Created in COMSOL Multiphysics 6.1

Terminal Falling Velocity of a Sand Grain

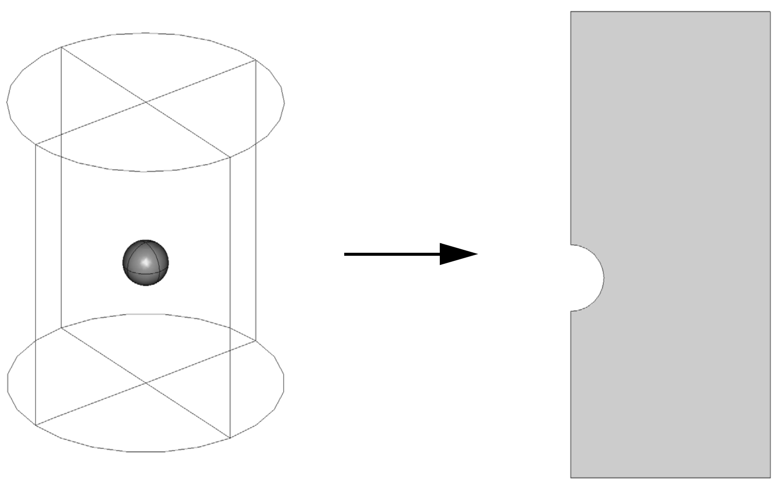
Introduction

The first stop for polluted water entering a water work is normally a large tank, where large particles are left to settle. More generally, gravity settling is an economical method of separating particles. If the fluid in the tank is moving at a controlled low velocity, the particles can be sorted in separate containers according to the time it takes for them to reach the bottom.

This application simulates a spherical sand grain falling in water. The grain accelerates from standstill and rapidly reaches its terminal velocity. The results agree with experimental studies. The model is an axially symmetric fluid-flow simulation in a moving coordinate system, coupled to an ordinary differential equation (ODE) describing the grain's motion.

Model Definition

The model couples the flow simulation in cylindrical coordinates with an ODE for the force balance of the particle. Due to axial symmetry, you can model the flow in 2D instead of 3D. The figure below shows the modeling domain.



DOMAIN EQUATIONS

The fluid flow is described by the Navier–Stokes equations

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p = \mathbf{F}$$

$$\nabla \cdot \mathbf{u} = 0$$

where ρ denotes the density (kg/m^3), $\mathbf{u}$ the velocity (m/s), μ the viscosity (Ns/m^2), and p the pressure (Pa). The fluid is water with a viscosity of $1.51 \cdot 10^{-3}$ Ns/m^2 and a density of 1000 kg/m^3. The model uses the accelerating reference system of the sand grain. This means that the volume force density $\mathbf{F}$ is given by:

$$F_r = 0, \qquad F_z = -\rho(a + g)$$

where a (m/s^2) is the acceleration of the grain and $g = 9.81$ m/s^2 is the acceleration due to gravity. The ODE that describes the force balance is:

$$m\ddot{x} = F_g + F_z$$

where m (kg) denotes the mass of the particle, x (m) the position of the particle, F_g (N) the gravitational force, and F_z the z-component of the force that the water exerts on the sand grain. The gravitational force is given by:

$$F_g = -\rho_{\mathrm{grain}} V_{\mathrm{grain}} g$$

where V_{grain} (m^3) is the volume of the sand grain and ρ_{grain} (kg/m^3) its density. The force that the water exerts on the grain could be calculated by integrating the normal component of the stress tensor over the surface of the particle:

$$F_z = 2\pi \int_S r\hat{\mathbf{z}} \cdot [-pI + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)]\mathbf{n}dS$$

where r (m) is the radial coordinate, $\hat{\mathbf{z}}$ the unit vector in the z direction and $\mathbf{n}$ is the normal vector on the surface of the grain.

An alternative approach, leading to a higher-order representation of the drag force on the grain, is used in this model. In the weak form of the Navier–Stokes equations, the stress is integrated by parts. When applying weak constraints for the Dirichlet condition on the velocity on the surface of the grain, the Lagrange multipliers enforcing the constraints for the two velocity components impose reaction forces (per unit area) on the fluid in the r and z directions. Because of the aforementioned integration by parts, the Lagrange multipliers become equal to the local surface stresses in the two directions. The drag force on the grain can thus be obtained as the surface integral of the negative value of the Lagrange multiplier in the z direction.

