Oil-Water Flow Through an Orifice — A Droplet Population Model

Introduction

This example considers the turbulent flow of an oil-water suspension through an orifice. The oil droplets are broken up into smaller droplets by the turbulent stresses as the suspension passes through the orifice. The aim of this model is to track the distribution of droplet sizes. The droplet size distribution is discretized into five populations of droplets with different diameters. The model uses the Phase Transport Mixture Model, Turbulent Flow, k-ε multiphysics interface, to compute the flow field and the transport of the different droplet populations.

Model Definition

The oil-water suspension flows through a pipe with a radius of 5 cm. The pipe contains an orifice with a radius of 1 cm. The section of the pipe that is taken to be the model geometry is 60 cm long, and the orifice is located 15 cm from the inlet boundary. The flow is assumed to be axially symmetric. See Figure 1 below for a graphic representation of the geometry.

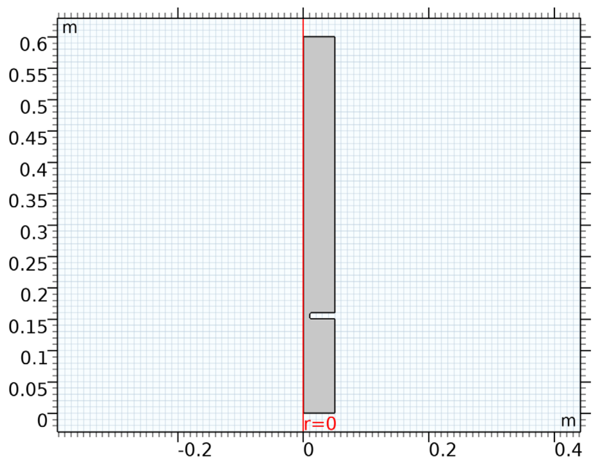

Figure 1: Cross section of the axially symmetric model geometry, which consists of a pipe with an orifice.

The stationary flow field for the mixture is computed using the Turbulent Flow, k-ε interface. The Mixture Model multiphysics coupling node computes from the mixture flow field the velocity fields of the dispersed phases. These dispersed phase velocity fields are used for the transport of the droplet populations in the Phase Transport interface which in this case is set up to solve for the volume fractions s_i of five different droplet sizes (diameters). The diameters in the populations with volume fraction s_1, …, s_5 are chosen to be $d_0/4$, $d_0/2$,d_0, $2d_0$, and $4d_0$, respectively, where the diameter d_0 is an input to the model. The diameter d_0 is taken to be 0.5 mm in this case.

The droplet break-up is modeled as follows: the maximum stable droplet size d_{max} is estimated using the relation

$$d_{\mathrm{max}} = \left(\frac{\sigma}{\rho_c}\right)^{0.6} \varepsilon^{-0.4} \tag{1}$$

where σ denotes the surface tension (taken to be 0.03 N/m), ρ_c the density of the continuous phase, and ε the energy dissipation rate of the flow. This relation can be derived (see Ref. 1) by assuming that a droplet will break up when the droplet Weber number is larger than a critical Weber number. The Weber number for a droplet with diameter d is defined by

$$\mathrm{We} = \frac{\rho_c(\Delta u)^2 d}{\sigma} \tag{2}$$

Here Δu is the difference of the flow velocity across the droplet. Using the relation

$$\Delta u = (\varepsilon d)^{1/3} \tag{3}$$

between Δu and ε, which is valid under the assumption of homogeneous isotropic turbulence, and setting the critical Weber number to 1, the relation in Equation 1 now follows. To derive an expression for the droplet break-up rate, it is assumed that the droplet break-up rate of the population of droplets with diameter d_i is inversely proportional to the turbulent dissipation time τ and proportional to the relative difference between the volumes $(d_i)^3/(6\pi)$ and $(d_{\mathrm{max}})^3/(6\pi)$, whenever d_i is larger than d_{max}. This leads to a mass sink in the conservation equation for the population of droplets with diameter d_i of the form

$$R_i = \frac{\rho_d s_i}{\tau} \max(0, (d_i^3 - d_{\mathrm{max}}^3)/d_i^3) \tag{4}$$

where ρ_d is the density of the dispersed phase. As mass is conserved, as coalescence is not taken into account, and as it is assumed that droplets of diameter d_i break up into droplets with diameter $d_{i-1} = d_i/2$, the total mass source for the population of droplets with diameter d_i is given by

$$Q_i = R_{i+1} - R_i \tag{5}$$

In addition R_1 and R_6 are taken to be zero: the smallest droplets do not break up into even smaller droplets, and there are no larger droplets that break up into droplets with diameter $4d_0$.

