Contaminant-Removal from Wastewater in a Secondary Clarifier

Introduction

Wastewater treatment is a several-step process for removing contaminants. First, large, solid particles are removed through sedimentation, flotation, and filtration. In a second step, biological treatment causes the smaller particles to aggregate, forming so called flocs. These flocs can more easily be removed, for instance through sedimentation. The present example studies the separation of flocs from water in a circular secondary clarifier. To model the turbulent multiphase flow in the tank, this example uses the Mixture Model, Turbulent Flow interface.

Model Definition

The model geometry is shown in Figure 1.

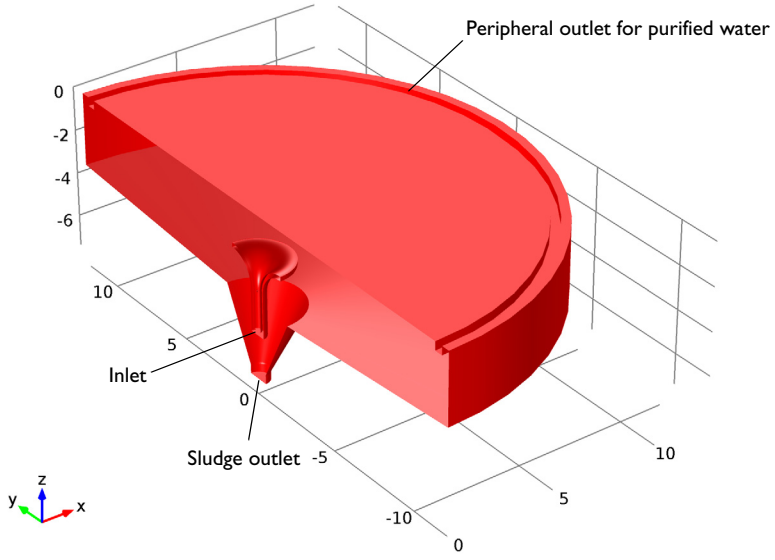


Figure 1: Cross section of the circular clarifier.

The clarifier has a diameter of 24 m and a gently sloping bottom, which makes the dept vary between 3.3 and 4 m, attached to a funnel at the center of the tank. The incoming sludge, consisting of a mixture of solid flocs and water, enters through the inlet in the middle of the tank. In the clarifier, gravity causes the flocs to settle to the bottom of the tank. The tank contains two outlets. One is located at the center, at the bottom of the funnel. The purpose of this outlet is to remove the sedimented flocs from the tank. There is also a peripheral outlet for the purified water as shown in the figure. Figure 2 shows the corresponding 2D axisymmetric model.

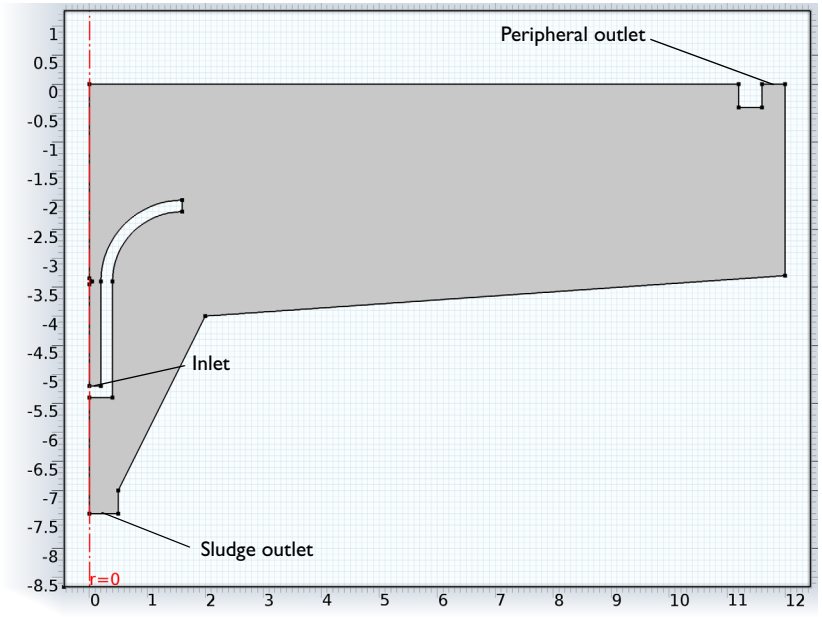


Figure 2: Axisymmetric representation of the clarifier geometry.

