Circulating Fluidized Bed

Introduction

This model simulates the operation of a circulating fluidized bed. In this apparatus the dispersed phase, consisting of solid spherical particles, is fluidized by a gas and transported through a vertical riser. Upon reaching the outlet, the dispersed phase is re-injected in the vicinity of the gas inlet at the bottom of the bed. To study the flow in the fluidized bed, the model uses the Euler-Euler model. The phase properties and model setup follow those in Ref. 1.

Model Definition

Figure 1 displays the geometry of the fluidized bed, and schematically describes its means of operation. The fluid phase, consisting of air, is injected at the bottom of the bed. The dispersed phase, particles with a diameter of 54 μm, are fluidized by the gas flow and transported upwards. At the outlet, particles and gas exit the bed. The particles are fed back into the bed through re-injection inlets on the vertical walls near the gas inlet. The dispersed phase flux matches that at the outlet, creating a circulating fluidized bed. Table 1 summarizes the bed geometries and the properties of the phases. The values correspond to those used in Ref. 1.

TABLE 1: FLUIDIZED BED PROPERTIES

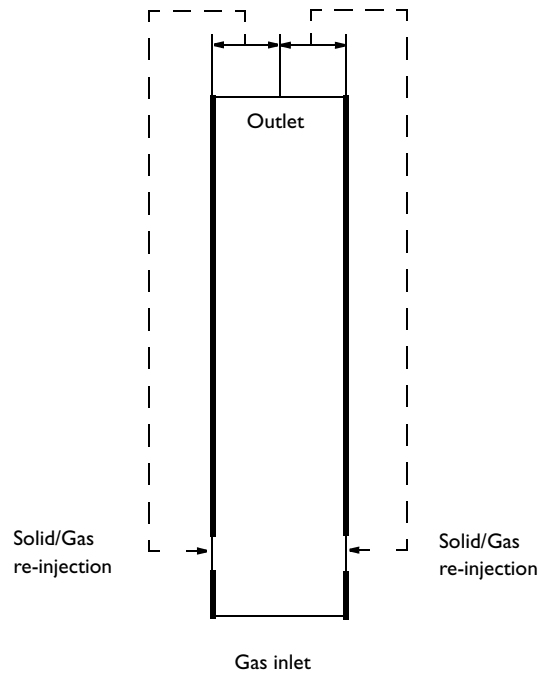| PROPERTY	VALUE
Particle diameter	54 μm
Particle density	930 kg/m^3
Gas density (air)	1.2 kg/m^3
Gas dynamic viscosity (air)	$1.8 \cdot 10^{-5}$ kg/m^3
Gas inlet velocity	1.52 m/s
Bed width	0.09 m
Bed height	10.6 m

Figure 1: Schematic of the circulating fluidized bed. The geometry is not to scale.

GOVERNING EQUATIONS

Use the Euler-Euler model to solve for the flow of the continuous and dispersed phase. Both phases are then modeled as inter penetrating continua governed by a separate set of Navier-Stokes equations. The model also includes a transport equation for the dispersed-phase volume fraction. For details of the equations and assumptions used, see the section Theory for the Euler-Euler Model, Laminar Flow Interface in the *CFD Module User's Guide*. To specify the transport properties of the fluidized bed, apply the settings below.

The viscosity of the dispersed phase is defined as (Ref. 1)

$$\mu_d = 0.5\phi_d \tag{1}$$

where ϕ_d is the dispersed-phase volume fraction.

Assume the momentum transfer to be dominated by the drag force and the drag acting on each phase is given by a drag coefficient β in the manner of

$$\mathbf{F}_{\text{drag,c}} = -\dot{\mathbf{F}}_{\text{drag,d}} = \beta \mathbf{u}_{\text{slip}} \tag{2}$$

Here, the subscripts "d" and "c" indicate properties of dispersed continuous phase, respectively, and the slip velocity is defined as

$$\mathbf{u}_{\text{slip}} = \mathbf{u}_{\text{d}} - \mathbf{u}_{\text{c}} \tag{3}$$

To model the drag coefficient, use the Gidaspow drag model (Ref. 2)

$$\beta = \frac{3\phi_c \phi_d \rho_c C_{\text{drag}}}{4 d_d} |\mathbf{u}_{\text{slip}}| \phi_c^{-2.65}$$

for $\phi_c > 0.8$, and

$$\beta = 150 \frac{\mu_c \phi_d^2}{\phi_c d_d^2} + 1.75 \frac{\phi_d \rho_c}{d_d} |\mathbf{u}_{\text{slip}}| \tag{4}$$

for $\phi_c < 0.8$.

