



Model created in COMSOL Multiphysics 6.4

Circulating Fluidized Bed

Introduction

This model simulates the operation of a circulating fluidized bed (CFB). 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 reinjected 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 $54\ \mu\text{m}$, are fluidized by the gas flow and transported upward. At the outlet, particles and gas exit the bed. The particles are fed back into the bed through reinjection 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	$54\ \mu\text{m}$
Particle density	$930\ \text{kg/m}^3$
Gas density (air)	$1.2\ \text{kg/m}^3$
Gas dynamic viscosity (air)	$1.8 \cdot 10^{-5}\ \text{kg/m}^3$
Gas inlet velocity	$1.52\ \text{m/s}$
Bed width	$0.09\ \text{m}$
Bed height	$10.6\ \text{m}$

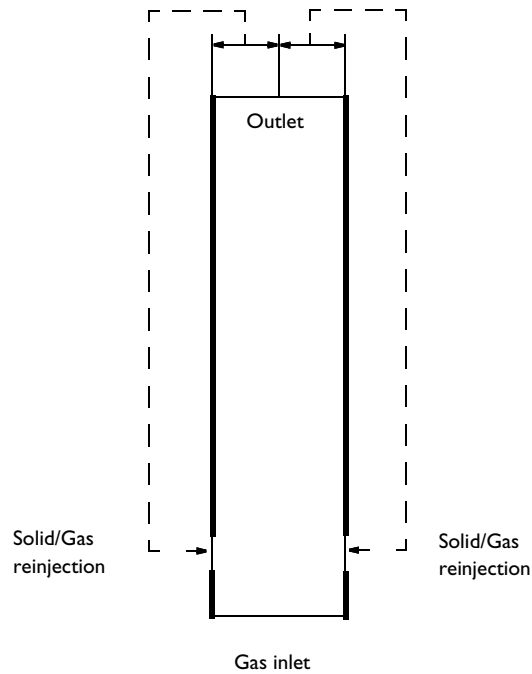


Figure 1: Schematic of the circulating fluidized bed. The geometry is not to scale.

GOVERNING EQUATIONS

Use the Euler–Euler model to solve for the flow of the continuous and dispersed phase. Both phases are then modeled as interpenetrating 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 theory section 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 from the Gidaspow model as

$$\mu_d = 0.5\phi_d$$

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

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

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

$$\mathbf{u}_{\text{slip}} = \mathbf{u}_d - \mathbf{u}_c$$

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

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

for $\phi_c > 0.8$ and

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

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, the following expression is used (Ref. 1):

$$\nabla p_s = -10^{-8.686\phi_c + 6.385} \nabla \phi_c$$

INITIAL CONDITIONS

Initially all dispersed particles are positioned in a packed bed section at the bottom of the column. The packed bed section is 1.855 m high and consists 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 reinjection 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 reinjection inlet on the respective side; see [Figure 1](#). Apply solid mass flux which matches the mass outflux at the top. Use a gas velocity that equals that of the particles, assuming that the particle reinjection drags the gas phase along with it.

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

This section shows the results from the simulation. Note that the model is not deterministic and the plots may differ between computations performed on different platforms.

[Figure 2](#) 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 upward 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 3](#). 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 11.5 kg/(m·s), which approximates to the experimental value reported in [Ref. 1](#). [Figure 4](#) shows snapshots of the dispersed-phase volume fraction during steady-state operation.

[Figure 5](#) 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 6](#).

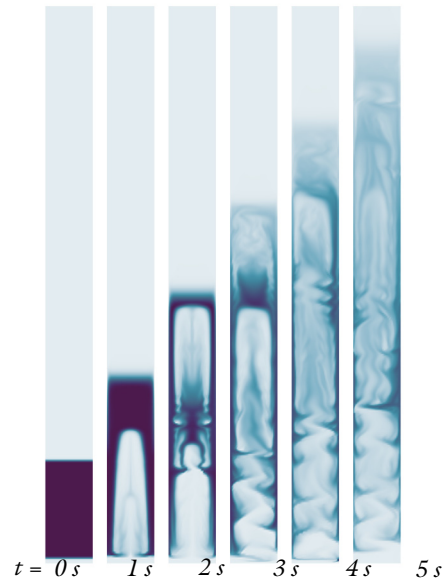


Figure 2: Snapshots of the dispersed-phase volume fraction during the startup phase. Black indicates high volume fraction. The bed width has been scaled by a factor of ten.

