Analyzing the Dynamics and Friction Forces on Spheres in Fluid Flows: Applications Across Engineering and Sports Science

Ming Zhang

School of Mechanical Engineering, University of Leeds, Leeds, The United Kingdom

mn21m3z@leeds.ac.uk

Abstract. This study explores the dynamics and friction forces of spherical objects in fluid flows, emphasizing their applications across engineering and sports science. By examining key theoretical concepts such as drag coefficient, Reynolds number, and fluid dynamics equations, the research highlights how these principles influence the motion of spheres in various fluid environments. The analysis encompasses computational fluid dynamics (CFD) and experimental approaches, addressing applications in ocean, aerospace, and vehicle engineering, as well as sports science. The findings demonstrate the critical role of understanding drag and friction characteristics in optimizing designs, improving performance, and driving technological innovation. Challenges such as multi-scale phenomena, nonlinear dynamics, and measurement techniques are identified, suggesting future research directions to refine theoretical models and enhance practical applications. The study not only advances the theoretical understanding of fluid dynamics but also provides essential insights for engineering optimization and sports performance enhancement.

Keywords: Fluid dynamics, Drag coefficient, Reynolds number, Computational Fluid Dynamics (CFD).

1. Introduction

Fluid mechanics plays an integral role in both engineering and sports science, providing insights into the interaction between fluid motion and objects within various environments. One particularly significant area of study within this field is the dynamics and friction forces experienced by spherical objects in fluid flows. These phenomena are vital due to their broad applications across different scenarios, ranging from optimizing the aerodynamic characteristics of aircraft, ships, and vehicles in engineering to enhancing the performance of athletes in sports science.

Recent advancements have focused on understanding the effects of skin friction on spheres, particularly in environments characterized by different Reynolds numbers. For instance, Cena's study utilized a novel experimental approach involving magnetic levitation to measure the rotational decay of spheres due to air resistance, providing insights into skin friction coefficients at low Reynolds numbers [1]. However, this research is limited to low Reynolds scenarios, and there is a need to extend these studies to higher Reynolds numbers to fully understand the sphere's behavior in various fluid dynamics conditions.

Research Content of This Paper: This paper aims to conduct a comprehensive analysis of the dynamics and friction of spherical objects within fluid flows, emphasizing the dual applications in both engineering optimization and sports performance enhancement. Specifically, the study will:

- Enhance engineering applications by designing more optimized shapes to reduce fluid drag, thereby improving the efficiency and performance of aerospace, automotive, and marine engineering designs.
- Provide sports science with deeper insights into how air resistance affects ball sports, offering athletes and coaches scientifically backed training methods and equipment designs.
- Refine and validate existing theoretical fluid dynamics models through a combination of experimental data and computational simulations, ensuring more accurate predictive tools for future research.

- Investigate new phenomena due to technological advances in materials and techniques, driving further innovation in fluid dynamics applications.

2. Relevant Theories

2.1. Drag Coefficient and Reynolds Number

Coefficient of drag (Cd): The coefficient of drag describes the amount of resistance an object experiences when moving through a fluid. It is related to the density of the fluid, the shape and size of the object, and the flow rate of the fluid. The drag coefficient can be determined experimentally or by numerical simulation and is usually related to the Reynolds number. In laminar flow (lower Re), the drag coefficient is usually higher, while in turbulent flow (higher Re), the drag coefficient may be lower.

Reynolds number (Re): The Reynolds number, a key metric in fluid dynamics, characterizes the nature of fluid flow, being calculated as the quotient of the fluid's inertial force over its viscous force. It is related to the velocity of the fluid, the characteristic length of the object (e.g. diameter) and the viscosity of the fluid. With lower Reynolds numbers, fluid flow is typically laminar, whereas at higher Reynolds numbers, it tends to become turbulent.

