Science in China Series E: Technological Sciences

New direction of computational fluid dynamics and its applications in industry

CHEN YaoSong[1†], SHAN XiaoWen[2] & CHEN HuDong[2]

[1] Department of Mechanics, Peking University, Beijing 100871, China;
[2] Exa Corporation, Burlington, MA 01803 USA

For the past ten years there has been much progress in computational fluid dynamics (CFD), among which the formation and development of the lattice Boltzmann method (LBM) are an important new direction. We give a review on the main aspect and the latest development of this method in this article, and at the same time we also discuss the related development of scientific software and its impact on the real-world applications in industry.

lattice method, Boltzmann equation, CFD

1 Introduction

Since the rapid advancement of the capability of modern computers, especially the appearance of large scale parallel computing, people have begun to realize the availability and necessity of solving scientific problems in industry using computational simulation methods. Fluid mechanics, as a classical subject of science, has gained new vein by the usage of computers. The application area of fluid mechanics concerns various aspects in industry, such as aerospace, weather forecast, oil/gas drilling, optimization of aero dynamics of automobiles, analysis and reduction of noise source of fans and moving objects, influence of heat conduction and transfer on the modules and environments, etc. As with the birth of some new industries, the application area of fluid mechanics has extended to the micro-flows, multi-phase flows, non-Newtonian flows and other complex fluids. Along with the rapid development of computer capability, a new direction in the high-tech scientific research and development in the world has emerged as to simulate the important large scale scientific and industrial projects using computational fluid dynamics (CFD) methods.

The real-world application of CFD has helped people to understand many physical mechanisms of fluids to the extent far beyond what can be reached by the classical methods. Not only

---

Recommended by Prof. GU SongFen, Member of Editorial Committer of Science in China, Series E: Technological Sciences
Received May 14, 2007; accepted July 30, 2007
doi: 10.1007/s11431-007-0075-4

† Corresponding author (email: chenys@pku.edu.cn)

does this have important impact on the R&D of new technology to reduce the energy consumption and increase the efficiency, but also it is crucially helpful for better understanding and optimizing industrial designs. For example, when an aircraft flies under large attack angles or makes maneuver movements, it is hard to gain the details of the complicated fluid effects using the traditional method of wind tunnel experiments. Actually under many circumstances such key information can hardly be obtained using experiment means. On the other hand, according to the understanding of fundamental physics of fluids, we know that the characteristics of fluids can have qualitative changes due to some minor changes of the environment. Also, the wind tunnel experiments usually take long period of processes and expensive budgets. All these aspects put severe constraints on the effect of wind tunnel experiment for aero-dynamics design. Therefore, an accurate and rapid CFD method can lead us from "knowing what it is" to "knowing why it is" on the problems of fluid mechanics, and give in-time and accurate feedback to the designers. Thus it plays a crucial role on the optimization of products and development of new technology.

The theoretic description of fluid mechanics is conventionally based on the Navier-Stokes equation (N-S equation). As the foundation of fluid mechanics, it has existed for more than a century. People take no objection to the physical reliability and accuracy of the equation in the normal scales.

In various academic organizations, the study of fluid mechanics is mainly based on the Navier-Stokes equation. But due to the nonlinear nature and difficulty of dealing with boundary conditions for the equation, it is a challenging task to solve the N-S equations analytically or numerically except for some simple problems. To show the existence and smoothness of the solutions for the N-S equation is still one of the millennium problems posted by the Clay Mathematics Institute.

Apart from the difficulty of solving the equation, as a description of the physics of the fluids different from the Newtonian dynamics equation for the classical mechanical motion or the Schrodinger equation for the quantum mechanical motion, the N-S equation starts from a more fundamental formulation, while ignoring reasonably some physical mechanisms, obtains itself by taking statistical averaging. In nature the N-S equation surely cannot describe the macroscopic phenomena induced by the ignored physical mechanisms, such as the phase transition in the fluid systems, the non-Newtonian strain-stress relations, and the physical phenomena in the scale of the mean free path of particles motion. The high-speed development of modern technology has extended the views of people to the more general physical phenomena than that of classical fluid mechanics, such as micro flows and the complex fluids. The N-S equation has obviously exhibited its limitations in these areas. At the same time, due to the limited capabilities of modern computers, some physical phenomenon of fluids, such as turbulence, can not be realized using the direct numerical simulation (DNS) approach. Various physical approximations have to be added. The most common and available one is the so called eddy-viscosity model[1].

Due to the difficulties, uncertainties, and errors caused by the various reasons above, CFD has not been able to fully replace the role of experiments in the area of science and technology, especially in real-world engineering applications. Nevertheless, CFD has become one of the fastest growing fields in the world. Not only has it exerted substantial influence on scientific and technological innovations as well as engineering and industrial applications, but also it represents a direction of development of the new industry in the world: the high-tech software industry.

