Two-Phase Flow Modeling of a Dense Suspension

Introduction

Liquid-solid mixtures (suspensions) are important in a variety of industrial fields, such as oil and gas refinement, paper manufacturing, food processing, slurry transport, and wastewater treatment. Several different modeling approaches have been developed, ranging from discrete, particle-based methods to macroscopic, semi-empirical two-phase descriptions. Particle-based methods are suitable when there is a limited number of solid particles. When, on the other hand, there are many particles, it is better to use a macroscopic, or averaged, model that tracks the volume fractions of the phases.

The following example illustrates how you can set up a macroscopic two-phase flow model in COMSOL Multiphysics using the Phase Transport Mixture Model, Laminar Flow multiphysics interface. The model is based on the "diffusive flux" model described in Ref. 1, Ref. 2, and Ref. 3, suitable for liquid-solid mixtures with high concentrations of solid particles. It accounts for not only buoyancy effects but also shear-induced migration; that is, the tendency of particles to migrate toward regions of lower shear rates.

The model simulates the flow of a dense suspension consisting of light, solid particles in a liquid placed between two concentric cylinders. The inner cylinder rotates while the outer one is fixed.

Model Definition

A suspension is a mixture of solid particles and a liquid. The dynamics of a suspension can be modeled by a momentum transport equation for the mixture, a continuity equation, and a transport equation for the solid phase volume fraction. The Phase Transport Mixture Model, Laminar Flow multiphysics interface automatically sets up these equations. It uses the following equation to model the momentum transport:

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla p - \nabla\cdot\mu[\nabla\mathbf{u} + \nabla\mathbf{u}^T] + \rho\mathbf{g} - \nabla\cdot[\rho_s\phi_s(1-c_s)\mathbf{u}_{\mathrm{slip}}\mathbf{u}_{\mathrm{slip}}{}^T]$$

where $\mathbf{u}$ is the mass-averaged mixture velocity (m/s), p denotes the pressure (Pa), $\mathbf{g}$ refers to the acceleration of gravity (m/s^2), and $\mathbf{u}_{\mathrm{slip}}$ gives the relative velocity between the solid and the liquid phases (m/s). Further, $\rho = (1-\phi_s)\rho_f + \phi_s\rho_s$ is the mixture density, where ρ_f and ρ_s are the pure-phase densities (kg/m^3) of liquid and solids, respectively, and where ϕ_s is the solid-phase volume fraction (m^3/m^3). $c_s = \phi_s\rho_s/\rho$ is the dimensionless particle mass fraction.

Finally, μ represents the mixture viscosity (Ns/m^2) according to the Krieger-type expression

$$\mu = \mu_f\left(1 - \frac{\phi_s}{\phi_{max}}\right)^{-2.5\phi} \tag{1}$$

where μ_f is the dynamic viscosity of the pure fluid and ϕ_{max} is the maximum packing concentration.

The mixture model uses the following form of the continuity equation

$$\frac{\partial}{\partial t}\rho + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{2}$$

The transport equation for the solid-phase volume fraction is

$$\frac{\partial}{\partial t}(\rho_s \phi_s) + \nabla \cdot (\rho_s \phi_s \mathbf{u}_s) = 0 \tag{3}$$

The solid-phase velocity, $\mathbf{u}_s$, is given by $\mathbf{u}_s = \mathbf{u} + (1 - c_s)\,\mathbf{u}_{slip}$. Assuming ρ_s is constant, this means that Equation 3 is equivalent to

$$\frac{\partial \phi_s}{\partial t} + \nabla \cdot (\phi_s \mathbf{u} + \phi_s (1 - c_s) \mathbf{u}_{slip}) = 0 \tag{4}$$

Rao and others (Ref. 2) formulate the continuity equation and the particle transport in a slightly different way. Instead of the slip velocity, $\mathbf{u}_{slip}$, they define a particle flux, $\mathbf{J}_s$ (kg $/(m^2{\cdot}s)$), and write the solid phase transport according to

$$\frac{\partial \phi_s}{\partial t} + \nabla \cdot (\phi_s \mathbf{u}) = -\frac{\nabla \cdot \mathbf{J}_s}{\rho_s} \tag{5}$$

By comparing Equation 5 with Equation 4, it is clear that they are equivalent if

$$\mathbf{u}_{slip} = \frac{\mathbf{J}_s}{\phi_s \rho_s (1 - c_s)}$$

In this example you use the model for the particle flux, $\mathbf{J}_s$, as suggested by Subia and others (Ref. 3) and Rao and others (Ref. 2), but the open and editable format of COMSOL Multiphysics makes it possible to specify the expression arbitrarily.

