Created in COMSOL Multiphysics 6.1

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.

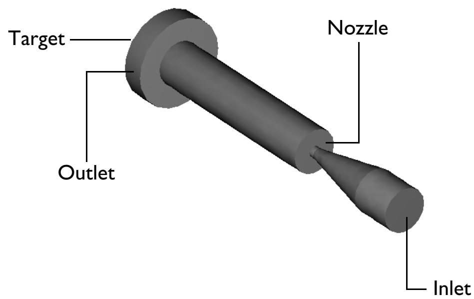


Figure 1: Geometry of the inkjet.

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

Level Set Method

The 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:

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

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

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

Here, ϕ acts as the volume fraction of the ink.

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 ($\mathrm{kg/m^3}$), μ equals the dynamic viscosity ($\mathrm{N{\cdot}s/m^2}$), $\mathbf{u}$ 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

$$\mathbf{F}_{\mathrm{st}} = \sigma\delta\kappa\mathbf{n}$$

where $\mathbf{n}$ is the interface normal, σ is the surface tension coefficient (N/m), $\kappa = -\nabla \cdot \mathbf{n}$ is the curvature, 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

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

INITIAL CONDITIONS

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

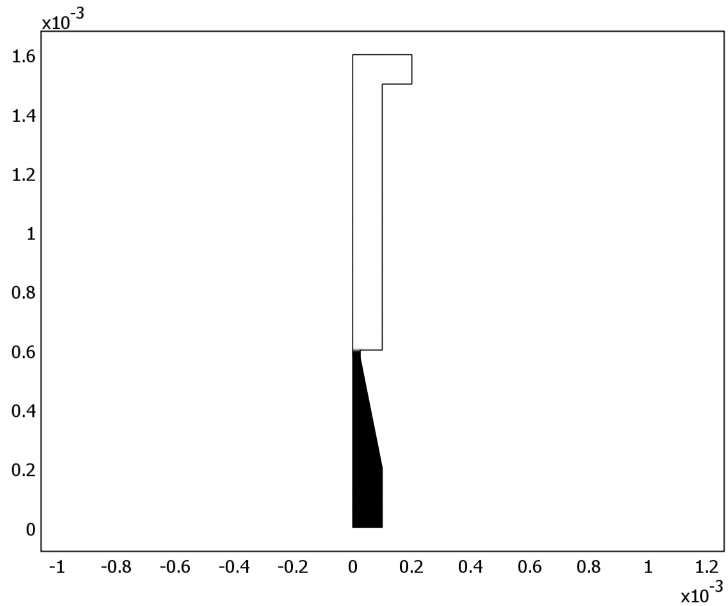


Figure 2: Initial distribution of ink. Black corresponds to ink and white corresponds to air.

BOUNDARY CONDITIONS

Inlet

The mean inlet velocity in the z direction increases from 0 to 0.56 m/s during the first 2 µs. The velocity is then maintained for 10 µs and finally decreases to 0 for another 2 µs. The time-dependent mean velocity in the z direction can then be defined as

$$v_{av}(t) = \mathrm{rect}(t) \cdot 0.56$$

where t is given in seconds and **rect**(t) is a smooth rectangular pulse function with the transition points at 1 µs and 13 µs with a 2 µs transition period. (see Figure 3).

The velocity profiled is defined by the fully developed flow option in the inlet feature.

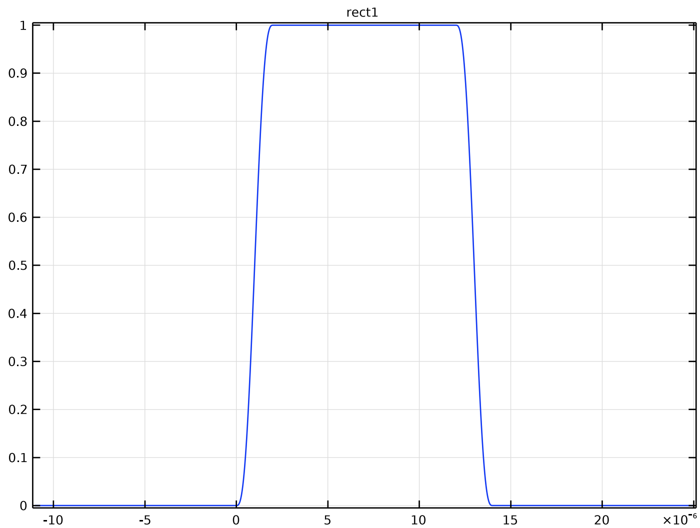


Figure 3: Smooth step function.

