Tutorial: Drop Ejection from a Printhead Nozzle

Purpose

The purpose of this tutorial is to provide guidelines during the transient simulation of drop ejection from the nozzle of the printhead in an ink jet printer. The volume of filling is used to predict the droplet shape. The time dependent boundary condition requires a user-defined function (UDF).

Prerequisites

This tutorial assumes that you are familiar with the FLUENT interface, and have a good understanding of basic setup and solution procedures. In this tutorial you will use the volume of fluid (VOF) multiphase model, so you should have some experience with it. This tutorial will not cover the mechanics of using this model; instead, it will focus on the application of this model to the drop ejection from a printhead nozzle.

If you have not used this feature before, it would be helpful to first review the Section 24.2 of FLUENT 6.2 User's Guide and Tutorial 16: Using the VOF Model of FLUENT 6.2 Tutorial Guide. If you have not used UDFs, it would be helpful to review the FLUENT 6.2 UDF Manual.

Problem Description

The problem to be solved in this tutorial is shown in Figure 1.

To capture the capillary effect of the ejected ink, the surface tension and prescription of the wetting angle are specified. The surface inside the nozzle is neutrally wettable, while the surface surrounding the nozzle orifice is non-wettable.

At time zero, ink fills the nozzle. 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 suddenly rises from 0 to 3.58 m/s and drops according to a cosine law. A user defined subroutine is provided in the Appendix for reference. 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.

Gravity is not included in the simulation. Due to the axial symmetry of the problem a 2D geometry is 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 the Table 1.

Table 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

Because the dimensions are small, the double precision version of FLUENT is used. The primary phase is air and the secondary phase is water-liquid. Patching is required to fill the ink chamber with the secondary phase.

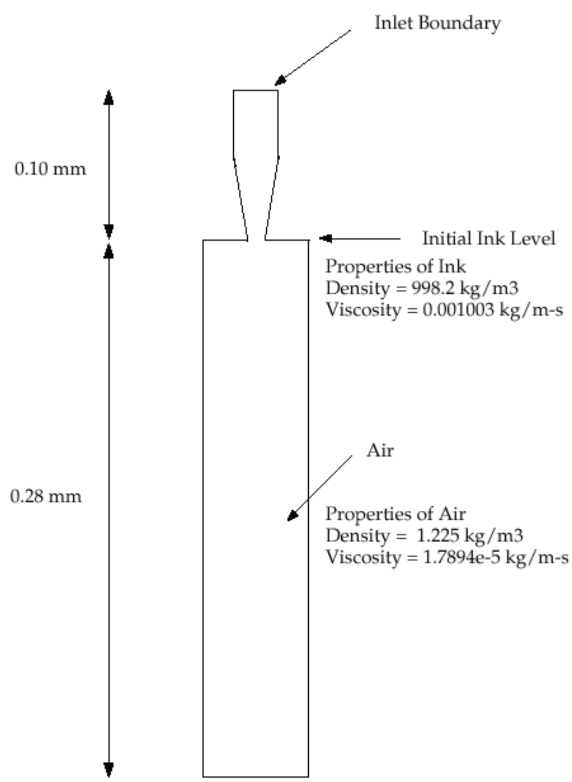

Figure 1: Schematic of the Problem

Preparation

1. Copy the files, `inkjet.msh` and `inlet1.c` to your working directory.
2. Start FLUENT 2DDP.

Setup and Solution

Step 1: Grid

1. Read in the mesh file, `inkjet.msh`.
2. Check the grid.
3. Scale the grid.
   (a) Scale the grid using `1e-6` as the scale factors for X and Y.
4. Define the units for the grid.
   (a) Set mm as the units of length.
   (b) Set dyn/cm as the units of surface-tension.
5. Display the grid.
6. Manipulate the grid display to show the full chamber upright.
   (a) Mirror the view across the axis.
      i. Under Mirror Planes, select axis and click Apply.
         The grid display will be updated to show both sides of the chamber.
   (b) Click on Camera... to rotate the model by 90 degrees to the left.
      The grid display of the chamber appears upright in the graphics window (Figure 2).

Step 2: Models

1. Define the solver settings.
   (a) Under Space, enable Axisymmetric.
   (b) Under Time, enable Unsteady.
2. Enable the Volume of Fluid multiphase model and accept the default settings.