Since the end of the eighties in the last century, a new theoretic description and effective computational method of the fluid mechanics has emerged. This is what people nowadays refer to as the lattice Boltzmann method (LBM). In the past twenty years, people's understanding of this method has undergone a zigzag path, and has obtained concrete progresses. Especially in the recent few years, the understanding of this method has gained qualitative progress. At the same time it has evolved from an academic theoretic model to a computational software tool with real-world engineering application values, and thus has been used in the research and development of the new products in some major industries[2]. For example, almost all the car manufacturing companies in the world are using the LBM-based CFD software to optimize the aero-dynamics property of their new car models. As the method gets further extended and improved, we believe that it can get further used in more general areas.

2 LBM and its new development

Fluid mechanics as a subject is formed based on the mathematical analysis of the continuous media modeling of fluids. The appearance of electronic computers has forged the formation of computational mathematics, thus the formation of computational fluid mechanics (CFD). In the early stage, due to the limited capabilities of the computing technology, the ultimate goal of CFD is to solve directly the Navier-Stokes equations. The Boltzmann equation as a model equation is more fundamental than the Navier-Stokes equation as far as the description of the fluid mechanics is concerned. Due to the rapid development of computing technology and the success of the lattice method, the lattice Boltzmann method has been developed in the area of computational fluid dynamics. By the usage of such method, CFD nowadays can analyze different fluid phenomena that can not be described by the Navier-Stokes equations.

For the early development of LBM, there have been numerous papers giving a complete description therein. Here we only give a simple introduction. From the historical point of view, LBM was evolved from the so called lattice gas model at first[3,4], while the latter is an abstract and simplified mathematical model of particle movement. As for each lattice gas model, the particle can only live in a discrete world. Its particle distribution, velocity and spatial location can only exist as integers. The relation between a lattice gas model and the real physics of fluids lies in the following physical assumption: physical systems with different microscopic details can correspond to the same macroscopic behavior. Take water and air as example, they have different physics at the microscopic level, but they all satisfy the Navier-Stokes equation approximately at the low Mach number and in the ordinary scales. Therefore, people want to construct possibly the simplest model such as a lattice gas model to describe the complicated macroscopic phenomena. In the meantime, it is quite easy to numerically simulate a lattice gas model. It also offers a simple and efficient path for solving the problems in fluid mechanics. The introduction of lattice Boltzmann method was originated from two motivations, one is to reduce the numerical noise caused by the integers[5], the other is to overcome the non-physical shortcomings existed in the lattice gas models[6−8]. Indeed, when an appropriate particle equilibrium distribution is chosen, the macroscopic behavior of the lattice Boltzmann system can be shown to satisfy the Navier-Stokes equation. Therefore, people can solve the Navier-Stokes equation indirectly by simulating a lattice Boltzmann system.

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

$$f_a(x+\xi_a \mathrm{d}t, t+\mathrm{d}t) = f_a(x,t) + C_a(f), \tag{1}$$

where f_a represents the particle distribution function, C_a represents the collision term. The subscripts in each term above represent the discrete particle velocity values. For the sake of simplicity, people usually adopt the normal unit convention. The most common form of the collision term is the so called BGK[9] model:

$$C_a(f) = -\frac{f_a - f_a^{\mathrm{eq}}}{\tau}, \tag{2}$$

where f_a^{eq} represents the equilibrium distribution, τ is the relaxation time for the particle distribution to converge to the equilibrium state by collision. In the early lattice Boltzmann formulation, the equilibrium distribution is a low Mach expansion containing some to-be-determined coefficients. The expansion coefficients are given by the inverse Chapman-Enskog expansion. It can be shown that when the discrete velocity set, ξ_a, satisfies certain symmetry requirements, the system represented by eq. (1) basically satisfies the Navier-Stokes equation at the macroscopic level.

Numerical simulation of the fluids by the lattice Boltzmann method has some obvious advantages. For example, its advection process is realized by a constant velocity. The corresponding computation is an extremely simple operation. When an appropriate lattice grid is chosen, the advection process can usually be realized by a complete translation. In the language of the conventional finite difference scheme in computational mathematics, it corresponds to an up-wind interpolation. But what is different is that its corresponding Courant number is unity. In contrast, the advection term in the Navier-Stokes equation is a nonlinear function depending on both time and space. As is well known, it is not an easy job to calculate. Also, the requirement of numerical stability forces people to use a Courant number much smaller than unity in practical computations. When the spatial resolution is given, a small Courant number means a small time step, thus it greatly enlarges the computational time. At the same time, a small Courant number amplifies the effect of numerical diffusion, thus it forces people to use higher-order schemes or implicit schemes. The outcome is that either the scheme becomes extremely complicated so that the parallel efficiency is greatly reduced, or the computation has to be restricted to treat the steady flows. In fact, a steady flow assumption is an extreme constraint of the fluid motion. Many important fluid problems, such as separated flows, even if we only care about the time-averaged results, cannot be approximated by a steady flow assumption. Here we would also like to mention another essential characteristic of the lattice Boltzmann method: all the nonlinear effect is contained in the collision term in the lattice Boltzmann method, and expresses itself in terms of purely local information. This further enhances the advantages of parallel computing. All these reasons mean that the lattice Boltzmann method is a desirable method for performing large scale parallel simulations for unsteady flow problems.