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.

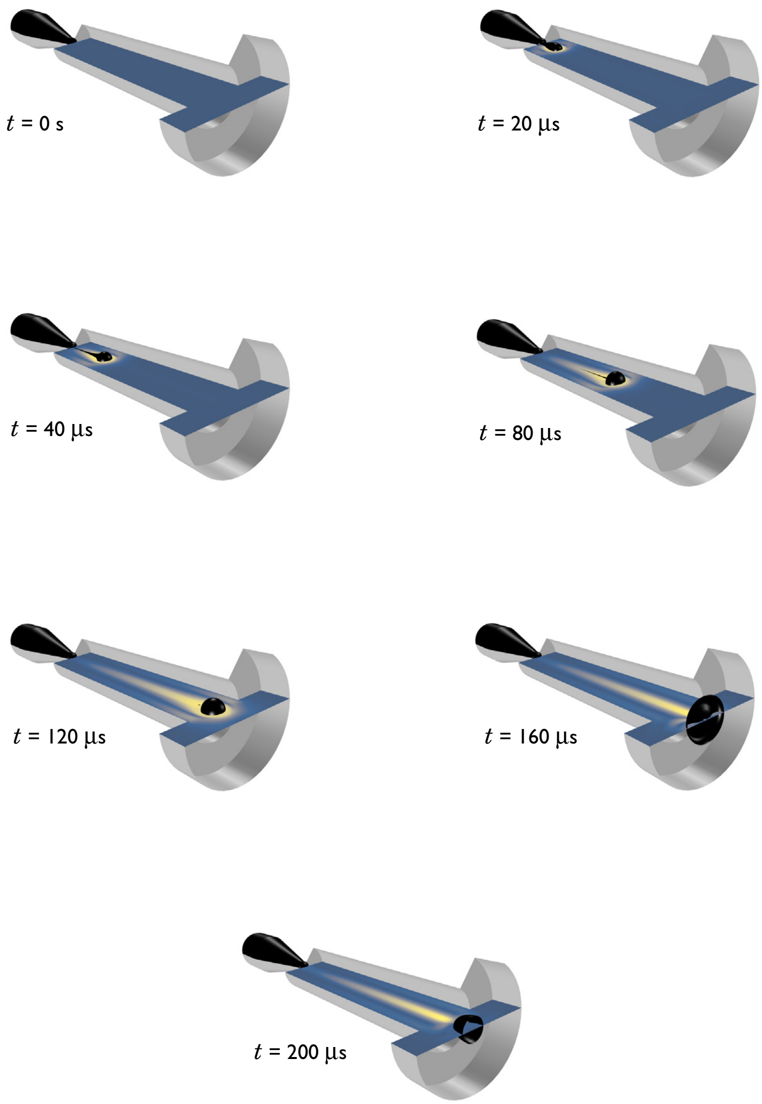


Figure 4: Position of air/ink interface and velocity field at various times.

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 $2 \cdot 10^{-10}$ kg.

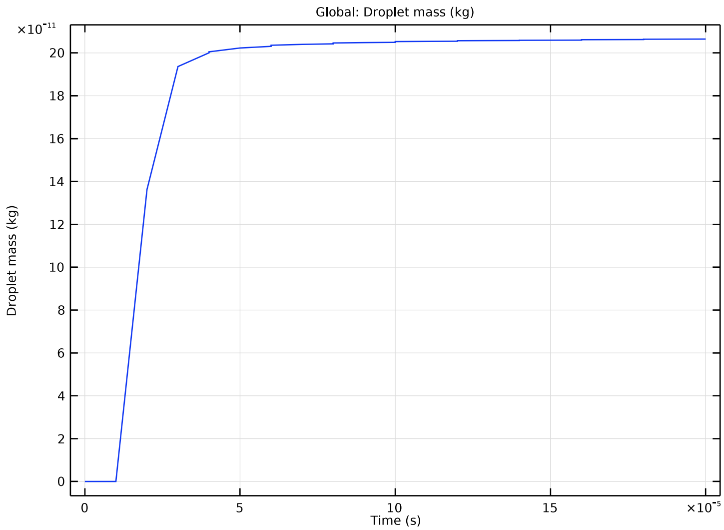


Figure 5: Amount of ink from just above the nozzle.

