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 model 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

\[
\rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \mathbf{u} + \nabla \cdot (\nabla \mathbf{u}) + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p = \mathbf{F}
\]

\[
\nabla \cdot \mathbf{u} = 0
\]

where \( \rho \) denotes the density (kg/m\(^3\)), \( \mathbf{u} \) the velocity (m/s), \( \eta \) the viscosity (Ns/m\(^2\)), and \( p \) the pressure (Pa). The fluid is water with a viscosity of \( 1.51 \times 10^{-3} \) 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:

\[
F_r = 0, \quad F_z = -\rho (a + g)
\]

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:

\[
F_g = -\rho_{\text{grain}} V_{\text{grain}} g
\]

where \( V_{\text{grain}} \) (m\(^3\)) is the volume of the sand grain and \( \rho_{\text{grain}} \) (kg/m\(^3\)) its density. The force that the water exerts on the grain is calculated by integrating the normal component of the stress tensor over the surface of the particle. Because the model is axially symmetric, only the force’s z-component is nonzero:

\[
F_z = 2\pi \int_S \mathbf{n} \cdot [-p I + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] dS
\]

where \( r \) (m) is the radial coordinate and \( \mathbf{n} \) is the normal vector on the surface of the grain.

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

**BOUNDARY CONDITIONS**

At the surface of the sphere, the velocity relative the sphere is zero. Therefore, the model uses a no-slip condition, \( \mathbf{u} = \mathbf{0} \). At the inlet of the fluid domain the velocity equals the falling velocity: \( \mathbf{u} = (0, \dot{x}) \). Symmetry, \( \mathbf{n} \cdot \mathbf{u} = 0 \), applies at the outer
boundary of the water domain, and a *neutral* condition, 
\[ \mathbf{n} \cdot [-pI + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] = 0, \]
describes the outlet. An axial symmetry condition models the symmetry axis at \( r = 0. \)

**Results**

*Figure 1* shows the velocity field at the final simulation time, \( t = 1 \text{ s} \), 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 0.291 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 (upper line) increases as the grain gains speed. The gravity force (lower line) remains the same, and the total force (middle 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:

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

where \(d\) (m) is the diameter of the sphere, \(\rho\) (kg/m\(^3\)) the fluid density, \(v\) (m/s) the velocity, and \(\text{Re} = (\rho v d)/\mu\) is the Reynolds number, with \(\mu\) (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 \(\rho_s\) (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 non-spherical 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


Model Library path: COMSOL_Multiphysics/Fluid_Dynamics/falling_sand

Modeling Instructions

MODEL WIZARD
1. Go to the Model Wizard window.
2. Click the 2D axisymmetric button.
3. Click Next.
5. Click Next.
6. In the Studies tree, select Preset Studies>Time Dependent.
7. Click Finish.
GLOBAL DEFINITIONS
A set of global parameters are provided in a text file.

Parameters
1 In the Model Builder window, right-click Global Definitions and choose Parameters.
2 Go to the Settings window for Parameters.
3 Locate the Parameters section. Click Load from File.
4 Browse to the model’s Model Library folder and double-click the file falling_sand_parameters.txt.

GEOMETRY 1

Rectangle 1
1 In the Model Builder window, right-click Model 1>Geometry 1 and choose Rectangle.
2 Go to the Settings window for Rectangle.
3 Locate the Size section. In the Width edit field, type 6e-3.
4 In the Height edit field, type 14e-3.
5 Locate the Position section. In the z edit field, type -6e-3.
6 Click the Build Selected button.

Circle 1
1 In the Model Builder window, right-click Geometry 1 and choose Circle.
2 Go to the Settings window for Circle.
3 Locate the Size and Shape section. In the Radius edit field, type 1e-3.
4 Click the Build Selected button.

Difference 1
1 In the Model Builder window, right-click Geometry 1 and choose Boolean Operations>Difference.
2 Go to the Settings window for Difference.
3 Locate the Difference section. Under Objects_to_add, click Activate Selection.
4 Select the object r1 only.
5 Under Objects_to_subtract, click Activate Selection.
6 Select the object c1 only.
7 Click the Build Selected button.
This completes the model geometry.
Define an integration coupling operator and a variable using this operator. Later, you will use this variable to calculate the drag force.

