Case Study

Terminal Velocity of a Falling Object using Ansys Fluent Software

Autumn Parker, Shanti Bhushan, and Es Sahli Omar
Mississippi State University

Edited by Gautham Varma from the Academic Development Team

education@ansys.com

Ansys Software Used

This resource uses Ansys Fluent®, a fluid simulation software.

Summary

In this study, we explore a classic fluid dynamics problem using the Falling Sphere Viscometer test case available on Ansys Innovation Space. The objective is to demonstrate how drag force acting on a body can be expressed as a function of the Reynolds number, a key non-dimensional parameter in fluid flow analysis. By deriving a functional relationship between drag and Reynolds number, we implement it within a numerical time integration scheme to predict the terminal velocity of a sphere falling through different fluids. This case effectively illustrates how computational fluid dynamics (CFD) can be leveraged to analyze and predict physical behavior in real-world engineering applications.

Table of Contents

1. Nomenclature

Re	Reynolds number (dimensionless)
V	Velocity of the object (m/s)
D	Diameter of the object (m)
ρ_f	Density of the fluid kgm^3
μ_f	Dynamic viscosity of the fluid (Pa·s)
C_d	Drag coefficient (dimensionless)
FD	Drag force (N)
A	Reference area of the object (m^2), πD2
g	Gravitational acceleration ms2
m_b	Mass of the object (kg)
V	Volume of the object (m^3), 16πD3
ρ_b	Density of the object kgm3
Dt	Time steps (s)
dVdt	Object acceleration ms2
e	Convergence criterion for terminal velocity ~ 10-6 ms2

2. Problem Statement

When an object is dropped into a fluid, it experiences various forces such as buoyancy, drag, and gravitational as shown in Figure 1. The buoyancy and gravitational forces depend on the object properties, i.e., volume and mass, respectively. But the drag force depends on the object velocity, shape and fluid viscosity. As the object moves through the fluid, the drag force changes dynamically resulting in a force equilibrium, where the acceleration is zero, causing the falling object to reach a terminal velocity. The key aspect that determines the terminal velocity of the object is to understand its drag characteristics. Computational Fluid Dynamics, CFD, can help in analyzing the correlation between drag and object velocity, that can help estimate the expected terminal velocity of the object.

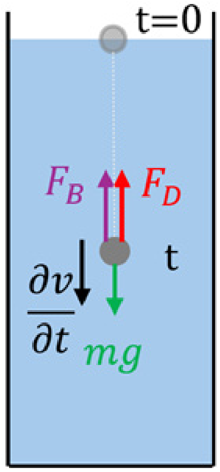

Figure 1: Forces acting on a sphere falling in a fluid.

3. Approach

The drag (D) characteristics of an object depends on its characteristic length (L), velocity (V), fluid density () and dynamic viscosity ()[1],

$$D = f(V, \rho, \mu, L) \tag{1}$$

By running a steady-state simulation in the Ansys Fluent software and assuming laminar flow, the total drag force can be calculated for a specified set of object, flow and fluid properties (V,ρ,μ,L). However, to make the predictions applicable for a wider range of flow and fluids, they can be cast into non-dimensional parameters using Buckingham Pi method. Note that above functional relation involves three (K) fundamental dimensions (mass, length and time) and five (N) physical variable (D,V,ρ,μ,L), we can obtain two (N-K) non-dimensional variables. Considering ρ,μ,L as repeating variables one can obtain:

$$C_D\left[=\frac{D}{\rho V^2 L^2}\right] = f\left(Re\left[=\frac{\rho V L}{\mu}\right]\right) \tag{2}$$

where, CD is the drag coefficient which is a function of a single parameter Reynolds number (Re). Herein, we will use the Ansys Innovation Simulation example – Falling Sphere Viscometer[2] will used for this project to obtain drag forces acting on a sphere for three different fluids at varying velocities. This will provide a wide range of CD and Re to obtain a functional form of the correlation. The following provides details of the CFD set-up.

4. Ansys Fluent Software Simulation Setup Overview

The simulations performed using were the Falling Sphere Viscometer test case available at Ansys Innovation website. The simulation is set-up using a pressure-based solver, configured with the absolute velocity formulation, runs in steady-state mode assuming laminar flow. The computational domain was designed around a sphere geometry with a diameter (D) of 0.02 meters, serving as the object for drag evaluation. The mesh consisted of approximately 436,730 polyhedral cells, providing a fine resolution to accurately capture flow details around the sphere (Fig. 2). The boundary conditions, as illustrated in Figure 3 (left panel), were set up with a velocity-inlet to specify the inflow speed and a pressure-outlet at the downstream boundary to allow fluid to exit the domain freely. This configuration enabled a realistic simulation of flow around the sphere under different fluid and velocity conditions.

