International Journal of Advanced Engineering and Technology
ISSN: 2456-7655
Impact Factor: RJIF 5.54
www.newengineeringjournal.com
Volume 2; Issue 2; May 2018; Page No. 40-41

Computational fluid dynamics

Matthew NO Sadiku, Yonghui Wang, Suxia Cui, Sarhan M Musa
Roy G. Perry College of Engineering, Prairie View A&M University, Prairie View, Texas, USA

Abstract
Computational fluid dynamics deals with the numerical solutions of equations governing transport of mass, momentum, and energy in moving fluids. It is essentially based on the Navier-Stokes equations, which describe the motion of viscous fluid. It involves solving a set of transport equations using numerical techniques such as finite different method and finite element method. It is the key design tool for mechanical engineers. This paper presents a brief introduction to computational fluid dynamics.

Keywords: computational fluid dynamics, partial different equations

Introduction

Fluid dynamics involves fluid motion and is usually studied in three ways: theoretical analysis, experiment, and numerically (computational fluid dynamics). These three ways are illustrated in Figure 1 [1].

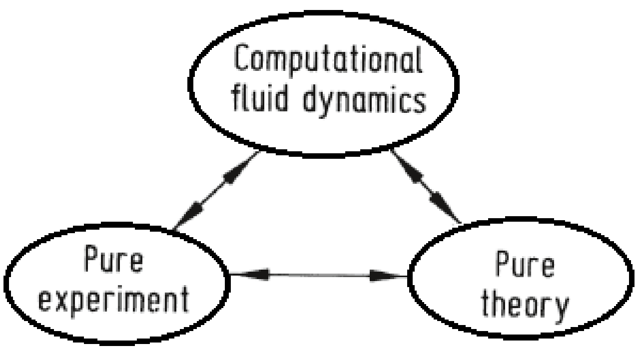

Fig 1: Relationship between CFD, theory, and experiment [1].

Computational fluid dynamics (CFD) is an emerging branch of fluid (gas and liquid) mechanics that applies numerical techniques to solve problems that involve fluid flows. It involves the numerical solutions of equations governing transport of mass, momentum and energy in moving fluids. In a CFD analysis, the examination of fluid flow is according to its physical properties such as velocity, pressure, temperature, density, and viscosity.

The basic mathematical description for all CDF problems models is given by the Navier-Stokes equations, representing the conservation of mass, momentum, and energy. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. Atmospheric currents, ocean was, the flow of air around moving vehicles, the flow of blood through the body are all in principle governed by the same set of partial different equations [2]. At the algorithm level, CFD is similar to computational electromagnetics, computational physics, computational biology, computational chemistry, and many other computational schemes in science and engineering [3].

Basic Procedure

The commercial software packages for solving CFD problems include COMSOL, ANSYS, CFX, and STARCCM. These codes are capable of producing accurate solutions of flows. All CFD codes has three elements: (1) a preprocessor, (2) a flow solver, and (3) a post-processor. Thus, the application of the CFD to analyze a fluid problem requires taking the following steps [4].

- *Preprocessing*: This is used to input the problem geometry. The physical geometry and boundary conditions (or initial conditions) of the problem are specified using computer aided design (CAD). The volume occupied by the fluid is divided into discrete cells (or mesh). The mesh may be uniform or non-uniform, structured or unstructured, consisting of a combination of hexahedral, tetrahedral, prismatic, pyramidal or polyhedral elements. The equations of fluid motion and the material parameters are defined.
- *Processing (Simulation)*: The partial differential equations describing the problem are solved iteratively as a steady-state or transient. Different methods can be used by the solver. These include the finite volume method (FVM), the finite element method (FEM), the finite difference method (FDM), the spectral element method, and the boundary element method.
- P*ostprocessing*: Postprocessing of the simulation results is performed to extract the desired information from the resulting solution. This may involve computing forces and other quantities of interest. This is used for the analysis and visualization of the results.

