Hindawi Publishing Corporation
Modelling and Simulation in Engineering
Volume 2011, Article ID 304983, 3 pages
doi:10.1155/2011/304983

Editorial

Advances in Computational Fluid Dynamics and Its Applications

Guan Heng Yeoh,[1,2] Chaoqun Liu,[3] Jiyuan Tu,[4] and Victoria Timchenko[2]

[1] *Australian Nuclear Science Technology Organisation (ANSTO), Locked Bag 2001, Kirrawee, Sydney, NSW 2233, Australia*
[2] *School of Mechanical and Manufacturing Engineering, The University of New South Wales, Sydney, NSW 2052, Australia*
[3] *Center for Numerical Simulation and Modeling, Department of Mathematics, The University of Texas at Arlington, Arlington, TX 76019-0408, USA*
[4] *School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, P.O. Box 71, Bundoora, Melbourne, VIC 3083, Australia*

Correspondence should be addressed to Guan Heng Yeoh, guan.yeoh@ansto.gov.au

Received 14 October 2011; Accepted 17 October 2011

Computational fluid dynamics, better known by CFD, is certainly very prevalent in many fields of engineering research and application. There is no doubt that we are certainly witnessing a renaissance of computer simulation technology where the ever-changing landscape is as a result of the rapid evolution of CFD techniques and models and the decreasing computer hardware costs accompanied by faster computing times. In spite of the many significant achievements, there remains much concerted development and advancement of CFD to meet the increasing demands bolstered from various emerging industries such as biomedical and bioengineering, uncharted areas in process, chemical, civil, and environmental engineering as well as traditionally renowned high-technology engineering areas in aeronautics and astronautics and automotive. In this special issue, we believe that significant coverage on the advances and applications of CFD has been achieved via the selected topics and papers which incidentally represent a good panel addressing the many different aspects of CFD in this special issue. Of course, we realise that these selected topics and papers are not an exhaustive representation. Nevertheless, they still represent the rich and many-faceted knowledge that we have the great pleasure of sharing with the readers. We certainly would like to thank the numerous authors for their excellent contributions and patience in assisting us. More importantly, the tiresome work of all referees on reviewing these papers is very warmly acknowledged.

This special issue contains sixteen papers. The advances and wide applications of CFD in each of the papers are detailed in the following.

In the first paper, "*New findings by high-order DNS for late flow transition in a boundary layer*," C. Liu et al. present a summary of new discoveries by direct numerical simulation (DNS) for late stages of flow transition in a boundary layer. Based on the new findings, ring-like vortex is found to be the only form existing inside the flow field and the result of the interaction between two pairs of counter-rotating primary and secondary streamwise vortices. Following first Helmholtz vortex conservation law, the primary vortex tube rolls up and is stretched due to the velocity gradient. In order to maintain vorticity conservation, a bridge must be formed to link two Λ-vortex legs. The bridge finally develops as a new ring. This process proceeds to form a multiple ring structure. U-shaped vortices are not new but existing coherent vortex structure. U-shaped vortex, which is a third level vortex, serves as a second neck to supply vorticity to the multiple rings. The small vortices can be found on the bottom of the boundary layer near the wall surface. It is believed that the small vortices, and thus turbulence, are generated by the interaction of positive spikes and other higher level vortices with the solid wall.

In the second paper, "*Implicit LES for supersonic microramp vortex generator: new discoveries and new mechanisms*," Q. Li and C. Liu present the application of an implicitly large eddy simulation (ILES) via fifth-order WENO scheme to study the flow around the microramp vortex generator (MVG) at Mach 2.5 and $Re_\theta = 1440$. A number of new discoveries on the flow around supersonic MVG have been made including spiral points, surface separation topology, source of the momentum deficit, inflection surface, Kelvin-Helmholtz instability, vortex ring generation, ring-shock interaction, 3D recompression shock structure, and influence of MVG decline angles. A new 5-pair-vortex-tube model near the MVG is given based on the ILES observation. The vortex ring-shock interaction is found as the new mechanism of the reduction of the separation zone induced by the shock-boundary layer interaction.

In the third paper, "*An initial investigation of adjoint-based unstructured grid adaptation for vortical flow simulations*," L. Li presents a CFD method utilizing unstructured grid technology to compute vortical flow around a 65° delta wing with sharp leading edge, specially known as the geometry of the second international vortex flow experiment (VFE-2). The emphasis of this paper is to investigate the effectiveness of an adjoint-base grid adaptation method for unstructured grid in capturing concentrated vortices generated at sharp edges or flow separation lines of lifting surfaces flying at high angles of attack. Earlier vortical flow simulations indicated that the vortex behaviour has been found to be highly dependent on the local grid resolution both on body surface and space. The basic idea of the adjoint-based adaptation method has been to construct a new adaptive sensor in a grid adaptation process with the intent to allocate where the elements should be smaller or larger through the introduction of an adjoint formulation to relate the estimated functional error to local residual errors of both the primal and adjoint solutions.

