Inkjet Nozzle—Level Set

Introduction

Inkjet printers are attractive tools for printing text and images because they combine low cost and high resolution with acceptable speed. The working principle behind inkjet technology is to eject small droplets of liquid from a nozzle onto a sheet of paper. Important properties of a printer are its speed and the resolution of the final images. Designers can vary several parameters to modify a printer's performance. For instance, they can vary the inkjet geometry and the type of ink to create droplets of different sizes. The size and speed of the ejected droplets are also strongly dependent on the speed at which ink is injected into the nozzle. Simulations can be useful to improve the understanding of the fluid flow and to predict the optimal design of an inkjet for a specific application.

Although initially invented to produce images on paper, the inkjet technique has since been adopted for other application areas. Instruments for the precise deposition of microdroplets often employ inkjets. These instruments are used within the life sciences for diagnosis, analysis, and drug discovery. Inkjets have also been used as 3D printers to synthesize tissue from cells and to manufacture microelectronics. For all of these applications it is important to be able to accurately control the inkjet's performance.

This example demonstrates how to model the fluid flow within an inkjet using the Laminar Two-Phase Flow, Level Set user interface.

Model Definition

Figure 1 shows the geometry of the inkjet studied in this example. Because of its symmetry you can use an axisymmetric 2D model. Initially, the space between the inlet and the nozzle is filled with ink. Additional ink is injected through the inlet during a period of 10 μs, and it consequently forces ink to flow out of the nozzle. When the injection stops, a droplet of ink snaps off and continues to travel until it hits the target.

©2013 COMSOL

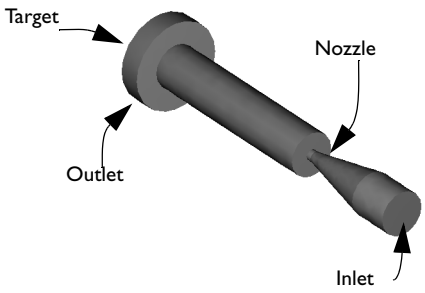

Figure 1: Geometry of the inkjet.

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

Level Set Method

The Laminar Two-Phase Flow, Level Set user interface uses a reinitialized, conservative level set method to describe and convect the fluid interface. The 0.5 contour of the level set function ϕ defines the interface, where ϕ equals 0 in air and 1 in ink. In a transition layer close to the interface, ϕ goes smoothly from 0 to 1. The interface moves with the fluid velocity, $\mathbf{u}$, at the interface. The following equation describes the convection of the reinitialized level set function:

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \mathbf{u}) + \gamma \left[\left(\nabla \cdot \left(\phi(1-\phi) \frac{\nabla \phi}{|\nabla \phi|} \right) \right) - \varepsilon \nabla \cdot \nabla \phi \right] = 0$$

The thickness of the transition layer is proportional to ε. For this model you can use $\varepsilon = h_c/2$, where h_c is the typical mesh size in the region passed by the droplet.

The parameter γ determines the amount of reinitialization. A suitable value for γ is the maximum magnitude occurring in the velocity field.

Beside defining the fluid interface, the level set function is used to smooth the density and viscosity jumps across the interface through the definitions

$$\rho = \rho_{\text{air}} + (\rho_{\text{ink}} - \rho_{\text{air}})\phi$$
$$\mu = \mu_{\text{air}} + (\mu_{\text{ink}} - \mu_{\text{air}})\phi$$

 ©2013 COMSOL

TRANSPORT OF MASS AND MOMENTUM

The incompressible Navier-Stokes equations, including surface tension, describe the transport of mass and momentum. Both ink and air can be considered incompressible as long as the fluid velocity is small compared to the speed of sound. The Navier-Stokes equations are

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u}\right) - \nabla \cdot (\mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)) + \nabla p = \mathbf{F}_{\mathrm{st}}$$

$$(\nabla \cdot \mathbf{u}) = 0$$

Here, ρ denotes density (kg/m^3), μ equals the dynamic viscosity (N·s/m^2), $\mathbf{u}$ represents the velocity (m/s), p denotes pressure (Pa), and $\mathbf{F}_{\mathrm{st}}$ is the surface tension force.