The aerodynamic characteristics of an object change at different Reynolds numbers. For example, at low Reynolds numbers, laminar separation is prevalent on the surface of the airfoil due to significant air viscosity effects, leading to deterioration of the aerodynamic characteristics, a sharp increase in drag coefficient and a sharp decrease in lift coefficient. With the escalation of the Reynolds number, the flow pattern might transition from an attached state to the emergence of a conventional elongated laminar separation bubble, and ultimately transform into a trailing edge laminar separation bubble, exhibiting substantial variations in the non-uniform flow structure [2].

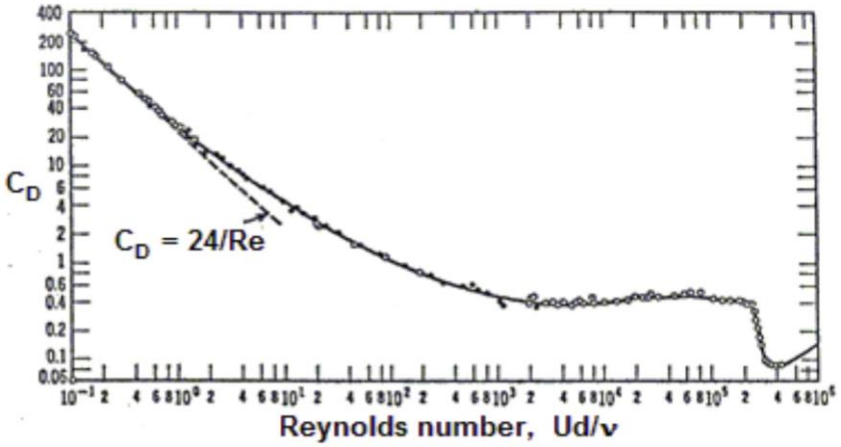

Fig. 1 Drag coefficient as a function of Reynolds number for sphere [3].

The correlation between the drag coefficient and the Reynolds number can be segmented into three phases: initially, the drag coefficient diminishes as the Reynolds number ascends to 5×10^4. Subsequently, the drag coefficient experiences a notable rise as the Reynolds number climbs to 1×10^5. Finally, the drag coefficient stabilizes and remains unchanged [3]. This relationship is also reflected in the figure 1.

2.2. Bernoulli's Theorem

Bernoulli's Theorem is a fundamental principle in fluid dynamics that describes the fact that in an ideal fluid (i.e., an incompressible, non-viscous fluid), the total energy of the fluid is conserved during the flow along the streamline. Bernoulli's theorem can be expressed as follows: In a flow line, the total energy of a fluid (including kinetic, potential and pressure energy) remains constant during the flow, provided that there is no work done by an external force, the fluid is flowing steadily and there is no heat exchange. The total energy can be expressed as:

$$E = \frac{1}{2}\rho v^2 + \rho g h + p \tag{1}$$

In which
E is total energy per unit volume of fluid.
ρ is density of the fluid.
v is velocity of the fluid.
g is gravitational acceleration.
h is height of the fluid relative to a reference point.
p is fluid pressure.

2.3. Newton's Viscosity Law

Newton's Viscosity Law is a fundamental law describing the viscous behavior of fluids and was introduced by Isaac Newton. The law states that under laminar flow conditions, shear stress in a fluid is directly proportional to its velocity gradient.

The mathematical expression for Newton's law of viscosity is:

$$\tau = \eta \frac{dv}{dx} \tag{2}$$

In which.
τ is Shear Stress
η is Dynamic Viscosity
$\frac{dv}{dx}$ is Velocity Gradient, which is the rate of change of fluid velocity in a certain direction (e.g., the x-direction).

Following Newton's law of viscosity, fluids are broadly categorized into two primary groups:

Newtonian Fluids: The viscosity of these fluids does not change with the change of shear stress, and their viscosity is constant, independent of the applied shear stress. For example, water, alcohol, mineral oil and petrol are Newtonian fluids.

Non-Newtonian Fluids: The viscosity of these fluids varies with shear stress, and their viscosity is not constant and is related to the applied shear stress. For example, toothpaste, ketchup and cosmetics.