The initial values for position and velocities are $u_0 = v_0 = x_0 = \dot{x}_0 = 0$.

BOUNDARY CONDITIONS

At the sphere's surface, the fluid velocity relative the sphere is zero, that is $\mathbf{u} = \mathbf{0}$ — a situation described by the *no slip* wall condition. At the inlet of the fluid domain the velocity equals the falling velocity: $\mathbf{u} = (0, \dot{x})$. At the outer boundary of the water domain, the normal velocity and the tangential shear stress both vanish, which means that a *symmetry* condition applies. Furthermore, a *neutral* condition, $\mathbf{n} \cdot [-pI + \eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] = 0$, describes the outlet, and an axial symmetry condition models the symmetry axis at $r = 0$.

Results

Figure 1 shows the velocity field at the final simulation time, $t = 1$ s, when the particle has reached steady state.

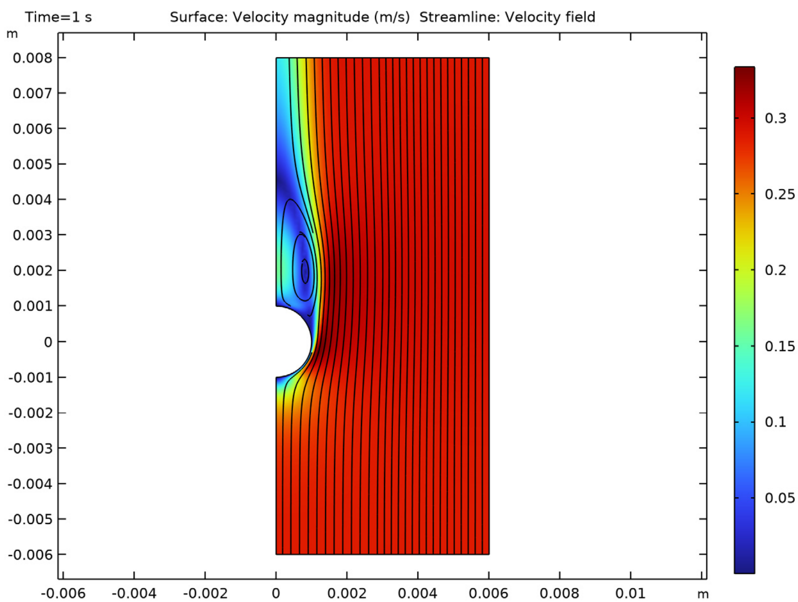


Figure 1: The velocity field at steady state. Note that the velocities are plotted in the reference system of the sand grain.

The following series of snapshots (Figure 2) displays velocity field from a moment just after the sand grain is released until it is approaching steady state. Notice the recirculation forming above the grain.