The surface tension force is computed as

$$\mathbf{F}_{\mathrm{st}} = \nabla \cdot \mathbf{T}$$

$$\mathbf{T} = \sigma(\mathbf{I} - (\mathbf{n}\mathbf{n}^T))\delta$$

where $\mathbf{I}$ is the identity matrix, $\mathbf{n}$ is the interface normal, σ is the surface tension coefficient (N/m), and δ equals a Dirac delta function that is nonzero only at the fluid interface. The normal to the interface is

$$\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|}$$

while the delta function is approximated by

$$\delta = 6|\phi(1-\phi)||\nabla \phi|$$

The following table gives the physical parameters of ink and air used in the model:

MEDIUM	DENSITY	DYNAMIC VISCOSITY	SURFACE TENSION
ink	10^3 kg/m^3	0.01 Ns/m^2	0.07 N/m
air	1.225 kg/m^3	$1.789 \cdot 10^{-5}$ Ns/m^2	

INITIAL CONDITIONS

Figure 2 shows the initial distribution ($t = 0$) of ink and air. The velocity is initially 0.

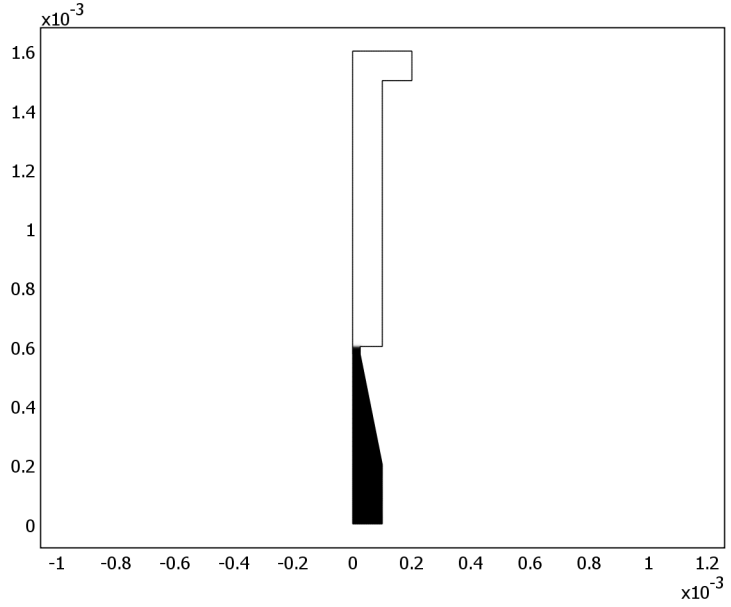

Figure 2: Initial distribution of ink. Black corresponds to ink and white corresponds to air.

BOUNDARY CONDITIONS

Inlet

The inlet velocity in the z direction increases from 0 to the parabolic profile

$$v(r) = 4.5\left(\frac{r + 0.1\ \text{mm}}{0.2\ \text{mm}}\right)\left(1 - \frac{r + 0.1\ \text{mm}}{0.2\ \text{mm}}\right)\ \text{m/s}$$

during the first 2 µs. The velocity is then $v(r)$ for 10 µs and finally decreases to 0 for another 2 µs. The time-dependent velocity profile in the z direction can then be defined as

$$v(r, t) = (\text{step}(t - 1 \cdot 10^{-6}) - \text{step}(t - 13 \cdot 10^{-6})) \cdot v(r)$$

where t is given in seconds and $\text{step}(t)$ is a smooth step function (see Figure 3).

 ©2013 COMSOL

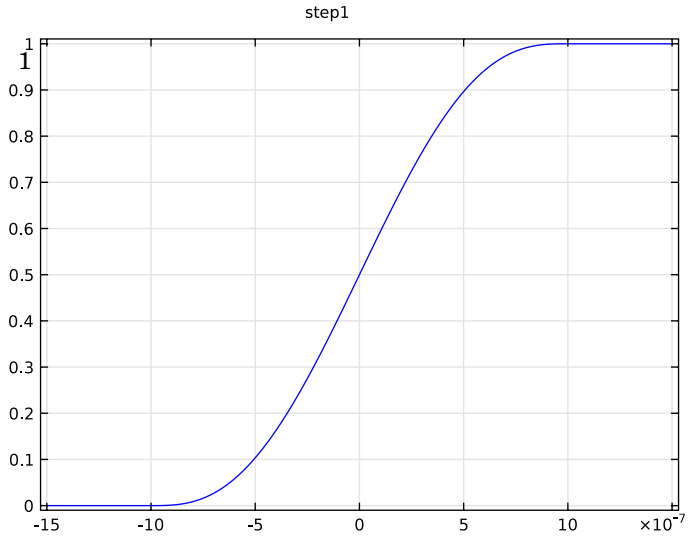

Figure 3: Smooth step function.

Use $\phi = 1$ as the inlet boundary condition for the level set variable.

Outlet

Set a constant pressure at the outlet. The value of the pressure given here is not important because the velocity depends only on the pressure gradient. You thus obtain the same velocity field regardless of whether the pressure is set to 1 atm or to 0.

