Created in COMSOL Multiphysics 6.3

Jet Instability — Level Set[1]

1. This model is courtesy of Prof. E. Furlani, Depts. of Chemical/Biological & Electrical Engineering, SUNY Buffalo, USA (formerly at Eastman Kodak, Rochester, NY, USA).

Introduction

The Marangoni effect results in slip velocity in the tangential direction on a fluid-fluid interface due to gradients in the surface tension coefficient. When the surface tension coefficient is constant, a two-fluid system can exist in static equilibrium. This is because the surface tension force is exactly balanced by a jump in the pressure across the interface. The pressure is discontinuous across the interface, but the velocity field is zero everywhere. The presence of a gradient in the surface tension coefficient means that the flow must be nonstationary. This is due to the fact that the force arising from the variability of the surface tension coefficient acts only in the tangential direction on the interface. This force must be balanced by viscous forces which are only present in a moving fluid. In this example an infinitely long liquid jet breaks up due to a spatially varying surface tension coefficient. Such a situation arises in the fluid jets emitted by continuous inkjet printers as a result of density or thermal gradients. The jet is moving at constant velocity so the problem can be treated in the inertial reference frame in which the jet is initially stationary.

Model Definition

A cylindrical fluid domain with radius 20 microns and 60 microns high contains an initial cylinder of water with radius 5 microns. The surface tension coefficient varies in the axial direction:

$$\sigma = \sigma_0\left(1 - 0.2\cos\left(\frac{2\pi z}{l}\right)\right)$$

where z is the z coordinate, l is the periodicity of the surface tension variation (60 μm) and σ_0 is the reference surface tension coefficient (0.07 N/m). The geometry and surface tension coefficient are shown in Figure 1. The spatial variation of the surface tension results in a force in the radial direction at $t = 0$ s. The two fluids are air and water, whose physical properties are 1.225 kg/m^3 and 1000 kg/m^3 for the density and $1.79 \cdot 10^{-5}$ Pa·s and 0.001 Pa·s for the dynamic viscosity.

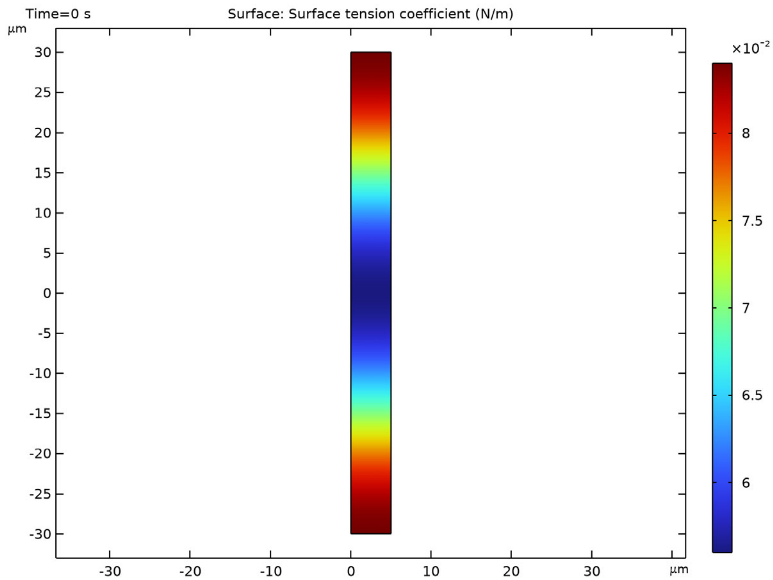

Figure 1: Plot of model geometry (moving mesh) showing surface tension coefficient at t = 0 s.

First, the problem is solved using the Laminar Two-Phase Flow, Moving Mesh interface. In this case the interface is represented by a boundary and has no thickness. Then the problem is formulated on a fixed mesh where the interface is tracked by a level-set function. In this case the thickness of the interface is greatly exaggerated. The Laminar Two-Phase Flow, Level Set interface is used in this case. For practical mesh densities the interface is represented more accurately by the Laminar Two-Phase Flow, Moving Mesh interface, but it cannot handle topological changes and therefore it can only be used prior to the breakup of the droplets.

