

The development and application of CFD technology in mechanical engineering

Yufeng Wei *

Beijing Institute of Technology, Beijing, China

*Corresponding author e-mail: weiyufeng715@163.com

Abstract. Computational Fluid Dynamics (CFD) is an analysis of the physical phenomena involved in fluid flow and heat conduction by computer numerical calculation and graphical display. The numerical method simulates the complexity of the physical problem and the precision of the numerical solution, which is directly related to the hardware speed of the computer and the hardware such as memory. With the continuous improvement of computer performance and CFD technology, it has been widely applied to the field of water conservancy engineering, environmental engineering and industrial engineering. This paper summarizes the development process of CFD, the theoretical basis, the governing equations of fluid mechanics, and introduces the various methods of numerical calculation and the related development of CFD technology. Finally, CFD technology in the mechanical engineering related applications are summarized. It is hoped that this review will help researchers in the field of mechanical engineering.

1. Introduction

With the rapid development of computer and computational fluid dynamics (CFD), CFD numerical simulation has become an important auxiliary method of engineering design [1, 2]. Computational fluid dynamics has been widely used in high-tech products such as aerospace, large-scale power equipment and nuclear reactors, and has promptly expanded to various types of general machinery products related to heat and flow, such as fluids, chemical machinery and equipment.

CFD came into being after the Second World War. In the 1960s, CFD became an independent discipline with the introduction of a CFD-related magazine “Journal of Computational Physics” and American scholar Harlow and Welch's presentation of staggered grids. After this, CFD technology developed rapidly: American scholars Thompson, Thames and Mastin proposed differential equations to generate aptamer coordinates; Lenard published the famous QUAICK format, which is a third-order precision convective discrete format; giant computer promotes the development of parallel algorithm, direct numerical simulation (DNS), research of turbulence with large eddy simulation (LES) and so on. [4]

In general, the experimental method is often limited by model size, flow field disturbances, personal safety, and measurement accuracy, while CFD compensates for these deficiencies. It involves many advantages, such as a shorter duration, easier operation, lower cost, and higher repeatability, which gives an access to selecting different flow parameters to perform various numerical experiments and to



carrying out multi-program comparison. And with a good flexibility and adaptability, it is also not restricted by physical models and experimental models.

In all, CFD, as an analog and simulation tool, has been widely used to study various transmission processes, including fluid flow, heat transfer and mass transfer and has been extensively recognized by researchers and users. It consequently plays an essential role in hydraulics scientific research and engineering design.

This paper is aimed at introduces the theoretical basis of CFD numerical simulation, the concrete example that CFD uses in engineering practice, the research achievements of CFD in hot engineering field in recent years, and the challenges as well as application prospects of CFD in mechanical engineering.

2. The theoretical basis and computational process of CFD

CFD can be regarded as a numerical simulation of the flow under the control of the basic equations of flow (mass conservation equation, momentum conservation equation, energy conservation equation), CFD uses the computer as a simulation method to use some computational techniques to find the complex problems of fluid mechanics and discrete numerical solution. Numerical solution is a discrete approximation of the calculation method, which generally follow the steps shown in Figure 1.