Walls

On all other boundaries except the target, set No slip conditions. Use the Wetted wall condition on the target, with a contact angle of $\pi/2$ and a slip length of 10 μm.

Results and Discussion

Figure 4 shows the ink surface and the velocity field at different times.

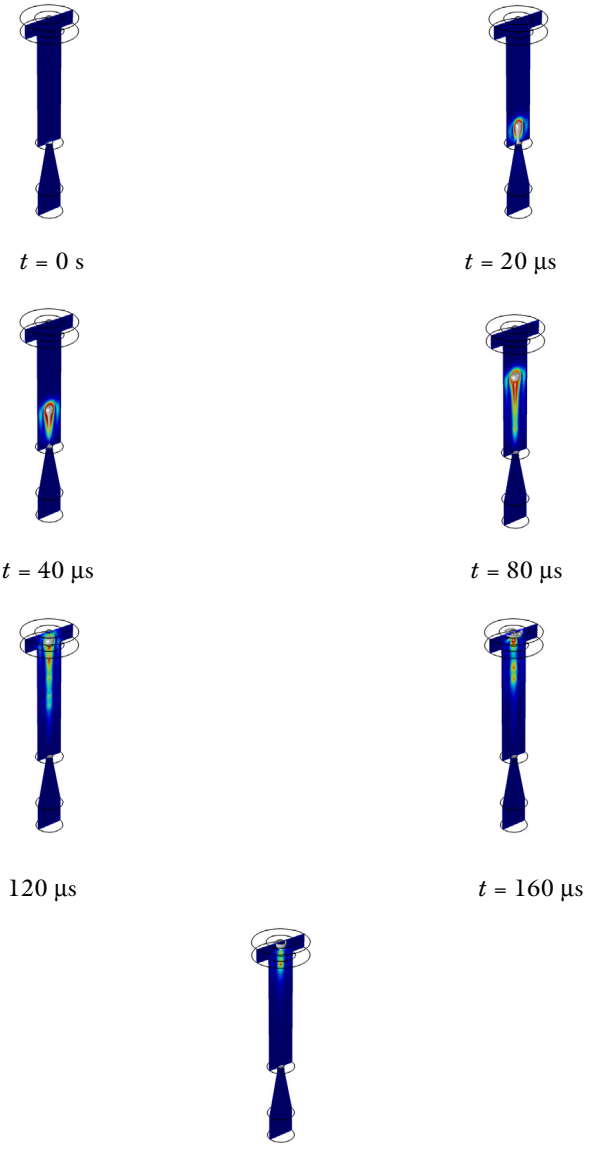

Figure 4: Position of air/ink interface and velocity field at various times.

©2013 COMSOL

Figure 5 illustrates the mass of ink that is further than 0.7 mm from the inlet. The figure shows that the mass of the ejected droplet is approximately $1.9 \cdot 10^{-10}$ kg.

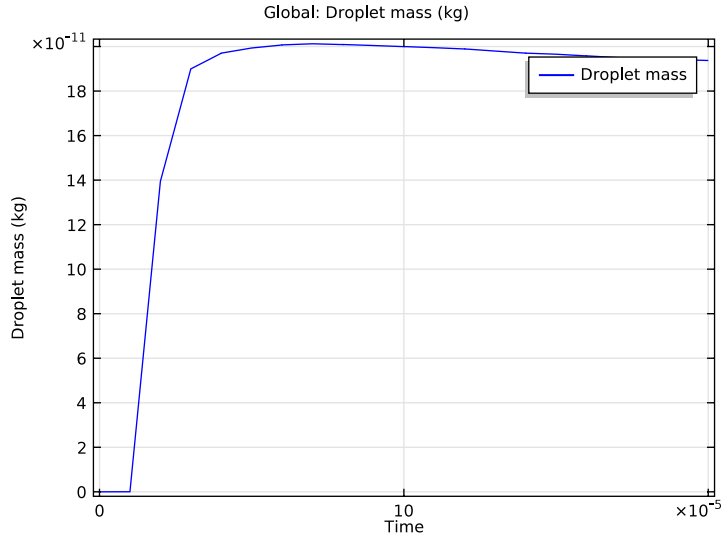

Figure 5: Amount of ink from just above the nozzle.

This example studies only one inkjet model, but it is easy to modify the model in several ways. You can, for example, change properties such as the geometry or the inlet velocity and study the influence on the size and the speed of the ejected droplets. You can also investigate how the inkjet would perform if the ink were replaced by a different fluid. It is also easy to add forces such as gravity to the model.

