Created in COMSOL Multiphysics 6.2

Three-Phase Mixer

Introduction

This example of a turbulent fluid-solid mixture in a mixer shows how to set up a time-dependent, turbulent three-phase flow simulation together with a rotating domain. The mixture is a suspension of a fluid and two particle populations, one population consisting of particles lighter than the fluid and the other of particles heavier than the fluid. Before the impeller starts to rotate, the mixture tends to separate since the light particles rise to the top of the tank and the heavy particles sediment at the bottom. When the suspension is stirred, the fluid and the particles mix again.

Model Definition

The mixer is an under-baffled stirred vessel. Figure 1 shows the model geometry. The tank is cylindrical with a three-bladed impeller placed close to the curved bottom. It has a diameter of 450 mm and a height of 300 mm. The tank is completely filled with the fluid-solid mixture and initially the distribution of both particle populations is homogeneous, with a volume fraction of 0.1.

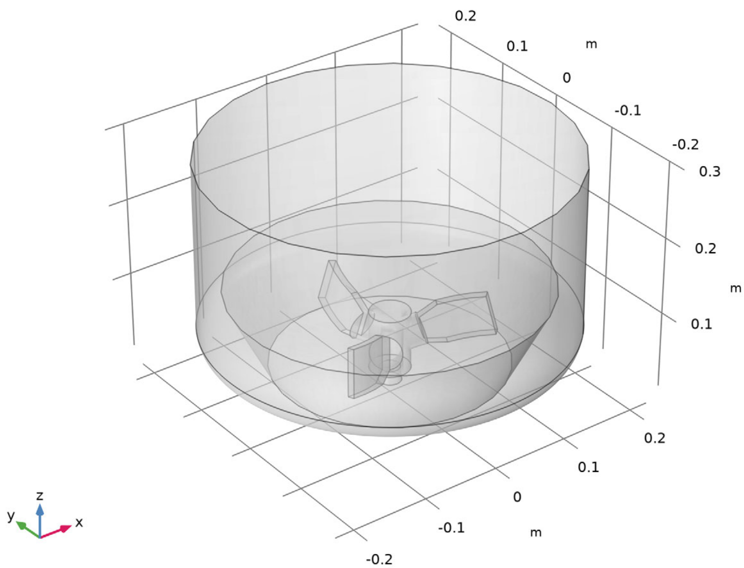

Figure 1: Geometry of the mixer with a three-bladed impeller.

The suspension consists of water (with density and viscosity taken from the Water, liquid material from COMSOL's built-in material library) and two particle populations with particle densities 1100 kg/m^3 and 850 kg/m^3, respectively. The particles in both populations have a diameter of 1 mm. The Hadamard–Rybczynski model is used to compute particle slip velocities, and the mixture viscosity is calculated using the Krieger model. The model uses a time-dependent simulation starting from rest. The impeller starts to rotate after 1.5 s and then the rotational velocity of the impeller is gradually increased until it reaches 100 rpm at t =2. 5 s. The simulation is continued until t = 10 s. The flow in the mixer is modeled using the Phase Transport Mixture Model Turbulent Flow, k-ε interface.

Results and Discussion

The results presented in this section concern the velocity of the mixture and the volume fractions of the two particle populations. During the initial 1.5 s, when the suspension is not stirred, the mixture separates, which is seen in Figure 2. The left column shows plots of the volume fraction of the heavy particles in the xz-cut plane at t = 0.5 s, t = 1 s, and t = 1.5 s. The heavy particles sediment at the bottom of the tank due to gravity. The right column shows the volume fraction of the light particles at the same time instances, and they accumulate at the top of the tank due to buoyancy forces.

