Editorial: Special Issue on Computational Problems in Applied Mathematics

Sukumar SENTHILKUMAR*

School of Computing Science and Engineering, Vellore Institute of Technology University, Vellore 632014, Tamilnadu, India

(*Corresponding author; e-mail: senthilkumar@vit.ac.in; ssenthilkumar1974@yahoo.co.in)

Computational Fluid Dynamics (CFD) is a highly interdisciplinary research area which lies at the interface of physics, applied mathematics, and computer science. CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. Theoretical and Computational Fluid Dynamics provides a forum for the cross-fertilization of notions, tools and techniques across all disciplines in which fluid flow plays a role, such as: aeronautical sciences, geophysical and environmental sciences, life sciences and materials sciences. Furthermore, computational fluid dynamics is considered an indispensable analysis/design tool in an ever-increasing range of diversified industrial applications. Practical flow problems are often so complex because a high level of ingenuity is needed. Therefore, besides the development of work in CFD, innovative CFD applications are also encouraged to solve real time problems. The accuracy and fidelity of modern CFD methods have significantly increased the level of design insight available to engineers throughout the design process and hence greatly reduces companies' exposure to technical risk when developing thermal and fluid-based products. The use of CFD in design generally leads to far fewer physical prototypes being necessary during development, far less prototype testing and consequently reduces the time-to-market and cost-to-market substantially. It is well known for the researchers working in the field of fluid (both gas and liquid) flows are governed by partial differential equations which represent conservation laws for the mass, momentum, and energy. CFD is the art of replacing such partial differential equation systems by a set of algebraic equations which can be solved using digital computers. Some of the practical application includes aerodynamics, industrial fluid dynamics, fluid structure interaction, heat transfer, hydrodynamics, and multi-phase flows etc.

In specific, fluid flows encountered in everyday life include, meteorological phenomena (rain, wind, hurricanes, floods, fires), environmental hazards (air pollution, transport of contaminants), heating, ventilation and air conditioning of buildings, cars etc., combustion in automobile engines and other propulsion systems, interaction of various objects with the surrounding air/water, complex flows in furnaces, heat exchangers, chemical reactors etc., processes in human body (blood flow, breathing, drinking, . . .) and so on. CFD provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations), numerical methods (discretization and solution techniques) and software tools (solvers, pre processing and post processing utilities). Moreover, CFD enables scientists and engineers to perform 'numerical experiments' (i.e. computer simulations) in a 'virtual flow laboratory' in order to obtain feasible results. In case of numerical simulations of fluid flow (will) enable architects to design comfortable and safe living environments, designers of vehicles to improve the aerodynamic characteristics, chemical engineers to maximize the yield from their equipment, petroleum engineers to devise optimal oil recovery strategies, surgeons to cure arterial diseases (computational hemodynamics), meteorologists to forecast the weather and warn of natural disasters, safety experts to reduce health risks from radiation and other hazards, military organizations to develop weapons and estimate the damage, CFD practitioners to make big bucks by selling colorful pictures. CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques. Therefore, it is essential to have simulation techniques to know better understanding of the behavior of fluids. Performing real time experiments is expensive, slow, sequential and single-purpose but in case of simulation, cheaper, faster, parallel and multi-purpose. It is to be noted that, the results of a CFD simulation are never 100 % reliable because of the input data may involve too much guessing or imprecision, the mathematical model of the problem at hand may be inadequate and the accuracy of the results is limited by the available computing power. The limitations of computational fluid dynamics includes that their solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.) and in addition CFD solutions can only be as accurate as the physical models on which they are based. Research is still being carried out in CFD on wide varieties of areas such as two-phase flows, heat transfer, acoustics, fluid-solid interaction, Navier-Strokes solution techniques for incompressible and compressible flows, convergence acceleration procedures, grid generation and adaptation techniques, distributed computing, turbulence, mesh-free methods, free-surfaces, chemical reactions and combustion, discretisation methods and schemes, unsteady flows etc. Flow simulation offers a wide range of physical models and fluid flow capabilities by covering wide range of applications in incompressible and compressible liquid, water vapor (steam), real gases, heat transfer in solids, non-Newtonian liquids (to simulate blood, honey, molten plastics), compressible gas, conjugate heat transfer, subsonic, transonic, and supersonic regimes, external and internal fluid flows, laminar, turbulent, and transitional flows, liquid and gas flow with heat transfer, time-dependent flow, gas mixture, liquid mixture, etc.

It is well known fact that CFD is a highly interdisciplinary research area which lies at the interface of physics, applied mathematics, and computer science. There are some basic requirements needed to carry out computational fluid dynamics analysis and its process such as Problem statement (information about the flow), Mathematical model, Mesh generation, Space discretization, Time discretization, Iterative solver, CFD solver, Simulation run, Post processing and Verification / validation. There are large number of commercial CFD packages available nowadays and CFD has established itself as a useful analysis and design tool. Commercial CFD tools usually come with user's guides and examples manuals that provide users with information of how to use that particular tool. But, in most cases, there is no explanation of the theoretical background which enables the user to make an informed choice of the technique used, or the type of boundary conditions to apply. Any CFD material / software should provide the end user with the theoretical background of basic CFD methods with / without going into deep detail of the mathematics or numerical algorithms. Some of the available CFD simulation software like ANSYS CFX, FLUENT, STAR-CD, FEMLAB and FEATFLOW helps end users to solve more complicated problems in order to obtain better solution. For instance, ANSYS computational fluid dynamics (CFD) simulation software is a comprehensive suite of products that allows end user to predict, with confidence, the impact of fluid flows on user product throughout design and manufacturing as well as during end use. The software's unparalleled fluid flow analysis capabilities can be employed to design and optimize novel equipment and to troubleshoot already existing installations. For different fluid flow phenomena such as single- or multi-phase, isothermal or reacting, compressible or not solutions can be obtained through ANSYS fluid dynamics. In specific, ANSYS renowned fluid analysis tools include the widely used and well-validated ANSYS Fluent and ANSYS CFX, available separately or together in the ANSYS CFD bundle. Combining Fluent or CFX with the full-featured ANSYS CFD-Post fluid flow post-processing tool allows end user to perform advanced quantitative analysis or create high-quality visualizations and animations. Finally, the ultimate goal of CFD is to provide a common platform for model/software developers and end users by balancing interdisciplinary contributions, disseminating information relating to development / refinement of mathematical and numerical models, software tools and their innovative applications. Since the technology has improved a lot, most of the application software is robust, rapid, highly scalable, providing efficient parallel calculations from a few to thousands of processing cores. Researchers across a wide range of industries can benefit from CFD-such as automotive, aerospace, defense, life science, machinery, and high tech. Indeed, almost every design encounters fluid dynamics at some point, whether heat or liquids, internal or external.

As a guest editor of this special issue, I would like to take this opportunity to acknowledge the effort of many reviewers, editorial board members and editor who helped us to shape the content of this volume perfect.