Notes About the COMSOL Implementation

You can readily set up the model using either the Laminar Two-Phase Flow, Level Set interface or the Laminar Two-Phase Flow, Phase Field interface. These physics interfaces set up the equations automatically, and you need only specify physical parameters of the fluids and the initial and boundary conditions.

In order to accurately resolve the interface between the air and ink, adaptive meshing is used. This means that as the interface moves during the simulation, the mesh will be updated in order to keep the mesh refined in the interface region.

©2013 COMSOL

The simulation procedure involves two consecutive computations. First you calculate a smooth initial solution for the level set variable. Using this initial solution, you then start the time-dependent simulation of the fluid motion.

To calculate the droplet's mass, use an integration coupling operator. To visualize the droplet in 3D, revolve the 2D axisymmetric solution to a 3D geometry.

References

1. J.-T. Yeh, "A VOF-FEM Coupled Inkjet Simulation," *Proc. ASME FEDSM'01*, New Orleans, Louisiana, 2001.

2. E. Olsson and G. Kreiss, "A Conservative Level Set Method for Two Phase Flow," *J. Comput. Phys.*, vol. 210, pp. 225–246, 2005.

3. P. Yue, J. Feng, C. Liu and J. Shen, "A Diffuse-Interface Method for Simulating Two-Phase Flows of Complex Fluids," *J. Fluid Mech.*, vol. 515, pp. 293–317, 2004.

Model Library path: `Microfluidics_Module/Two-Phase_Flow/inkjet_nozzle_ls`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D axisymmetric** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Fluid Flow>Multiphase Flow>Two-Phase Flow, Level Set>Laminar Two-Phase Flow, Level Set (tpf)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Transient with Initialization**.

7 Click **Finish**.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

 ©2013 COMSOL

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1

1 Right-click **Model 1>Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.1`.

4 In the **Height** edit field, type `0.2`.

5 Click the **Build Selected** button.

Bézier Polygon 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **r** to `0.1`.

5 In row **1**, set **z** to `0.2`.

6 In row **2**, set **z** to `0.2`.

7 Find the **Added segments** subsection. Click the **Add Linear** button.

8 Find the **Control points** subsection. In row **2**, set **z** to `0.575`.

9 Find the **Added segments** subsection. Click the **Add Linear** button.

10 Find the **Control points** subsection. In row **2**, set **r** to `0.025`.

11 Find the **Added segments** subsection. Click the **Add Linear** button.

12 Find the **Control points** subsection. Click the **Close Curve** button.

13 Click the **Build Selected** button.

14 Click the **Zoom Extents** button on the Graphics toolbar.

Rectangle 2

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.025`.

4 In the **Height** edit field, type `1.025`.

5 Locate the **Position** section. In the **z** edit field, type `0.575`.

6 Click the **Build Selected** button.

7 Click the **Zoom Extents** button on the Graphics toolbar.

Rectangle 3

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.1`.

4 In the **Height** edit field, type `1`.

5 Locate the **Position** section. In the ***r*** edit field, type `0`.

6 In the **z** edit field, type `0.6`.

7 Click the **Build Selected** button.

Rectangle 4

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.2`.

4 In the **Height** edit field, type `0.1`.

5 Locate the **Position** section. In the **z** edit field, type `1.5`.

6 Click the **Build Selected** button.

 ©2013 COMSOL

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Form Union** and choose **Build Selected**.

This completes the geometry modeling stage.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Density	rho	`1e3[kg/m^3]`
Dynamic viscosity	mu	`1e-2[Pa*s]`

4 Right-click **Model 1>Materials>Material 1** and choose **Rename**.

5 Go to the **Rename Material** dialog box and type `Ink` in the **New name** edit field.

6 Click **OK**.

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click **Add Material to Model**.

DEFINITIONS

Now, define a step function to use when defining the time dependence of the inlet velocity.

Step 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Functions>Step**.

2 In the **Step** settings window, click to expand the **Smoothing** section.

3 In the **Size of transition zone** edit field, type `2*1e-6`.

4 Click the **Plot** button.

Next, define an integration operator that you will use when defining a variable for the droplet mass.

Integration 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

After these preliminaries, you can define variables for the inlet velocity and the droplet mass.