The mixture enters the clarifier in the form of a jet. The Reynolds number based on the inlet velocity and diameter is $5 \cdot 10^5$, which means that the flow will be turbulent. Upon impact with the free surface, the mixture spreads out, causing the turbulence production to diminish with radial distance from the inlet. The turbulent flow in the tank tends to mix the phases together, and thus has a negative effect on the separation process. The object of this example is to study the turbulent multiphase flow within the circular secondary clarifier.

For simplicity, you can model the flocs as spherical solid particles of equal size. To solve for the mixture velocity, pressure and the phase volume fractions, you can use the Mixture Model, Turbulent Flow interface. The mixture model is a multiphase-flow

 ©2012 COMSOL

model, particularly well suited for suspensions of solid particles in liquid at low particle volume fractions. For the slip velocity, you can use the Hadamard-Rybczynski drag law. See Theory for the Mixture Model User Interfaces in the *CFD Module User's Guide* for more information.

BOUNDARY CONDITIONS

At the inlet, the velocity is fixed to 1.25 m/s and the dispersed phase volume fraction is 0.003. The turbulence intensity and length scale are set to 5% and $0.07*r_{in}$ (see Inlet Values for the Turbulence Length Scale and Turbulent Intensity in the *CFD Module User's Guide*), where r_{in}=0.2 m is the radius of the inlet. The velocity at the bottom outlet is set to 0.05 m/s, while a constant pressure is set at the peripheral outlet. A slip condition is applied on the free surface and an axial symmetry condition on the centerline.

INITIAL CONDITIONS

Initially, the velocity as well as the solid phase volume fraction is zero in the entire clarifier.

Results and Discussion

Figure 3 shows streamlines of the mixture velocity and the dispersed phase volume fraction after 12 hours, at which time the solution is close to the steady state. Opposing effects of gravity settling and turbulence induced particle dispersion produce volume-fraction gradients in the interior. The magnitude of the mixture strain rate (and hence the turbulence production) decreases with radial distance from the

centerline, and at the peripheral outlet settling dominates over turbulent dispersion. This allows for a relatively clear efflux.

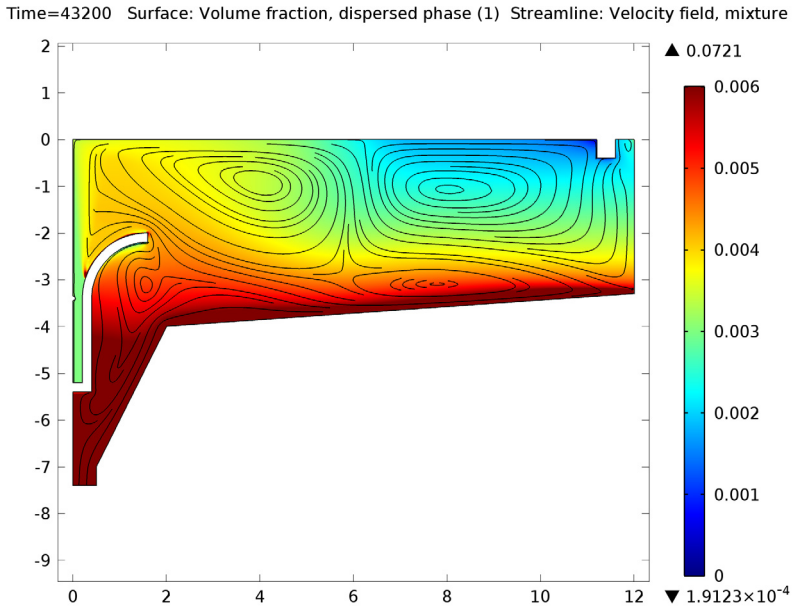


Figure 3: Mixture-velocity streamlines and solid phase volume fraction after 12 hours, when the flow has reached a steady state solution.

As you can see in Figure 3, the maximum volume fraction is only about 7.2 %. Hence, the Mixture Model should be sufficiently accurate. See Two-Phase Flow Modeling of a Dense Suspension in the Model Library on how to modify the mixture model for high volume fractions or use the Euler-Euler model for such cases.