1 Drag also depends on whether the flow is laminar or turbulent. For simplicity, flow is assumed to be laminar.

2 https://innovationspace.ansys.com/courses/courses/what-are-fluids/lessons/simulation-examples-homework-and-quizzes/topic/simulation-example-falling-sphere-viscometer/

In this study, the simulation set-up was modified to change the fluid types: Water, Engine Oil, and Glycerin, each with their respective physical properties, and the inflow velocity. This allowed drag predictions over a wide range of Reynolds numbers.

The numerical method of the simulation included Rhie-Chow interpolation method for the pressure-velocity coupling, with a distance-based flux type. This method is known to effectively reduce pressure-velocity decoupling issues, especially in low-speed and viscous flow regimes, which are characteristic of the current study. For spatial discretization, the following schemes were applied:

- Gradient Calculation: Least Squares Cell-Based method was selected to compute accurate gradients within the mesh.
- Pressure: Second Order discretization was used to capture pressure variations with higher accuracy.
- Momentum: Second Order Upwind scheme was adopted to enhance the resolution of velocity fields and improve solution stability in convection-dominated flows.

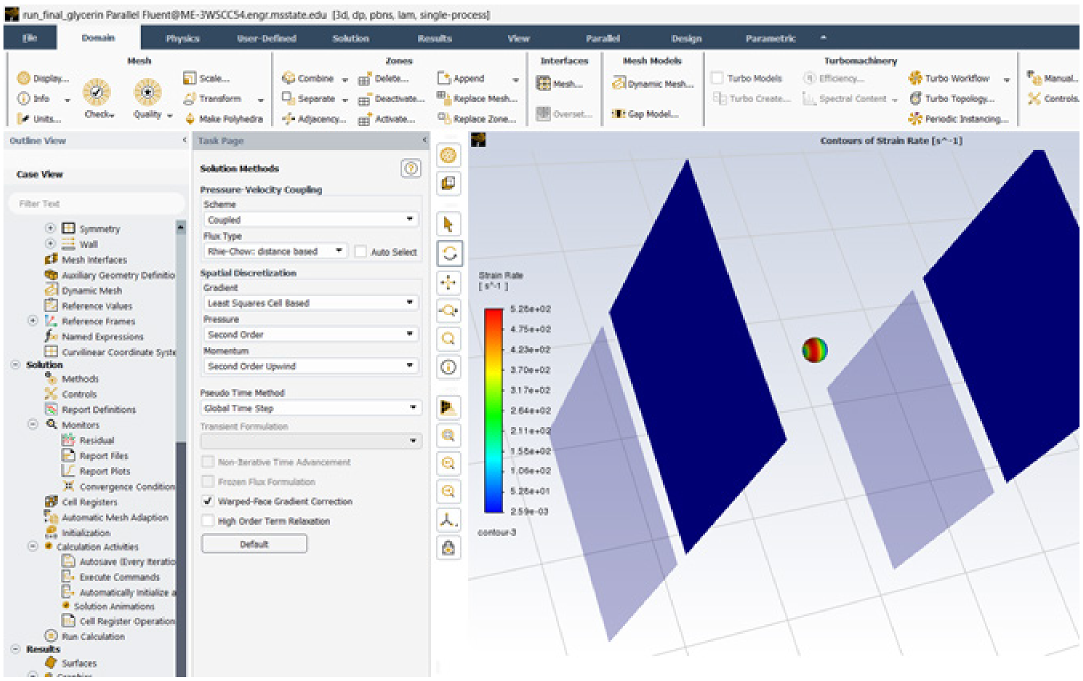

Figure 2: Ansys Fluent simulation Setup

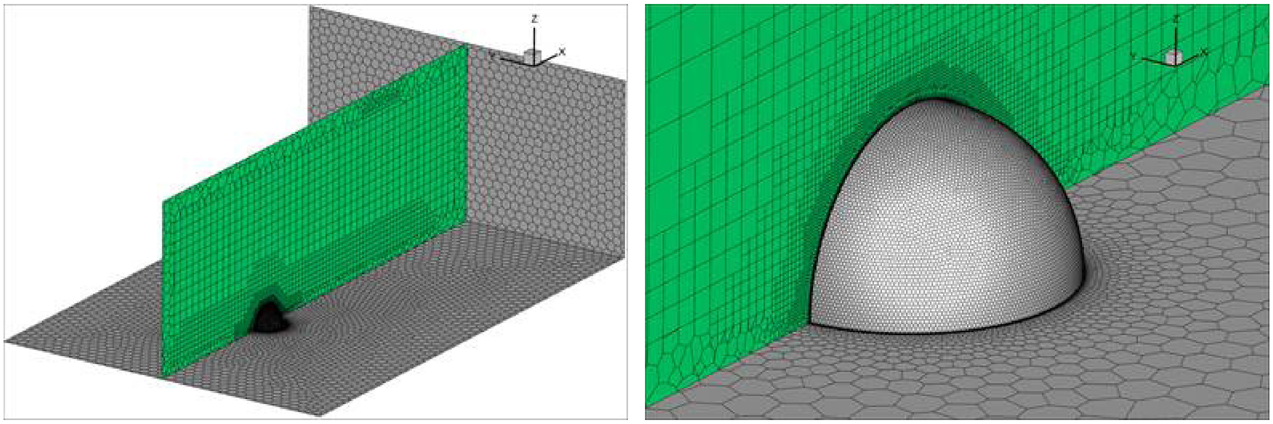

Figure 3: View of the mesh grid.