Variables 1

1 Right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
v_inr	4.5[m/s]*((r+0.1[mm])/0.2[mm])*(1-((r+0.1[mm])/0.2[mm]))	r-dependent inlet velocity factor
inlett	step1(t[1/s]-1e-6)-step1(t[1/s]-13e-6)	t-dependent inlet velocity factor

 ©2013 COMSOL

Name	Expression	Description
v_in	v_inr*inlett	Inlet velocity
m_d	intop1(1e3[kg/m^3]*phils*(z>0.7[mm])*2*pi*r)	Droplet mass

The model utilizes adaptive meshing and linear elements will then suffice.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu

LAMINAR TWO-PHASE FLOW, LEVEL SET

1 In the **Model Builder** window, under **Model 1** click **Laminar Two-Phase Flow, Level Set**.

2 In the **Laminar Two-Phase Flow, Level Set** settings window, click to expand the **Discretization** section.

3 From the **Discretization of fluids** list, choose **P1 + P1**.

Fluid Properties 1

1 In the **Model Builder** window, under **Model 1>Laminar Two-Phase Flow, Level Set** click **Fluid Properties 1**.

2 In the **Fluid Properties** settings window, locate the **Fluid 1 Properties** section.

3 From the **Fluid 1** list, choose **Air**.

4 Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, choose **Ink**.

5 Locate the **Surface Tension** section. From the **Surface tension coefficient** list, choose **User defined**. In the σ edit field, type `0.07`.

6 Locate the **Level Set Parameters** section. In the γ edit field, type `10`.

7 In the ε_{ls} edit field, type `2.5e-6`.

Initial Values 2

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Level Set** and choose **Initial Values**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 Click the **Fluid 2** button.

4 Select Domains 1–3 only.

Initial Interface 1

1 In the **Model Builder** window, under **Model 1>Laminar Two-Phase Flow, Level Set** click **Initial Interface 1**.

©2013 COMSOL

2 Select Boundary 8 only.

Inlet 1

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Level Set** and choose **Inlet**.

2 Select Boundary 2 only.

3 In the **Inlet** settings window, locate the **Boundary Condition** section.

4 In the ϕ edit field, type `1`.

5 Locate the **Velocity** section. In the U_0 edit field, type `v_in`.

Outlet 1

1 Right-click **Laminar Two-Phase Flow, Level Set** and choose **Outlet**.

2 Select Boundary 24 only.

Wall 2

1 Right-click **Laminar Two-Phase Flow, Level Set** and choose **Wall**.

2 Select Boundaries 11, 18, and 23 only.

3 In the **Wall** settings window, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Wetted wall**.

5 In the β edit field, type `10[um]`.

MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 2: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 2: Time Dependent**.

2 In the **Time Dependent** settings window, locate the **Study Settings** section.

3 In the **Times** edit field, type `range(0,10e-6,200e-6)`.

4 Click to expand the **Results While Solving** section. Select the **Plot** check box.

5 From the **Plot group** list, choose **Default**.

This choice means that the **Graphics** window will show a surface plot of the volume fraction of Fluid 1 while solving, and this plot will be updated at output time step, that is each 10th microsecond.

6 Locate the **Study Extensions** section. Select the **Adaptive mesh refinement** check box.

 ©2013 COMSOL

By adjusting the scaling of the fields manually, you can reduce the computation time.

1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

2 Expand the **Study 1>Solver Configurations** node.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1** node, then click **Study 1>Solver Configurations>Solver 1>Dependent Variables 2**.

2 In the **Dependent Variables** settings window, locate the **Scaling** section.

3 From the **Method** list, choose **Manual**.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Dependent Variables 2** node, then click **Velocity field (mod1.u)**.

5 In the **Field** settings window, locate the **Scaling** section.

6 From the **Method** list, choose **Manual**.

7 In the **Scale** edit field, type `10`.

8 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 2** click **Pressure (mod1.p)**.

9 In the **Field** settings window, locate the **Scaling** section.

10 From the **Method** list, choose **Manual**.

11 In the **Scale** edit field, type `1e4`.

12 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Volume Fraction, 3D (tpf)

1 In the **Model Builder** window, under **Results** right-click **Volume Fraction, 3D (tpf)** and choose **Slice**.

2 In the **Slice** settings window, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

4 In the **Planes** edit field, type `1`.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Laminar Two-Phase Flow, Level Set>Velocity magnitude (tpf.U)**.

6 Click the **Plot** button.

7 In the **Model Builder** window, click **Volume Fraction, 3D (tpf)**.

©2013 COMSOL

8 In the **3D Plot Group** settings window, locate the **Data** section.

9 From the **Time** list, choose **0**.

10 Click the **Plot** button.

Compare the resulting plot with that in the upper panel of Figure 4.

11 To create the remaining plots, plot the solution for the time values 2e-5, 4e-5, 8e-5, 1.2e-4, 1.6e-4, and 2e-4.

1D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 3**.

4 Right-click **Results>1D Plot Group 4** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Droplet mass (m_d)**.

6 Click the **Plot** button.

 ©2013 COMSOL