 ©2012 COMSOL

Figure 4 shows the dispersed phase mass flux at the inlet and the two outlets.

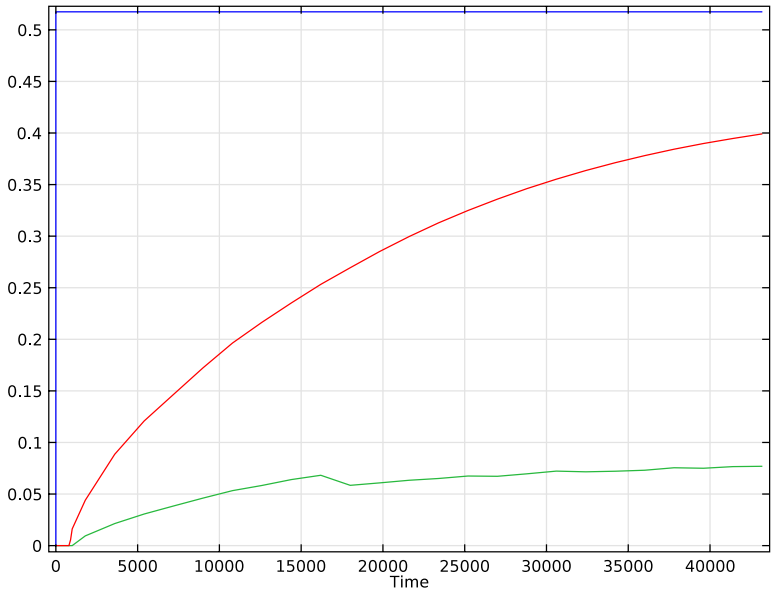


Figure 4: Mass flux of the dispersed phase at the inlet (blue), peripheral outlet (green) and central outlet (red).

The mass flux of the dispersed phase is given by

$$M_{\mathrm{d}} = \int \rho_{\mathrm{d}} \phi_{\mathrm{d}} \mathbf{u}_{\mathrm{d}} \cdot \mathbf{n} dS$$

Computing the removal rate from the results shows that the clarifier removes $0.52 - 0.075 = 0.45$ kg solid particles per second. The separation efficiency is thus higher than 85%.

Figure 5 shows a cut through the swept-out volume of the clarifier with streamlines for each phase after 12 hours.

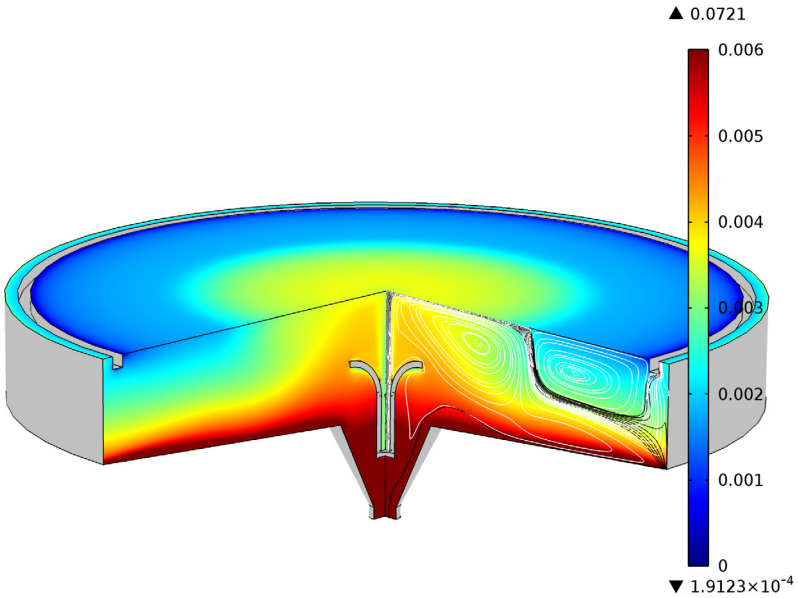


Figure 5: Volume fraction of the dispersed phase and streamlines for the dispersed (black) and continuous (white) velocity fields.