The pseudo time stepping approach was implemented with a global time step, allowing the simulation to evolve over time towards a steady-state terminal velocity condition. The automatic time step method was used to dynamically adjust the step size based on flow characteristics and residual behavior, improving convergence efficiency. To promote numerical stability and convergence, relaxation factors were set as follows:

- Pressure: 0.5
- Momentum: 0.5
- Density: 1.0
- Body Forces: 1.0

These values were chosen to balance solution accuracy with stability, particularly considering the varying viscosities of the test fluids.

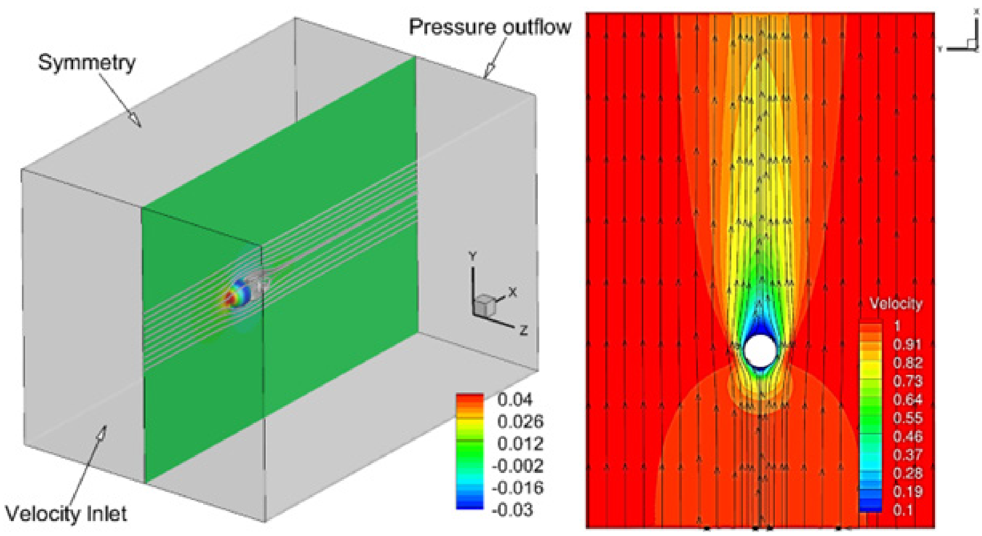

Figure 4: Simulation domain (left) and velocity contour of flow around a sphere (right).

5. Simulation Conditions

The simulations were conducted using a range of inflow velocities: U_0 = 2, 1, 0.5, 0.1, and 0.01 m/s. These values allowed us to observe how the drag behavior changes from relatively fast to very slow flow conditions. We also tested three different fluids, each with distinct viscosity and density values, to represent a wide range of flow behaviors:

- Water (H_2O)
  » Density: 998.2 kg/m³
  » Dynamic Viscosity: 0.001003 kg/(m·s)
  » A low-viscosity fluid, representing fast and less resistive flow.
- Engine Oil (E)
  » Density: 889 kg/m³
  » Dynamic Viscosity: 1.06 kg/(m·s)
  » A highly viscous fluid, providing strong resistance to motion.
- Glycerin (G)
  » Density: 1259.9 kg/m³
  » Dynamic Viscosity: 0.799 kg/(m·s)
  » A dense and viscous fluid, representing a middle ground between water and engine oil.

All three fluids were subjected to the same set of inflow velocities to ensure a fair and consistent comparison. This setup allowed us to clearly observe how viscosity and density affect the development of drag forces and the resulting terminal velocity in each case.