2.4. Fluid Dynamics Equations

The fluid dynamics equations constitute a foundational set that encapsulates the behavior of fluid motion, encompassing principles of mass conservation, momentum conservation, and energy conservation. Commonly known as the Navier-Stokes equations, they serve as the cornerstone of fluid dynamics theory.

Continuity Equation: This formula represents the principle of mass conservation and is utilized to characterize the incompressible nature of a fluid. In the context of an incompressible fluid, the continuity equation is articulated as follows:

$$\nabla \cdot \vec{V} = 0 \tag{3}$$

In which.
$\vec{V}$ is the fluid velocity field.

Navier-Stokes Equations: These are expressions for the conservation of momentum, which are used to describe the relationship between the acceleration of a fluid particle and the force acting on the fluid. For a Newtonian fluid, the Navier-Stokes equations can be expressed as:

$$\rho\left(\frac{\partial \vec{V}}{\partial t} + \vec{V} \cdot \nabla \vec{V}\right) = -\nabla p + \mu \nabla^2 \vec{V} + \vec{F} \tag{4}$$

In which.
ρ is fluid density
$\vec{V}$ is Fluid Velocity Vector

p is fluid pressure
μ is dynamic viscosity of the fluid
$\vec{F}$ is a volumetric force, such as gravity
∇^2 is Laplace operator

Energy Equation: This formula encapsulates the principle of energy conservation and is used to describe changes in energy within a fluid. It is usually more complex than the Navier-Stokes equation because it involves heat transfer, viscous dissipation, and possible heat sources or sinks. It can be expressed as:

$$\rho c_p \left(\frac{\partial T}{\partial t} + \vec{V} \cdot \nabla T\right) = k\nabla^2 T + Q + \Phi \tag{5}$$

In which.
c_p is specific heat capacity of a fluid
T is temperature of the fluid
k is thermal conductivity of fluids
Q is heat source term per unit volume
Φ is viscous dissipation function

Equation of State: For an ideal gas, the state equation can be formulated as: $p = \rho RT$, where R is the gas constant. For other types of fluids, the equation of state may be more complicated.

To achieve a precise depiction of fluid flow, these equations typically must be resolved alongside boundary and initial conditions. Given the nonlinearity inherent in the Navier-Stokes equations, numerical techniques are frequently employed for their resolution, including finite difference methods, finite element methods, or finite volume methods.

2.5. Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a field that employs numerical techniques and computational technology to analyze and resolve issues related to fluid flow. The essence of CFD is the numerical resolution of the fundamental fluid dynamics equations, which commonly encompass the equations for mass conservation (continuity equations), momentum conservation (Navier-Stokes equations), and energy conservation.

The main numerical solution methods for CFD include Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM) and Finite Analysis Method (FAM). These methods solve approximate solutions to the fluid control equations by discretizing a continuous physical field into a finite number of nodes and establishing algebraic relationships between these points.

3. System Analysis and Application Research

3.1. Ocean Engineering

In ocean engineering, fluid dynamics properties are critical for designing and optimizing marine structures. Sphere modeling can help us understand the interactions between fluids and structures, including buoyancy, drag, lift and stability. By analyzing the motion of a sphere in a fluid, we can predict and simulate the behavior of marine structures in real marine environments.

For example, in order to optimize the performance of the submarine, a sphere can be used as an analogy to analyze the various factors that cause fluids to affect a submarine or ship. In Ahmad Amin's article, there is research about falling behavior of large spheres in small particle suspensions at finite Reynolds number, with special reference to drag forces and average sphere surface stresses. In the research, a fluidized bed with a rectangular cross-section is used, which is filled with a mixture of water and particles. Water is injected through a porous medium at the bottom to create a uniform flow and a screen filter prevents particles from passing through. There is a sphere is suspended in the fluidized bed by means of nylon wires, and a high-speed video camera is used to record the descent

of the spheres to measure the terminal velocity of large spheres of different diameters in suspensions of different concentrations. It finds that the average surface stress during the fall of the sphere is independent of the sphere size. And the drag coefficient of a falling sphere is inversely proportional to the Reynolds number and is related to the Reynolds number of the particles, not the Reynolds number of the sphere. Two possible explanations are proposed: either the suspension may be inhomogeneous on large scales, or there may be particle slip velocities at the surface of the sphere [4].

