Fluid Dynamics and Material Processing FDMP, vol.15, no.4, pp.285-291, 2019

Review of the Special Issue on EFD and Heat Transfer

Liang Tian[1] and Zhigang Fang[2, 3, *]

The design of an aircraft's aerodynamic configuration has a direct influence on the flight performance and quality, not to mention a decisive influence on flight safety, efficiency, and economic benefits. The fundamentals of aerodynamic design involve the inverse problem of aerodynamics. Before the 1960s, aerodynamic design depended largely on a large number of wind-tunnel tests combined with the engineering experience of the designers, and employing the so-called trial-and-error method. Along with the development of modern computing technology, computational fluid dynamics (CFD) has played an increasingly important role in aerodynamic calculation, analysis, and design. For the purposes of solving potential flow equations, Euler equations, and numerical solutions of N-S equations, the number of grids (or meshes) involved in the computation has multiplied from what was tens of thousands to nearly a hundred million today. Such computations support the simulations of complex flow fields of 3D wing models, wing-body combinations, engines, and even the whole aircraft. However, in cases involving complex geometry as well as shock waves, separations, and other complex flow problems, CFD is still facing challenges in terms of computational complexity and accuracy. At present, aerodynamic design optimization based on high-credibility CFD mainly focused on academic research. To provide CFD aerodynamic design optimization with a unified platform for research and exchange, European and American countries have already begun to customize standard computation samples and research projects. In the United States, American Institute of Aeronautics and Astronautics (AIAA) organized the Aerodynamic Design Optimization Discussion Group (ADODG) in 2014 and published a series of standard problems of aerodynamic optimization in succession for researchers to optimize the standard model; in Europe, to look deeper into the capacity of the agent model-based method of global optimization in aerodynamic design, GARTEUR Action Group was founded early in 2013. Development directions for the methods of aerodynamic configuration design included the inverse design and optimization design methods, among many others. Extensive research on these methods have also been done by scholars at home and abroad. The inverse design method allows one to obtain the geometries of the corresponding wings or others parts through the given information about flow fields. The two-dimensional airfoil inverse design was first achieved by Lighthill with the method of conformal mapping. Nearing the end of last century, the

[1] School of Naval Architecture and Navigation, Wuhan Technical College of Communications, Wuhan, 430065, China.

[2] Hubei Key Laboratory of Advanced Technology of Automotive Components, Wuhan University of Technology, Wuhan, 430070, China.

[3] School of Automotive Engineering, Wuhan University of Technology, Wuhan, 430070, China.

[*] Corresponding Author: Zhigang Fang. Email: Zhigang_Fang@whut.edu.cn.

application of conformal mapping has not only branched out to solving three-dimensional wing models, but also shifted from solving incompressible flow models to compressible flow models, with specific methods including residual correction, surface curvature, etc. With such methods, scholars in China has made great achievements in the design of supercritical airfoils and wings. Into the 21st century, with the development of data mining technology, the inverse design method of POD airfoil is improved based on POD and data filling. The inverse design method features high computational efficiency and a smaller amount of CFD computing; however, an ideal target pressure distribution is still hard to obtain. In contrast to the inverse design method, the optimization design method combines CFD with the optimization theory and converts the design problems into solution of extreme values of the objective function under certain geometric and physical constraints; aided by a high-speed computer, it can realize more automatic and robust aerodynamic design. The optimization method includes gradient and evolutionary algorithm-based optimization and agent model-based global optimization. The simplest of the gradient method is one in which finite-difference method is used; however, since the computational complexity grew as the number of design variables increased, it was only used in cases of simple layout and fewer design variables. The introduction of the adjoint method, on the other hand, significantly reduced the time needed for the gradient calculation. Since the introduction of control theory to aerodynamic design for the first time, foreign research groups, represented by NASA Ames research center, Stanford University and University of Michigan, as well as many aviation research colleges and institutions at home have carried out systematic and in-depth research on the adjoint method. The most representative action of the evolutionary algorithms is the introduction of the genetic algorithm into aerodynamic configuration design. Later on, a large number of studies at home and abroad further developed improved versions of genetic algorithm, particle swarm search algorithm, adaptive differential evolutionary algorithm, etc., thus enhancing the capability of optimization search and the optimization efficiency. Disadvantages of the evolutionary algorithms, on the other hand, include slower convergence. Global optimization based on the agent model can both reduce the computation times of the original model and exert the global optimization capability of evolutionary algorithms, and was therefore heavily favored in engineering design and has gradually become a popular research method. In the aerodynamic designs of actual engineering projects, the results had to meet the requirements of different flight conditions and aerodynamic characteristics. In such a context, multi-objective and robust aerodynamic design optimization became a new research direction, covering multi-point weighted average, multi-objective evolution, robust design, and other methods. The combination of CFD and computational structural dynamics (CSD) with optimization theories has given birth to the multi-disciplinary optimization (MDO) design techniques, and would further influence the development of high-performance aircraft in the future. In the NASA-released research report on the prospects of CFD in year 2030, multi-disciplinary analysis and optimization was listed among the key areas of priority in the development.