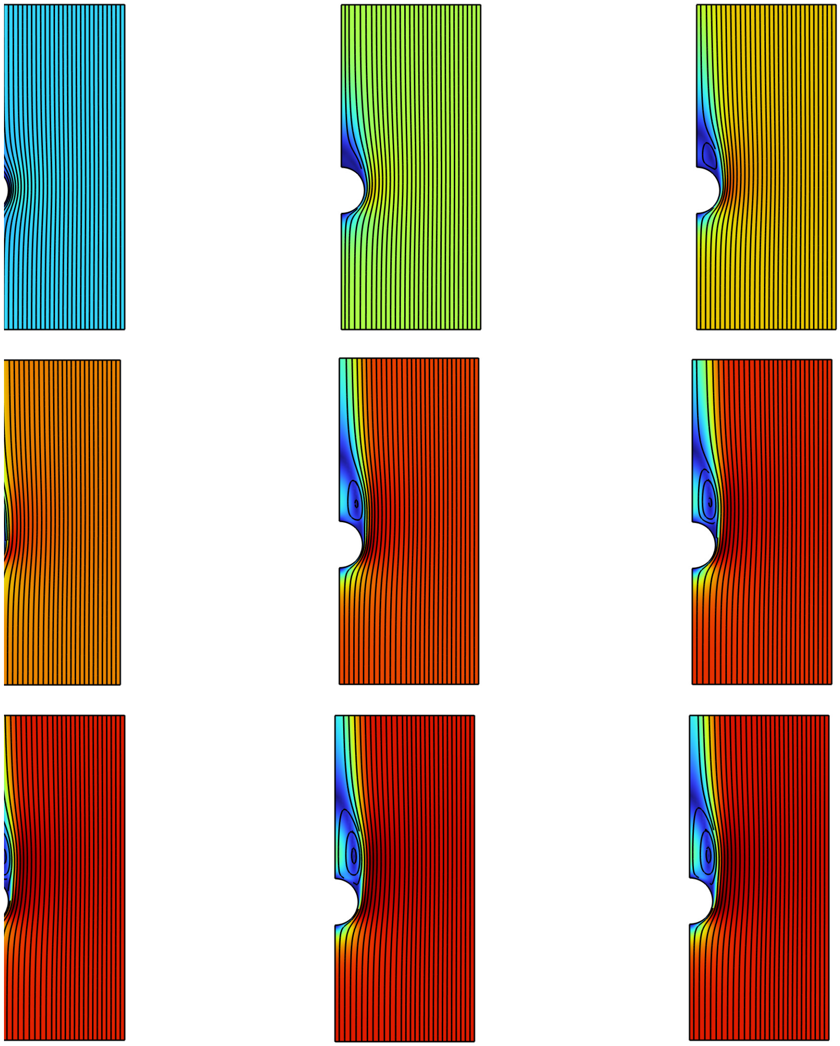


Figure 2: The velocity field around the sand grain at a series of times (t = 0.025 s, 0.05 s, 0.075 s, 0.1 s, 0.15 s, 0.2 s, 0.3 s, 0.5 s, and 0.75 s). For a color legend, see Figure 1.

Figure 3 shows the falling velocity of the grain as a function of time.

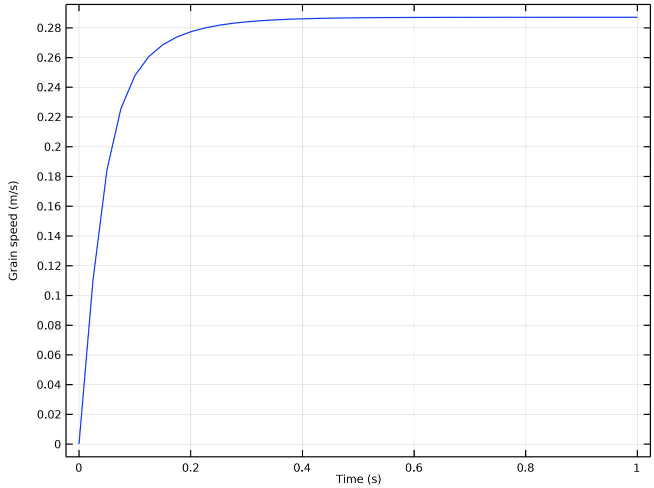


Figure 3: Falling velocity (m/s) of the grain versus time. After the solution time of 1 s, the velocity approaches the terminal velocity.

The terminal velocity equals 0.288 m/s. When this state is reached, the gravity and the forces from the water cancel out. Figure 4 shows the forces on the sand grain.

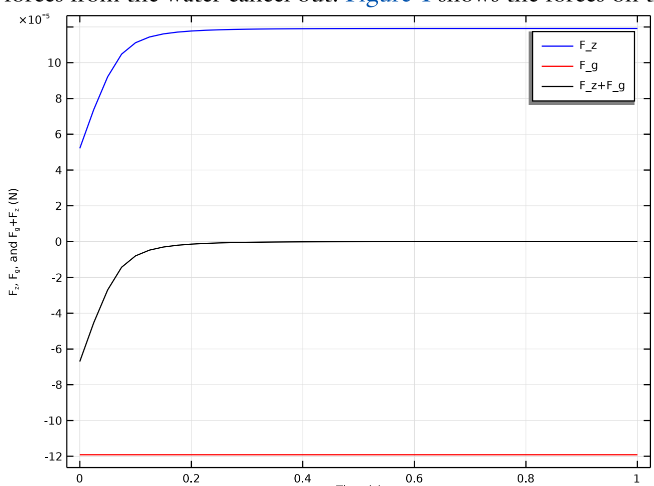


Figure 4: The forces on the sand grain. The force that the water exerts on the sphere (blue line) increases as the grain gains speed. The gravity force (red line) remains the same, and the total force (black line) tends toward zero as the solution approaches steady state.