3.2. Aerospace

In the field of aeronautical engineering, the application of fluid dynamics is crucial, which involves many aspects of vehicle design, propulsion systems, and aerodynamic performance optimization.

Aerodynamics is central to aeronautical engineering, which studies the mechanical effects on a vehicle as it moves through the air. Through Computational Fluid Dynamics (CFD) methods, engineers can simulate the aerodynamic performance of a vehicle under different flight conditions to reduce drag and increase lift.

In Denham's article, it be discussed that the conventional aircraft airworthiness certification process is predicated on the use of flight tests to ascertain if an aircraft's systems adhere to the essential criteria for airworthiness, ensuring flight safety and managing risk. As the flight testing's cost increases and Computational Fluid Dynamics becomes more accurate, there is a growing preference for analytical certification to take place of the flight testing. This paper describes a framework for predicting special case performance of existing certified aircraft with modifications. The system integrates current flight test data or detailed information from the original aircraft with simplified data from computational fluid dynamics (CFD) or wind tunnel experiments on both the original and altered setups, thereby generating a six-degree-of-freedom (6-DOF) flight dynamics model. Two approaches have been put forward to develop revised flight dynamics models for the altered aircraft configurations, leveraging insights from the original aircraft and making projections about the uncertainty in the model's form. Subsequently, the refined dynamics model and the assessed uncertainty are utilized to conduct probabilistic simulations, which take into account the effects of wind turbulence [5].

3.3. Vehicle Engineering

In the field of vehicle engineering, the application of fluid dynamics is crucial, and it involves a number of aspects such as the aerodynamic design of vehicles, the optimization of fluid power systems, and the performance of vehicles in fluid environments [6].

Vehicles encounter air resistance while traveling, which is caused by air friction against vehicle surfaces and the compression of air in front of the vehicle and the expansion of air at the rear. Air resistance can be significantly reduced by optimizing the shape and structure of the vehicle, such as designing a more streamlined body, a smooth underbody and effective aerodynamic accessories such as front spoilers, side skirts and rear diffusers [7]. Reducing air resistance not only increases a vehicle's top speed, it also improves fuel efficiency. Because a vehicle's engine requires more power to overcome air resistance, less air resistance means the engine can work more efficiently, resulting in less fuel consumption. Aerodynamic design is not only concerned with reducing drag, but also with utilizing air flow to improve vehicle stability. For example, by designing a suitable spoiler at the rear of the vehicle, additional downforce can be generated to help keep the vehicle in contact with the road at high speeds and improve handling stability [8]. Wind noise generated when a vehicle is in motion is a critical determinant of ride comfort. By optimizing the aerodynamic design of the vehicle, for example by using specific lines and shapes on the sides and top of the vehicle, vortices and noise generated by air flow can be reduced. Computational Fluid Dynamics (CFD) is a powerful tool for simulating and analyzing fluid flow and related physical phenomena. In vehicle engineering, CFD can be used to simulate the air flow around a vehicle and predict the air resistance, lift and lateral

forces on the vehicle. With CFD simulation, engineers can save time and costs by iterating and optimizing vehicle designs without building a physical model.

3.4. Sports Science

In the field of sports science, the performance of ball sports is significantly affected by fluid dynamics and friction. Taking ball sports as an example, systematic analysis and applied research on the dynamics and friction of a sphere in fluid flow can help us to understand the kinematic properties of a sphere and optimize the performance and technique of athletes [9].