In the fourth paper, "*Assessment of turbulence models for flow simulation around a wind turbine airfoil,*" F. Villalpando et al. present the investigation of flows over a NACA 63–415 airfoil at various angles of attack. With the aim of selecting the most suitable turbulence model to simulate flow around ice-accreted airfoils, this work concentrates on assessing the prediction capabilities of various turbulence models on clean airfoils at the large angles of attack that cause highly separated flows to occur. CFD simulations have been performed employing the one-equation Spalart-Allmaras model, the two-equation RNG k-ϵ and SST k-ω models, and Reynolds stress model.

In the fifth paper, "*Computation of ice shedding trajectories using cartesian grids, penalization, and level sets*," H. Beaugendre et al. present the modelling of ice shedding trajectories by an innovative paradigm based on Cartesian grids, penalization, and level sets. The use of Cartesian grids bypasses the meshing issue, and penalization is an efficient alternative to explicitly impose boundary conditions so that the body-fitted meshes can be avoided, making multifluid/multiphysics flows easy to set up and simulate. Level sets describe the geometry in a nonparametric way so that geometrical and topological changes due to physics and in particular shed ice pieces are straightforward to follow. The capabilities of the proposed CFD model are demonstrated on ice trajectories calculations for flow around iced cylinder and airfoil.

In the sixth paper, "*Aerodynamic optimization of an over-the-wing-nacelle-mount configuration,*" D. Sasaki and K. Nakahashi present the CFD investigation of an over-the-wing-nacelle-mount airplane configuration which is known to prevent the noise propagation from jet engines toward ground. Aerodynamic design optimization is conducted to improve aerodynamic efficiency in order to be equivalent to conventional under-the-wing-nacelle-mount configuration. The nacelle and wing geometry are modified to achieve high lift-to-drag ratio, and the optimal geometry is compared with a conventional configuration. Pylon shape is also modified to reduce aerodynamic interference effect. The final wing-fuselage-nacelle model is compared with the DLR F6 model to discuss the potential of over-the-wing-nacelle-mount geometry for an environmental-friendly future aircraft.

In the seventh paper, "*New evaluation technique for WTG design wind speed using a CFD-model-based unsteady flow simulation with wind direction changes*," T. Uchida et al. present the development of a CFD model called RIAM-COMPACT, based on large eddy simulation (LES), that can predict airflow and gas diffusion over complex terrain with high accuracy and simulates a continuous wind direction change over 360 degrees. The present paper proposes a technique for evaluating the deployment location of wind turbine generators (WTGs) since a significant portion of the topography in Japan is characterized by steep, complex terrain thereby resulting in a complex spatial distribution of wind speed, in which great care is necessary for selecting a site for the construction of WTG.

In the eighth paper, "*Comparisons between the wake of a wind turbine generator operated at optimal tip speed ratio and the wake of a stationary disk*," T. Uchinda et al. present the investigation that the wake of a wind turbine generator (WTG) operated at the optimal tip speed ratio is compared to the wake with its rotor replaced by a stationary disk. Numerical simulations are conducted with a large eddy simulation (LES) model. The characteristics of the wake of the stationary disk are significantly different from those of the WTG. The velocity deficit at a downstream distance of $10\,D$ (D being the rotor diameter) behind the WTG is approximately 30 to 40% of the inflow velocity. In contrast, flow separation is observed immediately behind the stationary disk ($\leq 2\,D$), and the velocity deficit in the far wake ($10\,D$) of the stationary disk is smaller than that of the WTG.

In the ninth paper, "*Experimental and numerical simulations predictions comparison of power and efficiency in hydraulic turbine,*" L. Castro et al. present the CFD simulations performed to principal components of a hydraulic turbine: runner and draft tube. On-site power and mass flow rate measurements have been conducted in a hydroelectric power plant (Mexico). Mass flow rate has been obtained using Gibson's water hammer-based method. Inlet boundary conditions for the runner have been obtained from a previous simulation conducted in the spiral case. Computed results at the runner's outlet are used to conduct the subsequent draft tube simulation. Numerical results from the runner's flow simulation provide data to compute the torque and the turbine's power. Power-versus-efficiency curves are built.

In the tenth paper, "*Simulating smoke filling in big halls by computational fluid dynamics,*" W. K. Chow et al. present the use of CFD in addressing the key hazard due to smoke filling within the enclosure in the event of a fire. This aspect primarily concerns the many tall halls of big space volume that have been and to be built in many construction projects in the Far East, particularly Mainland China, Hong Kong, and Taiwan. On the basis of the CFD simulations, it can be anticipated that better understanding of the smoke filling phenomenon and design of appropriate smoke exhaust systems can be realised.