The dispersed phase transport resulting from particle-particle interaction, collisions, friction between particles, and so on, is included by the solids pressure term in the dispersed phase momentum equations. To model this term, use the Gidaspow and Ettehadieh model (Ref. 3):

$$\nabla p_{\text{s}} = -10^{-8.76\phi_c + 5.43} \nabla \phi_c \tag{4-2}$$

INITIAL CONDITIONS

Initially all dispersed particles are positioned in a packed bed section at the bottom of the column. Use a packed bed section 1.855 m high consisting of 50% particles. To avoid discontinuities at the start and end of the packed bed section, use a smoothly varying rectangle function to define the packed bed column.

BOUNDARY CONDITIONS

To fluidize the bed, air is injected at the bottom. Set the dispersed phase volume fraction at the inlet to zero.

Use pressure normal flow conditions for both phases at the outlet. To prevent the dispersed phase from falling back into the bed at the outflow boundary, remove the

gravitational volume force in a region prior to the outlet using a smoothly varying step function.

At the vertical re-injection inlets, gas and solids are injected in a manner that matches the top outflux. This is achieved by using two coupling operators integrating the solid phase mass flux on the left and right sides of the bed center, each feeding the re-injection inlet on the respective side; see Figure 1. Apply a solid phase volume fraction of 0.5 at the inlets, and use injection velocities computed to match the mass outflux at the top. Use a gas velocity that equals that of the particles, assuming that the particle re-injection drags the gas phase along with it. To keep the total injected gas flow rate constant, compensate by reducing the bottom inlet velocity.

Along the solid bed walls, apply a no-slip condition for the continuous phase and a slip condition for the dispersed phase.

Results and Discussion

Figure 3 shows the startup phase of the fluidized bed simulation, plotting the dispersed phase volume fraction during the first five seconds of simulation. Note that the packed bed section is pushed upwards by the gas, primarily in the center of the bed. Due to the lower gas velocity close to the walls, the heavy solid phase is able to accumulate and fall down along the walls creating pockets with high concentration of solids. After about 6 s, the solid phase reaches the top of the bed and starts exiting the bed. You can verify this by inspecting the solids mass flux at the outlet; see Figure 4. This corresponds to the start of the re-injection of solids at the bottom. In the same figure, note that the bed reaches steady-state operations after about 15 s, after which the outflux of particles fluctuates around 105 kg/(m·s). Figure 5 shows snapshots of the dispersed phase volume fraction during steady-state operation.

Figure 6 shows profiles of the averaged gas-phase streamwise velocity, giving an insight in the flow development. The gas phase volume fraction profiles at the corresponding positions are displayed in Figure 7.
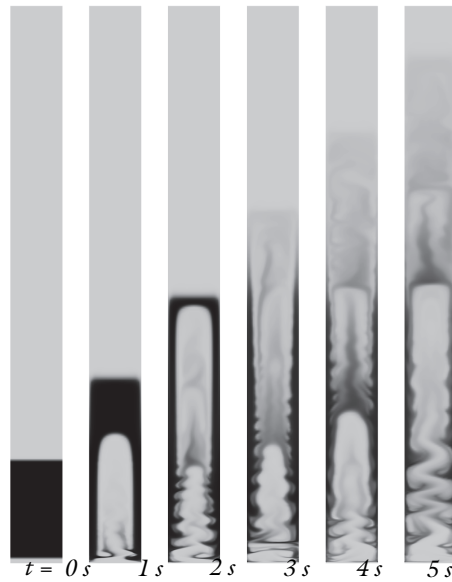
Figure 3: Snapshots of the dispersed phase volume fraction during the startup phase. Black indicates high volume fraction. The bed width has been scaled by a factor of ten.
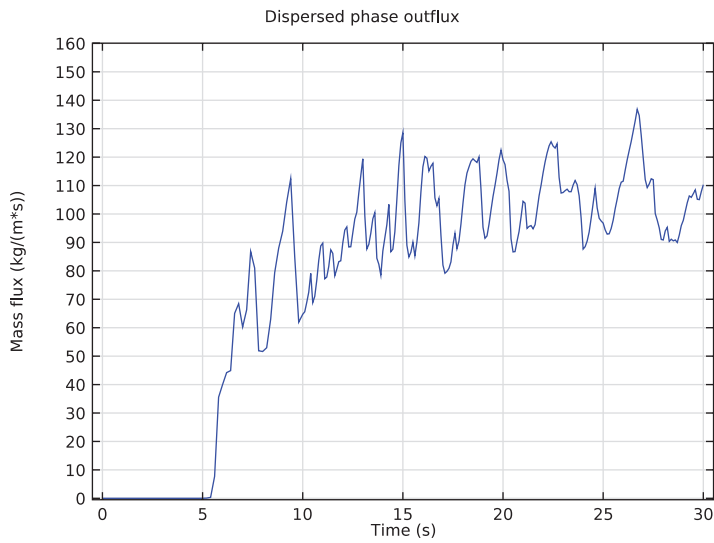