Nevertheless, the theoretical framework of the lattice Boltzmann method familiar to most people has some essential shortcomings. Since it uses the inverse Chapman-Enskog expansion to determine the key parameters in the equilibrium distribution function so as to reconstruct the macroscopic physical system, it becomes powerless for fluid physics beyond that of the Navier-Stokes equations, which usually lacks a reliable explicit macroscopic description. Apart from such a constraint, most known lattice Boltzmann models have some other obvious limitations. For example, their discrete velocity sets can only achieve the forth order isotropy requirement.

Also its equilibrium distribution can only be applied in the low Mach number, near zero temperature variation situations. This has placed great restriction on the application of the lattice Boltzmann method in simulation of compressible fluid flows. Here, we would like to mention some related misconceptions in the academic community about LBM. It confuses the lattice Boltzmann method with some specific existing lattice Boltzmann models. This has led to some wrong conclusion that this method can only treat the nearly incompressible and constant temperature Navier-Stokes fluid problems.

In the recent years, by in-depth exploration of the macroscopic representation form of the physical system of fluids, our understanding of the lattice Boltzmann method has undergone a qualitative advancement. As is well known, the statistical physical description of a many-body system can be related to the Hamilton equation of N particles, and the Liouville equation describing the combined probability distribution of N particles. In the realm of Newtonian mechanics, these can be regarded to be accurate. By making some assumptions on the collision effect of multiple particles, the motion of fluid can be described by the distribution function in the six dimensional phase space and its control equation, i.e., the Boltzmann equation. The physical quantities that people are interested in, such as the density, fluid velocity, and temperature, are all low-order moments of the distribution function in the velocity space. The fluid equations that people are familiar with, such as the Euler and Navier-Stokes equations are respectively the zero-th order and first-order approximate solutions of the continuous Boltzmann equation near the equilibrium state, obtained order by order via the Chapman-Enskog expansion. In the language of statistical physics, the part away from the equilibrium distribution in the distribution function (Maxwell-Boltzmann distribution) is totally neglected in the Euler equation, whereas in the Navier-Stokes equation only the linear contribution is preserved. But the Boltzmann equation itself contains non-equilibrium contribution of all orders, thus is more universal in the description of fluid mechanics. When the fluid is far away from the equilibrium, such as the cases of high Knudsen numbers and high Mach numbers, this universality is quite important.

On the other hand, under the current computing condition, information contained in the six-dimensional distribution function requires far beyond what can be dealt with by the present day computers, and is way beyond what is directly related to the realistic problems. To solve the problems of fluid mechanics by solving the Boltzmann equation is not only unrealistic, but also unnecessary. What are useful in the real problems are only the lower order moments of the distribution function. It thus prompts people to look for an efficient description of the continuous Boltzmann equation by its lower order moments. More than half a century ago, Grad first proposed expanding the distribution function using the Hermite polynomials[10] and solving the equation after making truncation in the spectral space. The Hermite polynomials give a complete ortho-normal basis in the velocity space. The coordinates in the Hilbert space spanned by the Hermite polynomials of an arbitrary distribution function correspond to its lower order moments. Therefore, the truncation of a distribution function in the Hermite space cannot induce any errors in the calculation of its lower order moments. Based on this idea, Grad derived his famous thirteen-moment model for fluid mechanics.

Due to its complexity, Grad's thirteen-moment model can not be directly used as an efficient computational method, but it offers a key indication to reconsider and expand the lattice Boltzmann method, i.e., the lattice Boltzmann method can be understood as an equivalent to a finite order Hermite polynomial expansion of the continuous Boltzmann equation[11,12]. When the first

N-th terms are determined, its lower order moments can be accurately expressed via the discrete values of the distribution function using the Gauss-Hermite quadrature. These discrete values give exactly the discrete velocity of particles in the lattice Boltzmann system. This totally new framework of formulation of the lattice Boltzmann method not only can offer another interpretation of the known lattice Boltzmann models, but also offers a systematic procedure for deriving higher-order models. Indeed, the usually known lattice Boltzmann models such as D3Q15 and D3Q19 correspond to a second order Hermite expansion. The higher order lattice Boltzmann models can then be naturally derived. This opens an efficient way to describe correctly the heat motion of fluid flows at finite Mach numbers or finite Knudsen numbers, the physical mechanism of latter is beyond the description of the Navier-Stokes description. What is worth emphasizing is this new framework of formulation of the lattice Boltzmann method is totally independent of whether there exists any corresponding description given by a macroscopic equation. We can know from the corresponding Gaussian integral that the higher the truncation moments, the larger the number of corresponding discrete velocity values. Models of different orders contain automatically the corresponding higher order fluid moments and the physical effects of higher order non-equilibrium states. This is not related to the closure problems of macroscopic description. Here we mention that this entirely new formulation not only gives a clear definition of the analytical form of the equilibrium distribution, but also offers a strict description for the non-equilibrium state and its evolution in the Hermite space, thus it offers a physically more reasonable, computationally more efficient multiple-relaxation-time (MRT) and adaptable Prandtl number model[13,14]. The latest progress of the Boltzmann equation has opened the door of using the method to compute the heat flows, the high Knudson number flows, and the finite Mach number flows.

