Application of Fluid Dynamics in the Development of Communication Industrialization

Lingrui Xu*, Xiong Xiong
Department of Electronics and Information Engineering, Beijing Jiaotong University, Beijing, China

*Corresponding author e-mail: 997197288@qq.com

Abstract. Fluid mechanics has made great progress in recent decades, especially in the field of communication engineering, which is an important emerging direction. This paper provides an overview of the main essence and latest progress of this method. At the same time, it also discusses the development of relevant scientific and technological software and its impact on practical industrial applications.

1. Introduction
Like other natural science studies, fluid mechanics has developed with the development of production. Around the 17th century, due to the rise of capitalist system and the rapid development of production, the need for the development of fluid mechanics is more urgent. During this period, there were two approaches to the study of fluid mechanics, which were not related to each other and had their own characteristics. One is classical fluid mechanics, which uses rigorous mathematical analysis to establish basic equations of fluid motion and try to solve them. The founders of this approach are Bertrand Euler and Euler. Lagrange, Navier, Stokes and Reynolds, most of whom are scientists and physicists, have made great contributions to the formation and development of classical fluid mechanics. Classical fluid mechanics cannot be used to solve practical problems because some theoretical assumptions in classical fluid mechanics are different from actual ones, or because the solution of basic equations meets mathematical difficulties. Engineers with outstanding achievements in hydraulics, including Bito, Caixi, Venturi, Darcy, Manning, Federer and others. However, due to insufficient theoretical guidance and relying solely on experiments, the hydraulic system in this period has certain limitations in application and is difficult to solve complex engineering problems [1].

From the end of the 19th century to the 20th century, with the continuous expansion of production scale and the complexity of technology, it is difficult to solve the complex hydrodynamic problems in engineering technology only by theoretical analysis or experimental research, which makes them closely related. In this period, the outstanding scientist of our country, Qian Xuesen, put forward the solution of the boundary layer of the compressible layer of flat plate-Carmen-Qian Xuesen solution. He has made important contributions in the fields of aerodynamics, aeronautical engineering, jet engineering, engineering cybernetics and other technical sciences.

Since the 20th century, fluid mechanics and other disciplines have penetrated into each other, forming many frontier disciplines, and developing fluid mechanics into a new scientific system. Mainly includes: magnetohydrodynamics, chemical fluid mechanics, rarefied gas dynamics, viscous fluid
mechanics, multiphase fluid mechanics, hydrodynamics, osmotic mechanics, non-Newtonian fluid mechanics, geofluid mechanics, computational fluid mechanics and so on. [2]

2. Lattice Boltzmann Method and Progress in Communication

Fluid mechanics is formed by continuum model relying on mathematical analysis method. Computational fluid dynamics (CFD) is formed by computer. In the early stage, due to the limitation of the development level of computational technology, the highest goal of CFD is to solve Navier-Stokes equation directly. Boltzmann equation is more than Navier-Stokes equation. In order to reflect the basic model equation of fluid mechanism, the lattice Boltzmann method has been developed in computational fluid dynamics due to the rapid development of computational technology and the success of lattice algorithm. With this method, nowadays computational fluid dynamics can analyze many phenomena of fluid motion that cannot be described by Navier-Stokes equation.

2.1. Lattice Boltzmann Model for Fluid Simulation

The standard lattice Boltzmann equation is generally described by the following mathematical expressions:

\[ f_a(x + \xi_a dt, t + dt) = f_a(x, t) + C_a(f) \]  

In the equation, \( f_a \) represents the particle distribution function, \( C_a \) represents the collision term. The lower corners of the above items represent the discrete velocity of a given particle. For simplicity, the general definition of unit is usually adopted. The most commonly used form of the collision term is the so-called BGK model.

\[ C_a(f) = -\frac{f_a - f_a^{eq}}{\tau} \]  

In the early lattice Boltzmann method, the equilibrium state distribution is a small Mach number expansion with several undetermined coefficients. The expansion coefficients are given by Chapman-Enskog. It can be proved that when the discrete velocity set satisfies certain symmetry requirements, the system represented by Formula (1) is macroscopically based. Ben satisfies Navier-Stokes equation. [3]