Figure 4: Dispersed phase bed outflux.

Figure 5: Snapshots of the dispersed phase volume fraction during the steady-state operation. Black indicates high volume fraction. The bed width has been scaled by a factor of ten.
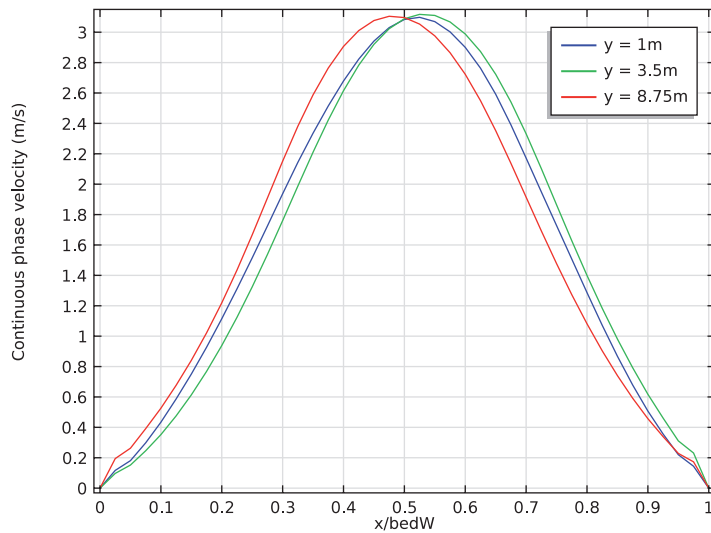
Figure 6: Averaged streamwise continuous phase velocities at three vertical positions.
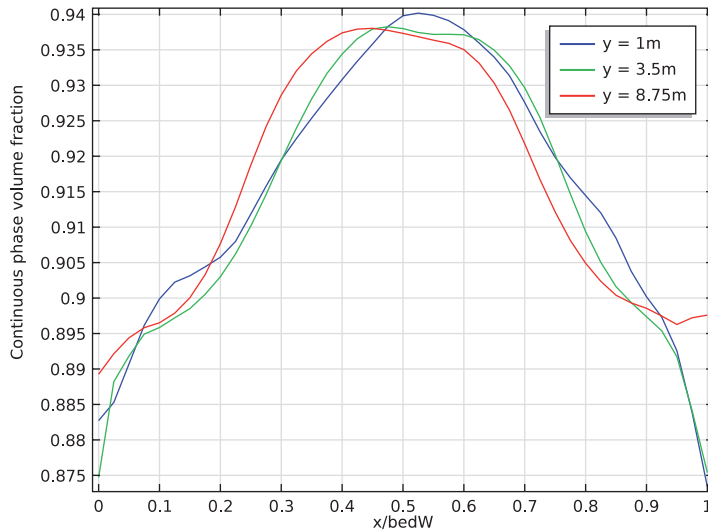

Figure 7: Averaged continuous phase volume fractions at three vertical positions.

