Rising Bubble

Introduction

This example shows how to model two immiscible fluids, tracking the fluid-fluid interface. An oil bubble rises through water and merges with oil already residing at the top of the container. Initially three different regions exist: the initially still oil bubble, the oil at the top of the container, and the water surrounding the bubble (see Figure 1). The container is cylindrical with a diameter of $1 \cdot 10^{-2}$ m and a height of $1.5 \cdot 10^{-2}$ m. The oil phase has a viscosity of 0.0208 Pa·s and a density of 879 kg/m^3. For water the viscosity is $1.01 \cdot 10^{-3}$ Pa·s and the density is 1000 kg/m^3. Buoyancy effects cause the oil bubble to rise through the water phase. As the bubble reaches the liquid-liquid interface, it merges with the oil phase.

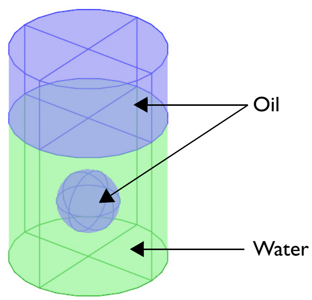

Figure 1: Initial bubble position. The geometry is axisymmetric.

As outlined above, the topology of the fluid interface changes with time. You start with three separate fluid regions and end up with two. The level set method as well as the phase field method are both well suited for modeling moving boundaries where topology changes occur. Both methods are available in the CFD Module as predefined physics interfaces. This example shows you how to use the Laminar Two Phase Flow interface.

Model Definition

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

The Level Set interface finds the fluid interface by tracing the isolines of the level set function, ϕ. The level set or isocontour $\phi = 0.5$ determines the position of the interface. The equation governing the transport and reinitialization of ϕ is

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

where $\mathbf{u}$ (m/s) is the fluid velocity, and γ (m/s) and ε (m) are reinitialization parameters. The ε parameter determines the thickness of the layer around the interface where ϕ goes from zero to one. When stabilization is used for the level set equation, you can typically use an interface thickness of $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The γ parameter determines the amount of reinitialization. A suitable value for γ is the maximum velocity magnitude occurring in the model.

Because the level set function is a smooth step function, it is also used to determine the density and dynamic viscosity globally by

$$\rho = \rho_w + (\rho_o - \rho_w)\phi$$

and

$$\mu = \mu_w + (\mu_o - \mu_w)\phi,$$

Here ρ_w, μ_w, ρ_o, and μ_o denote the constant density and viscosity of water and oil, respectively.

MASS AND MOMENTUM TRANSPORT

In both the Laminar Two-Phase Flow, Level Set and the Laminar Two-Phase Flow, Phase Field interface, the transport of mass and momentum is governed by the incompressible Navier-Stokes equations, including surface tension:

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

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

In the above equations, ρ (kg/m^3) denotes the density, $\mathbf{u}$ is the velocity (m/s), t equals time (s), p is the pressure (Pa), and μ denotes the viscosity (Pa·s). The momentum equations contain gravity, $\rho\mathbf{g}$, and surface tension force components, denoted by $\mathbf{F}_{st}$.

Surface Tension

The surface tension force is defined by

$$\mathbf{F}_{st} = \nabla\cdot\mathbf{T} = \nabla\cdot[\sigma\{\mathbf{I}+(-\mathbf{n}\mathbf{n}^T)\}\delta]$$

where σ is the surface tension coefficient, $\mathbf{I}$ is the identity matrix, $\mathbf{n}$ is the interface unit normal, and δ is a Dirac delta function, nonzero only at the fluid interface. The interface normal is calculated from

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

The level set parameter ϕ is also used to approximate the delta function by a smooth function defined by

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

INITIAL CONDITION

At $t = 0$, the velocity is zero. Figure 2 shows the initialized level set function. The Laminar Two-Phase Flow interface automatically calculates the initial level set function by solving Equation 1 with zero velocity. A suitable time for initialization is

$$t_{\text{init}} = \frac{5\varepsilon}{\gamma}$$

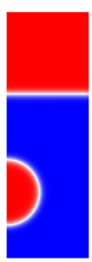

Figure 2: A surface and contour plot of the initialized level set function.

Boundary Conditions

Use no-slip conditions, $\mathbf{u} = 0$ at the top and bottom and a wetted wall condition on the right boundary. The left boundary corresponds to the symmetry axis.

Results and Discussion

Figure 3 and Figure 4 contain snapshots of the fluid interface. The snapshots show how the bubble travels up through the water and merges with the oil above. As the bubble rises, its shape remains spherical due to the surface tension and the high viscosity of the oil. As the droplet hits the water surface, it merges with the oil above and creates waves on the surface.