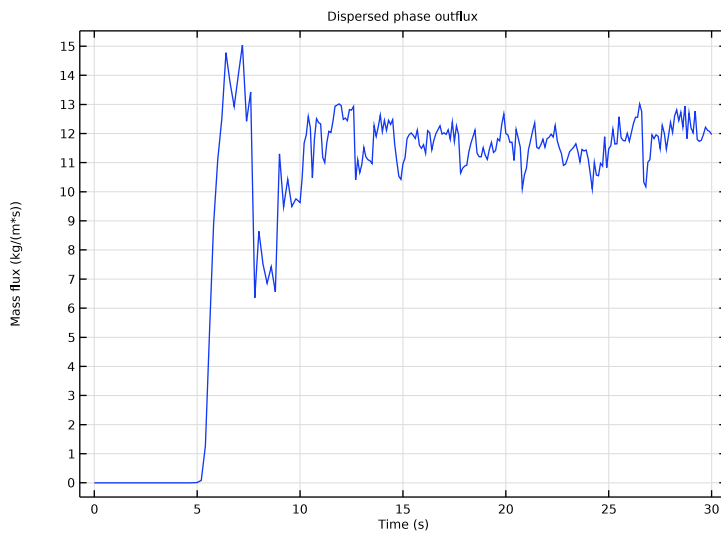


Figure 3: Dispersed-phase bed outflux.

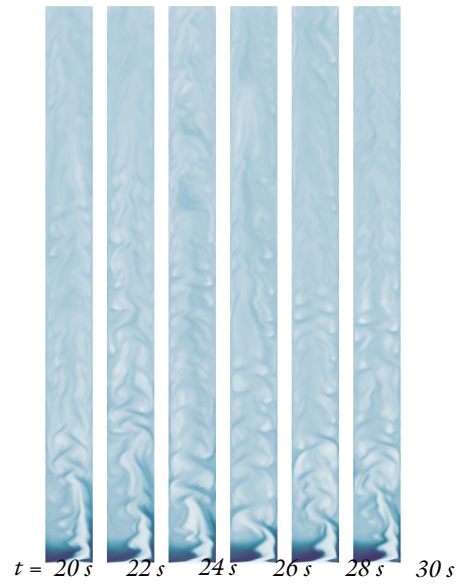


Figure 4: Snapshots of the dispersed-phase volume fraction during the steady-state operation. Black indicates high volume fraction. The bed width has been scaled by a factor of ten.

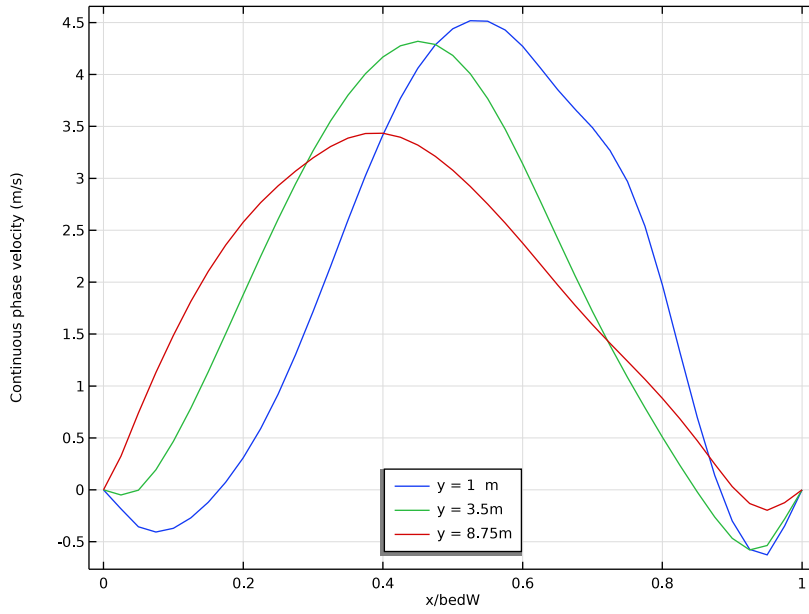


Figure 5: Averaged streamwise continuous-phase velocities at three vertical positions.

