Created in COMSOL Multiphysics 5.2a

Inkjet Nozzle — Level Set Method

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 performance.

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

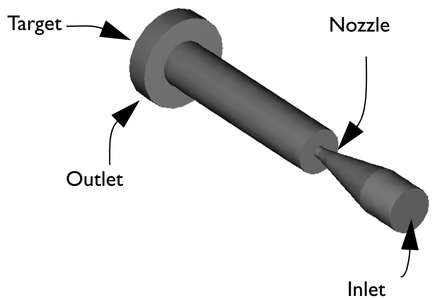


Figure 1: Geometry of the inkjet.

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

Level Set Method

The Laminar Two-Phase Flow, Level Set 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

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

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 N·s/m^2	0.07 N/m
air	1.225 kg/m^3	$1.789\cdot10^{-5}$ N·s/m^2	

INITIAL CONDITIONS

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

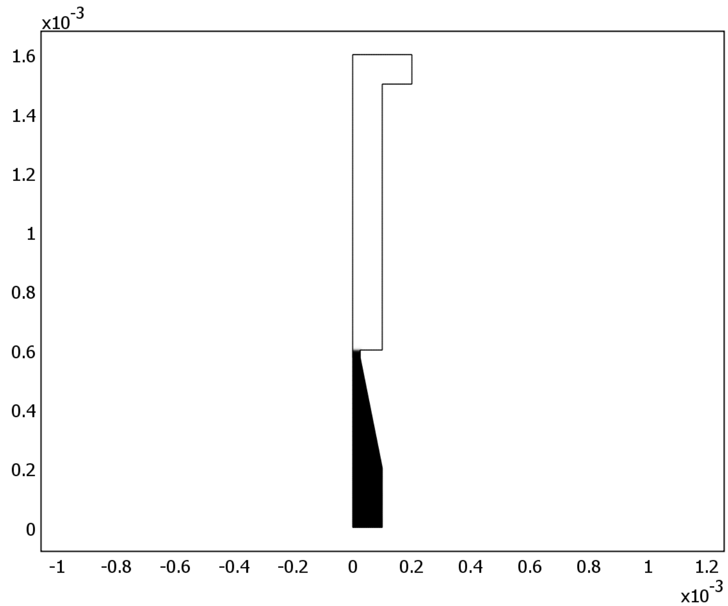


Figure 2: Initial distribution of ink. Black corresponds to ink and white corresponds to air.

BOUNDARY CONDITIONS

Inlet

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

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

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

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

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

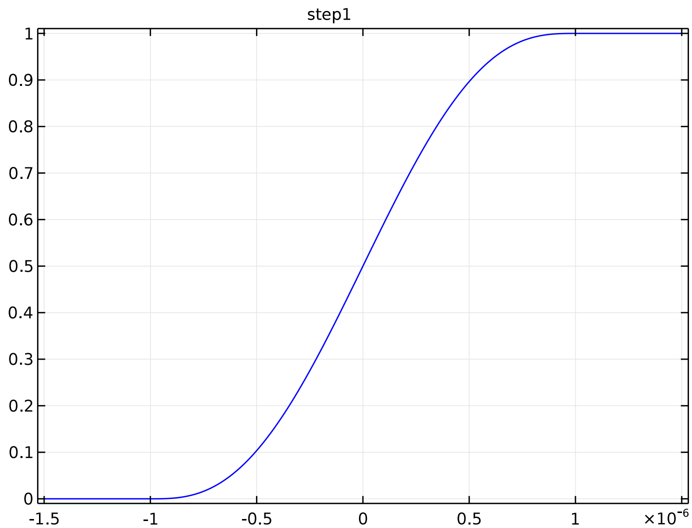


Figure 3: Smooth step function.

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.