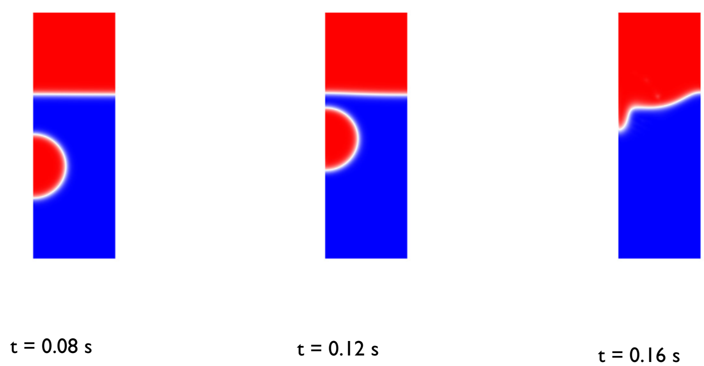

Figure 3: Snapshots showing the interface prior to and just after the bubble hits the surface.

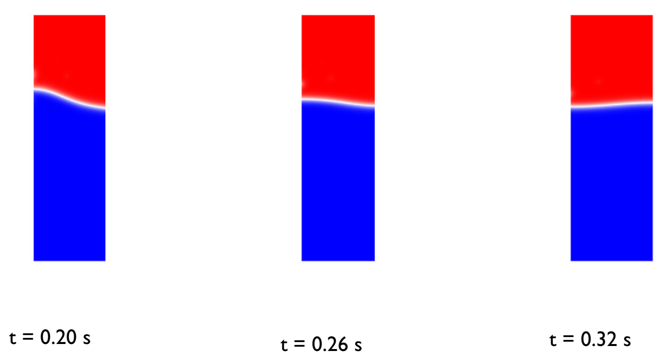

Figure 4: Snapshots showing the interface after the bubble has merged with the oil above.

One way to investigate the quality of the numerical results is to check the conservation of mass. Because there are no reactions and no flow through the boundaries, the total mass of each fluid should be constant in time. Figure 5 shows the total mass of oil as a function of time. The mass loss during simulation is less than 0.1%, showing that both models conserve mass well.

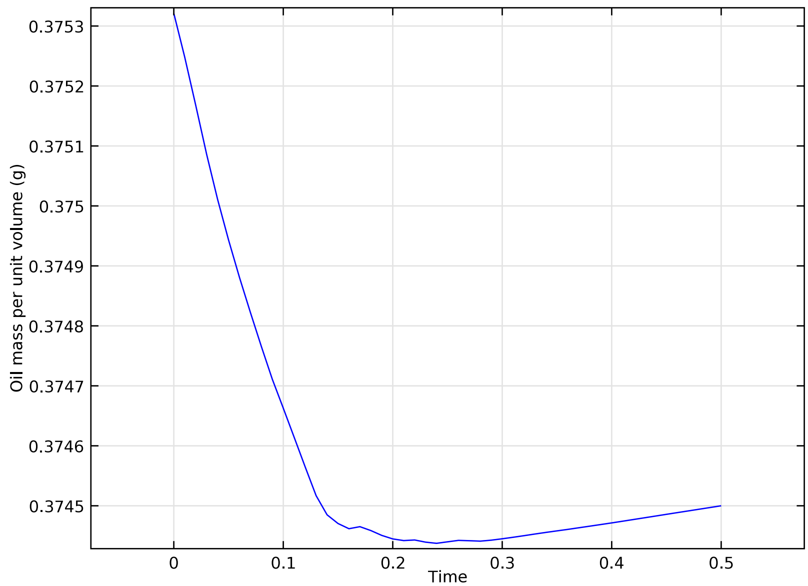

Figure 5: Total mass of oil as a function of time. The total mass loss during the simulation is very small, less than 0.3%.

Notes About the COMSOL Implementation

The model is straightforward to set up and solve using either the Laminar Two-Phase Flow, Level Set interface. Automatically, two study steps are created. The first one initializes the level set function, and the second one calculates the dynamic two phase flow problem. The following sections include step-by-step instructions for both applications modes.

Model Library path: `CFD_Module/Multiphase_Flow/rising_bubble_2daxi`

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

7 Click **Finish**.

GEOMETRY 1

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

2 Go to the **Settings** window for Geometry.

3 Locate the **Units** section. From the **Length unit** list, select **mm**.

Rectangle 1

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

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type `5`.

4 In the **Height** edit field, type `15`.

5 Click the **Build Selected** button.

Polygon 1

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

2 Go to the **Settings** window for Polygon.

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

4 In the **z** edit field, type `10`.

5 Click the **Build Selected** button.

Circle 1

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

2 Go to the **Settings** window for Circle.

3 Locate the **Size and Shape** section. In the **Radius** edit field, type `2`.

4 In the **Sector angle** edit field, type `180`.

5 Locate the **Position** section. In the **z** edit field, type `4`.

6 Locate the **Rotation Angle** section. In the **Rotation** edit field, type `-90`.

7 Click the **Build Selected** button.

Form Union

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

MATERIALS

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

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

3 Locate the **Materials** section. In the **Materials** tree, select **Liquids and Gases>Liquids>Transformer oil**.

4 Right-click and choose **Add Material to Model** from the menu.

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

6 Go to the **Material Browser** window.

7 Locate the **Materials** section. In the **Materials** tree, select **Liquids and Gases>Liquids>Water**.

8 Right-click and choose **Add Material to Model** from the menu.

You can leave the geometric scope empty at this stage; it will be defined when you use this material in the Fluid Properties feature.