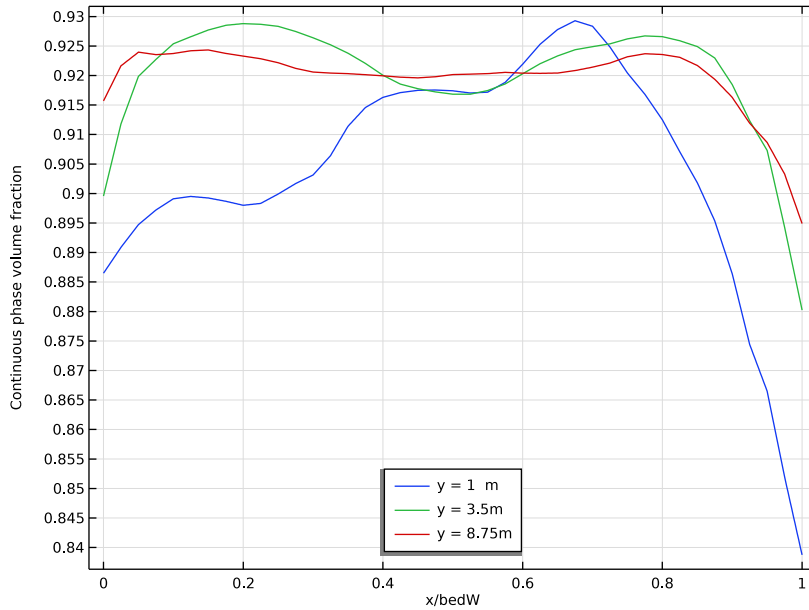


Figure 6: Averaged continuous-phase volume fractions at three vertical positions.

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 therefore hard to distinguish any details. To produce the plots in Figure 2 and Figure 4, the view was stretched ten times in the x direction.

The flow in the fluidized bed is turbulent with Reynolds number around 9000. Nevertheless, a laminar flow model is used here. The effects of additional stresses from meso-scale structures are included in the models for solid pressure and inter-phase drag. See Ref. 4 for further details.

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,” *J. Chem. Eng.*, 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.
4. K. Agrawal and others, “The Role of Meso-scale Structures in Rapid Gas-solid Flows,” *J. Fluid Mech.*, vol. 445, pp. 151–185, 2001.

Application Library path: CFD_Module/Multiphase_Flow/fluidized_bed




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**.
- 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 **General Studies** > **Time Dependent**.
- 6 Click  **Done**.


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 Click  **Load from File**.
- 4 Browse to the model’s Application Libraries folder and double-click the file `fluidized_bed_parameters.txt`.

GEOMETRY I


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 bedW.
- 4 In the **Height** text field, type bedH.
- 5 Locate the **Position** section. In the **y** text field, type in1H.


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 **Width** text field, type bedW.
- 4 In the **Height** text field, type solidH0.
- 5 Locate the **Position** section. In the **y** text field, type in1H.

Rectangle 3 (r3)

- 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 bedW.
- 4 In the **Height** text field, type in1H.



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 **y** text field, type 0.02.
- 6 Locate the **Endpoint** section. In the **y** text field, type 0.02+injH.

Line Segment 2 (ls2)

- 1 Right-click **Line Segment 1 (ls1)** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
- 3 In the **x** text field, type bedW.
- 4 Locate the **Endpoint** section. In the **x** text field, type bedW.

Point 1 (pt1)

- 1 In the **Geometry** toolbar, click  **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 $bedH+in1H$.
- 5 In the **Geometry** toolbar, click  **Build All**.


Edit the view scale manually to rescale the geometry in the x direction and reduce the aspect ratio of plots.


DEFINITIONS

View 1

In the **Model Builder** window, expand the **Component 1 (comp1) > Definitions** node.