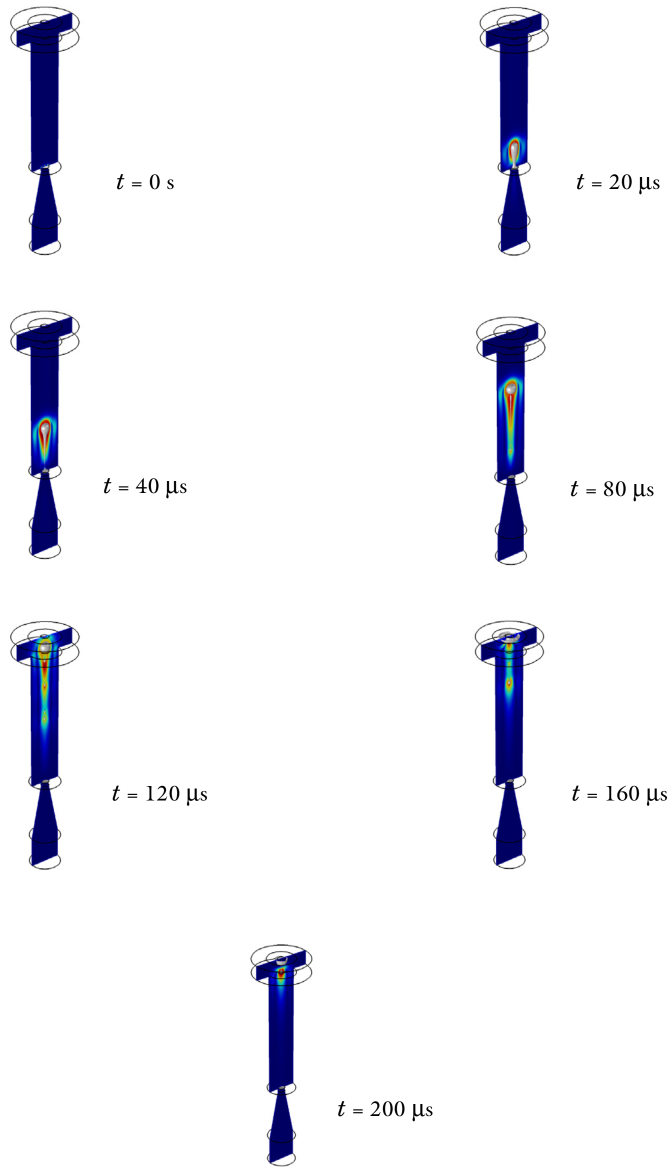


Figure 4: Position of air/ink interface and velocity field at various times.

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.

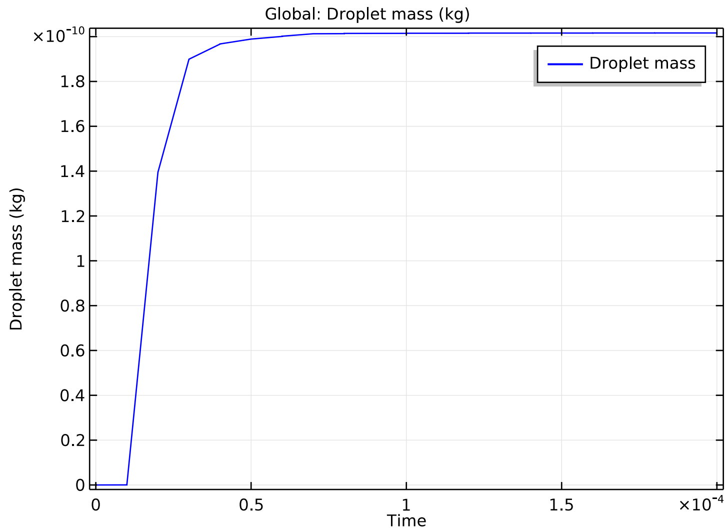


Figure 5: Amount of ink from just above the nozzle.

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 the Laminar Two-Phase Flow, Level Set interface. This interface adds 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, use the adaptive meshing. This means that as the interface moves during the simulation, the mesh is updated in order to keep the mesh refined in the interface region.

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 axially symmetric 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.

Application Library path: `CFD_Module/Multiphase_Tutorials/inkjet_nozzle_ls`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Fluid Flow>Multiphase Flow>Two-Phase Flow, Level Set>Laminar Two-Phase Flow, Level Set**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Transient with Phase Initialization**.

