International Journal of Information Science and Computing 3(2): December, 2016: p. 117-120
DOI : 10.5958/2454-9533.2016.00014.4

Introduction to Computational Fluid Dynamics

Purna Chandra Barman

Department of Physics, Raiganj University, West Bengal India.

Corressponding author : purna.barman@gmail.com

Abstract

Computational fluid dynamics (CFD) is an essential tool, which usually uses numerical methods for developing approximations of the governing equations of fluid mechanics in the fluid region of interest.It sufficiently predicts fluid flow, heat transfer, mass transfer, chemical reactions, andrelated phenomena by solving the mathematical equations which govern these processes using numerical methods. Applying the fundamental laws of mechanics to a fluid, a set of coupled non linear partial differential equations is obtained. For most engineering problems these equations are solved analytically. Considering physical characteristics of fluid such as conservation of matter, momentum, and energy etc. throughout the region of interest of that fluid mathematical models are composed. In order to solve easily the simplifying assumptions are made for appropriate initial and boundary conditions for the problem. To solve the fundamental equation of fluid dynamics, the Navier-Stokes equation and continuity equation, appropriate initial conditions and boundary conditions must be need to be applied. In spite of this, computational fluid dynamics can be used in the many areas of food processing industries for drying, sterilization, mixing, refrigeration etc. Drying is a common food manufacturing process.

Computational fluid dynamics (CFD) is the branch of science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and diverse related phenomena by solving the mathematical equations which govern these processes using a numerical process. It uses powerful computers and applied mathematics to model fluid flow situations at which it belongs. In today's society, the ability to simulate the numerical problems relating to fluid on the move has become part of everyday life. The increase in the volume and complexity of the simulation sent in this way has inevitably leaded to increasing demand on the CFD used to access it.

During last three decades, this field had met the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. However, Up until the last 10 years this field has just made a big jump especially due to the introduction of new computer hardware and software. As a developing science, CFD has received extensive attention throughout the international community since the advent of the digital computer. Since the late 1960s, there has been considerable growth in the development and application of CFD to all aspects of fluid dynamics [1]. As a result, CFD has become an integral part of the engineering design and analysis environment of many companies because of its ability to predict the performance of new designs or processes before they are ever manufactured or implemented [2]. In design and development, CFD programs are now considered to be standard numerical tools which predict not only fluid flow behavior, but also the transfer of heat, mass, phase change, chemical reaction, mechanical movement and stress or deformation of related solid structures [3]. Furthermore, CFD has been applied to deal with problems in environment and architecture.

How CFD works

It is a mathematical model of a physical problem used to analyze the fluids' flow. To formulate the problem at first we have to satisfied conservation of matter, momentum, and energy throughout the region of interest. Fluid properties are modeled empirically. Then simplifying assumptions are made in order to make the problem tractable for the steady-state, incompressible, inviscid, two-dimensional fluid. After formulating the problems appropriate boundary conditions are to be provided. CFD applies numerical methods which are recognized as discretization to develop approximations of the governing equations of fluid mechanics in the fluid region of interest.

A) Discretization methods: The stability of the discretisation is generally established numerically [4] rather than analytically as with simple linear problems. Special attention also is taken to ensure that the discretisation handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces. Some of the discretization methods being used are: a) Finite volume method, b) Finite element method and c) Finite difference method etc.

a) Finite volume method: The finite volume method is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows [5] In the finite volume method, the governing partial differential equations generally the Navier-Stokes equations, the mass and energy conservation equations, and the turbulence equations are develop in a conservative form, and then solved over discrete control volumes.

b) Finite element method: The finite element method is used in structural analysis of solids, but is also applicable to fluids. However, this formulation requires special care to ensure a conservative solution. Its formulation has been adapted for use with fluid dynamics governing equations. Although, the finite element method must be carefully formulated to be conservative, it is much more stable than the finite volume approach [6]. However, the finite element method can require more memory and has slower solution times than the finite volume method [7].

c) Finite difference method: The finite difference method has historical importance and is simple to program. It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids.

Applications of CFD

CFD has various applications for the flow and heat transfer in industrial processes such as boilers, heat exchangers, combustion equipment, pumps, blowers, piping, etc. Along with Aerodynamics of ground vehicles, aircraft, missiles, film coating, thermoforming in material processing applications. In chemical vapor deposition for integrated circuit manufacturing industries it can also be used for heat transfer for electronics packaging applications. CFD has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics. It allows for a deep analysis of the fluid mechanics and local effects in a lot of equipment. Most of the CFD results will give an improved performance, better reliability, more confident scale-up, improved product consistency, and higher plant productivity [8]. It provides a detailed understanding of flow distribution, weight losses, mass and heat transfer, particulate separation, etc. Consequently, all these will give plant managers a much better and deeper understanding of what is happening in a particular process or system. It makes it possible to evaluate geometric changes with much less time and cost than would be involved in laboratory testing. Many food processing operations such as chilling, drying, baking, mixing, freezing, cooking, pasteurization and sterilization rely on fluid flow. The transfer of CFD approaches to the food industry has provided food engineers new insight and understanding to the likely performance of food equipment at the design stage and confidence in the quality or safety of food products [9]. Equipment such as ovens, heat exchangers, refrigerated display cabinets and spray dryers has been improved through the application of CFD techniques in aiding the understanding of their operation and design process. CFD has become a powerful tool in the development, trouble shooting and optimization of food processes

Conclusion

Computational fluid dynamics (CFD) is a method to numerically use to calculate heat and mass transfer and fluid flow. Its main application is found as an engineering method, to provide data that is complementary to theoretical and experimental data. This is mainly the domain of commercially available codes and in-house codes at large companies. CFD can also be used for purely scientific studies. This is more common in academic institutions and government research laboratories. Codes are usually developed to specifically study a certain problem.

References

1. Parviz, M., John, K., 1997. Tackling turbulence with supercomputers. *Scientific American* 1, 276.

2. Schaldach, G., Berger, L., Razilov, I., Berndt, H., 2000. Computer simulation for fundamental studies and optimisation of ICP spray chambers. *ISAS (Institute of Spectrochemistry and Applied Spectroscopy) Current Research Reports*, Berlin, Germany.

3. Bin Xia, Da-Wen Sun, 2002. Applications of computational fluid dynamics (CFD) in the food industry: a review. *Computers and Electronics in Agriculture* 34, 5–24. https://en.wikipedia.org/wiki/Computational fluid dynamics.

4. Patankar, Suhas V. (1980). *Numerical Heat Transfer and Fluid FLow*. Hemisphere Publishing Corporation. ISBN 0891165223.

5. Surana, K.A.; Allu, S.; Tenpas, P.W.; Reddy, J.N. (February 2007). “k-version of finite element method in gas dynamics: higher-order global differentiability numerical solutions”. *International Journal for Numerical Methods in Engineering*. 69 (6): 1109–1157. Bibcode:2007IJNME..69.1109S. doi:10.1002/nme.1801.

6. Huebner, K. H.; Thornton, E. A.; and Byron, T. D. (1995). *The Finite Element Method for Engineers* (Third ed.). Wiley Interscience.

7. Bakker, A., Ahmad, H.H., Lanre, M.O., 2001. Realize greater benefits from CFD. *Fluid/Solids Handling* March, pp. 45–53.

8. FRPERC, 1995. CFD in the food industry. *Food Refrigeration and Process Engineering Research Centre Newsletter* 10, 1–2.