In the eleventh paper, "*Numerical simulations for a typical train fire in china*," W. K. Chow et al. present the use of CFD in addressing the fire safety in passenger trains due to big arson and accidental fires since railway is the key transport means in China including the Mainland, Taiwan, and Hong Kong. The predicted results such as the air flow, temperature distribution, smoke layer height, and smoke spread patterns inside a train compartment are useful for working out appropriate fire safety measures for train vehicles and determining the design fire for subway stations and railway tunnels.

In the twelfth paper, "*Simulation of pharyngeal airway interaction with air flow using low-Re turbulence model*," M. R. Rasani et al. present the CFD simulation of the interaction between a simplified tongue replica with expiratory turbulent air flow. A three-dimensional model with a low-Re SST turbulence model is adopted. An Arbitrary Eulerian-Lagrangian description for the fluid governing equation is coupled with the Lagrangian structural solver via a partitioned approach, allowing deformation of the fluid domain to be captured. Numerical simulations confirm expected predisposition of apneic patients with narrower airway opening to flow obstruction and suggest much severe tongue collapsibility if the pharyngeal flow regime is turbulent compared to laminar.

In the thirteenth paper, "*CFD-guided development of test rigs for studying erosion and large-particle damage of thermal barrier coatings*," M. A. Kuczmarski et al. present the use of CFD to accelerate the successful development and continuous improvement of combustion burner rigs for meaningful materials testing. Rig development is typically an iterative process of making incremental modifications to improve the rig performance for testing requirements. Application of CFD allows many of these iterations to be done computationally before hardware is built or modified, reducing overall testing costs and time, and it can provide an improved understanding of how these rigs operate. This paper focuses on the study of erosion and large-particle damage of thermal barrier coatings (TBCs) used to protect turbine blades from high heat fluxes in combustion engines. The steps used in this study—determining the questions that need to be answered regarding the test rig performance, developing and validating the model, and using it to predict rig performance—can be applied to the efficient development of other test rigs.

In the fourteenth paper, "*Investigation of swirling flows in mixing chambers*," J. J. Chen and C. H. Chen present the CFD investigation of the three-dimensional momentum and mass transfer characteristics arising from multiple inlets and a single outlet in micromixing chamber. The chamber consists of a right square prism, an octagonal prism, or a cylinder. Numerical results have indicated that the swirling flows inside the chamber dominate the mixing index. Particle trajectories demonstrate the rotational and extensional local flows which produce steady stirring, and the configuration of coloured particles at the outlet section expressed at different Reynolds numbers (Re) represented the mixing performance qualitatively. Effects of various geometric parameters and Re on the mixing characteristics have been investigated. An optimal design of the cylindrical chamber with 4 inlets is found. At $Re > 15$, more inertia causes more powerful swirling flows in the chamber, and the damping effect on diffusion is diminished, which subsequently increases the mixing performance.

In the fifteenth paper, "*Numerical computation and investigation of the characteristics of microscale synthetic jets*," A. Lee et al. present the CFD investigation of a synthetic jet generated as a result of the periodic oscillations of a membrane in a cavity. A novel moving mesh algorithm to simulate the formation of jet is presented. The governing equations are transformed into the curvilinear coordinate system in which the grid velocities evaluated are then fed into the computation of the flow in the cavity domain thus allowing the conservation equations of mass and momentum to be solved within the stationary computational domain. Numerical solution generated using this moving mesh approach is compared with experimental measurement. Comparisons between numerical results and experimental measurement on the streamwise component of velocity profiles at the orifice exit and along the centerline of the pulsating jet in microchannel as well as the location of vortex core indicate that there is good agreement, thereby demonstrating that the moving mesh algorithm developed is valid.

In the sixteenth paper, "*Recent efforts for credible CFD simulations in china*," L. Li et al. present the initiation of a series of workshops on credible CFD simulations similar to activities in the West such as ECARP and AIAA Drag Prediction Workshops. Another major effort in China is the ongoing project to establish a software platform for studying the credibility of CFD solvers and performing credible CFD simulations. The platform, named WiseCFD, has been designed to implement a seamless CFD process and to circumvent tedious repeating manual operations. A concerted focus is also given on a powerful job manager for CFD with capabilities to support plug and play (PnP) solver integration as well as distributed or parallel computations. Some future work on WiseCFD is proposed and also envisioned on how WiseCFD and the European QNET-CFD Knowledge Base can benefit mutually.

Guan Heng Yeoh
Chaoqun Liu
Jiyuan Tu
Victoria Timchenko