**DEFINITIONS**

**Integration 1**

1. In the *Model Builder* window, right-click *Model 1*>
   *Definitions* and choose *Model Couplings*>
   *Integration*.
2. Go to the *Settings* window for Integration.
3. Locate the *Source Selection* section. From the *Geometric entity level* list, select *Boundary*.
4. Select Boundaries 6 and 7 only.
5. Locate the *Advanced* section. From the *Method* list, select *Summation over nodes*.

**Variables 1**

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

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_z$</td>
<td>intop1(-reacf(w))</td>
<td>Drag force</td>
</tr>
</tbody>
</table>

Here, $w$ is the $z$-component of the fluid velocity and $\text{reacf()}$ is the reaction force operator.

**LAMINAR FLOW**

1. In the *Model Builder* window, click *Model 1*>
   *Laminar Flow*.
2. Go to the *Settings* window for Laminar Flow.
3. Locate the *Physical Model* section. From the *Compressibility* list, select *Incompressible flow*.

**Fluid Properties 1**

1. In the *Model Builder* window, expand the *Laminar Flow* node, then click *Fluid Properties 1*.
2. Go to the *Settings* window for Fluid Properties.
3. Locate the *Fluid Properties* section. From the $\rho$ list, select *User defined*. In the associated edit field, type $\text{rho_water}$.
4. From the $\mu$ list, select *User defined*. In the associated edit field, type $\text{mu_water}$.
**Volume Force 1**

1. In the **Model Builder** window, right-click **Laminar Flow** and choose **Volume Force**.
2. Go to the **Settings** window for Volume Force.
3. Locate the **Domain Selection** section. From the **Selection** list, select **All domains**.
4. Locate the **Volume Force** section. Specify the $F$ vector as

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>r</td>
</tr>
<tr>
<td>-$\rho_{water}(X_dott+g_{const})$</td>
<td>z</td>
</tr>
</tbody>
</table>

Next, add a Global Equation node for the equation of motion for the falling grain. By default, this feature is not visible.

5. In the **Model Builder** window’s toolbar, click the **Show** button and select **Advanced Physics Interface Options** in the menu.

**Global Equations 1**

1. In the **Model Builder** window, right-click **Laminar Flow** and choose **Global>Global Equations**.

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

2. Go to the **Settings** window for Global Equations.
3. Locate the **Global Equations** section. In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>NAME</th>
<th>$F(U, U_T, U_{TT}, T)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>$X_t - X_{dot}$</td>
</tr>
<tr>
<td>$X_{dot}$</td>
<td>$X_{dott} - (F_z + F_{g})/m_{grain}$</td>
</tr>
</tbody>
</table>

**Inlet 1**

1. In the **Model Builder** window, right-click **Laminar Flow** and choose **Inlet**.
2. Select Boundary 2 only.
3. Go to the **Settings** window for Inlet.
4. Locate the **Velocity** section. In the $U_0$ edit field, type $-X_{dot}$.

**Open Boundary 1**

1. In the **Model Builder** window, right-click **Laminar Flow** and choose **Open Boundary**.
2. Select Boundary 4 only.

**Symmetry 1**

1. In the **Model Builder** window, right-click **Laminar Flow** and choose **Symmetry**.
2. Select Boundary 5 only.

**MESH 1**

*Free Triangular 1*

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

**Size 1**

1. In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
2. Go to the **Settings** window for Size.
3. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, select **Boundary**.
4. Select Boundary 3 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7. In the associated edit field, type \(1.5 \times 10^{-4}\).

**Size 2**

1. In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
2. Go to the **Settings** window for Size.
3. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, select **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. Select the **Maximum element size** check box.
7. In the associated edit field, type \(0.75 \times 10^{-4}\).

**Size**

1. In the **Model Builder** window, click **Size**.
2. Go to the **Settings** window for Size.
3. Locate the **Element Size** section. From the **Predefined** list, select **Finer**.
4. Locate the **Element Size Parameters** section. In the **Maximum element growth rate** edit field, type \(1.05\).
5. Click the **Build All** button.
**STUDY 1**

*Step 1: Time Dependent*

1. In the Model Builder window, expand the Study 1 node, then click **Step 1: Time Dependent**.
2. Go to the **Settings** window for Time Dependent.
3. Locate the **Study Settings** section. In the **Times** edit field, type `range(0, 0.025, 1)`.
4. In the Model Builder window, right-click **Study 1** and choose **Show Default Solver**.
5. Expand the **Study 1>Solver Configurations** node.