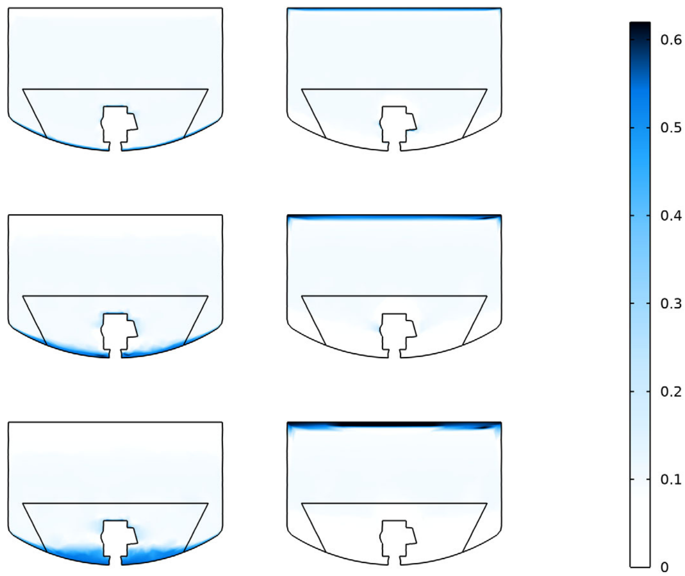

Figure 2: Volume fractions of the heavy (left column) and light particles (right column) at t = 0.5 s, t = 1 s, and t = 1.5 s, (top to bottom rows).

At $t = 1.5$ s the impeller starts rotating and at $t = 2.5$ s it is rotating full speed at 100 rpm. Figure 3 and Figure 4 show arrow and slice plots of the velocity (magnitude) at $t = 4$ s and $t = 10$ s, respectively. The plots show that, even though the impeller has been rotating at full speed for 1.5 s, at $t = 4$ s the fluid near the top of the tank is still accelerating, since it has not yet reached its final rotational velocity, which is shown in Figure 4.

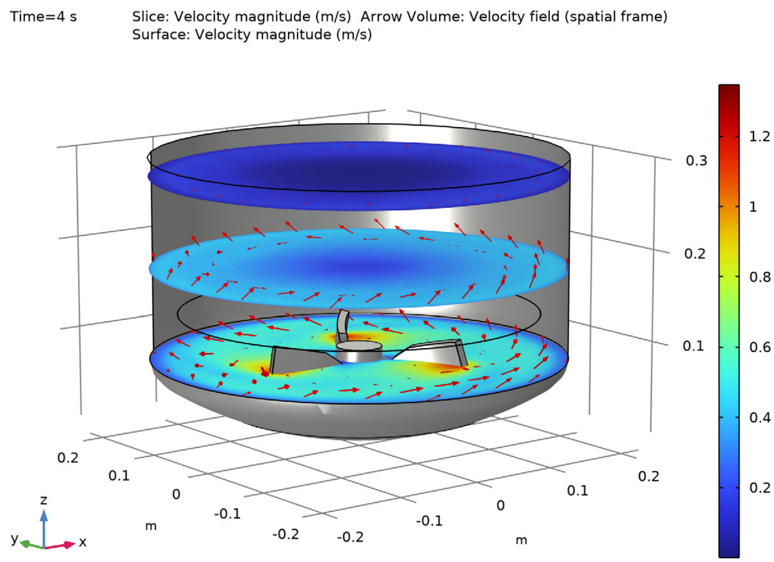

Figure 3: Velocity field at $t = 4$ s. The impeller is rotating at 100 rpm, but the rotation has not yet accelerated the fluid at the top of the tank to maximum speed.

Figure 5 through Figure 8 show slice plots of the volume fractions of both particle populations at $t = 3$ s, $t = 5$ s, $t = 7$ s, and $t = 9$ s to illustrate the progressive mixing: at $t = 3$ s, the heavy (blue gradient color scale) and the light (red gradient color scale) particles are still accumulated, respectively, at the bottom and top of the tank, while for later time instances the particles are more evenly spread. The plot for $t = 9$ s (Figure 8) shows that the particle distribution is still not completely homogeneous after 6.5 s of stirring at full speed. To obtain a more homogeneous distribution, a faster impeller speed is needed.

