz).

File → Read → Data...

2. Display filled contours of water volume fraction after 6 microseconds (Figure 18.5).

Graphics and Animations → Contours → Set Up...



- (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 from the Phase drop-down list.
 - (d) Click Display.
3. Similarly, display contours of water volume fraction after 12, 18, 24, and 30 microseconds (Figures 18.6—18.9).

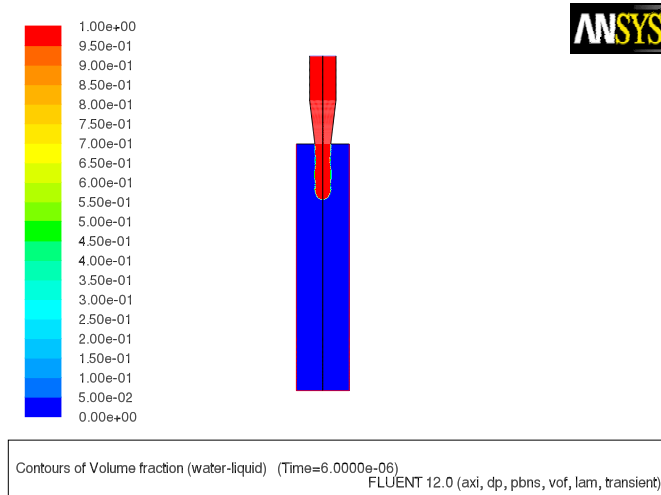


Figure 18.5: Contours of Water Volume Fraction After 6 μs

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.

For additional details about VOF multiphase flow modeling, see Section 16.3 in the separate [Theory Guide](#).

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 mesh. Mesh adaptation can also ensure that the solution is independent of the mesh. These steps are demonstrated in [Tutorial 1](#).

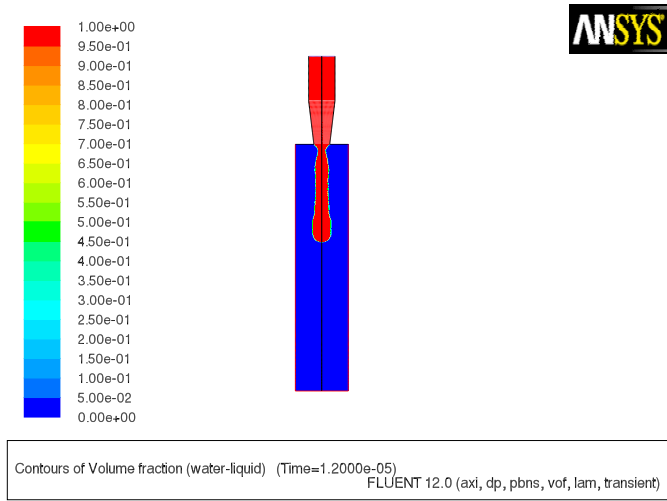


Figure 18.6: Contours of Water Volume Fraction After 12 μ s

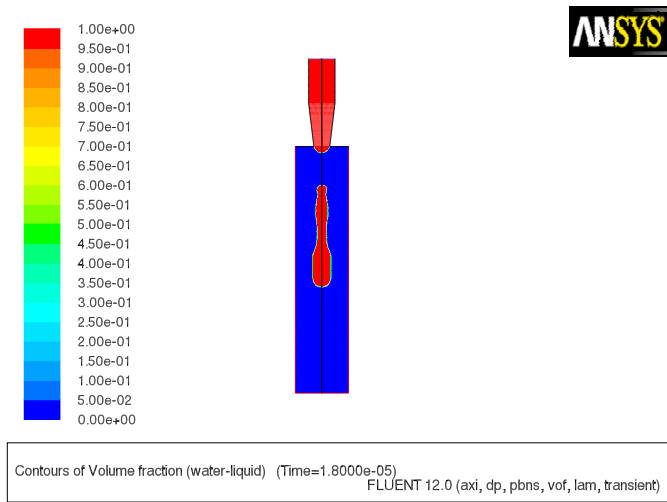


Figure 18.7: Contours of Water Volume Fraction After 18 μ s

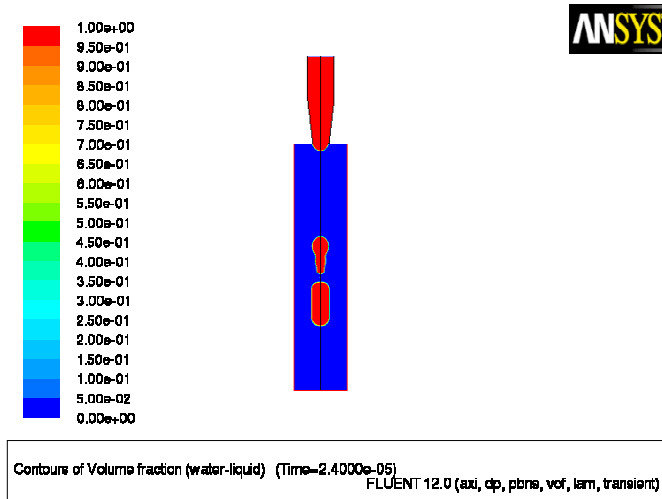


Figure 18.8: Contours of Water Volume Fraction After 24 μ s

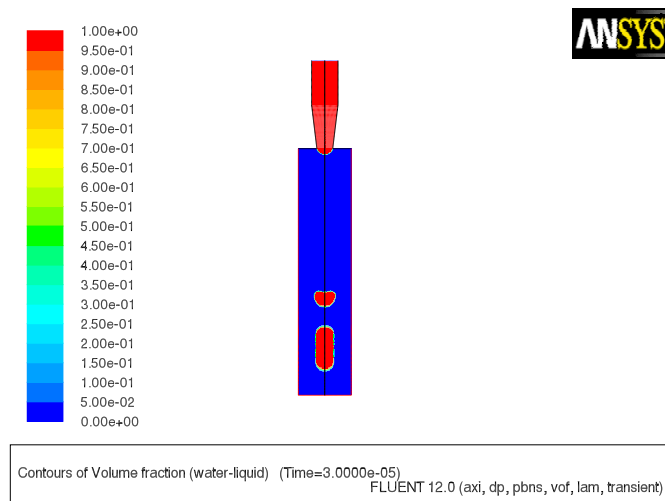


Figure 18.9: Contours of Water Volume Fraction After 30 μ s