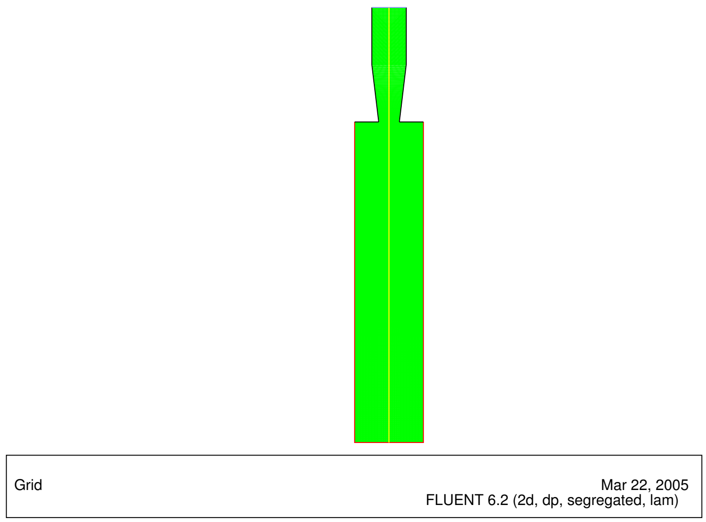

Figure 2: Grid Display of the Full Chamber

Step 3: Materials

The default properties of water and air are the correct values for this problem. You can check them in the Materials *panel.*

1. Define the material properties.
   (a) Accept the default settings for material air.
   (b) Define the new material by copying water-liquid (h20<l>) from the Fluent Database.
2. Define the primary (air) and secondary (water-liquid) phases.
   (a) Specify air as the primary phase and name it as `air`.
   (b) Specify water-liquid as the secondary phase and name it as `water-liquid`.
   (c) Specify the interphase interaction.
      i. In the Phase Interaction panel, enable Wall Adhesion so that contact angles can be prescribed.
      ii. Click the Surface Tension tab.
         The surface tension coefficient inputs will be displayed.
         A. In the drop-down list for Surface Tension Coefficient, select constant.
         B. Enter `73.5` dyn/cm for the Surface Tension Coefficient.

Step 4: Operating Conditions

1. Set the operating reference pressure location.

   Set the Reference Pressure Location *at a point where the fluid will always be 100% air.*

   (a) Enter `0.10` mm for X and `0.03` mm for Y.

Step 5: User-Defined Function (UDF)

1. Compile the UDF, inlet1.c, using the Interpreted UDFs panel.

   (a) Enter `inlet1.c` under Source File Name.

   Make sure that the C source code for your UDF and your mesh file reside in your working directory. If your source code is not in your working directory, then when you compile the UDF you must enter the file's complete path in the Interpreted UDFs *panel, instead of just the filename. For doing this, you should click on the* Browse... *button. This will open the* Select File *dialog box that enables you to choose a UDF for reading. You can use it to look at your system directories and select a UDF.*

   (b) Click Interpret to compile your UDF.

Step 6: Boundary Conditions

1. Set the boundary conditions for inlet.

   (a) Select udf membrane_speed in the Velocity Magnitude drop-down list.

   (b) In the Boundary Conditions panel, change the Phase to water-liquid and click the Set... button.

   (c) Set the water-liquid Volume Fraction to `1`.

2. Set the conditions for outlet.

   (a) Retain zero as the Backflow Volume Fraction.

3. Set the conditions for wall_no_wet.

   (a) In the Boundary Conditions panel, change the Phase to mixture and click the Set... button.

   (b) Input `175` degrees as the value under Contact Angles.

4. Set the conditions for wall_wet.

   (a) Retain `90` degrees as the value under Contact Angles.

Step 7: Solution

1. Set the solution parameters.
   (a) Set all Under-Relaxation Factors, except Momentum, to 1. Retain the default value of 0.7 for Momentum.
   (b) Under Discretization, set Pressure to PRESTO! and Momentum to Second Order Upwind.
   (c) Set Pressure-Velocity Coupling to PISO.
2. Enable the plotting of residuals during the calculation.
   (a) Under Options, select Plot.
   (b) Tighten the Convergence Criterion by a factor of 10 for all equations.
3. Initialize the gauge pressure, flow field, and water-liquid volume fraction to 0.
4. Define a register for the ink chamber region.
   (a) Enter 0 for X Min and 0.10 for X Max.
   (b) Enter 0 for Y Min and 0.03 for Y Max.
   (c) Click Mark.

   You can display and manipulate adaption registers, which are generated using the Mark *command, in the* Manage Adaption Registers *panel, which is opened by clicking on* Manage... *in the* Region Adaption *panel.*
5. Patch the initial distribution of the water-liquid.
   (a) In the drop-down list for Phase, select water-liquid.
   (b) In the Variable list, select Volume Fraction.
   (c) Select hexahedron-r0 in the Registers To Patch list.
   (d) Set the Value to 1.
   (e) Click Patch.
6. Set the time-stepping parameters.
   (a) Set the Time Step Size to 2.0e-8 seconds.
   (b) Set the Number of Time Steps to 1500.
   (c) Use the Fixed time stepping method.
   (d) Click Apply.
7. Request saving of data files every 100 steps.
8. Save the initial case and data files (`inkjet.gz`).
9. Run the calculation.