The simulation for each case was run for 100 iterations, sufficient to achieve convergence and obtain steady-state drag force values required for calculating terminal velocities. Simulation files are included with the case study document download.

CFD provides pressure and viscous forces that add up to equal drag force shown in Table 1. The velocity contour of a small sphere falling with a velocity of 2.0 ms in Engine Oil is shown in Figure 4.

6. Results and Analysis

The drag predictions (F_D) obtained for the different simulations were non-depersonalized into drag coefficient: $C_d = \frac{F_D}{\frac{1}{8}\rho_f V^2 \pi D^2}$ and Reynolds number: $Re = \frac{VD\rho_f}{\mu_f}$ following Eq. (2),

and results are summarized in Table 1.

Table 1: Simulation results for drag of sphere for different fluids.

Material	Density (kg/m^3)	Dynamic Velocity (kg/m-s)	Velocity (m/s)	FD (N)	Cd	Re
Water	998.2	0.001003	2	1.54E-01	6.12E-02	39808.5743
			1	4.11E-02	6.55E-02	19904.2871
			0.5	9.29E-03	5.92E-02	9952.1436
			0.1	3.65E-04	5.81E-02	1990.4287
			0.01	6.13E-06	9.77E-02	199.0429
Engine Oil	889	1.06	2	5.58E-01	2.50E-01	33.54716981
			1	2.15E-01	3.85E-01	16.77358491
			0.5	8.64E-02	6.18E-01	8.38679245
			0.1	1.26E-02	2.26E+00	1.67735849
			0.01	1.16E-03	2.08E+01	0.16773585
Glycerin	1259.9	0.799	2	5.53E-01	1.75E-01	63.0738423
			1	2.05E-01	2.59E-01	31.53692115
			0.5	7.92E-02	4.00E-01	15.76846058
			0.1	1.04E-02	1.31E+00	3.153692115
			0.01	8.69E-04	1.10E+01	0.315369212

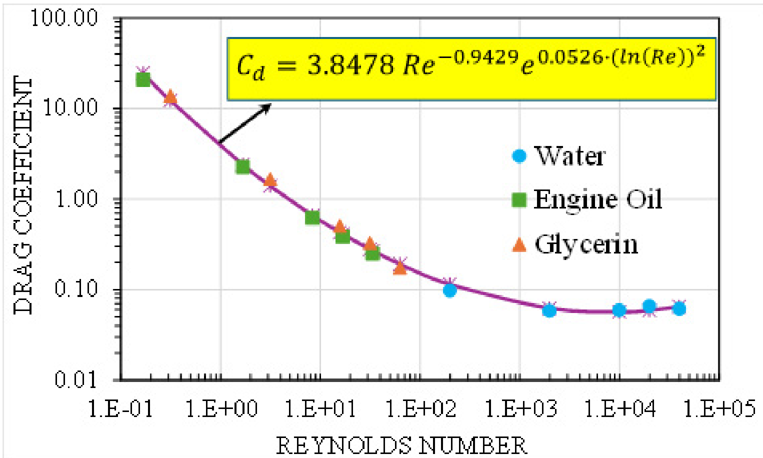

Figure 5: Prediction of drag coefficient vs. Reynolds number for different simulations.

Figure 5 illustrates this relationship, showing how the drag coefficient varies as a function of Reynolds number for all three fluids. Interestingly, despite the differences in viscosity and density among the fluids, all data points align along the same curve. This confirms that the drag coefficient is governed solely by the Reynolds number, and not directly by the individual physical properties of the fluids, as evaluated using Buckingham Pi method.

MATLAB poly-fit function was used to obtain a second-order polynomial to obtain the functional relationship between C_d and Re as below:

$$C_d = 3.8478\, Re^{-0.9429} e^{0.0526 \cdot (\ln(Re))^2} \quad (3)$$

From the fitted line equation, a time-marching solver shown below in Fig. 6 was developed using Microsoft Excel to calculate total force acting on falling sphere and its local velocity and acceleration. The sphere was assumed to have a diameter on 0.2 m with a density of $\rho_b = 1400\ kg/m^3$ resulting in a mass $m_b = 5.8643\ kg$.