To further examine the performance of the clarifier, you can easily modify the model in several ways. You can, for instance, modify the geometry by adding baffles, changing the inlet and outlet velocities, increasing the dispersed-phase volume fraction in the incoming sludge, or changing the density and size of the dispersed particles.

Notes About the COMSOL Implementation

It is straightforward to set up a multiphase flow model with the Mixture Model, Turbulent Flow interface. To simplify the startup of the transient calculation, you can gradually increase the inlet and outlet velocities from zero to their constant values. For this purpose, use a Step function feature to implement a smooth step function that gradually increases from zero to one.

Model Library path: `CFD_Module/Multiphase_Tutorials/sedimentation`

 ©2012 COMSOL

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D axisymmetric** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Fluid Flow>Multiphase Flow>Mixture Model>Mixture Model, Laminar Flow (mm)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

7 Click **Finish**.

GEOMETRY 1

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **2**, set **r** to 12.

5 Find the **Added segments** subsection. Click the **Add Linear** button.

6 Find the **Control points** subsection. In row **2**, set **z** to -3.3.

7 Find the **Added segments** subsection. Click the **Add Linear** button.

8 Find the **Control points** subsection. In row **2**, set **r** to 2.

9 In row **2**, set **z** to -4.

10 Find the **Added segments** subsection. Click the **Add Linear** button.

11 Find the **Control points** subsection. In row **2**, set **r** to 0.5.

12 In row **2**, set **z** to -7.

13 Find the **Added segments** subsection. Click the **Add Linear** button.

14 Find the **Control points** subsection. In row **2**, set **z** to -7.4.

15 Find the **Added segments** subsection. Click the **Add Linear** button.

16 Find the **Control points** subsection. In row **2**, set **r** to 0.

17 Find the **Added segments** subsection. Click the **Add Linear** button.

18 Find the **Control points** subsection. Click the **Close Curve** button.

Bézier Polygon 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **z** to `-5.4`.

5 In row **2**, set **r** to `0.4`.

6 In row **2**, set **z** to `-5.4`.

7 Find the **Added segments** subsection. Click the **Add Linear** button.

8 Find the **Control points** subsection. In row **2**, set **z** to `-3.4`.

9 Find the **Added segments** subsection. Click the **Add Quadratic** button.

10 Find the **Control points** subsection. In row **2**, set **z** to `-2.2`.

11 In row **3**, set **r** to `1.6`.

12 In row **3**, set **z** to `-2.2`.

13 Find the **Added segments** subsection. Click the **Add Linear** button.

14 Find the **Control points** subsection. In row **2**, set **z** to `-2`.

15 Find the **Added segments** subsection. Click the **Add Quadratic** button.

16 Find the **Control points** subsection. In row **2**, set **r** to `0.2`.

17 In row **3**, set **r** to `0.2`.

18 In row **3**, set **z** to `-3.4`.

19 Find the **Added segments** subsection. Click the **Add Linear** button.

20 Find the **Control points** subsection. In row **2**, set **z** to `-5.2`.

21 Find the **Added segments** subsection. Click the **Add Linear** button.

22 Find the **Control points** subsection. In row **2**, set **r** to `0`.

23 Find the **Added segments** subsection. Click the **Add Linear** button.

24 Find the **Control points** subsection. Click the **Close Curve** button.

Circle 1

1 Right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `0.05`.

4 Locate the **Position** section. In the **z** edit field, type `-3.4`.

 ©2012 COMSOL

5 Click the **Build Selected** button.

Square 1

1 Right-click **Geometry 1** and choose **Square**.

2 In the **Square** settings window, locate the **Size** section.

3 In the **Side length** edit field, type `0.4`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

5 In the **r** edit field, type `11.4`.

6 In the **z** edit field, type `-0.2`.

7 Click the **Build Selected** button.

Difference 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.