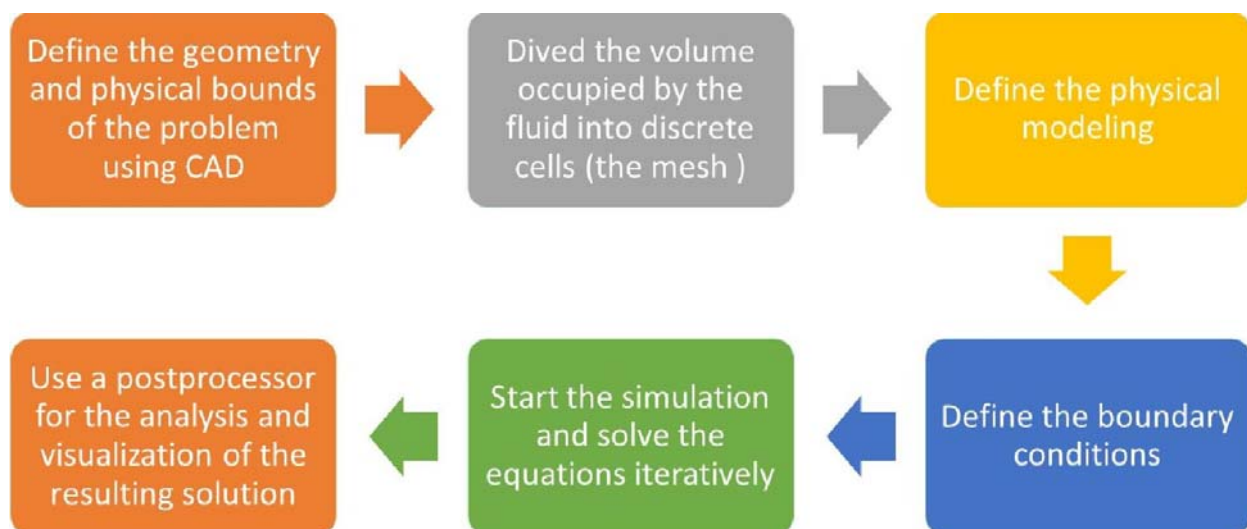


Figure 1. CFD numerical method for solving flowchart

3. The governing equations of fluid mechanics

Fluid flow is governed by conservation laws. The governing equation is a description of the conservation law, which includes the mass conservation equation (also called the continuity equation), the momentum conservation equation and the energy conservation equation.

3.1. Mass conservation equation

The mass conservation equation, also known as the continuity equation, represents an increase in the mass in the circulation element per unit time, equal to the net mass flowing into the micro-body at the same time interval. The equation is:

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{v}) = S_m, \quad (1)$$

div represents divergence, that is $\text{div}(a) = \frac{\partial a_x}{\partial x} + \frac{\partial a_y}{\partial y} + \frac{\partial a_z}{\partial z}$. Equation (1) is the general form of the mass conservation equation and is suitable for compressible flow and incompressible flow. And ρ is the density, t is the time, v is the velocity vector, and S_m is the mass added to the continuous terms, such as evaporation or heterogeneous reaction of the dropping.

3.2. Momentum conservation equation

The law of conservation of momentum is also the basic law that any flow system must satisfy, and its essence is Newton's second law. The law can be expressed as: the rate of change of the momentum of the fluid to time in the micro-element is equal to the sum of the forces acting on the micro-element. In the inertial coordinate system, the momentum equation is:

$$\begin{cases} \frac{\partial(\rho u)}{\partial t} + \text{div}(\rho u \vec{v}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \\ \frac{\partial(\rho v)}{\partial t} + \text{div}(\rho v \vec{v}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y \\ \frac{\partial(\rho w)}{\partial t} + \text{div}(\rho w \vec{v}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z \end{cases}, \quad (2)$$

Where p is the pressure on the microfluidic body; τ_{xx} , τ_{yx} and τ_{zx} are the components of the viscosity that acts on the surface of the micro-body due to the viscosity of the molecule; f_x , f_y and f_z are the components of the volume force (f) in the three directions acting on the unit mass microbody.

3.3. Energy conservation equation

The energy conservation equation is the basic law that must be satisfied by the flow system that contains heat exchange and it can be expressed as: the increase rate of energy in the micro-element is equal to the net heat flow into the micro-body and the work of the body and the force on the micro-body, and the law is actually the first law of thermodynamics.

When there is no heat source in the flow field, the differential energy equation is:

$$\begin{aligned} \frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho E \vec{V}) &= \rho F \cdot \vec{V} - \nabla \cdot (\tau \cdot \vec{V}) \\ \rho \dot{E} &= \rho e + \frac{\rho}{2} V^2 \end{aligned}, \quad (3)$$

Where E is the total energy, and e is the internal energy.

4. Numerical calculation method

In the past few decades, there has developed a variety of numerical solutions, their main difference lies in the regional discrete approach, the equation of discrete and algebraic equations to solve the three links. The numerical methods used in CFD numerical solution are: finite difference method, finite volume method and finite element method. The most widely used CFD field is the limited volume method.

4.1. Finite Difference Method

Finite difference method (FDM) is the earliest method of computer numerical simulation. The method divides the solution region into a separation grid, and uses a finite number of grid nodes instead of a continuous solution area. The finite difference method is used to discretize the derivative of the control equation with the difference of the function value on the grid node to establish the algebraic equations with unknown values on the grid nodes. The method is an approximate numerical solution that directly turns the differential problem into algebraic problem. The mathematical concept is intuitive and simple to express, which is an earlier and more mature numerical method.

4.2. Finite Volume Method

Finite Volume Method (FVM), also known as the control volume method, is a commonly used method for spatial discretization. It is mainly from the equations of conservation type, and makes integral in its control volume, and further solves the conservation equation in integral form. All control differential equations have a common form, this form of consistency is the basis of a common.