Notes About the COMSOL Implementation

The current model includes a slender geometry with an aspect ratio of about 100. When plotting and inspecting the solution over the whole modeling domain, it is thus hard to distinguish any details. To produce the plots in Figure 3 and Figure 5, the geometry was stretched ten times in the *x* direction. This was achieved in COMSOL using a Deformed Geometry interface including a Prescribed Deformation feature. Note that the rescaling does not require you to solve for the dependent variables of the Deformed Geometry interface; it suffices to update the solution. To return the geometry to its physical size, set the `xScale` parameter to 1 and update the solution.

References

1. N. Yang, W. Wang, W. Ge, and J. Li, "CFD simulation of concurrent-up gas–solid flow in circulating fluidized beds with structure-dependent drag coefficient," *Chemical Engineering Journal*, vol. 96, pp. 71–80, 2003.

2. D. Gidaspow, *Multiphase Flow and Fluidization*, Academic Press, San Diego, 1994.

3. H. Enwald, E. Peirano, and A.-E. Almstedt, "Eulerian Two-Phase Flow Theory Applied to Fluidization," *Int. J. Multiphase Flow,* vol. 22, pp. 21–66, 1996.

Model Library path: CFD_Module/Multiphase_Tutorials/fluidized_bed

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select physics** tree, select **Fluid Flow>Multiphase Flow>Euler-Euler Model>Euler-Euler Model, Laminar Flow (ee)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select study** tree, select **Preset Studies>Time Dependent**.

6 Click **Done**.

DEFINITIONS

Parameters

1 On the **Model** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file fluidized_bed_parameters.txt.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size** section.

3 In the **Width** text field, type bedW.

4 In the **Height** text field, type bedH.

5 Locate the **Position** section. In the **y** text field, type inlH.

6 Click the **Build Selected** button.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size** section.

3 In the **Width** text field, type bedW.

4 In the **Height** text field, type solidH0.

5 Locate the **Position** section. In the **y** text field, type inlH.

6 Click the **Build Selected** button.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size** section.

3 In the **Width** text field, type bedW.

4 In the **Height** text field, type inlH.

5 Click the **Build Selected** button.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **1**, set **y** to 0.02.

6 In row **2**, set **y** to 0.02+injH.

7 Click the **Build Selected** button.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **1**, set **x** to bedW.

6 In row **1**, set **y** to 0.02.

7 In row **2**, set **x** to bedW.

8 In row **2**, set **y** to 0.02+injH.

9 Click the **Build Selected** button.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **x** text field, type bedW/2.

4 In the **y** text field, type bedH+inlH.

5 Click the **Build Selected** button.

DEFINITIONS

<mark>Create a step function to use for ramping up the bottom inlet velocity from zero to its full value over the initial $2 \cdot 10^{-3}$ s.</mark>

Step 1 (step1)

1 On the **Model** toolbar, click **Functions** and choose **Local>Step**.

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

3 In the **Location** text field, type 1e-3.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 2e-3.

5 Click the **Plot** button to view the step function.

Next, create another step function to use for ramping up the volume fraction at the re-injection slots.

Step 2 (step2)

1 On the **Model** toolbar, click **Functions** and choose **Local>Step**.

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

3 In the **Location** text field, type Uin/100.

4 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `2*Uin/100`.

With these settings, the step function reaches its maximum value when there-injection velocity is two percent of the gas inlet velocity at the bottom.

Create a third step function to use for removing the gravitational volume force in a short region prior to the outlet. This is to ensure that the heavy solids are transported out of the bed at the outlet.

Step 3 (step3)

1 On the **Model** toolbar, click **Functions** and choose **Local>Step**.

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

3 In the **Location** text field, type `bedH`.

4 In the **From** text field, type `1`.

5 In the **To** text field, type `0`.

6 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `0.15`.

Rectangle 1 (rect1)

1 On the **Model** toolbar, click **Functions** and choose **Local>Rectangle**.

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

3 In the **Lower limit** text field, type `inlH`.

4 In the **Upper limit** text field, type `solidH0+inlH`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `bedW`.

Plot to view the initial conditions for the solid volume fraction.

6 Click the **Plot** button.

Create two integration operators at the outlet. You will use these to compute the dispersed phase outlet flux in the left and right bed halves as well as the corresponding re-injection velocities.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, type intopL in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 9 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, type intopR in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 10 only.