Following Rao and others, the particle flux is

$$\frac{\mathbf{J}_s}{\rho_s} = -[\phi D_\phi \nabla(\dot{\gamma}\phi) + \phi^2 \dot{\gamma} D_\eta \nabla(\ln \mu)] + f_h \mathbf{u}_{st} \phi$$

Here, $\mathbf{u}_{st}$ is the settling velocity (m/s) of a single particle surrounded by fluid and D_ϕ and D_η are empirically fitted parameters (m^2) given by

$$D_\phi = 0.41a^2$$
$$D_\eta = 0.62a^2$$

where a is the particle radius (m).

The shear rate tensor, $\dot{\underset{\sim}{\gamma}}$ (1/s), is given by

$$\dot{\underset{\sim}{\gamma}} = \nabla\mathbf{u} + \nabla\mathbf{u}^T$$

and its magnitude by

$$\dot{\gamma} = \sqrt{\frac{1}{2}(\dot{\underset{\sim}{\gamma}}:\dot{\underset{\sim}{\gamma}})}$$

which for a 2-dimensional problem is

$$\dot{\gamma} = \sqrt{\frac{1}{2}(4u_{x,x}^2 + 2(u_{x,y} + u_{y,x})^2 + 4u_{y,y}^2)}$$

The settling velocity, $\mathbf{u}_{st}$, for a single spherical particle surrounded by pure fluid is given by

$$\mathbf{u}_{st} = \frac{2}{9}\frac{a^2(\rho_s - \rho_f)}{\eta_0}\mathbf{g}$$

For several particles in a fluid, the settling velocity is lower. To account for the surrounding particles, the settling velocity for a single particle is multiplied by the hindering function, f_h, defined as

$$f_h = \frac{\mu_f(1-\phi_{av})}{\mu}$$

where ϕ_{av} is the average solid phase volume fraction in the suspension, μ_f is the dynamic viscosity of the pure fluid (Ns/m^2), and μ is the mixture viscosity (Equation 1).

The following table gives the physical properties of the solid and the liquid phases:

NAME	VALUE	DESCRIPTION
ρ_s	1180 kg/m^3	Density of particles
ρ_f	1253 kg/m^3	Density of pure fluid

NAME	VALUE	DESCRIPTION
a	678 µm	Particle radius
μ_f	0.589 Pa·s	Viscosity of pure fluid

BOUNDARY CONDITIONS

The suspension is placed in a Couette device, that is, between two concentric cylinders. The inner cylinder rotates while the outer one is fixed. The radii of the two cylinders are 0.64 cm and 2.54 cm, respectively. The inner cylinder rotates at a steady rate of 55 rpm. With the cylinder centered at (0,0), this corresponds to a velocity of

$$(u_x, u_y) = \frac{110\pi}{60}(y, -x)$$

The fluid and particle motion is small along the direction of the cylinder axis. You can therefore use a 2-dimensional model. Figure 1 shows the corresponding geometry.

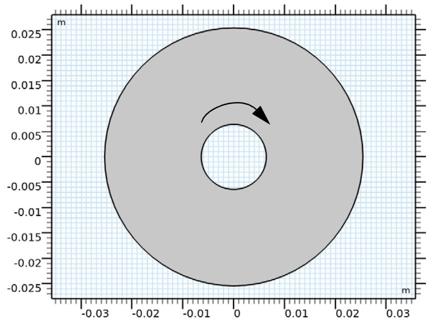

Figure 1: Geometry of the Couette device. The inner cylinder rotates, the outer one is fixed.

There is no particle flux through the boundaries, and the suspension velocity satisfies no slip conditions at all walls.

INITIAL CONDITIONS

There are two different initial particle distributions. In the first example, the particles are evenly distributed within the device. In the second example, the particles are initially gathered at the top of the device.

CASE I — INITIALLY EVENLY DISTRIBUTED PARTICLES

A suspension with particles lighter than the fluid is placed in a concentric Couette device. Initially, the particles are evenly distributed with a constant volume fraction of 0.35. The shear rate in the device varies radially across the gap and thus it is expected that particles migrate (shear-induced migration) from regions of high shear to regions of low shear (toward the outer wall). Because the particles are lighter than the fluid, they also rise.