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 a nonlocal integration coupling. 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_Flow/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 Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Time Dependent with Phase Initialization**.

6 Click **Done**.

It is convenient to add some parameters for the geometry.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
InletR	0.1[mm]	1E-4 m	Nozzle inlet radius
NozzleL	0.375[mm]	3.75E-4 m	Nozzle length
NozzleR	0.025[mm]	2.5E-5 m	Nozzle radius
ThroatL	0.025[mm]	2.5E-5 m	Throat length
TargetL	1 [mm]	0.001 m	Distance to target
AirW	0.1[mm]	1E-4 m	Air channel width

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 In the **Geometry** toolbar, click **Rectangle**.

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

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

4 In the **Height** text field, type `2*InletR`.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (mm)	z (mm)
0	2*InletR
InletR	2*InletR
NozzleR	2*InletR+NozzleL
0	2*InletR+NozzleL

4 Click **Build Selected**.

Rectangle 2 (r2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rectangle 1 (r1)** and choose **Duplicate**.

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

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

4 In the **Height** text field, type `ThroatL+TargetL`.

5 Locate the **Position** section. In the **z** text field, type `NozzleL+2*InletR`.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

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

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

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

5 Locate the **Position** section. In the **z** text field, type `ThroatL+NozzleL+2*InletR`.

Rectangle 4 (r4)

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

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

3 In the **Width** text field, type `2*AirW`.

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

5 Locate the **Position** section. In the **z** text field, type `ThroatL+NozzleL+2*InletR+TargetL-AirW`.

6 Click **Build Selected**.

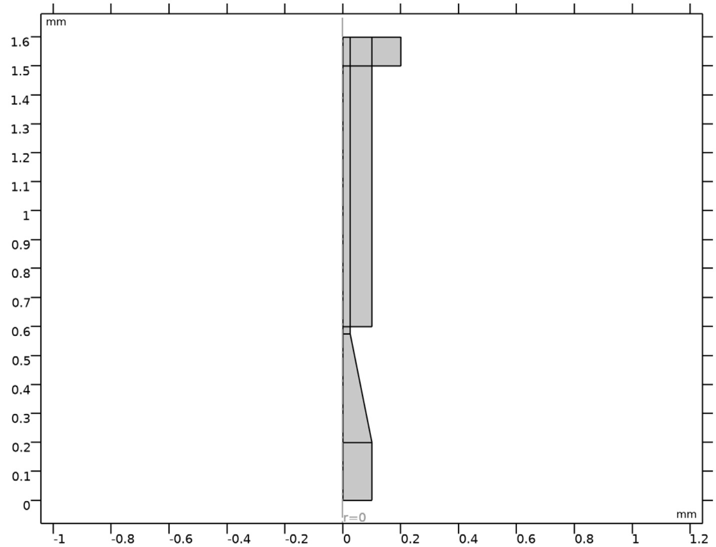
Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

This completes the geometry modeling stage.



Next, add a **Multiphase material** to get the effective material parameters from the volume fraction of each phase, and add the material parameters for each phase.

MULTIPHYSICS

Two-Phase Flow, Level Set 1 (tpf1)

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 **Material Properties** section.

3 Click **Add Multiphase Material**.

MATERIALS

Phase 1 (mpmat1.phase1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials> Multiphase Material 1 (mpmat1)** click **Phase 1 (mpmat1.phase1)**.

2 In the **Settings** window for **Phase**, locate the **Link Settings** section.

3 Click **Add Material from Library** . This button is found when expanding the options next to the **Material** list.

ADD MATERIAL TO PHASE 1 (MPMAT1.PHASE1)

1 Go to the **Add Material to Phase 1 (mpmat1.phase1)** window.

2 In the tree, select **Built-in>Air**.

3 Click **OK**.

MATERIALS

Phase 2 (mpmat1.phase2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials> Multiphase Material 1 (mpmat1)** click **Phase 2 (mpmat1.phase2)**.

