

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

- I 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 **M** Done.

GEOMETRY I

Rectangle 1 (r1)

- I 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)

- I 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

- I 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 I (mfmmI)

- I In the Physics toolbar, click A 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)

- I In the Model Builder window, under Component I (compl) 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 I

- I 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

- I In the Model Builder window, under Component I (compl)>Level Set (Is) click Initial Values, Fluid 2.
- 2 Select Domain 2 only.

Open Boundary I

- I In the Physics toolbar, click Boundaries and choose Open Boundary.
- **2** Select Boundary 10 only.

PHASE TRANSPORT (PHTR)

I In the Model Builder window, under Component I (compl) click Phase Transport (phtr).

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

Open Boundary I

Select Boundary 10 only.

Initial Values 2

- I 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)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Two-Phase Flow, Level Set I (tpfl).
- 2 In the Settings window for Two-Phase Flow, Level Set, locate the Fluid I Properties section.
- **3** From the p_1 list, choose **User defined**. In the associated text field, type mfmm1.rho.
- **4** From the μ_1 list, choose **Dynamic viscosity, mixture (mfmml)**.
- 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 I (wwI)

- I In the Model Builder window, click Wetted Wall I (wwI).
- **2** Select Boundaries 2 and 11–13 only.

Mixture Model I (mfmm1)

- I In the Model Builder window, click Mixture Model I (mfmmI).
- 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 (matl).
- 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[m²/s²].

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- **3** From the **Element size** list, choose **Fine**.

STUDY I

Step 2: Time Dependent

- I In the Model Builder window, under Study I 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 (soll)

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.

- I In the Study toolbar, click **here** Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **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

Volume Fraction of Fluid 1 (ls)

- I In the Model Builder window, under Results click Volume Fraction of Fluid I (Is).
- 2 In the Settings window for 2D Plot Group, click to expand the Plot Array section.

3 Select the **Enable** check box.

Contour I

- I In the Model Builder window, expand the Volume Fraction of Fluid I (Is) node.
- 2 Right-click Results>Volume Fraction of Fluid I (Is)>Contour I and choose Delete.
- **3** Click **Yes** to confirm.

Volume Fraction of Fluid 1 (ls)

- I In the Model Builder window, under Results click Volume Fraction of Fluid I (Is).
- 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

- I In the Model Builder window, click Surface I.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type 1s.Vf2.

Surface 2

- I Right-click Results>Volume Fraction of Fluid I (Is)>Surface I and choose Duplicate.
- 2 In the Settings window for Surface, locate the Data section.
- **3** From the **Dataset** list, choose **Study I/Solution I (sol1)**.
- 4 From the Time (s) list, choose 0.15.
- 5 Click to expand the Inherit Style section. From the Plot list, choose Surface I.

Repeat duplicating the **Surface** nodes and adjust the times 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 (Is) 1

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

- I In the Model Builder window, expand the Results>Volume Fraction of Fluid I (Is) I node, then click Volume Fraction of Fluid I (Is) I.
- 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

- I In the Model Builder window, click Isosurface I.
- 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 crossplatform 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

- I In the Model Builder window, right-click Volume Fraction of Fluid I (Is) I 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. Click Change Color Table.
- 5 In the Color Table dialog box, select Linear>GrayScale in the tree.
- 6 Click OK.
- 7 In the Settings window for Volume, locate the Data section.
- 8 From the Dataset list, choose Revolution 2D.
- 9 From the Solution parameters list, choose From parent.

Transparency I

- I Right-click Volume I and choose Transparency.
- 2 In the Volume Fraction of Fluid I (Is) I toolbar, click on Plot.
- **3** Click the **F Zoom Extents** button in the **Graphics** toolbar.

12 | DROPLET RISING THROUGH A SUSPENSION