Fluid-structure interaction (FSI) studies the interactions between the behavior of deformed solid under the effect of the flow field and the influence of solid movement on the flow field. Early recognition of FSI originated from the issue of aeroelasticity in aircraft engineering. Early in 1934, Theodorsen established a theoretical system of unsteady aerodynamics, which laid the foundation of study of stability, flutter, and other mechanisms under aeroelasticity [Theodorsen (1935); Theodorsen and Garrick (1942)]. Currently, aeroelasticity is mainly studied by coupling CFD and CSD, the related work of which included: POD-ROM model building based on time-lined frequency-domain Euler equation for the calculation of flutter boundaries of three-dimensional transonic wings; simulation of the limit cycle oscillation and flutter characteristics of transonic sail wings with simplified aerodynamic models; simulation of stalling flutter in low-speed wings with nonlinear aerodynamic models; etc. In the field of ship hydrodynamics, early research of fluid-solid coupling dynamics took the hull as a rigid floater and discussed the disturbance from the rigid motion of the ship on the water. Bishop and Price adopted the potential-flow theory and Green function to construct the velocity potential from the ship's motions, and also established a hydroelasticity theory by taking into account the elastic deformation of the ship. Other work included study on the linear and nonlinear response of small waterplane area twin hull (SWATH) with a pulsating source Green function. In China, academician Wu Yousheng developed a three-dimensional hydroelasticity theory of general linear-motion floaters on such a basis. In the practical FSI model of vertical vibration in two-dimensional linear high-speed craft established subsequently, the structural part and the fluid part employed two-dimensional finite element and boundary element method respectively [Wu, Zou, Tian et al. (2016)]. The model showed higher practical value in engineering operation. Later, some proposed a method of three-dimensional linear hydroelastic analysis for the treatment of fluid motion and structural deformation. The water-entry impact is another main direction of FSI research, which includes: study on the water-entry process of wedge and other shapes through experiments; study on the elastic performance in the process of structural water-entry impact using numerical simulation. The research of FSI issues also attracted much attention from other areas related to fluid matters. Vortex induced vibration (VIV) of marine flexible risers is also another typical FSI phenomenon. In this field, the strip theory has been extensively referred to in the calculation of VIV of flexible cylinders with low mass ratio and damping; the VIV characteristics of rigid cylinders were sometimes simulated by solving the RANS equation. In engineering, FSI problems in membrane structures and flow field loads can also be commonly found. Corresponding methods include building models of flexible membranes and computational methods interacting around viscous flow; FSI analysis of flow around flexible membranes with experimental and numerical methods; etc. In addition, some scholars used FSI to analyze the deformation and changes of flow parameters of membrane-structure airships in the stratosphere under the effect of the flow field, and to numerically simulate the fall of flexible parachutes experiencing large deformation in the air, etc. FSI analyses are also studied and applied in the biology-blood flow in elastic arteries, blood flow through the heart valves, air flow as a result of alveolar contraction and expansion, etc. FSI numerical methods consist of coupling strategies (methods of solution), mesh processing of the flow field, and interface data interpolation technology. Coupling strategies can be roughly divided into two categories: the monolithic approach (or full coupling) and the partitioned approach, i.e. tight coupling and loose coupling. Research patterns at home and abroad include coupling schemes, algorithms, and others of different precision. Upon the resolution of FSI problems with partitioned approach, interactions between the fluid and the structure can be reflected onto the boundary conditions of the fluid-structure interface in an explicit manner. Mesh processing technology of any deformation on the boundaries of the flow field caused by structural motion could also be divided into two types: uniform mesh and nonuniform mesh. The former includes the research related to the Eulerian-Lagrangian method; the latter includes VOF, Level-set, immersed boundary method (IBM), etc. The methods of dynamic mesh and interface data interpolation adopted by FSI have also been widely studied.