The density and viscosity of the continuous and dispersed phases are taken from the Water, liquid and Transformer oil materials from COMSOL's material databases.

The flow field is solved for three different values of the inflow velocity, which are computed from three different total mass flow rates: 1 kg/s, 5 kg/s, and 10 kg/s. The volume fraction of the dispersed phase at the inlet is 0.05, which is distributed over the inlet volume fractions s_{i0} of the different droplet populations as follows:

$$s_{i0} = 0.05\frac{x_i d_i^3}{\sum x_j d_j^3} \tag{6}$$

where the x_i are the fractions of the total inlet number density, given by $x_1 = x_5 = 0.1$, $x_2 = x_4 = 0.2$, and $x_3 = 0.4$.

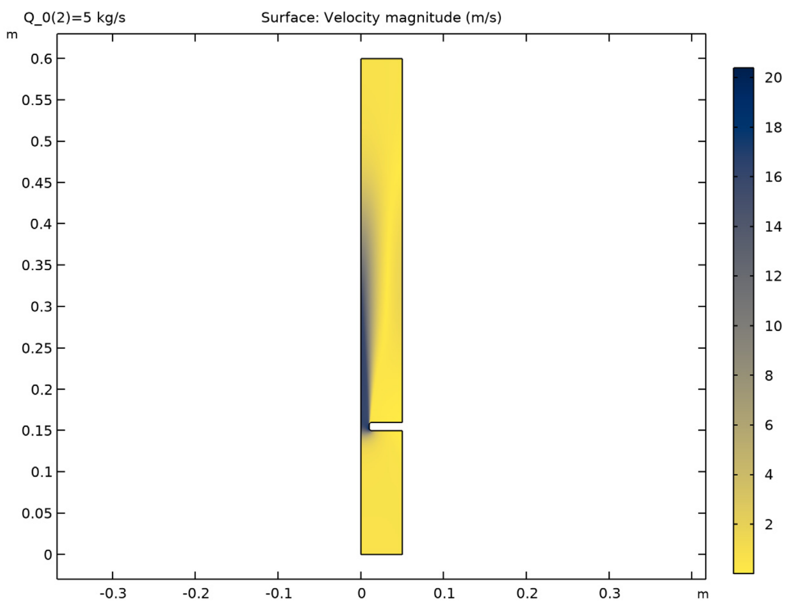

Figure 2: The magnitude of the volume-averaged velocity field of the mixture for a mass flow rate of 5 kg/s.

Results and Discussion

In Figure 2 the magnitude of the volume-averaged velocity field of the mixture is plotted for a mass flow rate of 5 kg/s. It can be seen that the oil-water mixture accelerates as it passes through the orifice, and the flow becomes more turbulent. There are no mass sources for the dispersed phase in the system, so the overall volume fraction of the oil droplets stays more or less constant throughout the pipe. However, due to turbulent stresses the volume fractions of the individual droplet populations change considerably as the mixture flows through the pipe.

.

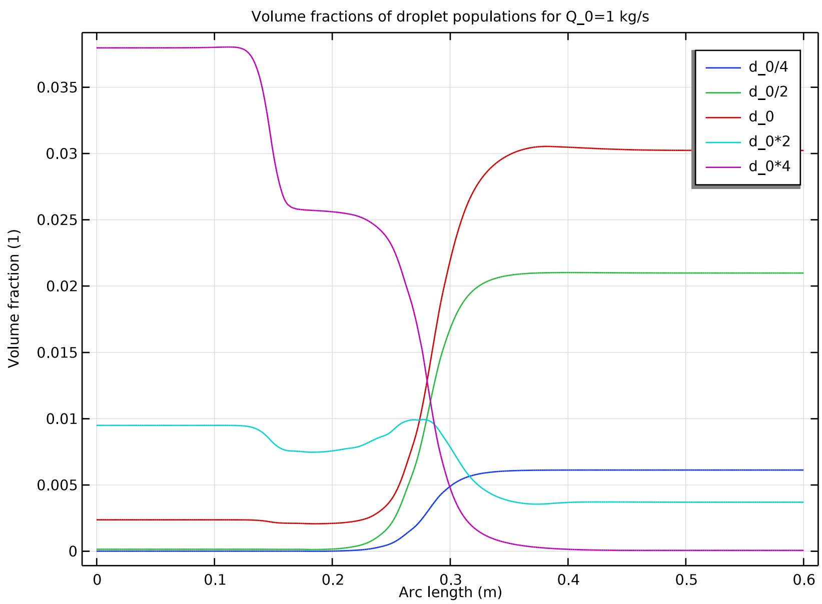

Figure 3: The volume fractions of the five droplet populations along the centerline of the pipe for a mass flow rate of 1 kg/s. The average oil droplet size decreases as the water-oil mixture flows through the orifice.

