# Experimental Investigation of Levitation Dynamics in Turbulent Airflow: A Study of Drag Force and Bernoulli’s Principle for Aerodynamic Application

Yuze Song

School of Mechanical and Automotive Engineering Shanghai University of Engineering Science Shanghai, China

13918516498@163.com

**Abstract.** This study is based on the study of Bernoulli's principle and repeated experiments to verify and summarize its principle. And speculate on its feasibility analysis in practical life. The main purpose of this study is to verify the correctness and feasibility of Bernoulli's principle. The research method of this study is self-experimentation and theoretical verification. This experiment confirms the correctness of Bernoulli's principle and its feasibility in practical life. The conclusion of this experiment is relatively simple and straightforward, as the experiment is not complicated and has certain limitations, without multiple control groups for comparative studies. The significance of this article lies in the verification of Bernoulli's principle, as well as the future prospects and practical applications of Bernoulli's principle after verification. The study also explores the potential applications of Bernoulli's principle in various fields, such as fluid mechanics, aerodynamics, and engineering design. By analyzing the experimental data, the research aims to provide insights into how Bernoulli's principle can be utilized to improve efficiency and performance in these areas. The article concludes on the importance of empirical research in validating theoretical principles and the value of continuous exploration in scientific discovery.

**Keywords:** Bernoulli principle, hydrodynamics, aircraft.

## 1. Introduction

Fluid mechanics is now widely used in military opera actions, with unmanned aerial vehicle systems being a classic example of fluid mechanics [1-2]. The power generated by the rotation of its wings is the practical result of Bernoulli's principle in fluid mechanics. The "HALO" triaxial-rotor UAV, a cutting-edge design, embodies the application of Bernoulli's principle to achieve enhanced stability and maneuverability. By understanding the relationship between fluid velocity and pressure, engineers have been able to optimize the UAV's rotor design, resulting in a vehicle that can operate effectively in various atmospheric conditions. The mathematical expression for Reynolds number and the formula for drag force is fundamental to the analysis of UAV’s performance. These principles not only guide the design process but also provide a framework for testing and validation, ensuring that theoretical knowledge translates into practical advantages in the field [3-4].

Since the successful flight of the first drone in the early 20th century, drones have been a hot topic of research both domestically and internationally. Due to its small size, easy operation, low noise, and strong concealment, it plays an increasingly important role in many fields such as military warfare and people's livelihood economy. In recent years, with the maturity of research technology and the reduction of research costs, drones are gradually moving from laboratories to the daily lives of ordinary people. The research on the aerodynamic performance of drones and airfoils provides a theoretical basis for improving and designing aircraft structures and creating drones that meet more practical requirements [5-6].

As technology advances, the application scenarios of drones are expanding. They are not only used for aerial photography and surveillance but also for precision agriculture, environmental monitoring, and emergency response. The versatility of drones has sparked the interest of both industry and academia, leading to a surge in investment and innovation in this field. Moreover, the development of autonomous flight capabilities and artificial intelligence integration has opened new possibilities for drones to operate in complex environments without human intervention.

The author used CFD numerical simulation calculation method to analyze the simplified aerodynamic performance models of two-dimensional airfoils and aircraft in aviation [1]. This article analyzes the variation of lift and drag coefficients of classic aerospace NACA airfoils at different flow velocities and angles of attack [7-8].

In addition, this article also calculated two different sets of geometric parameters of the airfoil and obtained the influence of the maximum relative curvature and maximum relative thickness of the airfoil within a certain range on the lift coefficient and drag coefficient [9-10]. For the simplified models of fixed wing unmanned aerial vehicles and quadcopter unmanned aerial vehicles, CFD numerical simulation calculation method and theoretical formula estimation method were used to analyze their respective takeoff mechanisms, and then compared with the parameters of the model itself to analyze the reasons for the errors [11-12].

In this study, simplified modeling was conducted on the fixed wing unmanned aerial vehicle in the laboratory, and CFD simulation calculation method was also used to obtain the opening coefficient and drag coefficient of the three-dimensional model of the wing at different inflow velocities. The principle of the fixed wing unmanned aerial vehicle needing runway takeoff was analyzed. And using the obtained data, the lift obtained by the fixed wing unmanned aerial vehicle during smooth forward flight was estimated, and compared with its flight weight to analyze the reasons for the error. The aerodynamic performance of the rotor blades of the quadcopter unmanned aerial vehicle in the laboratory was analyzed by combining slip theory and blade element theory. An approximate expression for the rotor pulling force was obtained, and the principle of vertical takeoff of the rotor aircraft was analyzed. And the obtained data was used to estimate the required rotational speed when the rotor tension and aircraft gravity were balanced, verifying that the brushless DC motor installed on the aircraft can meet the flight requirements.