Axis

- 1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.
- 2 In the **Settings** window for **Axis**, locate the **Axis** section.
- 3 From the **View scale** list, choose **Manual**.
- 4 In the **x scale** text field, type 10.
- 5 Click  **Update**.




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



View 2

- 1 In the **Definitions** toolbar, click  **View**.
- 2 In the **Settings** window for **View**, locate the **View** section.
- 3 Select the **Lock axis** checkbox.



Axis

- 1 In the **Model Builder** window, expand the **View 2** node, then click **Axis**.
- 2 In the **Settings** window for **Axis**, locate the **Axis** section.
- 3 In the **x minimum** text field, type -0.01.
- 4 In the **x maximum** text field, type 0.105.
- 5 In the **y maximum** text field, type 0.4.
- 6 In the **y minimum** text field, type -0.025.
- 7 From the **View scale** list, choose **Manual**.
- 8 In the **x scale** text field, type 10.
- 9 Click  **Update**.
- 10 In the **Graphics** window toolbar, click  next to  **Go to Default View**, then choose **Go to View 1**.

Create a step function to use for ramping up the bottom inlet velocity from zero to its full value over the initial 0.002 s.


GLOBAL DEFINITIONS

Step 1 (step1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Step**.
- 2 In the **Settings** window for **Step**, locate the **Parameters** section.
- 3 In the **Location** text field, type $1e-3[s]$.
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type $2e-3$.
- 5 Click  **Plot** to view the step function.



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

Step 2 (step2)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > 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`.

Create a rectangle function to specify the initial conditions for the solid volume fraction.


Rectangle 1 (rect1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Parameters** section.
- 3 In the **Lower limit** text field, type `in1H`.
- 4 In the **Upper limit** text field, type `solidH0+in1H`.
- 5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `bedW`.
- 6 Click  **Plot**.


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 reinjection velocities.

DEFINITIONS

Integration 1 (intop1)

- 1 In the **Definitions** toolbar, click  **Nonlocal 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 (top-left boundary)

Integration 2 (intop2)

- 1 In the **Definitions** toolbar, click  **Nonlocal 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 (top-right boundary)


Variables 1

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
outfluxd L	$\text{intopL}(\text{phid} * \max(\text{udy}, 0[\text{m/s}]))$	m ² /s	Dispersed phase outflux, left side
outfluxd R	$\text{intopR}(\text{phid} * \max(\text{udy}, 0[\text{m/s}]))$	m ² /s	Dispersed phase outflux, right side

Add a global probe for the dispersed phase outflux to get a plot automatically during the solver process.

Global Variable Probe: Dispersed phase outflux

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Global Variable Probe**.
- 2 In the **Settings** window for **Global Variable Probe**, type Global Variable Probe: Dispersed phase outflux in the **Label** text field.
- 3 Locate the **Expression** section. In the **Expression** text field, type $\text{rhod} * (\text{outfluxdL} + \text{outfluxdR})$.

EULER–EULER MODEL, LAMINAR FLOW (EE)



Phase Properties I

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Euler–Euler Model, Laminar Flow (ee)** click **Phase Properties I**.
- 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 μ_c^m list, choose **Pure phase value**.
- 8 From the μ_d^m list, choose **Gidaspow**.
- 9 Locate the **Drag Model** section. From the **Drag model** list, choose **Gidaspow**.
- 10 Locate the **Solid Pressure Model** section. From the **Solid pressure model** list, choose **User-defined modulus of elasticity**.
- 11 In the G text field, type $10^{(6.385-8.686*ee.phicReg)}$.

Initial Values I

- 1 In the **Model Builder** window, click **Initial Values I**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the *phid* text field, type $phid0*rect1(y)$.

Wall I

- 1 In the **Model Builder** window, click **Wall I**.
- 2 In the **Settings** window for **Wall**, locate the **Dispersed Phase Boundary Condition** section.
- 3 From the **Dispersed velocity boundary condition** list, choose **Slip**.
- 4 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 5 In the **Show More Options** dialog, click  **Select All**.
- 6 Click **OK**.
- 7 In the **Model Builder** window, click **Euler–Euler Model, Laminar Flow (ee)**.