6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Units** section.

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

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

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

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

Bézier Polygon 1 (b1)

1 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

3 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

4 Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **1**, set **r** to `0.1` and **z** to `0.2`.

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

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

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

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

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

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

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

13 Right-click **Bézier Polygon 1 (b1)** and choose **Build Selected**.

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

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

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

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

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

6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

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

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

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

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

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

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

Rectangle 4 (r4)

1 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

4 In the **Width** text field, type `0.2`.

5 In the **Height** text field, type `0.1`.

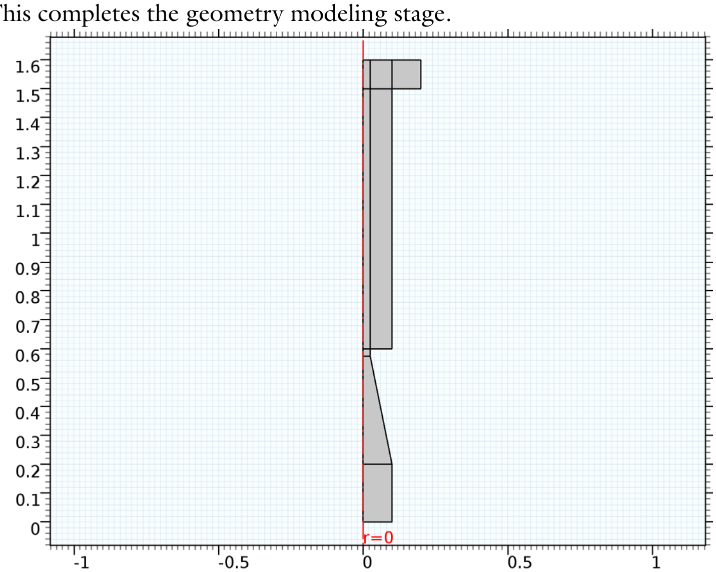
6 Locate the **Position** section. In the **z** text field, type `1.5`.

Form Union (fin)

1 Right-click **Rectangle 4 (r4)** and choose **Build Selected**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

This completes the geometry modeling stage.



MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, locate the **Material Contents** section.

2 In the table, enter the following settings:

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

3 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.

4 In the **Rename Material** dialog box, type `Ink` in the **New label** text field.

5 Click **OK**.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-In>Air**.

4 Click **Add to Component** in the window toolbar.

DEFINITIONS

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

Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Step**.

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

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

4 Click **Plot**.

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

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, 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 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for Variables, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
v_inr	4.5[m/s]*((r+0.1[mm])/0.2[mm])*(1-((r+0.1[mm])/0.2[mm]))	m/s	r-dependent inlet velocity factor
inlett	step1(t[1/s]-1e-6)-step1(t[1/s]-13e-6)		t-dependent inlet velocity factor
v_in	v_inr*inlett	m/s	Inlet velocity
m_d	intop1(1e3[kg/m^3]*phils*(z>0.7[mm])*2*pi*r)	kg	Droplet mass

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

LEVEL SET (LS)

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Level Set (ls)**.

3 In the **Settings** window for Level Set, click to expand the **Advanced settings** section.

4 Locate the **Advanced Settings** section. From the **Convective term** list, choose **Conservative form**.

5 In the **Model Builder** window, click **Level Set (ls)**.

Initial Interface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Initial Interface**.

2 Select Boundary 8 only.

3 In the **Model Builder** window, click **Level Set (ls)**.

4 In the **Settings** window for Level Set, click to expand the **Discretization** section.

5 In the **Settings** window for Level Set, locate the **Discretization** section.

6 From the **Level set variable** list, choose **Linear**.

LAMINAR FLOW (SPF)

On the **Physics** toolbar, click **Level Set (ls)** and choose **Laminar Flow (spf)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for Laminar Flow, click to expand the **Discretization** section.

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

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

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Two-Phase Flow, Level Set 1 (tpf1)**.

