



# Contaminant-Removal from Wastewater in a Secondary Clarifier by Sedimentation

## Introduction

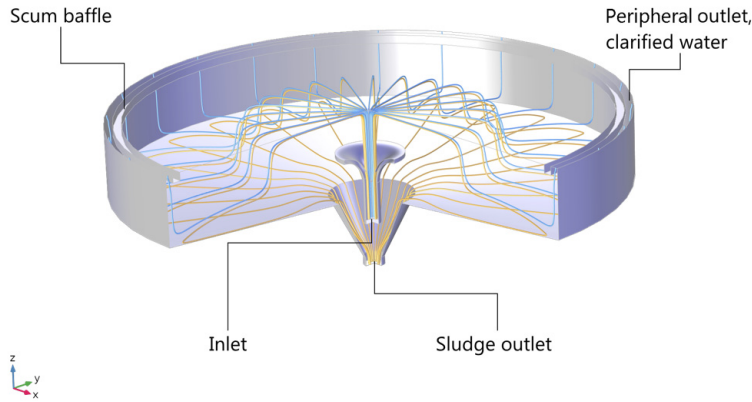
---

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 Phase Transport Mixture Model, Turbulent Flow,  $k$ - $\varepsilon$  multiphysics interface.

## Model Definition

---

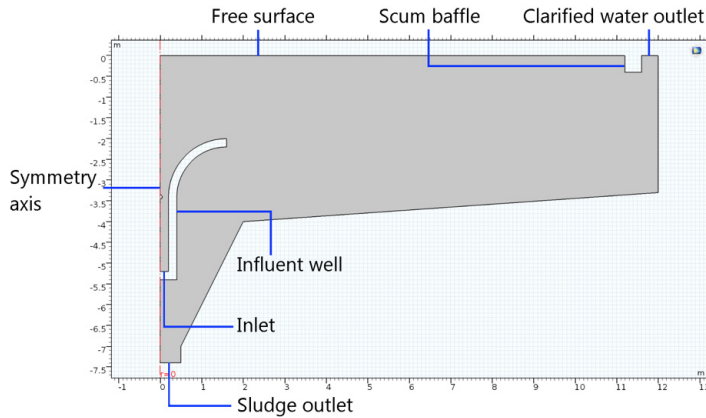
The model geometry is shown in [Figure 1](#).



*Figure 1: Cross section of the circular clarifier.*

The clarifier has a diameter of 24 m and a gently sloping bottom, which makes the depth 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 is 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 aim 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 and pressure you can use the Turbulent Flow Model,  $k$ - $\epsilon$  interface. Phase Transport interface can be used to solve the phase volume fractions. See the theory 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 is set to 5% (medium) and the turbulence length scale is automatically computed from the geometry. 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 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.

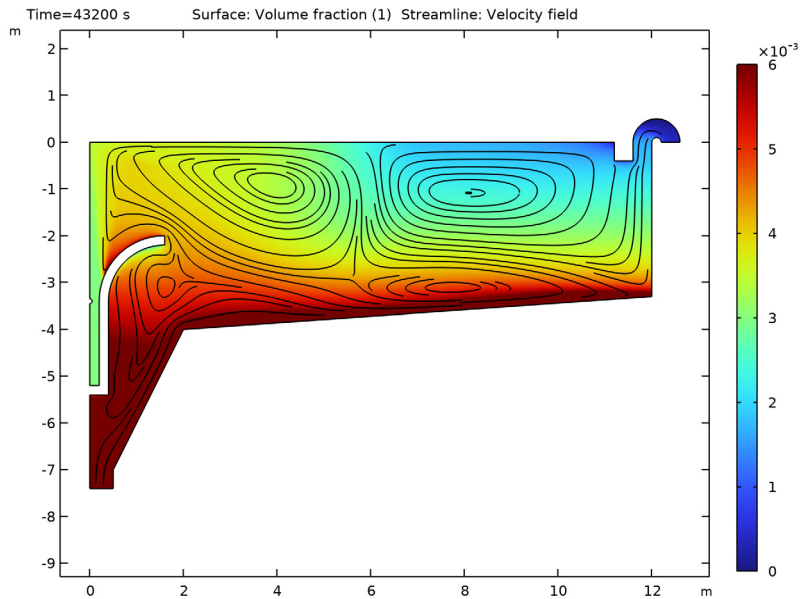


Figure 3: Mixture-velocity streamlines and solid phase volume fraction after 12 hours, when the flow has reached a steady-state solution.