In addition, this article is only a simple study that was conducted on the aerodynamic performance of airfoils and unmanned aerial vehicles, and many simplifications were made to the models of unmanned aerial vehicles, resulting in a small number of selected computational samples.

Simplified modeling of table tennis was conducted in the laboratory, and simulation calculation methods were used to obtain the resistance coefficient of the table tennis model at different inflow velocities. Analyzed the principle of table tennis ball suspension. And using the obtained data, the lift obtained by the table tennis ball during stable suspension was estimated and compared with the flight weight, and the reasons for the error were analyzed.

The problem with this article is that only a simple study was conducted on the aerodynamic performance of table tennis, and many simplifications were made to the blowing model, resulting in a small number of selected calculation samples.

## 2. Experimental model

In daily life, it is common to see many lightweight items that can float in the air or take off from the wind. For example, blowing ping pong balls with a hair dryer and taking off a plane. This sparked my thinking, so I went to study Bernoulli's principle and verified its feasibility and correctness. And discovered the application of aerodynamics based on Bernoulli's principle in practical life.

The experiment utilizes an infinitely large space filled with air as the control volume, within which a circular inlet of diameter $d$ is defined at a specific location on the bottom boundary of the control volume. The model is depicted in Fig. 1.

By measuring the inlet wind speed, inlet diameter, the distance of the sphere from the inlet, and the sphere's weight and diameter. Then use the publicly displayed calculation as shown as follows:

$$x = \frac{1}{\sqrt{\frac{1}{\frac{1}{2}\rho(5ud^2)\frac{\pi}{4}D^2C_D}}} \tag{1}$$

The experiment considers the scenario where a high-speed airflow emerges from a nozzle and enters a stagnant fluid, which typically results in turbulent flow characterized by the Reynolds number (Re). The Reynolds number is defined as:

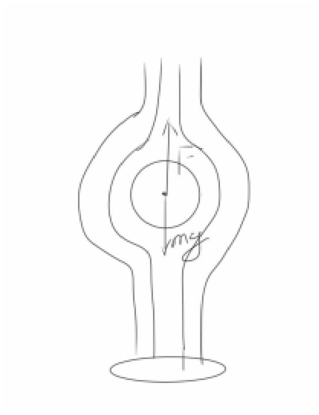


**Figure 1.** Experimental model

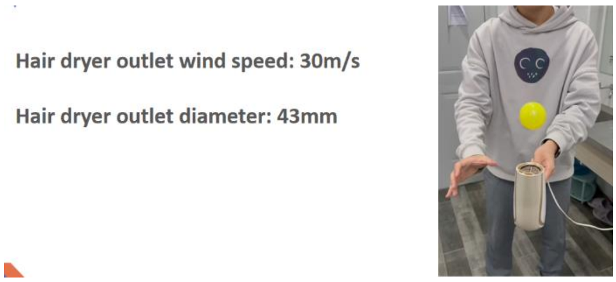


**Figure 2.** Experimental set

$$\mathrm{Re} = \rho U d / \mu \tag{2}$$

where $\rho$ is the fluid density, $U$ is the approach velocity, $d$ is the inlet diameter, and $\mu$ is the fluid viscosity. Assuming the fluid is atmospheric air at standard temperature and pressure, with the inlet, i.e., the hairdryer nozzle, having a diameter of 5 cm and an estimated wind speed of 15m/s, we roughly calculate the Reynolds number for this experimental model and find that $Re$ is significantly greater than 5000. Therefore, the jet in this model is turbulent, satisfying the assumption of turbulent flow.

The experiment mentions that the radius $R$ of the jet increases with the position $x$, and the jet expands at a constant angle of 11.8 degrees. The velocity profile of the jet is self-similar, with the maximum velocity at the centerline, and the velocity standard deviation increases with distance. In this experiment, to simplify the calculations, the relationship between the approach velocity and distance can be expressed as:

$$U(0,r) = Umax = 5Ud / x \tag{3}$$

where $U$ is the air velocity at the hairdryer nozzle exit, $d$ is the hairdryer outlet diameter, and $x$ is the distance along the streamline from the inside of the hairdryer vent to the object's head. And the fluid drag force can be described as follows:

$$FDrag = Cd\,\rho S v^2 / 2 \tag{4}$$

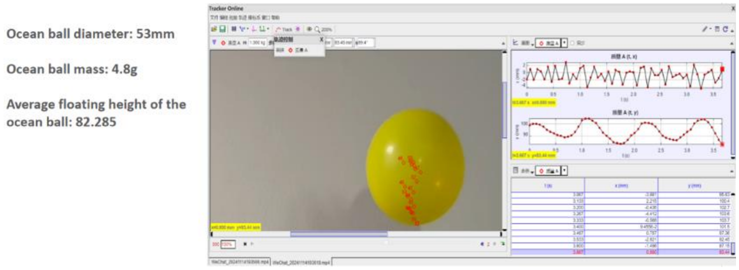


**Figure 3.** The diameter and weight data of the ocean ball measured in our experiment.

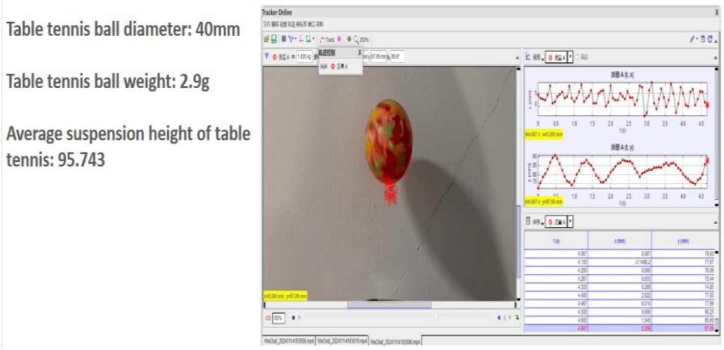


**Figure 4.** the diameter and weight data of the table tennis ball measured in our experiment.

here $FDrag$ is the drag force experienced by the sphere, directed opposite to the flow direction; $Cd$ is the drag coefficient, a dimensionless number that varies with the Reynolds number. $\rho$ is the fluid density; $S$ is the characteristic area through which the fluid flows, for a sphere this is its cross-sectional area, i.e., $S = \pi D^2/4$ ; $v$ is the velocity of the flow passing through the sphere.

## 3. Methodology

According to the above experimental model, when the air velocity of the hair dryer nozzle is large enough, the ball is suspended, and the product of its acceleration and mass is equal to the combined external force according to Newton's second law as follows:

$$F = ma \tag{5}$$

Them, substituting the expression of weight and Eq.5 into the equation,

$$FDrag - mg = 0 \tag{6}$$

Expand the resistance expression as follows:

$$FDrag = Cd\rho Sv^2/2 = mg \tag{7}$$

In addition to the equilibrium between the volume force of the research object and the differential pressure resistance caused by aerodynamics, it is also necessary to introduce the Reynolds number and the flow velocity calculation formulas of the center jet at different positions in the above-mentioned turbulence hypothetical model and jet velocity model in this experiment.

$$Re = \rho Ud/\mu \tag{8}$$

$$U(0, r) = Umax = 5Ud/x \tag{9}$$

In order to find the numerical outcome of the three equations above, we need to obtain the value of the parameters through experiments. We will first use ping-pang balls as the experiment objects, and later change the main characters of the lab, such as the weight, diameter of the ball, or the velocity of the jet. Besides, we will change the experiment objects into the one with a shape other than a sphere. What’s more, through independent thinking, we have conceived deep experimental models based on experimental phenomena and have tried to verify them.

## 4. Results

Fig. 3 shows the diameter and weight data of the ocean ball measured in our experiment, and the image on the right reflects the height suspension change of the center of the ocean ball under measurement. Finally, the average floating height of the ocean ball under the force of the blower shown in the previous data was obtained through multiple experiments.

Fig. 4 shows the diameter and weight data of the table tennis ball measured in our experiment, and the image on the right reflects the height suspension change of the center of the table tennis ball under measurement. Finally, multiple experiments were conducted to obtain the average floating height of the table tennis ball under the force of the hair dryer as shown in the previous data.

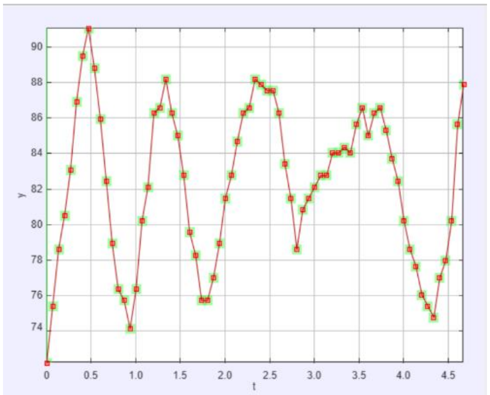


**Figure 5.** The data graph of table tennis ball suspension and compare it with the data graph of ocean ball suspension

Enlarge the data graph of table tennis ball suspension and compare it with the data graph of ocean ball suspension (Fig. 5). We can draw a conclusion that under the same external force conditions, table tennis balls with smaller volume and mass experience much greater changes in suspended external forces and acceleration from external forces than ocean balls.