Variables 1

1 On the **Definitions** toolbar, click **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
outfluxdL	intopL(phid*max(udy ,0[m/s]))	m²/s	Dispersed phase outflux, left side
outfluxdR	intopR(phid*max(udy ,0[m/s]))	m²/s	Dispersed phase outflux, right side
UinL	nojac(outfluxdL/ phidInj/injH)	m/s	Injector velocity, left side
UinR	nojac(outfluxdR/ phidInj/injH)	m/s	Injector velocity, right side
UinB	Uin-(UinL+UinR)*(1- phidInj)*injH/bedW	m/s	Bottom inlet velocity

EULER-EULER MODEL, LAMINAR FLOW (EE)

Phase Properties 1

1 In the **Model Builder** window, expand the **Euler-Euler Model, Laminar Flow (ee)** node, then click **Phase Properties 1**.

2 In the **Settings** window for Phase Properties, locate the **Continuous Phase Properties** section.

3 From the ρ_c list, choose **User defined**. In the associated text field, type rhoc.

4 From the μ_c list, choose **User defined**. In the associated text field, type muc.

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

6 In the d_d text field, type diam.

7 Locate the **Viscosity Model** section. From the **Viscosity model** list, choose **User defined**. Locate the **Drag Model** section. From the **Drag model** list, choose **Gidaspow**.

8 Locate the **Solid Pressure Model** section. From the **Solid pressure model** list, choose **Gidaspow-Ettehadieh**.

Wall 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Euler-Euler Model, Laminar Flow (ee)** click **Wall 1**.

2 In the **Settings** window for Wall, locate the **Dispersed Phase Boundary Condition** section.

3 From the **Dispersed velocity boundary condition** list, choose **Slip**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Euler-Euler Model, Laminar Flow (ee)** click **Initial Values 1**.

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

3 In the *phid* text field, type phid0*rect1(y).

Inlet 1

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

Zoom in to the lower part of the channel to be able to select the inlet boundary.

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

3 Select Boundary 2 only.

4 In the **Settings** window for Inlet, locate the **Continuous Phase** section.

5 Specify the $\mathbf{u}_{c0}$ vector as

0	x
UinB*step1(t)	y

6 Specify the $\mathbf{u}_{d0}$ vector as

0	x
UinB*step1(t)	y

Outlet 1

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

2 Select Boundaries 9 and 10 only.

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

4 From the **Mixture boundary condition** list, choose **Pressure normal flow**.

Gravity 1

1 On the **Physics** toolbar, click **Domains** and choose **Gravity**.

2 Select Domains 1–3 only.

3 In the **Settings** window for Gravity, locate the **Gravity** section.

4 Specify the **g** vector as

0	x
-g_const*step3(y)	y

Inlet 2

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

2 Select Boundary 3 only.

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

4 Specify the $\mathbf{u}_{c0}$ vector as

UinL*step1(t)	x
0	y

5 Specify the $\mathbf{u}_{d0}$ vector as

UinL*step1(t)	x
0	y

6 In the ϕ_{d0} text field, type `phidInj*step2(t)`.

Inlet 3

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

2 Select Boundary 12 only.

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

4 Specify the $\mathbf{u}_{c0}$ vector as

-UinR*step1(t)	x
0	y

5 Specify the $\mathbf{u}_{d0}$ vector as

-UinR*step1(t)	x
0	y

6 In the ϕ_{d0} text field, type `phidInj*step2(t)`.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for Mapped, locate the **Domain Selection** section.

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

4 Select Domains 2 and 3 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Boundary 6 only.

3 In the **Settings** window for Distribution, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Explicit distribution**.

5 In the **Explicit element distribution** text field, type `range(0,0.00225,0.09)`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 5 and 14 only.

3 In the **Settings** window for Distribution, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 120.

6 In the **Element ratio** text field, type 10.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 7 and 15 only.

3 In the **Settings** window for Distribution, locate the **Distribution** section.

4 In the **Number of elements** text field, type 250.

5 Click the **Build Selected** button.

Free Triangular 1

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

2 In the **Settings** window for Free Triangular, locate the **Domain Selection** section.

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

4 Select Domain 1 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type 0.003.

Size 2

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type 0.001.

7 Select Boundaries 3 and 12 only.

Size 3

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1>Size 2** and choose **Duplicate**.