As you can see in Figure 3, the maximum volume fraction is less than 1%.

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

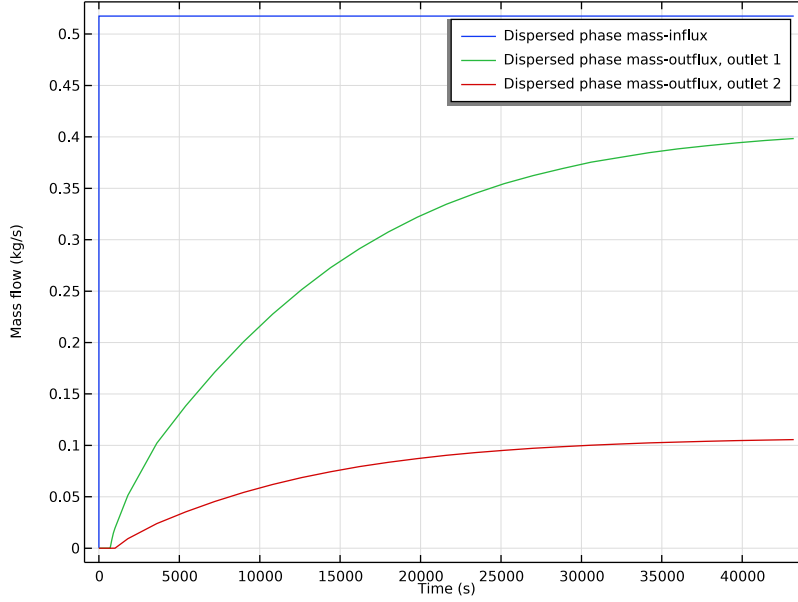


Figure 4: Mass flux of the dispersed phase at the inlet (blue), peripheral outlet (green), and central outlet (red).

The mass flux of the dispersed phase is given by

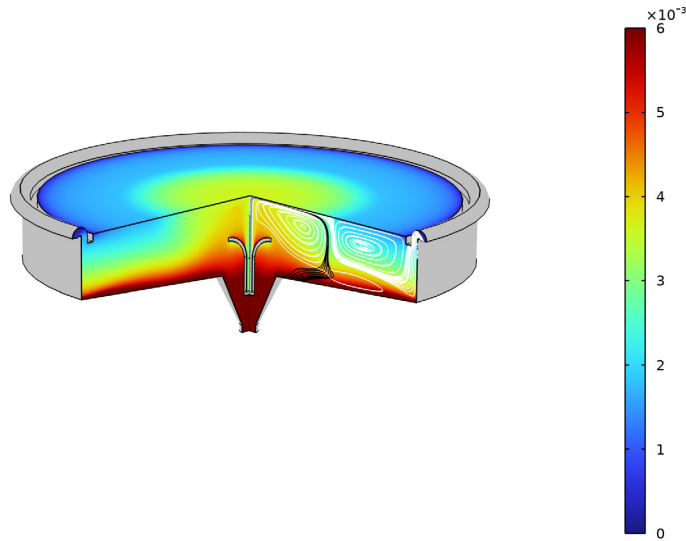
$$M_d = \int \rho_d \mathbf{j}_{d, \text{eff}} \cdot \mathbf{n} dS$$

where  $\mathbf{j}_{d, \text{eff}}$  is the effective dispersed phase flux

$$\mathbf{j}_{d, \text{eff}} = \phi_d (\mathbf{j} + (1 - \phi_d) \mathbf{u}_{\text{slip}}) - D_{md} \nabla \phi_d$$

Computing the removal rate from the results shows that the clarifier removes  $0.52 - 0.10 = 0.42$  kg solid particles per second. The separation efficiency is about 81%.

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 Phase Transport 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.

