Created in COMSOL Multiphysics 6.0

Droplet Rising Through a Suspension

Introduction

This example simulates an oil droplet rising through a suspension. The suspension is initially stratified, with a dense layer between two clear layers. The droplet is initially located in the bottom clear layer, and starts to rise through the different layers of the suspension. The particles in the suspension start to sediment toward the bottom of the flow domain, while a fraction of them is also dragged along with the rising droplet. The interface between the droplet and the suspension is tracked using the level set method, while the particles in suspension are tracked using the dispersed multiphase mixture model.

Model Definition

The flow domain is a cylinder with a diameter of 4 cm and a height of 9 cm. The spherical droplet with a diameter of 1 cm is initially located 0.5 cm from the bottom of the domain. The dense layer of the suspension is initially located between 3 and 5 cm from the bottom of the flow domain. The flow is assumed to be axially symmetric. See Figure 1 below for a graphic representation of the geometry.

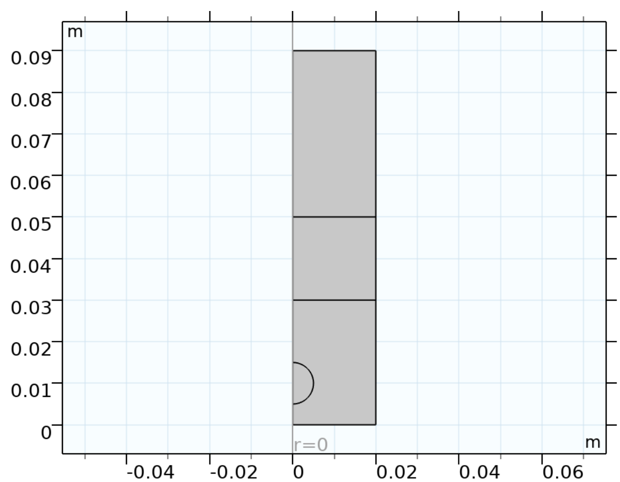

Figure 1: Cross section of the axially symmetric model geometry, with the internal boundaries indicating the initial distribution of the different phases.

The flow field in the suspension and droplet is computed using the Two-Phase Flow, Level Set, Laminar Flow multiphysics interface together with the Phase Transport interface and the Mixture Model multiphysics coupling.

The Mixture Model multiphysics coupling node uses the flow field from the Laminar Flow interface to compute the slip velocity and velocity field of the dispersed phase. This dispersed phase velocity field is then supplied to the Phase Transport interface where it is used to compute the evolution of the distribution of the particles in the suspension.

The suspension consists of water with particles with a diameter of 1 mm and a density of 1100 kg/m^3. The density and viscosity of the oil and water phases are taken from the Transformer oil material and Water, liquid material from the COMSOL's material database. The initial volume fraction of particles in the initial dense layer of the suspension is 0.5.

Results and Discussion

In Figure 2 the volume fraction of the oil phase is plotted at different moments in time. As the oil phase is lighter than the surrounding water phase, the oil droplet starts to rise through the suspension. After an initial startup phase, it travels upward with a more or less constant velocity until, after approximately 0.9 s, the oil droplet has completely left the flow domain through the top outlet boundary. The droplet shape is also seen to change from the initial sphere to a somewhat more flattened shape.

