

# 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 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 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

$$M_{\rm d} = \int \rho_{\rm d} \mathbf{j}_{\rm d, \, eff} \cdot \mathbf{n} dS$$

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

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

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

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

## GEOMETRY I

To simplify the instructions, import a ready-made geometry sequence. The complete stepby-step instructions for the geometry can be found in the Appendix — Geometry Modeling Instructions.

- I 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 1 (mcel)

I 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  $\leftarrow$  **Zoom Extents** button in the **Graphics** toolbar.

# 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.

Name	Expression	Value		Description
	,	U	U	

**3** In the table, enter the following settings:

rho_c	1000[kg/m^3]	1000 kg/m³	Continuous phase density
mu_c	1e-3[Pa*s]	0.001 Pa·s	Continuous phase viscosity
rho_d	1100[kg/m^3]	1100 kg/m³	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 I (step I)

- I In the Home toolbar, click f(X) 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.



I In the Home toolbar, click a= Variables and choose Local Variables.

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

Name	Expression	Unit	Description
v_in	1.25*step1(t[1/s])[m/s]	m/s	Inlet velocity
v_out	0.05*step1(t[1/s])[m/s]	m/s	Outlet velocity
phid_in	0.003*step1(t[1/s])		Inlet dispersed phase volume fraction
qd_out	2*pi*r*s2*(mfmm1.u_s2r* nr+mfmm1.u_s2z*nz)* mfmm1.rho_s2	kg/(m·s)	Dispersed phase mass-outflux

**3** In the table, enter the following settings:

## 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- $\varepsilon$ , locate the Physical Model section.
- **3** From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.
- **4** Select the **Include gravity** check box.

#### Initial Values 1

- I In the Model Builder window, expand the Turbulent Flow, k-ε (spf) node, then click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *p* text field, type -g\_const\*z\*rho\_c.
- 4 Clear the Compensate for hydrostatic pressure approximation check box.

## Wall 2

- I 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

- I 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  $I_{\rm T}$  list, choose **User defined**.
- **6** From the  $L_{\rm T}$  list, choose **User defined**.
- **7** In the text field, type 0.2\*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.

**IO** In the tree, select the check box for the node **Physics>Advanced Physics Options**.

II Click OK.

- 12 In the Model Builder window, click Inlet I.
- I3 In the Settings window for Inlet, click to expand the Constraint Settings section.

14 From the Apply reaction terms on list, choose All physics (symmetric).

#### Outlet I

- I 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

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

## PHASE TRANSPORT (PHTR)

In the Model Builder window, click Phase Transport (phtr).

#### Volume Fraction 1

- I 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 I

- I In the Physics toolbar, click Boundaries and choose Outflow.
- 2 Select Boundaries 2 and 18 only.

## MULTIPHYSICS

#### Mixture Model I (mfmm1)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Mixture Model I (mfmml).
- 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 ρ<sub>c</sub> 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 I

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

#### Size

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

#### Size I

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

Corner Refinement I

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

#### Size 2

- I In the Model Builder window, right-click Mesh I 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

- I In the Model Builder window, click Free Triangular I.
- 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

- I In the Model Builder window, click Boundary Layers I.
- 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

- I In the Model Builder window, expand the Boundary Layers I node, then click Boundary Layer Properties I.
- 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 I

#### Step 1: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step I: 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 (soll)

- I In the Study toolbar, click The Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Dependent Variables I node, then click Turbulent dissipation rate (compl.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 I>Solver Configurations>Solution I (soll)> Dependent Variables I click Turbulent kinetic energy (compl.k).
- 8 In the Settings window for Field, locate the Scaling section.
- 9 From the Method list, choose Manual.
- **IO** In the **Scale** text field, type **0.1**.
- II In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables I click Pressure (compl.p).
- 12 In the Settings window for Field, locate the Scaling section.
- **I3** From the **Method** list, choose **Manual**.
- **I4** In the **Scale** text field, type 1e4.
- I5 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables I click Velocity field (compl.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 I>Solver Configurations>Solution I (soll) click Time-Dependent Solver I.
- 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 I>Solver Configurations> Solution I (soll)>Time-Dependent Solver I node, then click Segregated I.
- 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**.