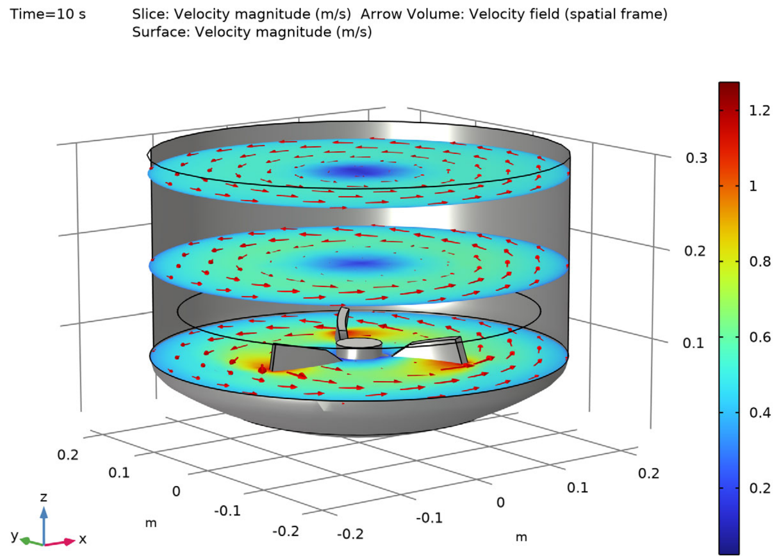

Figure 4: Velocity field at t *= 10 s.*

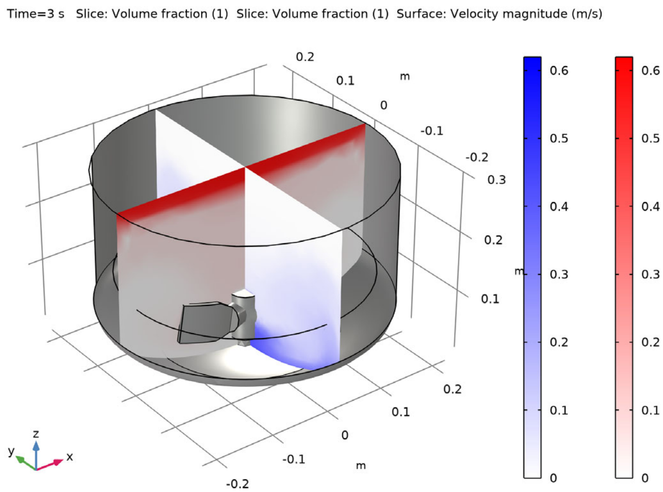

Figure 5: Volume fractions of the light particles (red gradient color scale) and heavy particles (blue gradient color scale) at $t = 3$ *s.*

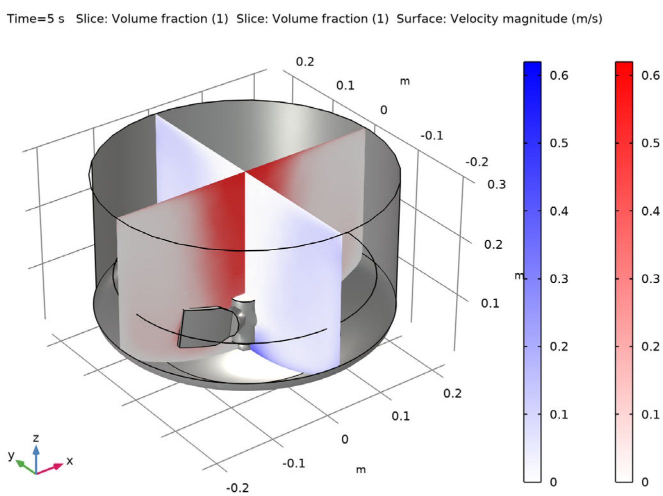

Figure 6: Volume fractions of the light particles (red gradient color scale) and heavy particles (blue gradient color scale) at $t = 5$ *s.*

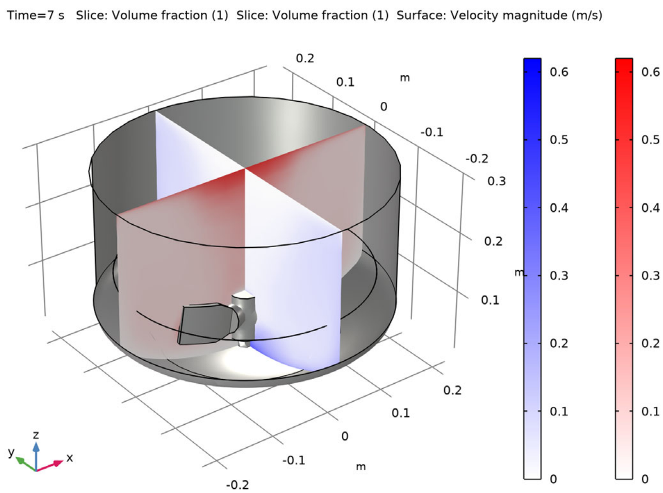

Figure 7: Volume fractions of the light particles (red gradient color scale) and heavy particles (blue gradient color scale) at t = 7 s.