8 In the **Settings** window for **Euler–Euler Model, Laminar Flow**, click to expand the **Consistent Stabilization** section.

Use **Limit small time steps effect on stabilization time scale** instead of **Use dynamic subgrid time scale** to avoid loss of pressure stabilization at small time steps.

9 Clear the **Use dynamic subgrid time scale** checkbox.

10 Select the **Limit small time steps effect on stabilization time scale** checkbox.

Inlet 1

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

2 Select Boundary 2 only (bottom)

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

4 Specify the $\mathbf{u}_{c,0}$ vector as

0	x
$U_{in} \cdot \text{step1}(t)$	y

5 Locate the **Dispersed Phase Boundary Condition** section. Specify the $\mathbf{u}_{d,0}$ vector as

0	x
$U_{in} \cdot \text{step1}(t)$	y

6 From the **Dispersed phase boundary condition** list, choose **No flux**.

Outlet 1

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




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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Acceleration of Gravity** section. Specify the \mathbf{g} vector as

0	x
$-g_const \cdot \text{step2}(y) \cdot \text{step1}(t)$	y

Inlet 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Inlet**, in the **Graphics** window toolbar, click  next to  **Go to Default View**, then choose **Go to View 2**.



- 4 Locate the **Two-Phase Inlet Type** section. From the **Two-phase inlet type** list, choose **Dispersed phase**.
- 5 Locate the **Continuous Phase Boundary Condition** section. From the **Continuous phase boundary condition** list, choose **Slip**.
- 6 Locate the **Dispersed Phase Boundary Condition** section. Specify the $\mathbf{u}_{d,0}$ vector as

$U_{in} \cdot \text{step1}(t)$	x
0	y

- 7 From the **Dispersed phase boundary condition** list, choose **Mass flux**.
- 8 In the \dot{m}_d text field, type $\max(\text{intopL}(\text{ee.rhod} \cdot \text{phid} \cdot \text{udy}), 0) / \text{injH}$.

Inlet 3

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

2 Select Boundary 12 only.



3 In the **Settings** window for **Inlet**, locate the **Two-Phase Inlet Type** section.

4 From the **Two-phase inlet type** list, choose **Dispersed phase**.

5 Locate the **Continuous Phase Boundary Condition** section. From the **Continuous phase boundary condition** list, choose **Slip**.

6 Locate the **Dispersed Phase Boundary Condition** section. Specify the $\mathbf{u}_{d,0}$ vector as

$-U_{in} \cdot \text{step1}(t)$	x
0	y

7 From the **Dispersed phase boundary condition** list, choose **Mass flux**.

8 In the \dot{m}_d text field, type $\max(\text{intopR}(\text{ee.rhod} \cdot \text{phid} \cdot \text{udy}), 0) / \text{injH}$.

DEFINITIONS

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Maximum**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.


3 In the **Settings** window for **Maximum**, locate the **Source Selection** section.

4 From the **Selection** list, choose **All domains**.



5 In the list box, select **2**.

EULER-EULER MODEL, LAMINAR FLOW (EE)

Wall 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Wall**.
- 2 Select Boundaries 1, 4, 11, and 13 only.



- 3 In the **Settings** window for **Wall**, locate the **Continuous Phase Boundary Condition** section.
- 4 From the **Continuous velocity boundary condition** list, choose **Slip**.
- 5 Locate the **Dispersed Phase Boundary Condition** section. From the **Dispersed velocity boundary condition** list, choose **Slip**.
- 6 In the **Graphics** window toolbar, click  next to  **Go to Default View**, then choose **Go to View 1**.


DEFINITIONS

View 2

In the **Model Builder** window, under **Component 1 (comp1) > Definitions** right-click **View 2** and choose **Delete**.

MESH 1

Mapped 1

- 1 In the **Mesh** toolbar, click  **Mapped**.
- 2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Entire geometry**.

Distribution 1

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

2 Select Boundaries 6 and 8 only.

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

4 In the **Number of elements** text field, type 40.

Distribution 2

1 In the **Model Builder** window, 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 In the **Number of elements** text field, type 53.