2 Select Boundary 2 only.

3 In the **Settings** window for Size, locate the **Element Size Parameters** section.

4 In the **Maximum element size** text field, type 0.002.

Size 4

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1>Size 3** and choose **Duplicate**.

2 Select Boundary 6 only.

3 In the **Settings** window for Size, locate the **Element Size Parameters** section.

4 In the **Maximum element size** text field, type 0.005.

5 Click the **Build All** button.

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 **Settings** window for Time Dependent, locate the **Study Settings** section.

3 In the **Times** text field, type range(0,0.2,10) range(10.1,0.1,30).

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

5 In the associated text field, type 0.005.

Solution 1

1 On the **Study** toolbar, click **Show Default Solver**.

Due to the highly nonlinear problem and the long simulation time, take the following measures to ensure the accuracy of the solution: decrease the relative tolerance, apply manual scales for the dependent variables and prescribe a maximum time step.

2 In the **Model Builder** window, expand the **Solution 1** node.

3 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1** click **Dependent Variables 1**.

4 In the **Settings** window for Dependent Variables, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1>Dependent Variables 1** click **Pressure (comp1.p)**.

7 In the **Settings** window for Field, locate the **Scaling** section.

8 From the **Method** list, choose **Manual**.

9 In the **Scale** text field, type 5e3.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1>Dependent Variables 1** click **Volume fraction, dispersed phase (comp1.phid)**.

11 In the **Settings** window for Field, locate the **Scaling** section.

12 From the **Method** list, choose **Manual**.

13 In the **Scale** text field, type phid0.

14 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1>Dependent Variables 1** click **Velocity field, continuous phase (comp1.uc)**.

15 In the **Settings** window for Field, locate the **Scaling** section.

16 From the **Method** list, choose **Manual**.

17 In the **Scale** text field, type Uin.

18 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1>Dependent Variables 1** click **Velocity field, dispersed phase (comp1.ud)**.

19 In the **Settings** window for Field, locate the **Scaling** section.

20 From the **Method** list, choose **Manual**.

21 In the **Scale** text field, type Uin.

22 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1** click **Time-Dependent Solver 1**.

23 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

24 Locate the **Time Stepping** section. Select the **Initial step** check box.

25 In the associated text field, type 1e-5.

26 Select the **Maximum step** check box.

27 In the associated text field, type 5e-4.

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, click to expand the **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

4 On the **Study** toolbar, click **Compute**.

RESULTS

Disable geometry edges in the plots.

Continuous Phase (ee)

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

2 Clear the **Plot data set edges** check box.

Dispersed Phase (ee)

1 In the **Model Builder** window, under **Results** click **Dispersed Phase (ee)**.

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

3 Clear the **Plot data set edges** check box.

Before working with plots, disable the default plot when selected (update plot automatically). Due to the size of the data sets, it is more convenient to generate plots actively.

1 In the **Model Builder** window, click **Results**.

2 In the **Settings** window for Results, locate the **Result Settings** section.

3 Clear the **Automatic update of plots** check box.

Now add a Deformed Geometry interface that you can use to rescale the geometry in the x direction.

ADD PHYSICS

1 On the **Model** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the **Add physics** tree, select **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.

4 Click **Add to Component** in the window toolbar.

Add a parameter for the rescaling.

DEFORMED GEOMETRY (DG)

DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
xScale	10	10.00	x-axis scale factor

DEFORMED GEOMETRY (DG)

Prescribed Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Deformation**.

2 Select Domains 1–3 only.

3 In the **Settings** window for Prescribed Deformation, locate the **Prescribed Mesh Displacement** section.

4 In the d_x text-field array, type xScale*Xg on the first row.

Update the solution for the re-scaling to take effect.

STUDY 1

On the **Study** toolbar, click **Update Solution**.

RESULTS

Follow these instructions to reproduce the series of dispersed-phase volume fraction plots in Figure 3. Note the use of Deformation features to include a number of plots side-by-side in the same figure.