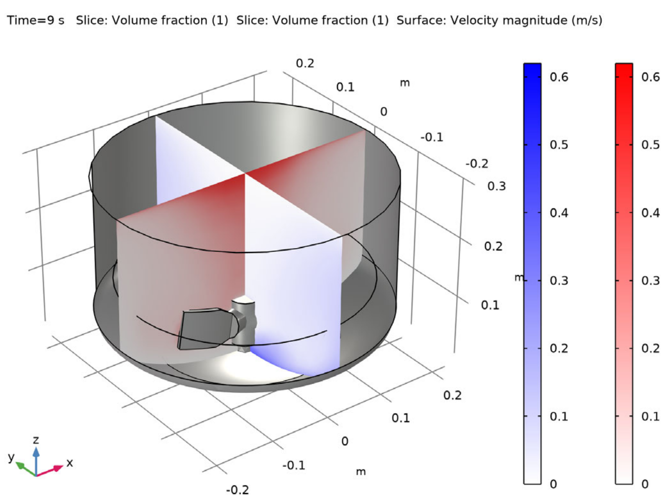

Figure 8: Volume fractions of the light particles (red gradient color scale) and heavy particles (blue gradient color scale) at t = 9 s.

Application Library path: `Mixer_Module/Tutorials/three_phase_mixer`

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 **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Multiphase Flow>Rotating Machinery, Phase Transport Mixture Model>Turbulent Flow>Turbulent Flow, k-ε**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

DEFINITIONS

Ramp 1 (rm1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Ramp**.
2 In the **Settings** window for **Ramp**, locate the **Parameters** section.
3 In the **Location** text field, type `1.55`.
4 Select the **Cutoff** check box.
5 Click to expand the **Smoothing** section. Select the **Size of transition zone at start** check box.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Water, liquid**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MOVING MESH

Rotating Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Moving Mesh** click **Rotating Domain 1**.
2 Select Domain 2 only.
3 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.
4 In the *f* text field, type `100[rpm]*rm1(t*1[1/s])`.

TURBULENT FLOW, K-ε (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.
2 In the **Settings** window for **Turbulent Flow, k-ε**, locate the **Physical Model** section.
3 Select the **Include gravity** check box.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2 Select Boundary 13 only.
3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

4 From the **Translational velocity** list, choose **Zero (Fixed wall)**.

The **Translational velocity** is set to **Zero (Fixed Wall)** to ensure that the wall is not moving. If **Automatic from frame** is selected, the wall will rotate due to the angular velocity of the **Rotating Domain**.

Pressure Point Constraint 1

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

2 Select Point 2 only.

PHASE TRANSPORT (PHTR)

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

2 In the **Settings** window for **Phase Transport**, click to expand the **Dependent Variables** section.

3 Click **Add Volume Fraction**.

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 $s_{0,s2}$ text field, type `0.1`.

4 In the $s_{0,s3}$ text field, type `0.1`.

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 In the μ text field, type `mfmm1.muc*max(1-max(0,min(s2+s3,0.999*0.62))/0.62, eps)^(-2.5*0.62)`.

4 From the **Slip model** list, choose **Hadamard-Rybczynski**.

5 Locate the **Dispersed Phase 2 Properties** section. From the ρ_{s2} list, choose **User defined**. In the associated text field, type `1100[kg/m^3]`.

6 Locate the **Dispersed Phase 3 Properties** section. From the ρ_{s3} list, choose **User defined**. In the associated text field, type `850[kg/m^3]`.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Coarser**.

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(0,0.5,10)`.

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 From the **Steps taken by solver** list, choose **Strict**.

6 Find the **Algebraic variable settings** subsection. In the **Fraction of initial step for Backward Euler** text field, type `1`.

7 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1>Segregated 1** node, then click **Velocity u, Pressure p**.