4.3. Finite Element Method

The finite element method (FEM) is based on the variation principle and the weighted residual method. The basic solution idea is to divide the calculation area into a finite number of non-overlapping units. In each unit, select some of the appropriate nodes as the interpolation point for the function. Then, the variables in the differential equation are rewritten as a linear expression (shape function) consisting of the node value of each variable or its derivative and the selected interpolation function. Finally, the differential equation is discretized and solved by means of the variation principle or the weighted residual method. Different finite element methods are used to form different weight functions and interpolation functions.

5. The development of CFD related technologies

5.1. Grid technology

Grid technology, is the entire flow field for meshing, and thus related to the calculation. The principle [5] is the equation of the control pointing on the predetermined region (finite difference method) region (finite element method and the finite volume method) or discrete, so that it is converted into algebraic equations which is defined at each grid point or region. Moreover, if it is a nonlinear system, it is usually also linearized and then use the method of linear algebraic iterative solution. By calculating the position variables at these points, the physical component distribution over the entire region is obtained. The so-called grid, which is, in accordance with certain laws distributed in the flow field of discrete points of the collection. In the CFD calculation process, grid technology is very important, because it is directly related to the success of the CFD model calculation results.

In the 1980s-1990s, there is the rapid development of high-performance computer era, but also CFD theory and technology in the golden period of development. Structured meshes, unstructured grids, hybrid grids, deformed grids, overlapping or splicing grid techniques [6-8] are all generated during this period. Improving the accuracy and calculation time, they greatly enhance the ability of CFD to solve complex problems, and further expand the CFD computing range and application areas. In short, they laid the CFD in the field of engineering status.

5.2. The development of CFD calculation software

With the widespread use of CFD, its commercial software has been fully developed. Researchers can use the software to calculate the flow field, and with the help of these software to improve work efficiency. At present, the CFD software commonly used in engineering calculation is PHOENICS, CFX, STAR-CD, FIDAP, FLUENT, etc., among them, FLUENT software is the most widely used.

PHOENICS is the first set of commercial software about computational fluid dynamics and heat transfer in the world, which was presented by the scholars Spalding and Patanka, and was developed

primarily by the British CHAM [9]. CFX was developed by British AEA Technology. CFX software is now widely used in aerospace, rotating machinery, energy, petrochemical, machinery manufacturing, automotive, biotechnology, water treatment, fire safety, metallurgy, environmental protection and other fields [10]. CFX is unique in that it uses a finite volume method based on finite element method, which guarantees numerical accuracy. The STAR-CD is the first commercial software package for fluid analysis using a completely unstructured grid generation technology and a finite volume method to study the complex flow in the industrial sector [11]. FIDAP is based on the finite element method and can be used to solve the problems of various flows and turbulence in polymer, film coating, biomedicine, semiconductor crystal growth, metallurgy, glass processing and other fields. It provides an accurate and effective solution to problems involving fluid flow, heat transfer, mass transfer, discrete phase flow, free surface, liquid-solid phase transition, fluid-solid coupling, and so on. FLENT is currently the most comprehensive, most widely used CFD software, its design based on "CFD computer software group concept." For each physical problem of physical characteristics, FLENT will be used for its numerical solution, so that the software's computing speed, stability and accuracy are to achieve the best. Different areas of computing software are combined to become CFD software group, so as to efficiently solve the complex flow of computing problems in various fields. Using a unified post-processing tool and being able to calculate the flow field, heat transfer and chemical reactions, all different software can be exchanged between the values directly, which established the foundation for generalization of FLUENT [12].

6. The application of CFD technology in mechanical engineering

6.1. The application of CFD in aeronautical engineering field

In the field of aeronautical engineering, CFD's contribution and achievements are remarkable. Under the support of the more complete fluid mechanics theory and large-scale parallel computing technology, CFD is applied almost exclusively to every field of aeronautical aerodynamics research:

Pneumatic integrated optimization design based on CFD, greatly improved the aerodynamic shape to optimize the design quality, which greatly shorten the design cycle and improve the aerodynamic shape selection of aircraft design efficiency. At the same time, the integrated optimization technology based on CFD and other disciplines is also applied in the design of aeronautical aircraft, which plays an important role in the integration design of pneumatic stealth and aerodynamic structure [13-19].