Heat transfer studies the transmission patterns of heat energy caused by temperature difference, whose applications cover energy and power, metallurgy, chemistry, transportation, construction materials, and machinery, in addition to traditional industries like food, light industry, textile, and medicine, as well as high and new technology fields like aerospace, nuclear power, microelectronics, materials, biomedical engineering, environmental engineering, and new energy. As theoretical and experimental research continue to close its gaps with production and life, microscale heat transfer, phase change of biological heat transfer, multiphase flow heat transfer, low-temperature heat transfer, and many other interdisciplinary studies and subdisciplines were derived therefrom. In recent years, with the development of micromachining and the growing demand for miniaturization of industrial products, microscale heat transfer has become a hot topic in the research of engineering thermophysics. Studies of microscale heat transfer could be divided into two aspects-spatial and temporal, of which the spatial microscale included microchannels and microstructure on the heat exchange surface. In this respect, existing research is focused on the similarities and differences between microchannels and regular channels in flow boiling and condensation mechanisms and the influence of surface characteristics (hydrophilicity/hydrophobicity, micro/nano-structure) and others on boiling bubbles, droplet nucleation and heat exchange efficiency; methods and mechanisms enhancing heat transfer have also been proposed. Meanwhile, fundamental research of heat transfer involving phase change on the microscale level is conducive to the optimization design of high heat flux cooling technology and microelectronic chip cooling technology. On the nanoscale level, research on the mechanisms of nanoparticles' plasma absorption of sunlight and the generation of boiling bubbles in nanofluids help with solar energy storage and development as well as its application in energy, medicine, pollutant treatment, and other fields. Heat transfer in fluids is closely related to the boundary layer and its external flow. Related research at home and abroad mainly focuses on: flows of special geometric effect, compressible and high-speed flows, numerical simulation and analysis technology, thin film and interface flows, flows of special fluids, combustion and reaction flows, etc. Experimental and numerical calculations of heat transfer in pipes and channels have also received much attention, especially in the field of microchannel heat transfer, which grows the fastest in terms of literature quantity over the past few years. The related studies are mainly centered on: flows in straight-walled channels and pipes, flows in horizontal and vertical heat exchangers, flows in pipes with heat-transfer-enhancement fin profile, flows and heat transfer in complex channels, multiphase flows and flows in non-Newtonian fluid channels, etc. Flow separation is a common phenomenon accompanying fast channel change or strong adverse pressure gradient, which has a significant effect on laminar and turbulent flow, as well as the efficiency and regularity of heat transfer. In this respect, recent and popular topics include the application of large eddy simulation (LES) in the numerical simulation of turbulence, rotational flow, chemical reaction flow, combustion, and other areas. Natural convection is a flow driven by temperature difference and without any other external driving force. Its popular research topics include: Rayleigh-Benard convection of specific geometry, simulation of thermocapillary flow of evaporation and transient oscillation, natural convection in multi-heat source pipes, air/water and electrochemical mass transfer, etc. Heat transfer accompanying flow in porous media covers a wide range of physical phenomena and related technologies. In this field, research has experienced a rapid growth in the past twenty years, during which multiphase transportation and porous media combustion systems, and effective heat transfer properties of fluids in porous systems have received much attention. Its specific applications include hybrid heat and mass transfer equipment, combustion systems, phase-transition systems, systems of forced convection, fluid-bed and packed-bed reactors, etc. Other heat transfer and thermal-energy-related research areas include boiling and evaporation, freezing and melting, thermal radiation transfer, and biological heat transfer. CFD numerical simulation technology of heat transfer in flow has been widely applied to the above basic research and industrial practice, and has continuously generated improved methods and new methods for higher computing capacity, precision, and wider applicability.