---

**Application Library path:** CFD\_Module/Multiphase\_Flow/sedimentation\_ptmm


---

## 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>Time Dependent**.
- 6 Click  **Done**.


### GEOMETRY I

To simplify the instructions, import a ready-made geometry sequence. The complete step-by-step instructions for the geometry can be found in the [Appendix — Geometry Modeling Instructions](#).

- 1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
- 2 Browse to the model's Application Libraries folder and double-click the file `sedimentation_ptmm_geom_sequence.mph`.
- 3 In the **Geometry** toolbar, click  **Build All**.

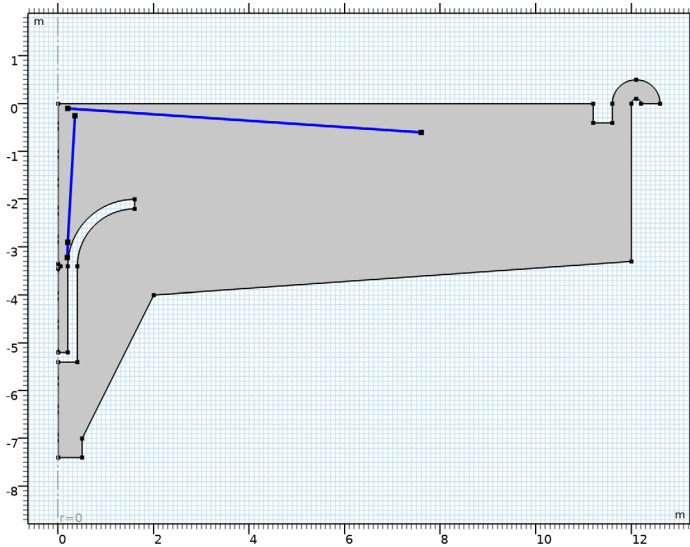
Use **Mesh Control Edges** to obtain a mesh that is aligned with the turbulent shear.


#### *Mesh Control Edges I (mceI)*


- 1 In the **Geometry** toolbar, click  **Virtual Operations** and choose **Mesh Control Edges**.

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

It might be easier to select the boundaries by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



- 3 In the **Geometry** toolbar, click  **Build All**.

- 4 Click the  **Zoom Extents** button in the **Graphics** toolbar.

## GLOBAL DEFINITIONS

### *Parameters I*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.





3 In the table, enter the following settings:

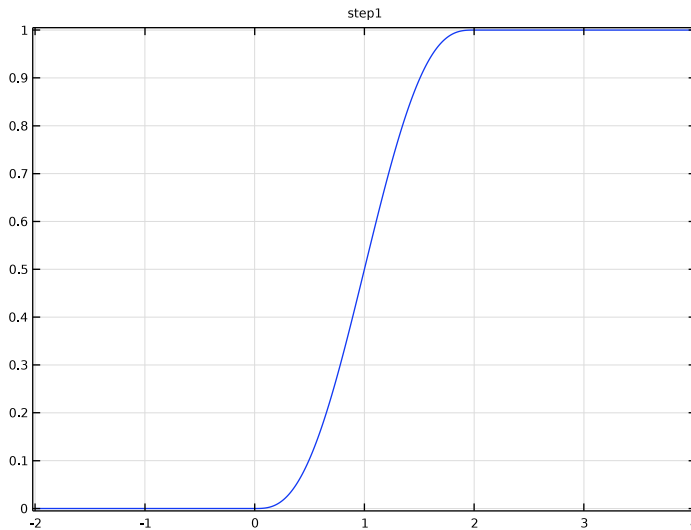
Name	Expression	Value	Description
rho_c	1000[kg/m^3]	1000 kg/m <sup>3</sup>	Continuous phase density
mu_c	1e-3[Pa*s]	0.001 Pa·s	Continuous phase viscosity
rho_d	1100[kg/m^3]	1100 kg/m <sup>3</sup>	Dispersed phase density
d_d	0.2[mm]	2E-4 m	Dispersed phase particle diameter

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

## DEFINITIONS

### Step 1 (step1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Local>Step**.
- 2 In the **Settings** window for **Step**, click to expand the **Smoothing** section.
- 3 Locate the **Parameters** section. In the **Location** text field, type 1.
- 4 Locate the **Smoothing** section. In the **Size of transition zone** text field, type 2.
- 5 Click  **Plot**.