On the contrary, there exist fundamental difficulties using the traditional computing methods based on the macroscopic description (Navier-Stokes equation or Burnett type of equations) to solve problems mentioned above. Besides the boundary conditions, there still exist many questions and disputes over the mathematical validity of these macroscopic equations beyond the Navier-Stokes equation using various closure assumptions. There exist similar problems in the calculation of multi-phase flows. As is well known, the physical mechanism for multi-phases flows is the long distance interaction between particles. This mechanism is far beyond the physical phenomenon that can be described by the usual fluid equations. Computational methods for the multi-phase flows based on the fluid equations must rely on additional modeling to simulate the physical phenomenon not included in the fluid equation itself. Apart from the problems displayed by the real numerical results, such approach contains serious fundamental physical flaws in its essence. The flaw exhibits itself mainly in the accurate description of the interface of phases. That is, near an interface of very sharp phase transition, it is quite questionable as far as the approximation given by the near equilibrium description of Navier-Stokes equation is concerned. This is also exhibited in the contradiction between the boundary condition for phase interface and the no-slip condition on a solid wall. To overcome such a shortcoming, people have to introduce various empirical slip models. On the other hand, the lattice Boltzmann method based on the mesoscopic description of fluids can tolerate larger non-equilibrium state and achieve generally more precise boundary conditions. Besides, the equation of state for pressure in the mesoscopic phenomenon is a natural outcome of the interaction between the particles, thus it is not input nor dealt with separately. In the situation involving phase transitions, the macroscopic characteristic of objects will express discontinuity, while the corresponding microscopic and mesoscopic mechanism is still the same. The lattice Boltzmann method has extensive applications in simulating the multiple phase flows[15,16].

3 The application of LBM in CFD

Professor C. C. Lin has explained more than once the mission of “applied mathematics”. He gave an explicit definition of “generalized applied mathematics” in the centennial anniversary in memory of Professor Zhou P. Y’s birth: Establish the appropriate mathematical model for the target > Make prediction of the scientific problem using mathematical methods (including solving it), constructing novel mathematical model if necessary > Go back to the original real-world problem and to explain it. The emergence of electronic computers and the corresponding computational methods has greatly extended the intermediate step in the “generalized applied mathematics”, and a lot of tasks that had to be solved via experimental means due to the mathematical difficulty can now be solved by computational methods.

The theoretic research of the lattice Boltzmann method has undergone concrete progress due the efforts of a lot of people in many years. An enormous number of articles on the lattice Boltzmann method have appeared, and the earlier work can be found in the survey paper such as ref. [17]. But just like what we have said before, some qualitative progresses have made since then. At the same time there are many comparisons between the results obtained by LBM to those obtained by the experiments or other methods[18]. All these have demonstrated that even though LBM remains to be made perfect, it has proved to be an accurate and efficient computational method for the fluid mechanics. Meanwhile the computational software based on the lattice Boltzmann method has played an un-ignorable role in the real-world engineering as well as the research and development of the new products. This is closely related to the explicit and implicit advantages of LBM. Among them the main reason is that it can perform a large scale parallel computing of unsteady fluid problems. Besides, we think that the lattice Boltzmann method has at least the following obvious characteristics:

First of all, since it has the property that the Courant number equals to one, it has much less numerical error than the simple finite difference scheme of the same order. Thus under the same grid resolution, it can capture finer behavior of the turbulence. It offers some advantage to compute the high Reynolds number fluid problems. According to an estimate measure, the spatial resolution needed to solve a fluid problem directly is in the order of the Reynolds number. As for computation with turbulence modes or the large eddy simulations, the requirement on the spatial resolution is at least in the order of the square root of the Reynolds number. Usually the Reynolds number of the ordinary engineering problems is around millions. This means that we cannot perform direct simulation of a 3-dimensional fluid problem. Even if we conduct a large eddy simulation, the number above is still unreachable for the current computers. Therefore if some computational method can allow for a higher resolution, then it should be a more desirable candidate.

