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 1.5×10^{-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 1.01×10^{-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
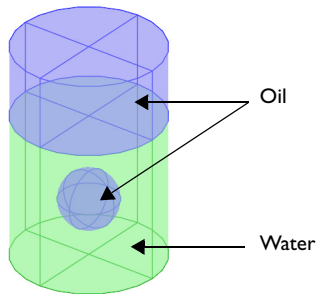

Figure 1: Initial bubble position. The geometry is axisymmetric.

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, Level Set 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

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

where $\mathbf{u}$ (SI unit: m/s) is the fluid velocity, and γ (SI unit: m/s) and ε (SI unit: 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 the Laminar Two-Phase Flow, Level Set interface, the transport of mass and momentum is governed by the incompressible Navier-Stokes equations, including surface tension:

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

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

In the above equations, ρ (SI unit: kg/m^3) denotes the density, $\mathbf{u}$ is the velocity (SI unit: m/s), t equals time (SI unit: s), p is the pressure (SI unit: Pa), and μ denotes the viscosity (SI unit: 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 initial level set function. This is automatically computed using a Phase Initialization study step by solving for the geometrical distance to the initial interface, D_{wi}. The initialized level set function is then defined from the analytical steady state solution for a straight fluid-fluid interface:

$$\phi_{1,0} = \frac{1}{1+e^{D_{wi}/\varepsilon}}, \quad \phi_{2,0} = \frac{1}{1+e^{-D_{wi}/\varepsilon}},$$

in the domains initially filled with Fluid 1 and Fluid 2 respectively,

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.
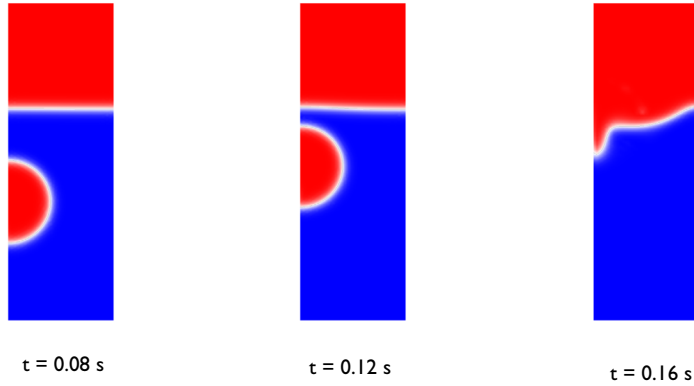

| t = 0.08 s | t = 0.12 s | t = 0.16 s |

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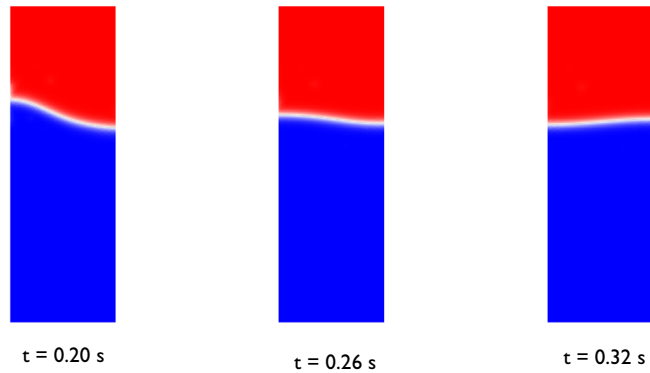

| t = 0.20 s | t = 0.26 s | t = 0.32 s |

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

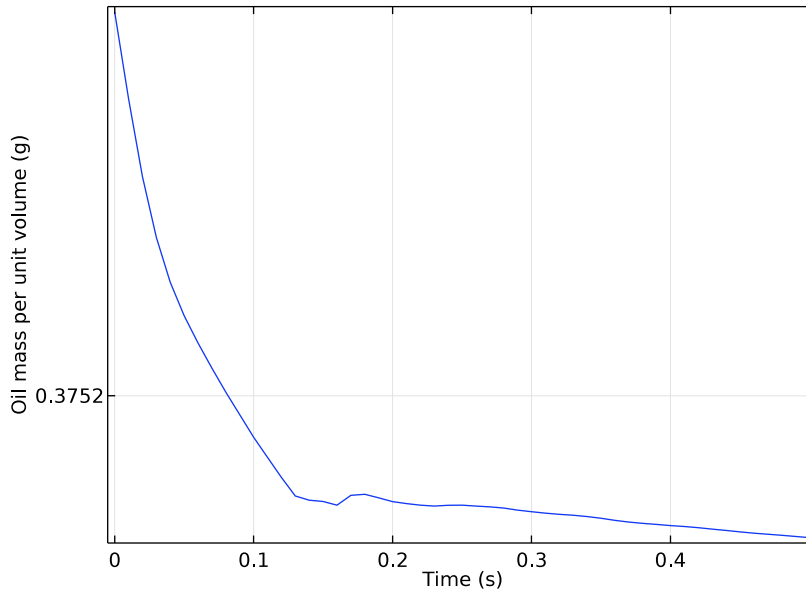of time. The mass loss during simulation is less than 0.2%, showing that the model conserve mass well.