*Solver 1*

1. In the Model Builder window, expand the Study 1>Solver Configurations>Solver 1 node, then click **Time-Dependent Solver 1**.
2. Go to the **Settings** window for Time-Dependent Solver.
3. Click to expand the **Time Stepping** section.
4. From the **Steps taken by solver** list, select **Intermediate**.
   This setting forces the solver to take at least one step in each of the time intervals you specified.
5. In the Model Builder window, right-click **Study 1** and choose **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.

1. In the Model Builder window, right-click Results>Velocity (spf) and choose **Streamline**.
2. Go to the **Settings** window for Streamline.
3. Locate the **Streamline Positioning** section. From the **Positioning** list, select **Magnitude controlled**.
4. Locate the **Coloring and Style** section. From the **Color** list, select **Black**.
5. Click the **Plot** button.
6. Click the **Zoom Extents** button on 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.
1 In the **Model Builder** window, click **Surface 1**.

2 Go to the **Settings** window for Surface.

3 Click to expand the **Range** section.

4 Select the **Manual color range** check box.

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

6 Click the **Plot** button.

7 In the **Model Builder** window, click **Velocity (spf)**.

8 Go to the **Settings** window for 2D Plot Group.

9 Locate the **Data** section. From the **Time** list, select 0.025.

10 Click the **Plot** button.

11 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.

Click the **Player** button on the main toolbar.

**Export**

COMSOL Multiphysics generates a movie and then plays it. To replay the movie, click the Play button on the Graphics toolbar. If you want to export a movie in GIF, Flash, or AVI format, right-click Report and create an Animation feature.

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

**1D Plot Group 4**

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

2 Go to the **Settings** window for 1D Plot Group.

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

4 In the associated edit field, type **Time (s)**.

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

6 In the associated edit field, type **Grain speed (m/s)**.

7 Select the **Title** check box.

8 Clear the associated edit field.

9 Right-click **Results>1D Plot Group 4** and choose **Point Graph**.

10 Select Vertex 1 only.

11 Go to the **Settings** window for Point Graph.
12 Locate the y-Axis Data section. In the Expression edit field, type -Xdot.

13 Click the Plot button.

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

**1D Plot Group 5**

1. In the Model Builder window, right-click Results and choose 1D Plot Group.
2. Go to the Settings window for 1D Plot Group.
3. Locate the Plot Settings section. Select the x-axis label check box.
4. In the associated edit field, type Time (s).
5. Select the y-axis label check box.
6. In the associated edit field, type $F_z$, $F_g$, and $F_z + F_g$ (N).
   The HTML tags 'sub' and 'sup' give subscripts and superscripts, respectively.
7. Select the Title check box.
8. Clear the associated edit field.
9. Right-click Results>1D Plot Group 5 and choose Point Graph.
10. Select Vertex 1 only.
11. Go to the Settings window for Point Graph.
12. In the upper-right corner of the y-Axis Data section, click Replace Expression.
13. From the menu, choose Definitions>Drag force ($F_z$).
14. Click to expand the Coloring and Style section.
15. Find the Line style subsection. From the Color list, select Blue.
16. Click to expand the Legends section.
17. Select the Show legends check box.
18. From the Legends list, select Manual.
19. In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>LEGENDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_z$</td>
</tr>
</tbody>
</table>

20 Click the Plot button.

21 In the Model Builder window, right-click 1D Plot Group 5 and choose Point Graph.
22 Select Vertex 1 only.
23 Go to the Settings window for Point Graph.
24 Locate the y-Axis Data section. In the Expression edit field, type \( F_g \).

25 Locate the Coloring and Style section. Find the Line style subsection. From the Color list, select Red.

26 Locate the Legends section. Select the Show legends check box.

27 From the Legends list, select Manual.

28 In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>LEGENDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>( F_g )</td>
</tr>
</tbody>
</table>

29 Click the Plot button.

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

31 Select Vertex 1 only.

32 Go to the Settings window for Point Graph.

33 Locate the y-Axis Data section. In the Expression edit field, type \( F_z + F_g \).

34 Locate the Coloring and Style section. Find the Line style subsection. From the Color list, select Black.

35 Locate the Legends section. Select the Show legends check box.

36 From the Legends list, select Manual.

37 In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>LEGENDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>( F_z + F_g )</td>
</tr>
</tbody>
</table>

38 Click the Plot button.