Model created in COMSOL Multiphysics 6.4

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 has a density of 879 kg/m^3. For water the viscosity is $1.01 \cdot 10^{-3}$ Pa·s and the density is 998.2 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.

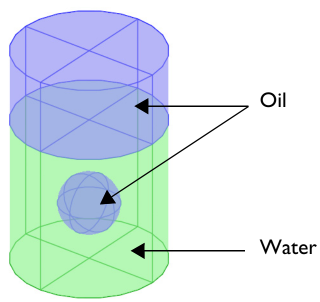

Figure 1: Initial bubble position. The geometry is axisymmetric.

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 multiphysics 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 force denoted by $\rho\mathbf{g}$, and surface tension force, 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:

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

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.

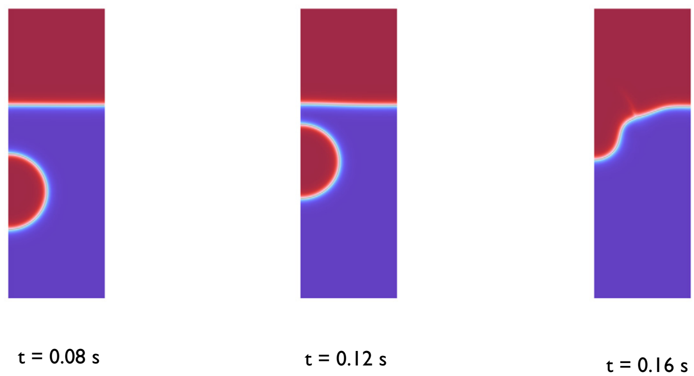

Figure 3: Snapshots showing the interface prior to and just after the bubble hits the surface.

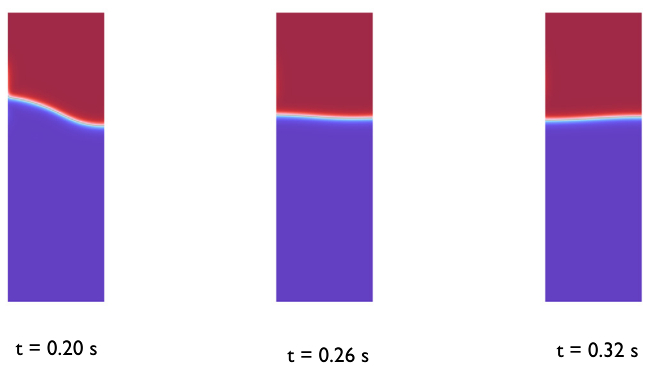

Figure 4: Snapshots showing the interface after the bubble has merged with the oil above.

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 small, showing that the model conserves mass.

Exact mass conservation is obtained when using the conservative level set form. However, the conservative form is less suited for numerical calculations and convergence is harder.

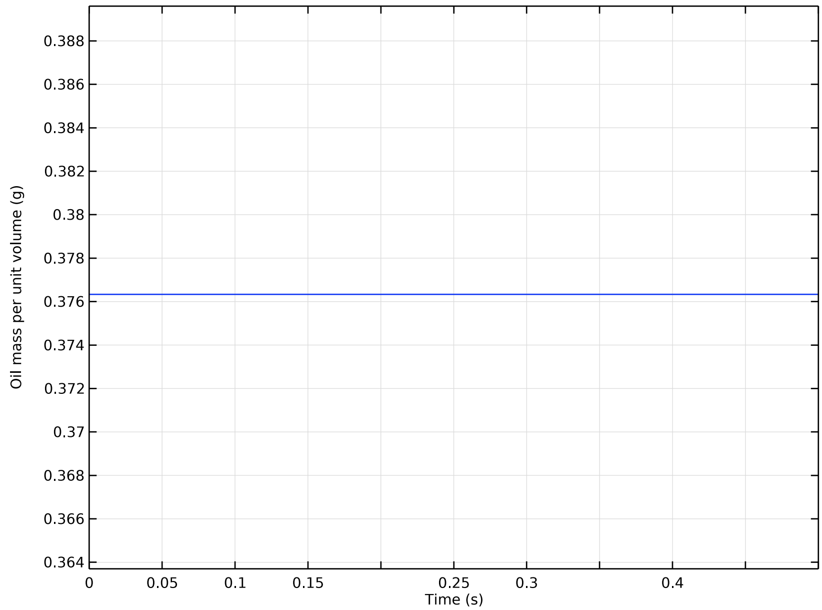

Figure 5: Total mass of oil as a function of time. The total mass loss during the simulation is conserved.

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_Flow/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 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 **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 `5`.
4 In the **Height** text field, type `15`.
5 Click **Build Selected**.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3 From the **Data source** list, choose **Vectors**.
4 In the **r** text field, type `0 5`.
5 In the **z** text field, type `10`.
6 Click **Build Selected**.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

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

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

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

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

7 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**.

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**.

4 In the **Model Builder** window, click **Two-Phase Flow, Level Set 1 (tpf1)**.

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

6 From the **Surface tension coefficient** list, choose **Library coefficient, liquid/liquid interface**.

7 From the list, choose **Olive oil/Water, 20°C**.

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

3 Click **Add Material**.

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

ADD MATERIAL TO PHASE 2 (MPMAT1.PHASE2)

1 Go to the **Add Material to Phase 2 (mpmat1.phase2)** window.

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

3 Click **Add Material**.

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 `0.2`.

Initial Values, Fluid 2

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

2 Select Domain 1 only.

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**, locate the **Physical Model** section.

3 Select the **Include gravity** checkbox.

Pressure Point Constraint 1

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

2 Select Point 8 only.

3 In the **Settings** window for **Pressure Point Constraint**, locate the **Pressure Constraint** section.

4 Clear the **Compensate for hydrostatic pressure** checkbox.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.
2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3 Clear the **Compensate for hydrostatic pressure** checkbox.

MULTIPHYSICS

Wetted Wall 1 (ww1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Wetted Wall 1 (ww1)**.
2 Select Boundaries 9 and 10 only.

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.

DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click a= **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/m^3	Oil mass per unit volume

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 From the **Predefined** list, choose **Finer**.
5 Click **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(0,0.5/50,0.5)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Settings** window for **Solution**, click **Run**.

RESULTS

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

Surface Integration 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration** > **Surface Integration**.

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

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **rho_oil - Oil mass per unit volume - kg/m³**.

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

Expression	Unit	Description
`rho_oil`	`g`	`Oil mass per unit volume`

6 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

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

RESULTS

1D Plot Group 6

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** checkbox.

4 In the **x minimum** text field, type `0`.

5 In the **x maximum** text field, type `0.5`.

6 In the **y minimum** text field, type `0.3637`.

7 In the **y maximum** text field, type `0.3896`.

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

Volume Fraction of Fluid 1 (ls)

1 In the **Model Builder** window, 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 dataset edges** checkbox.

Surface 1

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

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

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

Volume Fraction of Fluid 1 (ls)

To reproduce the plots 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.

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

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

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

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

Repeat the 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.

Slice 1

1 In the **Model Builder** window, expand the **Results** > **Velocity, 3D (spf)** node.

2 Right-click **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 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

Isosurface 1

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

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

3 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 **0.11**.

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

Edge 2D 1

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

2 Select Boundaries 2 and 8–10 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 180.

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**.

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 **Plot Settings** section.

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

4 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

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

Finally, create a movie using the current plot group.

Animation 1

1 In the **Volume Fraction of Fluid 1 (ls) 1** toolbar, click **Animation** and choose **Player**.

2 Click the **Play** button in the **Graphics** toolbar.