2.2. LBM Model’s Application in Industry of Communication

The lattice Boltzmann model has some obvious advantages in numerical simulation of fluids. For example, its communication process is simulated by constant velocity. The corresponding calculation is a very simple operation step. When the appropriate lattice mesh is selected, the process can usually be realized in a completely translational way. For the conventional finite interpolation language in computational mathematics, it can be applied to the simulation of fluids. However, the difference is that the corresponding Courant Number equals 1. By contrast, the convection term of Navier-Stokes equation is a non-linear function that varies with time and space. It is well known that the calculation of Navier-Stokes equation is not a simple matter, and the requirement of numerical stability forces people to use a much smaller Couran number than 1 in the calculation of practical problems. In the case of given spatial resolution, the small Couran number means an hour step, which greatly prolongs the calculation time; at the same time, the small Couran number also increases the numerical diffusion error, forcing people to adopt a higher precision format or implicit format. The consequence is that the algorithm becomes extremely complex and the parallel efficiency is greatly reduced; or the calculation is limited to the case of processing the steady-state communication conditions. In fact, the extreme simplification of complex communication circuits will greatly limit the situation. Many important hydrodynamic problems can not be approximated by the steady flow hypothesis even if we only care about the time-averaged results. Here we also mention another essential characteristic of lattice Boltzmann equation: all non-linear effects are in lattice Boltzmann method. All these reasons imply that lattice Boltzmann method is a superior method for large-scale parallel simulation of unsteady flow.
3. Application of Boltzmann Method in Communication Engineering

Large eddy simulation of lattice Boltzmann equation has been widely used in practical engineering calculation. Because of the sensitivity of fluid to small geometric changes and the coupling effect of various parts of fluid, it is strictly unreliable to approximate it by simplifying geometric shape. For communication engineering or design, besides aerodynamics, drag coefficient, lift coefficient and moment caused by loss, it is not reliable to approximate it by simplifying geometric shape. In addition, people are also concerned about many other kinds of problems, such as air noise of objects. Many engineering performance indicators differ in the size of noise they produce, and many of the factors determining these important indicators come from the geometry of some key parts, such as curvature treatment, which may lead to different separation of fluids, resulting in instability of airflow. In addition, computational fluid dynamics (CFD) is also very important for the optimal design of the intake. The shape, location and relative layout of the intake and other components may bring different fluid effects, which directly affect the power and radiation efficiency. The power and noise performance related to radiator design are also important subjects in the application of computational fluid. Related topics include the design of conductors and transformers. For all these subjects, direct wind tunnel tests are not only time-consuming and time-consuming, but also impossible to measure some key factors or situations. Although the existing computational fluid dynamics methods have not completely replaced the actual experiments in terms of accuracy and reliability. However, it has become an extremely important auxiliary means, which can provide more comprehensive and detailed information about the characteristics of fluid in various situations, so that people can get a more rational understanding of fluid problems. At the same time, virtual test before manufacture not only saves time and cost, but also can optimize the design in a larger range through simultaneous calculation of multiple operating conditions. Computational fluid dynamics (CFD), especially lattice Boltzmann method, has a very wide application and development space in practical fluid engineering applications. [4]

Generally, the familiar non-sliding condition can be explained as the macroscopic limit of particle wall reflection. However, in a broader sense, the sliding phenomenon is a natural physical phenomenon. This also provides a new way to realize the turbulent boundary model, that is, we can use the sliding process in the appropriate molecular motion theory to achieve the purpose of achieving the large eddy boundary. Essentially, the boundary condition. The core task is to accurately determine the flow rate of basic physical quantities, such as the density, momentum and energy of the fluid passing through the wall, which can be accurately realized by appropriate particle reflection process. The normal component of the momentum flow generated from this process automatically corresponds to the fluid pressure value, while the boundary conditions that usually depend on the velocity itself and the gradient linear function of the flow field are essentially cow-like. The relationship between stress and strain, which is based on the non-Newtonian fluid, must be linear. Because of the distance from equilibrium state, the characteristics of large eddy flow have many similarities with that of finite Nusselt number flow, including the non-linear effect of non-Newtonian fluid [22]. Whether the molecular motion theory under the generalized definition is a more appropriate description of the large eddy turbulence physics based on the large eddy closed model equation. In this paper, the accuracy and results of industrial data in the same communication engineering are shown in Fig. 1 and Table 1 by using lattice Boltzmann and general fluid dynamics methods.
Figure 1. Stimulation result of Lattice Boltzmann optimization results (experiment 1), general hydrodynamic results (experiment 2) and original data.