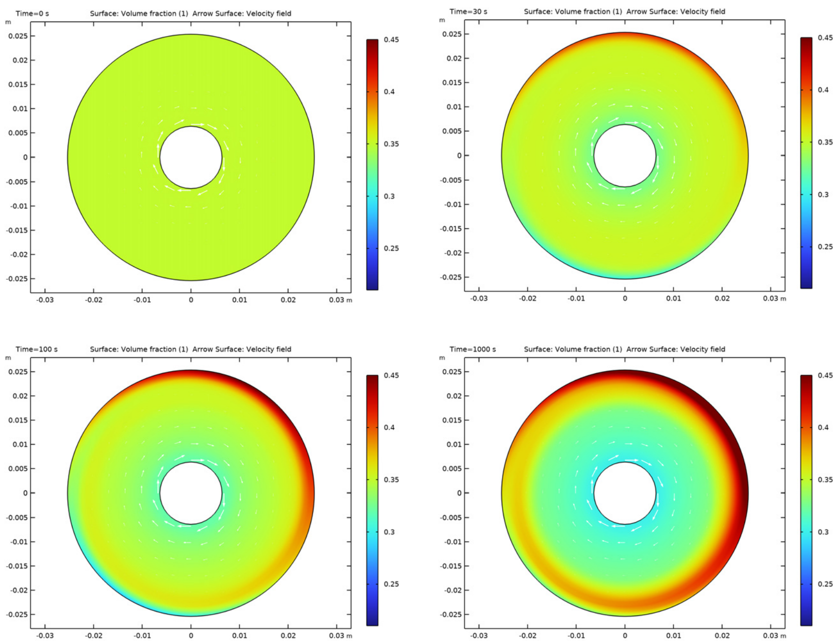

Figure 2: The particle concentration ϕ_s at different times. The particles move to regions with lower shear rate and rise because of buoyancy.

Figure 2 shows the particle concentration ϕ_s in the device at $t = 0$ s, $t = 30$ s, $t = 100$ s and $t = 1000$ s. The migration of the particles toward the outer wall is apparent. As a result of the shear induced migration and gravity, the solid phase volume fraction approaches the value for maximum packing close to the upper right outer wall. The suspension viscosity thus becomes high in this region. The results compare well with those presented in Ref. 2.

CASE 2 — PARTICLES INITIALLY GATHERED AT THE TOP OF THE DEVICE

In this case the particles are initially gathered at the top of the device. The particle volume fraction is initially zero in the lower part, while it is 0.59 at the top.

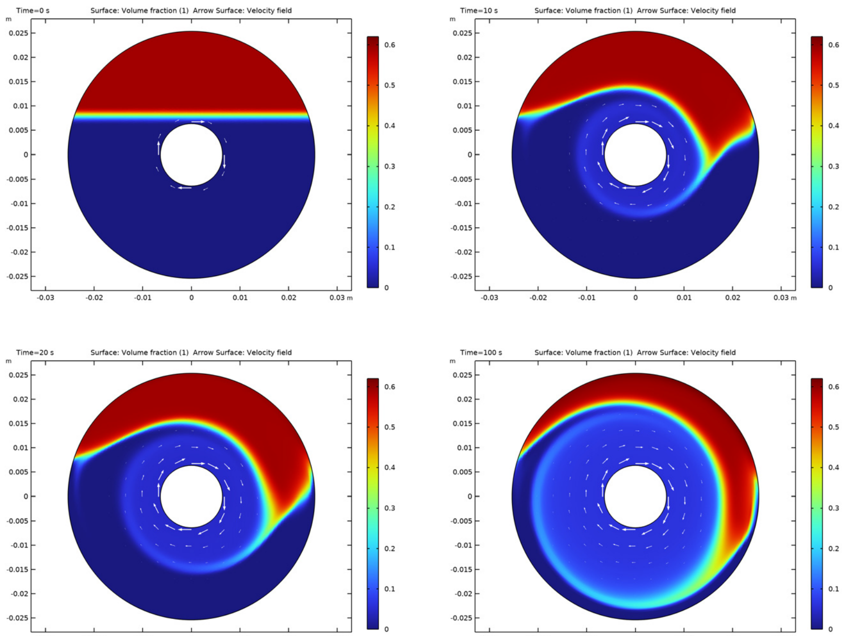

Figure 3: Particle concentrations for t = 0 s, 10 s, 20 s, and100 s with particles initially at the top. Note that the same color range values are used in each of the plots.