### 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 In the table, enter the following settings:

Name	Expression	Unit	Description
v_in	$1.25 \cdot \text{step1}(t[1/s])$ [m/s]	m/s	Inlet velocity
v_out	$0.05 \cdot \text{step1}(t[1/s])$ [m/s]	m/s	Outlet velocity
phid_in	$0.003 \cdot \text{step1}(t[1/s])$		Inlet dispersed phase volume fraction
qd_out	$2 \cdot \pi \cdot r^2 \cdot (mfmm1.u_{s2} \cdot r + mfmm1.u_{s2z} \cdot n_z) \cdot mfmm1.\rho_{s2}$	kg/(m·s)	Dispersed phase mass-outflux


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

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k- $\epsilon$  (spf)**.
- 2 In the **Settings** window for **Turbulent Flow, k- $\epsilon$** , 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- $\epsilon$  (spf)** click **Initial Values 1**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the  $p$  text field, type  $-g_{\text{const}} \cdot z \cdot \rho_{s,c}$ .
- 4 Clear the **Compensate for hydrostatic pressure approximation** check box.

#### Wall 2


- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Wall**.
- 2 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.
- 3 From the **Wall condition** list, choose **Slip**.
- 4 Select Boundary 7 only.

#### Inlet 1


- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
- 4 In the  $U_0$  text field, type v\_in.
- 5 Locate the **Turbulence Conditions** section. From the  $I_T$  list, choose **User defined**.

- 6 From the  $L_T$  list, choose **User defined**.
- 7 In the text field, type  $0.2 \times 0.07$ .
- 8 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 9 In the **Show More Options** dialog box, select **Physics>Advanced Physics Options** in the tree.
- 10 In the tree, select the check box for the node **Physics>Advanced Physics Options**.
- 11 Click **OK**.
- 12 In the **Settings** window for **Inlet**, click to expand the **Constraint Settings** section.
- 13 From the **Apply reaction terms on list**, choose **All physics (symmetric)**.

#### *Outlet 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outlet**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Outlet**, locate the **Boundary Condition** section.
- 4 From the list, choose **Velocity**.
- 5 Locate the **Velocity** section. In the  $U_0$  text field, type  $v_{out}$ .
- 6 Click to expand the **Constraint Settings** section. From the **Apply reaction terms on list**, choose **All physics (symmetric)**.


#### *Outlet 2*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outlet**.
- 2 Select Boundary 18 only.
- 3 In the **Settings** window for **Outlet**, locate the **Constraint Settings** section.
- 4 From the **Apply reaction terms on list**, choose **All physics (symmetric)**.

### **PHASE TRANSPORT (PHTR)**

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

#### *Volume Fraction 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Volume Fraction**.
- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Volume Fraction**, locate the **Volume Fraction** section.
- 4 Select the **Phase s2** check box.
- 5 In the  $s_{0,s2}$  text field, type  $phid_{in}$ .

#### *Outflow 1*

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

- 2 Select Boundaries 2 and 18 only.

## MULTIPHYSICS

### *Mixture Model 1 (mfmm1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Mixture Model 1 (mfmm1)**.
- 2 Select Domain 1 only.
- 3 In the **Settings** window for **Mixture Model**, locate the **Physical Model** section.
- 4 From the **Slip model** list, choose **Hadamard-Rybczynski**.
- 5 From the **Mixture viscosity model** list, choose **Krieger type**.
- 6 Locate the **Continuous Phase Properties** section. From the  $\rho_c$  list, choose **User defined**. In the associated text field, type rho\_c.
- 7 From the  $\mu_c$  list, choose **User defined**. In the associated text field, type mu\_c.
- 8 Locate the **Dispersed Phase 2 Properties** section. From the  $\rho_{s2}$  list, choose **User defined**. In the associated text field, type rho\_d.
- 9 In the  $d_{s2}$  text field, type d\_d.

## MESH 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
- 2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
- 3 From the list, choose **User-controlled mesh**.

### *Size*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Extra fine**.

### *Size 1*

- 1 In the **Model Builder** window, click **Size 1**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Extremely fine**.