Applications

CFD is widely applied in areas of fluid and heat transfer. It is used in automotive, aerospace, biomedical, marine, oil and gas, hydraulics, chemical processing, food industry, biomedical industry, chemical processing industry or any field where the fluid motion plays a critical role. Applications of CFD include [5, 6]:

- Flow and heat transfer in industrial processes (boilers, combustion equipment, heat exchangers, etc.)
- Aerodynamics of ground vehicles, aircraft, and missiles
- Heating, cooling, and ventilation flows in building
- Model the flow of blood in the heart and vessels in flow of heart assist devices
- Simulation of the blood flow in the human heart
- Simulation of gas flow in models of the human lung
- Simulation of many processes in the food industry
- Simulation for spray-freezing in a cold gas
- Simulation of different parts of an industrial freeze-drying equipment
- Design and characterization of cooling systems in electrical machines
- Circuit-breaker flow analysis
- Fire and smoke modeling in buildings
- Simulation and analysis of greenhouse solar dryer

Benefits and Challenges

CFD has low, affordable cost compared to physical experiments, is fast, is able to simulate real conditions, can handle complex problems, and yields a comprehensive set of parameters for examination. Although some tools for solving CFD problems are commercially available to the engineer, the strengths of these tools must be weighed against their limitations when making decisions. The danger lies in using the tools as a kind of black box where we input data and produce output. We should be aware of the limitations of these tools.

One must be careful when interpreting the results produced by CFD techniques because many potential sources of error may be involved. Solving problems using computers introduces numerical errors such as truncation error and round-off error.

Conclusion

Computational fluid dynamics is a field that uses numerical tools to simulate and analyze problems involving fluid flow. It has evolved from being a research tool to a powerful design tool. It has been regarded as the new “third dimension” in fluid dynamics, the other two dimensions being theory and experiment. Since many interesting fluid problems require large amounts of time, parallel CFD models have been developed to significantly reduce computation time [7].

CFD has gained acceptance in engineering education at graduate and undergraduate levels. Students use FORTRAN, MATLAB or Excel (spreadsheet) to solve problems. They are also exposed to commercial CFD packages [8, 9]. For more information on computational fluid dynamics, one should consult [1, 10-16] and several other books on CFD available on Amazon.com.

References

1. Wendt JF. (ed.), Computational Fluid Dynamics: An Introduction. Berlin: Springer, 3rd ed., 2009, 7.
2. Quarteroni A, *et al*. Computational fluid dynamics at CRS4, Italy, Computing in Science & Engineering, 2012, 10-19.
3. Khajeh-Saeed A, Perot JB. Computational fluid dynamics simulation using many graphics processors, computing in Science & Engineering, 2012, 10-19.
4. Computational fluid dynamics, Wikipedia, the free encyclopedia https://en.wikipedia.org/wiki/Computational_fluid_dynamics
5. Bakker A. Applied computation fluid dynamics, http;//www.bakker.org
6. Lin CL, *et al*., Computational fluid dynamics, IEEE Engineering in Medicine and Biology Magazine, 2009, 25-33.
7. Succi S, Papetti F. An Introduction to Parallel Computational Fluid Dynamics. Commack, NY: Nova Science Publishers, 1997.
8. Adair D, Jaeger M. Incorporating computational fluid dynamics code development into an undergraduate engineering course, International Journal of Mechanical Engineering Education. 2015; 43(3):153-167.
9. Barber T, Timchenko V. Student-specific projects for greater engagement in a computational fluid dynamics course, Australasian Journal of Engineering Education. 2011; 17(2):129-138.
10. Lohner R. Applied Computational Fluid Dynamics Techniques: An Introduction Based on Finite Element Methods. Chichester, UK: John Wiley & Sons, 2008.
11. Sharma A. Introduction to Computational Fluid Dynamics: Development, Application and Analysis. John Wiley & Sons, 2017.
12. Tu J, Yeoh GH, Liu C. Computational Fluid Dynamics. Elsevier, 3rd edition, 2019.
13. Versteeg HK, Malalasekera W. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Essex, UK: Prentice Education, 2nd edition, 2007.
14. Andersson B, *et al*. Computational Fluid Dynamics for Engineers. New York: Cambridge University Press, 2012.
15. Roache PJ. Fundamentals of Computational Fluid Dynamics. Hermosa Publishers, 1998.
16. Abbott MB, Basco DR. Computational Fluid Dynamics - An Introduction for Engineers. John Wiley & Sons, 1989.