Jet Instability—Moving Mesh[1]

Introduction

The Marangoni effect results in a 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 non-stationary. 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.

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).

©2012 COMSOL

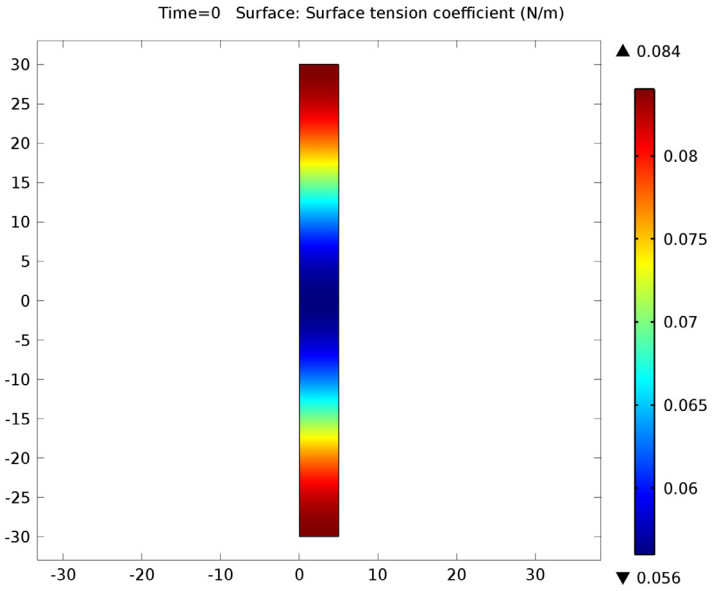

Figure 1: Plot of model geometry (moving mesh) showing surface tension coefficient at t = 0 s.

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 will be 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:

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho((\mathbf{u} - \mathbf{u}_m)\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I} + \eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] + \rho\mathbf{g} \qquad (1)$$

and

 ©2012 COMSOL

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

Here $\mathbf{u}_m$ 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

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

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:

$$\mathbf{u}_m = \mathbf{u} \tag{6}$$

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.

©2012 COMSOL

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

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

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

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 (ϕ=1 in the ink and ϕ=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:

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

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

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

Point Settings

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

 ©2012 COMSOL

Results and Discussion

Figure 3 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.

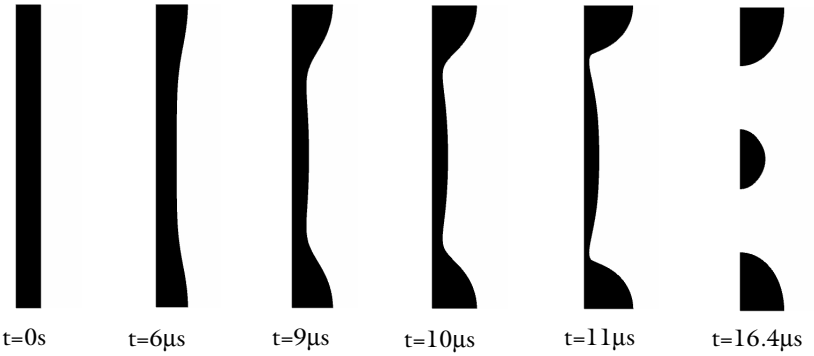

Figure 2: Liquid regions of the model (black), obtained from the level set method, shown at 6 different times.

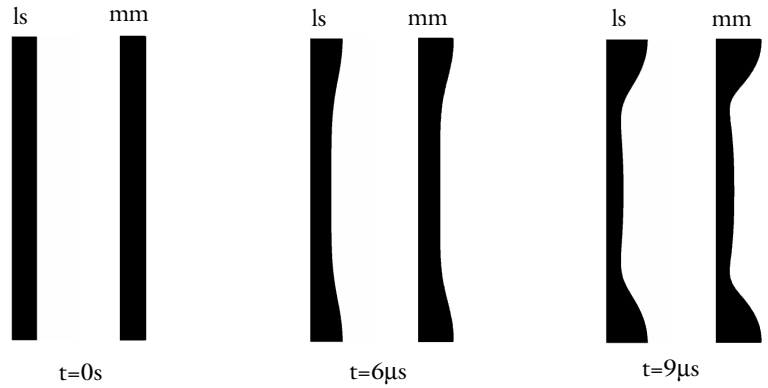

Figure 3: Comparison of solutions obtained using the moving mesh (mm) and level-set (ls) methods.

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.