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 300.

Distribution 4

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

2 Select Boundaries 3 and 12 only.

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

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

5 In the **Number of elements** text field, type 7.

6 In the **Element ratio** text field, type 3.

7 Select the **Symmetric distribution** checkbox.

Distribution 5

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

2 Select Boundaries 4 and 13 only.


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

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

5 In the **Number of elements** text field, type 6.

6 In the **Element ratio** text field, type 7.

Distribution 6

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 1 and 11 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Element ratio** text field, type 3.
- 6 Select the **Reverse direction** checkbox.
- 7 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 (0,0.2,10) range (10.1,0.1,30).
- 4 Right-click **Study 1 > Step 1: Time Dependent** and choose **Get Initial Value for Step**.

RESULTS

Continuous Phase Velocity (ee)

- 1 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.
- 2 Clear the **Plot dataset edges** checkbox.

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**, click to expand the **Results While Solving** section.
- 3 Select the **Plot** checkbox to monitor the velocity field throughout the computation.

Solver Configurations

In the **Model Builder** window, expand the **Study 1 > Solver Configurations** node.

Solution 1 (sol1)

- 1 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Dependent Variables 1** node, then click **Velocity Field, Continuous Phase (comp1.uc)**.

- 2 In the **Settings** window for **Field**, locate the **Scaling** section.
- 3 From the **Method** list, choose **Manual**.
- 4 In the **Scale** text field, type **Uin**.
- 5 In the **Model Builder** window, click **Velocity Field, Dispersed Phase (comp1.ud)**.
- 6 In the **Settings** window for **Field**, locate the **Scaling** section.
- 7 From the **Method** list, choose **Manual**.
- 8 In the **Scale** text field, type **Uin**.
- 9 Click  **Run**.

RESULTS

Follow these instructions to reproduce the series of dispersed-phase volume fraction plots in [Figure 2](#). Note the use of Deformation features to include a number of plots side-by-side in the same figure.

Continuous Phase Volume Fraction (ee)

- 1 In the **Model Builder** window, under **Results** click **Continuous Phase Volume Fraction (ee)**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.
- 3 Clear the **Plot dataset edges** checkbox.
- 4 Click to expand the **Plot Array** section. Select the **Enable** checkbox.

Surface 1

- 1 In the **Model Builder** window, expand the **Continuous Phase Volume Fraction (ee)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
- 4 From the **Time (s)** list, choose **0**.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 6 Click to expand the **Range** section. Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.

Surface 2

- 1 Right-click **Results > Continuous Phase Volume Fraction (ee) > Surface 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **1**.
- 4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Surface 3

- 1 Right-click **Surface 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **2**.



Surface 4

- 1 Right-click **Surface 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **3**.

Surface 5


- 1 Right-click **Surface 4** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **4**.

Surface 6

- 1 Right-click **Surface 5** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **5**.
- 4 In the **Continuous Phase Volume Fraction (ee)** toolbar, click  **Plot**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Use the plot group you just created as the starting point for reproducing [Figure 4](#).

Dispersed Phase Volume Fraction at Steady-State

- 1 In the **Model Builder** window, right-click **Continuous Phase Volume Fraction (ee)** and choose **Duplicate**.
- 2 In the **Settings** window for **2D Plot Group**, type Dispersed Phase Volume Fraction at Steady-State in the **Label** text field.
- 3 In the **Dispersed Phase Volume Fraction at Steady-State** toolbar, click  **Plot**.

Surface 1

- 1 In the **Model Builder** window, expand the **Dispersed Phase Volume Fraction at Steady-State** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **20**.

Surface 2

- 1 In the **Model Builder** window, click **Surface 2**.

- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **22**.

Surface 3

- 1 In the **Model Builder** window, click **Surface 3**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **24**.



Surface 4

- 1 In the **Model Builder** window, click **Surface 4**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **26**.

Surface 5


- 1 In the **Model Builder** window, click **Surface 5**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **28**.

