The development of fluid dynamics and its future application

Heyang Ni 1,*†, Shengtao Yao 2,†, and Zhaoliang Zhou 3,†

1Tilton School, Tilton, New Hampshire, 03276, United States
2Jiangsu Tianyi High School, Wuxi, Jiangsu, 214000, China
3Jiangxi University of technology high school, Nanchang, Jiangxi, 330000, China
*Corresponding author's e-mail: jincheng@dipont-edu.com
†These authors contributed equally.

Abstract. The development of fluid dynamics is essential to many related topics in mechanics. With the emerging high technology, people have begun to discover more functions suitable for fluid mechanics. This article mainly summarizes the following two aspects. One is to review the historical development of fluid mechanics on the time axis, and to summarize the important achievements of fluid mechanics in recent years. The second is to summarize the practical applications of fluid mechanics in aviation, engineering application, shipbuilding and daily life. It points out in detail the shortcomings and deficiencies in the development of computational fluid mechanics so far, and points out the direction for the future development of computational fluid mechanics.

1. Introduction
Throughout history, Computational Fluid dynamics grew for the reason of aerospace engineering, in the purpose of describing the physical movement and phenomenon of real fluid. The main way to describe it is through high-speed computer calculation. Now the CFD has a wide application in aerospace engineering, transportation, shipbuilding, hydraulic pressure, and many other fields in engineering. CFD is a greatly advocated method for simulation as it can minimize the planning time and saves cost for experiment methods. And compared to theoretical analysis, its overweighing advantages are fewer limits on assumption and wide application form. This article mainly describes the CFD's development through a historical view and to modern-day CFD simulation. Its importance will be stressed at the end of the article, following possible future application.

2. The development of computational fluid dynamics
In the 1930s, due to the needs of the aircraft industry, it was required to use the theory of fluid mechanics to understand and guide the design of aircraft. At that time, because the flight speed was very low, the viscosity and vortex could be ignored, so the flow model was the Laplace equation. The focus of the research work was the numerical solution of the elliptic equation [1]. The analytical solution is obtained by using the complex function theory and the superposition method. As aircraft shape design becomes more complex, a method to solve the singular boundary integral equation (BEE) appears. Later, to consider the viscous effect, the numerical calculation method of the boundary layer equation was developed. It was developed to combine the potential equation as the outflow equation with the inner boundary layer equation to solve the viscous interference flow field by an iterative method.
At the same time, the partial differential equation is discretized, and then it is proved that the discrete system converges to the continuous system. Then the existence of difference decomposition is determined by the algebraic method. They also developed the famous stability criterion, the CFL condition. These jobs are different. The mathematical theoretical basis of the method. During the same period, many mathematicians studied the mathematical theory of partial differential equations. Hadamard, Courant, Friedrichs, et al. studied the basic characteristics of partial differential equations, the applicability of mathematical formulation, and the propagation characteristics of physical waves. After developing the theory of hyperbolic partial differential equations, Courant, Friedrichs, Lewy, et al. published classic papers [2], proving the existence and uniqueness theorem of solutions for continuous elliptic, parabolic, and hyperbolic systems of equations. Because of the initial value problem of linear equations, the partial differential equations were discretized firstly. Then it is proved that the discrete system converges to the continuous system. Finally, the existence of the difference decomposition is determined by using the algebraic method. They also developed the famous stability criterion [3], the CFL condition. These works are the theoretical basis of the difference method. In the 1940s, Vonneumann, Richtmyer, Hopf, Lax, and some other scholars established the numerical method theory of the conservation law of linear hyperbolic equations, which laid a theoretical foundation for the numerical simulation of gas flows with shock waves [4].

In the 1950s, it was not enough to study the complex nonlinear flow phenomena only by the method of fluid mechanics at that time. Especially it could not meet the needs of the research on the characteristics of the flow field around the rapidly developed space vehicle [5]. For this kind of situation, some scholars began to be based on hyperbolic equation time correlation method based on the mathematical theory is used for solving the aerospace craft gas unsteady flow field around a problem. This method may require spending more computer, but due to the well-posed mathematical formulation, and have a good theoretical basis, and can simulate the unsteady process of fluid motion. So in the 1960s this was a general method that was widely used [6]. The stability theory of differential approximation of unsteady partial differential equations, which was later given by Lax, Kreiss, and other authors, further promoted the time-dependent method [7]. There were also some special algorithms developed for specific problems [8].

In the 1970s, the research on numerical calculation of transonic flow around aircraft was a great success in computational fluid dynamics [9]. First, Murman and Cole used the relaxation method to solve the small perturbation equation of potential flow and numerically simulated the transonic flow field with shock waves, thus solving the mixing problem in transonic flow [10]. In their work, the upwind scheme was first applied to the simulation of aerodynamic problems. Not long after, Jameson proposed the rotation scheme and extended the Muhlman-Kohler method to solve the full potential flow equation of three-dimensional transonic flow around it and achieved success [3]. In the solution of compressible N-S equation, great progress has been made in calculation methods, such as switching function method, concatenation factor method, compact upwind scheme, advance iteration method, dissipation scheme with no fluctuation and no free parameters, bounded value scheme, and dissipation analogy method, etc. [11]. These studies further improve the accuracy of the calculation method, improve the solution's efficiency, and the numerical simulation of the flow field shock wave has a high resolution.