Continuous Phase (ee)

2D Plot Group 3

1 On the **Model** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 Clear the **Plot data set edges** check box.

4 Right-click **Results>2D Plot Group 3** and choose **Surface**.

5 In the **Settings** window for Surface, locate the **Data** section.

6 From the **Data set** list, choose **Study 1/Solution 1**.

7 From the **Time (s)** list, choose **0.000**.

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

9 In the **Minimum** text field, type -0.1.

10 In the **Maximum** text field, type 0.5.

11 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

12 Select the **Reverse color table** check box.

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

14 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

15 Right-click **Results>2D Plot Group 3>Surface 1** and choose **Duplicate**.

16 In the **Settings** window for Surface, locate the **Data** section.

17 From the **Time (s)** list, choose **1.000**.

18 Right-click **Results>2D Plot Group 3>Surface 2** and choose **Deformation**.

19 In the **Settings** window for Deformation, locate the **Expression** section.

20 In the **x component** text field, type 1.5.

21 In the **y component** text field, type 0.

22 Locate the **Scale** section. Select the **Scale factor** check box.

23 In the associated text field, type 1.

24 Right-click **Surface 2** and choose **Duplicate**.

25 In the **Settings** window for Surface, locate the **Data** section.

26 From the **Time (s)** list, choose **2.000**.

27 In the **Model Builder** window, expand the **Results>2D Plot Group 3>Surface 3** node, then click **Deformation 1**.

28 In the **Settings** window for Deformation, locate the **Expression** section.

29 In the **x component** text field, type 3.

30 In the **Model Builder** window, under **Results>2D Plot Group 3** right-click **Surface 3** and choose **Duplicate**.

31 In the **Settings** window for Surface, locate the **Data** section.

32 From the **Time (s)** list, choose **3.000**.

33 In the **Model Builder** window, expand the **Results>2D Plot Group 3>Surface 4** node, then click **Deformation 1**.

34 In the **Settings** window for Deformation, locate the **Expression** section.

35 In the **x component** text field, type 4.5.

36 In the **Model Builder** window, under **Results>2D Plot Group 3** right-click **Surface 4** and choose **Duplicate**.

37 In the **Settings** window for Surface, locate the **Data** section.

38 From the **Time (s)** list, choose **4.000**.

39 In the **Model Builder** window, expand the **Results>2D Plot Group 3>Surface 5** node, then click **Deformation 1**.

40 In the **Settings** window for Deformation, locate the **Expression** section.

41 In the **x component** text field, type 6.

42 In the **Model Builder** window, under **Results>2D Plot Group 3** right-click **Surface 5** and choose **Duplicate**.

43 In the **Settings** window for Surface, locate the **Data** section.

44 From the **Time (s)** list, choose **5.000**.

45 In the **Model Builder** window, expand the **Results>2D Plot Group 3>Surface 6** node, then click **Deformation 1**.

46 In the **Settings** window for Deformation, locate the **Expression** section.

47 In the **x component** text field, type 7.5.

48 On the **2D plot group** toolbar, click **Plot**.

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

Use the plot group you just created as the starting point for reproducing Figure 5.

2D Plot Group 4

1 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Duplicate**.

2 In the **Model Builder** window, expand the **2D Plot Group 4** node, then click **Surface 1**.

3 In the **Settings** window for Surface, locate the **Data** section.

4 From the **Time (s)** list, choose **20.00**.

5 In the **Model Builder** window, under **Results>2D Plot Group 4** click **Surface 2**.

6 In the **Settings** window for Surface, locate the **Data** section.

7 From the **Time (s)** list, choose **22.00**.

8 In the **Model Builder** window, under **Results>2D Plot Group 4** click **Surface 3**.

9 In the **Settings** window for Surface, locate the **Data** section.

10 From the **Time (s)** list, choose **24.00**.

11 In the **Model Builder** window, under **Results>2D Plot Group 4** click **Surface 4**.

12 In the **Settings** window for Surface, locate the **Data** section.

13 From the **Time (s)** list, choose **26.00**.

14 In the **Model Builder** window, under **Results>2D Plot Group 4** click **Surface 5**.

15 In the **Settings** window for Surface, locate the **Data** section.

16 From the **Time (s)** list, choose **28.00**.

17 In the **Model Builder** window, under **Results>2D Plot Group 4** click **Surface 6**.

18 In the **Settings** window for Surface, locate the **Data** section.

19 From the **Time (s)** list, choose **30.00**.

20 On the **2D plot group** toolbar, click **Plot**.

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

The following steps reproduce the plot shown in Figure 4.