The numerical simulation of static and dynamic aerodynamic elasticity of aircraft based on CFD play an important role in the design of aircraft type frame shape, flutter characteristic analysis and flutter boundary assessment [20-26]. It provides effective guidance for structural strength/stiffness design, rigid/center of gravity configuration, and it is coupled with classical control theory and modern control theory, and further provides a very effective simulation method for chatter suppression control rate design [27].

The aeronautical aerodynamic noise prediction method is based on CFD, including direct and mixed methods. The hybrid method can realize the acoustic optimization design of some aircraft components under the existing calculation conditions. The direct method can verify the performance of the noise reduction design and carry on the related mechanism research [28-30].

Digital flight and control law validation is also one of the ways applying CFD. Based on the Navier-Stokes equations, the numerical solution and deformation, overlapping grid technology and six-degree-of-freedom equations are coupled, so that the numerical model of aircraft-wide flight is beginning to appear on the stage of aeronautical engineering. The world's aircraft development departments have invested in scientific research for digital simulation research, and to some extent applied to the actual model [31-33].

CFD-based multi-body separation safety boundary assessment, greatly improved the efficiency of the relevant research, and significantly reduces the research costs and risks. It has played an important role in predicting the trajectory of the movement of the object and the assessment of the safety boundary.

It has been successfully applied in the field of plug-in/buried airborne missile separation [34], sub-shells [35] and civil aircraft ice [36].

6.2. The application of CFD in vehicle engineering field

The rise of CFD technology has promoted the development of experimental research methods and theoretical analysis methods in the field of vehicle engineering, and put forward more basis for the simplification of the flow model, which has made many analysis methods developed and perfected. Nowadays, CFD technology has become a basic tool in the field of vehicle engineering [37] for all aspects of automotive design.

6.2.1 Based on CFD, the numerical simulation of the external flow field of automobile is directly applied to the exterior design of automobile. In the early stages of body design, CFD can be crossed with computer-aided design to predict, analyze and optimize automotive aerodynamic characteristics, and further provide the basis for vehicle selection and modeling [38]. In the early 1980s, the simulation object of CFD was limited to the basic shape of the body. With the development of computer technology and turbulence theory, CFD has been applied to the simulation and design of complex body such as rearview mirror, spoiler, complex floor, mobile ground, engine compartment, wheel and so on. Moreover, CFD can also carry out the wind stability and cross wind over the uniqueness of the two cars meet the transient aerodynamic characteristics of the simulation [39-41].

6.2.2 The CFD-based noise control technology is applied to the engine noise, tire noise and wind noise controlling fields. For the noise generated by the engine, the design of exhaust muffler is the main means of noise control. CFD is used to analyze the flow field and temperature field of the engine under typical operating conditions. It uses the surface temperature of sensitive parts such as discarded streamlines, fuel tanks and side skirts to evaluate the optimal design of the muffler so as to achieve the effect of controlling the engine noise [42-43]; the noise generated by the tire, during tire rolling outflow field characteristics is the main cause of the noise. Based on the theory of fluid-solid coupling, the researchers couple the fluid and solid of the outer wall surface when the tire is rolling, so that the finite element method and the CFD method are combined to simulate the flow field of the tire rolling process. As a result, the transient aerodynamic characteristics of the rolling tires were proposed, which laid the foundation for the control of the noise produced by tires [44]; The noise caused by wind vibration is mainly caused by the opening of the car side window or skylight. Simulation of CFD simulation of vehicle side windows, skylights, and wind noise generated by the accurate prediction has been achieved. Further, the influence of the factors such as the front window, the rear window, the speed, the compartment volume, the yaw angle, the number of passengers and so on to the noise has also been found, which has made the researchers propose a variety of ways to reduce the the noise [45-46]. All in all, CFD has made a significant contribution to vehicle noise control. Unlimited potential in CFD will be tapped out and it will change the area a lot in the future.

7. Conclusion

CFD technology is an advanced virtual design method, which simulates the influence of the structure on its performance by simulating the flow field inside the mechanical equipment. It can simulate the flow field inside the mechanical equipment, including the speed distribution, pressure distribution, concentration and temperature distribution of the flow of various detailed and intuitive information. This paper introduces the theoretical basis and calculation method of CFD, expatiates the application status of CFD technology in mechanical engineering, summarizes a series of key technologies of CFD in engineering application, and some difficulties and challenges. In the future, the development of CFD technology in the field of mechanical engineering should focus on the development of multidisciplinary coupling calculation, general engineering software and the extraction of massive flow field data, so as to improve the calculation precision and efficiency of CFD and the ability to process data.