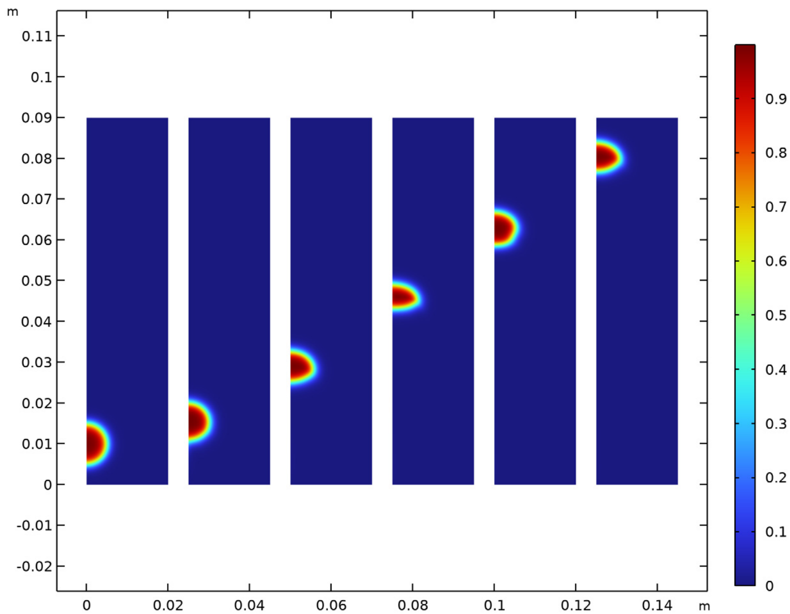

Figure 2: The volume fraction of the oil phase for (left to right) t=0 s, t=0.15 s, t=0.3 s, t=0.45 s, t=0.6 s, and t=0.75 s, respectively. After approximately 0.9 s the droplet has completely left the flow domain.

In Figure 3 the volume fraction of the particles in the suspension are plotted at different moments in time. As the particles in suspension are heavier than the surrounding water phase, they start to sediment downward. The particles at the bottom boundary of the dense layer, settling into a clear water layer, will travel downward faster than the particles at the top and inside the dense layer, which are hindered by the surrounding particles. This causes a rarefaction fan at the bottom of the dense layer, which is clearly seen in the plot for t = 0.3 s and subsequent time instances. In the same plots, however, it can be seen that the rising droplet pushes the particles aside, and also drags along a fraction of the particles upward (see the plots for the subsequent time instances), even all the way to the top of the flow domain. The last plot (for t = 1.5 s) also shows that the sedimentation process has started to form a thin denser layer at the bottom of the flow domain.

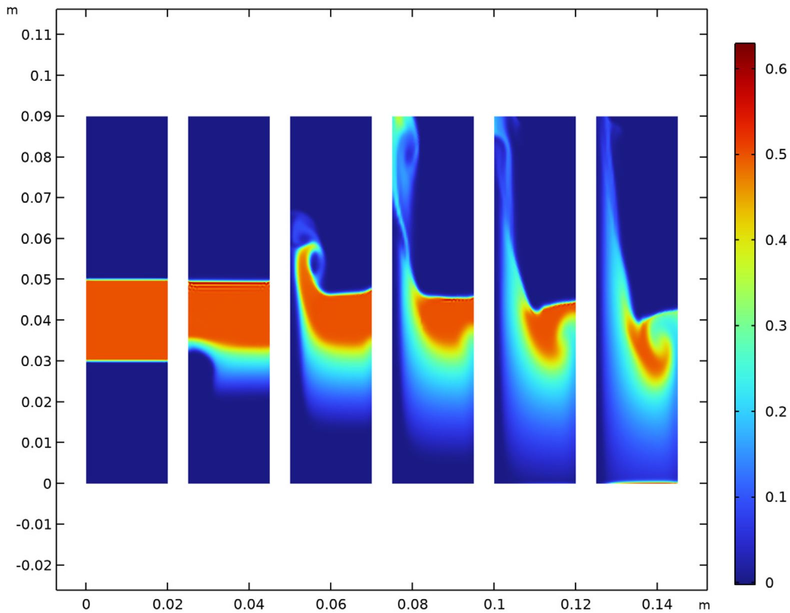

Figure 3: The volume fraction of the particles in the suspension for (left to right) t=0 s, t=0.3 s, t=0.6 s, t=0.9 s, t=1.2 s, and t=1.5 s, respectively.

Application Library path: `CFD_Module/Multiphase_Flow/droplet_suspension`

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 In the **Select Physics** tree, select **Fluid Flow>Multiphase Flow>Phase Transport>Phase Transport (phtr)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Time Dependent with Phase Initialization**.

8 Click **Done**.

GEOMETRY 1

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 `0.02`.

4 In the **Height** text field, type `0.09`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.03
Layer 2	0.02

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 `0.005`.

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

5 Locate the **Position** section. In the **z** text field, type `0.01`.

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

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in>Water, liquid**.

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

5 In the tree, select **Liquids and Gases>Liquids>Transformer oil**.

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

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Transformer oil (mat2)

Select Domain 2 only.

MULTIPHYSICS

Mixture Model 1 (mfmm1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Mixture Model**.