2 Select the object **b1** only to add it to the **Objects to add** list.

3 In the **Difference** settings window, locate the **Difference** section.

4 Under **Objects to subtract**, click **Activate Selection**.

5 Select the objects **b2**, **c1**, and **sq1** only.

Bézier Polygon 3

1 Right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **r** to `0.19`.

5 In row **1**, set **z** to `-3.22`.

6 In row **2**, set **r** to `0.2`.

7 In row **2**, set **z** to `-2.9`.

Bézier Polygon 4

1 Right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **r** to `0.2`.

5 In row **1**, set **z** to `-2.9`.

6 In row **2**, set **r** to `0.35`.

7 In row **2**, set **z** to `-0.25`.

Bézier Polygon 5

1 Right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **r** to `0.2`.

5 In row **1**, set **z** to `-0.1`.

6 In row **2**, set **r** to `7.6`.

7 In row **2**, set **z** to `-0.6`.

Use **Mesh Control Edges** to obtain a mesh which is aligned with the turbulent shear.

8 Right-click **Geometry 1** and choose **Virtual Operations>Mesh Control Edges**.

Mesh Control Edges 1

1 In the **Model Builder** window, under **Model 1>Geometry 1** click **Mesh Control Edges 1**.

2 On the object **fin**, select Boundaries 8, 10, and 11 only.

3 Click the **Build All** button.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
rho_c	1000[kg/m^3]	Continuous phase density
mu_c	1e-3[Pa*s]	Continuous phase viscosity
rho_d	1100[kg/m^3]	Dispersed phase density
d_d	0.2[mm]	Dispersed phase particle diameter

DEFINITIONS

Add a **Step** function feature to use for implementing a gradual increase of the inlet and outlet velocities from zero to their constant values.

Step 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Functions>Step**.

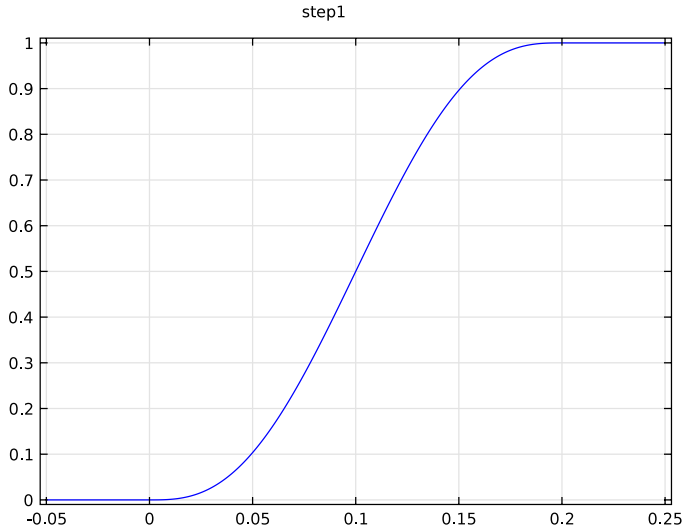
 ©2012 COMSOL

2 In the **Step** settings window, click to expand the **Smoothing** section.

3 Locate the **Parameters** section. In the **Location** edit field, type `0.1`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** edit field, type `0.2`.

5 Click the **Plot** button.



Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
v_in	1.25*step1(t[1/s])[m/s]	Inlet velocity
v_out	-0.05*step1(t[1/s])[m/s]	Outlet velocity
phid_in	0.003	Inlet dispersed phase volume fraction
qd_out	2*pi*r*(mm.udr*nr+mm.udz*nz)*phid*mm.rhod	Dispersed phase mass-outflux

MIXTURE MODEL, LAMINAR FLOW

1 In the **Model Builder** window, under **Model 1** click **Mixture Model, Laminar Flow**.

2 In the **Mixture Model, Laminar Flow** settings window, locate the **Physical Model** section.

3 From the **Slip model** list, choose **Hadamard-Rybczynski**.

4 From the **Turbulence model type** list, choose **RANS, k-ε**.

MIXTURE MODEL, TURBULENT FLOW

Mixture Properties 1

1 In the **Model Builder** window, expand the **Model 1>Mixture Model, Turbulent Flow** node, then click **Mixture Properties 1**.

2 In the **Mixture Properties** settings window, locate the **Continuous Phase Properties** section.

3 From the ρ_c list, choose **User defined**. In the associated edit field, type `rho_c`.

4 From the μ_c list, choose **User defined**. In the associated edit field, type `mu_c`.

5 Locate the **Dispersed Phase Properties** section. From the ρ_d list, choose **User defined**. In the associated edit field, type `rho_d`.