### *Corner Refinement 1*

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

### *Size 2*

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

- 2 Drag and drop **Size 2** below **Size 1**.
- 3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 4 From the **Geometric entity level** list, choose **Boundary**.
- 5 Select Boundaries 26–28 only.
- 6 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 7 From the **Predefined** list, choose **Extremely fine**.


#### *Free Triangular 1*

- 1 In the **Model Builder** window, click **Free Triangular 1**.
- 2 In the **Settings** window for **Free Triangular**, click to expand the **Control Entities** section.
- 3 In the **Number of iterations** text field, type 8.
- 4 In the **Maximum element depth to process** text field, type 8.

#### *Boundary Layers 1*

- 1 In the **Model Builder** window, click **Boundary Layers 1**.
- 2 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.
- 3 From the **Handling of sharp corners** list, choose **Splitting**.
- 4 Click to expand the **Transition** section. In the **Number of iterations** text field, type 8.
- 5 In the **Maximum element depth to process** text field, type 16.

#### *Boundary Layer Properties 1*


- 1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
- 2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
- 3 In the **Number of layers** text field, type 12.
- 4 In the **Thickness adjustment factor** text field, type 1.
- 5 Click  **Build All**.


### **STUDY 1**

#### *Step 1: Time Dependent*

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 In the **Output times** text field, type `range(0,0.01,0.1) range(1,10) 100*range(1,10) 1800*range(1,24)`.


*Solution 1 (sol1)*

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Turbulent dissipation rate (comp1.ep)**.
- 4 In the **Settings** window for **Field**, locate the **Scaling** section.
- 5 From the **Method** list, choose **Manual**.
- 6 In the **Scale** text field, type 0.1.
- 7 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Turbulent kinetic energy (comp1.k)**.
- 8 In the **Settings** window for **Field**, locate the **Scaling** section.
- 9 From the **Method** list, choose **Manual**.
- 10 In the **Scale** text field, type 0.1.
- 11 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Pressure (comp1.p)**.
- 12 In the **Settings** window for **Field**, locate the **Scaling** section.
- 13 From the **Method** list, choose **Manual**.
- 14 In the **Scale** text field, type 1e4.
- 15 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Velocity field (comp1.u)**.
- 16 In the **Settings** window for **Field**, locate the **Scaling** section.
- 17 From the **Method** list, choose **Manual**.
- 18 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.
- 19 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 20 Select the **Initial step** check box. In the associated text field, type 0.005.
- 21 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Segregated 1**.
- 22 In the **Settings** window for **Segregated**, locate the **General** section.
- 23 Select the **Limit on nonlinear convergence rate** check box. In the associated text field, type 10.


- 24 In the **Study** toolbar, click  **Compute**.
- 25 Click **Scroll Lock** in the window toolbar.
- 26 Click **Scroll Lock** in the window toolbar.

**RESULTS**


*Line Integration 1*

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Integration> Line Integration**.
- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Line Integration**, locate the **Expressions** section.
- 4 In the table, enter the following settings:



Expression	Unit	Description
-qd_out	kg/s	Dispersed phase mass-influx

- 5 Locate the **Integration Settings** section. Clear the **Compute surface integral** check box.
- 6 Click  **Evaluate**.


*Line Integration 2*

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Integration> Line Integration**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Line Integration**, locate the **Expressions** section.
- 4 In the table, enter the following settings:

Expression	Unit	Description
qd_out	kg/s	Dispersed phase mass-outflux

- 5 Locate the **Integration Settings** section. Clear the **Compute surface integral** check box.
- 6 Click  next to  **Evaluate**, then choose **Table 1 - Line Integration 1**.



*Line Integration 3*

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Integration> Line Integration**.
- 2 Select Boundary 18 only.
- 3 In the **Settings** window for **Line Integration**, locate the **Expressions** section.