Step 8: Postprocessing

1. Read the data file, `inkjet0300.dat`.
   (a) Display filled contours of water volume fraction after six microseconds (Figure 3).
      i. Under Options, select Filled.
      ii. Select Phases... and Volume fraction in the Contours Of drop-down lists
      iii. Select water-liquid in the Phase drop-down list and click Display.

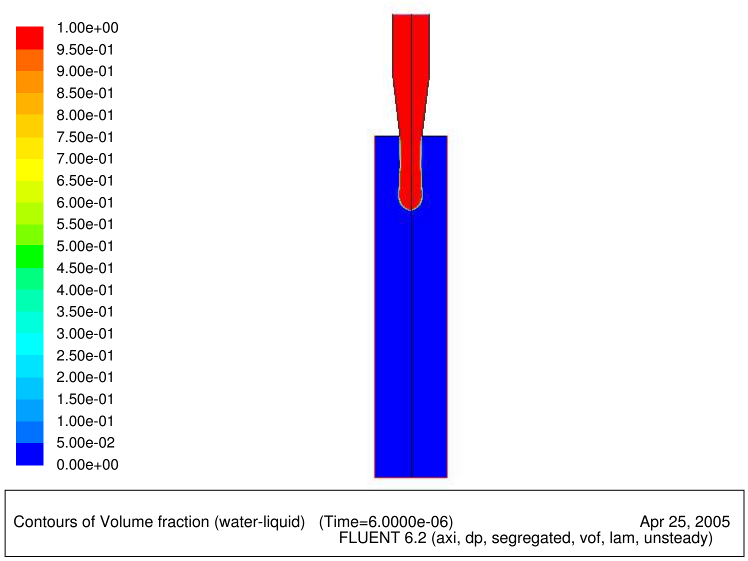

Figure 3: Contours of water-liquid Volume Fraction after 6 ms

2. Similarly, display contours of water volume fraction after 12, 18, 24, and 30 microseconds (Figures 4-7).

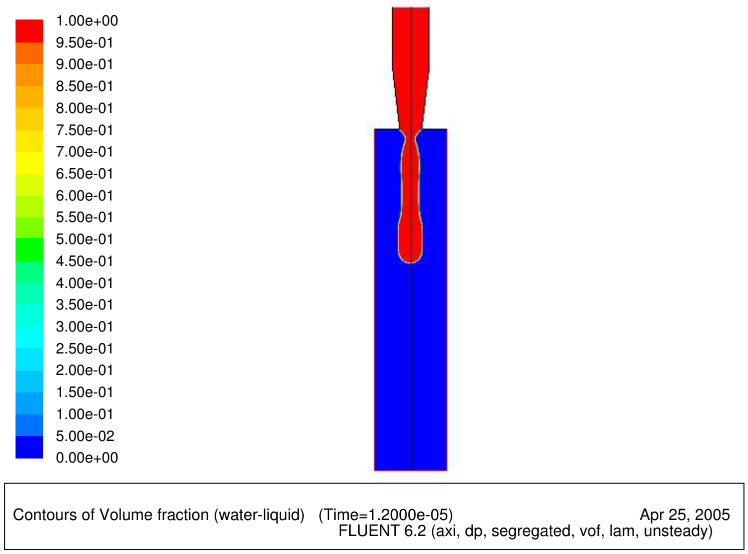

Figure 4: Contours of water-liquid Volume Fraction after 12 ms

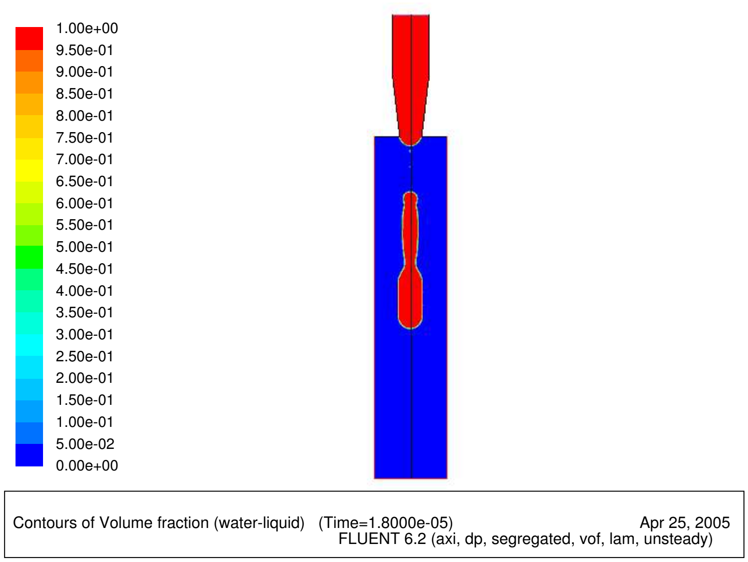

Figure 5: Contours of water-liquid Volume Fraction after 18 ms

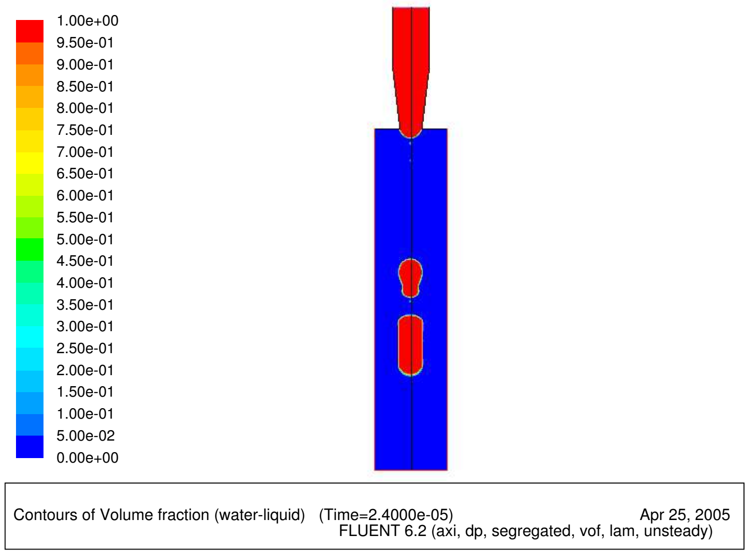

Figure 6: Contours of water-liquid Volume Fraction after 24 ms

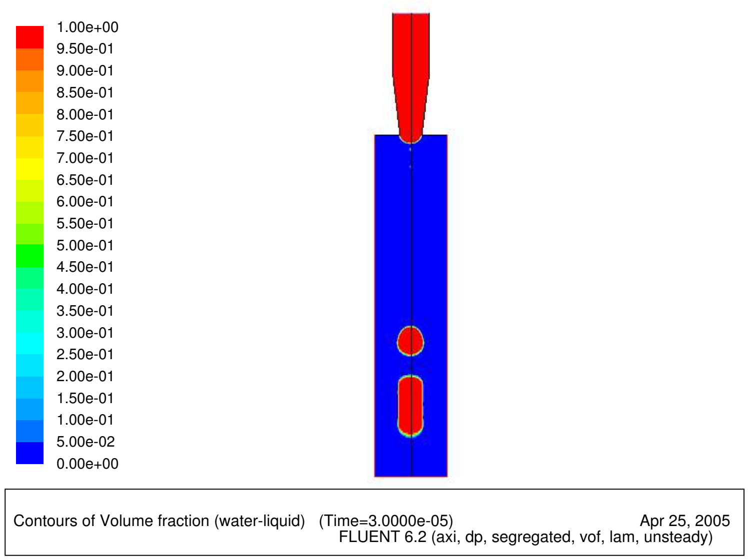

Figure 7: Contours of water-liquid Volume Fraction after 30 ms

Results

The VOF model in FLUENT is able to adequately predict the formation and development of an ink droplet ejected from the printhead of an ink jet printer.

Summary