References

- [1] JD Anderson. Computational Fluid Dynamics: The Basics with Applications [M]. Beijing: Tsinghua University Press, 2002.
- [2] Zhang W Z, Yu S R, Zhang X H, et al. Numerical simulation and experimental analysis of interior flow field in control valves [J]. Journal of Lanzhou University of Technology, 2008, 34(3): 65-68.
- [3] Cao Y H. Modern Helicopter Rotor Aerodynamics [M]. Beijing: Beijing University of Aeronautics and Astronautics Press, 2015.04: 77.
- [4] Yao R T, Guo D P. Computational Fluid Mechanics Foundation and STAR-CD Engineering Application [M]. Beijing: National Defense Industry Press, 2015.07: 2.
- [5] Zhang C. Based on the CFD method of sliding bearing dynamic characteristics analysis [D]. Southeast University, 2011.
- [6] V Venkatakrishnan. A perspective on unstructured grid low solvers [J]. AIAA Journal, 1995, 34(3): 533-547.
- [7] Y Kallinderis, A Khawaja, H McMorris. Hybrid prismatic/tetrahedral grid generation for viscous flows around complex geometries [J]. AIAA Journal, 1996, 34(2): 291-298.
- [8] Ye Z Y, Yang Y N, Zhong C W. The method investigation in unstructured grid generation technique [J]. Aeronautical Computer Technique, 1998, 28(1): 44-47.
- [9] Zhai J H. Review of commercial CFD software [J]. Journal of Hebei University of Science and Technology, 2005, 21(9): 35-36.
- [10] Yao R T, Guo D P. Computational Fluid Mechanics Foundation and STAR-CD Engineering Application [M]. Beijing: National Defense Industry Press, 2015: 143-144.
- [11] Han Y X, Jin H. The introduction of STAR-CD software [J]. Gansu Science and Technology, 2005, 21(9): 35-36.
- [12] Li Y, Liu Z Y, An Y R. A brief introduction to Fluent-a general purpose CFD code [J]. Journal of Hydrodynamics, 2001, 16(2): 254-258.
- [13] Sobieszczanski-Sobieski J. Sensitivity analysis and multidisciplinary optimization for aircraft design: Recent advances and results [J]. Journal of Aircraft, 1990, 27(12): 993-1001.
- [14] Yu X Q, Ding Y L. Multidisciplinary design optimization a survey of its algorithms and applications to aircraft design [J]. Acta Aeronautica et Astronautica Sinica, 2000, 21(1): 1-6.
- [15] Xia L, Gao Z H, Li T. Investigation of integrated multi-disciplinary and multi-objective optimization of the aircraft configuration design method [J]. Acta Aerodynamica Sinica, 2003, 21(3): 275-281.
- [16] He L S, Wang S H, Zhang Y Z. The new algorithm for aircraft multidisciplinary integrated design[J]. Acta Aeronautica et Astronautica Sinica, 2004, 25(5): 465-469.
- [17] Su W, Gao Z H, Xia L. Multi-objective optimization design of aerodynamic configuration constrained by stealth performance [J]. Acta Aerodynamica Sinica, 2006, 24(1): 137-140.
- [18] Tang W, Gui Y W, Wang A L. Proposal of thermal configuration optimization design for a maneuverable vehicle [J]. Journal of Astronautics, 2009, 30(5): 1803-1807.
- [19] Viana F A C, Simpson T W, Balabanov V, et al. Metamodeling in multidisciplinary design optimization: how far have we really come [J]. AIAA Journal, 2014, 52(4): 670-690.
- [20] Yang Z C, Xia W. Analytical models, numerical solutions and advances in the study of panel flutter [J]. Advances in Mechanics, 2010, 40(1): 81-98.
- [21] Zhang W W, Zhong H S, Xiao H, et al. Review and prospect of flutter boundary prediction methods for flight flutter testing [J]. Acta Aerodynamica Sinica, 2015, 36(5): 1367-1384.
- [22] Huang J, Gao WZ, J Bai, Z Zhou, K Zhao. Aircraft jig shape design based on radial basis functions and Delaunay graphic mapping [J]. Acta Aerodynamica Sinica, 2014, 32(3): 328-333.
- [23] Liang Q, Yang Y N, Ye Z Y. Analysis of jig-shape design for elastic wing [J]. Journal of Northwestern Poly-technical University, 2002, 20(2): 262-264.
- [24] Huang J T, Gao Z H, Bai J Q, et al. Aircraft jig shape design based on radial basis functions and Delaunay graphic mapping [J]. Acta Aerodynamica Sinica, 2014, 32(3): 328-333.