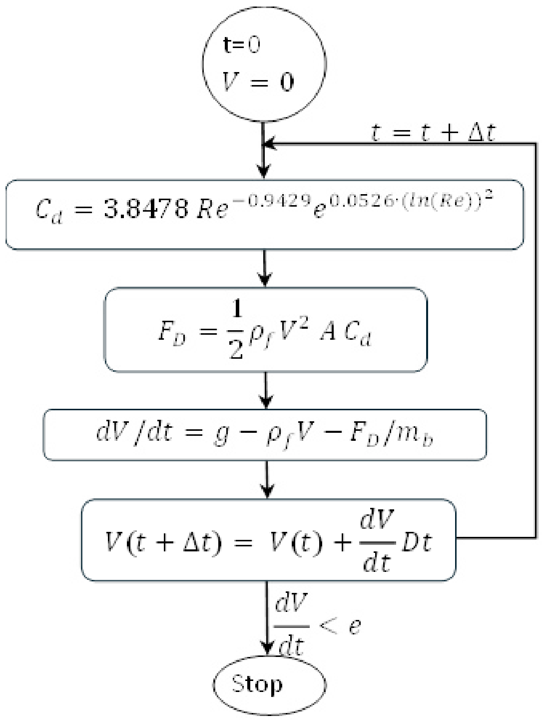

Figure 6: Time-marching process to evaluate velocity of a falling sphere.

The time marching was started at time = 0 with a small velocity of approximately V = 0.01 ms. The total force on the sphere was calculated considering the sphere weight, buoyancy force and flow drag estimated using Eq. 3. Once the acceleration is approximately zero, the sphere reaches terminal velocity.

The terminal velocity of the sphere varies depending on the fluid's properties. The higher the viscosity, the lower the terminal velocity. Note that engine oil has the highest viscosity (refer Table 1) whereas water has the lowest viscosity out of the three fluids tested. Figure 7 below shows that in water the sphere will reach a much higher terminal velocity of 0.21 ms, whereas in the engine oil the sphere will stop accelerating at a much lower velocity, 0.02 ms due to the fluid's thickness. In glycerin the sphere reached a terminal velocity of 0.1 ms.

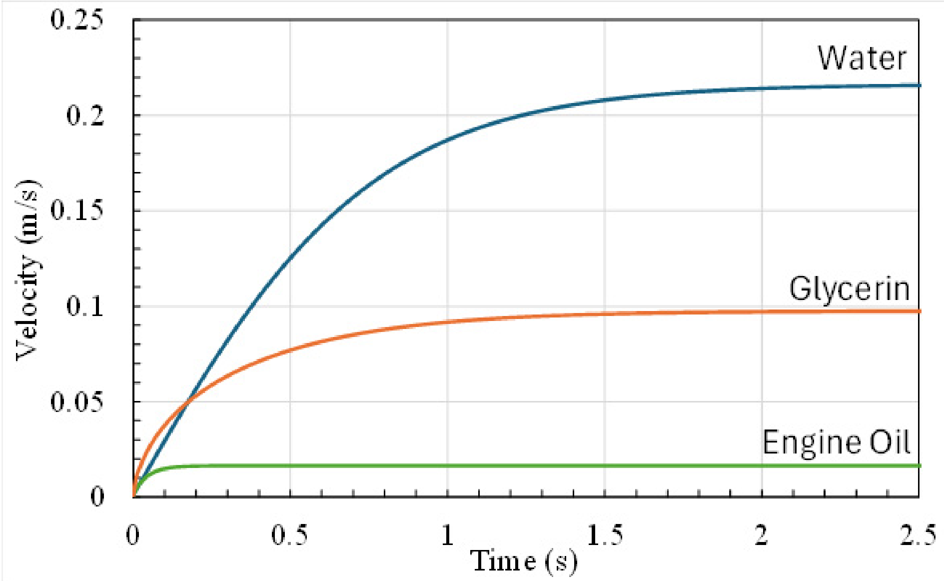

Figure 7: Terminal velocity of various fluids.

7. Conclusions

In this case study, Ansys Fluent software was employed to determine functional relationship between sphere drag coefficient and flow Reynolds number, which was then used within a numerical time integrator to obtain terminal velocities of a sphere falling in various fluids, including water, glycerin, and engine oil. The analysis was based on a viscometer setup, where multiple simulations were conducted for each fluid to derive a second-order polynomial equation representing the drag coefficient. Among the fluids studied, engine oil exhibited the lowest terminal velocity at 0.02 m/s due to its high viscosity, while water, having the lowest viscosity, reached the highest terminal velocity of 0.21 m/s. Glycerin, with intermediate viscosity, had a terminal velocity of 0.1 m/s. Ansys Fluent software proved to be an invaluable tool for this project, offering accurate flow modeling capabilities and flexibility to handle complex fluid behavior across a range of viscosities.

Use and Reproduction
The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found using this link.

Document Information
This case study is part of a set of teaching resources to help introduce students to topics related to engineering, simulation, and materials.

Ansys Education Resources
To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback
Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit **www.ansys.com** for more information