2 In the **Settings** window for **Phase**, locate the **Link Settings** section.

3 Click **Blank Material** . This button is found when expanding the options next to the **Material** list.

GLOBAL DEFINITIONS

Ink

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material 2 (mat2)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1e3[kg/m^3]	kg/m³	Basic
Dynamic viscosity	mu	1e-2[Pa*s]	Pa·s	Basic

4 In the **Label** text field, type `Ink`.

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

Rectangle 1 (rect1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Rectangle**.

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

3 Locate the **Parameters** section. In the **Lower limit** text field, type `1e-6`.

4 In the **Upper limit** text field, type `13e-6`.

5 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `2e-6`.

6 Click **Plot**.

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

DEFINITIONS

Integration 1 (intop1)

1. In the **Definitions** toolbar, click **Nonlocal 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. In 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_in	0.56[m/s]*rect1(t[1/s])	m/s	Inlet velocity
m_d	intop1(1e3[kg/m^3]*phils*(z>0.7[mm]))	kg	Droplet mass

MULTIPHYSICS

Two-Phase Flow, Level Set 1 (tpf1)

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 **Surface Tension** section.
3. Select the **Include surface tension force in momentum equation** check box.
4. From the **Surface tension coefficient** list, choose **User defined**. In the σ text field, type 0.07.

LAMINAR FLOW (SPF)

Inlet 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.
2. Select Boundary 2 only.

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

4 From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type v_in.

Outlet 1

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

2 Select Boundary 24 only.

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, Fluid 2

1 In the **Model Builder** window, click **Initial Values, Fluid 2**.

2 Select Domains 1–3 only.

Inlet 1

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

2 Select Boundary 2 only.

3 In the **Settings** window for **Inlet**, locate the **Level Set Condition** section.

4 From the list, choose **Fluid 2 (**φ **= 1)**.

Outlet 1

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

2 Select Boundary 24 only.

MULTIPHYSICS

Wetted Wall 1 (ww1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Wetted Wall 1 (ww1)**.

2 In the **Settings** window for **Wetted Wall**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `11-13, 15, 18-20, 22, 23` in the **Selection** text field.

5 Click **OK**.

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

7 From the **Slip length** list, choose **User defined**. In the β text field, type `10[um]`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 2: Time Dependent

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

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

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

4 Click to expand 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 fraction of Fluid 1 and velocity field while solving, and this plot will be updated at each output time step.

5 Click to expand the **Adaptation** section. From the **Adaptive mesh refinement** list, choose **Adaptive mesh refinement**.

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

Solution 1 (sol1)

1 In 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 **Study** toolbar, click **Compute**.

RESULTS

Edge 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Edge 2D**.

2 Select Boundaries 12, 13, 15, 19, 20, 22, and 24 only.

Revolution 2D 3

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Edge 2D 1**.

4 Click to expand the **Revolution Layers** section. In the **Revolution angle** text field, type 230.

Edge 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Edge 2D**.

2 Select Boundaries 12, 13, and 19 only.

Revolution 2D 4

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Edge 2D 2**.

4 Locate the **Revolution Layers** section. In the **Start angle** text field, type 230.

5 In the **Revolution angle** text field, type 130.

Isosurface 1

1 In the **Model Builder** window, expand the **Results>Volume Fraction of Fluid 1 (ls) 1** node, then click **Isosurface 1**.

2 In the **Settings** window for **Isosurface**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Black**.

Slice 1

1 In the **Model Builder** window, right-click **Volume Fraction of Fluid 1 (ls) 1** and choose **Slice**.

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

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

4 In the **Planes** text field, type 1.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

7 Click **OK**.

Surface 1

1 Right-click **Volume Fraction of Fluid 1 (ls) 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 3**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Surface 2

1 Right-click **Volume Fraction of Fluid 1 (ls) 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 4**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

Volume Fraction of Fluid 1 (ls) 1

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

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

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

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 In the **Volume Fraction of Fluid 1 (ls) 1** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

7 Locate the **Data** section. From the **Time (s)** list, choose **0**.

8 In 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 In 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 **Dataset** list, choose **Study 1/Refined Mesh Solution 1 (sol3)**.

4 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **1D Plot Group 6** and choose **Global**.

2 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 (comp1)>Definitions>Variables>m_d - Droplet mass - kg**.

3 In the **1D Plot Group 6** toolbar, click **Plot**.