Figure 5: Total mass of oil as a function of time. The total mass loss during the simulation is very small, less than 0.2%.

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.

Application Library path: `CFD_Module/Multiphase_Tutorials/`
`rising_bubble_2daxi`

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 5.

4 In the **Height** text field, type 15.

Polygon 1 (pol1)

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

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

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

4 In the **r** text field, type 0 5.

5 In the **z** text field, type 10.

Circle 1 (c1)

1 Right-click **Polygon 1 (pol1)** and choose **Build Selected**.

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

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

4 In the **Radius** text field, type 2.

5 In the **Sector angle** text field, type 180.

6 Locate the **Position** section. In the **z** text field, type 4.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.

8 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Form Union (fin)

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

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 **Liquids and Gases>Liquids>Transformer oil**.

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

ADD MATERIAL

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

2 In the tree, select **Liquids and Gases>Liquids>Water**.

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

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

You can leave the **Geometric Entity Selection** empty at this stage; it will be defined when you use this material in the Fluid Properties feature.

LEVEL SET (LS)

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

Initial Interface 1

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

2 Select Boundaries 7, 11, and 12 only.

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 **Transformer oil (mat1)**.

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

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

6 Select **Olive oil/Water, 20 C$^{\mathrm{o}}$** from the list.

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 γ text field, type 0.1.

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 ($\phi = 1$)**.

4 Select Domain 1 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 ($\phi = 0$)**.

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)**.

Volume Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Volume Force**.

2 In the **Model Builder** window, right-click **Volume Force 1** and choose **Rename**.

3 In the **Rename Volume Force** dialog box, type Gravity in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for Volume Force, locate the **Domain Selection** section.

6 From the **Selection** list, choose **All domains**.

7 Locate the **Volume Force** section. Specify the **F** vector as

0	r
-g_const*spf.rho	z

Pressure Point Constraint 1

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

2 Select Point 7 only.

MULTIPHYSICS

Wetted Wall 1 (ww1)

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

2 Select Boundaries 9 and 10 only.

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 On the **Home** toolbar, click **Variables** and choose **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
rho_oil	tpf1.rho1*tpf1.Vf1	kg/m³	Oil mass per unit volume

MESH 1

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

Size

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

2 Click the **Custom** button.

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

4 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(0,0.5/50,0.5)`.

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 0.01.

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 100.

11 In the **Model Builder** window, click **Solution 1 (sol1)**.

12 In the **Settings** window for Solution, click **Compute**.

RESULTS

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

Surface Integration 1
On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

Derived Values

1 In the **Settings** window for Surface Integration, locate the **Selection** section.

2 From the **Selection** list, choose **All domains**.

3 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Variables>rho_oil - Oil mass per unit volume**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
rho_oil	g	Oil mass per unit volume

5 Locate the **Integration Settings** section. Select the **Compute volume integral** check box.

6 Click **Evaluate**.

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

1D Plot Group 6
Compare the result to that in Figure 5. As the plot shows, mass is conserved to within 0.1% accuracy.

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

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

3 Clear the **Plot data set edges** check box.

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

5 In the **Settings** window for Surface, locate the **Coloring and Style** section.

6 From the **Color table** list, choose **WaveLight**.

To reproduce the plots of reproduced in Figure 2 and Figure 3, plot the solution for the time values 0 0.08 0.12, 0.16, 0.20, 0.26, and 0.32.

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

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

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

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

Repeat last two steps for the time values 0.08 0.12, 0.16, 0.20, 0.26, and 0.32 s.

Volume Fraction of Fluid 1 (ls) 1
Add a slice plot of the velocity magnitude to the axisymmetric model revolved into 3D

1 In the **Model Builder** window, expand the **Results>Velocity (spf) 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 **0.11**.

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

Volume Fraction of Fluid 1 (ls) 1

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

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

Finally, create a movie using the current plot group.

3 Click **Animation** and choose **Player**.

Export

In the **Model Builder** window, under **Results>Export** right-click **Animation 1** and choose **Play**.