## RESULTS

Line Integration 1

- I In the Results toolbar, click <sup>8,85</sup><sub>e-12</sub> 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

- I In the Results toolbar, click <sup>8.85</sup><sub>e-12</sub> 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  $\checkmark$  next to  $\equiv$  Evaluate, then choose Table I Line Integration I.

Line Integration 3

- I In the Results toolbar, click <sup>8,85</sup><sub>e-12</sub> 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 **v** next to **Evaluate**, then choose **Table I** - **Line Integration I**.

## ID Plot Group 7

- I In the Results toolbar, click  $\sim$  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 I

- I 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 Model Builder window, click Table Graph I.
- 7 Locate the **Data** section. From the **Table** list, choose **Table I**.
- 8 In the ID Plot Group 7 toolbar, click 💿 Plot.

#### Surface

- I 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 **I** 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

- I 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**

- I 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

- I In the Model Builder window, expand the Volume Fraction (phtr) I 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

- I In the Model Builder window, right-click Volume Fraction (phtr) I 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 I (compl)>Turbulent Flow, kɛ>Turbulence variables>spf.Delta\_wPlus - Wall resolution in viscous units.
- 3 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 4 From the Color list, choose Gray.

#### Streamline 1

- I Right-click Volume Fraction (phtr) I and choose Streamline.
- In the Settings window for Streamline, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
  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 range (0.01, 0.02, 0.19).
- **7** In the **y** text field, type **0**.
- 8 In the z text field, type -1\*1^range(1,10).
- 9 Locate the Coloring and Style section. Find the Point style subsection. From the Color list, choose Black.

Streamline 2

- I Right-click Volume Fraction (phtr) I and choose Streamline.
- In the Settings window for Streamline, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
  Phase Transport>mfmml.ucontr,...,mfmml.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 range(0,0.02,0.2) range(0.5,0.5,12).
- **7** In the **y** text field, type **0**.
- 8 In the z text field, type -1^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)

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

r (m)	z (m)
0	0
12	0
12	-3.3
2	- 4
0.5	-7
0.5	-7.4
0	-7.4
0	0

**3** In the table, enter the following settings:

4 Click 📄 Build Selected.

Polygon 2 (pol2)

- I 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 (ca1)

- I 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 I (Is I)

- I 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)

- I 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)

- I 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**.
- 6 In the z text field, type -3.4 -5.2 -5.2 -5.2 -5.2 -5.4.

#### Convert to Solid 1 (csol1)

- I In the Geometry toolbar, click 🔣 Conversions and choose Convert to Solid.
- 2 Select the objects cal, ca2, ls1, pol2, and pol3 only.

#### Circle 1 (c1)

- I In the **Geometry** toolbar, click  $\bigcirc$  **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 틤 Build Selected.

#### Rectangle 1 (r1)

- I In the **Geometry** toolbar, click **Rectangle**.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.4**.
- 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 틤 Build Selected.

## Difference I (dif1)

- I 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** Find the **Objects to subtract** subsection. Click to select the **Calculate Selection** toggle button.
- 5 Select the objects cl, csoll, and rl only.

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

## Fillet I (fill)

- I 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)

- I In the **Geometry** toolbar, click  $\cdot$  **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)

- I In the Geometry toolbar, click i Booleans and Partitions and choose Difference.
- 2 Select the objects difl and fill only.
- 3 In the Settings window for Difference, locate the Difference section.
- **4** Find the **Objects to subtract** subsection. Click to select the **Selection** toggle button.
- **5** Select the object **c2** only.
- 6 Clear the Keep interior boundaries check box.
- 7 Click 틤 Build Selected.

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

#### Polygon 4 (pol4)

- I 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:

r (m)	z (m)	
0.19	-3.22	
0.2	-2.9	

4 Click 틤 Build Selected.

Polygon 5 (pol5)

- I 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:

r (m)	z (m)
0.2	-2.9
0.35	-0.25

## 4 Click 🔚 Build Selected.

Polygon 6 (pol6)

- I 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:

r (m)	z (m)
0.2	-0.1
7.6	-0.6

4 Click 틤 Build Selected.

Form Union (fin)

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

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

 $24 \mid$  contaminant-removal from wastewater in a secondary clarifier by sedimentation