Modern CFD technology is mainly based on the N-S (Navier-Stokes) equation system for the solution of complex flow fields. Its application in many fields of engineering benefited from the continued development of basic fluid mechanics and numerical implementation. Currently, the most commonly used numerical discretization methods of CFD include: finite difference method (FDM), finite volume method (FVM), finite element method (FEM), boundary element method (BEM) and spectral methods. Research and development of CFD technology was mainly carried out along two main paths: computational method and computational scheme. With respect to the CFD computational method, early in the 1930s, people simplified mathematical models of fluid flow into linear potential-flow models to study the numerical solution of this elliptic Laplace's equation. After that, in order to also take viscosity, compressibility, etc., into account, the numerical solution of the Euler equation or N-S equation became the most important content of CFD. Many mathematicians have studied the mathematical theory of partial differential equation (PDE). Courant, Fredric et al. studied the basic characteristics of PDE, well-posedness of the mathematical formulation, propagation characteristics of physical waves, and other problems, and developed the hyperbolic theory of PDE. Later, Courant, Friedrichs, and Lewy published a classic paper demonstrating the existence and uniqueness theorems of continuous elliptic, parabolic, and hyperbolic solutions to the equations; addressing the problem of the initial value of linear equations, they first discretized PDE, then proved the convergence of the discrete system into a continuous system, and determined the existence of the difference solutions via algebraic approach and proposed the famous stability criterion in the end: the CFL condition. Von Neumann, Richtmyer, Hopf, Lax, and other scholars developed the numerical theory of the conservation law of nonlinear hyperbolic equations, laying the theoretic foundation for numerical simulation of flows along discontinuities containing physical quantities. Lax and Wendroff jointly developed the time-dependent method (also known as time marching method) for the solution to the problems of steady gas around the flow field of aircraft, which was further enriched by Richtmyer and Morton. Afterwards, the stability theory of the difference approximation of unsteady PDE proposed by Lax, Kreiss, and other scholars further promoted the time-dependent method. People continued with the study on the use of computational methods in solving the Euler equation and the N-S equation with the time-dependent method and came up with the characteristic-line-based operator splitting (CBOS) method and three-layer time calculation. Patankar and Spalding put forward the famous "semi-implicit method for solution of pressure coupled equations" (SIMPLE method), an important method for solving incompressible flow fields based on the "prediction-correction" process that has become one of the most important methods in the engineering application of CFD [Patankar and Spalding (1972)]. Thereafter, scholars continued to put forward several improvements, such as: the SIMPLEC method from Van Doormal and Raithby, the PISO method from Issa, etc. Recently, along with the improvement of calculation methods and the development of parallel computing, high-precision numerical simulation has become possible. Since the 1950s, research of CFD computational schemes in China has obtained splendid results. In general, computational schemes could be divided into two categories: the upwind scheme and the central scheme. The former, which considers the dissemination characteristics of the flow information, has been regarded as the most popular computational scheme in CFD today. Courant et al. took the lead in numerical computation of the Euler equation and proposed the explicit upwind scheme of the first-order accuracy. Lax and Friedrichs also developed the first-order accuracy scheme for linear convection equations. Godunov published the famous Godunov first-order upwind scheme, which opened a new approach for computational scheme construction through the calculation of Riemann discontinuity decomposition. The subsequent explicit Lax-Wendroff scheme of central difference and second-order accuracy and its follow-up development placed the cornerstone of modern CFD. Lerat developed the primitive Lax-Wendroff scheme to an implicit one. However, these schemes might show non-physical numerical oscillation near flow interruptions and thereby cause calculation failure. This has significantly limited their application. In such a context, Van Leer creatively proposed the MUSCL scheme, expanding Godunov and other first-order schemes to ones of second-order accuracy through monotone interpolation. Such an interpolation method, later known as "limiters", has been a universal method for almost all high-resolution schemes today. Steger, Warming, and Van Leer proposed new upwind schemes named after their own names respectively-Flux Vector Splitting (FVS) scheme. Almost at the same time, Roe and Osher also put forward another upwind scheme named after their own names-the Flux Difference Splitting (FDS) scheme, which actually belonged to the Godunov scheme, only with much lower computational complexity. The FDS scheme and the FVS scheme have now become the main CFD computational schemes. Harten, Hyman, Lax et al. also put forward the idea of total variation diminishing (TVD) difference scheme, and developed high-resolution TVD scheme of second-order accuracy [Harten, Hyman, Lax, et al. (1994)]. Nevertheless, at this point, TVD schemes still suffered from such defects as the local reduction of extreme points and the second order accuracy being the upper limit. To improve the TVD scheme, Harten proposed the essentially non-oscillatory (ENO) scheme of uniform higher-order accuracy, which, however, met great challenges when expanded to higher dimensions. Later, Liu, Osher, and Chan developed the WENO scheme, which shows remarkable improvement [Liu, Osher and Chan (1994)]. Aside from such foreign-based research, in China, Zhang Hanxin et al. developed a non-oscillatory dissipative difference scheme of second-order accuracy without free parameters that can meet the conditions of entropy increase and can automatically capture shock waves-the NND scheme. It possesses the nature of TVD and hence deemed to fall into the same category as the TVD scheme [Zhang and Zhuang (1991)]. At present, the research and application involving schemes of higher-order accuracy remained the popular topics in CFD.