After entering the 1980s, computer hardware technology has developed by leaps and bounds, tens of millions of times the machine, hundreds of millions of times the machine gradually into the scope of people's practical activities [12]. With the continuous improvement of calculation methods and the development of numerical analysis theory, high-precision numerical simulation is no longer a fantasy. Simultaneously, with the continuous development of human production practice and the rapid development of science and technology, a large number of high-tech industries have put forward new requirements for computational fluid dynamics and provide new opportunities for the development of computational fluid dynamics. The continuous interaction between practice and theory forms a new hot spot and new power of computational fluid dynamics, thus promoting the continuous development of computational fluid dynamics.
At present, the focus of CFD research is as follows: research on computational methods, including parallel algorithms and various new algorithms; Eddy motion and turbulence, including direct numerical simulation of compressible and incompressible turbulence, large eddy simulation and turbulence mechanism; Research grid generation technology and computer optimization design; Computational fluid dynamics (CFD) is studied to solve practical flow problems, including computational biomechanics, computational acoustics, numerical simulation of micro-mechanical flows, multiphase flows and turbine mechanical flows.

3. Application of computational fluid dynamics

Computational fluid dynamics (CFD) was first applied to the aerospace field and later followed it. With continuous development and maturity, it is widely used in shipbuilding, chemistry, industrial design, etc. In the field, and from the actual situation has also achieved a good result.

The following is an example of the application of CFD in the automotive field. Using CFD technology to study automobile flow field is not only low cost and short period, but also the understanding of fluid motion is deeper and more detailed than experiments in a sense. Not only can we understand the results of motion, but also the overall and local The meticulous process is getting more and more attention. Most of the applications of CFD in automobiles are focused on the simulation of the external flow field of the automobile.

By calculating the flow field of the car body, the pressure field, velocity field, aerodynamic force and aerodynamic moment of the car body surface can be obtained, and then a series of design parameters that people are concerned about, such as wind resistance coefficient, can be obtained. The flow process and flow field distribution have a more detailed understanding. Based on the comparison with the experiment, the optimization design can be achieved by modifying the geometric parameters of the car body. Figure 1 shows the external flow field diagram of a certain car body.

![Fig. 1 The distribution of the flow field on the surface of the automobile](image)

In addition to the calculation of the external flow field of the whole vehicle, the calculation and analysis of the local flow field such as rearview mirrors, flow deflectors and wheels can design reasonable parts or installation dimensions to further reduce wind resistance. In addition, for the driving conditions of plural vehicles (three types of meeting, overtaking and platooning), because its aerodynamic characteristics are a transient process, foreign countries have only conducted preliminary qualitative analysis, but this is undoubtedly an important research direction. It plays an important role in studying the handling stability in this state. Figure 2 is the static pressure distribution diagram when a car overtakes.
Fig. 2 Static pressure diagram when two cars are overtaking

CFD can also analyze the internal flow field of the car. The heat transfer model can be used to easily obtain the distribution of the temperature field and the velocity field in the air-conditioned car. By analyzing the distribution characteristics of the temperature field and the velocity field in the car, it is possible to find out the effect of the air-conditioning refrigeration. The key is to find ways to reduce the power consumed by the air conditioner and reduce fuel consumption. Figure 3 is an example of the front seat of a car produced by Dongfeng Motor Co., Ltd., considering the radiant heat transfer of the geometric surface and the convective heat transfer in the space, the temperature distribution of the driver’s seat. Due to the contact between the human body and the seat, the air flow in the buttocks and back is not smooth, and the temperature in these positions is too high. Through the analysis of the air distribution in the car, in addition to guiding people to improve the seat, it can also guide the position of the air conditioner, the air inlet and the air conditioner. Design of car windows, etc.

Fig. 3 Surface temperature field distribution of front seats

In addition, CFD is also widely used in the design of oil pumps, hydraulic torque converter blades, cooling system cooling fan blades and hydraulic brakes. Figure 4 shows the streamline distribution near the blades in the turbine runner of the torque converter. The black part in Figure 4 is the turbine blade, and it is obvious that the streamline above the right blade forms a vortex. Through the three-dimensional simulation calculation of the internal flow field of the hydraulic torque converter, the generation of secondary flow, outflow and vortex can be accurately simulated, and the calculation
results can be verified through experiments. A design analysis method based on the three-dimensional flow field theory can be proposed. Effectively solve the problems of transmission efficiency, low design accuracy and long development period of the hydraulic torque converter.