1D Plot Group 5

1 On the **Model** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type Dispersed phase outflux.

4 Locate the **Plot Settings** section. Select the **y-axis label** check box.

5 In the associated text field, type Mass flux (kg/(m*s)).

6 On the **1D plot group** toolbar, click **Global**.

7 In the **Settings** window for Global, locate the **y-Axis Data** section.

8 In the table, enter the following settings:

Expression	Unit	Description
rhod*(outfluxdL+outfluxdR)	kg/(m*s)	

9 Click to expand the **Legends** section. Clear the **Show legends** check box.

10 On the **ID plot group** toolbar, click **Plot**.

To reproduce the plots in Figure 6 and Figure 7, begin by creating three cut-line data sets.

Data Sets

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for Cut Line 2D, locate the **Line Data** section.

3 In row **Point 1**, set **y** to 1.

4 In row **Point 2**, set **y** to 1.

5 In row **Point 2**, set **x** to bedW.

6 Clear the **Bounded by points** check box.

7 Right-click **Results>Data Sets>Cut Line 2D 1** and choose **Duplicate**.

8 In the **Settings** window for Cut Line 2D, locate the **Line Data** section.

9 In row **Point 1**, set **y** to 3.5.

10 In row **Point 2**, set **y** to 3.5.

11 Right-click **Results>Data Sets>Cut Line 2D 2** and choose **Duplicate**.

12 In the **Settings** window for Cut Line 2D, locate the **Line Data** section.

13 In row **Point 1**, set **y** to 8.75.

14 In row **Point 2**, set **y** to 8.75.

Now plot the averaged streamwise continuous phase velocities at the three y positions defined by the cut-line data sets you just defined. Compare the resulting plot with that in Figure 6.

1D Plot Group 6

1 On the **Results** toolbar, click **1D Plot Group**.

2 On the **1D plot group** toolbar, click **Line Graph**.

3 In the **Settings** window for Line Graph, locate the **Data** section.

4 From the **Data set** list, choose **Cut Line 2D 1**.

5 From the **Time selection** list, choose **First**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type
`1-timeavg(10,30,phid)`.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `Xg/bedW`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

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

11 In the table, enter the following settings:

Legends
y = 1m

12 Right-click **Results>1D Plot Group 6>Line Graph 1** and choose **Duplicate**.

13 In the **Settings** window for Line Graph, locate the **Data** section.

14 From the **Data set** list, choose **Cut Line 2D 2**.

15 Locate the **Legends** section. In the table, enter the following settings:

Legends
y = 3.5m

16 Right-click **Results>1D Plot Group 6>Line Graph 2** and choose **Duplicate**.

17 In the **Settings** window for Line Graph, locate the **Data** section.

18 From the **Data set** list, choose **Cut Line 2D 3**.

19 Locate the **Legends** section. In the table, enter the following settings:

Legends
y = 8.75m

20 In the **Model Builder** window, click **1D Plot Group 6**.

21 In the **Settings** window for 1D Plot Group, locate the **Title** section.

22 From the **Title type** list, choose **None**.

23 Locate the **Plot Settings** section. Select the **x-axis label** check box.

24 In the associated text field, type `x/bedW`.

25 Select the **y-axis label** check box.

26 In the associated text field, type `Continuous phase volume fraction`.

27 On the **1D plot group** toolbar, click **Plot**.

Finally, reproduce Figure 7 by duplicating and adapting the plot group you just created.

1D Plot Group 7

1 Right-click **1D Plot Group 6** and choose **Duplicate**.

2 In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.

3 In the **y-axis label** text field, type Continuous phase velocity (m/s).

4 In the **Model Builder** window, expand the **1D Plot Group 7** node, then click **Line Graph 1**.

5 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

6 In the **Expression** text field, type timeavg(10,30,ucy).

7 In the **Model Builder** window, under **Results>1D Plot Group 7** click **Line Graph 2**.

8 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

9 In the **Expression** text field, type timeavg(10,30,ucy).

10 In the **Model Builder** window, under **Results>1D Plot Group 7** click **Line Graph 3**.

11 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

12 In the **Expression** text field, type timeavg(10,30,ucy).

13 On the **1D plot group** toolbar, click **Plot**.