Like the research in Obligado's article, the experiments were carried out in the LEGI Laboratory's 'Lespinard' large wind tunnel, which is a closed-loop type with a $0.75\,m \times 0.75\,m \times 4\,m$ measurement section. A passive grille was used to generate a background flow field with about 3% turbulence intensity. Experiments were performed at three different mean flow velocities, $U_{z1} = 7.33\,m/s$, $U_{z2} = 9.89\,m/s$ and $U_{z3} = 12.44\,m/s$. A $2.08\,m$ long polyamide-nylon filament was used to tow the particles, which experienced restricted flow along the z-axis while being unrestricted in movement across the x- and y-axes. A high-speed video camera was used to record the trajectories of the particles in the xy plane at a frame rate of $3000\,fps$. For every particle category and each unobstructed flow velocity, a standard procedure involved capturing approximately 40 video recordings, totaling more than 1.3×10^3 carrier-flow integration timescales. Analysis of the particle trajectories was accomplished by spectral analysis to remove high-frequency noise and to generate Probability Density Functions (PDFs) of velocity and acceleration. It finds that the PDFs of particle velocity and acceleration show Gaussian distributions, which are different from the typical non-Gaussian acceleration statistics of free-moving particles. The fluctuations in velocity and acceleration decrease with increasing particle inertia, which is consistent with the predictions of low-pass filtering through particle inertia. The interaction between particles and turbulence is measured by an adjusted Stokes number (St), which reflects patterns associated with the particle's viscous response time (τ_p) and the velocity of the unobstructed flow (U_z). The behavior of the particles is characterized using a second-order bandpass filter, the transfer function of which is connected to the particle's viscous response time and the velocity of the free stream. The main features of the particle dynamics can be predicted by simplifying the model, which assumes that the coupling of the particle to the turbulence is mainly driven by drag. Experimental observations agree qualitatively with the model and also quantitatively after adjusting the particle response time. The dynamic characteristics of the particles are related to their inertia and traction configuration in the turbulence, independently of the traction line tension. These results contribute to a better understanding of the motion of sphere in turbulence, especially the dynamic properties when considering particle inertia and traction configurations [10].

4. Challenges

4.1. Multi-scale Phenomena

In fluid dynamics, the scale of a sphere can vary from microscopic to macroscopic, and the behavior of the fluid at different scales may exhibit significant differences. For example, a sphere at the nanoscale may be affected by intermolecular forces, while a sphere at the macroscale may be dominated by inertial forces. The nature of the fluid and the dynamical behavior of the sphere may change nonlinearly during the transition from the microscale to the macroscale, requiring the researcher to be able to understand and model this scale transition. Development of computational models capable of handling both micro and macro scales to accurately predict the dynamical behavior of spheres at different scales.

To address this challenge, it can use the multiscale computational methods, such as the coupling of molecular dynamics and continuum mechanics, are used to simulate the hydrodynamic behavior of spheres at different scales. The physical mechanisms of fluid-sphere interaction at different scales are explored through experimental and theoretical analysis.

4.2. Nonlinear Dynamics

The motion of a sphere in a fluid may exhibit complex nonlinear dynamical behaviors such as chaos, periodic variations, and bifurcations. In engineering and sports science, the stability of a sphere is a key factor, and the complexity of nonlinear dynamics makes stability analysis difficult. The development of mathematical models capable of accurately predicting the nonlinear behavior of a sphere in a fluid is essential for practical applications.

To address these challenges, these approaches can be used that Employ nonlinear kinetic theories, such as chaos theory, to analyze the complex motion of a sphere in a fluid. Predictive models based on machine learning and data mining are developed to improve the accuracy of predicting nonlinear dynamical behavior.

4.3. Measurement Techniques

Measuring the kinetic parameters of a sphere in a fluid, such as velocity, acceleration and force, requires highly accurate measurement techniques. Real-time monitoring of the dynamics of a sphere in a fluid is critical to understanding its dynamic behavior, but real-time measurement techniques are often limited by equipment and environmental conditions. Noise and disturbances in the fluid environment may affect the accuracy of measurements, and it is a challenge to minimize these disturbances.