8 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

9 From the **Jacobian update** list, choose **On every iteration**.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1>Segregated 1** click **Volume Fractions**.

11 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

12 From the **Jacobian update** list, choose **On every iteration**.

13 Click **Compute**.

RESULTS

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

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

3 From the **Plane** list, choose **xz-planes**.

2D Plot Group 6

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 From the **Dataset** list, choose **Cut Plane 1**.

Surface 1

1 Right-click **2D Plot Group 6** and choose **Surface**.

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

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

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

5 In the **Maximum** text field, type `0.62`.

6 Locate the **Coloring and Style** section. Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

8 Click **OK**.

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

10 From the **Color table transformation** list, choose **Reverse**.

2D Plot Group 6

1 In the **Model Builder** window, click **2D Plot Group 6**.

2 In the **Settings** window for **2D Plot Group**, click to expand the **Plot Array** section.

3 Select the **Enable** check box.

4 From the **Array shape** list, choose **Square**.

5 From the **Order** list, choose **Column-major**.

6 Locate the **Data** section. From the **Time (s)** list, choose **1.5**.

Surface 1

In the **Model Builder** window, right-click **Surface 1** and choose **Duplicate**.

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 **Dataset** list, choose **Cut Plane 1**.

4 From the **Time (s)** list, choose **1**.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

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

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 **0.5**.

Surface 1

In the **Model Builder** window, right-click **Surface 1** and choose **Duplicate**.

Surface 4

1 In the **Model Builder** window, click **Surface 4**.

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

3 In the **Expression** text field, type `s3`.

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

5 Right-click **Surface 4** and choose **Duplicate**.

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 **Dataset** list, choose **Cut Plane 1**.

4 From the **Time (s)** list, choose **1**.

5 Right-click **Surface 5** and choose **Duplicate**.

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 **0.5**.

2D Plot Group 6

1 Click the **Show Grid** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **2D Plot Group 6**.

3 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

4 From the **Title type** list, choose **None**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

The previous instructions created the plots of the volume fractions as shown in Figure 2.

The following instructions create the plots in Figure 3 and Figure 4.

Slice

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Slice**.
2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
3 From the **Plane** list, choose **xy-planes**.
4 In the **Planes** text field, type `3`.
5 Select the **Interactive** check box.
6 In the **Shift** text field, type `0.03`.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Arrow Volume**.
2 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.
3 Find the **x grid points** subsection. In the **Points** text field, type `10`.
4 Find the **y grid points** subsection. In the **Points** text field, type `10`.
5 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.
6 In the **Coordinates** text field, type `0.08 0.18 0.28`.

Surface 1

Right-click **Velocity (spf)** and choose **Surface**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `2, 3, 9-34` in the **Selection** text field.
5 Click **OK**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.
2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
3 From the **Appearance** list, choose **Custom**.
4 From the **Material type** list, choose **Steel**.

Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

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

4 In the **Velocity (spf)** toolbar, click **Plot**.

This reproduces the plot in Figure 3. Reproduce the plot in Figure 4 by choosing **10** from the **Times (s)** list.

The following last set of instructions creates the plots in Figure 5 to Figure 8.

Volume Fraction (phtr)

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

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **Frame** list, choose **Spatial (x, y, z)**.

Slice

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

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

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

4 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Gradient**.

6 From the **Bottom color** list, choose **Blue**.

7 From the **Color table transformation** list, choose **Reverse**.

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

9 In the **Minimum** text field, type `0`.

10 In the **Maximum** text field, type `0.62`.

11 Right-click **Slice** and choose **Duplicate**.

Slice 2

1 In the **Model Builder** window, click **Slice 2**.

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

3 In the **Expression** text field, type `s3`.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 In the **Planes** text field, type `1`.

6 Locate the **Coloring and Style** section. From the **Bottom color** list, choose **Red**.

Surface 1

In the **Model Builder** window, under **Results>Velocity (spf)** right-click **Surface 1** and choose **Duplicate**.

Surface 2

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

Drag and drop below **Volume Fraction (phtr)>Slice 2**.

Volume Fraction (phtr)

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

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

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

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

This reproduces the plot in Figure 5. To reproduce the remaining plots, choose different time instances from the **Times (s)** list.