![Streamline distribution near turbine blades](image)

**Fig. 4** Streamline distribution near turbine blades

Computational fluid dynamics (CFD) is reasonable in rotor calculation and flow field calculation of application and has achieved good results. The results are shown by computational fluid dynamics. It has a good application value in solving practical engineering problems. In the process of physical science of computational stream, the specific process is as follows:

1) Build a model, and then use mathematical methods according to the corresponding professional knowledge to express.

2) Apply the corresponding software to complete the solution and points of various problems analysis.

Computational fluid dynamics (CFD) is applied in practical engineering problem solving. It has certain advantages, mainly reflected in the following aspects:

First, a more thorough analysis of fluid flow. In-depth analysis of the actual transmission situation in matter and energy can be carried out. Second, in the process of problem analysis, CFD can make a proper change to Test parameters, conditions, and other contents involved in the test process that cannot be easily obtained by traditional research approach. Thirdly, it greatly shortens the time of design and research. Fourth, can be applied to high-risk, high-temperature environments; Fifthly, through simulation data, the whole process can be optimized [4].

In recent years, computer hydrodynamics engineering technology has been widely used and gradually developed in the direction of the software. Various types of commercial software can be used as professional software for special purposes. Through the application of CFX, FLUEN, and other different types of software, we can complete the processing of different difficult and complex problems involved in the field of engineering technology so that the processing of each problem becomes simple [5]. However, it should be noted that the application of computational fluid software in some fields is not mature. So it is necessary to strengthen its research and improve the aspects of function enhancement, calculation accuracy, and simplification of operation so that its application effect can be further improved.

The current practical application of computational fluid dynamics, combining its computational fluid dynamics simulation tool with modern computers creates a new concept. It reduces the cost of specific design and further reduces the actual development time. At present, the primary task of CFD researchers in the design and research process is to quickly and reasonably develop efficient and accurate viscous flow calculation methods to push the concrete application of CFD to a new height and promote its development pace.
4. Conclusion
Computational fluid dynamics is a branch of fluid mechanics, and at the same time, it has a wide range of applications in many aspects. This article reviews the development of computational fluid dynamics and its engineering application. And it is proposed that the future development of computational fluid dynamics is mainly in two aspects: on the one hand, it is to study the unsteady and stable characteristics of the flow, the bifurcation solution and the mechanism of turbulent flow, and the more complex unsteady, multi-scale flow characteristics, high precision and high Resolution calculation methods and parallel algorithms; on the other hand, computational fluid dynamics is directly used to simulate various actual flows to solve various problems raised in industrial production. Therefore, relevant researchers should continue to strengthen their analytical work to make computational fluid dynamics more widely used.

References
[1] Liu, G. The status, development and development trend of computational fluid dynamics. Aeronautical computing technology, 1994(1): 15-21.
[2] Guo, Z. (1999) Principle of reinforced concrete. Tsinghua University Press, Beijing.
[3] Qiu, H. (2002) Ganping Shu.design of building structure. Southeast University Press, Nanjing.
[4] Shu, Q. A review of discontinuous Galerkin methods in computational fluid dynamics. Mechanical progress, 2013, 43(5): 541.
[5] Bo, X., Liu, J., Xu, J. Research on the application of computational fluid dynamics in urban planning and design. Journal of Sichuan University (Engineering Science Edition), 2002, 34(6): 36-391.
[6] Wulandari, R., Sihassaleh, P., Ramadani, et al. Goose foot water turbine performance with variation of fins quantity and turbine depth using computational fluid dynamics (CFD) approach. IOP Conference Series Materials Science and Engineering, 2021, 1034(1): 012062.
[7] Permanasari, A. A., Rusli, M. H., Puspitasari, P., et al. Computational fluid dynamics heat transfer analysis of double pipe heat exchanger using nanofluid MnFe2O4 with ethylene glycol/water. IOP Conference Series: Materials Science and Engineering, 2021, 1034(1): 012065 (9pp).
[8] Aoki, R., Fujioka, S., Terasaka, K. Experimental Study and Prediction by Computational Fluid Dynamics on Self-induced Sloshing Due to Bubble Flow in a Rectangular Vessel. Journal of chemical engineering of Japan, 2021, 54(2): 51-57.
[9] Heil, M., Hazel, A. L. Fluid-Structure Interaction in Internal Physiological Flows. Annual Review of Fluid Mechanics, 2011, 43(1): 141-162.
[10] Smolianski, A., Haario, H., Luukka, P. Vortex shedding behind a rising bubble and two-bubble coalescence: A numerical approach. Applied Mathematical Modelling, 2005, 29(7): 615-632.
[11] Li, Z., Bouscasse, B., Ducrozet, G., et al. Spectral Wave Explicit Navier-Stokes Equations for wave-structure interactions using two-phase Computational Fluid Dynamics solvers. Ocean Engineering, 2021, 221(2021): 108513.
[12] Mahalov, A., Titi, E. S., Leibovich, S. Invariant Helical Subspaces for the Navier-Stokes Equations. Archive for Rational Mechanics and Analysis, 1990, 112(3): 193-222.