In Figure 3 the volume fractions of the five droplet populations are plotted along the centerline of the pipe for a mass flow rate of 1 kg/s. The plot shows that the droplet size distribution does not change until the mixture reaches the orifice. When the mixture does reach the orifice, the volume fraction of the population with the largest droplets starts to decrease, indicating that the largest droplets break up into smaller ones, and it decreases until all the largest droplets are broken up. Similarly, the population of the second largest droplets decreases as the mixture passes through the orifice. The populations with the three smallest droplet sizes, however, all increase. This means that, as is expected, the average oil droplet size has decreased after the mixture has passed through the orifice. For higher mass flow rates, it is expected that the average droplet size will decrease even further. Figure 4, which shows the population volume fractions for a mass flow rate of 10 kg/s, illustrates that this is indeed the case: the volume fractions of all droplet populations vanish as the mixture flows through the orifice, except for the population with the smallest droplets.

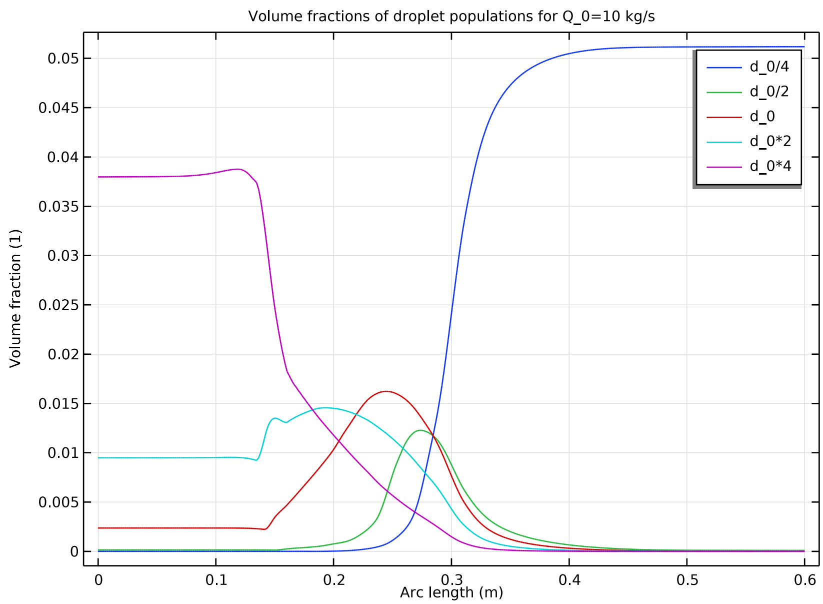

Figure 4: The volume fractions of the five droplet populations along the centerline of the pipe for a mass flow rate of 10 kg/s. Only the population with the smallest droplet size remains after the orifice.

Reference

1. M.J. van der Zande and W.M.G.T. van den Broek, *The Effect of Tubing and Choke Valve on Oil-Droplet Break-up*, proceedings of the 1st North American Conference on Multiphase Technology, Banff, Canada, June 10–11, 1998, pp. 89–100.

Application Library path: `CFD_Module/Multiphase_Flow/droplet_population_model`

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>Phase Transport Mixture Model>Turbulent Flow>Turbulent Flow, k-ε**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `droplet_population_model_parameters.txt`.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `droplet_population_model_variables.txt`.

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 `r_pipe`.
4 In the **Height** text field, type `h_pipe`.

Rectangle 2 (r2)

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 `r_pipe-r_orifice`.
4 In the **Height** text field, type `h_orifice`.
5 Locate the **Position** section. In the **r** text field, type `r_orifice`.
6 In the **z** text field, type `h_orifice_pos`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the object **r1** only.
3 In the **Settings** window for **Difference**, locate the **Difference** section.
4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.
5 Select the object **r2** only.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Fillet**.
2 On the object **dif1**, select Points 3 and 4 only.
3 In the **Settings** window for **Fillet**, locate the **Radius** section.
4 In the **Radius** text field, type `0.003`.
5 Click **Build All Objects**.

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 1 only.