References

Harten, A.; Hyman, J. M.; Lax, P. D.; Keyfitz, B. (1976): On finite-difference approximations and entropy conditions for shocks. *Communications on Pure and Applied Mathematics*, vol. 29, no. 3, pp. 297-322.

Liu, X. D.; Osher, S.; Chan, T. (1994): Weighted essentially non-oscillatory schemes. *Journal of Computational Physics*, vol. 115, no. 1, pp. 200-212.

Patankar, S. V.; Spalding, D. B. (1972): A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal of Heat and Mass Transfer*, vol. 15, no. 10, pp. 1787-1806.

Theodorsen, T. (1935): General theory of aerodynamic instability and the mechanism of flutter. *NASA: Ames Research Center Classical Aerodynamic Theory*, pp. 291-311.

Theodorsen, T.; Garrick, I. E. (1942): Nonstationary flow about a wing-aileron-tab combination including aerodynamic balance. *NACA Technical Reports*, pp. 129-138.

Wu, Y. S.; Zou, M. S.; Tian, C.; Sima, C.; Qi, L. B. et al. (2016): Theory and applications of coupled fluid-structure interactions of ships in waves and ocean acoustic environment. *Journal of Hydrodynamics*, vol. 28, no. 6, pp. 923-936.

Zhang, H. X.; Zhuang, F. G. (1991): NND schemes and their applications to numerical simulation of two- and three-dimensional flows. *Advances in Applied Mechanics*, vol. 29, pp. 193-256.