4 In the table, enter the following settings:

Expression	Unit	Description
qd_out	kg/s	Dispersed phase mass-outflux

5 Locate the **Integration Settings** section. Clear the **Compute surface integral** check box.

6 Click  next to  **Evaluate**, then choose **Table 1 - Line Integration 1**.

#### *ID Plot Group 7*

1 In the **Results** toolbar, click  **ID Plot Group**.

2 In the **Settings** window for **ID Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type Mass flow (kg/s).

#### *Table Graph 1*

1 Right-click **ID Plot Group 7** and choose **Table Graph**.


2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Dispersed phase mass-influx
Dispersed phase mass-outflux, outlet 1
Dispersed phase mass-outflux, outlet 2

6 In the **ID Plot Group 7** toolbar, click  **Plot**.

#### *Surface*

1 In the **Model Builder** window, expand the **Volume Fraction (phtr)** node, then click **Surface**.

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

3 In the **Expression** text field, type s2.

4 In the **Volume Fraction (phtr)** toolbar, click  **Plot**.

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


6 In the **Minimum** text field, type 0.

7 In the **Maximum** text field, type 0.006.

#### *Streamline 1*

1 In the **Model Builder** window, right-click **Volume Fraction (phtr)** and choose **Streamline**.



- 2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
- 3 From the **Positioning** list, choose **Uniform density**.
- 4 In the **Separating distance** text field, type 0.02.
- 5 In the **Volume Fraction (phtr)** toolbar, click  **Plot**.

#### *Revolution 2D*

- 1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Revolution 2D**.
- 2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.
- 3 In the **Start angle** text field, type 0.
- 4 In the **Revolution angle** text field, type 270.

#### *Surface*

- 1 In the **Model Builder** window, expand the **Volume Fraction (phtr) 1** node, then click **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type  $s2$ .
- 4 Click to expand the **Range** section. Select the **Manual color range** check box.
- 5 In the **Minimum** text field, type 0.
- 6 In the **Maximum** text field, type 0.006.

#### *Surface 2*


- 1 In the **Model Builder** window, right-click **Volume Fraction (phtr) 1** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Turbulent Flow, k-ε>Turbulence variables>spf.Delta\_wPlus - Wall resolution in viscous units - 1**.
- 3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 4 From the **Color** list, choose **Gray**.

#### *Streamline 1*

- 1 Right-click **Volume Fraction (phtr) 1** and choose **Streamline**.
- 2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Phase Transport>mfmml.u\_s2r,...,mfmml.u\_s2z - Convective velocity dispersed phase s2**.
- 3 Locate the **Expression** section. In the **phi-component** text field, type 0.

- 4 Select the **Description** check box. In the associated text field, type Dispersed phase (black).
- 5 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.
- 6 In the **x** text field, type  $\text{range}(0.01, 0.02, 0.19)$ .
- 7 In the **y** text field, type 0.
- 8 In the **z** text field, type  $-1 \cdot 1^{\text{range}(1, 10)}$ .
- 9 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Black**.

#### *Streamline 2*


- 1 Right-click **Volume Fraction (phtr) I** and choose **Streamline**.
- 2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component I (comp I) > Phase Transport > mfmm I.ucontr,...,mfmm I.ucontz - Velocity field, continuous phase**.
- 3 Locate the **Expression** section. In the **phi-component** text field, type 0.
- 4 Select the **Description** check box. In the associated text field, type Continuous phase (white).
- 5 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.
- 6 In the **x** text field, type  $\text{range}(0, 0.02, 0.2) \text{ range}(0.5, 0.5, 12)$ .
- 7 In the **y** text field, type 0.
- 8 In the **z** text field, type  $-1^{\text{range}(1, 35)}$ .
- 9 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **White**.
- 10 In the **Volume Fraction (phtr) I** toolbar, click  **Plot**.

### *Appendix — Geometry Modeling Instructions*

---

#### **GEOMETRY I**

##### *Polygon I (poll)*


- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:


<b>r (m)</b>	<b>z (m)</b>
0	0
12	0
12	-3.3
2	-4
0.5	-7
0.5	-7.4
0	-7.4
0	0

4 Click  **Build Selected**.


#### *Polygon 2 (pol2)*

- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
- 3 From the **Type** list, choose **Open curve**.
- 4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.
- 5 In the **r** text field, type 0 0.4 0.4 0.4.
- 6 In the **z** text field, type -5.4 -5.4 -5.4 -3.4.