TURBULENT FLOW, K-ε (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.

2 In the **Settings** window for **Turbulent Flow, k-ε**, locate the **Physical Model** section.

3 Select the **Include gravity** check box.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Turbulent Flow, k-ε (spf)** click **Initial Values 1**.

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

3 Specify the **u** vector as

0	r
u_0	z

Inlet 1

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

2 Select Boundary 2 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type u_0.

Outlet 1

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

2 Select Boundary 3 only.

PHASE TRANSPORT (PHTR)

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

2 In the **Settings** window for **Phase Transport**, click to expand the **Dependent Variables** section.

3 In the **Number of phases** text field, type `6`.

4 Locate the **Phases** section. From the **From volume constraint** list, choose **s6**.

Initial Values 1

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

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

3 In the $s_{0,s1}$ text field, type `s1_0`. Similarly, type `s2_0`, `s3_0`, `s4_0`, and `s5_0`, respectively, in the next text fields.

Mass Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Mass Source**.

2 Select Domain 1 only.

3 In the **Settings** window for **Mass Source**, locate the **Mass Source** section.

4 In the Q_{s1} text field, type `R2`.

5 In the Q_{s2} text field, type `R3-R2`.

6 In the Q_{s3} text field, type `R4-R3`.

7 In the Q_{s4} text field, type `R5-R4`.

8 In the Q_{s5} text field, type `-R5`.

Outflow 1

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

2 Select Boundary 3 only.

Volume Fraction 1

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

2 Select Boundary 2 only.

3 In the **Settings** window for **Volume Fraction**, locate the **Volume Fraction** section.

4 Select the **Phase s1** check box.

5 In the $s_{0,s1}$ text field, type `s1_0`.

6 Repeat the last two steps for the remaining phases: select the check boxes, and type `s2_0`, `s3_0`, `s4_0`, and `s5_0`, respectively, in the text fields.

MULTIPHYSICS

Mixture Model 1 (mfmm1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Mixture Model 1 (mfmm1)**.

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

3 From the **Dispersed phase** list, choose **Liquid droplets/bubbles**.

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

5 From the **Mixture viscosity model** list, choose **Volume averaged**.

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

7 Locate the **Dispersed Phase 1 Properties** section. From the **Phase s1** list, choose **Transformer oil (mat2)**.

8 In the d_{s1} text field, type `d_0/4`.

9 Repeat the last two steps for the remaining dispersed phases: choose Transformer oil as material and set the diameter of the droplets of the different phases to `d_0/2`, `d_0`, `d_0*2`, and `d_0*4`, respectively.

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 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Q_0 (Mass flow rate)	1 5 10	kg/s

6 In the **Home** toolbar, click = **Compute**.

RESULTS

1D Plot Group 7

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3 From the **Parameter selection (Q_0)** list, choose **From list**.
4 In the **Parameter values (Q_0 (kg/s))** list, select **1**.
5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
6 In the **Title** text area, type `Volume fractions of droplet populations for Q_0=1 kg/s`.

Line Graph 1

1 In the **1D Plot Group 7** toolbar, click **Line Graph**.
2 Select Boundary 1 only.
3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `s1`.
5 Click to expand the **Legends** section. Select the **Show legends** check box.
6 From the **Legends** list, choose **Manual**.
7 In the table, enter the following settings:

Legends
d_0/4

8 Right-click **Line Graph 1** and choose **Duplicate**.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.
2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3 In the **Expression** text field, type `s2`.
4 Locate the **Legends** section. In the table, enter the following settings:

Legends
d_0/2

Line Graph 3-5

Repeat the four previous instructions for the remaining three phases: duplicate the previous line graph, set the **Expression** text field in the **y-Axis Data** section to `s3_0`, `s4_0`, and s5_0, respectively, and change the settings for the legends accordingly. And then, as a final step to produce Figure 3:

Line Graph 3

1 In the **Model Builder** window, click **Line Graph 3**.

2 In the **1D Plot Group 7** toolbar, click **Plot**.

Velocity (spf)

The following instructions create the plot that is used as the model thumbnail.

Revolution 2D 2

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **Revolution 2D**.

3 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

4 From the **Dataset** list, choose **Exterior Walls**.

5 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type -90.

6 In the **Revolution angle** text field, type 225.

Surface

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

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

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

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

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

Streamline 1

1 In the **Model Builder** window, right-click **Velocity, 3D (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 In the **Points** text field, type 40.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

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

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

7 From the **Color table transformation** list, choose **Reverse**.