6 In the d_d edit field, type `d_d`.

Gravity 1

1 In the **Model Builder** window, right-click **Mixture Model, Turbulent Flow** and choose **Gravity**.

2 Select Domain 1 only.

Wall 2

1 Right-click **Mixture Model, Turbulent Flow** and choose **Wall**.

2 Select Boundary 7 only.

3 In the **Wall** settings window, locate the **Mixture Boundary Condition** section.

4 From the **Mixture boundary condition** list, choose **Slip**.

Inlet 1

1 Right-click **Mixture Model, Turbulent Flow** and choose **Inlet**.

2 Select Boundary 5 only.

3 In the **Inlet** settings window, locate the **Mixture Boundary Condition** section.

4 In the $\mathbf{u}_0$ table, enter the following settings:

0	r
v_in	z

 ©2012 COMSOL

5 In the L_T edit field, type `0.2*0.07`.

6 Locate the **Dispersed Phase Boundary Condition** section. In the ϕ_{d0} edit field, type `phid_in`.

Outlet 1

1 Right-click **Mixture Model, Turbulent Flow** and choose **Outlet**.

2 Select Boundary 2 only.

3 In the **Outlet** settings window, locate the **Mixture Boundary Condition** section.

4 From the **Mixture boundary condition** list, choose **Velocity**.

5 In the $\mathbf{u}_0$ table, enter the following settings:

0	r
v_out	z

Outlet 2

1 Right-click **Mixture Model, Turbulent Flow** and choose **Outlet**.

2 Select Boundary 17 only.

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extra fine**.

4 From the **Sequence type** list, choose **User-controlled mesh**.

Size 1

In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Size 1** and choose **Build Selected**.

Size 2

1 Right-click **Mesh 1** and choose **Size**.

2 Select Boundaries 23–25 only.

3 In the **Size** settings window, locate the **Element Size** section.

4 From the **Calibrate for** list, choose **Fluid dynamics**.

5 From the **Predefined** list, choose **Extremely fine**.

Corner Refinement 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Corner Refinement 1** and choose **Disable**.

2 Right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

2 In the **Time Dependent** settings window, locate the **Study Settings** section.

3 In the **Times** edit field, type `range(0,0.01,0.1) range(1,10) 100*range(1,10) 1800*range(1,24)`.

4 Select the **Relative tolerance** check box.

5 In the associated edit field, type `0.01`.

6 Click to expand the **Results While Solving** section. Select the **Plot** check box.

Solver 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

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

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

4 From the **Method** list, choose **Generalized alpha**.

5 In the **Amplification for high frequency** edit field, type `0.5`.

6 From the **Predictor** list, choose **Constant**.

7 Select the **Initial step** check box.

8 In the associated edit field, type `0.001`.

RESULTS

2D Plot Group 1

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 1** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Mixture Model, Turbulent Flow>Volume fraction, dispersed phase (phid)**.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** edit field, type `0`.

 ©2012 COMSOL

6 In the **Maximum** edit field, type `0.006`.

7 Click the **Zoom Extents** button on the Graphics toolbar.

8 In the **Model Builder** window, right-click **2D Plot Group 1** and choose **Streamline**.

9 In the **Streamline** settings window, locate the **Streamline Positioning** section.

10 From the **Positioning** list, choose **Uniform density**.

11 In the **Separating distance** edit field, type `0.02`.

MIXTURE MODEL, TURBULENT FLOW

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Mixture Model, Turbulent Flow** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the *p* edit field, type `-g_const*z*rho_c`.

RESULTS

2D Plot Group 1

1 In the **Model Builder** window, under **Results** click **2D Plot Group 1**.

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

3 Click the **Go to Source** button.

DEFINITIONS

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

2 Click the **Zoom Extents** button on the Graphics toolbar.

Derived Values

Calculate the inflow and outflow rates of the dispersed phase. Start with the inflow rate.

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Integration>Line Integration**.

2 Select Boundary 5 only.

3 In the **Line Integration** settings window, locate the **Expression** section.

4 In the **Expression** edit field, type `-qd_out`.

The negative sign is used since `qd_out` is defined as the outward flux.

5 Click the **Evaluate** button.

The result shown in **Table 1** should be close to 0.52 kg/s.