2 In the **Settings** window for Two-Phase Flow, Level Set, locate the **Fluid 1 Properties** section.

3 From the **Fluid 1** list, choose **Air (mat2)**.

4 Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, choose **Ink (mat1)**.

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

LEVEL SET (LS)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Level Set (ls)**.

Level Set Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Level Set (ls)** click **Level Set Model 1**.

2 In the **Settings** window for Level Set Model, locate the **Level Set Model** section.

3 In the ε_{ls} text field, type `2.5e-6`.

4 In the γ text field, type `10`.

Initial Values 2

1 On the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 In the **Settings** window for Initial Values, locate the **Initial Values** section.

3 From the **Domain initially** list, choose **Fluid 2 (ϕ = 1)**.

4 Select Domains 1–3 only.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Level Set (ls)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, locate the **Initial Values** section.

3 From the **Domain initially** list, choose **Fluid 1 (ϕ = 0)**.

LAMINAR FLOW (SPF)

On the **Physics** toolbar, click **Level Set (ls)** and choose **Laminar Flow (spf)**.

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 2 only.

3 In the **Settings** window for Inlet, locate the **Velocity** section.

4 In the U_0 text field, type `v_in`.

LEVEL SET (LS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Level Set (ls)**.

2 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

3 Select Boundary 2 only.

4 In the **Settings** window for Inlet, locate the **Inlet** section.

5 In the ϕ text field, type `1`.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 24 only.

LEVEL SET (LS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Level Set (ls)**.

2 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

3 Select Boundary 24 only.

MULTIPHYSICS

Wetted Wall 1 (ww1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Boundary>Wetted Wall**.

2 Select Boundaries 11, 18, and 23 only.

3 In the **Settings** window for Wetted Wall, locate the **Wetted Wall** section.

4 In the β text field, type `10[um]`.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, click **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 **Settings** window for Time Dependent, locate the **Study Settings** section.

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

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

   This choice means that the **Graphics** window will show a contour line of the volume function of Fluid 1 and velocity field while solving, and this plot will be updated at each output time step.

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

6 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Adaptive mesh refinement** check box.

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

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 2** node, then click **Velocity field (comp1.u)**.

4 In the **Settings** window for Field, locate the **Scaling** section.

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

6 In the **Scale** text field, type `10`.

7 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 2** click **Pressure (comp1.p)**.

8 In the **Settings** window for Field, locate the **Scaling** section.

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

10 In the **Scale** text field, type `1e4`.

11 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 2** click **Level set variable (comp1.phils)**.

12 In the **Settings** window for Field, locate the **Scaling** section.

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

14 On the **Study** toolbar, click **Compute**.

RESULTS

Volume Fraction of Fluid 1 (ls) 1

1 In the **Model Builder** window, expand the **Volume Fraction of Fluid 1 (ls) 1** node.

2 Right-click **Results>Volume Fraction of Fluid 1 (ls) 1** and choose **Slice**.

3 In the **Settings** window for Slice, locate the **Plane Data** section.

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

5 In the **Planes** text field, type `1`.

6 In the **Model Builder** window, click **Volume Fraction of Fluid 1 (ls) 1**.

7 In the **Settings** window for 3D Plot Group, locate the **Data** section.

8 From the **Time (s)** list, choose **4E-5**.

9 On the **Volume Fraction of Fluid 1 (ls) 1** toolbar, click **Plot**.

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

11 In the **Model Builder** window, click **Volume Fraction of Fluid 1 (ls) 1**.

12 In the **Settings** window for 3D Plot Group, locate the **Data** section.

13 From the **Time (s)** list, choose **0**.

14 On the **Volume Fraction of Fluid 1 (ls) 1** toolbar, click **Plot**.

Compare the resulting plot with that in the upper panel of Figure 4.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 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for 1D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Refined Mesh Solution 1 (sol3)**.

Global 1

On the **1D Plot Group 6** toolbar, click **Global**.

1D Plot Group 6

1 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>m_d - Droplet mass**.
2 On the **1D Plot Group 6** toolbar, click **Plot**.