- [25] Yang Z C, Xia W. Analytical models, numerical solutions and advances in the study of panel flutter [J]. *Advances in Mechanics*, 2010, 40(1): 81-98.
- [26] Zhang W W, Zhong H S, Xiao H, et al. Review and prospect of flutter boundary prediction methods for flight flutter testing [J]. *Acta Aerodynamica Sinica*, 2015, 36(5): 1367-1384 (in Chinese).
- [27] Xiang J, Yan Y, Li D. Recent advance in nonlinear aero-elastic analysis and control of the aircraft[J]. *Chinese Journal of Aeronautics*, 2014, 27(1): 12-22.
- [28] Wang M, Freund J B, Lele S K. Computational prediction of flow-generated sound [J]. *Annual Review of Fluid Mechanics*, 2006, 38: 483-512.
- [29] Farassat F, Casper J H. Towards an airframe noise prediction methodology: Survey of current approaches [C]//44th AIAA Aerospace Sciences Meeting and Exhibit. Reston: AIAA, 2006.
- [30] Wagner C, Huttl T, Sagaut P. Large-eddy simulation for acoustics [M]. London: Cambridge University Press, 2007: 441.
- [31] Tao Y, Fan Z L, Wu J F. CFD based virtual flight simulation of square cross-section missile with control in longitudinal flight [J]. *Chinese Journal of Theoretical and Applied Mechanics*, 2010, 42(2): 169-176.
- [32] Da X Y, Tao Y, Zhao Z L. Numerical simulation of virtual flight based on prediction-correction coupling method and chimera grid [J]. *Acta Aeronautica et Astronautica Sinica*, 2012, 33(6): 977-983.
- [33] Chang X H, Ma R, Zhang L P, et al. Study on CFD-based numerical virtual flight technology and preliminary application [J]. *Chinese Journal of Theoretical and Applied Mechanics*, 2015, 47(4): 596-604.
- [34] Liu G, Xiao Z Y, Wang J T, et al. Numerical simulation of missile air-launching process under rail slideway constraints [J]. *Acta Aerodynamica Sinica*, 2015, 33(2): 192-197.
- [35] Zhang Y D, Ji C Q. The numerical simulation of submunition separation processes from dispenser [J]. *Acta Aerodynamica Sinica*, 2003, 21(1): 47-52.
- [36] Wang J T, Yi X, Xiao Z Y, et al. Numerical simulation of ice shedding from ARJ21-700 [J]. *Acta Aerodynamica Sinica*, 2013, 31(4): 430-436.
- [37] Huang T. Preliminary computational investigation of vehicle aerodynamics [J]. *Journal of Hunan University*, 1995, 22(1): 477-82.
- [38] Jiang L H, Gu Z Q. The application of CFD methods to automobile aerodynamics [J]. *Journal of Hunan University*, 1997, 24(4): 52-56.
- [39] Shen J, Fu L M, Li M H, et al. CFD Software and the application of CFD in the field of automobile[J]. *Automobile Research and Development*, 2000, 5: 26-28.
- [40] Zhang Y J, Lu Z H, Xu S A. A review on numerical simulation of automotive aerodynamics [J]. *Automotive Engineering*, 2001, 23(2): 82-91.
- [41] Song Z H, Zhu B Z. The development situation of research on the flow simulation around automobile with the application of CFD [J]. *Automobile Science and Technology*, 2002, 1: 5-7.
- [42] Zou X H. Simulation the vehicle muffler with CFD software and development a test-bed for evaluating the pressure loss of the vehicle muffler [D]. *Wuhan University of technology, Power Mechanical Engineering*, 2007, 5.
- [43] Jiang G F. CFD design optimization for the silencer exit [J]. *Automobile Science and Technology*, 2011, 6: 34-47.
- [44] Liu C C. Fluid dynamics analysis of tire under rolling condition [D]. *Donghua University. Mechanical design and theory*. 2014, 1.
- [45] Gu Z Q. Review of CFD simulation on vehicle wind buffeting [J]. *Noise and Vibration Control*, 2007, 4: 65-68.
- [46] Wang N, Gu Z Q, Liu S C, et al. Wind buffeting noise analysis and control for high-speed vehicle side-windows [J]. *Journal of Aerospace Power*, 2013, 28(1): 112-119.