6 In the **Model Builder** window, right-click **Derived Values** and choose **Integration>Line Integration**.

7 Select Boundary 17 only.

8 In the **Line Integration** settings window, locate the **Expression** section.

9 In the **Expression** edit field, type `qd_out`.

10 Right-click **Results>Derived Values>Line Integration 2** and choose **Evaluate>Table 1 - Line Integration 1 (-qd_out)**.

The result should be close to 0.075 kg/s.

11 Right-click **Derived Values** and choose **Integration>Line Integration**.

12 Select Boundary 2 only.

13 In the **Line Integration** settings window, locate the **Expression** section.

14 In the **Expression** edit field, type `qd_out`.

15 Right-click **Results>Derived Values>Line Integration 3** and choose **Evaluate>Table 1 - Line Integration 1 (-qd_out)**.

16 In the **Table** window, click **Table Graph**.

Data Sets

1 In the **Model Builder** window, expand the **Data Sets** node.

2 Right-click **Solution 1** and choose **Add Selection**.

3 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 1 only.

6 In the **Model Builder** window, right-click **Data Sets** and choose **Revolution 2D**.

7 In the **Revolution 2D** settings window, click to expand the **Revolution Layers** section.

8 In the **Revolution angle** edit field, type `270`.

9 Right-click **Data Sets** and choose **Edge 2D**.

10 Select Boundaries 3, 5, 8–16, and 18–22 only.

11 Right-click **Data Sets** and choose **Revolution 2D**.

12 In the **Revolution 2D** settings window, locate the **Data** section.

 ©2012 COMSOL

13 From the **Data set** list, choose **Edge 2D 2**.

14 Click to expand the **Revolution Layers** section. In the **Revolution angle** edit field, type `270`.

15 Click the **Plot** button.

3D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Revolution 2D 3**.

4 Right-click **Results>3D Plot Group 3** and choose **Surface**.

5 In the **Surface** settings window, locate the **Data** section.

6 From the **Data set** list, choose **Revolution 2D 4**.

7 Locate the **Expression** section. In the **Expression** edit field, type `1`.

8 Select the **Description** check box.

9 Clear the associated edit field.

10 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

12 In the **Model Builder** window, right-click **3D Plot Group 3** and choose **Volume**.

13 In the **Volume** settings window, locate the **Data** section.

14 From the **Data set** list, choose **Revolution 2D 3**.

15 Locate the **Expression** section. Select the **Description** check box.

16 Click to expand the **Range** section. Select the **Manual color range** check box.

17 In the **Minimum** edit field, type `0`.

18 In the **Maximum** edit field, type `0.006`.

19 Right-click **3D Plot Group 3** and choose **Streamline**.

20 In the **Streamline** settings window, locate the **Expression** section.

21 In the **x component** edit field, type `mm.udr`.

22 In the **y component** edit field, type `0`.

23 In the **z component** edit field, type `mm.udz`.

24 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.

25 In the **x** edit field, type `range(0.01,0.02,0.19)`.

26 In the **y** edit field, type `0`.

27 In the **z** edit field, type `-1*1^range(1,10)`.

28 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

29 Locate the **Expression** section. Select the **Description** check box.

30 In the associated edit field, type `dispersed phase (black)`.

31 Right-click **3D Plot Group 3** and choose **Streamline**.

32 In the **Streamline** settings window, locate the **Expression** section.

33 In the **x component** edit field, type `mm.ucr`.

34 In the **y component** edit field, type `0`.

35 In the **z component** edit field, type `mm.ucz`.

36 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.

37 In the **x** edit field, type `range(0,0.02,0.2) range(0.5,0.5,12)`.

38 In the **y** edit field, type `0`.

39 In the **z** edit field, type `-1^range(1,35)`.

40 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

41 Click the **Plot** button.

42 Locate the **Expression** section. Select the **Description** check box.

43 In the associated edit field, type `continuous phase (white)`.

View 3D 2

1 In the **Model Builder** window, expand the **Results>Views** node, then click **View 3D 2**.

2 In the **View 3D** settings window, locate the **View** section.

3 Clear the **Show grid** check box.

4 Clear the **Show axis orientation** check box.

 ©2012 COMSOL