LAMINAR TWO-PHASE FLOW, LEVEL SET

Fluid Properties 1

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

2 Go to the **Settings** window for Fluid Properties.

3 Locate the **Fluid 1 Properties** section. From the **Fluid 1** list, select **Transformer oil**.

4 Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, select **Water**.

5 Locate the **Surface Tension** section. From the **Surface tension coefficient** list, select **Library coefficient, liquid/liquid interface**.

6 From the **Surface tension coefficient** list, select **Library coefficient, liquid/liquid interface**, then select **Olive oil/Water, 20°C** from the list underneath.

7 Locate the **Level Set Parameters** section. In the γ edit field, type `0.1`.

Initial Values 2

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Level Set** and choose **Initial Values**.
2 Go to the **Settings** window for Initial Values.
3 Locate the **Initial Values** section. Click the **Fluid 2** button.
4 Select Domain 1 only.

Initial Interface 1

1 In the **Model Builder** window, click **Initial Interface 1**.
2 Select Boundaries 7, 11, and 12 only.

Gravity 1

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Level Set** and choose **Gravity**.
2 Go to the **Settings** window for Gravity.
3 Locate the **Domain Selection** section. From the **Selection** list, select **All domains**.

Pressure Point Constraint 1

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Level Set** and choose **Points>Pressure Point Constraint**.
2 Select Vertex 7 only.

Wall 2

1 In the **Model Builder** window, right-click **Laminar Two-Phase Flow, Level Set** and choose **Wall**.
2 Select Boundaries 9 and 10 only.
3 Go to the **Settings** window for Wall.
4 Locate the **Boundary Condition** section. From the **Boundary condition** list, select **Wetted wall**.

DEFINITIONS

Before creating the mesh, add a variable for computing the mass of oil in the model domain. You will use this variable later to test mass conservation.

Variables 1

1 In the **Model Builder** window, right-click **Model 1>Definitions** and choose **Variables**.
2 Go to the **Settings** window for Variables.

3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
rho_oil	tpf.rho1*tpf.Vf1	Oil mass per unit volume

MESH 1

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

Size

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

2 Go to the **Settings** window for Size.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `0.2`.

4 Click the **Build All** button.

STUDY 1

Step 2: Time Dependent

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

2 Go to the **Settings** window for Time Dependent.

3 Locate the **Study Settings** section. In the **Times** edit field, type `range(0,0.5/50,0.5)`.

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

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

Solver 1

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

2 Go to the **Settings** window for Dependent Variables.

3 Locate the **Scaling** section. From the **Method** list, select **Manual**.

4 In the **Model Builder** window, expand the **Dependent Variables 2** node, then click **mod1_u**.

5 Go to the **Settings** window for Field.

6 Locate the **Scaling** section. From the **Method** list, select **Manual**.

7 In the **Scale** edit field, type `0.1`.

8 In the **Model Builder** window, click **Dependent Variables 2>mod1_p**.

9 Go to the **Settings** window for Field.

10 Locate the **Scaling** section. From the **Method** list, select **Manual**.

11 In the **Scale** edit field, type 100.

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

RESULTS

Next, test to what degree the total mass of oil is conserved.

Derived Values

1 In the **Model Builder** window, right-click **Results>Derived Values** and choose **Integration>Surface Integration**.

2 Go to the **Settings** window for Surface Integration.

3 Locate the **Selection** section. From the **Selection** list, select **All domains**.

4 In the upper-right corner of the **Expression** section, click **Replace Expression**.

5 From the menu, choose **Definitions>Oil mass per unit volume (rho_oil)**.

6 Locate the **Expression** section. From the **Unit** list, select **g**.

7 Click to expand the **Integration Settings** section.

8 Select the **Compute volume integral** check box.

9 Click the **Evaluate** button.

10 In the **Results** window, click **Plot**.

1D Plot Group 4

Compare the result to that in Figure 5. As the plot shows, mass is conserved to within 0.3% accuracy.

Volume Fraction (tpf)

To reproduce the figures in Figure 3 and Figure 4, follow the instructions below.

1 In the **Model Builder** window, click **Results>Volume Fraction (tpf)**.

2 Go to the **Settings** window for 2D Plot Group.

3 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

4 In the **Model Builder** window, expand the **Volume Fraction (tpf)** node, then click **Surface 1**.

5 Go to the **Settings** window for Surface.

6 Locate the **Coloring and Style** section. From the **Color table** list, select **WaveLight**.

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

8 Go to the **Settings** window for 2D Plot Group.

9 Locate the **Data** section. From the **Time** list, select **0**.

10 Click the **Plot** button.

11 From the **Time** list, select **0.08**.

12 Click the **Plot** button.

To reproduce the remaining five plots, plot the solutions for the time values **0.12**, **0.16**, **0.20**, **0.26**, and **0.32**.

Finally, create a movie where the results of the axisymmetric model are revolved into 3D.

Volume Fraction, 3D (tpf)

Click the **Player** button on the main toolbar.