Application of the volume of fluid method with surface tension effects has bee demonstrated. The FLUENT calculation agrees well with the prediction published in the literature using the same injection scenario. Noticeably good agreement was achieved in predicting the number and volume of the ink droplets. Because these parameters are crucial to the print quality, it can be concluded that the FLUENT VOF model can be used in the design of ink jet printheads. (Reference: W. J. Rider, D.B. Kothe, E.G. Pucket, I. D. Aleinov "Accurate and robust methods for variable density incompressible flow with discontinuities", 1996, Proc. of ICASE/LaRC workshop on barriers and challenges in CFD, NASA Langley Research Center, Hampton, Virginia, August 5-7, M. Salas (Ed.), Kluwer Academic Publishers (in press).

Appendix

inlet1. c

```
#include "udf.h"
  #include "sg.h"
  #include "sg_mphase.h"
  #include "flow.h"

  #define PI 3.141592654
  DEFINE_PROFILE(membrane_speed,        /* function name   */
                 th    ,                /* thread          */
                 nv)                    /* variable number */
  {
     face_t f;
     real x[ND_ND];
     real f_time = RP_Get_Real("flow-time");
     begin_f_loop (f,th)
        {
            F_CENTROID(x,f,th);
            if (f_time<=10e-6)
            {F_PROFILE(f,th,nv) = 3.58*cos(PI*f_time/30e-6);
            }
            else F_PROFILE(f,th,nv) = 0;
        }
     end_f_loop (f,th)
 }
```