In order to meet these challenges, there are some methods like utilizing advanced sensor technology and signal processing to improve measurement accuracy and reliability. Develop measurement equipment that can adapt to complex fluid environments, such as sensors and data acquisition systems with strong anti-interference capability. Adopt multi-sensor fusion technology to improve the accuracy and robustness of measurement results through data fusion.

5. Conclusion

This paper has contributed significantly to the field of fluid dynamics by analyzing the dynamics and friction forces exerted on spherical objects within fluid flows, with broad applications across engineering and sports science. The research has systematically explored the complex interactions between spherical objects and fluid environments through a combination of theoretical studies and practical experiments, involving computational fluid dynamics (CFD) and physical experimental methods.

The findings have demonstrated the pivotal role of fluid dynamics in understanding and predicting the behavior of spherical objects under various flow conditions. By examining key parameters such as drag coefficient and Reynolds number, this study has provided deeper insights into the aerodynamic performance of spheres, which is crucial for optimizing designs in aerospace, oceanic, and automotive engineering. Additionally, the research has significant implications for sports science, offering enhanced methodologies for athletes to improve performance through a better understanding of how air resistance and friction affect ball dynamics. Looking ahead, several areas of research are poised to further advance our understanding and application of fluid dynamics in relation to spherical objects.

References

[1] Cena C., Guizado L. F. S., Ferreira J. V. B. Determination of skin friction on a rotating sphere in magnetic levitation. Revista Brasileira de Ensino de Física, 2024, 46(Suppl. 1): e20230351. https://doi.org/10.1590/1806-9126-RBEF-2023-0351.

[2] Liu Q., Liu Q., Bai P., Li F. Aerodynamic characteristics of airfoil and evolution of laminar separation at different Reynolds numbers. Acta Aeronautica et Astronautica Sinica, 2017, 38 (4): 120338. https://doi.org/10.7527/S1000-6893.2016.0257.

[3] Zhang Y., Xu H., Zhu X. Detection and quantization technique of optical distributed acoustic coupling based on φ-OTDR. J. Shanghai Jiaotong Univ. (Sci.), 2020, 25: 208–213.

[4] Zeno A., Darwish M., Alee A., Awad A. An experimental study of drag coefficient for smooth thin circular plates with central hole. International Journal of GEOMATE, 2021, 20 (77): 149-154. https://doi.org/10.21660/2020.77.76739.

[5] Zhu X., Zhang Y., Zhao Z., Zuo J. Radio frequency sensing based environmental monitoring technology. In Fourth International Workshop on Pattern Recognition, 2019, Vol. 11198: 187-191. SPIE.

[6] Amin A., Girolami L., Risso F. Fall of a large sphere in a suspension of small fluidized particles. Physical Review Fluids, 2022, 7(8): L082301. https://doi.org/10.1103/ PhysRevFluids.7.L082301.

[7] Zhang Y., Zhao H., Zhu X., Zhao Z., Zuo J. Strain measurement quantization technology based on DAS system. In 2019 IEEE 3rd Advanced Information Management, Communicates, Electronic and Automation Control Conference (IMCEC), 2019, 214-218. IEEE.

[8] Denham C. L., Patil M., Roy C. J., Alexandrov N. Framework for estimating performance and associated uncertainty for modified aircraft configurations. Aerospace, 2022, 9: 490. https://doi.org/10.3390/ aerospace9090490.

[9] Obligado M., Bourgoin M. Dynamics of towed particles in a turbulent flow. Journal of Fluids and Structures, 2022, 114: 103704. https://doi.org/10.1016/j.jfluidstructs.2022.103704.

[10] Ge W., Wang L., Xu J., Chen F., Zhou G., Lu L., Li J. Discrete simulation of granular and particle-fluid flows: from fundamental study to engineering application. Reviews in Chemical Engineering, 2017, 33 (6): 551-623.