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

Modeling Instructions—Moving Mesh Method

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, Moving Mesh>Laminar Two-Phase Flow, Moving Mesh (tpfmm)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

7 Click **Finish**.

Since the viscosity of the air is negligible compared to that of the ink, only a domain for the ink is necessary.

GEOMETRY 1

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

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

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

 ©2012 COMSOL

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 `5`.

4 In the **Height** edit field, type `60`.

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

6 Click the **Build All** button.

Define parameters for the ink properties.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

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

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
rhoink	1e3[kg/m^3]	Density, ink
etaink	1e-3[Pa*s]	Dynamic viscosity, ink
sigma0	0.07[N/m]	Reference surface tension coefficient

Define a spatially varying function for the surface tension coefficient. Making the surface tension dependent on Z rather than z ensures that it is defined in the material frame. This means that as the surface moves in the spatial frame, it retains its original surface tension coefficient.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 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
sigma	sigma0*(1-0.2*cos(2*pi*Z/60[um]))	Surface tension coefficient

Set the fluid viscosity and density to the previously defined parameters.

LAMINAR TWO-PHASE FLOW, MOVING MESH

Fluid Properties 1

1 In the **Model Builder** window, expand the **Model 1>Laminar Two-Phase Flow, Moving Mesh** node, then click **Fluid Properties 1**.

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

3 From the ρ list, choose **User defined**. In the associated edit field, type `rhoink`.

4 From the μ list, choose **User defined**. In the associated edit field, type `etaink`.

Define the initial pressure to be 1 atmosphere.

Initial Values 1

1 In the **Model Builder** window, expand the **Laminar Two-Phase Flow, Moving Mesh** node, then click **Initial Values 1**.

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

3 In the p edit field, type `1[atm]`.

The viscosity of the air is neglected and only the air pressure is accounted for in the model. The External Fluid Interface boundary condition can therefore be used. The surface tension is set to the previously defined function.

External Fluid Interface 1

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Moving Mesh** and choose the boundary condition **External Fluid Interface**.

2 Select Boundary 4 only.

3 In the **External Fluid Interface** settings window, locate the **Surface Tension** section.

4 In the σ edit field, type `sigma`.

5 In the p_{ext} edit field, type `1[atm]`.

Use the Wall-Fluid Interface boundary condition to apply contact forces to the fluid. The default contact angle of 90 degrees is appropriate for a symmetry boundary.

Wall-Fluid Interface 1

1 Right-click **Model 1>Laminar Two-Phase Flow, Moving Mesh>External Fluid Interface 1** and choose **Wall-Fluid Interface**.

Boundary conditions for the mesh also need to be applied on all the exterior boundaries (except for the external fluid interface, which automatically includes the mesh constraint).

 ©2012 COMSOL

Prescribed Mesh Displacement 2

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Moving Mesh** and choose the boundary condition **Moving Mesh>Prescribed Mesh Displacement**.

2 Select Boundary 1 only.

3 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.

4 Select the **Prescribed r displacement** check box.

5 Clear the **Prescribed z displacement** check box.

Prescribed Mesh Displacement 3

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Moving Mesh** and choose the boundary condition **Moving Mesh>Prescribed Mesh Displacement**.

2 Select Boundaries 2 and 3 only.

3 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed r displacement** check box.

5 Select the **Prescribed z displacement** check box.

Prescribed Mesh Displacement 1

1 In the **Model Builder** window, under **Model 1>Laminar Two-Phase Flow, Moving Mesh** click **Prescribed Mesh Displacement 1**.

2 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.

3 Clear the **Prescribed r displacement** check box.

4 Clear the **Prescribed z displacement** check box.

The Navier Slip boundary condition must be used on all boundaries with a two phase contact. In this case the boundaries are symmetry boundaries.

Navier Slip 1

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Moving Mesh** and choose the boundary condition **Navier Slip**.

2 Select Boundaries 2 and 3 only.

The flow is laminar, so a free triangular mesh can be used.

MESH 1

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

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

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

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

4 From the **Predefined** list, choose **Extra coarse**.

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

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Times** edit field, type `range(0,0.5e-6,1e-5)`.

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

RESULTS

2D Plot Group 3

1 In the **Model Builder** window, expand the **Results>2D Plot Group 3** node, then click **Surface 1**.

2 In the **Surface** settings window, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

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

5 Click the **Plot** button.

Compare the resulting plot with Figure 3 at different time steps.

 ©2012 COMSOL