Secondly, the lattice Boltzmann method has many advantages in dealing with the boundary conditions of complicated geometry[19]. Firstly, its constant-speed-advection characteristic makes the near boundary process available by just using constant geometrical weights. Such information can be completed during the pre-processing stage before performing the simulation, thus makes the computing of extremely complicated geometrical objects more efficient. On the contrary, the grid generation for a complicated geometrical shape is a tedious job in the traditional computational fluid dynamics based on the Navier-Stokes equation. Secondly, what is more important is that it can deal with more general physical boundary conditions[20]. The familiar no-slip boundary condition can be interpreted as a macroscopic limit of the particles reflecting on the wall. But under a more general situation, a slip phenomenon is a natural physical process[21]. This gives a new way of realizing turbulence boundary layer models, i.e., we can make use of the appropriate slip process in the particle motion to realize the turbulent large eddy at the boundary. In essence, the core task of the boundary condition is to determine accurately the flow rate of basic physical quantities, such as the flow rate of density, momentum, and energy of the fluids passing the boundary. All these conditions can be realized accurately using some appropriate reflection process of particles. The resulting normal component of the momentum flux corresponds naturally to the fluid pressure, while the conventional boundary condition that depends on the velocity of fluid itself and the linear gradient of the flow field is based in essence on the Newtonian flow, i.e., the stress must depend linearly on the rate of strain. Since far away from the equilibrium, the characteristic of large eddy flow has much similarity to the finite Knudson number flow, including the nonlinear effect of the Non-Newtonian flow[22]. Thus a fundamental question on a deeper academic level is worthy of exploring: compared to the large eddy modeling of the Navier-Stokes equation, whether an extended kinetic theory is a more suitable description for the physics of large eddy turbulence?

Thirdly, the kinetic theory based Boltzmann equation has the property of entropy in statistical physics or the so-called H-theorem. Its existence offers a more robust parameter control on the stability (such as the lattice Boltzmann equation)[23]. Hence it is more generally applicable for stability analysis than linear stability. The latter is the conventional approach in performing stability analysis in computational fluid dynamics. Due to its uncertainty, it might lead to an over conservative choice of parameter or multiple repetition of computations. While for the lattice Boltzmann equation, its choice of parameters does not change due to the specific distribution of fluids. This is crucial in simulation of complex fluid flow applications.

Fourth, compared to the usual equations in fluid mechanics, the Boltzmann equation is more close to the physical reality. After resolving the computational complexity issue, the scheme starting from the Boltzmann equation allows for more direct simulation of more general physical mechanism. For example, as for the multiphase flow, the simulation of the non-Newtonian fluid stress-strain relation can be realized by simulating the microscopic mechanism that leads to these macroscopic physical phenomena.

Fifth, the kinetic equation has a simple form and easier computational scheme. All the nonlinear characteristic of fluids is contained in the local collision term. The description of the state far away from equilibrium can be realized by increasing truncation orders, as opposed to be realized via additional equations.

There are many examples of applying the lattice Boltzmann method to directly computing the fluid problems. Here we specifically mention one pertaining to flows around a cylinder[18]. Figure 1 is a typical picture of stream lines compared with experiment.

Figure 2 is flux vs. kundsen number in a micro-channel. At high kundsen unmbers, the results are apparently devicated from that of the Navier-Stokes equation.

For all such projects, the direct wind tunnel experiments are not only time-consuming and cumbersome, but also unable to measure some key factors or situations. Although the current method of CFD has not reached the extent to completely replace the real experiments as far as the accuracy and reliability are concerned, it has become an extremely important auxiliary tool, capable of providing more complete and detailed information of the properties of fluids under different situations, thus enabling people to obtain more rational knowledge of the problems of fluids. At the same time, the virtual experiments before production of real objects can not only save time and cut the cost, but also optimize the design on a larger scale by performing a simultaneous computing of various working situations. CFD, especially the lattice Boltzmann method has a broad area of applications and development space in the application of real engineering problems of fluids.

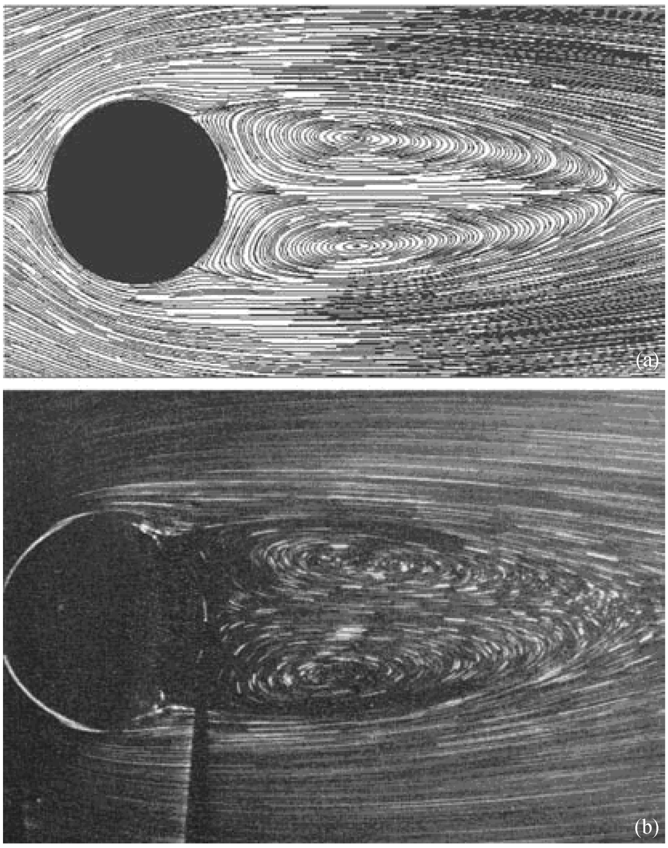

Figure 1[18] Flow passing a cylinder. (a) Comparison between the stream lines in a LBM simulation; (b) with streak lines in experiments.