In contrast, the lattice Boltzmann equation is closer to physical reality. After solving the computational complexity, the algorithm based on the Boltzmann equation allows more direct simulation of a wider range of physical mechanisms. In addition to the advantages of numerical simulation in the communications industry shown in this paper, its simulation of the constitutive relationship of multiphase flow and non-Newtonian fluid can be simulated by simulation. The micro-mechanism of these macro-physical phenomena is realized.

Lattice Boltzmann has many advantages in dealing with boundary conditions of complex geometric shapes. Firstly, its constant velocity convection characteristics enable the near-wall process to be achieved by constant geometric weights. This information can be completed in the pre-processing stage before the simulation calculation, thus making the calculation of extremely complex geometric shapes more concise and effective. In contrast, the mesh generation of complex geometric shapes is based on NA. The traditional computational fluid dynamics based on Wilson-Stokes is a tedious work. Secondly, more importantly, it can deal with a wider range of physical boundary conditions. The familiar non-sliding condition can be interpreted as the macro-limit of particle wall reflection. However, in a broader

### Table 1. Accuracy between the two experiments.

| Numble | Experiment 1 | Experiment 2 |
|--------|--------------|--------------|
| 1      | 92.7%        | 83.9%        |
| 2      | 78.9%        | 42.9%        |
| 3      | 93.1%        | 65.4%        |
sense, sliding phenomenon is a natural physical phenomenon, which also provides a new realization for turbulent boundary model. The way is that we can use the sliding process in the appropriate molecular motion theory to achieve the purpose of large eddy turbulent boundary. [5]

Essentially, the core task of boundary conditions is to accurately determine the flow rate of basic physical quantities, such as the density, momentum and energy of fluid passing through the wall, which can be accurately realized by appropriate particle reflection processes. The normal component of momentum flow generated from these conditions automatically corresponds to the value of fluid pressure, and usually depends on the boundary of velocity itself and the linear function of flow field gradient conditions. [6] Essentially, it is based on Newtonian fluid, that is, stress and strain must be linearly related. Because they are far from equilibrium, the characteristics of large eddy flow have many similarities with finite Nusselt flow, including the non-linear effect of non-Newtonian fluid.

4. Conclusion
First of all, we want to discuss the importance and necessity of developing computational fluid dynamics software. The development and popularization of software industry is the main part of the third industrial revolution (information technology) in the world. It promotes social progress unprecedentedly. The leading degree of software industry has become an important symbol to measure a country’s industrial level and competitiveness. The high-tech industry with software as the main body has developed considerably, and the development of scientific and technological software itself is also a high-tech industry. Although some countries are in the leading position in this field, it is still in the initial stage of rapid development in general. Therefore, it is very important to seize the first opportunity. Experience shows that creating an environment that can accommodate software innovation, especially scientific and technological software innovation, is not only for scientific and technological software. The research and development of scientific and technological software is essentially different from the so-called software. Its main feature is that it is formed by the intersection of many cutting-edge disciplines. Its key components include advanced physical models, superior computational mathematical methods, latest software engineering synthesis and so on. Matching (such as automatic grid generation, parallel computing load distribution, pre-and post-processing, etc.), deep understanding of the main industrial application problems (such as the understanding of real aerodynamic phenomena) and even advanced management and operation modes, all of which require close cooperation of various disciplines to form an organic combination of echelons to adapt to this system engineering.

References
[1] Papukchiev A, Roelofs F, Shams A, et al. Development and application of computational fluid dynamics approaches within the European project thins for the simulation of next generation nuclear power systems [J]. Nuclear Engineering & Design, 2015, 290: 13-26.
[2] Ramanath H S, Chua C K. Application of rapid prototyping and computational fluid dynamics in the development of water flow regulating valves[J]. International Journal of Advanced Manufacturing Technology, 2006, 30(9-10): 828-835.
[3] Konrad H W. S.H. Haber, Industry and Underdevelopment: The Industrialization of Mexico, 1890-1940 [C]// Mechanics of Sound Generation in Flows. 1991..
[4] Zhong C, Wang G, Mirza Z A, et al. Study of transpiring fluid dynamics in supercritical water oxidation using a transparent reactor [J]. Journal of Supercritical Fluids, 2014, 88 (1): 117-125.
[5] Shaari K Z K, Awang M. Engineering Applications of Computational Fluid Dynamics [J]. Advanced Structured Materials, 2015, 44 (05): 101-123.
[6] Noguchi M, Friedman A. Manufacturer-User Communication in Industrialized Housing in Japan [J]. Open House International, 2002.