Figure 3 shows the numerically predicted particle concentration at times 0 s, 10 s, 20 s, and 100 s. Initially, the particle motion is dominated by inertia and the effect of the shear-induced migration is not visible. At later times, shear-induced migration causes the particles to move toward the outer boundary. In this case also, the results agree well with the results in Ref. 2.

References

1. R.J. Phillips, R.C. Armstrong, R.A. Brown, A.L. Graham, and J.R. Abbot, “A Constitutive Equation for Concentrated Suspensions that Accounts for Shear-induced Particle Migration,” *Phys. Fluids A*, vol. 4, pp. 30–40, 1992.

2. R. Rao, L. Mondy, A. Sun, and S. Altobelli, “A Numerical and Experimental Study of Batch Sedimentation and Viscous Resuspension,” *Int. J. Num. Methods in Fluids*, vol. 39, pp. 465–483, 2002.

3. S.R. Subia, M.S. Ingber, L.A. Mondy, S.A. Altobelli, and A.L. Graham, “Modelling of Concentrated Suspensions Using a Continuum Constitutive Equation,” *J. Fluid Mech.*, vol. 373, pp. 193–219, 1998.

Notes About the COMSOL Implementation

The shear rate is discretized as an additional equation to improve accuracy because the particle flux contains derivatives of this quantity, which in turn depend on the derivatives of the velocity.

Application Library path: `CFD_Module/Verification_Examples/dense_suspension`

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>Phase Transport Mixture Model>Laminar Flow**.

3 Click **Add**.

4 In the **Added physics interfaces** tree, select **Phase Transport (phtr)**.

5 In the **Volume fractions** table, enter the following settings:

phic
phid

6 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>General Form PDE (g)**.

7 Click **Add**.

8 In the **Dependent variables** table, enter the following settings:

gamma_dot

9 Click **Define Dependent Variable Unit**.

10 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	1/s

11 Click **Define Source Term Unit**.

12 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1/s

13 Click **Study**.

14 In the **Select Study** tree, select **General Studies>Time Dependent**.

15 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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 `dense_suspension_parameters.txt`.

GEOMETRY 1

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `0.0064`.

4 Click **Build Selected**.

Circle 2 (c2)

1 In the **Geometry** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `0.0254`.

4 Click **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **c2** only.

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

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **c1** only.

6 Click **Build Selected**.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `dense_suspension_variables.txt`.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 Select the **Include gravity** check box.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 1 only.

Wall 2

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

2 Select Boundaries 3, 4, 6, and 7 only.

3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

4 From the **Translational velocity** list, choose **Manual**.

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

`c_vel*y`	x
`-c_vel*x`	y

PHASE TRANSPORT (PHTR)

Initial Values 1

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

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

3 In the $s_{0,\mathrm{phid}}$ text field, type `phi0`.

GENERAL FORM PDE (G)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **General Form PDE (g)**.

2 In the **Settings** window for **General Form PDE**, click to expand the **Discretization** section.

3 From the **Element order** list, choose **Linear**.

General Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE (g)** click **General Form PDE 1**.

2 In the **Settings** window for **General Form PDE**, locate the **Conservative Flux** section.

3 Specify the Γ vector as

0	x
0	y

4 Locate the **Source Term** section. In the f text field, type `gamma_dot-sqrt(0.5*(4*ux^2+2*(uy+vx)^2+4*vy^2)+eps)`.

5 Locate the **Damping or Mass Coefficient** section. In the d_a text field, type `0`.

Flux/Source 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Flux/Source**.