Several approximate equations have been proposed for the terminal velocity of a sphere falling in a fluid. Ref. 1 cites the following expression for the total force that the fluid exerts on the sphere, as a function of the velocity:

$$F = \frac{\pi}{4}d^2\rho v^2(1.84\mathrm{Re}^{-0.31} + 0.293\mathrm{Re}^{0.06})^{3.45}$$

Here d (m) is the diameter of the sphere, ρ (kg/m^3) is the fluid density, v (m/s) is the velocity, and $\mathrm{Re} = (\rho v d)/\mu$ is the Reynolds number, with μ (Ns/m^2) being the viscosity of the fluid. The gravity force is given analytically as $F_g = \pi d^3(\rho - \rho_s)g/6$, where ρ_s (kg/m^3) is the density of the sphere. Equating the two forces and introducing the values used in the simulation gives an approximate terminal velocity of 0.284 m/s.

The same reference discusses correction factors for nonspherical particles. You can easily adapt the model to hold for a general axially symmetric object (by redrawing the geometry) or even an arbitrarily shaped object (by modeling in 3D).

Reference

1. J.M. Coulson and J.F. Richardson, *Chemical Engineering vol. 2*, 4th ed., 1993.

Application Library path: `COMSOL_Multiphysics/Fluid_Dynamics/falling_sand`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

A set of global parameters is provided in a text file.

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 `falling_sand_parameters.txt`.

GEOMETRY 1

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 `6e-3`.

4 In the **Height** text field, type `14e-3`.

5 Locate the **Position** section. In the **z** text field, type `-6e-3`.

6 Click **Build Selected**.

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 `1e-3`.

4 Click **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **r1** 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**.

This completes the model geometry.

DEFINITIONS

Define a nonlocal integration coupling and a variable using this coupling. Later, you will use this variable to calculate the drag force.

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 Select Boundaries 6 and 7 only.

Variables 1

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Description
`F_z`	`intop1(-w_lm[Pa])`	`Drag force`

Here, `w_lm` is a Lagrange multiplier and represents the reaction force (per unit area) when imposing the Dirichlet condition for the z-component of the fluid velocity at the wall by means of weak constraints.

LAMINAR FLOW (SPF)

Fluid Properties 1

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

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

3 From the ρ list, choose **User defined**. In the associated text field, type `rho_water`.

4 From the μ list, choose **User defined**. In the associated text field, type `mu_water`.

Volume Force 1

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

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

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

4 Locate the **Volume Force** section. Specify the **F** vector as

0	r
-rho_water*(Xdott+g_const)	z

Next, use weak constraints to impose the wall boundary condition on the grain. Add **Global Equations** nodes for the equation of motion for the falling grain, and constrain the Lagrange multiplier corresponding to the horizontal velocity at the extreme points of the wall so that it respects the symmetry of the problem. By default, these features are not visible.

5 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

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

8 Click **OK**.

Wall 1

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

2 In the **Settings** window for **Wall**, click to expand the **Constraint Settings** section.

3 From the **Constraints** list, choose **Use weak constraints**.

Pointwise Constraint 1

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

2 Select Points 2 and 3 only.

3 In the **Settings** window for **Pointwise Constraint**, locate the **Pointwise Constraint** section.

4 In the **Constraint expression** text field, type `u_lm`.

5 Click to expand the **Discretization** section. Find the **Base geometry** subsection. From the **Element order** list, choose **Linear**.

Enter the second-order ODE as a system of two coupled first-order ODEs:

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
X	Xt-Xdot	0	0

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `displacement` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Displacement (m)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `velocity` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Velocity (m/s)**.

14 Click **OK**.

Global Equations 2

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
Xdot	Xdott-(F_z+F_g)/ m_grain	0	0

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, select **General>Velocity (m/s)** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Global Equations**, locate the **Units** section.