#### *Circular Arc 1 (cal)*


- 1 In the **Geometry** toolbar, click  **More Primitives** and choose **Circular Arc**.
- 2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.
- 3 From the **Specify** list, choose **Endpoints and start angle**.
- 4 Locate the **Starting Point** section. In the **r** text field, type 1.6.
- 5 In the **z** text field, type -2.2.
- 6 Locate the **Endpoint** section. In the **r** text field, type 0.4.
- 7 In the **z** text field, type -3.4.
- 8 Locate the **Angles** section. In the **Start angle** text field, type 90.

#### *Line Segment 1 (ls1)*


- 1 In the **Geometry** toolbar, click  **More Primitives** and choose **Line Segment**.
- 2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
- 3 From the **Specify** list, choose **Coordinates**.

- 4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
- 5 Locate the **Starting Point** section. In the **r** text field, type 1.6.
- 6 Locate the **Endpoint** section. In the **r** text field, type 1.6.
- 7 Locate the **Starting Point** section. In the **z** text field, type -2.2.
- 8 Locate the **Endpoint** section. In the **z** text field, type -2.


#### *Circular Arc 2 (ca2)*

- 1 In the **Geometry** toolbar, click  **More Primitives** and choose **Circular Arc**.
- 2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.
- 3 From the **Specify** list, choose **Endpoints and start angle**.
- 4 Locate the **Starting Point** section. In the **r** text field, type 1.6.
- 5 In the **z** text field, type -2.
- 6 Locate the **Endpoint** section. In the **r** text field, type 0.2.
- 7 In the **z** text field, type -3.4.
- 8 Locate the **Angles** section. In the **Start angle** text field, type 90.



#### *Polygon 3 (pol3)*

- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
- 3 From the **Type** list, choose **Open curve**.
- 4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.
- 5 In the **r** text field, type 0.2 0.2 0.2 0 0 0.
- 6 In the **z** text field, type -3.4 -5.2 -5.2 -5.2 -5.2 -5.4.



#### *Convert to Solid 1 (csol1)*

- 1 In the **Geometry** toolbar, click  **Conversions** and choose **Convert to Solid**.
- 2 Select the objects **ca1**, **ca2**, **ls1**, **pol2**, and **pol3** only.



#### *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 0.05.
- 4 Locate the **Position** section. In the **z** text field, type -3.4.
- 5 Click  **Build Selected**.


#### *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 0.4.
- 4 In the **Height** text field, type 0.4.
- 5 Locate the **Position** section. In the **r** text field, type 11.2.
- 6 In the **z** text field, type -0.4.
- 7 Click  **Build Selected**.


#### *Difference 1 (dif1)*

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **poll** only to add it to the **Objects to add** list.
- 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 objects **c1**, **csol1**, and **r1** only.


#### *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 **Height** text field, type 0.5.
- 4 Locate the **Position** section. In the **r** text field, type 11.6.

#### *Fillet 1 (fil1)*



- 1 In the **Geometry** toolbar, click  **Fillet**.
- 2 On the object **r2**, select Points 3 and 4 only.
- 3 In the **Settings** window for **Fillet**, locate the **Radius** section.
- 4 In the **Radius** text field, type 0.5.

#### *Circle 2 (c2)*

- 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 0.1.
- 4 Locate the **Position** section. In the **r** text field, type 12.1.


#### *Difference 2 (dif2)*

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.

- 2 Select the objects **dif1** and **fill** 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 **c2** only.
- 6 Clear the **Keep interior boundaries** check box.
- 7 Click  **Build Selected**.

Add a few lines to obtain a better control of the mesh in the turbulent shear regions.


#### *Polygon 4 (pol4)*

- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
- 3 In the table, enter the following settings:

<b>r (m)</b>	<b>z (m)</b>
0.19	-3.22
0.2	-2.9

- 4 Click  **Build Selected**.


#### *Polygon 5 (pol5)*

- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
- 3 In the table, enter the following settings:

<b>r (m)</b>	<b>z (m)</b>
0.2	-2.9
0.35	-0.25

- 4 Click  **Build Selected**.

#### *Polygon 6 (pol6)*

- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
- 3 In the table, enter the following settings:

<b>r (m)</b>	<b>z (m)</b>
0.2	-0.1
7.6	-0.6

4 Click  **Build Selected**.

*Form Union (fin)*

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click  **Build Selected**.