Surface 6

- 1 In the **Model Builder** window, click **Surface 6**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **30**.
- 4 In the **Dispersed Phase Volume Fraction at Steady-State** toolbar, click  **Plot**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.

The following steps reproduce the plot shown in [Figure 3](#).


Dispersed Phase Bed Outflux

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Dispersed Phase Bed Outflux in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Dispersed phase outflux.
- 5 Locate the **Plot Settings** section.
- 6 Select the **y-axis label** checkbox. In the associated text field, type Mass flux (kg/(m*s)).


Global 1

- 1 Right-click **Dispersed Phase Bed Outflux** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
$\text{rhod}*(\text{outfluxdL}+\text{outfluxdR})$	kg/(m*s)	

- 4 Click to expand the **Legends** section. Clear the **Show legends** checkbox.
- 5 In the **Dispersed Phase Bed Outflux** toolbar, click  **Plot**.
To reproduce the plots in [Figure 6](#) and [Figure 5](#), begin by creating three cut line datasets.

Cut Line 2D 1

- 1 In 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 and **x** to bedW.
- 5 Clear the **Bounded by points** checkbox.

Cut Line 2D 2


- 1 Right-click **Cut Line 2D 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.
- 3 In row **Point 1**, set **y** to 3.5.
- 4 In row **Point 2**, set **y** to 3.5.

Cut Line 2D 3

- 1 Right-click **Cut Line 2D 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.
- 3 In row **Point 1**, set **y** to 8.75.
- 4 In row **Point 2**, set **y** to 8.75.

Now plot the time-averaged continuous phase volume fraction at the three y positions defined by the cut line datasets you just defined. Compare the resulting plot with that in [Figure 6](#).

ID Plot Group 6

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

Line Graph 1

- 1 Right-click **ID Plot Group 6** and choose **Line Graph**.
- 2 In the **Settings** window for **Line Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Line 2D 1**.
- 4 From the **Time selection** list, choose **First**.
- 5 Locate the **y-Axis Data** section. In the **Expression** text field, type $1 - \text{timeavg}(10, 30, \text{phid})$.
- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 7 In the **Expression** text field, type Xg / bedW .
- 8 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 9 From the **Legends** list, choose **Manual**.
- 10 In the table, enter the following settings:

Legends

 $y = 1 \text{ m}$

Line Graph 2

- 1 Right-click **Line Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Line 2D 2**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

Legends

 $y = 3.5\text{m}$

Line Graph 3


- 1 Right-click **Line Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Line 2D 3**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

Legends

 $y = 8.75\text{m}$

Averaged Continuous Phase Volume Fraction

- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 6**.

- 2 In the **Settings** window for **ID Plot Group**, type Averaged Continuous Phase Volume Fraction in the **Label** text field.
- 3 Locate the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section.
- 5 Select the **x-axis label** checkbox. In the associated text field, type $x/bedW$.
- 6 Select the **y-axis label** checkbox. In the associated text field, type Continuous phase volume fraction.
- 7 Locate the **Legend** section. From the **Position** list, choose **Lower middle**.
- 8 In the **Averaged Continuous Phase Volume Fraction** toolbar, click  **Plot**.

Finally, reproduce [Figure 5](#) by duplicating and adapting the plot group you just created.

Averaged Continuous Phase Velocity

- 1 Right-click **Averaged Continuous Phase Volume Fraction** and choose **Duplicate**.
- 2 In the **Model Builder** window, click **Averaged Continuous Phase Volume Fraction 1**.
- 3 In the **Settings** window for **ID Plot Group**, type Averaged Continuous Phase Velocity in the **Label** text field.
- 4 Locate the **Plot Settings** section. In the **y-axis label** text field, type Continuous phase velocity (m/s).


Line Graph 1

- 1 In the **Model Builder** window, click **Line Graph 1**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `timeavg(10,30,ucy)`.

Line Graph 2

- 1 In the **Model Builder** window, click **Line Graph 2**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `timeavg(10,30,ucy)`.

Line Graph 3

- 1 In the **Model Builder** window, click **Line Graph 3**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `timeavg(10,30,ucy)`.
- 4 In the **Averaged Continuous Phase Velocity** toolbar, click  **Plot**.