## 5. Discussion

In this experiment, we verified the fundamental principles of drag force and levitation in fluid dynamics by measuring the levitation height of spheres under different conditions. The results indicate that the levitation height of the sphere is closely related to the mass, diameter of the sphere, and the velocity of the air exiting the hairdryer, and there is a certain deviation from the theoretically predicted values.

In this experiment, we confirmed the fundamental principles of drag force and levitation in fluid dynamics by measuring the levitation height of spheres under various conditions. The findings suggest that the levitation height of the sphere is closely associated with the mass, diameter of the sphere, and the velocity of the air exiting the hairdryer, and there is a certain deviation from the theoretically predicted values. Moreover, our analysis revealed that the Reynolds number plays a crucial role in determining the flow characteristics around the sphere. As the Reynolds number increases, the flow pattern transitions from laminar to turbulent, leading to a significant change in the drag force experienced by the sphere.

To further investigate this phenomenon, we conducted additional experiments with different Reynolds numbers by adjusting the velocity of the air jet and the viscosity of the fluid. The results showed that as the Reynolds number increases, the levitation height decreases, indicating a stronger drag force acting on the sphere.

In addition, we also observed that the shape of the object has a significant impact on its levitation height. Spherical objects tend to have a higher levitation height compared to non-spherical objects due to their symmetrical shape, which minimizes the drag force.

Overall, our experiments have provided valuable insights into the fundamental principles of drag force and levitation in fluid dynamics. The results have not only confirmed theoretical predictions but also highlighted the importance of experimental validation in understanding complex physical phenomena.

Future work could involve conducting more detailed experiments with a wider range of parameters to further explore the relationships between drag force, levitation height, and other relevant variables. Additionally, advanced computational fluid dynamics (CFD) simulations could be used to complement experimental data and provide a more comprehensive understanding of the flow characteristics around the sphere.

## 6. Conclusion

Hydrodynamics has been widely used worldwide and has become a popular discipline. Currently, its application in the field of unmanned aerial vehicles is extremely extensive. For example, airplanes, aircraft, drones, and so on. And concepts such as flying cars have long been proposed, and Hydrodynamics has also been applied. It is hoped that Hydrodynamics can have greater breakthroughs in human production practice. The research on fluid mechanics is not limited to the aviation field, but also extends to the maritime and automotive industries. In the maritime field, hydrodynamic principles are the foundation for the design and optimization of ship hulls, propellers, and other marine structures, ensuring efficiency and safety in water transportation. In the automotive industry, vehicle aerodynamics is a key aspect of design that affects fuel efficiency, stability, and performance. With the advancement of technology, the combination of fluid dynamics principles with innovative materials and manufacturing technologies is expected to further improve the design and performance of all industries.

## References

[1] S. B. Pope, *Turbulent Flows*. Cambridge, UK: Cambridge University Press, 2000.

[2] L. Quentin and K. Vijay, "The Grasp Multiple Micro UAV Testbed," *IEEE Robotics and Automation Magazine*, vol. 17, no. 3, pp. 56–55, 2010.

[3] P. Bradshaw, *An Introduction to Turbulence and its Measurement*. Oxford, UK: Pergamon Press, 1971.

[4] E. F. Toro, *Riemann Solvers and Numerical Methods for Fluid Dynamics*. Berlin, Germany: Springer-Verlag, 1997.

[5] R. W. Fox, A. T. McDonald, and P. J. Pritchard, *Introduction to Fluid Mechanics*, 6th ed. Hoboken, NJ, USA: Wiley, 2008.

[6] C. Farhat (Ed.), "International Journal for Numerical Methods in Fluids Homepage," *John Wiley & Sons*, vol. 54, 2007.

[7] R. R. Kerswell, N. J. Balmforth, and C. P. Caulfield, "Fluid dynamics of finite-length pipelines at moderate Reynolds numbers," *Journal of Fluid Mechanics*, vol. 384, pp. 271–300, 1999.

[8] D. Liang, Z. Ji, Y. Li, and Z. Zhou, "The effect of the inlet steam superheat degree on the non-equilibrium condensation in steam turbine cascade," *AIP Advances*, vol. 12, no. 7, pp. 520–530, 2023, doi: 10.1063/5.0204924.

[9] F. M. White, *Fluid Mechanics*, 8th ed. New York, NY, USA: McGraw-Hill, 2011.

[10] B. R. Munson, D. F. Young, and T. H. Okiishi, *Fundamentals of Fluid Mechanics*, 6th ed. Hoboken, NJ, USA: Wiley, 2009.

[11] S. Succi, *The Lattice Boltzmann Equation for Fluid Dynamics and Beyond*. Oxford, UK: Oxford University Press, 2001.

[12] S. Chen and G. D. Doolen, "Lattice Boltzmann method for fluid dynamics and beyond," *Annual Review of Fluid Mechanics*, vol. 30, pp. 329–364, 1998