Figure 3 is spatial distribution of surface pressure and stream lines (vorticity lines) of some aircraft model under a certain attack angle.

Figure 4 is the comparison of the computed results of lifting dray-force and torque The traditional N-S equation (Fluent) can only calculate the attack angle up to 50°. The results computed by using Euler equation by some institute are displayed as well.

4 Discussion

Here we would like to discuss the importance and necessity of developing scientific CFD software. The development and popularity of software industry is a main component of the Third Technology Revolution (information technology) in the world, it has led to the unprecedented

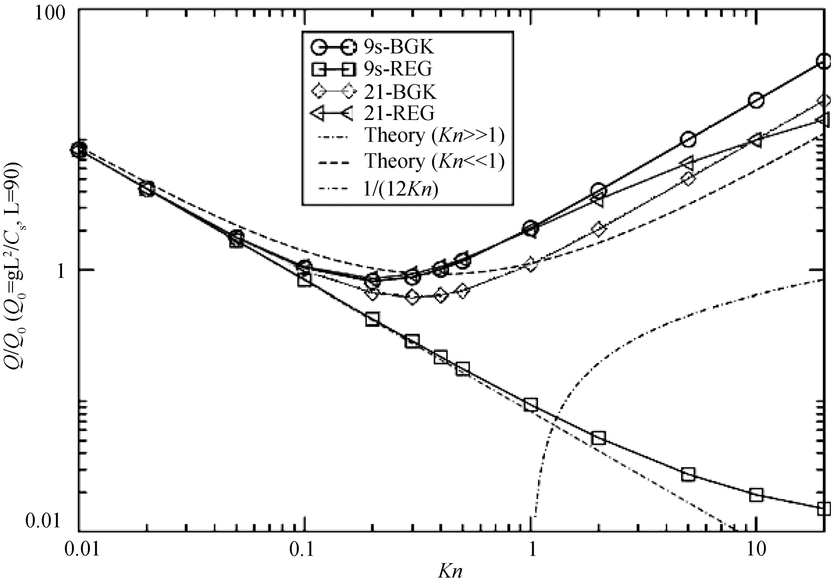

Figure 2[13] Knudsen paradox: flux vs. Knudsen number in a micro-channel. Streight line is the result of Navier-Stokes; dashed lines are the kinetic solutions in limiting conditions; symbols are simulation results of several lattice Boltzmann models.

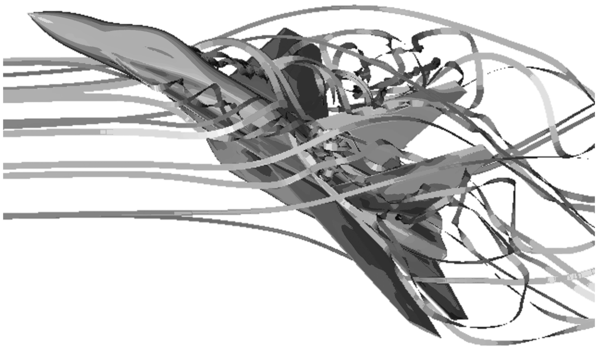

Figure 3[24] Surface pressure and spatial distribution of stream lines obtained using LBM-based CFD software PowerFLOW for model aircraft CT1 at the attack angle of 30°.

advancement of the society. Developing scientific software has a fundamental difference from developing ordinary software. The main characteristic lies in that it is formed by intertwining multiple cutting edge fields. Its key components include advanced physical models, expedient computational methods, the latest software engineering integration and accommodation (such as automatic mesh generation, paralleling computing, load allocation, front end and back end processing, etc.), deep understanding of hard application problems of major industry (such as the understanding of realistic aero-dynamics phenomena), as well as the advanced management and operation. All this needs a close integration of the relevant fields, and forms an organically organized team to reach such a systematic objective. Here we only want to emphasize the importance of pre- and post-processing and the user interface. Maybe we can use a metaphor that might not be so appropriate: before the appearance of the Windows software, the personal computers were merely specialized tools for a few computer specialists. The popularization of personal