2 In the **Settings** window for **Mixture Model**, locate the **Domain Selection** section.

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

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** check box.

Outlet 1

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

2 Select Boundary 10 only.

3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

4 Clear the **Suppress backflow** check box.

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.

Open Boundary 1

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

2 Select Boundary 10 only.

PHASE TRANSPORT (PHTR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Phase Transport (phtr)**.

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

Open Boundary 1

Select Boundary 10 only.

Initial Values 2

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 3 only.

3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 In the $s_{0,s2}$ text field, type `0.5`.

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 ρ_1 list, choose **User defined**. In the associated text field, type `mfmm1.rho`.

4 From the μ_1 list, choose **Dynamic viscosity, mixture (mfmm1)**.

5 Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, choose **Transformer oil (mat2)**.

6 Locate the **Surface Tension** section. Select the **Include surface tension force in momentum equation** check box.

7 From the **Surface tension coefficient** list, choose **User defined**. In the σ text field, type `3.2e-2[N/m]`.

Wetted Wall 1 (ww1)

1 In the **Model Builder** window, click **Wetted Wall 1 (ww1)**.

2 Select Boundaries 2 and 11–13 only.

Mixture Model 1 (mfmm1)

1 In the **Model Builder** window, click **Mixture Model 1 (mfmm1)**.

2 In the **Settings** window for **Mixture Model**, locate the **Physical Model** section.

3 From the **Slip model** list, choose **Schiller-Naumann**.

4 From the **Mixture viscosity model** list, choose **Krieger type**.

5 Locate the **Continuous Phase Properties** section. From the **Continuous phase** list, choose **Water, liquid (mat1)**.

6 Locate the **Dispersed Phase 2 Properties** section. From the ρ_{s2} list, choose **User defined**. In the associated text field, type `1100[kg/m^3]`.

7 In the $slipvel_{0,s2}$ text field, type `0.05[m^2/s^2]`.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

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(0,0.05,1.5)`.

Solution 1 (sol1)

Perform the next steps to tune manually the solver sequence for better performance. In particular, increase the initialization time-step from the default value since the initial time-step is already rather small. This will make the initialization easier.

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 Find the **Algebraic variable settings** subsection. In the **Fraction of initial step for Backward Euler** text field, type `0.1`.

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

RESULTS

Contour 1

1 In the **Model Builder** window, expand the **Results>Volume Fraction of Fluid 1 (ls)** node.

2 Right-click **Results>Volume Fraction of Fluid 1 (ls)>Contour 1** and choose **Delete**.

3 Click **Yes** to confirm.

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**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Click **Plot First**.

Surface 1

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

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

3 In the **Expression** text field, type `ls.Vf2`.

Surface 2

1 Right-click **Results>Volume Fraction of Fluid 1 (ls)>Surface 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 From the **Time (s)** list, choose **0.15**.

Deformation 1

1 Right-click **Surface 2** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `1`.

5 Locate the **Expression** section. In the **r component** text field, type `0.025`.

Repeat duplicating the **Surface** nodes and adjust the times and deformations to plot the volume fraction at t=0 s, t=0.15 s, t=0.3 s, t=0.45 s, t=0.6 s, and t=0.75 s as in Figure 2.

Volume Fraction (phtr)

To create the plot in Figure 3, perform similar steps for this plot group as explained in the previous section.

Volume Fraction of Fluid 1 (ls) 1

The last steps of the instructions create the plot that is used as the model thumbnail.

1 In the **Model Builder** window, expand the **Results>Volume Fraction of Fluid 1 (ls) 1** node, then 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 **0.5**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

7 Click the **Show Legends** button in the **Graphics** toolbar.

8 Click the **Show Grid** button in the **Graphics** toolbar.

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 **Custom**.

4 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

5 Click **Define custom colors**.

6 Set the RGB values to 196, 106, and 72, respectively.

7 Click **Add to custom colors**.

8 Click **Show color palette only** or **OK** on the cross-platform desktop.

Volume 1

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

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

3 In the **Expression** text field, type `s1`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

5 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D**.

6 From the **Solution parameters** list, choose **From parent**.

Transparency 1

1 Right-click **Volume 1** and choose **Transparency**.

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

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