PROBLEM FORMULATION — MOVING MESH

Domain Equations

In domains, the incompressible Navier–Stokes equations are solved:

$$\rho\frac{\partial\mathbf{u}}{\partial t}+\rho((\mathbf{u}-\mathbf{u}_{\mathrm{m}})\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)]+\rho\mathbf{g} \tag{1}$$

and

$$\nabla\cdot\mathbf{u} = 0 \tag{2}$$

Here $\mathbf{u}_\mathrm{m}$ denotes the mesh velocity and arises from the definition of time derivatives in the coordinate system of the deformed mesh. Poisson's equation is solved for the mesh displacement.

Boundary Conditions

In the absence of mass flow across the boundary the correct boundary condition on the fluid/fluid interface is

$$\mathbf{n}\cdot(\mathbf{T}_1-\mathbf{T}_2) = \sigma(\nabla\cdot\mathbf{n})\mathbf{n}-\nabla\sigma \tag{3}$$

where the total stress tensor, $\mathbf{T}$ is defined for either fluid 1 or fluid 2 as

$$\mathbf{T}_{1,2} = -p_{1,2}\mathbf{I}+\mu_{1,2}(\nabla\mathbf{u}_{1,2}+(\nabla\mathbf{u}_{1,2})^T)$$

This boundary condition can be decomposed into a normal component

$$\mathbf{n}\cdot\mathbf{T}_1\cdot\mathbf{n}-\mathbf{n}\cdot\mathbf{T}_2\cdot\mathbf{n} = \sigma(\nabla\cdot\mathbf{n}) \tag{4}$$

and a tangential component

$$\mathbf{n}\cdot\mathbf{T}_1\cdot\mathbf{t}-\mathbf{n}\cdot\mathbf{T}_2\cdot\mathbf{t} = \nabla\sigma\cdot\mathbf{t} \tag{5}$$

The term on the right-hand side of Equation 4 is the force per unit area due to local curvature of the interface. The term on the right-hand side of Equation 5 is a tangential stress associated with gradients in the surface tension coefficient. Equation 5 reveals that whenever gradients in the surface tension coefficient exist, the flow must be nonstationary. This is because the pressure is continuous in the tangential direction so the gradient in the surface tension coefficient must be balanced by the tangential component of the viscous stress. A mesh velocity equal to the fluid velocity is imposed on the interface:

$$\mathbf{u}_\mathrm{m} = \mathbf{u} \tag{6}$$

Equation 1 and Equation 2 along with the equation for the mesh displacement describe the evolution of the fluid at the domain level. Equation 3 and Equation 6 are suitable boundary conditions for the problem. These equations and boundary conditions are solved in the Two-Phase Flow, Moving Mesh interface.

PROBLEM FORMULATION — LEVEL SET METHOD

When the problem is transformed onto a fixed mesh, the interface is approximated from the spatial derivatives of a level-set function.

Domain Equations

The velocity field and pressure for the liquid phase are described by the Navier–Stokes equations

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) + \sigma(\mathbf{I} - \mathbf{n}\mathbf{n}^T)\delta] + \rho\mathbf{g}$$

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

Here $\mathbf{u}$ is the fluid velocity and the last term inside the divergence operator on the right-hand side is the force due to surface tension. Implementing the surface tension force in this way is convenient because the Marangoni effect is taken into account naturally.

In the level set method the fluid-fluid interface is represented as the 0.5 contour of the level-set function. The level-set function, ϕ, represents the volume fraction of the ink ($\phi = 1$ in the ink and $\phi = 0$ in the air). The transport of the fluid interface separating the two phases is computed by solving the level-set equation:

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

where ε is a parameter that controls the thickness of the interface.

Boundary Conditions

Periodic boundary conditions are used on the top and bottom boundaries to mimic the effect of the jet being infinitely long in length:

$$\mathbf{u}|_{\mathrm{top}} = \mathbf{u}|_{\mathrm{bot}}$$

$$p|_{\mathrm{top}} = p|_{\mathrm{bot}}$$

$$\phi|_{\mathrm{top}} = \phi|_{\mathrm{bot}}$$

Point Settings

In order to make the pressure unique, the pressure is constrained at a point.

Results and Discussion

Figure 2 shows the solution obtained from the level set method at 6 different time steps. The water (indicated by the red color) initially forms a perfectly axial column but the force due to the variation in surface tension coefficient perturbs the jet. Once this occurs the force due to surface curvature takes over and the jet breaks up into two main lobes and a satellite drop.