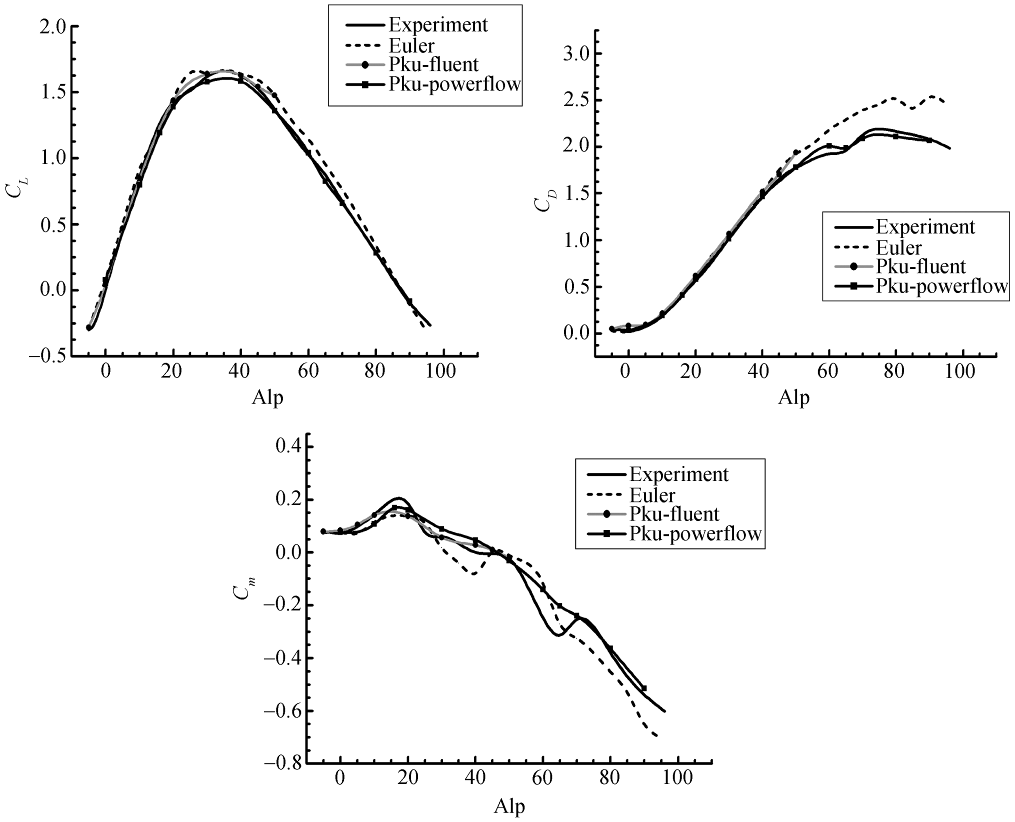

Figure 4 The comparison of the computed results of lifting drag-force and torque coefficient with the experiments[25].

computers after the appearance of the Windows software not only promoted the advancement of many scientific and technical fields, but also made a great impact on various areas in society as well as culture. Therefore, user friendly capabilities of pre- and post-processesing and user interface are a necessity for the scientific software to transform from the product of a computer specialist to a useful tool for engineers, since a senior aero-dynamic engineer may not be a specialist in computational mathematics or software programming. The central difficulty that needs to be overcome in the development of scientific software lies in the disconnection between the scientific research and the software development as well as the real application. For scientists, the traditional way of doing research is through individual effort. For example, research on various turbulence models is a separate behavior under most circumstances. The corresponding tests are also only limited to some typical benchmark cases of fluid flows. Even for the scientific computing, software is usually developed by the field researchers themselves. Except to serve a few people of themselves, it can not reach the standard by the professional software developers, let alone considering the universal adaptability and mobility. There is also the hidden issue that scientists are not used to or not interested in the transformation of technology into real-world application products. From another point of view, the software developers and business organizers do not have deep understanding of physics and mathematics, resulting in having a sense of helplessness. All these are fundamental problems faced by developing scientific software compared to ordinary software. Thus, we recognize that the real intertwining and organic integration of the frontier fields is in itself an important field that we must face. We can experience such a metaphor: No matter how high the playing skill of a musician himself, the effect that they play individually can never be comparable with the symphony orchestrated by them playing together.

In summary, CFD as a field of science is extremely promising and the prospective of CFD is extremely optimistic. The modern industrial design has tended to use computers as an auxiliary tool (Computer Aided Engineering, CAE). The development and distribution of scientific software is a direction of the future for the high-tech industry. The impact on the science and technology due to the maturity of its development is unparalleled by the traditional method.