2 In the **Settings** window for **Flux/Source**, locate the **Boundary Selection** section.

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

MULTIPHYSICS

Mixture Model 1 (mfmm1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Mixture Model 1 (mfmm1)**.
2 In the **Settings** window for **Mixture Model**, locate the **Physical Model** section.
3 From the **Slip model** list, choose **User defined**. From the **Mixture viscosity model** list, choose **Krieger type**.
4 In the ϕ_{max} text field, type `phi_max`.
5 Locate the **Continuous Phase Properties** section. From the ρ_c list, choose **User defined**. In the associated text field, type `rho_f`.
6 From the μ_c list, choose **User defined**. In the associated text field, type `eta_f`.
7 Locate the **Dispersed Phase 2 Properties** section. From the ρ_{phid} list, choose **User defined**. In the associated text field, type `rho_s`.
8 Specify the $\mathbf{u}_{slip,phid}$ vector as

`J1/(phid*rho_s*(1-cd))`	x
`J2/(phid*rho_s*(1-cd))`	y

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Calibrate for** list, choose **Fluid dynamics**.
4 From the **Predefined** list, choose **Finer**.
5 Click the **Custom** button.
6 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.0012`.
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 `0 30 100 1000`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.
4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
5 In the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

To visualize the volume fraction of the **Dispersed Phase** as a surface plot along with the mixture velocity field as an arrow surface plot, follow the steps given below.

Surface

1 In the **Model Builder** window, expand the **Volume Fraction (phtr)** node, then click **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `phid`.
4 Click to expand the **Range** section. Select the **Manual color range** check box.

   Fix the color range so that the solutions for different time values use the same color legend.
5 In the **Minimum** text field, type `0.21`.
6 In the **Maximum** text field, type `0.45`.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Volume Fraction (phtr)** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.
3 From the **Color** list, choose **White**.

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

Volume Fraction (phtr)

1 In the **Model Builder** window, click **Volume Fraction (phtr)**.

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

3 From the **Time (s)** list, choose **0**.

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

Repeat the two previous instructions for the times 30 s, 100 s and 1000 s to generate Figure 2.

This completes Case 1. Now model the case with the particles initially gathered at the top.

DEFINITIONS

Define the particle concentration distribution using the **Step** function.

Step 1 (step1)

1 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 In the **Size of transition zone** text field, type `2*2`.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
phi0_2	step1(y[1/mm]-8)*0.59		Initial concentration

PHASE TRANSPORT (PHTR)

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

Initial Values 2

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

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

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

4 Locate the **Initial Values** section. In the $s_{0,\mathrm{phid}}$ text field, type `phi0_2`.

In order to avoid dividing by zero in the region where the particle volume fraction is initially zero, modify the expressions for the slip velocity.

MULTIPHYSICS

Mixture Model 1 (mfmm1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Mixture Model 1 (mfmm1)**.

2 In the **Settings** window for **Mixture Model**, locate the **Dispersed Phase 2 Properties** section.

3 Specify the $\mathbf{u}_{\mathrm{slip,phid}}$ vector as

J1/(phid*rho_s*(1-cd)+eps)	x
J2/(phid*rho_s*(1-cd)+eps)	y

COMPONENT 1 (COMP1)

Create a finer mesh compared to the one used in the previous case.

MESH 2

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

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

2 From the **Predefined** list, choose **Extra fine**.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.0006`.

5 Click **Build All**.

Next, add a **Time Dependent** study to compute the particle distribution.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

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

2 In the **Output times** text field, type `range(0,10,100)`.

3 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 2

4 In the **Home** toolbar, click **Compute**.

RESULTS

To visualize the volume fraction of the dispersed phase as a surface plot along with the arrow plot of the mixture velocity (Figure 3), follow the steps given below.

Volume Fraction (phtr) 1

1 In the **Model Builder** window, under **Results** click **Volume Fraction (phtr) 1**.

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

3 From the **Time (s)** list, choose **0**.

Surface

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

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

3 In the **Expression** text field, type `phid`.

4 Locate 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 `0.62`.

Arrow Surface 1

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

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **White**.

4 In the **Volume Fraction (phtr) 1** toolbar, click **Plot**.

Volume Fraction (phtr) 1

1 In the **Model Builder** window, click **Volume Fraction (phtr) 1**.

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

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

4 In the **Volume Fraction (phtr) 1** toolbar, click **Plot**.

5 From the **Time (s)** list, choose **20**.

6 In the **Volume Fraction (phtr) 1** toolbar, click **Plot**.

7 From the **Time (s)** list, choose **100**.

8 In the **Volume Fraction (phtr) 1** toolbar, click **Plot**.

To make **Study 1** behave as when it was first created, the feature **Initial Values 2** added for **Study 2** must be disabled.

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 **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Phase Transport (phtr)>Initial Values 2**.

5 Click **Disable**.