

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

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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, $k \cdot \varepsilon$ 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

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

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

We =
$$\frac{\rho_{\rm c}(\Delta u)^2 d}{\sigma}$$
 (2)

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

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

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 with 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 $(d_{\max})^3/(6\pi)$, 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

$$R_{i} = \frac{\rho_{\rm d}s_{i}}{\tau} \max(0, (d_{i}^{3} - d_{\rm max}^{3})/d_{i}^{3})$$
(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

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

3 | OIL-WATER FLOW THROUGH AN ORIFICE — A DROPLET POPULATION MODEL

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

- I 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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

- I In the Home toolbar, click $\partial =$ 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 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 r_pipe.
- 4 In the **Height** text field, type h_pipe.

Rectangle 2 (r2)

- 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 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 I (dif I)

- I In the Geometry toolbar, click P Booleans and Partitions and choose Difference.
- 2 Select the object rI only.
- 3 In the Settings window for Difference, locate the Difference section.
- **4** Find the **Objects to subtract** subsection. Click to select the **Calculate Selection** toggle button.
- 5 Select the object r2 only.

Fillet I (fill)

- I In the **Geometry** toolbar, click *Fillet*.
- 2 On the object difl, 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

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 1 only.

TURBULENT FLOW, $K-\epsilon$ (SPF)

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

- I In the Model Builder window, under Component I (compl)>Turbulent Flow, k-ε (spf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** Specify the **u** vector as

Inlet 1

- I 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 I

- I In the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 3 only.

PHASE TRANSPORT (PHTR)

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

- I In the Model Builder window, under Component I (compl)>Phase Transport (phtr) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- 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 I

- I 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 qs_{s1} text field, type R2.
- **5** In the qs_{s2} text field, type R3-R2.
- 6 In the qs_{s3} text field, type R4-R3.
- 7 In the qs_{s4} text field, type R5-R4.
- 8 In the qs_{s5} text field, type -R5.

Outflow I

- I In the Physics toolbar, click Boundaries and choose Outflow.
- 2 Select Boundary 3 only.

Volume Fraction 1

- I 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 sI 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 I (mfmm1)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Mixture Model I (mfmml).
- 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 (matl).
- 7 Locate the Dispersed Phase I Properties section. From the Phase sI 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 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 1: Stationary

- I In the Model Builder window, under Study I click Step I: 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

ID Plot Group 7

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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 I.
- 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 I

- I In the ID 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

Line Graph 2

- I Right-click Line Graph I and choose Duplicate.
- 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

- I In the Model Builder window, click Line Graph 3.
- 2 In the ID Plot Group 7 toolbar, click 💿 Plot.

Velocity (spf)

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

Revolution 2D 2

- I 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

- I 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

- I 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

- I Right-click Streamline I 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.