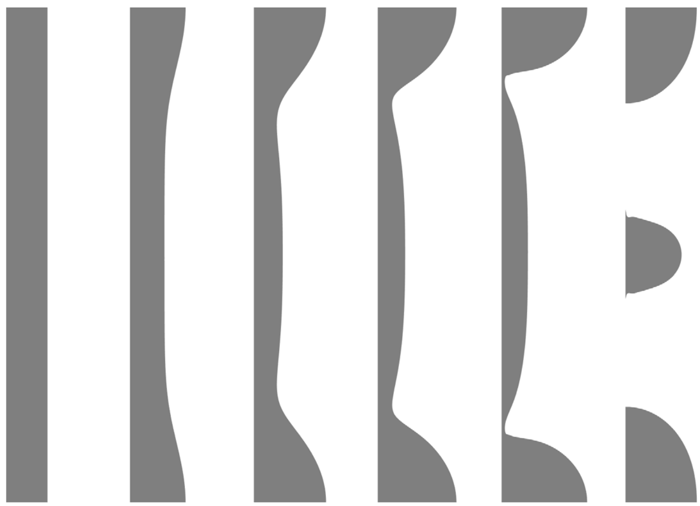

Figure 2: Liquid regions of the model (black), obtained from the level set method, shown at 6 different times:0, 6, 9, 10, 11, and 16.4 μ*s.*

Figure 3: Comparison of solutions obtained using level-set (top row) and the moving mesh (bottom row) methods.

Figure 3 compares the solution using the moving mesh and level set methods. Because the moving mesh method cannot handle a topological change, the comparison is only provided during the initial onset of the instability. The agreement between the two methods is good which indicates that the smoothing of the interface over several mesh elements in the level-set method still results in an accurate solution for the evolution of the surface.

Notes About the COMSOL Implementation

The model is straightforward to set up using either the Laminar Two-Phase Flow, Moving Mesh or the Laminar Two-Phase Flow, Level Set interface. The moving mesh simulation can only be used for early time steps, because topological changes cannot be handled by the interface.

Application Library path: `Microfluidics_Module/Two-Phase_Flow/jet_instability_ls`

Modeling Instructions — Level Set Method

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**.

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 **µm**.

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 `5`.
4 In the **Height** text field, type `60`.
5 Locate the **Position** section. In the **z** text field, type `-30`.

Rectangle 2 (r2)

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 15.

4 In the **Height** text field, type 60.

5 Locate the **Position** section. In the **z** text field, type -30.

Point 1 (pt1)

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

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

3 In the **r** text field, type 15.

4 Click **Build All Objects**.

MATERIALS

Material 1 (mat1)

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

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	1000	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 2 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1.225	kg/m³	Basic
Dynamic viscosity	mu	1.789e-5	Pa·s	Basic

Define a spatially varying function for the surface tension.

DEFINITIONS

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
`sigma`	`0.07*(1-0.2*cos(2*pi*z/60[um]))`		`Surface tension expression`

This model can benefit from using second order elements.

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 **P2+P2**.

LEVEL SET (LS)

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

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

3 From the **Level set variable** list, choose **Quadratic**.

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 **Fluid 1 Properties** section.

3 From the **Fluid 1** list, choose **Material 1 (mat1)**.

4 Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, choose **Material 2 (mat2)**.

5 Locate the **Surface Tension** section. Select the **Include surface tension force in momentum equation** checkbox.

6 From the **Surface tension coefficient** list, choose **User defined**. In the σ text field, type `sigma`.

LAMINAR FLOW (SPF)

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

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 6 only.

LEVEL SET (LS)

Initial Values, Fluid 2

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

2 Select Domain 2 only.

LAMINAR FLOW (SPF)

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

Symmetry 1

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

2 Select Boundaries 2, 3, 5, and 6 only.

LEVEL SET (LS)

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

2 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

Select Boundaries 2, 3, 5, and 6 only.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 In the **Model Builder** window, 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(0,2e-7,2e-5)`.

4 In the **Study** toolbar, click **Compute**.

RESULTS

2D Plot Group 6

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 Clear the **Plot dataset edges** checkbox.

Contour 1

1 Right-click **2D Plot Group 6** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 In the **Expression** text field, type `ls.Vf1`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.5`.

6 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

7 From the **Color table** list, choose **GrayScale**.

8 From the **Color table transformation** list, choose **Reverse**.

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

10 In the **2D Plot Group 6** toolbar, click **Plot**.

Compare the results with those in Figure 2 at different time steps.