As a final note, we discuss the teaching related issue: the popularity of scientific CFD software has also offered an efficient tool for teaching. Students are able to perform experiments as long as they enter a virtual lab, thus establish an intuitive as well as rational understanding of the physical characteristics of fluid mechanics. Science is just like sports, its advancement and enhancement are closely related to its popularity. Said in a folklore language, this is the impact caused from few people's "orchestra" to the mass's "Rock & Roll". Computers have replaced quite a part of the brain labor of the people, and the key to popularize the application of computers lies in the cultivation of high-level scientific and technology experts. From the point of view of teaching, it is to speed up the cultivation of experts of applications of computing technology. No improvement can be accomplished unless there is a popularization of computers. The article titled "Tsian's idea on the technology science directs the construction of engineering mechanics seminar in the Tsinghua University" details a revolutionary method to develop the high level experts in the mechanics –"Seminar in Mechanics", and it can be a good reference to cultivate the computer-application experts nowadays. Professor Tsian said very clearly in the article: "In 1960s, the chip electronic computers capable of fast computing emerged,which caused a revolution in the computing capability. Now the computers capable of trillion floating point operations per second have also appeared. As the mechanical computing capability progresses, to use the theoretical method of mechanics in the design of industry has become the main path, whereas experimentation will be a supplement. Thus, if we look forward to the twenty-first century, mechanics combined with electronic computers will be the main tool in the new design of industry, and the design of industrial model will be displayed by the electronic computers together with the pictures. All is virtual, not real, so we should call it 'virtual prototyping', at last it is the manufacturing of real products. The direction proposed by Professor Tsian in those days is the task of the era faced by us today.

1 Wilcox D. Turbulence modeling for CFD. DCW Industries, Inc, 1993

2 Chen H, Kandasamy S, Orszag S, et al. Extended boltzmann kinetic equation for turbulent flows. Science, 2003. 301—633

3 Frisch U, Hasslacher B, Pomeau Y. Lattice-gas automata for the Navier–Stokes equation. Phys Rev, 1986, 56: 1505

4 Wolfram S. Cellular automaton fluid 1. Basic theory. J Stat Phys, 1986, 45: 471—526

5 McNamara G R, Zanetti G. Use of the Boltzmann equation to simulate lattice-gas automata. Phys Rev Lett, 1988, 61: 2332

6 Chen S, Chen H, Martinez D, et al. Lattice Boltzmann model for simulation of magneto hydrodynamics. Phys Rev, 1991, 67: 3776

7 Chen H, Chen S, Matthaeus W H. Recovery of Navier–Stokes equations using a lattice- gas Boltzmann method. Phys Rev A, 1992, 45: R5339—R5342

8 Qian Y H, d'Humieres D, Lallemand P. Lattice BGK models for Navier–Stokes equation. Europhys Lett, 1992, 17: 479—484

9 Bhatnagar P, Gross E, Krook M. A model for collision processes in gases. I. small amplitude processes in charged and neu-tral one-component system. Phys Rev, 1954, 94: 511—525

10 Grad H. On the kinetic theory of rarefied gases. Commun Pure Appl Math, 1949, 2: 331

11 Shan X, He X. Discretization of the velocity space in solution of the Boltzmann equation. Phys Rev, 1998, 80: 65

12 Shan X, Yuan X F, Chen H. Kinetic theory representation of hydrodynamics: a way beyond the Navier-Stokes equation. J Fluid Mech, 2006, 550: 413

13 Zhang R, Shan X, Chen H. Efficient kinetic method for fluid simulation beyond the Navier-Stokes equation. Phys Rev E, 2006, 74: 046703

14 Shan X, Chen H. A general multiple-relaxation-time Boltzmann collision model. Int J Mod Phys, C, 2007 (to appear)

15 Shan X, Chen H. Lattice Boltzmann model for simulating flows with multiple phases and components. Phys Rev E, 1993, 47: 1815－1819

16 Shan X, Chen H. Simulation of non-ideal gases and liquid–gas phase transitions by lattice Boltzmann equation. Phys Rev E, 1994, 49: 2941－2948

17 Chen S, Doolen G. Lattice Boltzmann method for fluid flows. Annu Rev Fluid Mech, 1998, 30**:** 329－364

18 Li Y, Shock R, Zhang R, et al. Numerical study of flow past an impulsively started cylinder by the lattice-Boltzmann method. J Fluid Mech, 2004, 519: 273

19 Chen H, Teixeira C, Molvig K. Realization of fluid boundary conditions via discrete Boltzmann dynamics. Intl J Mod Phys, 1998, 9(8): 1281

20 Cercignani C. Theory and Application of the Boltzmann Equation. Scottish: Elsevier, 1975

21 Chen H, Orszag S, Staroselsky I. Macroscopic description of arbitrary Knudsen number flow using Boltzmann-BGK kinetic theory. J Fluid Mech, 2007, 574: 495

22 Chen H, Orszag S, Staroselsky I, et al. Expanded analogy between kinetic theory of fluids and turbulence. J Fluid Mech, 2004, 519: 307

23 Succi S, Karlin I, Chen H. Role of the H theorem in lattice Boltzmann hydrodynamic simulations. Rev Mod Phys, 2002, 74(4): 1203

24 Li X, Zhang X, Wang Y, et al. Lattice Boltzmann method used for the aircraft characteristics computation with high angle attack. Acta Aerod Sinica, 2007, 25 (accepted)

25 Wu J, Fan Z, He Z, et al. Research on the tests technology at high angles of attack in 2. 4m transonic wind tunnel. Exp & Meas in Fluid Mech, 2004, 18(4): 43