8 Click **Select Source Term Quantity**.

9 In the **Physical Quantity** dialog box, select **General>Acceleration (m/s^2)** in the tree.

10 Click **OK**.

Inlet 1

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

2 Select Boundary 2 only.

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

4 In the U_0 text field, type `-Xdot`.

Open Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.
2 Select Boundary 4 only.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.
3 From the **Wall condition** list, choose **Slip**.
4 Select Boundary 5 only.

MESH 1

Free Triangular 1

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

Size 1

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 Select Boundary 3 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Maximum element size** check box. In the associated text field, type `1.5e-4`.

Size 2

1 In the **Model Builder** window, 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 Select Boundaries 6 and 7 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Maximum element size** check box. In the associated text field, type `0.75e-4`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Finer**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type `1.05`.

6 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(0,0.025,1)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.001`.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

4 From the **Steps taken by solver** list, choose **Intermediate**.

This setting forces the solver to take at least one step in each of the time intervals you specified.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

The default plot shows the velocity magnitude as a surface plot. Add a streamline plot of the velocity field by following the steps below.

Streamline 1

1 Right-click **Velocity (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Minimum distance** text field, type `0.01`.

5 In the **Maximum distance** text field, type `0.04`.

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

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Visualize the approach to this steady state by creating the series of snapshots of the velocity field (Figure 2).First, fix the color range for the surface plot to get the same color-to-velocity mapping for all time steps.

Surface

1. In the **Model Builder** window, click **Surface**.
2. In the **Settings** window for **Surface**, click to expand the **Range** section.
3. Select the **Manual color range** check box.
4. Locate the **Coloring and Style** section. Clear the **Color legend** check box.
5. In the **Velocity (spf)** toolbar, click **Plot**.

Velocity (spf)

1. In the **Model Builder** window, click **Velocity (spf)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Time (s)** list, choose **0.025**.
4. In the **Velocity (spf)** toolbar, click **Plot**.

To reproduce the remaining plots, plot the solution for the time values 0.05 s, 0.075 s, 0.1 s, 0.15 s, 0.2 s, 0.3 s, 0.5 s, and 0.75 s.

Finally, generate a movie.

Animation 1

1. In the **Velocity (spf)** toolbar, click **Animation** and choose **Player**.

   COMSOL Multiphysics generates a movie and then plays it. To replay the movie, click the **Play** button in the **Graphics** toolbar.

   If you want to export a movie in GIF, Flash, or AVI format, right-click **Export** and create an Animation feature.

Next, visualize the grain’s downward velocity as a function of time.

1D Plot Group 4

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.
3. From the **Title type** list, choose **None**.
4. Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Time (s)`.
6 Select the **y-axis label** check box. In the associated text field, type `Grain speed (m/s)`.

Point Graph 1

1 Right-click **1D Plot Group 4** and choose **Point Graph**.
2 Select Point 1 only.
3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `-Xdot`.
5 In the **1D Plot Group 4** toolbar, click **Plot**.

To view all the forces in the same figure, follow the steps given below.

1D Plot Group 5

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.
3 From the **Title type** list, choose **None**.
4 Locate the **Plot Settings** section.
5 Select the **x-axis label** check box. In the associated text field, type `Time (s)`.
6 Select the **y-axis label** check box. In the associated text field, type F_z, F_g, and F_g+F_z (N).
   The HTML tags 'sub' and 'sup' give subscripts and superscripts, respectively.

Point Graph 1

1 Right-click **1D Plot Group 5** and choose **Point Graph**.
2 Select Point 1 only.
3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>F_z - Drag force - N**.
4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Blue**.
5 Click to expand the **Legends** section. Select the **Show legends** check box.
6 From the **Legends** list, choose **Manual**.
7 In the table, enter the following settings:

Legends
F_z

8 In the **1D Plot Group 5** toolbar, click **Plot**.

Point Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `F_g`.

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

6 Locate the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

Legends
F_g

9 In the **1D Plot Group 5** toolbar, click **Plot**.

Point Graph 3

1 Right-click **1D Plot Group 5** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `F_z+F_g`.

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

6 Locate the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

Legends
F_z+F_g

9 In the **1D Plot Group 5** toolbar, click **Plot**.